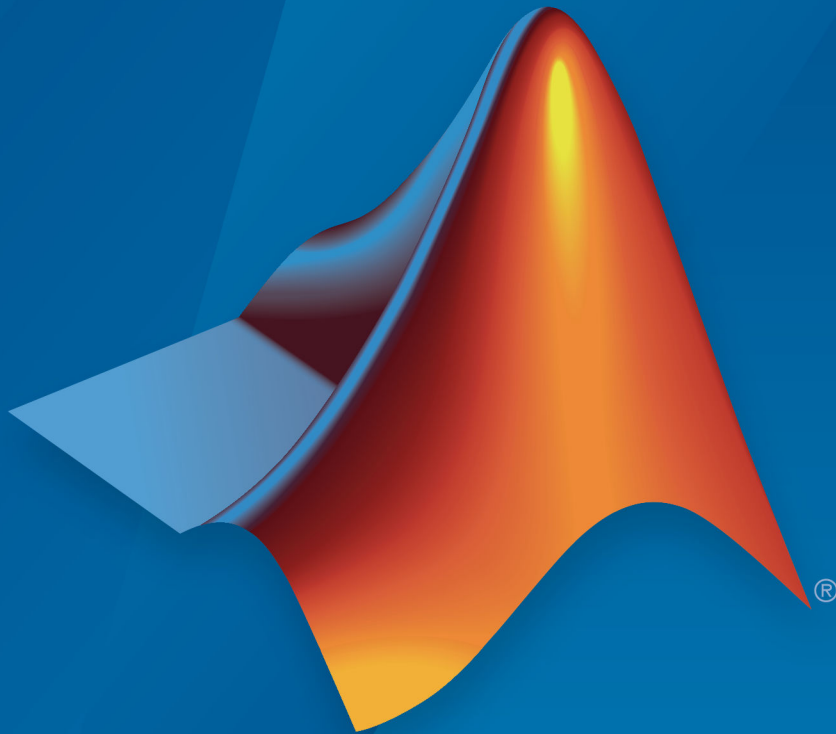


Simscape™ Fluids™

Reference



MATLAB® & SIMULINK®

R2017b



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 2006–2017 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 <i>SimHydraulics</i> [®] <i>Reference</i>)
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)

Blocks — Alphabetical List

1

Bibliography

A

Glossary

Blocks — Alphabetical List

2-Position Valve Actuator

Actuator for two-position valves



Library

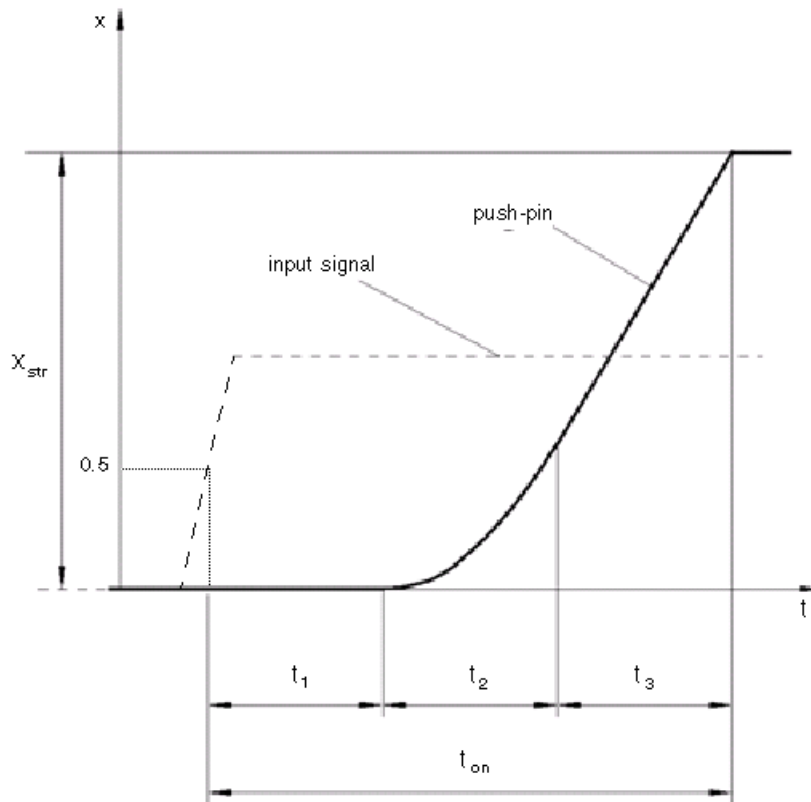
Valve Actuators

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.

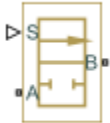
See Also

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 $5e-5 \text{ m}^2$. 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 $5e-3 \text{ m}$. 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 $1e-12 \text{ m}^2$.

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.

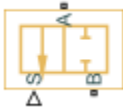
See Also

3-Way Directional Valve | 4-Way Directional Valve

Introduced in R2006a

2-Way Directional Valve (TL)

Two-port, two-position directional control valve

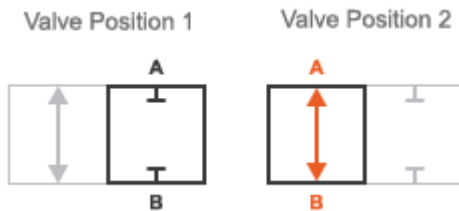


Library

Thermal Liquid/Valves/Directional Control Valves

Description

The 2-Way Directional Valve (TL) block represents a generic two-port, two-position directional control valve for the thermal liquid domain. The valve positions correspond to closed and open configurations between the thermal liquid conserving ports A and B. A closed valve allows only leakage flow between the two ports.



Valve Positions

The physical signal input port S sets the valve position through a control member displacement signal. The valve is in position 1 at a zero control member displacement. It is in position 2 at the maximum allowable displacement specified through the **Maximum valve opening** block parameter.

Parameters

Valve area parameterization

Parameterization used to calculate the valve opening area. Select `Linear area-opening relationship` to calculate this area from the valve control member displacement. Select `Tabulated data – Area vs. opening` to specify the valve open area as a lookup table parameterized in terms of the valve control member displacement. The default setting is `Linear area-opening relationship`.

Maximum valve opening

Control member displacement at which the valve opening area reaches its maximum value. The physical signal input S saturates at this value. This parameter is active only when the **Orifice area parameterization** parameter is set to `Analytical`. The default value is 0.005 m .

Maximum opening area

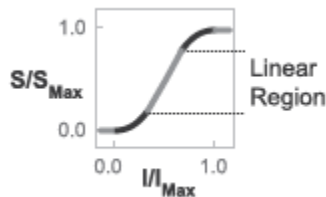
Valve cross-sectional area in the fully open position. This area corresponds to the maximum control member displacement. This parameter is active only when the **Orifice area parameterization** parameter is set to `Analytical`. The default value is $1\text{e-}4\text{ m}^2$.

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 $1\text{e-}10\text{ m}^2$.

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

Enter a value of 0 for zero smoothing. Enter a value of 1 for full-curve smoothing. The default value is 0.01. This parameter is visible only when the **Valve area parameterization** parameter is set to Linear area-opening relationship.

Valve opening vector

Vector of control member displacements at which to specify the valve cross-sectional area. This parameter is visible only when the **Orifice area parameterization** parameter is set to Tabulated. The default vector is a five-element vector ranging from -0.002 to 0.015 m.

Opening area vector

Vector of valve opening areas corresponding to the control member displacements specified through the **Valve opening vector** parameter. This parameter is visible only when the **Orifice area parameterization** parameter is set to Tabulated. The default vector is a five-element vector ranging from $1.0e-9$ to 0.00034356 m^2 .

Valve opening offset

Control member offset from the zero position. The control member position is the sum of the offset specified and the physical signal input S. The offset value does not affect the hard stop locations. The default value is 0 m.

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.

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

3-Way Directional Valve (TL) | 4-Way Directional Valve (TL) | Check Valve (TL)

Introduced in R2016a

3-Position Valve Actuator

Actuator for three-position valves



Library

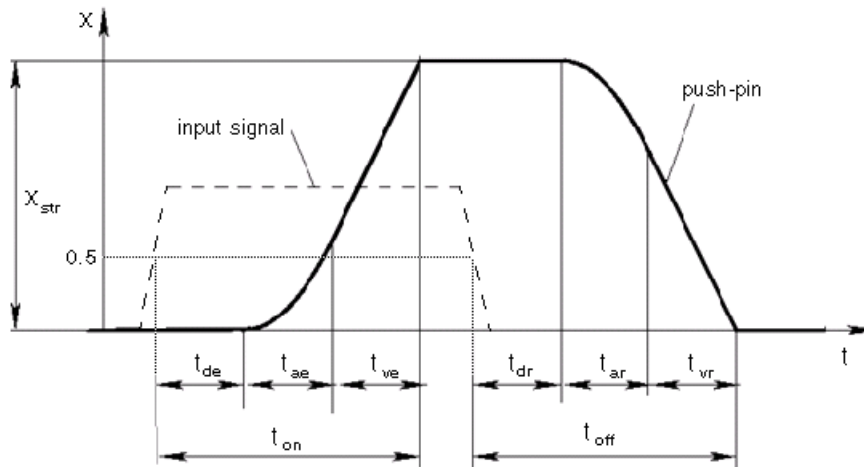
Valve Actuators

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

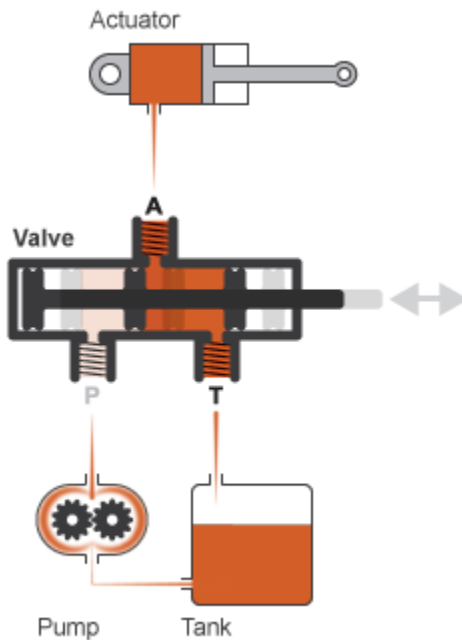
Three-port two-position directional control valve

Library: Hydraulics (Isothermal) / Valves / Directional Valves



Description

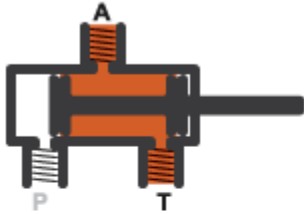
The 3-Way Directional Valve block represents a directional control valve with three ports and two positions, or flow paths. The ports connect to what in a typical model are a hydraulic pump (port **P**), a storage tank (port **T**), and a single-acting actuator (port **A**). Fluid can flow from the pump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



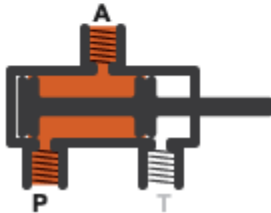
Typical Valve Setup

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:

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

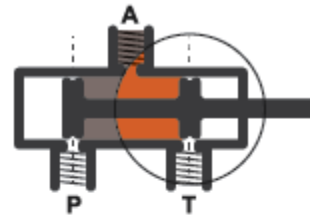
$$h_{AT} = h_{AT0} - x,$$

where:

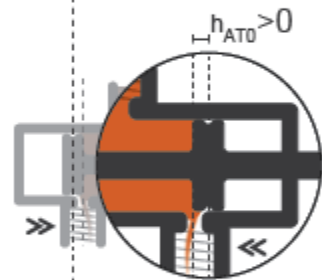
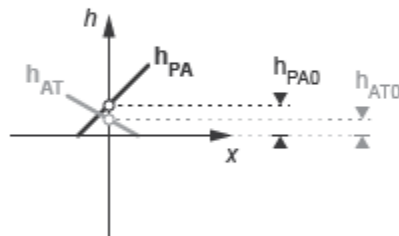
- h_{PA} and h_{AT} are the orifice openings of the **P-A** and **A-T** flow paths. The orifice openings are computed during simulation.
- h_{PA0} and h_{AT0} are the opening offsets of the **P-A** and **A-T** flow paths. The opening offsets are specified in the **Valve opening offsets** tab.
- x is 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.

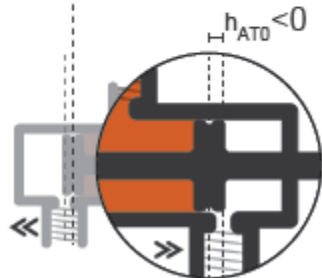
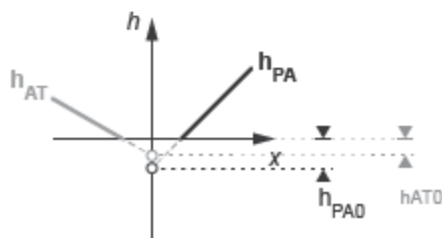
I.



II.



III.



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

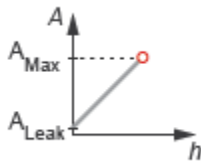
Opening Area Parameterizations

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:

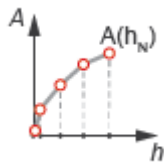
- Maximum area and opening — Specify the maximum opening area and the corresponding orifice opening. The opening area is a linear function of the orifice opening,

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

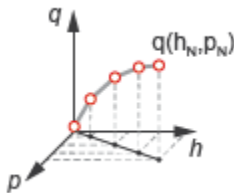
where A is the opening area and h the orifice opening of a given flow path. The subscript `Max` refers to a fully open orifice and the subscript `Leak` to a fully closed orifice—one with internal leakage flow area only. The figure shows a plot of the linear function $A(h)$.



- 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.

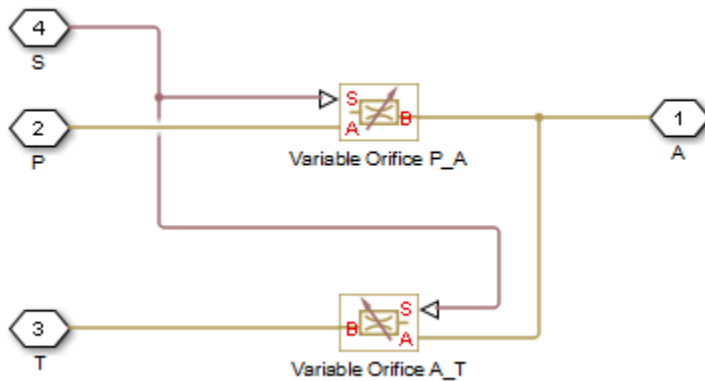
The `Maximum area and opening` and `Area vs. opening` table parameterizations also account for a small leakage area associated with the valve spool-to-bore clearance. The leakage area ensures that portions of the hydraulic network never become isolated when a flow path is closed. Isolated, or “hanging”,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

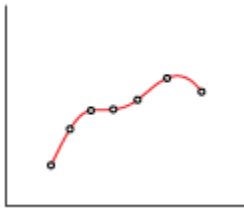
Interpolation method — Method of computing values inside the lookup table range

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.



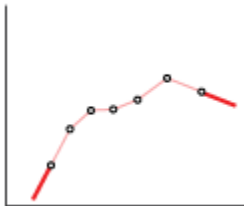
Dependencies

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

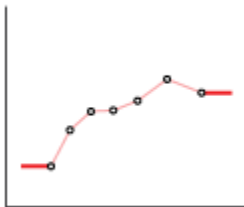
Extrapolation method — Method of computing values outside of the lookup table range

`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 m² (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 1-24.

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 m² (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 1-27.

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 m² (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 1-27.

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 m² (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 1-27.

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] m² (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] m² (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] m² (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 m³/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 m³/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 (TL)

Three-port two-position directional control valve

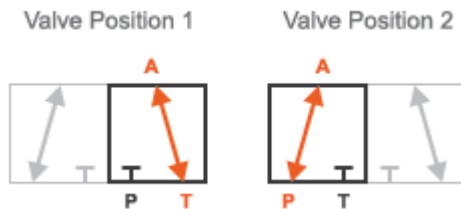


Library

Thermal Liquid/Valves/Directional Control Valves

Description

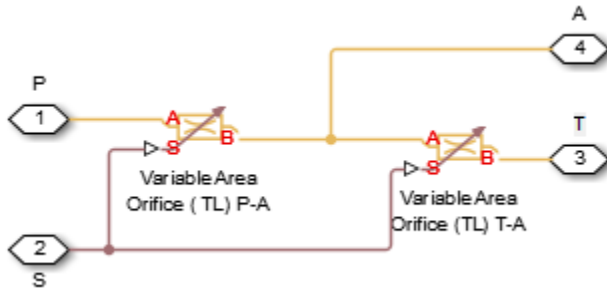
The 3-Way Directional Valve (TL) block represents a three-port two-position directional control valve for the thermal liquid domain. Valve position 1 allows fluid flow between ports A and T. Valve position 2 allows fluid flow between ports A and P. The figure shows the two valve positions.



Physical signal input port S sets the valve position through a control member displacement signal. The valve is in position 1 at a zero control member displacement. It is in position 2 at the maximum allowable displacement specified through the **Maximum valve opening** block parameter.

This block is a composite component based on two Simscape Fluids Variable Area Orifice (TL) blocks. The figure shows the equivalent block diagram for this component. Each block corresponds to a valve flow path. The block names identify the associated flow

paths. For example, the Variable Area Orifice (TL) P-A block corresponds to the flow path between ports P and A.



Composite Component Diagram

Physical signal S controls all Variable Area Orifice (TL) blocks, and therefore all flow paths, simultaneously. The control member position for a given orifice is a function of the **Control member offset** value specified in the block dialog box and the control member displacement specified through port S. The control member position for the P-A orifice is

$$l_{P-A} = l_{P-A,0} + d_S,$$

while for the T-A orifice it is

$$l_{T-A} = l_{A-T,0} - d_S,$$

where:

- l_{P-A} and l_{T-A} are the control member positions for the P-A and T-A orifices.
- $l_{P-A,0}$ and $l_{T-A,0}$ are the initial control member offsets for the P-A and T-A orifices.
- d_S is the control member displacement specified through physical signal input port S.

Parameters

Parameters Tab

Valve area parameterization

Parameterization used to calculate the valve opening area. Select `Linear area-opening relationship` to calculate this area from the valve control member displacement. Select `Tabulated data - Area vs. opening` to specify the valve

open area as a lookup table parameterized in terms of the valve control member displacement. The default setting is `Linear area-opening relationship`.

Maximum valve opening

Control member displacement at which the valve opening area reaches its maximum value. The physical signal input S saturates at this value. This parameter is active only when the **Orifice area parameterization** parameter is set to `Analytical`. The default value is `0.005 m`.

Maximum opening area

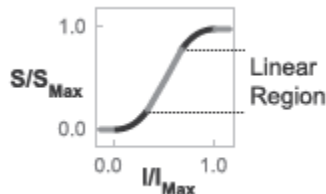
Valve cross-sectional area in the fully open position. This area corresponds to the maximum control member displacement. This parameter is active only when the **Orifice area parameterization** parameter is set to `Analytical`. The default value is `1e-4 m2`.

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-10 m2`.

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

Enter a value of 0 for zero smoothing. Enter a value of 1 for full-curve smoothing. The default value is `0.01`. This parameter is visible only when the **Valve area parameterization** parameter is set to `Linear area-opening relationship`.

Valve opening vector

Vector of control member displacements at which to specify the valve cross-sectional area. This parameter is visible only when the **Orifice area parameterization** parameter is set to `Tabulated`. The default vector is a five-element vector ranging from `-0.002` to `0.015 m`.

Opening area vector

Vector of valve opening areas corresponding to the control member displacements specified through the **Valve opening vector** parameter. This parameter is visible only when the **Orifice area parameterization** parameter is set to *Tabulated*. The default vector is a five-element vector ranging from $1.0e-9$ to 0.00034356 m^2 .

Cross-sectional area at ports A, P, and T

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 .

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.

Valve Opening Offsets Tab**Between ports P and A**

Control member offset from the zero position between ports P and A. The control member position is the sum of the offset specified and the physical signal input S. The offset value does not affect the hard stop locations. The default value is 0 m.

Between ports A and T

Control member offset from the zero position between ports A and T. The control member position is the sum of the offset specified and the physical signal input S. The offset value does not affect the hard stop locations. The default value is 0 m.

Ports

- A — Thermal liquid conserving port representing valve inlet A
- P — Thermal liquid conserving port representing valve inlet P
- T — Thermal liquid conserving port representing valve inlet T
- S — Physical signal input port for the control member displacement

See Also

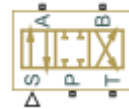
2-Way Directional Valve (TL) | 4-Way Directional Valve (TL) | Check Valve (TL)

Introduced in R2016a

4-Way Directional Valve

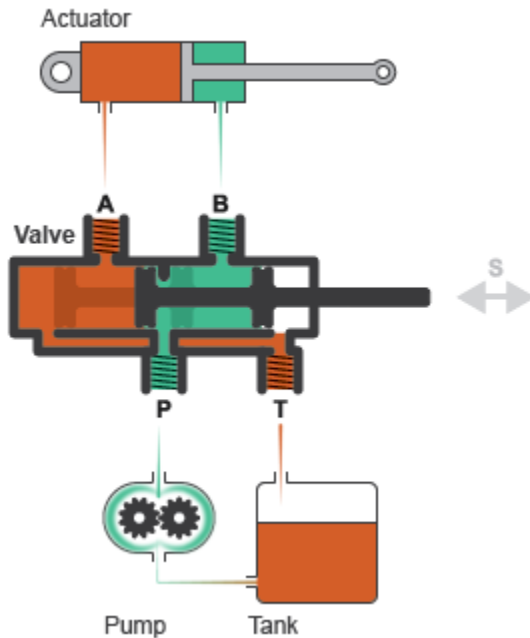
Four-port three-position directional control valve

Library: Hydraulics (Isothermal) / Valves / Directional Valves



Description

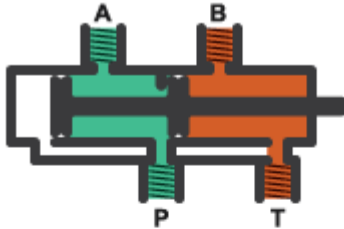
The 4-Way Directional Valve block represents a directional control valve with four ports and three positions, or flow paths. The ports connect to what in a typical model are a hydraulic pump (port **P**), a storage tank (port **T**), and a double-acting actuator (ports **A** and **B**). Fluid can flow 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.



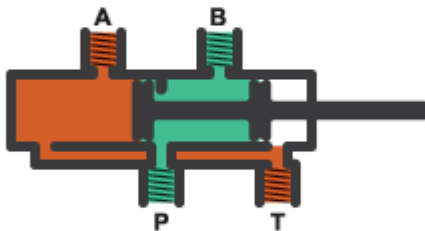
Typical Valve Setup

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.

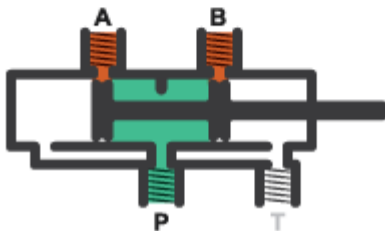
I.



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:

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

$$h_{PB} = h_{PB0} - x$$

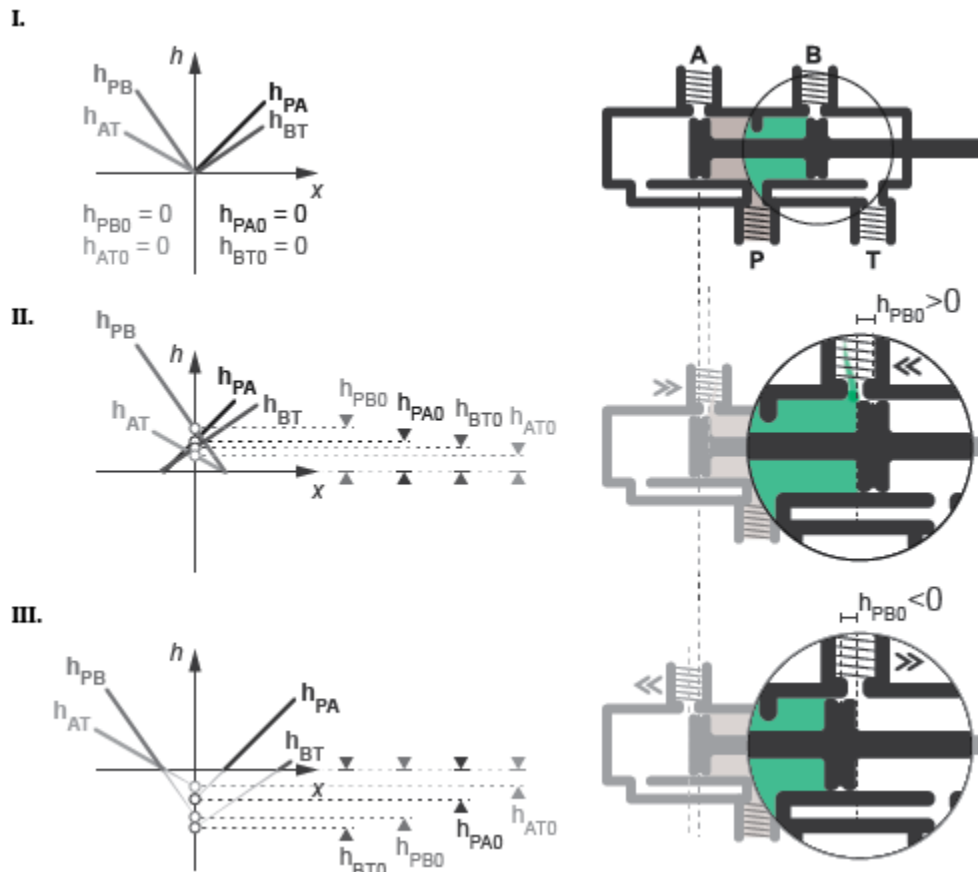
$$h_{AT} = h_{AT0} - x$$

$$h_{BT} = h_{BT0} + x$$

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_{BT0} 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

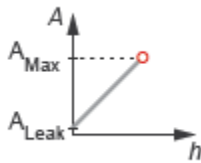
Opening Areas

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:

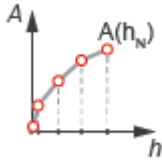
- **Maximum area and opening** — Specify the maximum opening area and the corresponding orifice opening. The opening area is a linear function of the orifice opening,

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

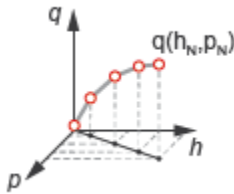
where A is the opening area and h the orifice opening of a given flow path. The subscript *Max* refers to a fully open orifice and the subscript *Leak* to a fully closed orifice—one with internal leakage flow area only. The figure shows a plot of the linear function $A(h)$.



- **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.

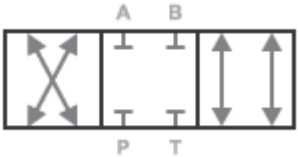
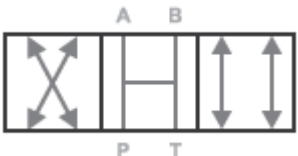
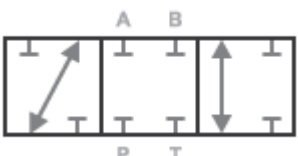
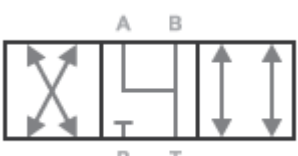
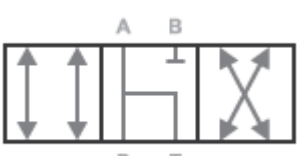
The Maximum area and opening and Area vs. opening table parameterizations also account for a small internal leakage area even in the fully closed state. The leakage area ensures that portions of the hydraulic network do not become isolated when a flow path is closed. Isolated, or “hanging”, network portions affect the computational efficiency of the model and can cause simulation to fail.

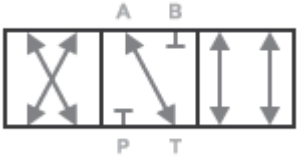
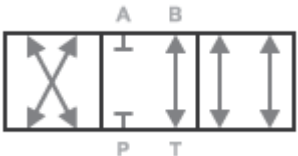
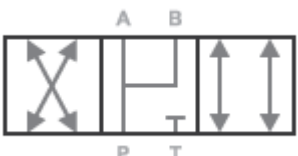
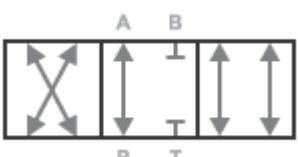
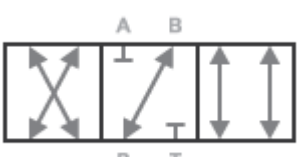
The effects of flow regime and internal leakage are assumed to be reflected in the tabulated flow rate data specified directly in the Pressure-flow characteristic parameterization.

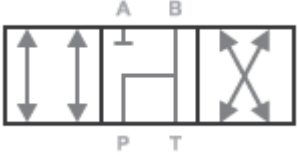
Valve Configurations

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 1-51 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
	<ul style="list-style-type: none"> • Between ports P and A < 0 • Between ports P and B < 0 • Between ports A and T < 0 • Between ports B and T < 0
	<ul style="list-style-type: none"> • Between ports P and A > 0 • Between ports P and B > 0 • Between ports A and T > 0 • Between ports B and T > 0
	<ul style="list-style-type: none"> • 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
	<ul style="list-style-type: none"> • Between ports P and A < 0 • Between ports P and B < 0 • Between ports A and T > 0 • Between ports B and T > 0
	<ul style="list-style-type: none"> • Between ports P and A > 0 • Between ports P and B < 0 • Between ports A and T > 0 • Between ports B and T < 0

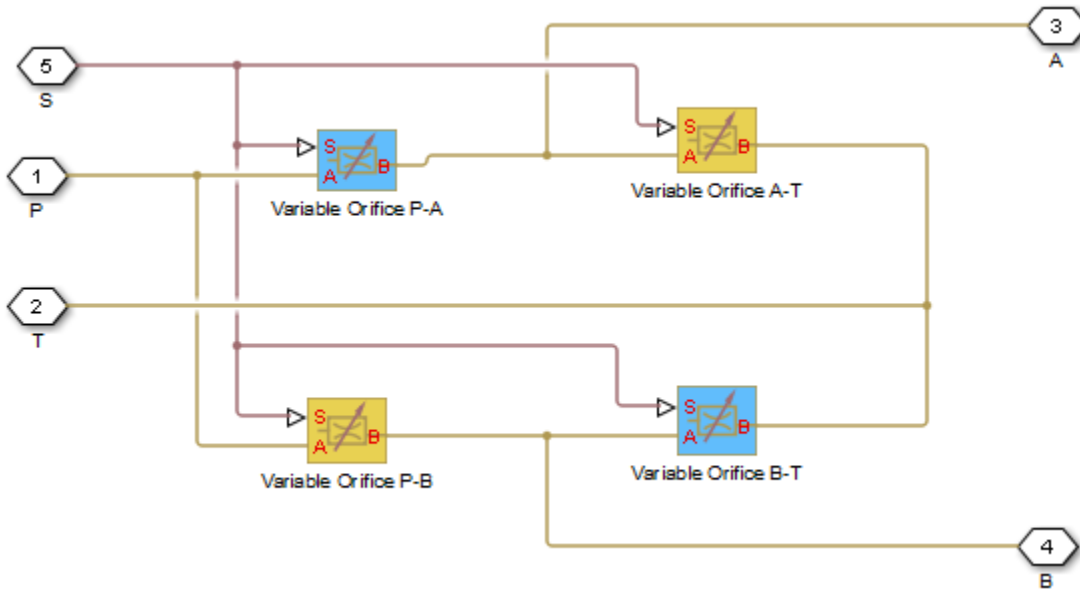
Configuration	Valve opening offsets
	<ul style="list-style-type: none"> • Between ports P and A < 0 • Between ports P and B < 0 • Between ports A and T > 0 • Between ports B and T < 0
	<ul style="list-style-type: none"> • Between ports P and A < 0 • Between ports P and B < 0 • Between ports A and T < 0 • Between ports B and T > 0
	<ul style="list-style-type: none"> • Between ports P and A > 0 • Between ports P and B > 0 • Between ports A and T < 0 • Between ports B and T < 0
	<ul style="list-style-type: none"> • Between ports P and A > 0 • Between ports P and B < 0 • Between ports A and T < 0 • Between ports B and T < 0
	<ul style="list-style-type: none"> • 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
	<ul style="list-style-type: none"> • 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.

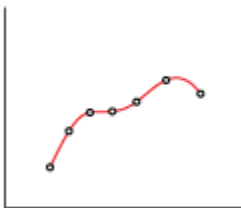
Interpolation method — Method of computing values inside the lookup table range

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.



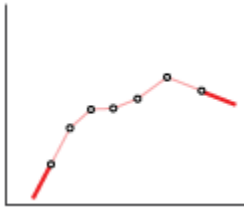
Dependencies

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

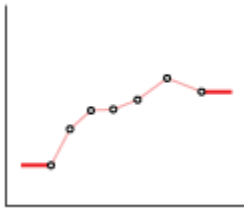
Extrapolation method — Method of computing values outside of the lookup table range

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 1-52.

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 m² (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 1-54.

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 m² (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 1-54.

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 m² (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 1-54.

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 m² (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 1-54.

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 m² (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 1-54.

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] m² (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] m² (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] m² (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]` m² (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] m² (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` m³/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 m³/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 m³/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 m³/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 m³/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

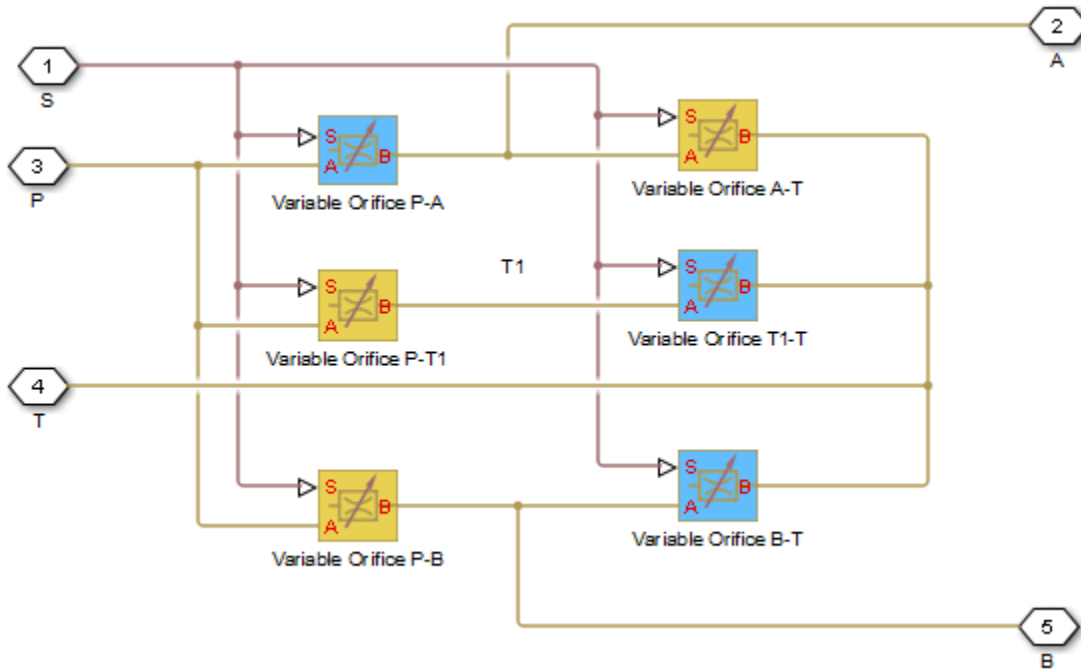


Library

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 1-83
- “Valve Opening Offsets Tab” on page 1-87

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 $5e-5 \text{ m}^2$. 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 $5e-3 \text{ m}$. 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 $1e-12 \text{ m}^2$.

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

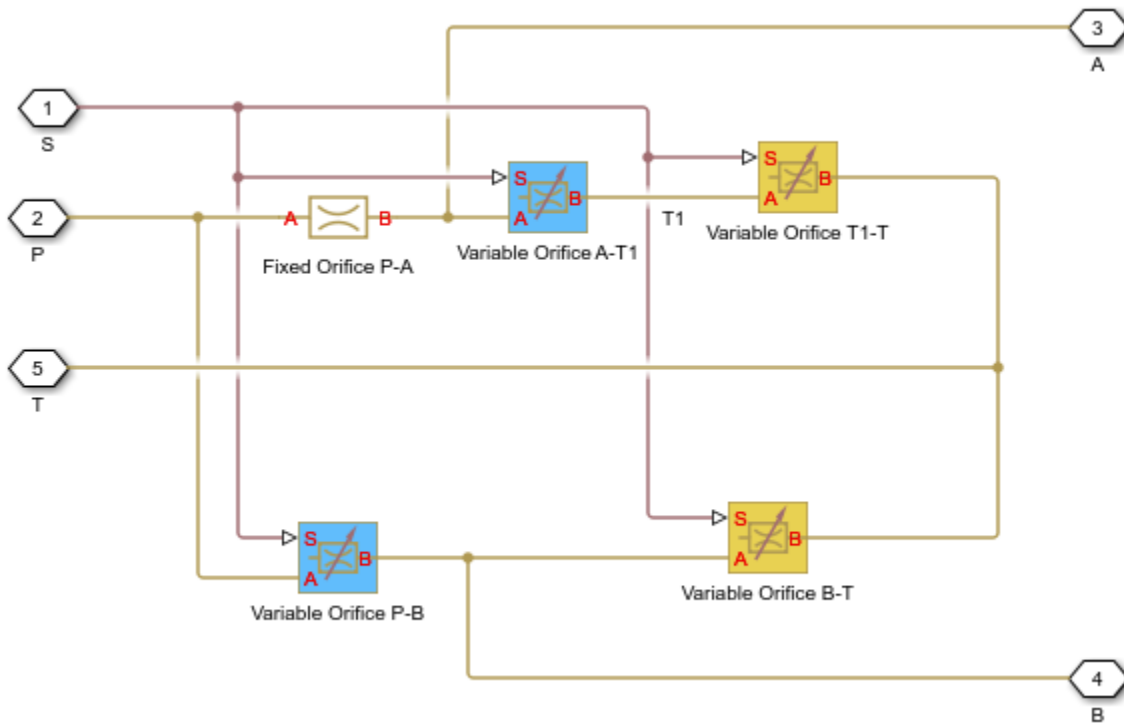


Library

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 1-91
- “Valve Opening Offsets Tab” on page 1-95

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 $5e-5 \text{ m}^2$. 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 $5e-5 \text{ m}^2$. 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 $5e-3 \text{ m}$. 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 m³/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 $1e-12 \text{ m}^2$.

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 2.5×10^{-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 2.5×10^{-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 2.5×10^{-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 2.5×10^{-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

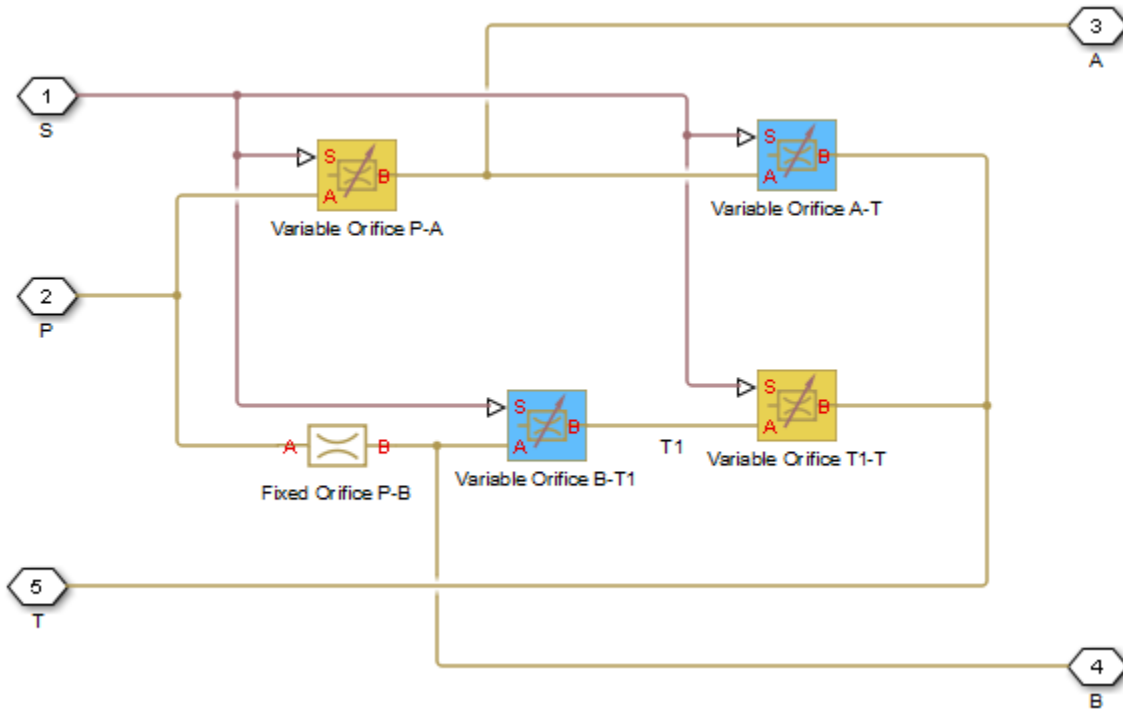


Library

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 1-99
- “Valve Opening Offsets Tab” on page 1-103

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 $5e-5 \text{ m}^2$. 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 $5e-5 \text{ m}^2$. 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 $5e-3 \text{ m}$. 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 m³/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 $1e-12 \text{ m}^2$.

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

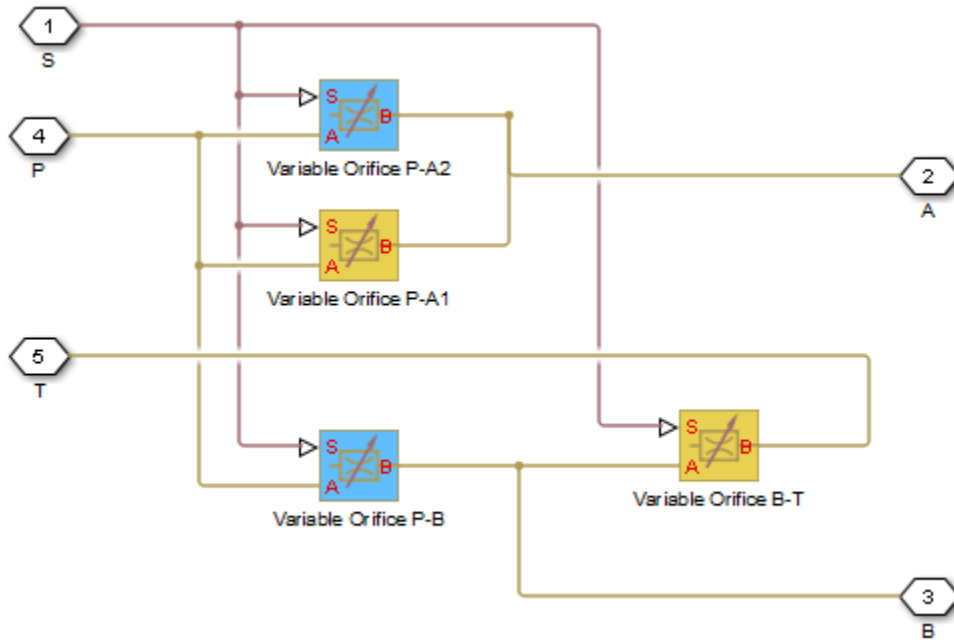


Library

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 1-107
- “Valve Opening Offsets Tab” on page 1-110

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 $5e-5 \text{ m}^2$. 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 $5e-3 \text{ m}$. 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 $1e-12$ m².

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

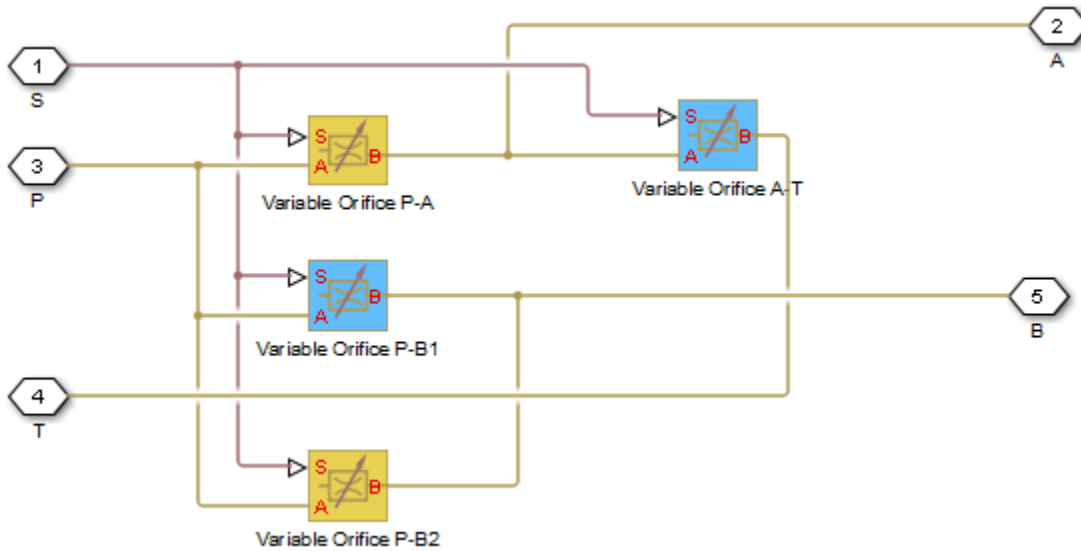


Library

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 1-115
- “Valve Opening Offsets Tab” on page 1-118

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 $5e-5 \text{ m}^2$. 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 $5e-3 \text{ m}$. 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 $1e-12$ m².

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

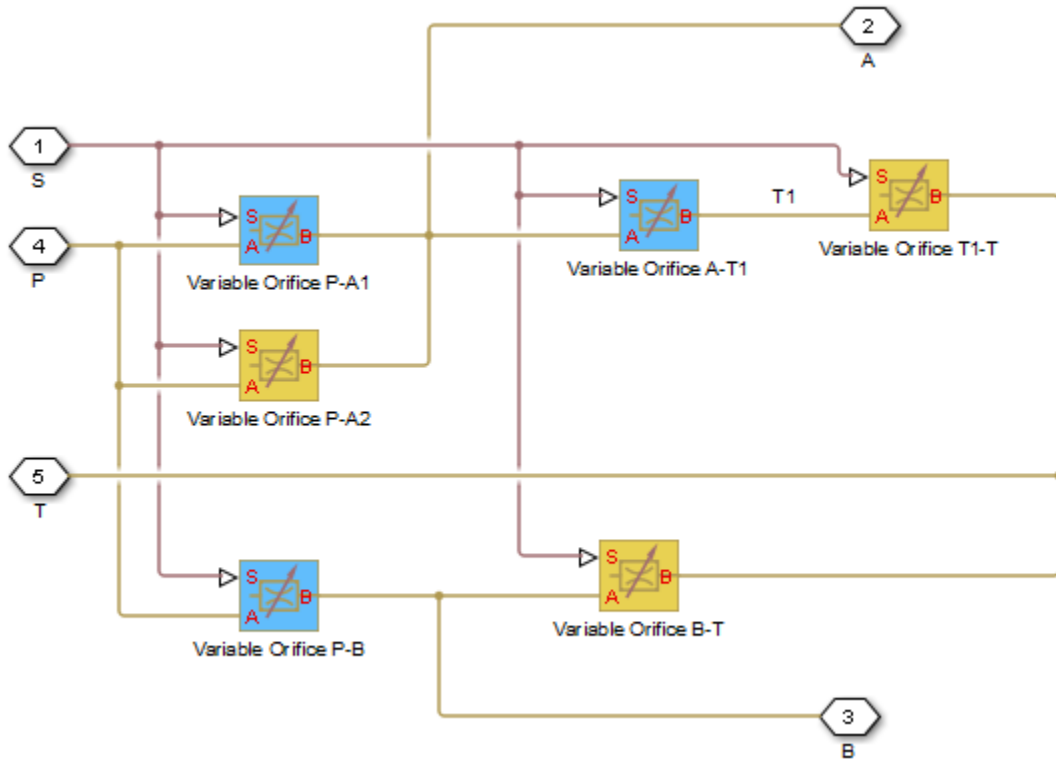


Library

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 1-123
- “Valve Opening Offsets Tab” on page 1-127

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 $5e-5 \text{ m}^2$. 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 $5e-3 \text{ m}$. 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 m³/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 $1e-12 \text{ m}^2$.

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

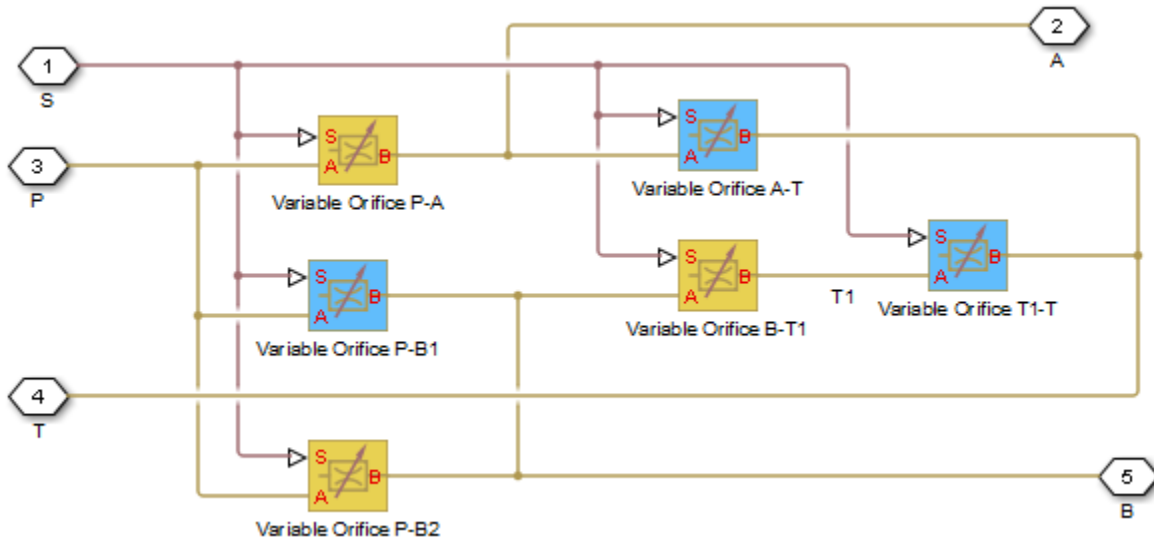


Library

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 1-131
- “Valve Opening Offsets Tab” on page 1-134

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 $5e-5 \text{ m}^2$. 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 $5e-3 \text{ m}$. 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 m³/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 $1e-12 \text{ m}^2$.

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

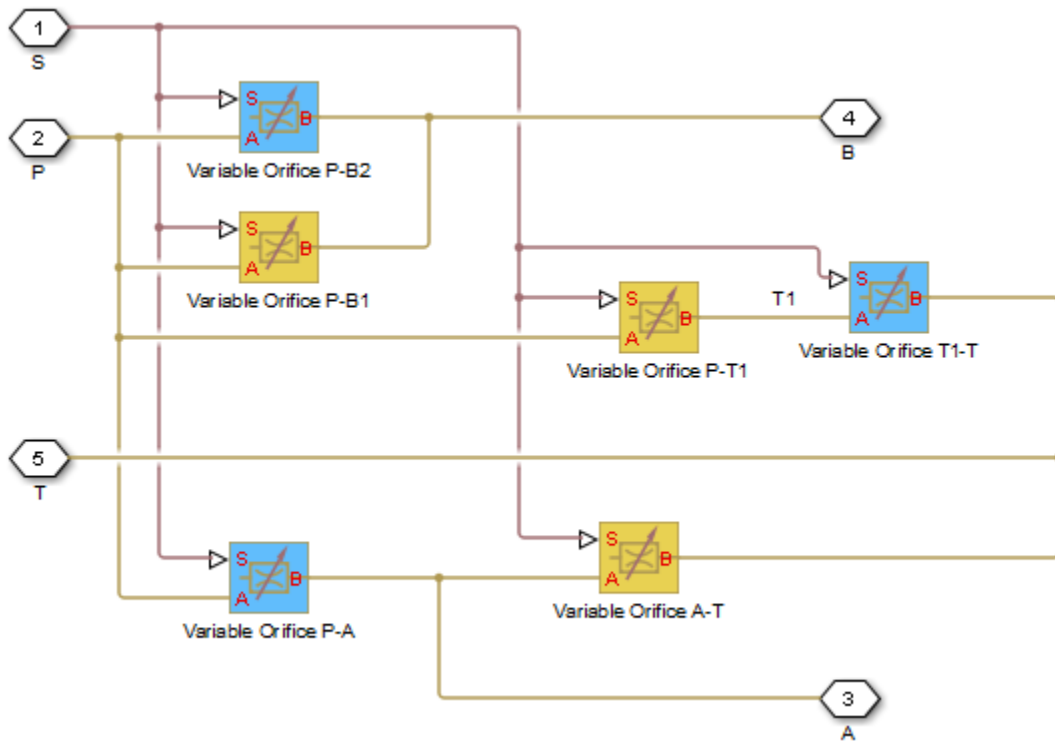


Library

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 1-139
- “Valve Opening Offsets Tab” on page 1-143

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 $5e-5 \text{ m}^2$. 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 $5e-3 \text{ m}$. 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 m³/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 $1e-12 \text{ m}^2$.

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

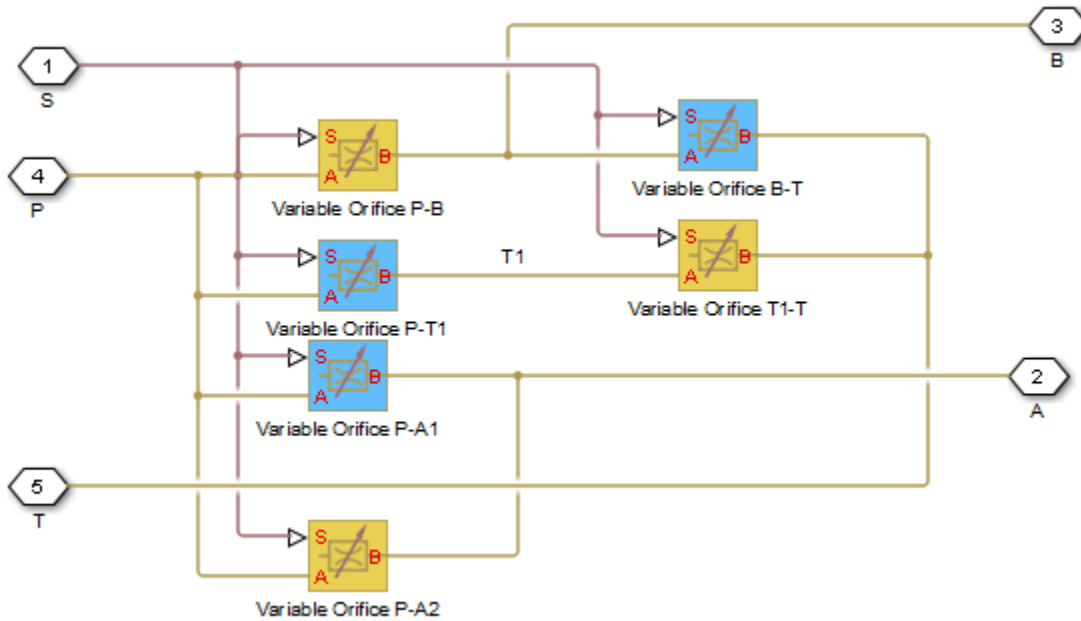


Library

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 1-147
- “Valve Opening Offsets Tab” on page 1-151

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 $5e-5 \text{ m}^2$. 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 $5e-3 \text{ m}$. 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 m³/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 $1e-12 \text{ m}^2$.

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 (TL)

Four-port, three-position directional control valve

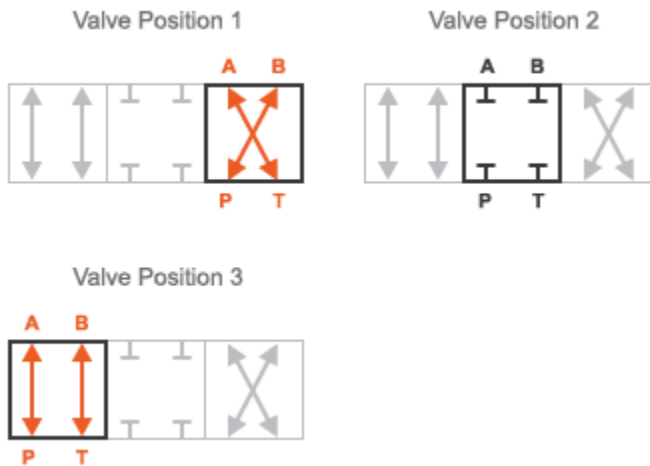


Library

Thermal Liquid/Valves/Directional Control Valves

Description

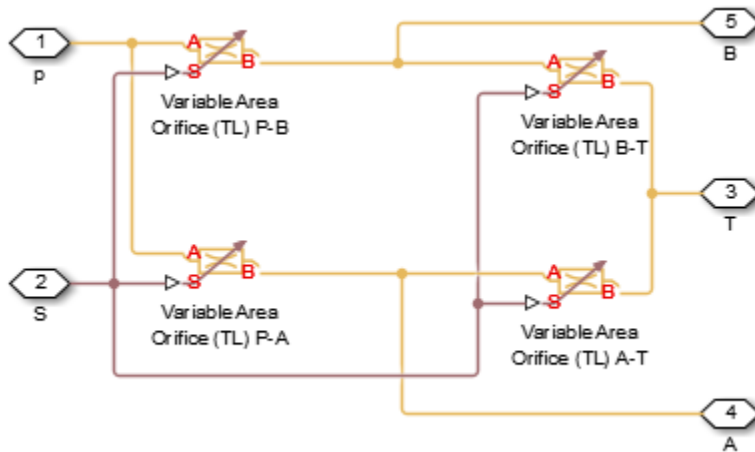
The 4-Way Directional Valve (TL) block represents a generic four-port, three-position directional control valve for the thermal liquid domain. Valve position 1 allows fluid flow between ports A and T and between ports B and P. Valve position 2 allows no fluid flow between any ports. Valve position 3 allows fluid flow between ports A and T and between ports B and P.



Valve Positions

The physical signal input port S sets the valve position through a control member displacement signal. The valve is in position 1 at a zero control member displacement. It is in position 3 at the maximum allowable displacement specified through the **Maximum valve opening** block parameter.

This block is a composite component based on four Simscape Fluids Variable Area Orifice (TL) blocks. The figure shows the equivalent block diagram for this component. Each block corresponds to a valve flow path. The block names identify the associated flow paths. For example, the Variable Area Orifice (TL) P-B block controls the flow path between ports P and B.



Composite Component Diagram

Control Member Offsets

The control member position for a given orifice is a function of the **Control member offset** value specified in the block dialog box and the control member displacement specified through port S. The control member position for the P-B orifice is

$$l_{P-A} = l_{p-A,0} + d_S,$$

while for the P-A orifice it is

$$l_{P-B} = l_{p-B,0} - d_S,$$

for the A-T orifice it is

$$l_{A-T} = l_{A-T,0} - d_S,$$

and for the B-T orifice it is

$$l_{B-T} = l_{B-T,0} - d_S,$$

where:

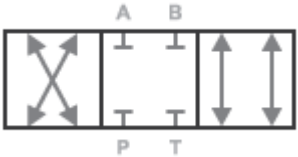
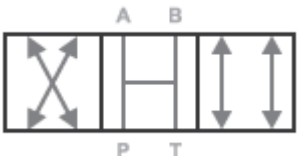
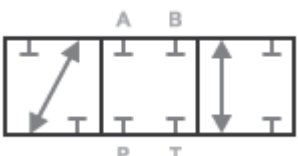
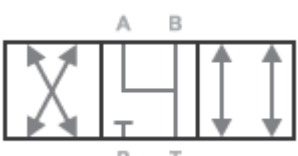
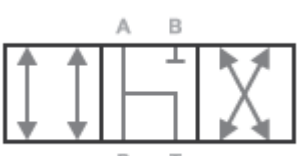
- l_{P-A} , l_{P-B} , l_{A-T} , and l_{B-T} are the control member positions for the P-A, P-B, A-T, and B-T orifices.
- $l_{p-A,0}$, $l_{p-B,0}$, $l_{A-T,0}$, and $l_{B-T,0}$ are the initial control member offsets for the P-A, P-B, A-T, and B-T orifices.

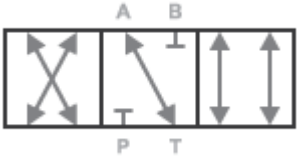
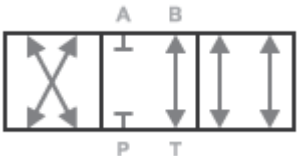
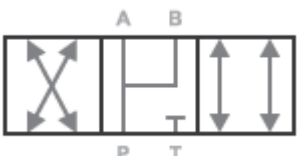
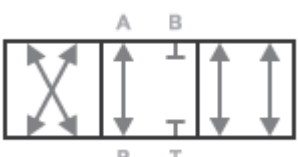
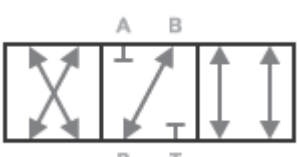
- d_S is the control member displacement specified through physical signal input port S.

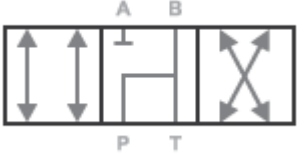
Valve Configurations

The opening offsets are by default zero. This configuration corresponds to a valve with all flow paths closed in the neutral position. 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
	<ul style="list-style-type: none"> • Between ports P and A < 0 • Between ports P and B < 0 • Between ports A and T < 0 • Between ports B and T < 0
	<ul style="list-style-type: none"> • Between ports P and A > 0 • Between ports P and B > 0 • Between ports A and T > 0 • Between ports B and T > 0
	<ul style="list-style-type: none"> • 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
	<ul style="list-style-type: none"> • Between ports P and A < 0 • Between ports P and B < 0 • Between ports A and T > 0 • Between ports B and T > 0
	<ul style="list-style-type: none"> • 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
	<ul style="list-style-type: none"> • Between ports P and A < 0 • Between ports P and B < 0 • Between ports A and T > 0 • Between ports B and T < 0
	<ul style="list-style-type: none"> • Between ports P and A < 0 • Between ports P and B < 0 • Between ports A and T < 0 • Between ports B and T > 0
	<ul style="list-style-type: none"> • Between ports P and A > 0 • Between ports P and B > 0 • Between ports A and T < 0 • Between ports B and T < 0
	<ul style="list-style-type: none"> • Between ports P and A > 0 • Between ports P and B < 0 • Between ports A and T < 0 • Between ports B and T < 0
	<ul style="list-style-type: none"> • 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
	<ul style="list-style-type: none"> • Between ports P and A < 0 • Between ports P and B > 0 • Between ports A and T < 0 • Between ports B and T > 0

Parameters

Parameters Tab

Valve area parameterization

Parameterization used to calculate the valve opening area. Select **Linear area-opening relationship** to calculate this area from the valve control member displacement. Select **Tabulated data – Area vs. opening** to specify the valve open area as a lookup table parameterized in terms of the valve control member displacement. The default setting is **Linear area-opening relationship**.

Maximum valve opening

Control member displacement at which the valve opening area reaches its maximum value. The physical signal input **S** saturates at this value. This parameter is active only when the **Orifice area parameterization** parameter is set to **Analytical**. The default value is 0.005 m.

Maximum opening area

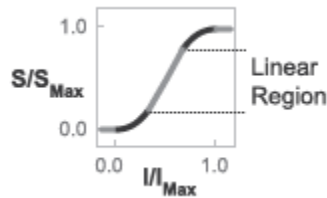
Valve cross-sectional area in the fully open position. This area corresponds to the maximum control member displacement. This parameter is active only when the **Orifice area parameterization** parameter is set to **Analytical**. The default value is $1e-4 \text{ m}^2$.

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 $1e-10 \text{ m}^2$.

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

Enter a value of 0 for zero smoothing. Enter a value of 1 for full-curve smoothing. The default value is 0.01. This parameter is visible only when the **Valve area parameterization** parameter is set to Linear area-opening relationship.

Valve opening vector

Vector of control member displacements at which to specify the valve cross-sectional area. This parameter is visible only when the **Orifice area parameterization** parameter is set to Tabulated. The default vector is a five-element vector ranging from -0.002 to 0.015 m.

Opening area vector

Vector of valve opening areas corresponding to the control member displacements specified through the **Valve opening vector** parameter. This parameter is visible only when the **Orifice area parameterization** parameter is set to Tabulated. The default vector is a five-element vector ranging from $1.0e-9$ to 0.00034356 m^2 .

Cross-sectional area at ports A, B, P, and T

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.

Valve Opening Offsets Tab**Between ports P and A**

Control member offset from the zero position between ports P and A. The control member position is the sum of the offset specified and the physical signal input S. The default value is 0 m.

Between ports B and T

Control member offset from the zero position between ports B and T. The control member position is the sum of the offset specified and the physical signal input S. The default value is 0 m.

Between ports P and B

Control member offset from the zero position between ports P and B. The control member position is the sum of the offset specified and the physical signal input S. The default value is 0 m.

Between ports A and T

Control member offset from the zero position between ports A and T. The control member position is the sum of the offset specified and the physical signal input S. The default value is 0 m.

Ports

- A — Thermal liquid conserving port representing valve inlet A
- B — Thermal liquid conserving port representing valve inlet B
- P — Thermal liquid conserving port representing valve inlet P
- T — Thermal liquid conserving port representing valve inlet T
- S — Physical signal input port for the control member displacement

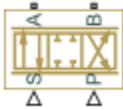
See Also

2-Way Directional Valve (TL) | 3-Way Directional Valve (TL) | Check Valve (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:

$$q = x C_D b \sqrt{\frac{1}{\rho} (p_S - \text{abs}(p_A - p_B)) \text{sign}(p_A - p_B)}$$

where

q	Flow rate
x	Valve displacement, $-x_{max} \leq x \leq x_{max}$
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 $5e-5 \text{ m}^2$.

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



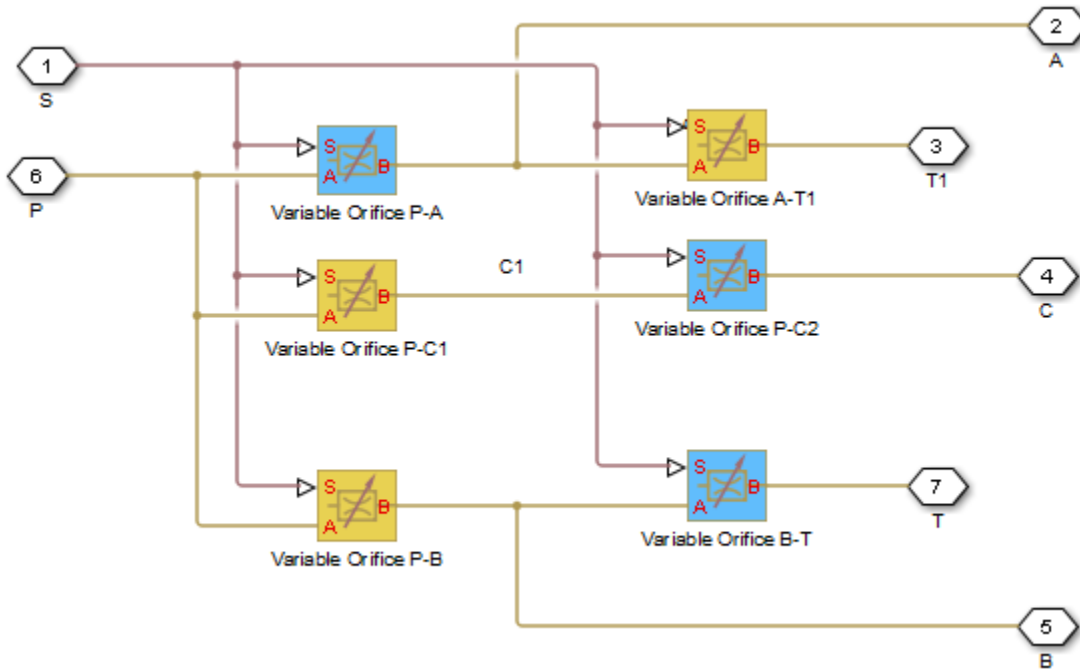
Library

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 1-168
- “Valve Opening Offsets Tab” on page 1-172

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 $5e-5 \text{ m}^2$. 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 $5e-3 \text{ m}$. 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 $1e-12 \text{ m}^2$.

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

Angle Sensor

Ideal angle sensor with measuring range from 0 to 360 degrees



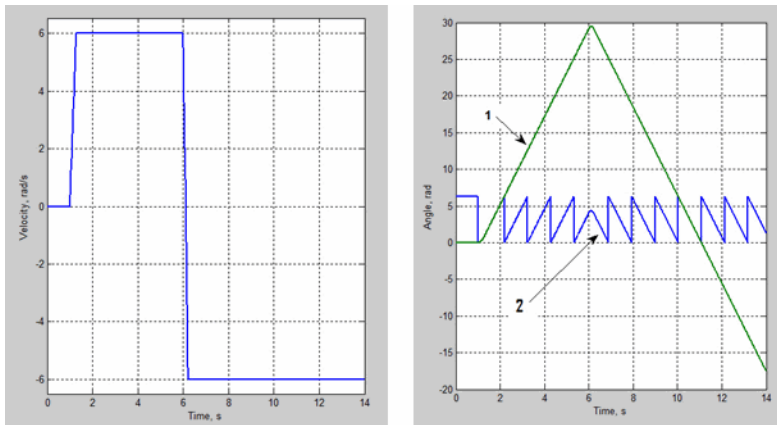
Library

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.

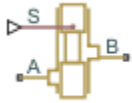
See Also

Ideal Rotational Motion Sensor

Introduced in R2011a

Annular Orifice

Hydraulic variable orifice created by circular tube and round insert

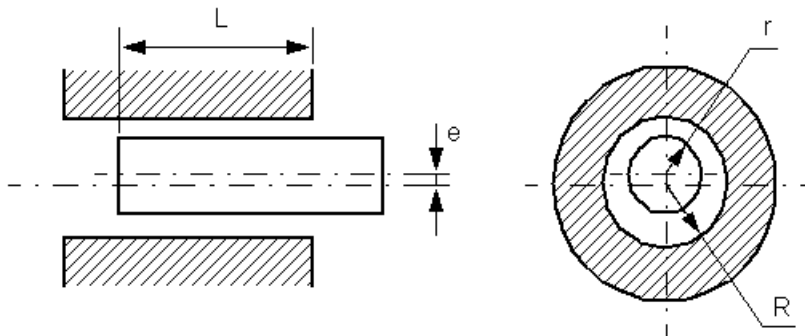


Library

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 1-180):

$$q = \frac{\pi R(R-r)^3}{6\nu\rho L} \left(1 + \frac{3}{2}\varepsilon^2 \right) p$$

$$\varepsilon = \frac{e}{R-r}$$

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

$p = p_A - p_B$. 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

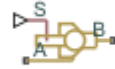
Constant Area Hydraulic Orifice | Fixed Orifice | Orifice with Variable Area Round Holes | Orifice with Variable Area Slot | Variable Area Hydraulic Orifice | Variable Orifice

Introduced in R2006b

Ball Valve

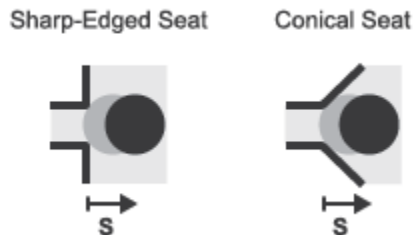
Valve with a sliding ball control mechanism

Library: 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:

$$A(h) = \pi r_O \left(1 - \left[\frac{r_B}{d_{OB}} \right]^2 \right) d_{OB}(h),$$

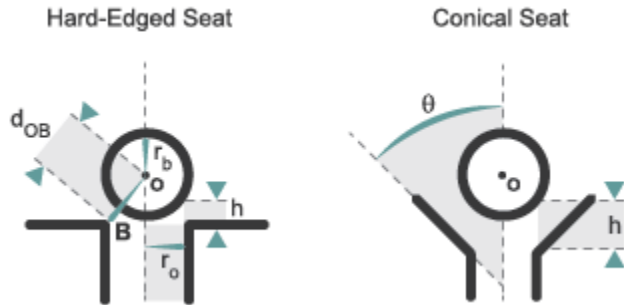
where:

- A is the opening area at a given valve lift value.
- r_B is the ball radius.
- $d_{OB}(h)$ 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:

$$A(h) = \pi h \cos(\theta) \sin(\theta) (d_{OB}(h) + h \sin(\theta)),$$

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:

$$q = C_D A(h) \sqrt{\frac{2}{\rho} \frac{\Delta p}{(\Delta p)^2 + p_{Cr}^2}}^{1/4},$$

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:

$$p_{Cr} = \frac{\rho}{2} \left(\frac{Re_{Cr} v}{C_D D_H} \right)^2,$$

where:

- Re_{Cr} is the critical Reynolds number.

- ν (nu) is the hydraulic fluid dynamic viscosity.
- D_H is the orifice hydraulic diameter:

$$D_H = \begin{cases} D_H^{Min}, & \text{if } h \leq 0 \\ D_H^{Max} + D_H^{Min}, & \text{if } h \geq h_{Max}, \\ \frac{4A}{l} + D_H^{Min}, & \text{otherwise} \end{cases}$$

in which:

- D_H^{Min} is the minimum hydraulic diameter, corresponding to the smallest attainable flow area, the leakage flow area.
- D_H^{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 m² (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 m³/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 (TL)

Flow control valve actuated by longitudinal motion of ball element

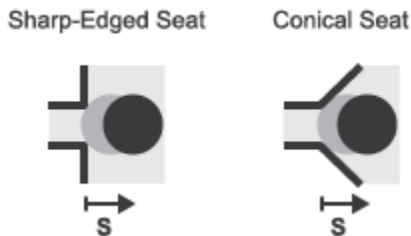


Library

Thermal Liquid/Valves/Flow Control Valves

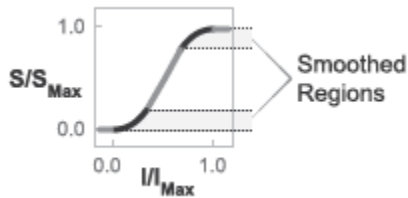
Description

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

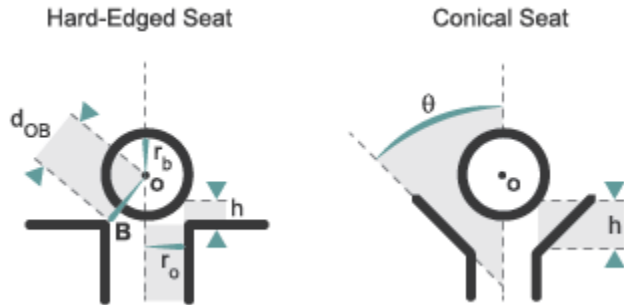
Valve Opening Area

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

$$\bar{h} = \frac{h - (h_{max} - \Delta h_{smooth})}{h_{max} - (h_{max} - \Delta h_{smooth})}$$

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.
- 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

$$S_R = \begin{cases} S_{Leak}, & h \leq 0 \\ S_{Leak} (1 - \lambda_L) + (A + S_{Leak}) \lambda_L, & h < \Delta h_{smooth} \\ A + S_{Leak}, & h \leq h_{Max} - \Delta h_{smooth}, \\ (A + S_{Leak}) (1 - \lambda_R) + (S_{Leak} + S_{Max}) \lambda_R, & h < h_{Max} \\ S_{Leak} + S_{Max}, & h \geq h_{Max} \end{cases}$$

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

$$p_A - p_B = \frac{\dot{m} \sqrt{\dot{m}^2 + \dot{m}_{cr}^2}}{2\rho_{Avg} C_d^2 S^2} \left[1 - \left(\frac{S_R}{S} \right)^2 \right] PR_{Loss},$$

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:

$$\dot{m}_{cr} = Re_{cr} \mu_{Avg} \sqrt{\frac{\pi}{4} S_R}.$$

- 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:

$$PR_{Loss} = \frac{\sqrt{1 - (S_R/S)^2 (1 - C_d^2)} - C_d (S_R/S)}{\sqrt{1 - (S_R/S)^2 (1 - C_d^2)} + C_d (S_R/S)}.$$

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 $1e-12$ m².

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 m².

Characteristic longitudinal length

Distance traversed by the fluid between the valve inlets. The default value is 0.1 m².

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

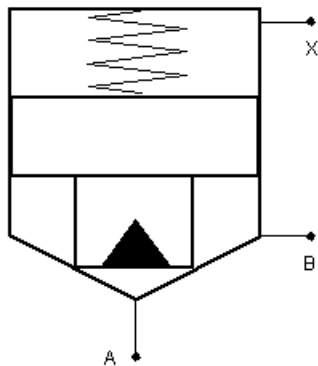


Library

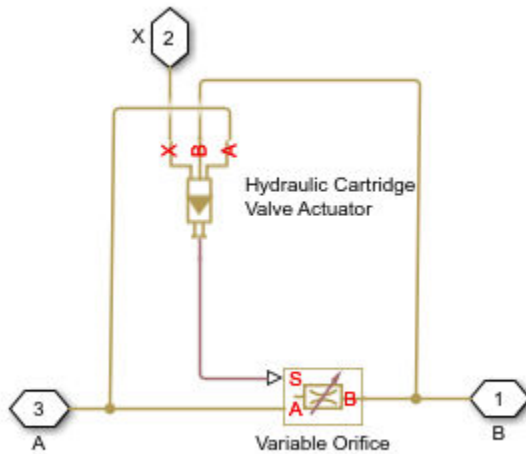
Directional Valves

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 $2e-4 \text{ m}^2$.

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 $1.4e4 \text{ N/m}$.

Poppet stroke

Maximum poppet stroke. The parameter value must be greater than or equal to zero. The default value is $5e-3 \text{ m}$. 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 $5e-5 \text{ m}^2$. 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 m³/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 $1e-12 \text{ m}^2$.

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

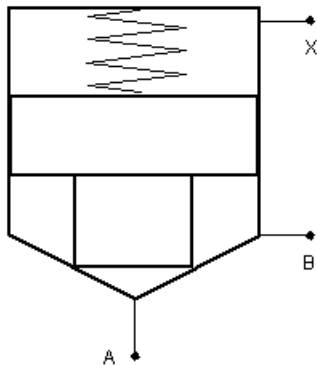


Library

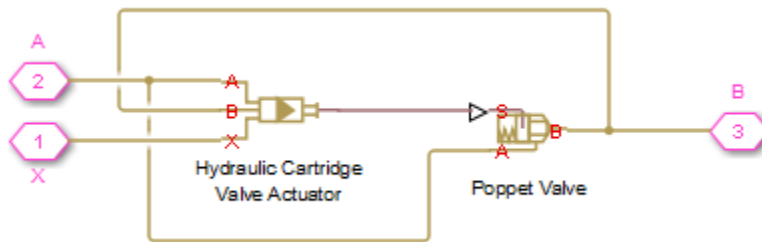
Directional Valves

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 $2e-4 \text{ m}^2$.

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 $1e-12$ m².

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:

$$F = \frac{\pi\rho\omega^2}{4} \left(r_o^4 - r_i^4 - 2r_p^2 (r_o^2 - r_i^2) \right)$$

where

F	Centrifugal force
r_o	Piston outer radius
r_i	Piston inner radius
r_p	Fluid entry radius. For design purposes, $r_p < \sqrt{(r_o^2 + r_i^2)}/2$
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/m³.

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

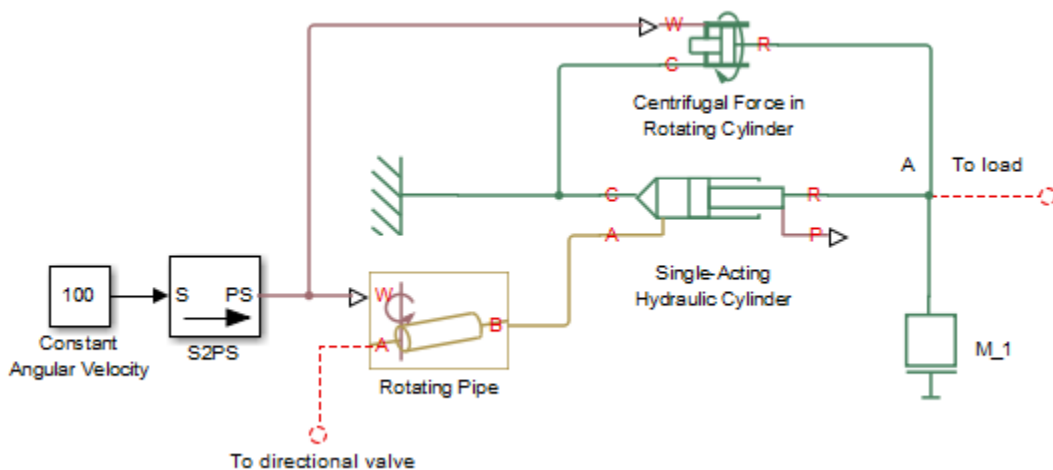
When your model is in Restricted editing mode, you cannot modify the following parameter:

- **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 1-213
- “Parameterizing the Pump by Pressure Differential and Brake Power Versus Pump Delivery” on page 1-217
- “Parameterizing the Pump by Pressure Differential and Brake Power Versus Pump Delivery at Different Angular Velocities” on page 1-218

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 1-225, 2 on page 1-225], which for a given pump, angular velocity, and fluid can be represented as the following:

$$p_{ref} = k \cdot p_E - p_{HL} - p_D$$

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 1-225, 2 on page 1-225] 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

$$p_E = \rho_{ref} (c_0 - c_1 \cdot q_{ref})$$

where

ρ_{ref}	Fluid density
c_0, c_1	Approximating coefficients. They can be determined either analytically from the Euler equation [1 on page 1-225, 2 on page 1-225] 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

$$p_D = \rho_{ref} \cdot c_3 (q_D - q_{ref})^2$$

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:

$$p_{ref} = \rho_{ref} \left(k(c_0 - c_1 q_{ref}) - c_2 q_{ref}^2 - c_3 (q_D - q_{ref})^2 \right)$$

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

$$q_{ref} = q \frac{\omega_{ref}}{\omega}$$

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” on page 1-215 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” on page 1-215 describes pump characteristic for $\omega > 0$ and $q \geq 0$. Outside this range, the characteristic is approximated with the following relationships:

$$p = \begin{cases} -k_{leak} \cdot q & \text{for } \omega \leq 0 \\ p_{max} - k_{leak} \cdot q & \text{for } \omega > 0, q < 0 \\ -k_{leak} \cdot (q - q_{max}) & \text{for } \omega > 0, q > q_{max} \end{cases}$$

$$q_{max} = \frac{-b + \sqrt{b^2 + 4ac}}{2a}$$

$$a = (c_2 + c_3) \cdot \alpha^2$$

$$b = (k \cdot c_1 - 2c_3 \cdot q_D) \cdot \alpha$$

$$c = k \cdot c_0 - c_3 \cdot q_D^2$$

$$\alpha = \frac{\omega}{\omega_{ref}}$$

$$q_{max} = \rho \frac{1}{\alpha^2} \left(k \cdot c_0 - c_3 \cdot q_D^2 \right)$$

where

k_{leak}	Leakage resistance coefficient
q_{max}	Maximum pump delivery at given angular velocity. The delivery is determined from “Equation 1-2” on page 1-215 at $p = 0$.
p_{max}	Maximum pump pressure at given angular velocity. The pressure is determined from “Equation 1-2” on page 1-215 at $q = 0$.
k	Correction factor, as described in “Equation 1-1” on page 1-213.

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:

$$N_{fr} = (T_0 + k_p \cdot p) \cdot \omega$$

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/m³). 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 $1.097e7 \text{ Pa}\cdot\text{s}^2/(\text{kg}\cdot\text{m}^3)$. 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 $2.136e5 \text{ Pa}\cdot\text{s}^2/(\text{kg}\cdot\text{m}^3)$. 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 $1.77e3 \text{ rpm}$. 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 $920 \text{ kg}/\text{m}^3$.

Leak resistance

Leakage resistance coefficient (see “Equation 1-4” on page 1-215). The default value is $1e+8 \text{ Pa}/(\text{m}^3/\text{s})$. 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 $0.1 \text{ N}\cdot\text{m}$. 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 $1\text{e-}6 \text{ N}\cdot\text{m}/\text{Pa}$. 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

Check Valve

Hydraulic valve that allows flow in one direction only

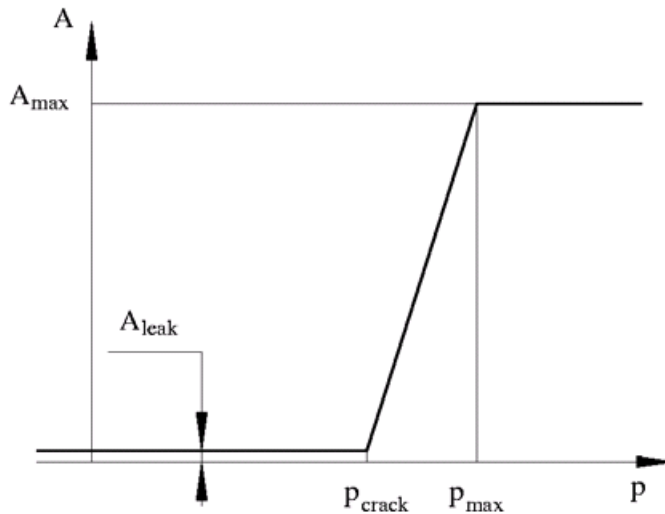


Library

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 $p = p_A - p_B$.



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:

$$q = C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

$$A(p) = \begin{cases} A_{leak} & \text{for } p \leq p_{crack} \\ A_{leak} + k \cdot (p - p_{crack}) & \text{for } p_{crack} < p < p_{max} \\ A_{max} & \text{for } p \geq p_{max} \end{cases}$$

$$k = \frac{A_{max} - A_{leak}}{p_{max} - p_{crack}}$$

$$p = p_A - p_B$$

$$p_{cr} = \frac{\rho}{2} \left(\frac{Re_{cr} \cdot \nu}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

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

$$P = P_A - P_B.$$

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 $1e-4 \text{ m}^2$.

Cracking pressure

Pressure level at which the orifice of the valve starts to open. The default value is $3e4 \text{ Pa}$.

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 $1.2e5 \text{ Pa}$.

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 $1e-12 \text{ m}^2$.

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 $1e-12 \text{ m}^2$.

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 (TL)

One-way directional valve with pressure-actuated control element



Library

Thermal Liquid/Valves/Directional Control Valves

Description

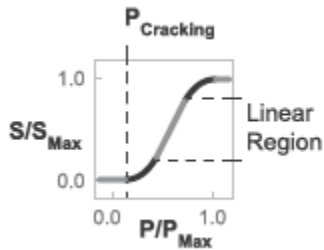
The Check Valve (TL) block represents a one-way directional valve for the thermal liquid domain. The valve allows flow from port A to port B only. A minimum pressure, known as the cracking pressure, is required at port A in order to force the valve open.



Check Valve Positions

The block accepts the cracking pressure parameter as a gauge pressure at port A or as a pressure differential between ports A and B. The cracking pressure marks only the point at which the valve begins to open. The valve opens gradually with rising pressure until it is fully open.

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 cracking and maximum opening pressures. 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

$$S_{Linear} = \left(\frac{S_{Max} - S_{Leak}}{P_{Max} - P_{Crack}} \right) (P_{Control} - P_{Crack}) + S_{Leak},$$

where:

- S_{Linear} is the linear valve opening area.
- 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.
- p_{Max} is the pressure at which the valve reaches the fully open position. This parameter depends on the **Pressure control specification** setting in the block dialog box—
Pressure at port A or Pressure differential:

$$p_{Max} = \begin{cases} p_{Max, Gauge} + p_{Atm}, & \text{'Pressure at port A'} \\ p_{Max, Diff}, & \text{'Pressure differential'} \end{cases}$$

In the equation, $p_{Max, Gauge}$ is the gauge pressure at which the valve reaches the fully open position, p_{Atm} is atmospheric pressure, and $p_{Max, Diff}$ is pressure difference between ports A and B at which the valve reaches the fully open position.

- p_{Crack} is the pressure at which the valve begins to open. This parameter depends on the **Pressure control specification** setting in the block dialog box—Pressure at port A or Pressure differential:

$$p_{Crack} = \begin{cases} p_{Crack, Gauge} + p_{Atm}, & \text{'Pressure at port A'} \\ p_{Crack, Diff}, & \text{'Pressure differential'} \end{cases}$$

In the equation, $p_{Crack, Gauge}$ is the gauge pressure at which the valve first begins to open, p_{Atm} is atmospheric pressure, and $p_{Crack, Diff}$ is the pressure difference between ports A and B at which the valve first begins to open.

- $p_{Control}$ is the valve control pressure. This parameter depends on the **Pressure control specification** setting in the block dialog box—Pressure at port A or Pressure differential:

$$p_{Control} = \begin{cases} p_A, & \text{'Pressure at port A'} \\ p_A - p_B, & \text{'Pressure differential'} \end{cases}$$

In the equation, p_A is the pressure at port A and p_B the pressure at port B.

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

$$\lambda_L = 3\bar{p}_L^{-2} - 2\bar{p}_L^3$$

and

$$\lambda_R = 3\bar{p}_R^{-2} - 2\bar{p}_R^{-3}$$

where:

$$\bar{p}_L = \frac{p_{Control} - p_{Crack}}{\Delta p_{smooth}}$$

and

$$\bar{p}_R = \frac{p_{Control} - (p_{Max} - \Delta p_{smooth})}{\Delta p_{smooth}}.$$

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.
- Δp_{smooth} is the temperature smoothing region:

$$\Delta p_{smooth} = f_{smooth} \frac{P_{Max} - P_{Set}}{2},$$

where f_{smooth} is a smoothing factor between 0 and 1.

The smoothed valve opening area is given by the piecewise conditional expression

$$S_R = \begin{cases} S_{Leak}, & p_{control} \leq p_{crack} \\ S_{Leak} (1 - \lambda_L) + S_{Linear} \lambda_L, & p_{control} < p_{crack} + \Delta p_{smooth} \\ S_{Linear}, & p_{control} < P_{Max} - \Delta p_{smooth} \\ S_{Linear} (1 - \lambda_R) + S_{Max} \lambda_R, & p_{control} < P_{Max} \\ S_{Max}, & p_{control} \geq P_{Max} \end{cases}$$

where:

- S_R is the smoothed valve opening area.

Parameters

Parameters Tab

Pressure control specification

Specification method for the valve set pressure data. Options include Pressure at port A and Pressure differential.

Cracking pressure (gauge)

Minimum gauge pressure at port A required to force the valve partially open. The valve opening continues to expand as the fluid pressure rises above the cracking pressure. 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.

Cracking pressure differential

Minimum pressure differential, measured from port A to port B, required to force the valve partially open. The valve opening continues to expand as the pressure differential rises above this value. This parameter is active only when the **Pressure control specification** parameter is set to Pressure differential. The default value is 0.01 MPa.

Maximum opening pressure (gauge)

Gauge pressure at port A required to fully open the valve. The valve opening area stays constant above this pressure. This parameter is active only when the **Pressure control specification** parameter is set to Pressure at port A. The default value is 0.2 MPa.

Maximum opening pressure differential

Pressure differential from port A to port B required to fully open the valve. The valve opening area stays constant above this pressure differential. This parameter is active only when the **Pressure control specification** parameter is set to Pressure differential. The default value is 0.02 MPa.

Maximum opening area

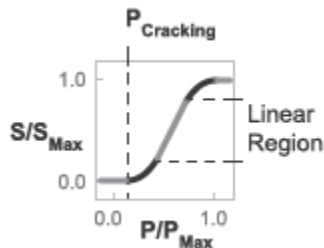
Valve cross-sectional area in the fully open position. This area corresponds to the maximum control member displacement. The default value is $1e-4 \text{ m}^2$.

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 $1e-10 \text{ m}^2$.

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**

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

Approximate length of the valve. This parameter provides a measure of the longitudinal scale of the valve. The default value is 0.1 m².

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

2-Way Directional Valve (TL) | 3-Way Directional Valve (TL) | 4-Way Directional Valve (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:

$$q = \sqrt{\frac{1}{K}} \cdot A_p \sqrt{\frac{2}{\rho}} \cdot \frac{P_{loss}}{(P_{loss}^2 + P_{cr}^2)^{1/4}}$$

$$p_{cr} = K \frac{\rho}{2} \left(\frac{\text{Re}_{cr} \cdot v}{d} \right)^2$$

The Critical Reynolds number is set to 15.

The pressure at the tank inlet is computed with the following equations:

$$p = p_{elev} - p_{loss} + p_{pr}$$

$$p_{elev} = \rho \cdot g \cdot H$$

$$A_p = \frac{\pi \cdot d^2}{4}$$

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

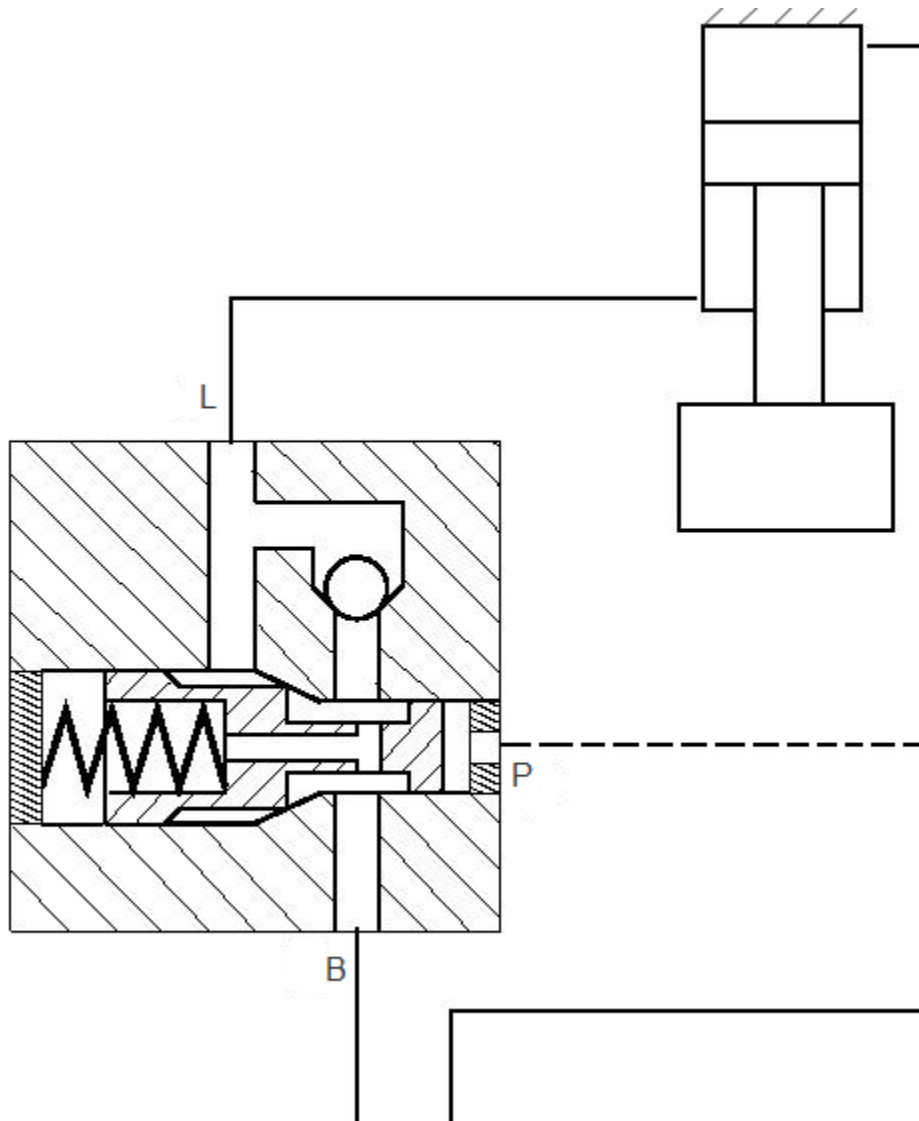


Library

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

$$F_0 + c \cdot x = p_{pilot} \cdot A_{pilot} + p_{load} \cdot A_{load} - p_{back} \cdot A_{back}$$

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” on page 1-245 takes the form

$$p_{set} + c_p \cdot x = p_{pilot} \cdot k_{pilot} + p_{load} - p_{back} \cdot k_{back}$$

$$p_{set} = F_0 / A_{load}$$

$$c_p = c / A_{load}$$

$$k_{pilot} = A_{pilot} / A_{load}$$

$$k_{back} = A_{back} / A_{load}$$

where

p_{set}	Valve pressure setting
c_p	Spring pressure stiffness (Pa/m)
x	Valve opening
k_{pilot}	Pilot ratio
k_{back}	Backpressure ratio

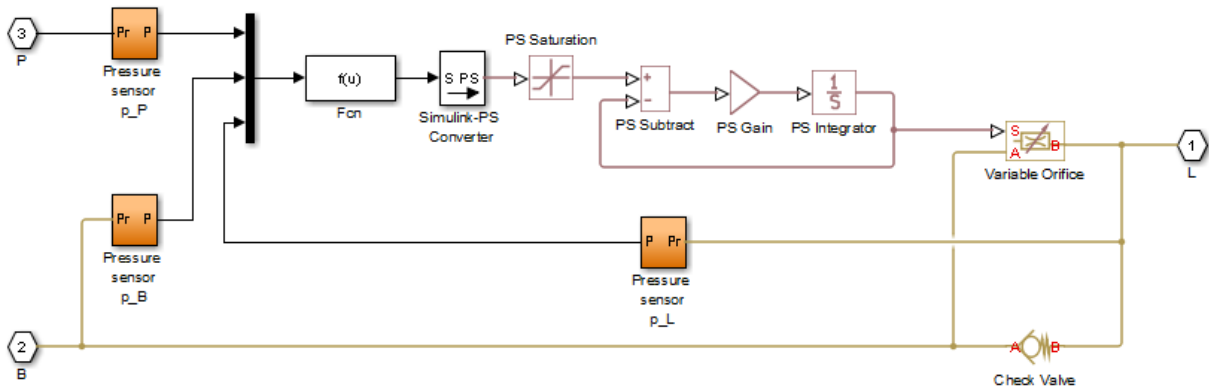
The valve displacement is determined from “Equation 1-6” on page 1-245

$$x = \frac{p_{pilot} \cdot k_{pilot} + p_{load} - p_{back} \cdot k_{back} - p_{set}}{c_p}$$

$$0 \leq x \leq x_{max}$$

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” on page 1-246. 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 1-247
- “Variable Orifice Tab” on page 1-248
- “Check Valve Tab” on page 1-251

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 50×10^5 Pa.

Valve spring stiffness

The valve spring pressure stiffness c_p . The default value is 1×10^8 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 $1e-9 \text{ m}^2$.

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 $8e-5 \text{ m}^2$. 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 $5e-4 \text{ m}$. 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 $1e-4 \text{ m}^2$.

Check valve cracking pressure

Pressure level at which the orifice of the valve starts to open. The default value is $3e4 \text{ Pa}$.

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 $1.2e5 \text{ Pa}$.

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

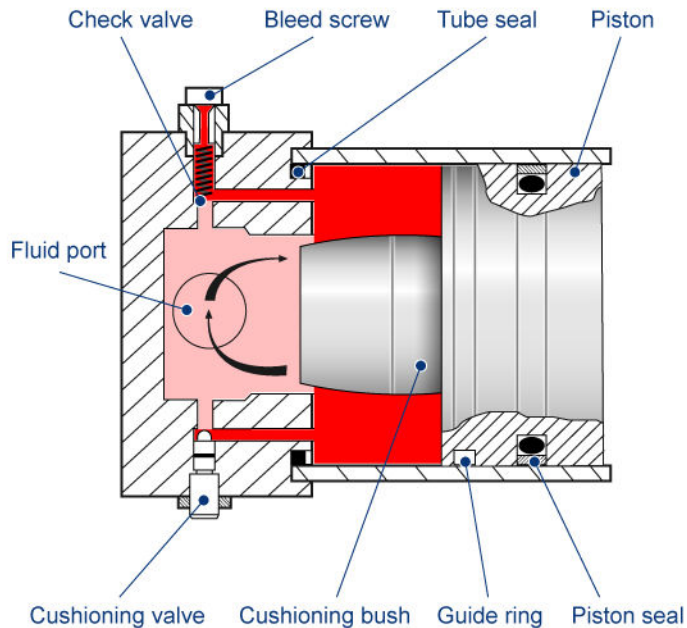


Library

Hydraulic Cylinders

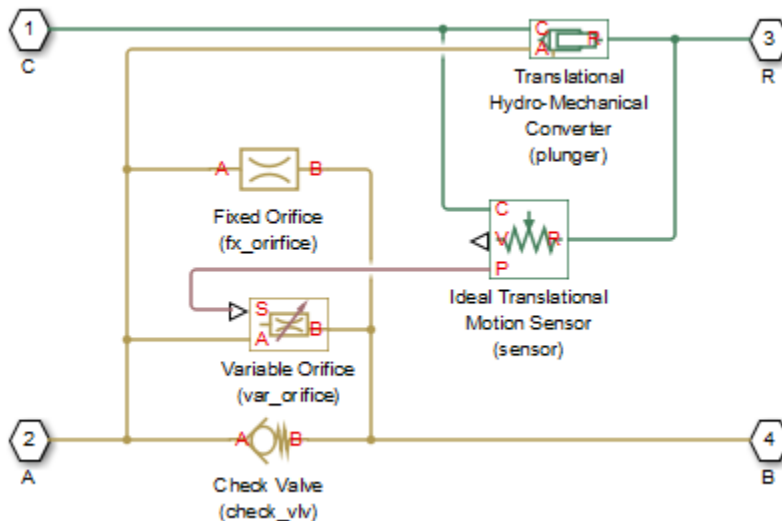
Description

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 1-264.



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.

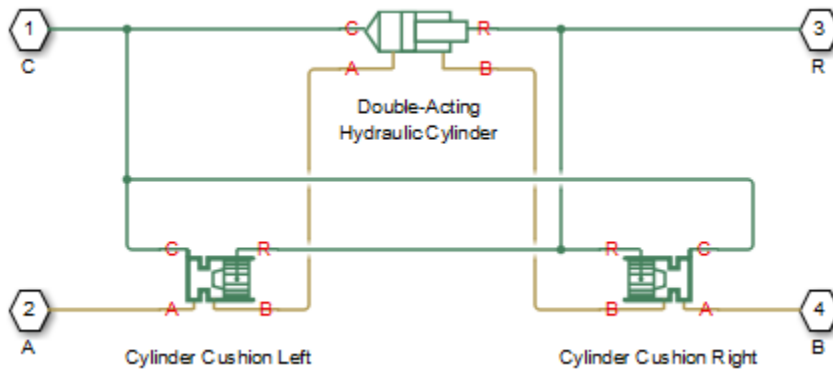
The block is implemented as a structural model that replicates a cushioning device, as shown in this diagram.



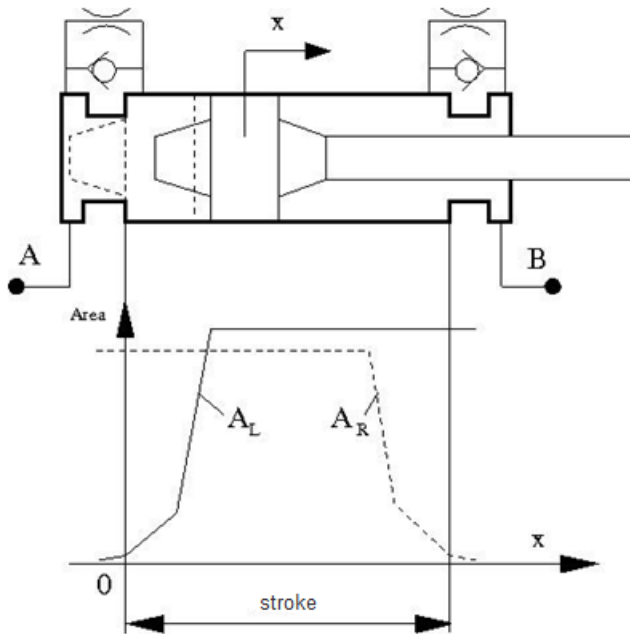
The Variable Orifice block represents a variable gap between the bush and the cavity machined in the end cap. The lookup table of the Variable Orifice block implements the relationship between the orifice area and the piston displacement. The Fixed Orifice and the Check Valve blocks simulate the cushioning 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.

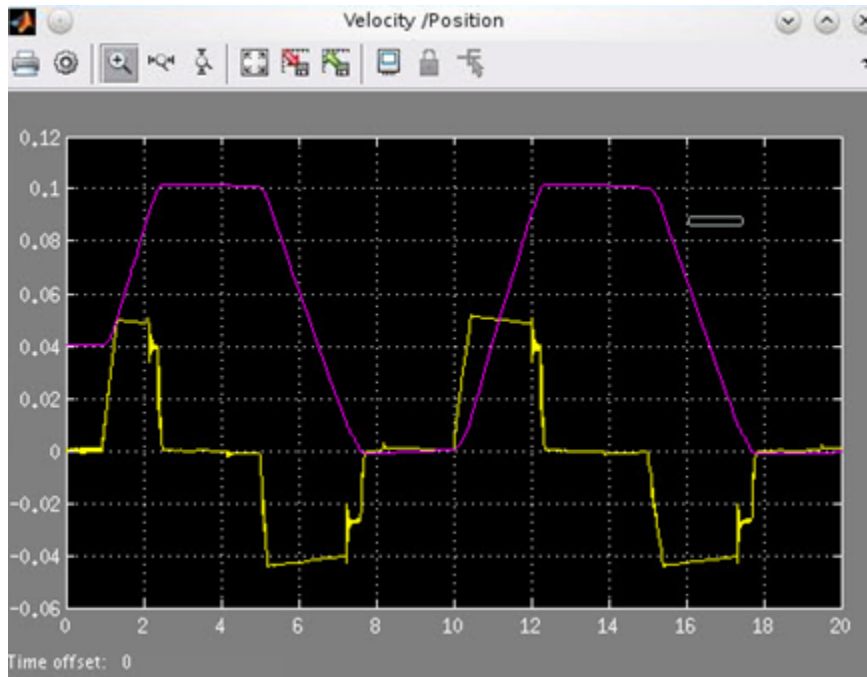


You can adjust the cushioning effect by changing the area of the fixed orifice and the profile of the cushioning bush (variable orifice). Specify the profile using the one-dimensional lookup table of the orifice area versus piston displacement. The next figure shows a typical configuration of a double-acting 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 1-260
- “Fixed Orifice Tab” on page 1-260
- “Variable Orifice Tab” on page 1-261
- “Check Valve Tab” on page 1-262

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 $1e-4 \text{ m}^2$.

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 $1e-6 \text{ m}^2$.

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 $1e-12 \text{ m}^2$.

Check Valve Tab

Check valve maximum area

Valve passage maximum cross-sectional area. The default value is $1e-4 \text{ m}^2$.

Check valve cracking pressure

Pressure level at which the orifice of the valve starts to open. The default value is $0.5e5 \text{ Pa}$.

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 $1.5e5 \text{ Pa}$.

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 $1e-12 \text{ m}^2$.

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:

$$F = F_C \cdot (1 + (K_{brk} - 1) \cdot \exp(-c_v |v|)) \cdot \text{sign}(v) + f_{vfr} \cdot v$$

$$F_C = F_{pr} + f_{cfr} (p_A + p_B)$$

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 $|v| \leq v_{th}$ is introduced around zero velocity, where friction force is assumed to be linearly proportional to velocity:

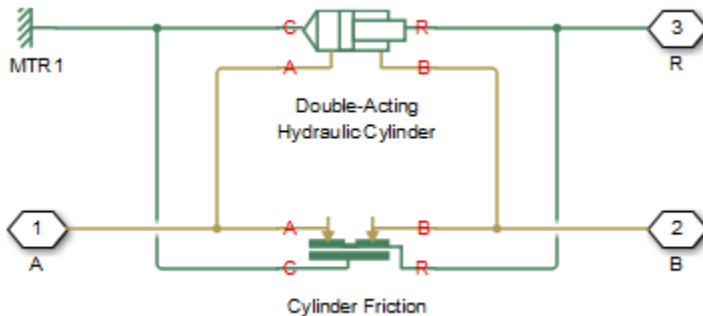
$$F = K \cdot v$$

$$K = \frac{F_C (1 + (K_{brk} - 1) \cdot \exp(-c_v v_{th})) + f_{vfr} \cdot v_{th}}{v_{th}}$$

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 1-267
- “Variables Tab” on page 1-268

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 $c_v \approx 4/v_{\min}$, 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



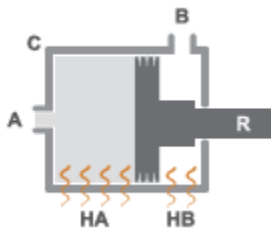
Library

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

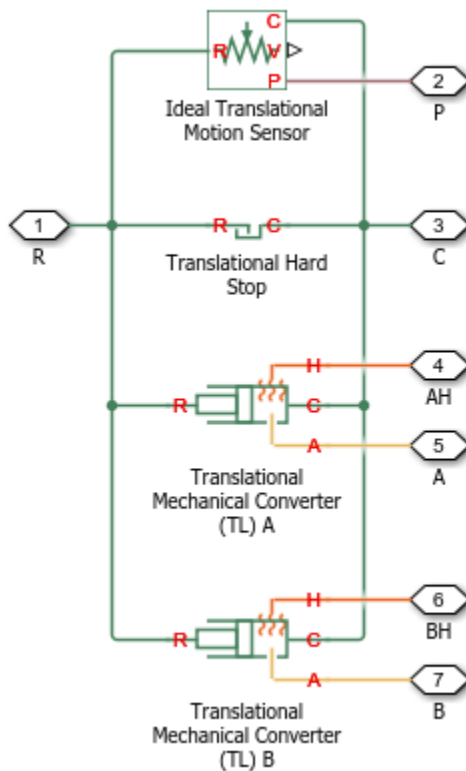
The direction of the piston motion depends on the mechanical orientation setting in the block dialog box. If the mechanical orientation is positive, then a higher pressure at port A yields a positive piston translation relative to the actuator case. The direction of motion reverses for a negative mechanical 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 $1\text{e-}5 \text{ m}^3$.

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 $1\text{e-}5 \text{ m}^3$.

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 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 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

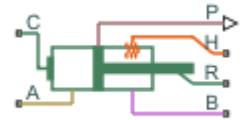
Single-Acting Actuator (TL)

Introduced in R2016a

Double-Acting Actuator (IL-G)

Linear actuator with piston motion controlled by opposing hydraulic and gas chambers

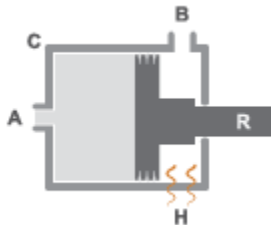
Library: Interfaces / Actuators



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

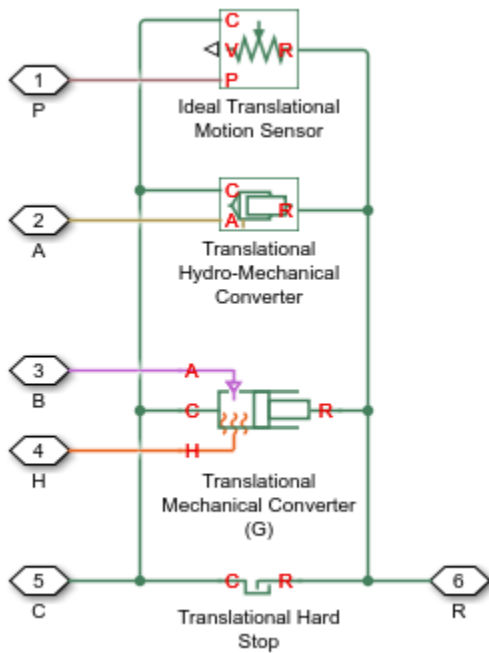
mechanical orientation is negative, then a higher pressure at port A yields a negative piston translation instead.

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 chamber2e-3 m² (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 m³ (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 m² (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 chamber1e-3 m² (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

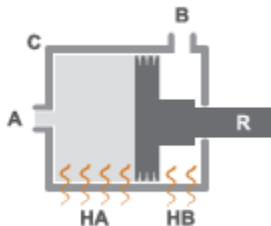
Library: Interfaces / Actuators



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

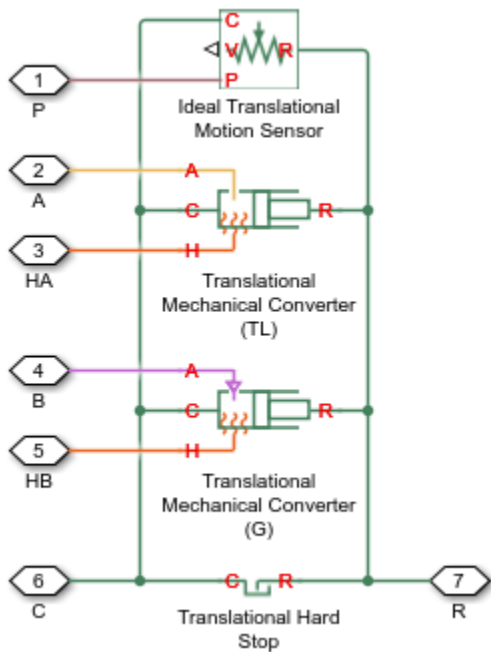
mechanical orientation is negative, then a higher pressure at port A yields a negative piston translation instead.

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 m² (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 m³ (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 m² (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 m² (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 m³ (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



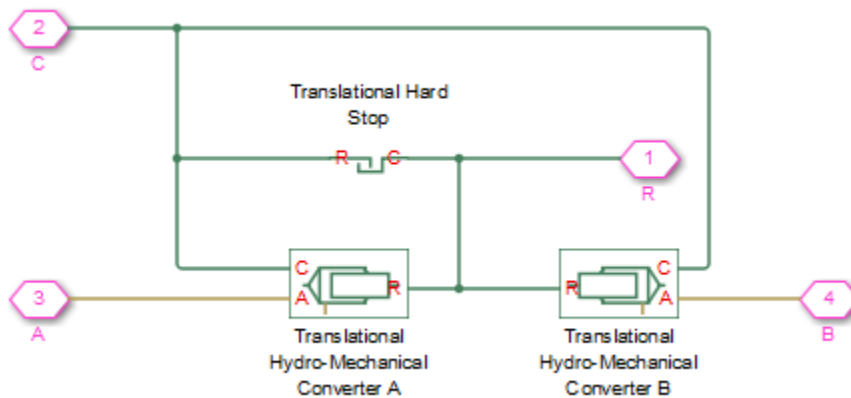
Library

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 1-295
- “Hard Stop Properties Tab” on page 1-296
- “Initial Conditions Tab” on page 1-297

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 $5\text{e-}5 \text{ m}^2$.

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 $1e-4 \text{ m}^3$.

Dead volume B

Fluid volume in chamber B that remains in the chamber after the rod is fully extended. The default value is $1e-4 \text{ m}^3$.

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 $1e6 \text{ N/m}$.

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 $5e-4 \text{ m}^2$.

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



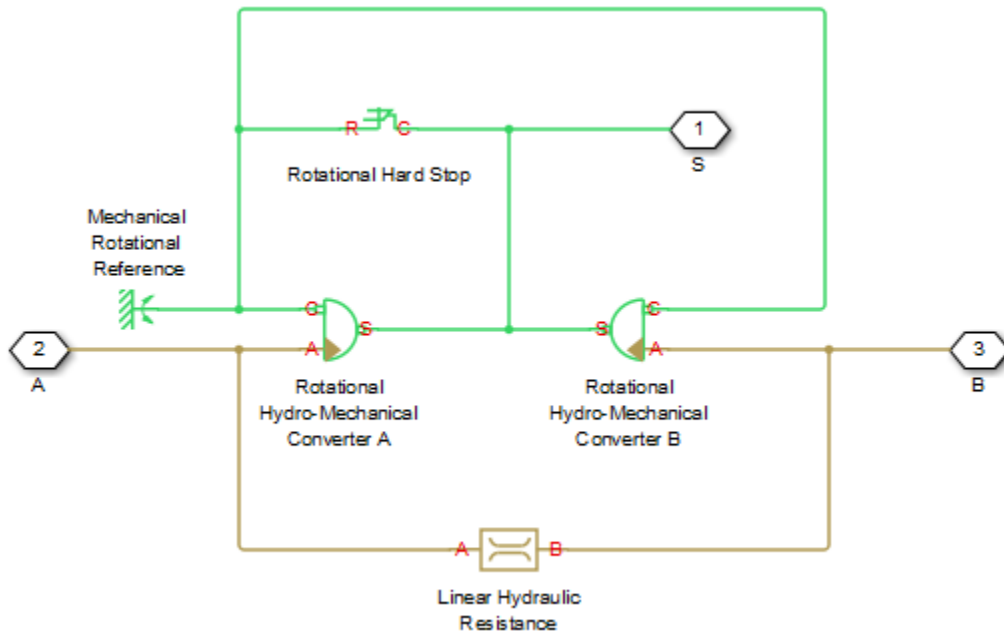
Library

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 $4.5 \times 10^{-5} \text{ m}^3/\text{rad}$.

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 $1 \times 10^{-4} \text{ m}^3$.

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 $1 \times 10^{-4} \text{ m}^3$.

Leak coefficient

Leak coefficient for the Linear Hydraulic Resistance block. The default value is $1 \times 10^{-14} \text{ (m}^3/\text{s)/Pa}$.

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 $1 \times 10^6 \text{ N}\cdot\text{m}/\text{rad}$.

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.

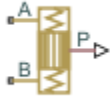
See Also

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

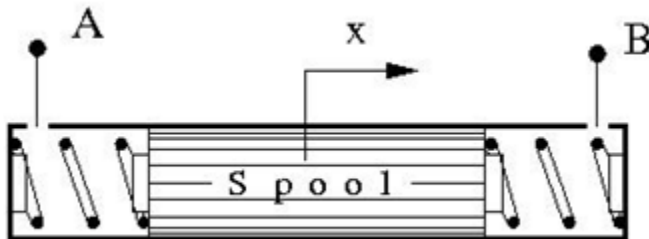


Library

Valve Actuators

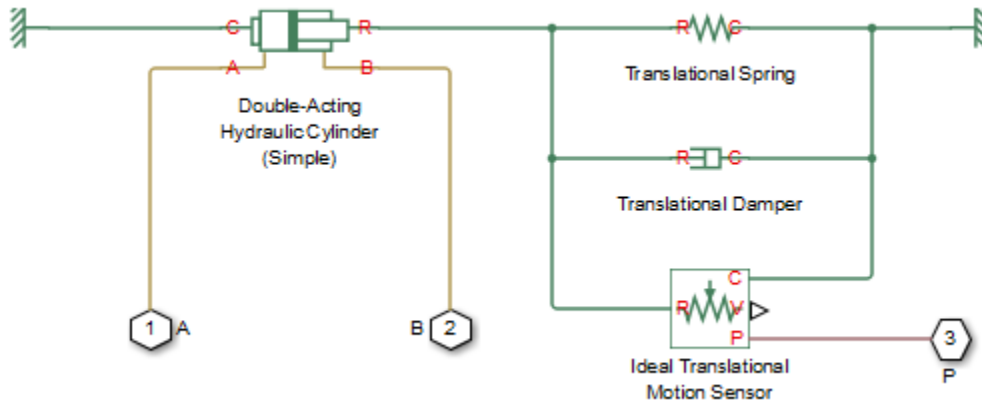
Description

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:

$$A(p_A - p_B) = k \cdot x + B \cdot v + F_{HS}$$

$$F_{HS} = \begin{cases} (x - S/2) \cdot K_p \cdot v & \text{if } x > S/2, v > 0 \\ (x + S/2) \cdot K_p \cdot \text{abs}(v) & \text{if } x < -S/2, v < 0 \\ 0 & \text{otherwise} \end{cases}$$

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 $1e-4 \text{ m}^2$.

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 $1e12 \text{ N/m/(m/s)}$.

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 1-318, 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.

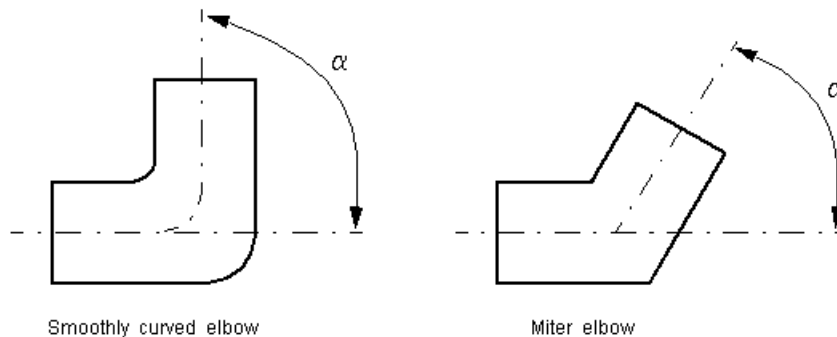
The core data for the pressure loss coefficient computation is the table-specified relationship between the friction factor f_T and the internal diameter for clean commercial steel pipes, with flow in the zone of complete turbulence (see [1] on page 1-318, p. A-26). For smoothly curved, standard 90° elbows, the pressure loss coefficient is determined with the formula

$$K = 30f_T$$

For elbows with different angles, the coefficient is corrected with the relationship presented in [2] on page 1-318, Fig.4.6:

$$K_{corr} = \alpha(0.0142 - 3.703 \cdot 10^{-5} \alpha)$$

where α is the elbow angle in degrees ($0 \leq \alpha \leq 90$).



Therefore, the pressure loss coefficient for smoothly curved, standard elbows is determined with the formula

$$K_{SCE} = 30f_T \cdot \alpha(0.0142 - 3.703 \cdot 10^{-5} \alpha)$$

For sharp-edged, miter bends the pressure loss coefficient is determined according to the table provided in [1] on page 1-318, p. A-29, as a function of the elbow diameter and angle

$$K_{ME} = f(d, \alpha)$$

where $5 \leq d \leq 100$ mm and $0 \leq \alpha \leq 90$ 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

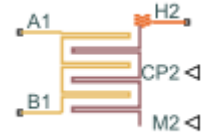
See Also

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: 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.

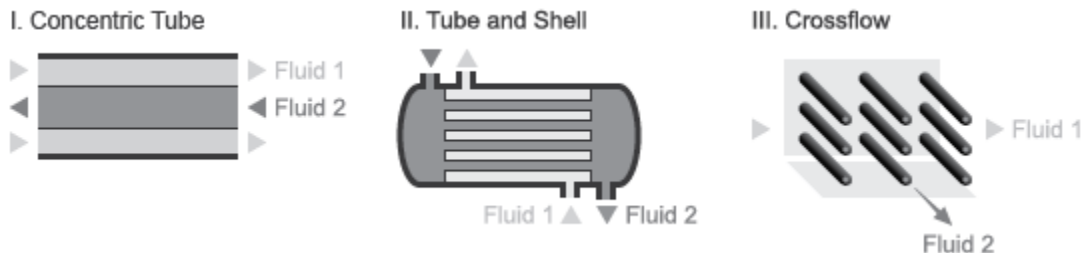
The equations underpinning the model, and therefore the available parameterizations and their required inputs, depend on the active block variant. To see the active variant or to change it, right-click the block and select **Simscape** > **Block Choices**. There are two variants with differing degrees 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.

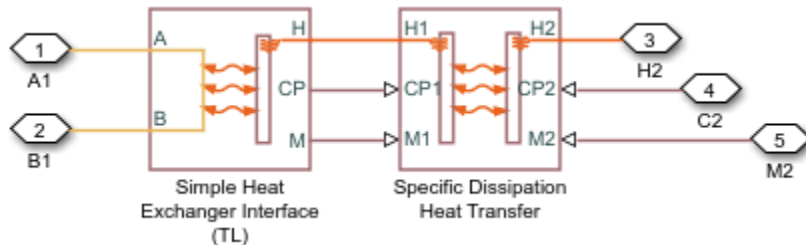


Heat Exchanger Configurations

Component Structure

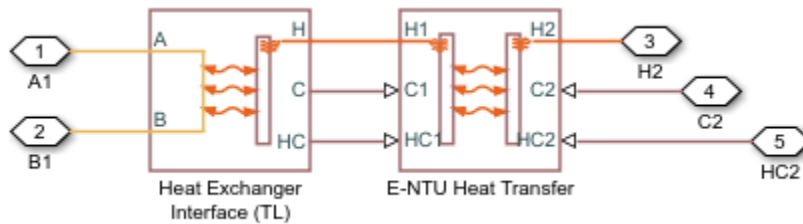
This block is a composite component assembled from simpler components found in the **Fundamental Components** library. The structural block diagram depends on the active block variant. The figures show the blocks comprising each variant and their respective connections.

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, $\mathit{mdot1}$ — 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, $\mathit{mdot2}$ — 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, $\mathit{SD}(\mathit{mdot1}, \mathit{mdot2})$ — 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.

Thermal Liquid 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 m³ (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 m² (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

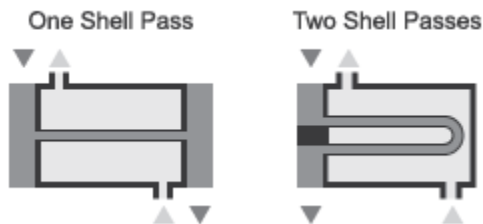
Parallel or counter flow (default) | Shell and tube | Cross flow | Generic - effectiveness table

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 m² (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 m³ (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 data12-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 number12-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 fluids0.4 m² (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 Prandtl 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 Prandtl 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/(m² * 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 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: Fluid Network Interfaces / Heat Exchangers



Description

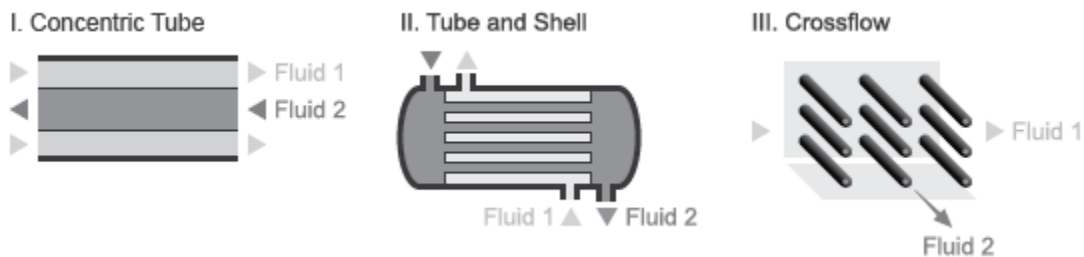
The Heat Exchanger (TL-TL) block models the heat transfer and fluid dynamics of a heat exchanger located between two thermal liquid networks. Each network must connect to a separate Thermal Liquid Settings (TL) block. The fluids do not mix nor do they change phase.

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.

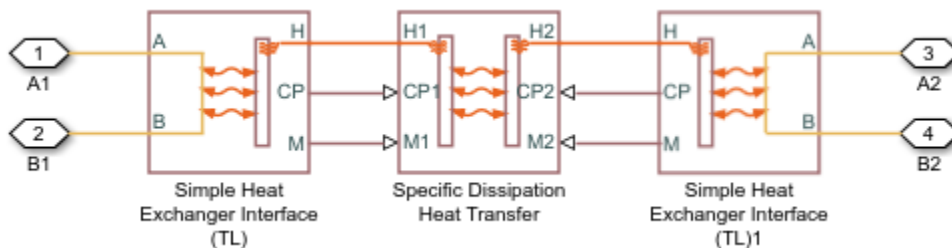


Heat Exchanger Configurations

Component Structure

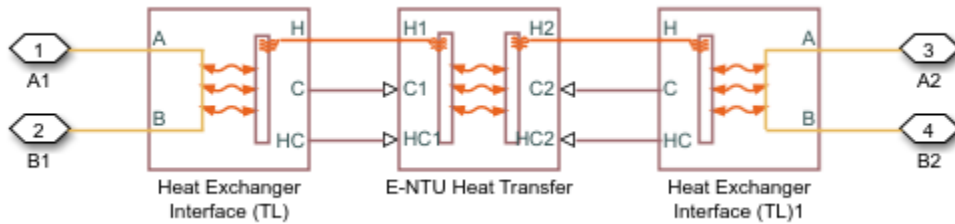
This block is a composite component assembled from simpler components found in the **Fundamental Components** library. The structural block diagram depends on the active block variant. The figures show the blocks comprising each variant and their respective connections.

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 m³ (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 m² (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

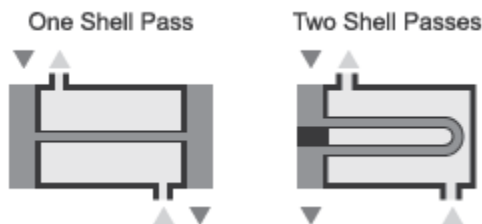
Parallel or counter flow (default) | Shell and tube | Cross flow | Generic - effectiveness table

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 m² (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 m³ (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 m² (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 Prandtl 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 Prandtl 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/(m² * 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.

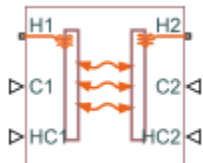
See Also

E-NTU Heat Transfer

Introduced in R2016a

E-NTU Heat Transfer

Detailed heat transfer model between two general fluids



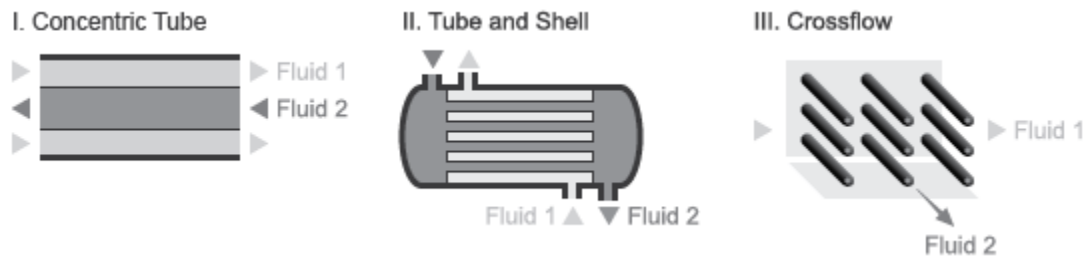
Library

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

$$Q_{Max} = C_{Min} (T_{1,In} - T_{2,In}),$$

where:

- C_{Min} is the minimum value of the thermal capacity rate:

$$C_{Min} = \min(\dot{m}_1 C_{p,1}, \dot{m}_2 C_{p,2})$$
 - $T_{1,In}$ 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.

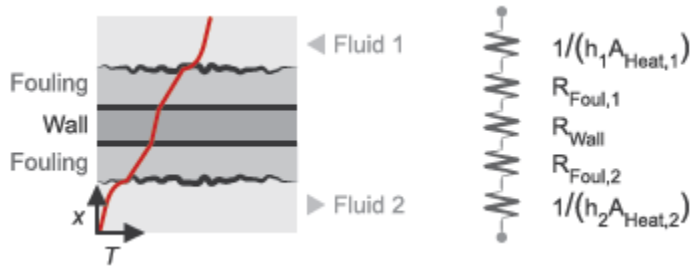
The overall heat transfer coefficient and thermal resistance used in the NTU calculation are functions of the heat transfer mechanisms at work. These mechanisms include convective heat transfer between the fluids and the heat exchanger interface and conduction through the interface wall [2 on page 1-353]:

$$R_{Overall} = \frac{1}{U_{Overall} A_{Heat}} = \frac{1}{h_1 A_{Heat,1}} + R_{Foul,1} + R_{Wall} + R_{Foul,2} + \frac{1}{h_2 A_{Heat,2}},$$

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 1-353]. 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	$\varepsilon = \begin{cases} \frac{1 - \exp[-NTU(1 - C_{rel})]}{1 - C_{rel} \exp[-NTU(1 - C_{rel})]}, & \text{if } C_{rel} < 1 \\ \frac{NTU}{1 + NTU}, & \text{if } C_{rel} = 1 \end{cases}$
Parallel Flow	$\varepsilon = \frac{1 - \exp[-NTU(1 + C_{rel})]}{1 + C_{rel}}$
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	$\varepsilon = \frac{\left[\frac{(1 - \varepsilon_1 C_{rel})}{(1 - \varepsilon_1)} \right]^N - 1}{\left[\frac{(1 - \varepsilon_1 C_{rel})}{(1 - \varepsilon_1)} \right]^N - C_{rel}}$
Cross Flow (Single Pass)	
Both Fluids Unmixed	$\varepsilon = 1 - \exp\left(\frac{\exp(-C_{rel} NTU^{0.78}) - 1}{C_{rel} NTU^{-0.22}} \right)$
Both Fluids Mixed	$\varepsilon = \frac{1}{\frac{1}{1 - \exp(-NTU)} + \frac{C_{rel}}{1 - \exp(-C_{rel} NTU)} - \frac{1}{NTU}}$
C_{Max} mixed, C_{Min} unmixed	$\varepsilon = \frac{1}{C_{rel}} \left(1 - \exp(-C_{rel} (1 - \exp(-NTU))) \right)$
C_{Max} unmixed, C_{Min} mixed	$\varepsilon = 1 - \exp\left(-\frac{1}{C_{rel}} (1 - \exp(-C_{rel} NTU)) \right)$

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

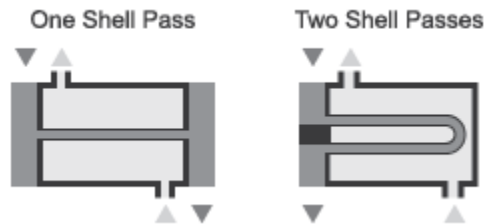
Heat exchanger geometry. Common geometries that you can select include Parallel or counter flow, Shell and tube, and Cross flow. Select Generic -

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 1.6×10^{-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 $1 \times 10^{-4} \text{ m}^2 \cdot \text{K/W}$.

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 $5 \text{ W}/(\text{m}^2 \cdot \text{K})$.

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 1-347.

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 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

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:

$$q = C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

$$p = p_A - p_B$$

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:

$$p_{cr} = \frac{\rho}{2} \left(\frac{Re_{cr} \cdot v}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

where

D_H	Orifice hydraulic diameter
v	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

$$p = p_A - p_B.$$

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 $1e-4 \text{ m}^2$.

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

$$p = p_A - p_B.$$

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 m³/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:

$$q = C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p_r}{(p_r^2 + p_{cr}^2)^{1/4}}$$

$$p = p_{in} + p_r$$

$$p_{in} = \rho \frac{L}{A} \frac{dq}{dt}$$

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:

$$p_{cr} = \frac{\rho}{2} \left(\frac{Re_{cr} \cdot v}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

where

D_H	Orifice hydraulic diameter
v	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 $p = p_A - p_B$.

Parameters

Orifice area

Cross-sectional area of the orifice. The default value is $1e-4 \text{ m}^2$.

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

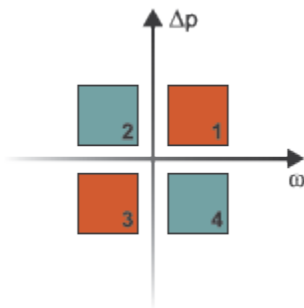
Library: Hydraulics (Isothermal) / Pumps and Motors



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

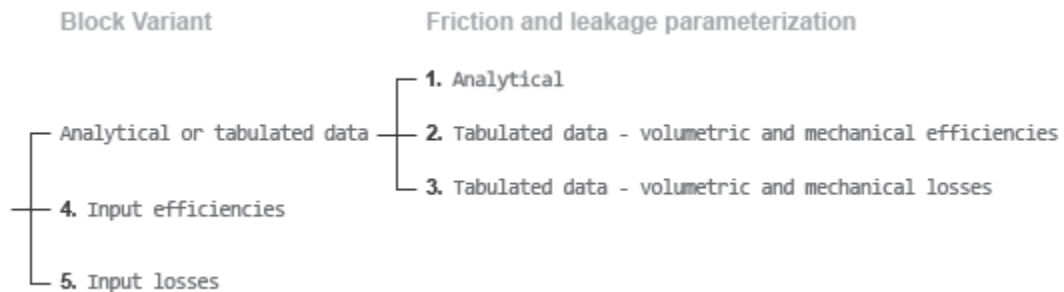
Modes” on page 1-365 figure maps the modes to the octants of a Δp - ω - D chart. The modes are labeled 1–4:

- 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 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

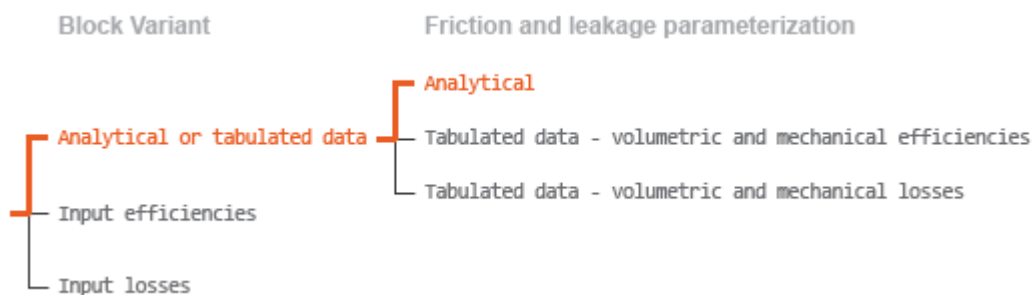
$$\tau_{\text{Ideal}} = D\Delta p,$$

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

$$\tau_{\text{Friction}} = (\tau_0 + K_{\text{TP}}|\Delta p|)\tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right),$$

where:

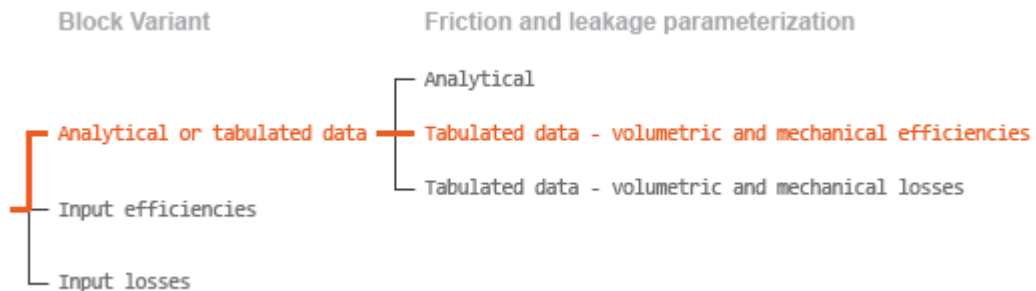
- K_{HP} is the Hagen-Poiseuille coefficient for laminar pipe flows. This coefficient is computed from the specified nominal parameters.
- K_{TP} is the specified value of the **Friction torque vs pressure drop coefficient** block parameter.
- τ_0 is the specified value of the **No-load torque** block parameter.
- $\omega_{\text{Threshold}}$ 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

$$K_{\text{HP}} = \frac{v_{\text{Nom}}}{\rho v} \frac{\rho_{\text{Nom}} \omega_{\text{Nom}} D}{\Delta p_{\text{Nom}}} \left(\frac{1}{\eta_{v,\text{Nom}}} - 1 \right),$$

where:

- v_{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.
- v 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.
- $\eta_{v,\text{Nom}}$ 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

$$\alpha = \tanh\left(\frac{4\Delta p}{\Delta p_{\text{Threshold}}}\right) \tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right),$$

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

$$q_{\text{Leak,Motor}} = (1 - \eta_v) q,$$

in motor mode and through the equation

$$q_{\text{Leak,Pump}} = -(1 - \eta_v) q_{\text{Ideal}}.$$

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

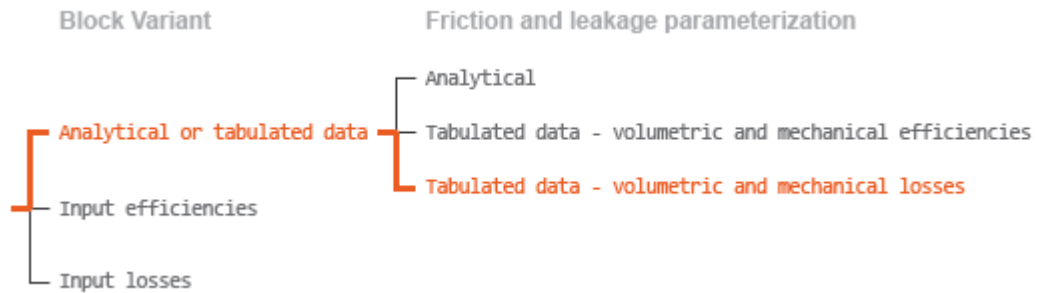
$$\tau_{\text{Friction, Motor}} = (1 - \eta_m) \tau_{\text{Ideal}},$$

in motor mode and through the equation

$$\tau_{\text{Friction, Pump}} = -(1 - \eta_m) \tau.$$

in pump mode, where:

- η_m is the mechanical efficiency obtained through interpolation or extrapolation of the **Mechanical efficiency table, $e_m(\mathbf{dp}, \mathbf{w})$** parameter data.



Case 3: Loss Tabulated Data

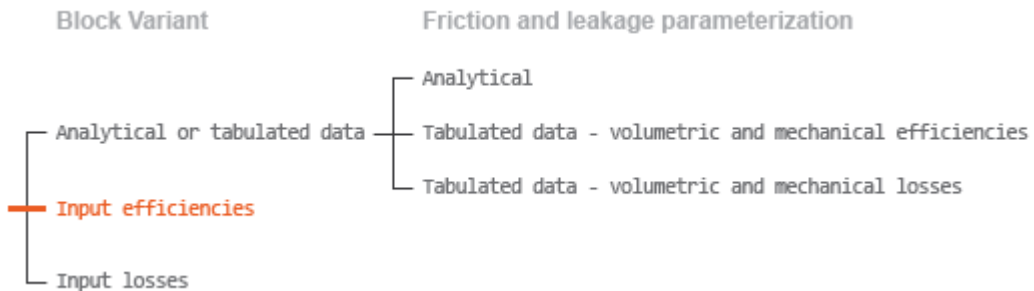
If the active block variant is Analytical or tabulated data and the **Leakage and friction parameterization** parameter is set to Tabulated data – volumetric and mechanical losses, the leakage flow rate equation is

$$q_{\text{Leak}} = q_{\text{Leak}}(\Delta p, \omega).$$

and the friction torque equation is

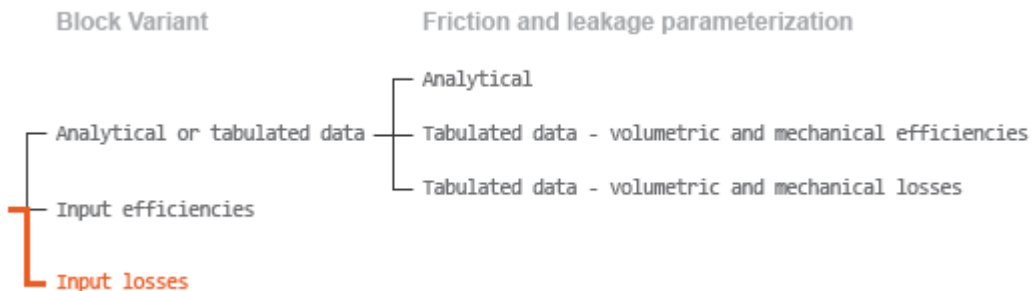
$$\tau_{\text{Friction}} = \tau_{\text{Friction}}(\Delta p, \omega),$$

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, $q_{\text{loss}}(\mathbf{dp}, \mathbf{w})$** and **Mechanical loss table, $\text{torque_loss}(\mathbf{dp}, \mathbf{w})$** 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 1-366.

Variant 1: Analytical or tabulated data**Displacement — Fluid volume displaced per unit shaft rotation angle**

30 cm³/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/m³ (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 cm³/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 cm³/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

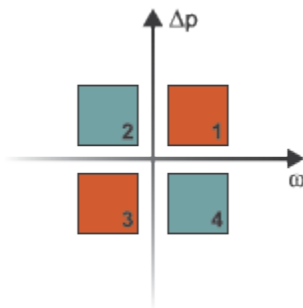
Library: Thermal Liquid / Pumps & Motors



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**

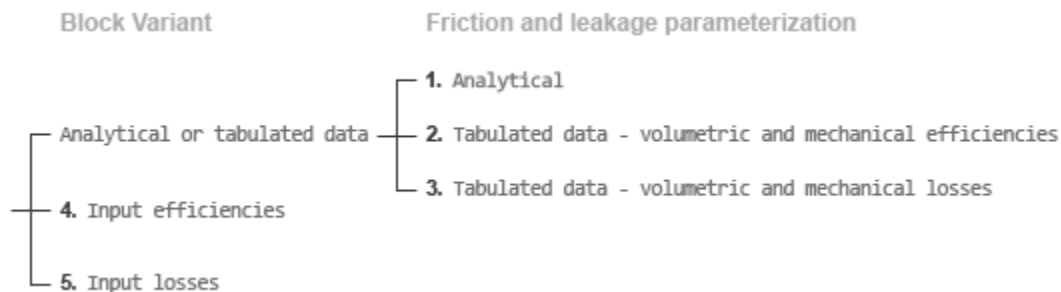
(ω). The “Operation Modes” on page 1-387 figure maps the modes to the octants of a Δp - ω - D chart. The modes are labeled 1–4:

- 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

$$\dot{m} = \dot{m}_{\text{Ideal}} + \dot{m}_{\text{Leak}},$$

where:

- \dot{m} is the actual mass flow rate.
- \dot{m}_{Ideal} is the ideal mass flow rate.
- \dot{m}_{Leak} is the internal leakage mass flow rate.

The torque generated at the motor is

$$\tau = \tau_{\text{Ideal}} - \tau_{\text{Friction}},$$

where:

- τ is the actual torque.
- τ_{Ideal} is the ideal torque.
- τ_{Friction} is the friction torque.

The ideal mass flow rate is

$$\dot{m}_{\text{Ideal}} = \rho D \omega,$$

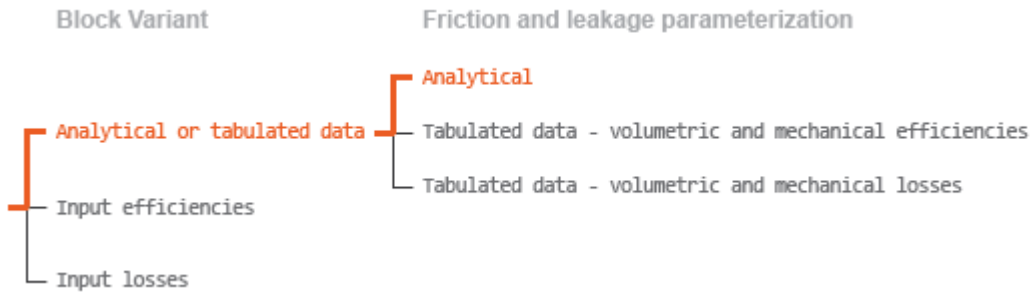
and the ideal generated torque is

$$\tau_{\text{Ideal}} = D \Delta p,$$

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

$$\dot{m}_{\text{Leak}} = K_{\text{HP}} \Delta p,$$

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.
- K_{TP} is the specified value of the **Friction torque vs pressure drop coefficient** block parameter.

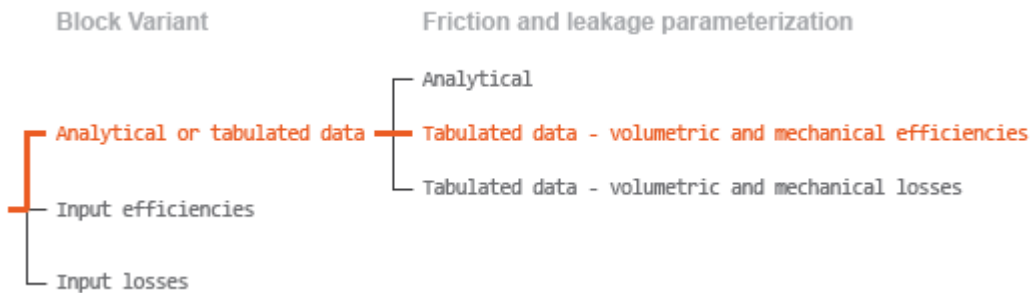
- τ_0 is the specified value of the **No-load torque** block parameter.
- ω_{Nom} is the specified value of the **Nominal shaft angular velocity** block parameter.

The Hagen-Poiseuille coefficient is determined from nominal fluid and component parameters through the equation

$$K_{\text{HP}} = \frac{D\omega_{\text{Nom}}\mu_{\text{Nom}}\left(\frac{1}{\eta_{v,\text{Nom}}} - 1\right)}{\Delta p_{\text{Nom}}},$$

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.
- $\eta_{v,\text{Nom}}$ 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

$$\alpha = \tanh\left(\frac{4\Delta p}{\Delta p_{\text{Threshold}}}\right) \tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right),$$

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 1-387 figure), the leakage flow rate is:

$$\dot{m}_{\text{Leak, Motor}} = (1 - \eta_v) \dot{m},$$

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:

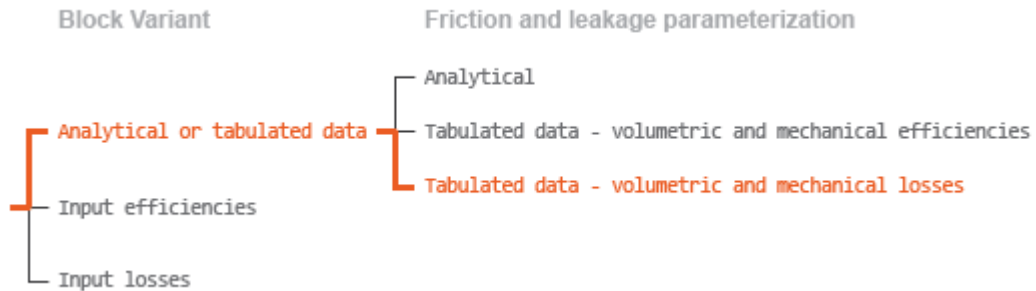
$$\dot{m}_{\text{Leak, Pump}} = -(1 - \eta_v) \dot{m}_{\text{Ideal}}.$$

The friction torque is similarly calculated from the mechanical efficiency, a quantity that is specified in tabulated form over the Δp - ω domain via the **Mechanical efficiency table** block parameter. When operating in motor mode (quadrants 1 and 3 of the Δp - ω chart):

$$\tau_{\text{Friction, Motor}} = (1 - \eta_m) \tau_{\text{Ideal}},$$

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 chart):

$$\tau_{\text{Friction, Pump}} = -(1 - \eta_m) \tau.$$



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 domain:

$$q_{\text{Leak}} = q_{\text{Leak}}(\Delta p, \omega).$$

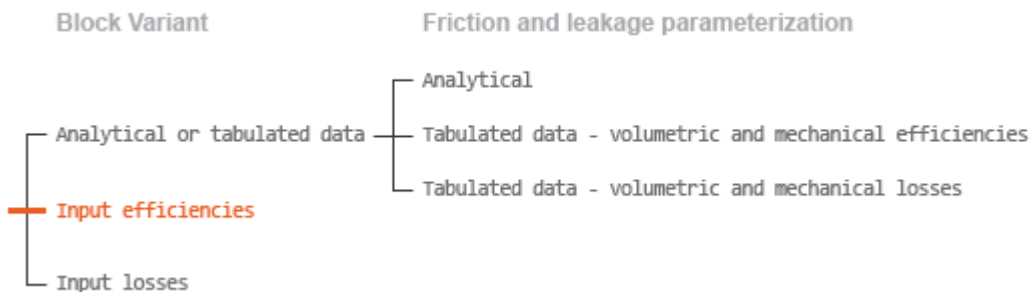
The mass flow rate due to leakage is calculated from the volumetric flow rate:

$$\dot{m}_{\text{Leak}} = \rho q_{\text{Leak}}.$$

The friction torque is similarly specified in tabulated form:

$$\tau_{\text{Friction}} = \tau_{\text{Friction}}(\Delta p, \omega),$$

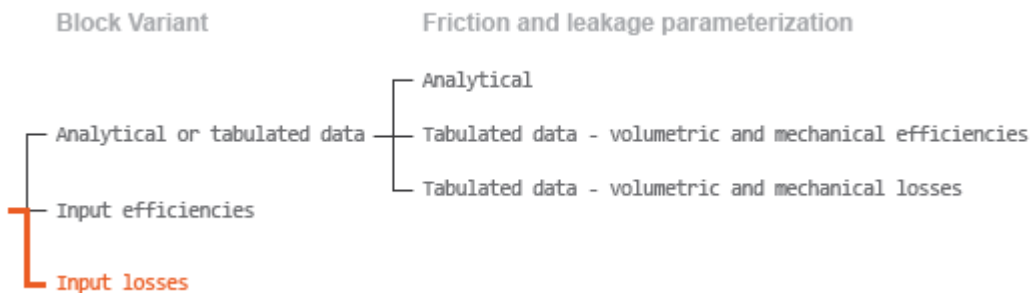
where $q_{\text{Leak}}(\Delta p, \omega)$ and $\tau_{\text{Friction}}(\Delta p, \omega)$ are the volumetric and mechanical losses, obtained through interpolation or extrapolation of the tabulated data specified via the **Volumetric loss table** and **Mechanical loss table** block parameters.



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 1-955.

Variant 1: Analytical or tabulated data

Displacement — Fluid volume displaced per unit shaft rotation angle

30 cm³/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 m² (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, Δp — 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 $(\omega, \Delta p)$ 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 $(\omega, \Delta p)$ 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 $(\omega, \Delta p)$ 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 $(\omega, \Delta p)$ 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, Δp — 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 $(\omega, \Delta p)$ 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, \mathbf{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 $(\omega, \Delta p)$ 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`.

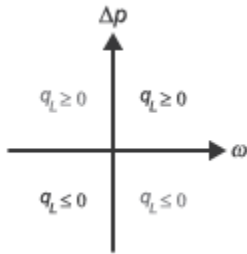
Volumetric loss table, $\mathbf{q_loss}(\mathbf{dp}, \mathbf{w})$ — Internal leakage flow rates at the specified pressure drops and angular velocities

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, \mathbf{dp}** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, \mathbf{w}** parameter.

The tabulated data need not encompass all quadrants of operation—those of a $(\omega, \Delta p)$ 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`.

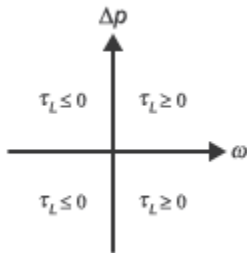
Mechanical loss table, torque_loss(dp,w) — Friction torques at the specified pressure drops and angular velocities

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 cm³/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 outlet0.01 m² (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 cm³/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 m² (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

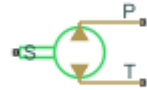
Fixed-Displacement Pump (TL)

Introduced in R2016a

Fixed-Displacement Pump

Mechanical-to-hydraulic power conversion device

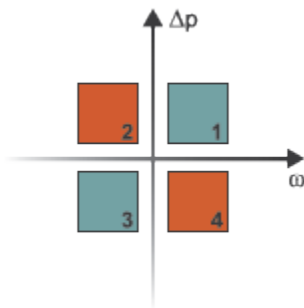
Library: Hydraulics (Isothermal) / Pumps and Motors



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

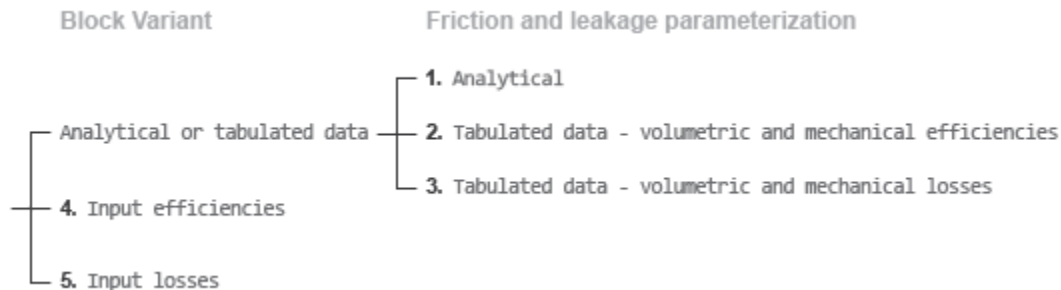
Modes” on page 1-409 figure maps the modes to the quadrants of a Δp - ω chart. The modes are labeled 1–4:

- 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 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

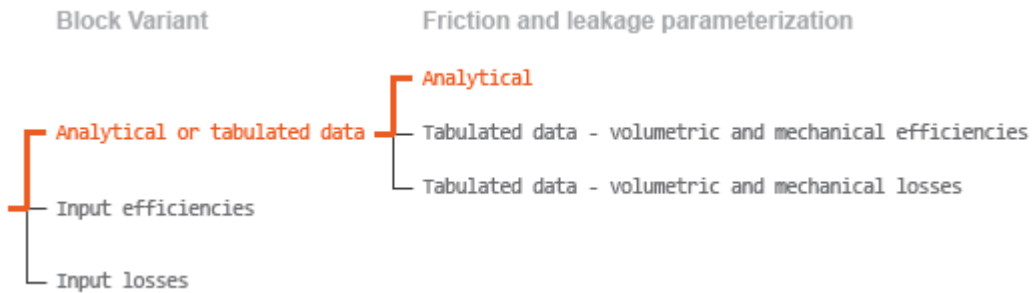
$$\tau_{\text{Ideal}} = D\Delta p,$$

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

$$\tau_{\text{Friction}} = (\tau_0 + K_{\text{TP}}|\Delta p|)\tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right),$$

where:

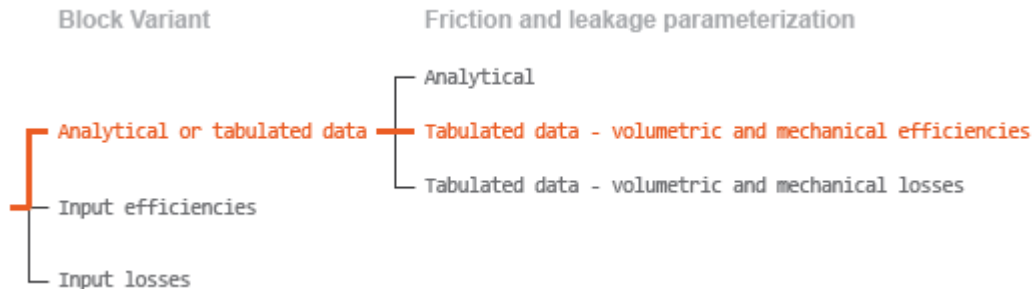
- K_{HP} is the Hagen-Poiseuille coefficient for laminar pipe flows. This coefficient is computed from the specified nominal parameters.
- K_{TP} is the specified value of the **Friction torque vs pressure gain coefficient** block parameter.
- τ_0 is the specified value of the **No-load torque** block parameter.
- $\omega_{\text{Threshold}}$ 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

$$K_{\text{HP}} = \frac{v_{\text{Nom}}}{\rho v} \frac{\rho_{\text{Nom}} \omega_{\text{Nom}} D}{\Delta p_{\text{Nom}}} (1 - \eta_{v,\text{Nom}}),$$

where:

- v_{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.
- v 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.
- $\eta_{v,\text{Nom}}$ 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

$$\alpha = \tanh\left(\frac{4\Delta p}{\Delta p_{\text{Threshold}}}\right) \tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right),$$

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

$$q_{\text{Leak,Pump}} = (1 - \eta_v) q_{\text{Ideal}},$$

in pump mode and through the equation

$$q_{\text{Leak,Motor}} = -(1 - \eta_v) q,$$

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

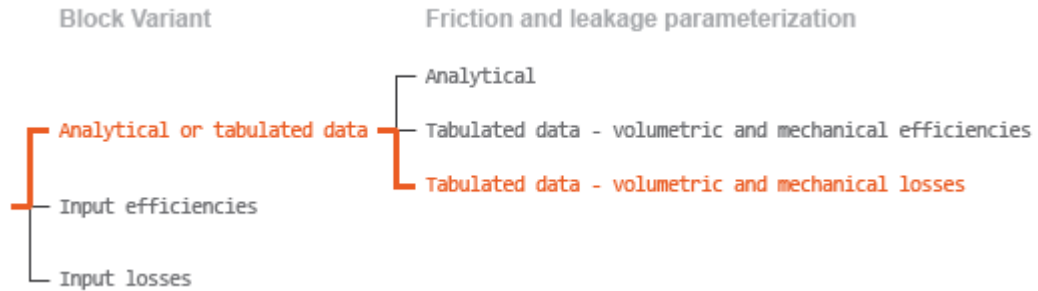
$$\tau_{\text{Friction,Pump}} = (1 - \eta_m) \tau,$$

in pump mode and through the equation

$$\tau_{\text{Friction, Motor}} = -(1 - \eta_m) \tau_{\text{Ideal}},$$

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

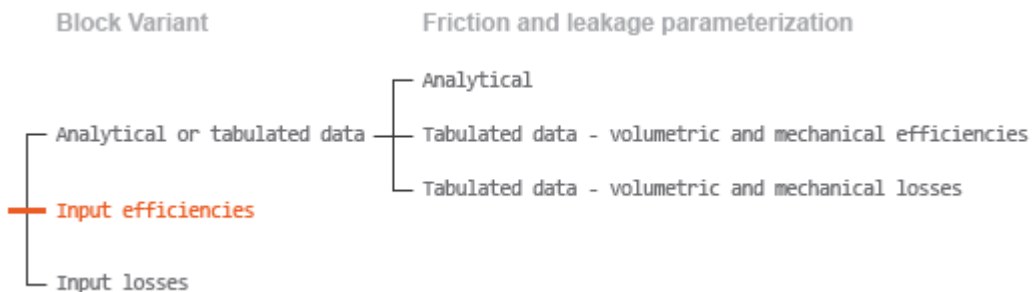
If the active block variant is Analytical or tabulated data and the **Leakage and friction parameterization** parameter is set to Tabulated data – volumetric and mechanical losses, the leakage flow rate equation is

$$q_{\text{Leak}} = q_{\text{Leak}}(\Delta p, \omega).$$

and the friction torque equation is

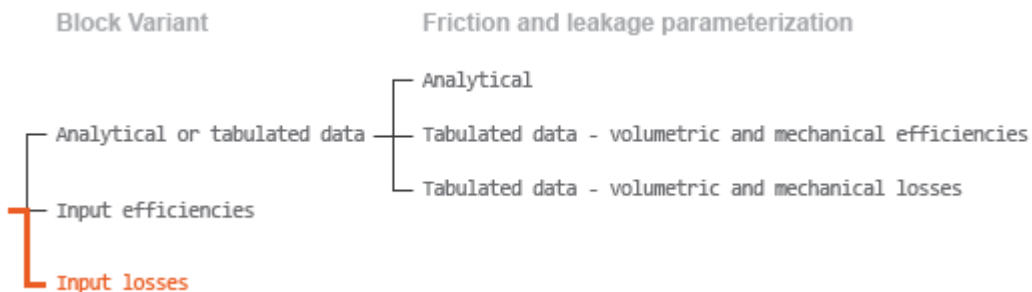
$$\tau_{\text{Friction}} = \tau_{\text{Friction}}(\Delta p, \omega),$$

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, m³/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 1-410.

Variant 1: Analytical or tabulated data**Displacement — Fluid volume displaced per unit shaft rotation angle**

30 cm³/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/m³ (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 cm³/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 cm³/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 modeNone (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 pressureNone (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

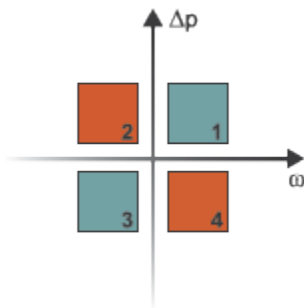
Library: Thermal Liquid / Pumps & Motors



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**

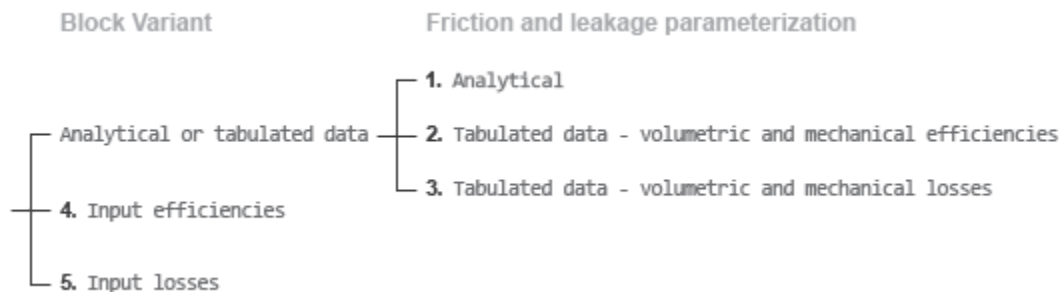
(ω). The “Operation Modes” on page 1-431 figure maps the modes to the quadrants of a Δp - ω chart. The modes are labeled 1–4:

- **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

$$\dot{m} = \dot{m}_{\text{Ideal}} - \dot{m}_{\text{Leak}},$$

where:

- \dot{m} is the actual mass flow rate.
- \dot{m}_{Ideal} is the ideal mass flow rate.
- \dot{m}_{Leak} is the internal leakage mass flow rate.

The driving torque required to power the pump is

$$\tau = \tau_{\text{Ideal}} + \tau_{\text{Friction}},$$

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

$$\dot{m}_{\text{Ideal}} = \rho D \omega,$$

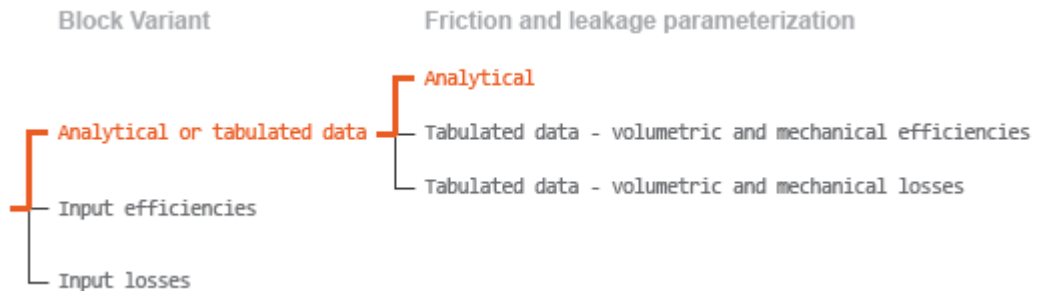
and the ideal generated torque is

$$\tau_{\text{Ideal}} = D \Delta p,$$

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

$$\dot{m}_{\text{Leak}} = K_{\text{HP}} \Delta p,$$

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.
- K_{TP} is the specified value of the **Friction torque vs pressure drop coefficient** block parameter.

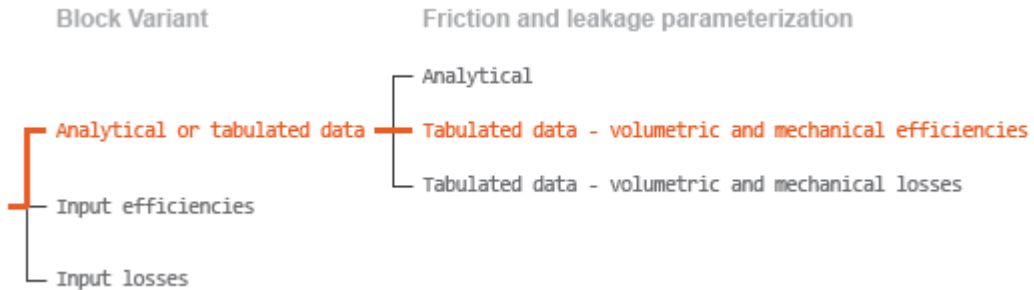
- τ_0 is the specified value of the **No-load torque** block parameter.
- ω_{Nom} is the specified value of the **Nominal shaft angular velocity** block parameter.

The Hagen-Poiseuille coefficient is determined from nominal fluid and component parameters through the equation

$$K_{\text{HP}} = \frac{D\omega_{\text{Nom}}\mu_{\text{Nom}}(1 - \eta_{v,\text{Nom}})}{\Delta p_{\text{Nom}}},$$

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.
- $\eta_{v,\text{Nom}}$ 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

$$\alpha = \tanh\left(\frac{4\Delta p}{\Delta p_{\text{Threshold}}}\right) \tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right),$$

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 figure), the leakage flow rate is:

$$\dot{m}_{\text{Leak,Pump}} = (1 - \eta_v) \dot{m}_{\text{Ideal}},$$

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:

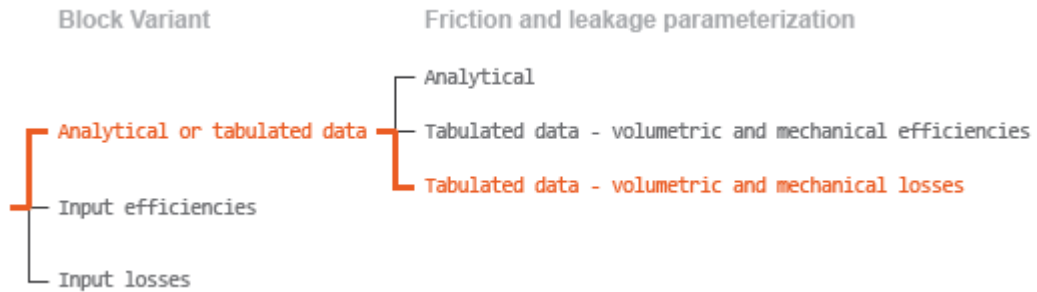
$$\dot{m}_{\text{Leak,Motor}} = -(1 - \eta_v) \dot{m}.$$

The friction torque is similarly calculated from the mechanical efficiency, a quantity that is specified in tabulated form over the Δp – ω domain via the **Mechanical efficiency table** block parameter. When operating in pump mode (quadrants **1** and **3** of the Δp – ω chart):

$$\tau_{\text{Friction,Pump}} = (1 - \eta_m) \tau,$$

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 chart):

$$\tau_{\text{Friction,Motor}} = -(1 - \eta_m) \tau_{\text{Ideal}}.$$



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 domain:

$$q_{\text{Leak}} = q_{\text{Leak}}(\Delta p, \omega).$$

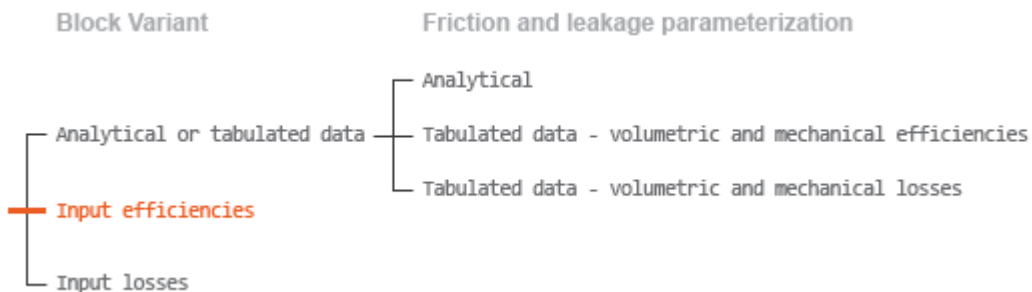
The mass flow rate due to leakage is calculated from the volumetric flow rate:

$$\dot{m}_{\text{Leak}} = \rho q_{\text{Leak}}.$$

The friction torque is similarly specified in tabulated form:

$$\tau_{\text{Friction}} = \tau_{\text{Friction}}(\Delta p, \omega),$$

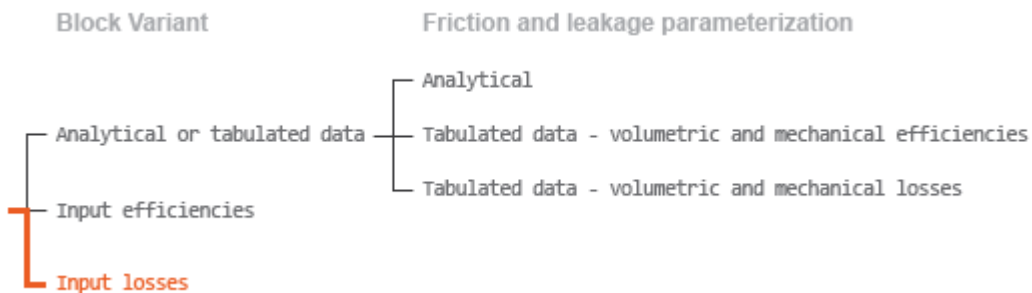
where $q_{\text{Leak}}(\Delta p, \omega)$ and $\tau_{\text{Friction}}(\Delta p, \omega)$ are the volumetric and mechanical losses, obtained through interpolation or extrapolation of the tabulated data specified via the **Volumetric loss table** and **Mechanical loss table** block parameters.



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 1-1014.

Variant 1: Analytical or tabulated data

Displacement — Fluid volume displaced per unit shaft rotation angle

30 cm³/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 m² (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 $(\omega, \Delta p)$ 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 $(\omega, \Delta p)$ 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 $(\omega, \Delta p)$ 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 $(\omega, \Delta p)$ 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, Δp — 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 $(\omega, \Delta p)$ 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, \mathbf{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 $(\omega, \Delta p)$ 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`.

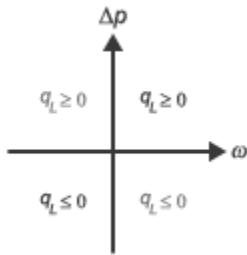
Volumetric loss table, $\mathbf{q_loss}(\Delta p, \mathbf{w})$ — Internal leakage flow rates at the specified pressure gains and angular velocities

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, Δp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, \mathbf{w}** parameter.

The tabulated data need not encompass all quadrants of operation—those of a $(\omega, \Delta p)$ 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`.

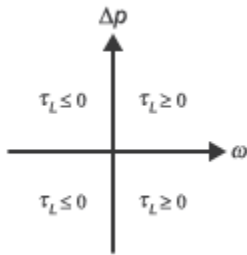
Mechanical loss table, `torque_loss(dp,w)` — Friction torques at the specified pressure gains and angular velocities

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 $(\omega, \Delta p)$ 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 cm³/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 outlet0.01 m² (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 cm³/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 m² (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)

Introduced in R2016a

Flow Coefficient Parameterized Valve (TL)

Generic valve model with flow coefficient parameterization



Library

Thermal Liquid/Valves

Description

The Flow Coefficient Parameterized Valve (TL) block models a generic valve using a parameterization based on the valve flow coefficient, a constant relating the flow rate to the pressure drop. The block accepts the flow coefficient in metric units or imperial units:

- C_v — Valve flow coefficient in imperial units of USG/min. C_v data is often provided at a temperature of 60 °F and a valve pressure differential of 1 lb/in² [1 on page 1-458].
- K_v — Valve flow coefficient in metric units of m³/h. K_v data is often provided at a temperature at 5–30 °C and a valve pressure differential of 1 bar [2 on page 1-458].

The physical signal L sets the valve opening fraction as a numerical value from 0 to 1. A value of 0 corresponds to a fully closed valve and leakage flow. A value of 1 corresponds to a fully open valve and maximum flow. The input signal saturates at these values.

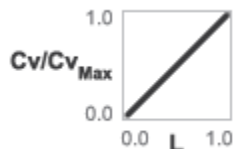
Valve Opening Characteristics

At valve opening fractions between 0 and 1, the opening area depends on the valve opening parameterization selected in the block dialog box. The block provides three parameterizations:

- **Linear** — Model the valve open fraction $f(L)$ as a linear function of the lift input signal L:

$$f(L) = L$$

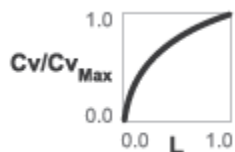
This behavior is suitable at constant pressure drops in steady-state systems. The figure shows the relationship between the valve flow coefficient, expressed as a fraction of the maximum flow coefficient, and the valve lift input signal.



- Quick opening — Model the valve open fraction $f(L)$ as a power function of the lift input signal L :

$$f(L) = L^{1/\alpha}$$

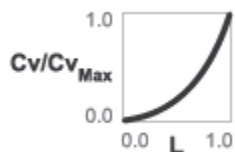
The parameter α is an exponent number that you specify. This behavior is suitable for pressure-relief valves that must open quickly from a fully closed state. The figure shows the relationship between the valve flow coefficient, expressed as a fraction of the maximum flow coefficient, and the valve lift input signal.



- Equal percentage — Model the valve open fraction $f(L)$ as an exponential function of the valve lift input signal:

$$f(L) = R^{L-1}$$

The parameter R is the valve rangeability—the ratio of the maximum and minimum valve flow rates. The figure shows the relationship between the valve flow coefficient, expressed as a fraction of the maximum flow coefficient, and the valve lift input signal.



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

$$p_A - p_B = \frac{\dot{m} \sqrt{\dot{m}^2 + \dot{m}_{cr}^2}}{\rho_{Avg} S^2},$$

where:

- p_A is the pressure at port A.
- p_B is the pressure at port B.
- ρ_{Avg} is the average fluid density.
- S is the valve opening area.
- \dot{m}_{cr} is the critical mass flow rate.

The valve opening area is

$$S = \begin{cases} S_{Max} f(L), & S_{Max} f(L) > S_{Min} \\ S_{Min}, & \text{Else} \end{cases}$$

where:

- S_{Max} is the maximum valve opening area.
- S_{Min} is the minimum valve opening area.
- $f(L)$ is the inherent valve flow characteristic.

The critical mass flow rate is

$$\dot{m}_{cr} = Re_{cr} \mu_{Avg} \sqrt{\frac{\pi}{4} S},$$

where:

- Re_{cr} is the critical Reynolds number at which the flow regime transitions between laminar and turbulent.
- μ_{Avg} is the average dynamic viscosity.

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

Flow coefficient specification

Flow coefficient to use in the block calculations. Options include Cv, defined in imperial units, and Kv, defined in metric units. The default setting is Cv coefficient (USG/min).

Cv coefficient at maximum flow

Valve flow coefficient in the fully open position, specified in imperial units. The default value is 1. This parameter is active only when the **Flow coefficient specification** parameter is set to Cv coefficient (USG/min).

Kv coefficient at maximum flow

Valve flow coefficient in the fully open position, specified in metric units. The default value is 1. This parameter is active only when the **Flow coefficient specification** parameter is set to Kv coefficient (m³/h).

Cv coefficient at minimum flow

Valve flow coefficient in the fully closed position, specified in imperial units. The default value is $1e-4$. This parameter is active only when the **Flow coefficient specification** parameter is set to `Cv coefficient (USG/min)`.

Kv coefficient at minimum flow

Valve flow coefficient in the fully closed position, specified in metric units. The default value is $1e-4$. This parameter is active only when the **Flow coefficient specification** parameter is set to `Kv coefficient (m3/h)`.

Valve opening characteristics

Valve opening response to the lift input signal specified through port L. The block provides three valve opening models:

- **Linear** — Models the valve open fraction $f(L)$ as a linear function of the lift input signal L:

$$f(L) = L$$

- **Quick opening** — Models the valve open fraction $f(L)$ as a power function of the lift input signal L:

$$f(L) = L^{1/\alpha}$$

- **Equal percentage** — Models the valve open fraction $f(L)$ as an exponential function of the valve lift input signal:

$$f(L) = R^{L-1}$$

Exponent number

Parameter α in the power expression of the quick opening valve model:

$$f(L) = L^{1/\alpha}$$

The exponent number determines how rapidly the valve open fraction approaches its maximum value. This parameter is active only when the **Valve opening characteristics** parameter is set to `Quick opening`. The exponent number must be greater than zero. The default value is 2.

Valve rangeability

Parameter R in the exponential expression of the equal-percentages valve model:

$$f(L) = R^{L-1}$$

The rangeability parameter determines the minimum valve opening fraction, a number generally different from zero. This parameter is active only when the **Valve opening characteristics** parameter is set to Equal-percentages. Typical values range from 20 to 50. The default value is 50.

Cross-sectional area at ports A and B

Area normal to the direction of flow at inlets A and B. This area is assumed the same for the two inlets. The default value is 0.01 m^2 .

Characteristic longitudinal length

Distance traversed by the fluid between inlets A and B. The default value is 0.1 m^2 .

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
- L — Physical signal input port for the valve opening control signal

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

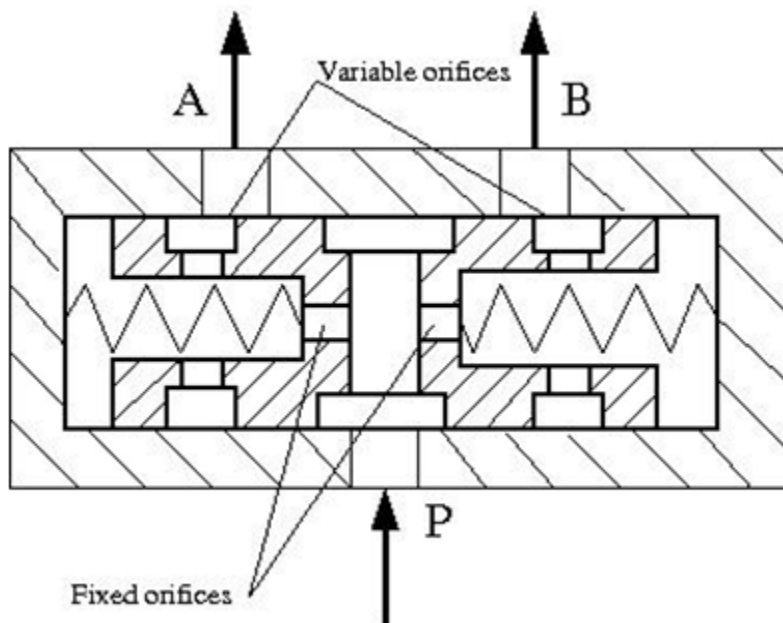


Library

Flow Control Valves

Description

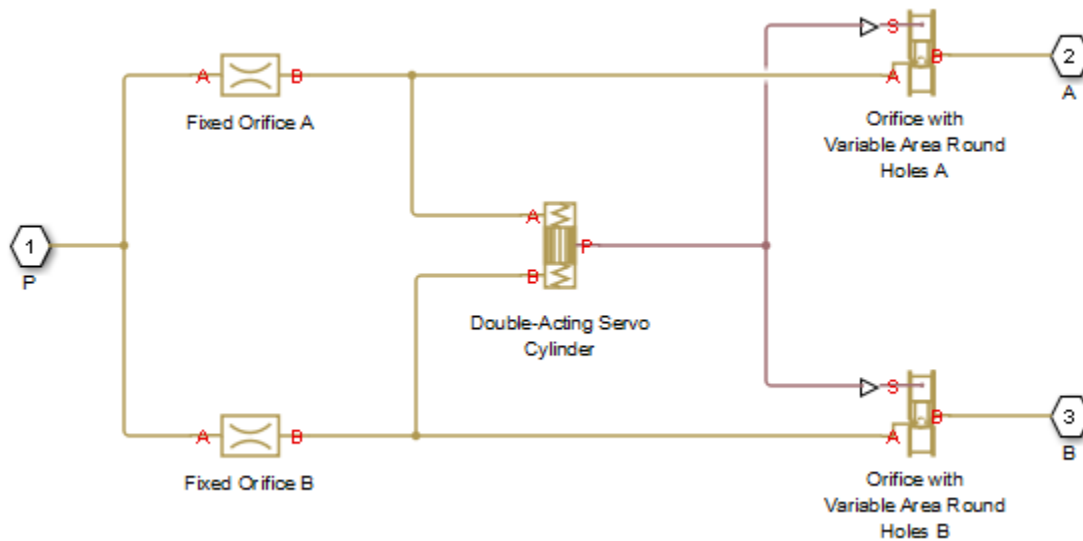
The Flow Divider block simulates a hydraulic two-path flow divider, which consists of a spring-centered spool installed in a case, as shown in the figure.



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 1-463
- “Variable Orifices Tab” on page 1-464

- “Servo Cylinder Tab” on page 1-466

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 $1e-4 \text{ m}^2$.

Fixed orifice B area

The cross-sectional passage area of the fixed orifice in the P–B path. The default value is $1e-4 \text{ m}^2$.

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 $1e-9 \text{ m}^2$.

Servo Cylinder Tab

Servo cylinder piston area

The face area of the piston in the servo cylinder. The default value is $1.6e-4 \text{ m}^2$.

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 $1e12 \text{ N/m/(m/s)}$.

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



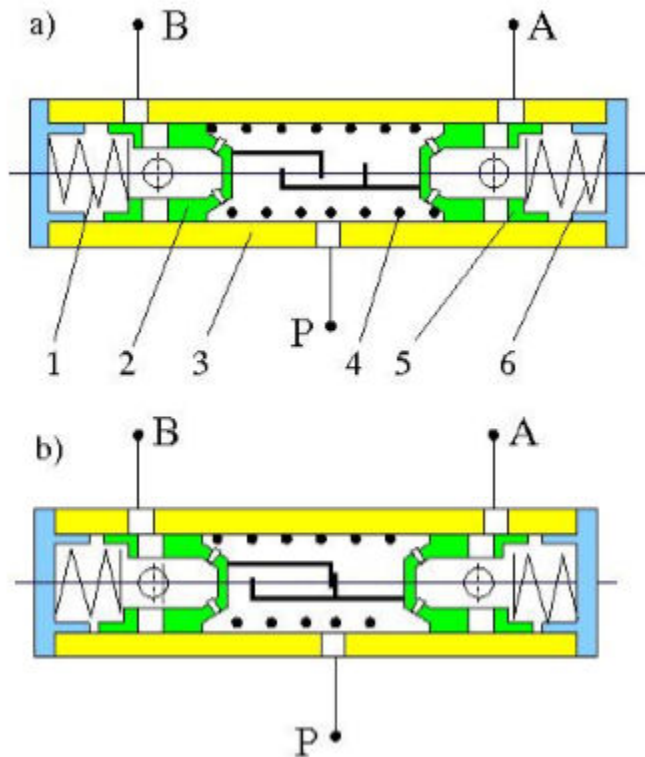
Library

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.

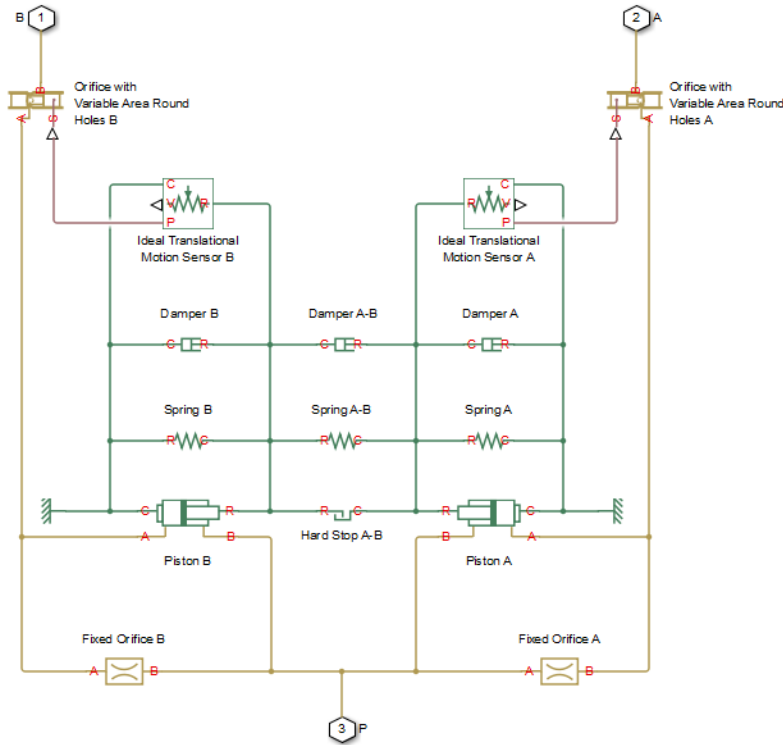
The figure shows a schematic for the flow divider-combiner valve: a) in the divider mode, and b) in the combiner mode.



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
Hard Stop A-B	Hard stop between pistons 2 and 5	hard_stop_A_B

Name in the block diagram	Purpose (numbers refer to the valve schematic)	Name in the actual component file
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 1-472
- “Pistons Tab” on page 1-473
- “Springs/Dampers Tab” on page 1-474
- “Variable Orifices Tab” on page 1-475
- “Hard Stop Between Pistons Tab” on page 1-476

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 $1.5e-5 \text{ m}^2$.

Fixed orifice B area

The cross-sectional passage area of the fixed orifice in piston 2 (the P–B path). The default value is $1.5e-5 \text{ m}^2$.

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 $2e-4 \text{ m}^2$.

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 $2e-4 \text{ m}^2$.

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 $1e12 \text{ s}^* \text{N}/\text{m}^2$.

Springs/Dampers Tab

Spring A rate

Spring rate of Spring A (spring 6). The default value is $1e3 \text{ N}/\text{m}$.

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 $150 \text{ N}/(\text{m}/\text{s})$.

Spring B rate

Spring rate of Spring B (spring 1). The default value is $1e3 \text{ N}/\text{m}$.

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 $150 \text{ N}/(\text{m}/\text{s})$.

Spring A-B rate

Spring rate of Spring A-B (spring 4). The default value is $1e3 \text{ N}/\text{m}$.

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 $150 \text{ N}/(\text{m}/\text{s})$.

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 $1e-9 \text{ m}^2$.

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 $1e8 \text{ N/m}$.

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

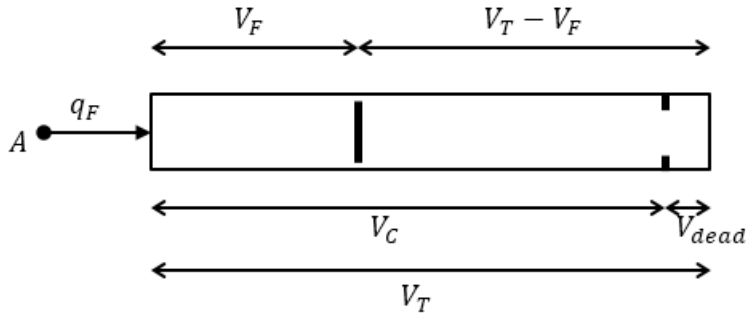
This block models a gas-charged accumulator. The accumulator consists of a precharged gas chamber and a fluid chamber. The fluid chamber is connected to a hydraulic system. The chambers are separated by a bladder, a piston, or any kind of a diaphragm.

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

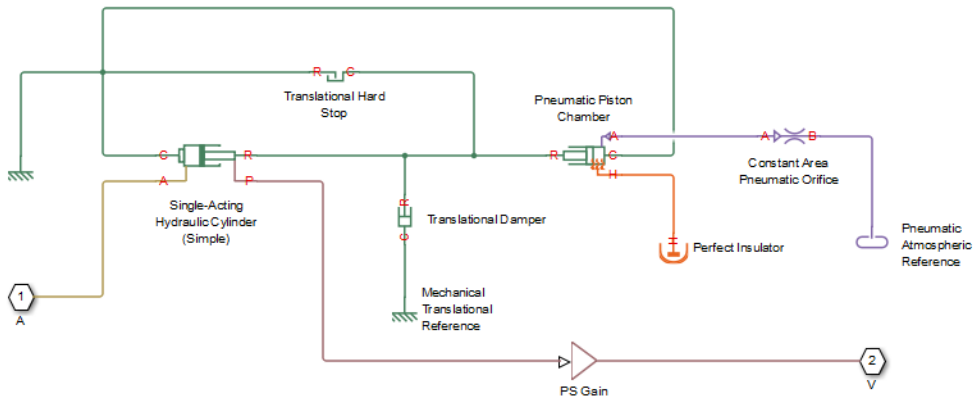
The flow rate into the accumulator is the rate of change of the fluid volume:

$$q_F = \frac{dV_F}{dt}$$

At $t = 0$, the initial condition is $V_F = V_{\text{init}}$, 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 1-481
- “Variables Tab” on page 1-482

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 $8e-3 \text{ m}^3$.

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 $4e-5 \text{ m}^3$.

Precharge pressure (gauge)

Pressure (gauge) in the gas chamber when the fluid chamber is empty. The default value is $10e5 \text{ Pa}$.

Specific heat ratio

Specific heat ratio (adiabatic index). To account for heat exchange, you can set it to a value 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 $1e10 \text{ Pa/m}^3$.

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 $1e10 \text{ Pa}\cdot\text{s/m}^6$.

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

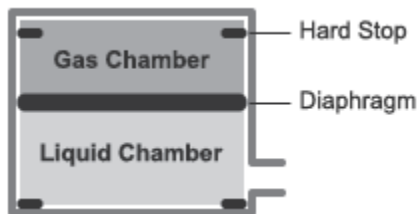


Library

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.

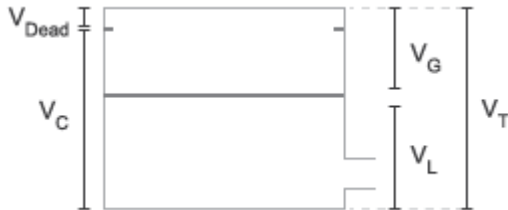
Chamber Volumes

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

$$\frac{dV_L}{dt} = \begin{cases} \frac{\dot{p}_L}{kp_{pr} V_T^k V_G^{(-k-1)} + K_s + K_d \dot{m}^+ / \rho_L}, & V_L \geq V_C \\ \frac{\dot{p}_L}{kp_{pr} V_T^k V_G^{(-k-1)} + K_s - K_d \dot{m}^- / \rho_L}, & V_L \leq 0 \\ \frac{\dot{p}_L}{kp_{pr} V_T^k V_G^{(-k-1)}}, & \text{Else} \end{cases},$$

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:

$$p_{HS} = \begin{cases} (V_L - V_C)(K_s + K_d \dot{m}^+ / \rho), & V_L \geq V_C \\ V_L (K_s - K_d \dot{m}^- / \rho), & V_L \leq 0 \\ 0, & \text{Else} \end{cases},$$

Energy Balance

The energy conservation equation in the liquid chamber volume is

$$\frac{d}{dt}(\rho_L u_L V_L) = \phi_A + \phi_H,$$

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 $8e-3 \text{ m}^3$.

Minimum gas volume

Remnant gas volume in the accumulator in a completely filled state. The default value is $4e-5 \text{ m}^3$.

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 $1e4 \text{ MPa/m}^3$.

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 $1e4 \text{ s*MPa/m}^6$.

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

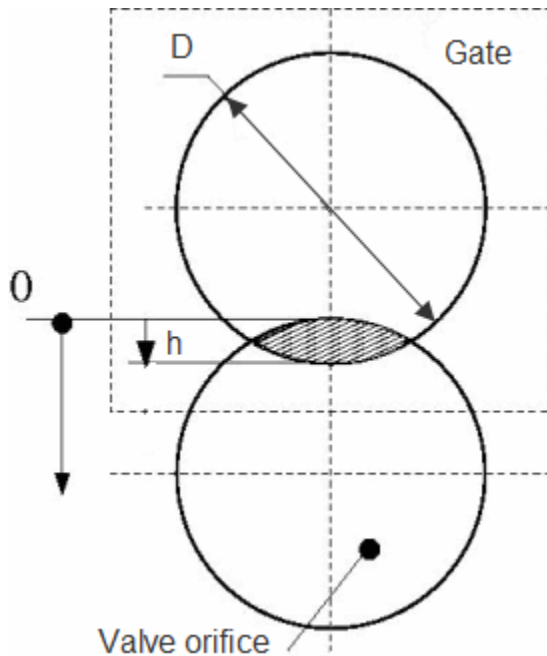


Library

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:

$$q = C_D \cdot A(h) \sqrt{\frac{2}{\rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

$$p = p_A - p_B$$

$$h = x_0 + x$$

$$A(h) = \begin{cases} A_{leak} & \text{for } h \leq 0 \text{ or } h > 2D \\ D^2 \cdot \left(\frac{\alpha}{2} - \frac{\sin(2\alpha)}{4} \right), \alpha = \alpha \cos \left| 1 - \frac{h}{D} \right| & \text{for } 0 < h \leq 2D \end{cases}$$

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:

$$p_{cr} = (p_{\text{avg}} + p_{\text{atm}})(1 - B_{\text{lam}})$$

$$p_{\text{avg}} = (p_A + p_B)/2$$

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:

$$p_{cr} = \frac{\rho}{2} \left(\frac{\text{Re}_{cr} \cdot v}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

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 $p = p_A - p_B$. 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 $1e-12 \text{ m}^2$.

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 (TL)

Flow control valve actuated by transverse motion of circular gate

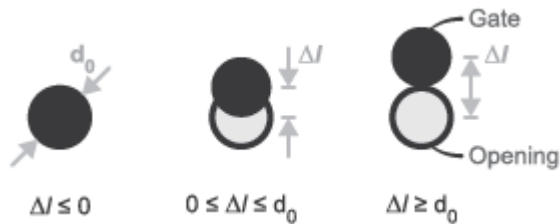


Library

Thermal Liquid/Valves/Flow Control Valves

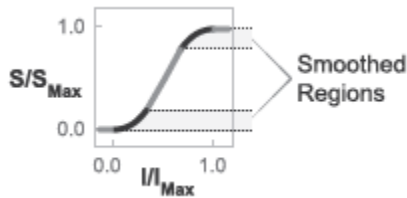
Description

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:

$$A_{Covered} = \frac{d_0^2}{2} \arccos\left(\frac{\Delta l}{d_0}\right) - \frac{\Delta l}{2} \sqrt{d_0^2 - (\Delta l)^2}$$

- Δl is the net displacement of the gate center relative to the orifice center.

$$\Delta l = \begin{cases} 0, & (x_0 + S_d) \leq 0 \\ d_0, & (x_0 + S_d) \geq d_0, \\ (x_0 + S), & \text{Else} \end{cases}$$

- 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

$$\lambda_L = 3\bar{\Delta}L^2 - 2\bar{\Delta}L^3$$

and

$$\lambda_R = 3\bar{\Delta}L_R^2 - 2\bar{\Delta}L_R^3$$

where:

$$\bar{\Delta}L = \frac{\Delta l}{\Delta l_{smooth}}$$

and

$$\bar{\Delta}L_R = \frac{\Delta l - (d_0 - \Delta l_{smooth})}{d_0 - (d_0 - \Delta l_{smooth})}$$

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.
- 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

$$S_R = \begin{cases} S_{Leak}, & \Delta l \leq 0 \\ S_{Leak} (1 - \lambda_L) + (A + S_{Leak}) \lambda_L, & \Delta l < \Delta l_{smooth} \\ A + S_{Leak}, & \Delta l \leq d_0 - \Delta l_{smooth} \\ (A + S_{Leak}) (1 - \lambda_R) + (S_{Leak} + S_{Max}) \lambda_R, & \Delta l < d_0 \\ S_{Leak} + S_{Max}, & \Delta l \geq d_0 \end{cases}$$

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:

$$\dot{m}_{cr} = Re_{cr} \mu_{Avg} \sqrt{\frac{\pi}{4} S_R}.$$

- 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:

$$PR_{Loss} = \frac{\sqrt{1 - (S_R/S)^2 (1 - C_d^2)} - C_d (S_R/S)}{\sqrt{1 - (S_R/S)^2 (1 - C_d^2)} + C_d (S_R/S)}.$$

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.

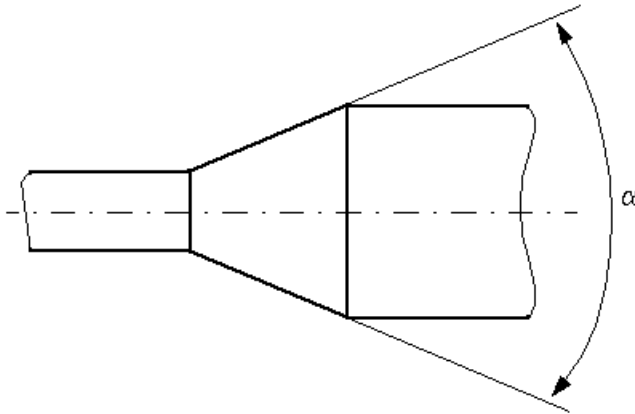
If you choose to apply the semi-empirical formulas, you provide geometric parameters of the resistance, and the pressure loss coefficient is determined according to the A.H. Gibson equations (see [1] on page 1-509 and [2] on page 1-509):

$$K_{GE} = \begin{cases} K_{cor} \left(1 - \frac{A_s}{A_L}\right)^2 \cdot 2.6 \sin \frac{\alpha}{2} & \text{for } 0 < \alpha \leq 45^\circ \\ K_{cor} \left(1 - \frac{A_s}{A_L}\right)^2 & \text{for } 45^\circ < \alpha < 180^\circ \end{cases}$$

$$K_{GC} = \begin{cases} K_{cor} \cdot 0.5 \left(1 - \frac{A_s}{A_L} \right)^{0.75} \cdot 1.6 \sin \frac{\alpha}{2} & \text{for } 0 < \alpha \leq 45^\circ \\ K_{cor} \cdot 0.5 \left(1 - \frac{A_s}{A_L} \right)^{0.75} \cdot \sqrt{\sin \frac{\alpha}{2}} & \text{for } 45^\circ < \alpha < 180^\circ \end{cases}$$

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

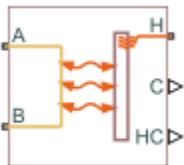
See Also

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

$$p_A - p_B = \Delta p_{\text{Loss,A}} - \Delta p_{\text{Loss,B}}.$$

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

$$\Delta p_{Loss,A} = \begin{cases} \dot{m}_A \mu_A (CP)_{Loss} Re_L \frac{1}{4D_{h,p} \rho_A S_{Min}}, & Re_A \leq Re_L \\ (CP)_{Loss} \frac{\dot{m}_A |\dot{m}_A|}{4\rho_A S_{Min}^2}, & Re_A \geq Re_T \end{cases},$$

while in the half volume adjacent to port B it is

$$\Delta p_{Loss,B} = \begin{cases} \dot{m}_B \mu_B (CP)_{Loss} Re_L \frac{1}{4D_{h,p} \rho_B S_{Min}}, & Re_B \leq Re_L \\ (CP)_{Loss} \frac{\dot{m}_B |\dot{m}_B|}{4\rho_B S_{Min}^2}, & Re_B \geq Re_T \end{cases},$$

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

$$\Delta p_{Loss,A} = \begin{cases} \dot{m}_A \mu_A \lambda \frac{(L_{press} + L_{add})}{4D_{h,p} \rho_A S_{Min}}, & Re_A \leq Re_L \\ f_{T,A} \frac{(L_{press} + L_{add})}{4D_{h,p}} \frac{\dot{m}_A |\dot{m}_A|}{\rho_A S_{Min}^2}, & Re_A \geq Re_T \end{cases},$$

while in the half volume adjacent to port B it is

$$\Delta p_{Loss,B} = \begin{cases} \dot{m}_B \mu_B \lambda \frac{(L_{press} + L_{add})}{4D_{h,p} \rho_B S_{Min}}, & \text{Re}_B \leq \text{Re}_L \\ f_{T,B} \frac{(L_{press} + L_{add}) \dot{m}_B |\dot{m}_B|}{4D_{h,p} \rho_B S_{Min}^2}, & \text{Re}_B \geq \text{Re}_T \end{cases},$$

where:

- L_{press} is the flow path length from inlet to outlet.
- L_{add} is the aggregate equivalent length of local resistances.
- $f_{T,A}$ and $f_{T,B}$ 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

$$f_{T,A} = \frac{1}{\left[-1.8 \log_{10} \left(\frac{6.9}{\text{Re}_A} + \frac{r}{3.7D_{h,p}} \right)^{1.11} \right]^2},$$

while in the half volume adjacent to port B it is

$$f_{T,B} = \frac{1}{\left[-1.8 \log_{10} \left(\frac{6.9}{\text{Re}_B} + \frac{r}{3.7D_{h,p}} \right)^{1.11} \right]^2},$$

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

$$\Delta p_{Loss,A} = \begin{cases} \dot{m}_A \mu_A \lambda \frac{L_{press}}{4D_{h,p}^2 \rho_A S_{Min}}, & \text{Re}_A \leq \text{Re}_L \\ f(\text{Re}_A) \frac{L_{press}}{4D_{h,p}} \frac{\dot{m}_A |\dot{m}_A|}{\rho_A S_{Min}^2}, & \text{Re}_A \geq \text{Re}_T \end{cases},$$

while in the half volume adjacent to port B it is

$$\Delta p_{Loss,B} = \begin{cases} \dot{m}_B \mu_B \lambda \frac{L_{press}}{4D_{h,p}^2 \rho_B S_{Min}}, & \text{Re}_B \leq \text{Re}_L \\ f(\text{Re}_B) \frac{L_{press}}{4D_{h,p}} \frac{\dot{m}_B |\dot{m}_B|}{\rho_B S_{Min}^2}, & \text{Re}_B \geq \text{Re}_T \end{cases},$$

where:

- λ is the shape factor for laminar flow viscous friction.
- $f(\text{Re}_A)$ and $f(\text{Re}_B)$ 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

$$\Delta p_{Loss,A} = \begin{cases} \dot{m}_A \mu_A \text{Eu}(\text{Re}_L) \text{Re}_L \frac{1}{4D_{h,p} \rho_A S_{Min}}, & \text{Re}_A \leq \text{Re}_L \\ \text{Eu}(\text{Re}_A) \frac{\dot{m}_A |\dot{m}_A|}{\rho_A S_{Min}^2}, & \text{Re}_A \geq \text{Re}_T \end{cases},$$

while in the half volume adjacent to port B it is

$$\Delta p_{Loss,B} = \begin{cases} \dot{m}_B \mu_B \text{Eu}(\text{Re}_L) \text{Re}_L \frac{1}{4D_{h,p} \rho_B S_{Min}}, & \text{Re}_B \leq \text{Re}_L \\ \text{Eu}(\text{Re}_B) \frac{\dot{m}_B |\dot{m}_B|}{\rho_B S_{Min}^2}, & \text{Re}_B \geq \text{Re}_T \end{cases},$$

where:

- $\text{Eu}(\text{Re}_L)$ 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

$$\frac{dp}{dt} \frac{du}{dp} + \frac{dT}{dt} \frac{du}{dT} = \phi_A + \phi_B + \phi_H,$$

where:

- u is the specific internal energy in the internal interface volume.
- Φ_A and Φ_B are the energy flow rates into the heat exchanger interface through ports A and B.
- Φ_H is the heat flow rate into the thermal liquid in the internal interface volume.

The specific internal energy derivative terms are

$$\frac{du}{dp} = \left[\frac{1}{\beta} (\rho u + p) - T\alpha \right] V$$

and

$$\frac{du}{dT} = \left[c_p - \alpha \left(u + \frac{p}{\rho} \right) \right] \rho V.$$

If the **Fluid dynamic compressibility** parameter is set to Off, the energy balance is

$$\frac{de}{dt} = \phi_A + \phi_B + Q,$$

where e is the total internal energy of the thermal liquid volume:

$$e = \rho u V.$$

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_{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

$$Nu_A = \begin{cases} Nu_L, & Re_A \leq Re_L \\ \frac{\left(\frac{f_{T,A}}{8}\right)(Re_A - 1000) Pr_A}{1 + 12.7 \left(\frac{f_{T,A}}{8}\right)^{1/2} (Pr_B^{2/3} - 1)}, & Re_A \geq Re_T \end{cases},$$

while at port B it is

$$Nu_B = \begin{cases} Nu_L, & Re_B \leq Re_L \\ \frac{\left(\frac{f_{T,B}}{8}\right)(Re_B - 1000) Pr_B}{1 + 12.7 \left(\frac{f_{T,B}}{8}\right)^{1/2} (Pr_B^{2/3} - 1)}, & Re_B \geq Re_T \end{cases},$$

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

$$Nu_A = j(Re_{A,\text{heat}}) Re_{A,\text{heat}} Pr_A^{1/3},$$

while at port B it is

$$Nu_B = j(Re_{B,\text{heat}}) Re_{B,\text{heat}} Pr_B^{1/3},$$

where:

- $j(Re_{A,\text{heat}})$ and $j(Re_{B,\text{heat}})$ 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.

- $Re_{A,heat}$ and $Re_{B,heat}$ 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

$$Re_{A,heat} = \frac{\dot{m}_A D_{h,heat}}{S_{Min} \mu_A},$$

and at port B as

$$Re_B = \frac{\dot{m}_B D_{h,heat}}{S_{Min} \mu_B}.$$

If the heat transfer specification is set to Tabulated data – Nusselt number vs. Reynolds number & Prandtl number, the Nusselt number at port A is

$$Nu_A = Nu(Re_{A,heat}, Pr_A),$$

while at port B it is

$$Nu_B = Nu(Re_{B,heat}, Pr_B).$$

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

$$D_{h,heat} = \frac{4(\pi/4)(D_o^2 - D_i^2)}{\pi D_i} = \frac{D_o^2 - D_i^2}{D_i},$$

while the hydraulic diameter for pressure calculations is

$$D_{h,p} = \frac{4(\pi/4)(D_o^2 - D_i^2)}{\pi(D_i + D_o)} = D_o - D_i,$$

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 $1e-4 \text{ m}^2\text{K/W}$.

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

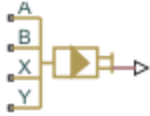
See Also

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

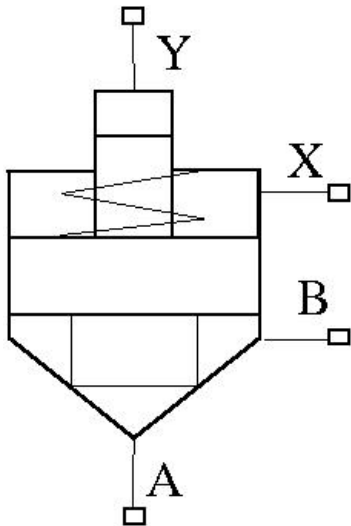


Library

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:

$$F_P = p_A \cdot A_A + p_B \cdot A_B - p_X \cdot A_X - p_Y \cdot A_Y$$

$$x_s = \begin{cases} 0 & \text{for } F_P \leq F_{PR} \\ (F_P - F_{PR}) / k_{spr} & \text{for } F_{PR} < F_P < F_{PR} + k_{spr} \cdot x_{\max} \\ x_{\max} & \text{for } F_P \geq F_{PR} + k_{spr} \cdot x_{\max} \end{cases}$$

$$x = x_s \frac{1}{T_s + 1}$$

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 $3.3e-4 \text{ m}^2$.

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 $1.65e-4 \text{ m}^2$.

Preload force

Spring preload force. The default value is 26 N .

Spring rate

Spring rate. The default value is $1.4e4 \text{ N/m}$.

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 $3.3 \times 10^{-4} \text{ m}^2$.

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 $1.4 \times 10^4 \text{ N/m}$.

Poppet stroke

Maximum poppet stroke. The parameter value must be greater than or equal to zero. The default value is $5 \times 10^{-3} \text{ m}$.

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

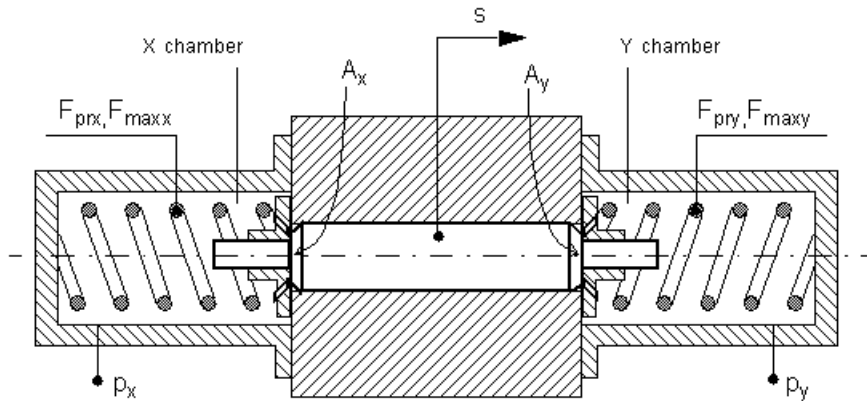


Library

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:

$$F = p_x \cdot A_x - p_y \cdot A_y$$

$$L_x = \frac{str_x}{F_{maxx} - F_{prx}}$$

$$L_y = \frac{str_y}{F_{maxy} - F_{pry}}$$

If $F \geq 0$,

$$s = \begin{cases} 0 & \text{for } F \leq F_{pry} \\ L_y \cdot (F - F_{pry}) \cdot or & \text{for } F_{pry} < F < F_{maxy} \\ str_y \cdot or & \text{for } F \geq F_{maxy} \end{cases}$$

If $F < 0$,

$$s = \begin{cases} 0 & \text{for } |F| \leq F_{prx} \\ -L_x \cdot (|F| - F_{prx}) \cdot or & \text{for } F_{prx} < |F| < F_{maxx} \\ -str_x \cdot or & \text{for } |F| \geq F_{maxx} \end{cases}$$

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 $2e-4 \text{ m}^2$.

Piston area at port Y

Effective piston area at port Y. The parameter value must be greater than zero. The default value is $2e-4 \text{ m}^2$.

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 $5e-3 \text{ m}$.

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 $5e-3 \text{ m}$.

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

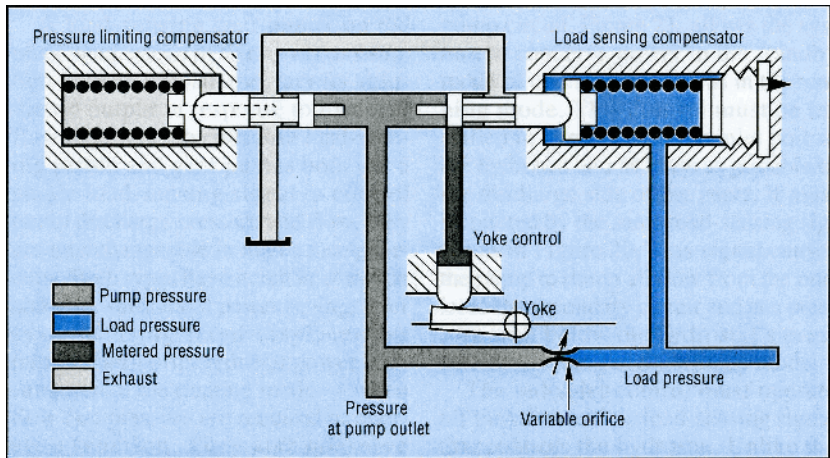
Hydraulic conserving port associated with the valve Y chamber.

P

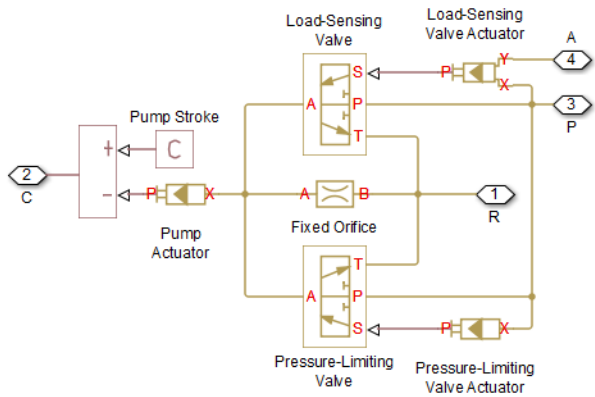
Physical signal port that outputs piston displacement.

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 1-539). 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:

$$E = E_l \frac{1 + \alpha \left(\frac{p_a}{p_a + p} \right)^{1/n}}{1 + \alpha \frac{p_a^{1/n}}{n \cdot (p_a + p)^{\frac{n+1}{n}}} E_l}$$

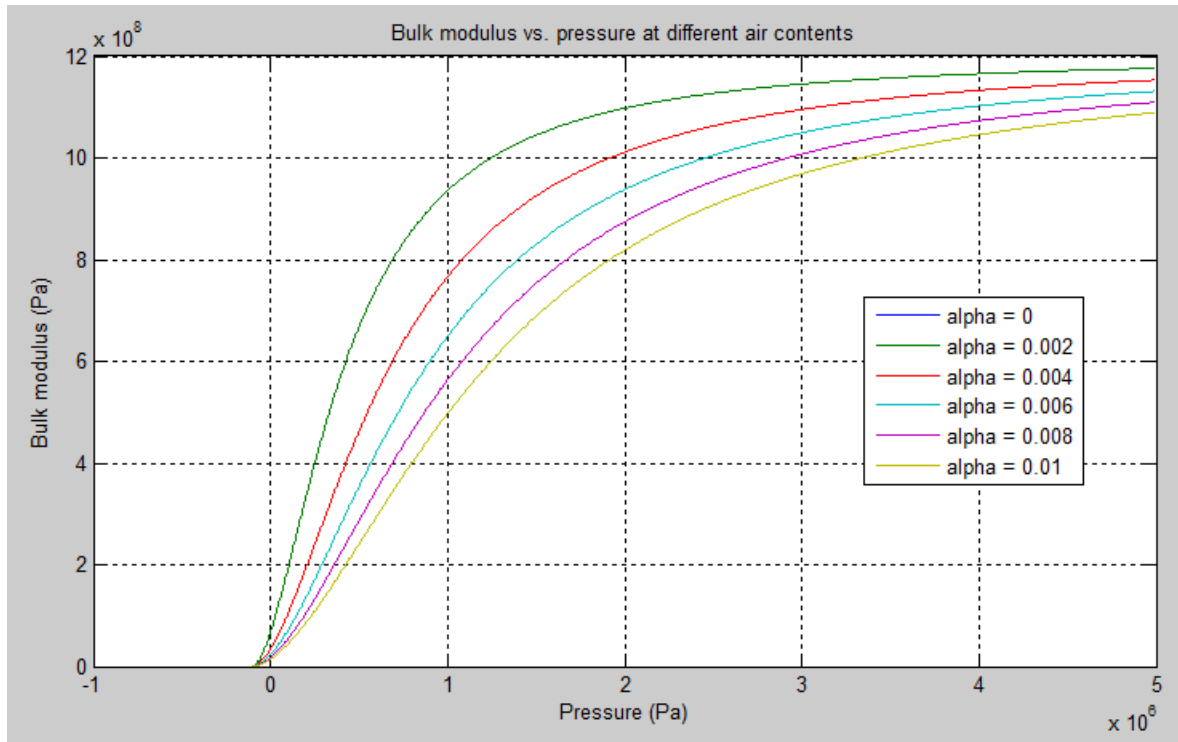
where

E_L	Pure liquid bulk modulus
p_a	Atmospheric pressure
α	Relative gas content at atmospheric pressure, $\alpha = V_G/V_L$
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 $p \rightarrow p_a$, thus considerably slowing down further pressure change.

At high pressure, $p \gg p_a$, 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 1-544 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:

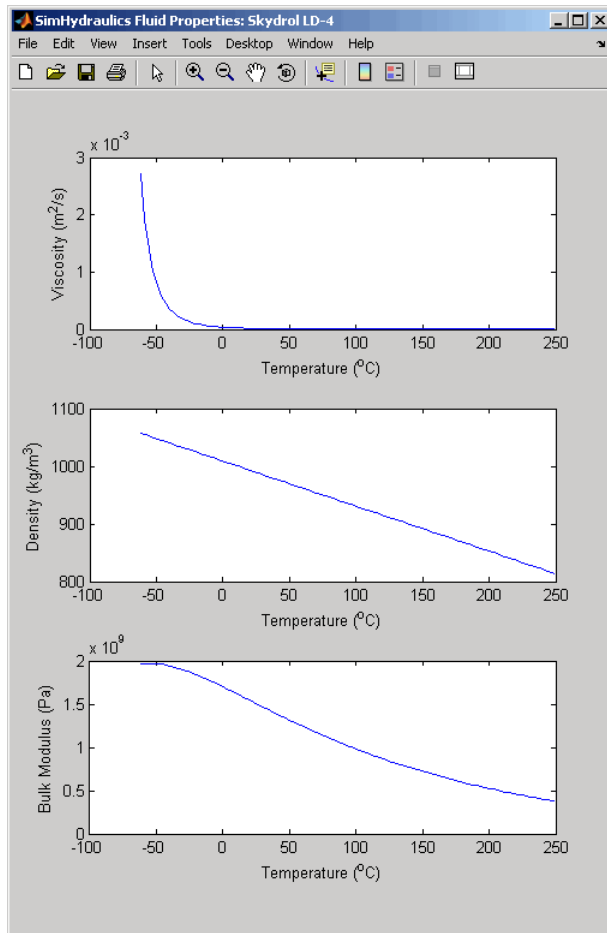
```
props =  
  
    skydrol_ld_4: [1x1 struct]  
    skydrol_500_4: [1x1 struct]  
    skydrol_5: [1x1 struct]  
    hy_jet: [1x1 struct]
```

```
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



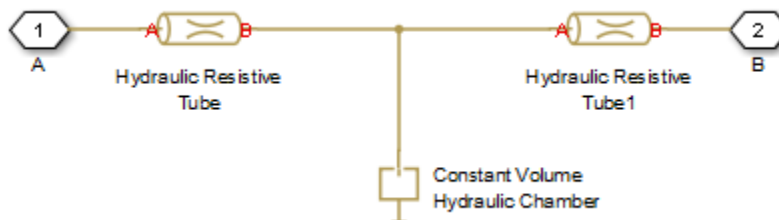
Library

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 $1e-4 \text{ m}^2$.

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 1-551]. 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 1.5×10^{-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 2×10^{-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



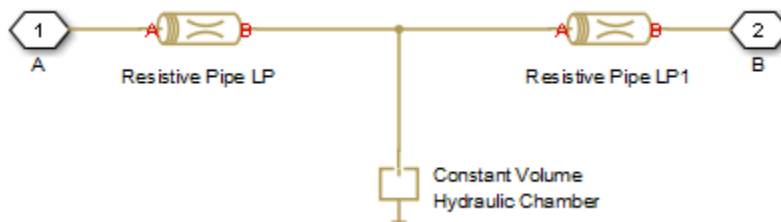
Library

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 1-553
- “Wall Compliance Tab” on page 1-555
- “Vertical Position Tab” on page 1-555

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 $1e-4 \text{ m}^2$.

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 1-551]. 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 1.5×10^{-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 0m.

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 0m.

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



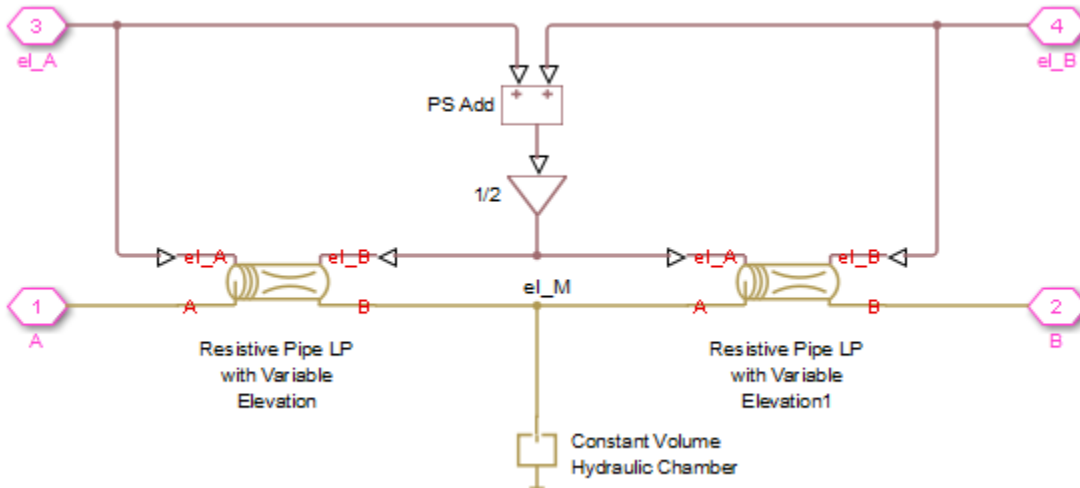
Library

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 1-560
- “Wall Compliance Tab” on page 1-562

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 $1e-4 \text{ m}^2$.

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 1-551]. 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 1.5×10^{-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.

e1_A

Physical signal input port that controls pipe elevation at port A.

e1_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

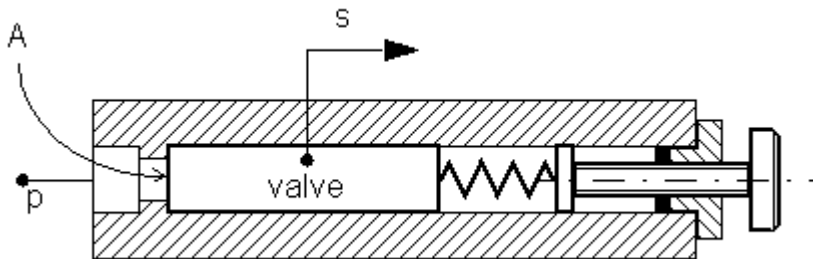


Library

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:

$$F = p \cdot A$$

$$L = \frac{\textit{stroke}}{F_{\max} - F_{pr}}$$

$$s = \begin{cases} 0 & \text{for } F \leq F_{pr} \\ L \cdot (F - F_{pr}) \cdot or & \text{for } F_{pr} < F < F_{\max} \\ \textit{stroke} \cdot or & \text{for } F \geq F_{\max} \end{cases}$$

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
\textit{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 $2e-4 \text{ m}^2$.

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 $5e-3 \text{ m}$.

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

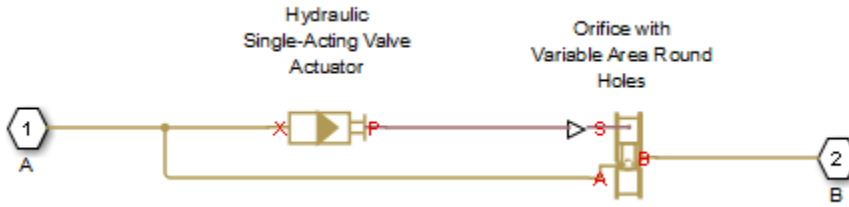
Hydraulic conserving port associated with the valve chamber.

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

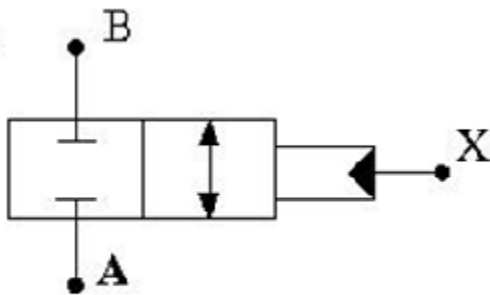


Library

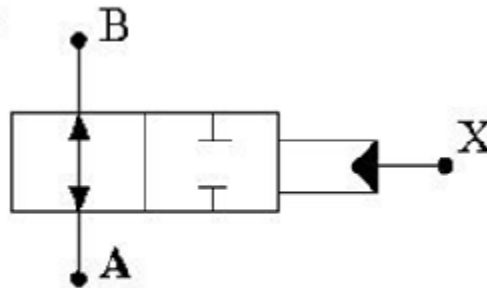
Directional Valves

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

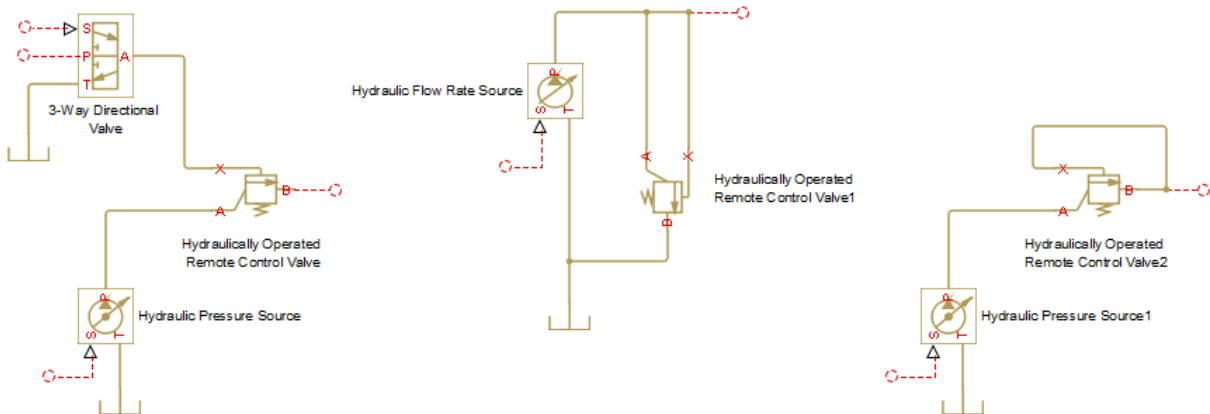


Normally open
valve configuration

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:

$$q = C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

For the normally closed valve, the instantaneous orifice passage area $A(p)$ is computed with the equations:

$$A(p) = \begin{cases} A_{leak} & \text{for } p_p \leq p_{crack} \\ A_{leak} + gain \cdot (p_p - p_{crack}) & \text{for } p_{crack} < p_p < p_{max} \\ A_{max} & \text{for } p_p \geq p_{max} \end{cases}$$

For the normally open valve, the equations are similar:

$$A(p) = \begin{cases} A_{max} & \text{for } p_p \leq p_{crack} \\ A_{max} - gain \cdot (p_p - p_{crack}) & \text{for } p_{crack} < p_p < p_{max} \\ A_{leak} & \text{for } p_p \geq p_{max} \end{cases}$$

The rest of the equations apply to both valve configurations:

$$gain = \frac{A_{max} - A_{leak}}{p_{max} - p_{crack}}$$

$$p = p_A - p_B$$

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:

$$p_{cr} = \frac{\rho}{2} \left(\frac{Re_{cr} \cdot v}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

where

D_H	Orifice hydraulic diameter
v	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 $p = p_A - p_B$.

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 $1e-4 \text{ m}^2$.

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 $3e4 \text{ Pa}$.

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 $1.2e5 \text{ Pa}$.

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 $1e-12 \text{ m}^2$.

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 $1e-12 \text{ m}^2$.

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: Fluid Network Interfaces

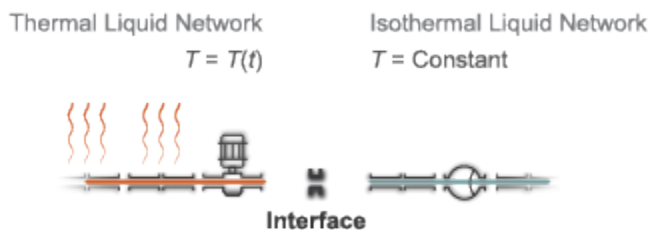


Description

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:

$$\dot{m}_A + q_B \rho_B = 0,$$

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:

$$p_A = p_B + 1\text{atm},$$

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:

$$T_I = T_{\text{Hyd}},$$

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 m² (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

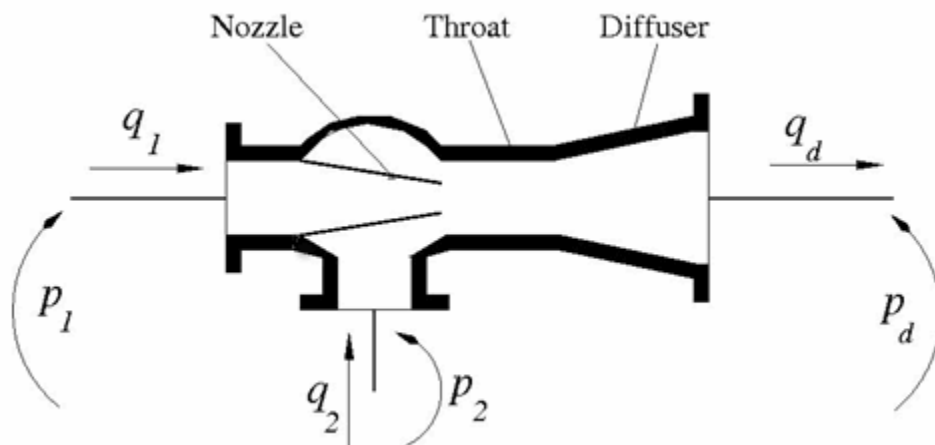


Library

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 1-584]:

$$q_1 = \frac{A_n}{\sqrt{1 + K_n}} \sqrt{\frac{2}{\rho} (p_1 - p_0)}$$

$$q_2 = \frac{A_n \cdot c}{\sqrt{1 + K_{en}}} \sqrt{\frac{2}{\rho} (p_2 - p_0)}$$

$$p_d - p_0 = Zb^2 \left(\frac{2}{b} + \frac{2}{1-b} M^2 - (1+M)^2 \cdot (1 + K_{th} + K_{di} + a^2) \right)$$

$$b = \frac{A_n}{A_{th}}$$

$$c = \frac{1-b}{b}$$

$$Z = \rho \frac{V_n^2}{2} = \rho \frac{q_1^2}{2A_n^2}$$

$$M = \frac{q_2}{q_1}$$

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” on page 1-581 describes the nozzle, “Equation 1-9” on page 1-581 – throat entry, and “Equation 1-10” on page 1-581 – 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 1-584].

The pump parameters are closely related to each other, and the methodology described in [1 on page 1-584] 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 cm².

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 cm².

Diffuser inlet/outlet area ratio

The ratio between the inlet and outlet diffuser areas. For a standard pump with a $5^\circ - 7^\circ$ 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

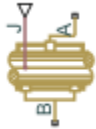
See Also

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

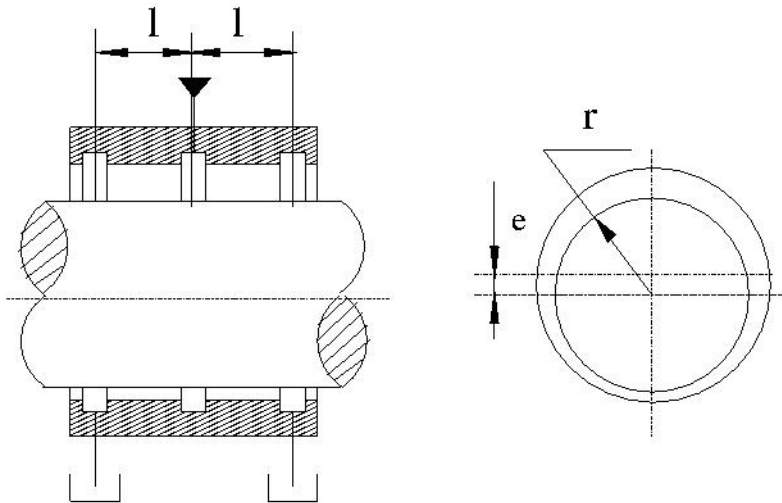


Library

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 1-588):

$$q = \frac{\pi p r c^3}{3\mu l} (1 + 1.5\epsilon^2)$$

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, $\epsilon = e / r$
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 $p = p_A - p_B$. 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:

$$p = p_A - p_B$$

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:

$$q = A \sqrt{\frac{2}{K \cdot \rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

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:

$$p = K(\text{Re})\rho \frac{q|q|}{2A^2}$$

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:

$$\text{Re} = \frac{q \cdot D_H}{A \cdot \nu}$$

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:

$$p_{cr} = K \frac{\rho}{2} \left(\frac{\text{Re}_{cr} \cdot v}{D_H} \right)^2$$

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 $p = p_A - p_B$.

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 1-592
- “Variables Tab” on page 1-594

Parameters Tab

Resistance area

The smallest passage area. The default value is $1e-4 \text{ m}^2$.

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 1-596). 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

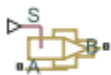
See Also

Elbow | Gradual Area Change | Pipe Bend | Sudden Area Change | T-junction

Introduced in R2006a

Needle Valve

Hydraulic needle valve

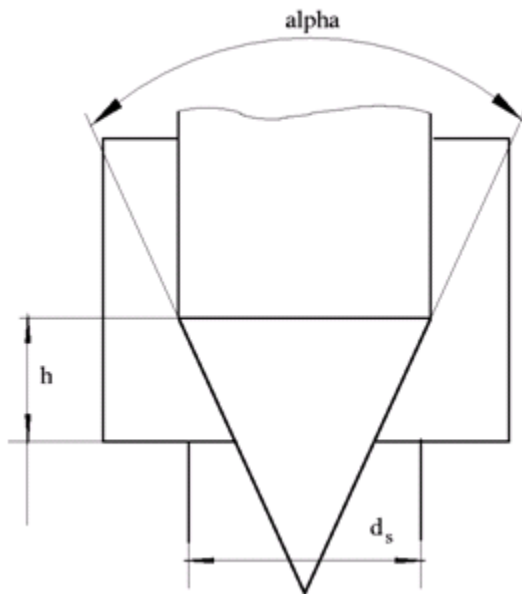


Library

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:

$$q = C_D \cdot A(h) \sqrt{\frac{2}{\rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

$$p = p_A - p_B$$

$$h = x_0 + x$$

$$h_{\max} = d_s \frac{1 - \sqrt{1 - \cos \frac{\alpha}{2}}}{\sin \alpha}$$

$$A(h) = \begin{cases} A_{leak} & \text{for } h \leq 0 \\ \pi \cdot \left(d_s - h \cdot \sin \frac{\alpha}{2} \cdot \cos \frac{\alpha}{2} \right) \cdot h \cdot \sin \frac{\alpha}{2} + A_{leak} & \text{for } 0 < h < h_{\max} \\ A_{\max} + A_{leak} & \text{for } h \geq h_{\max} \end{cases}$$

$$A_{\max} = \frac{\pi d_s^2}{4}$$

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:

$$p_{cr} = \frac{\rho}{2} \left(\frac{Re_{cr} \cdot v}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

where

D_H	Valve instantaneous hydraulic diameter
v	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

$p = p_A - p_B$. 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 $1e-12 \text{ m}^2$.

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.

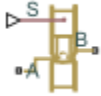
See Also

Ball Valve | Gate Valve | Poppet Valve | Pressure-Compensated Flow Control Valve

Introduced in R2006a

Orifice with Variable Area Round Holes

Hydraulic variable orifice shaped as set of round holes drilled in sleeve



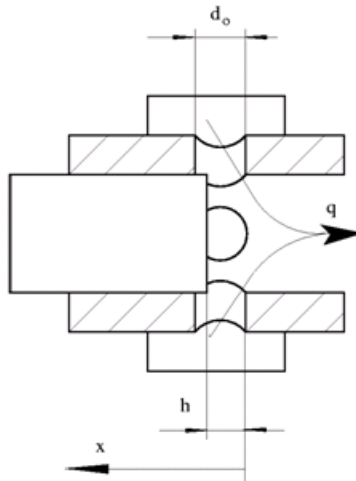
Library

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



The flow rate is determined according to the following equations:

$$q = C_D \cdot A(h) \sqrt{\frac{2}{\rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

$$p = p_A - p_B$$

$$h = x_0 + x \cdot \text{or}$$

$$A(h) = \begin{cases} A_{leak} & \text{for } h \leq 0 \\ \left(\frac{1}{8} z \cdot d_0^2 \left(2 \arccos \left(1 - \frac{2h}{d_0} \right) - \sin \left(2 \arccos \left(1 - \frac{2h}{d_0} \right) \right) \right) \right) + A_{leak} & \text{for } 0 < h < d_0 \\ A_{max} + A_{leak} & \text{for } h \geq d_0 \end{cases}$$

$$A_{max} = \frac{\pi d_0^2}{4} \cdot z$$

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
d_0	Hole diameter
z	Number of holes
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
A_{max}	Fully open orifice passage 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:

$$p_{cr} = \frac{\rho}{2} \left(\frac{Re_{cr} \cdot \nu}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

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

$p = p_A - p_B$. 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

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 $1e-15 \text{ m}^2$.

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:

$$q = C_D \cdot A(h) \sqrt{\frac{2}{\rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

$$p = p_A - p_B$$

$$h = x_0 + x \cdot or$$

$$A(h) = \begin{cases} A_{leak} & \text{if } b \cdot h < A_{leak} \\ b \cdot h & \text{otherwise} \end{cases}$$

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:

$$p_{cr} = \frac{\rho}{2} \left(\frac{Re_{cr} \cdot v}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

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

$p = p_A - p_B$. 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 $1e-12 \text{ m}^2$.

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

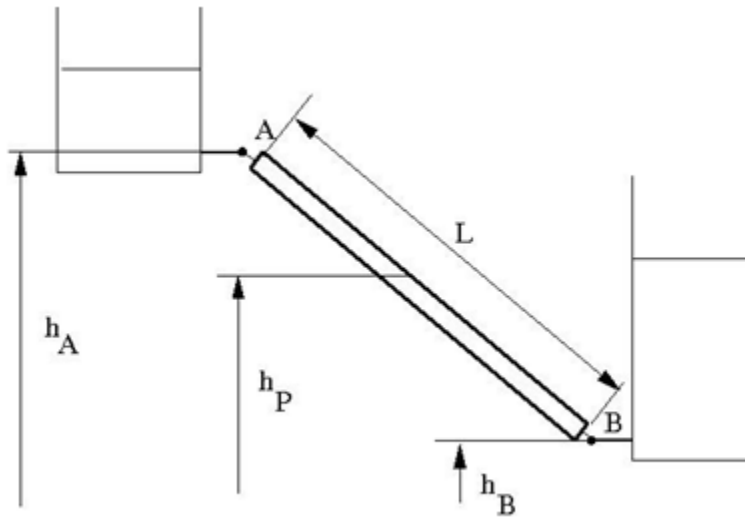
Low-Pressure Blocks

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 1-619. 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 1-623]:

$$f = \frac{1}{\left(-1.8 \log_{10} \left(\frac{6.9}{Re} + \left(\frac{k / D_H}{3.7} \right)^{1.11} \right) \right)^2}$$

$$Re = \frac{q \cdot D_H}{A \cdot \nu}$$

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

$$f = \frac{8}{Re}$$

where s is the geometrical shape factor, or Poiseuille number [1 on page 1-623], 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:

$$q_A = \begin{cases} 0 & \text{for } p \geq 0, V < V_{\min} \\ A \sqrt{\frac{1}{f} \frac{D_H}{L_{ef}}} \sqrt{\frac{2}{\rho}} \cdot p & \text{for } p \geq 0, V \geq V_{\min}, Re \geq Re_{cr} \\ 2 \cdot A \cdot \frac{D_H^2}{s \cdot v \cdot \rho \cdot L_{ef}} \cdot p & \text{for } p \geq 0, V \geq V_{\min}, Re < Re_{cr} \\ 0 & \text{for } p < 0, L_F \leq L \\ 2 \cdot A \cdot \frac{D_H^2}{s \cdot v \cdot \rho \cdot L_{ef}} \cdot p & \text{for } p < 0, L_F > L, Re < Re_{cr} \\ A \sqrt{\frac{1}{f} \frac{D_H}{L_{ef}}} \sqrt{\frac{2}{\rho}} \cdot |p| & \text{for } p < 0, L_F > L, Re \geq Re_{cr} \end{cases}$$

$$q_B = \begin{cases} 0 & \text{for } p \geq 0, V_F < V_{\min} \\ A \sqrt{\frac{1}{f} \frac{D_H}{L_{ef}}} \sqrt{\frac{2}{\rho}} \cdot |p| \cdot \text{sign}(p) & \text{for } p \geq 0, V_F \geq V_{\min}, Re \geq Re_{cr} \\ 2 \cdot A \cdot \frac{D_H^2}{s \cdot v \cdot \rho \cdot L_{ef}} \cdot p & \text{for } p \geq 0, V_F \geq V_{\min}, Re < Re_{cr} \end{cases}$$

$$q_A - q_B = \frac{dV_p}{dt}$$

$$L_F = \frac{V_p}{A}$$

$$L_{ef} = L_F \left(1 + \frac{L_{ad}}{L} \right)$$

$$h_p = L_F \frac{h_A - h_B}{L}$$

$$p = p_A - p_B + \rho \cdot g \cdot h_p$$

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 1-620
- “Vertical Position Tab” on page 1-622

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 1-623]. 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 $1e-4 \text{ m}^3$.

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 $5e-5 \text{ m}$.

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

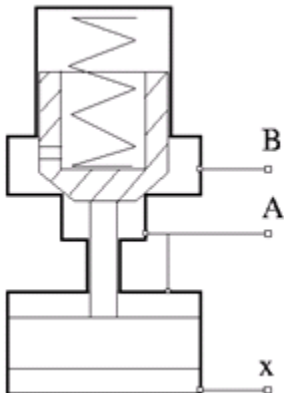


Library

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 determined as

$$F = p_A \cdot A_A + p_X \cdot A_X - p_B \cdot A_B$$

where

p_A, p_B	Gauge pressures at the valve terminals
p_X	Gauge pressure 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

This equation is commonly used in a slightly modified form

$$p_e = p_A + p_X \cdot k_p - p_B$$

where $k_p = A_X / A_A$ is usually referred to as pilot ratio and p_e is the equivalent pressure differential across the poppet. 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 (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:

$$q = C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

$$p_e = p_A + p_X \cdot k_p - p_B$$

$$A(p) = \begin{cases} A_{leak} & \text{for } p_e \leq p_{crack} \\ A_{leak} + k \cdot (p_e - p_{crack}) & \text{for } p_{crack} < p_e < p_{max} \\ A_{max} & \text{for } p_e \geq p_{max} \end{cases}$$

$$k = \frac{A_{max} - A_{leak}}{p_{max} - p_{crack}}$$

$$p = p_A - p_B$$

$$p_{cr} = \frac{\rho}{2} \left(\frac{Re_{cr} \cdot v}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

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

$$p = p_A - p_B.$$

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.
- No flow consumption is associated with the pilot chamber.

Parameters

Maximum passage area

Valve passage maximum cross-sectional area. The default value is $1e-4 \text{ m}^2$.

Cracking pressure

Pressure level at which the orifice of the valve starts to open. The default value is $3e4 \text{ Pa}$.

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 $1.2e5 \text{ Pa}$.

Pilot ratio

Ratio between effective area in the pilot chamber to the effective area in the inlet chamber. The default value is 5.

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 $1e-12 \text{ m}^2$.

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 $1e-12 \text{ m}^2$.

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

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 1-636, 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.

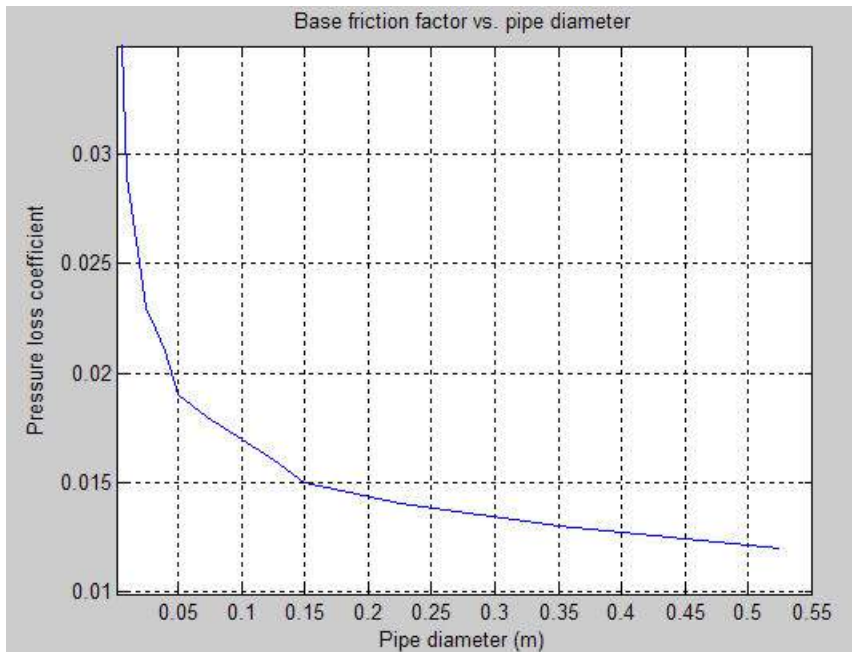
The pressure loss coefficient is determined according to recommendation provided in [1] on page 1-636:

$$K = K_d \cdot K_r \cdot K_\alpha$$

where

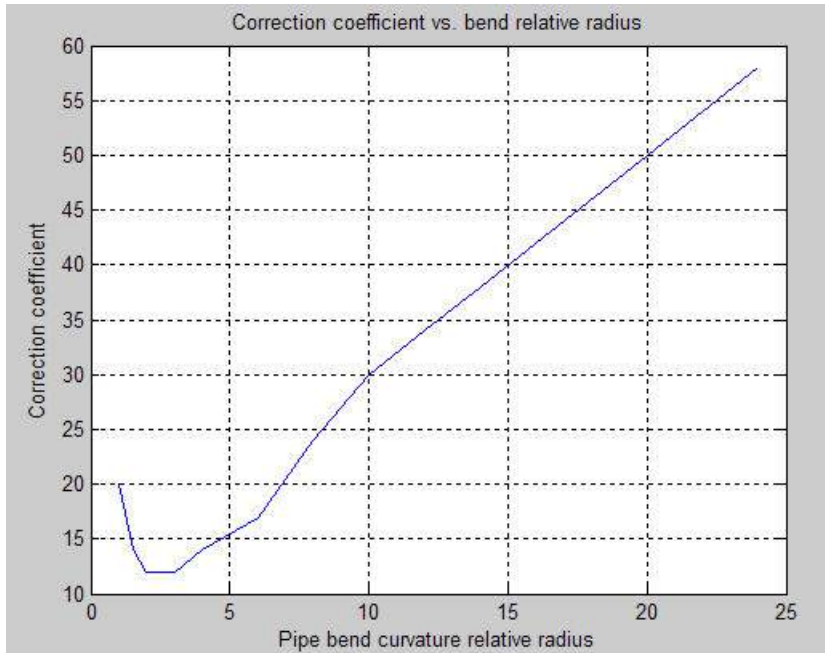
K_d	Base friction factor coefficient
K_r	Correction coefficient accounting for the bend curvature
K_α	Correction coefficient accounting for the bend angle

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

$$r = \text{bend radius} / \text{pipe diameter}$$

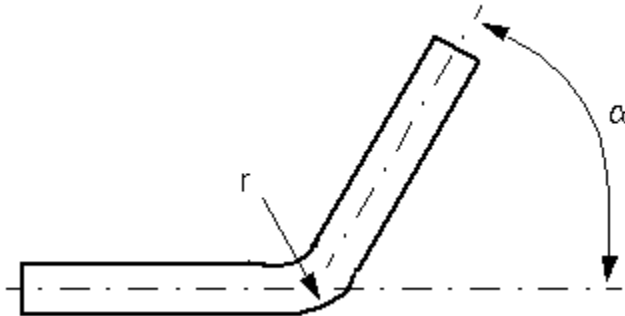
Note For pipes with the bend curvature relative radius value outside the range of $1 > r > 24$, correction coefficients are determined by extrapolation.

Correction for non-90° bends is performed with the empirical formula (see [2] on page 1-636, Fig. 4.6):

$$K_{\alpha} = \alpha(0.0142 - 3.703 \cdot 10^{-5} \alpha)$$

where

α	Bend angle in degrees ($0 \leq \alpha \leq 180$)
----------	--



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 1.5×10^{-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)

Rigid or flexible thermal liquid conduit

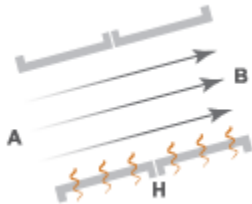


Library

Thermal Liquid/Pipes & Fittings

Description

The Pipe (TL) block models the flow dynamics of a thermal liquid through a pipe segment or pipeline. The pipe model accounts for viscous friction losses and convective heat transfer with the pipe wall. Dynamic compressibility, fluid inertia, and pipe wall compliance are optional model features selectable through the block dialog box.



Pipe Schematic

The pipe wall can be rigid or flexible. A flexible pipe wall can expand and contract radially in response to internal pressure fluctuations. This radial compliance allows the pipe fluid volume to vary during simulation. The **Pipe wall specification** parameter provides the choice of rigid or flexible pipe walls.

The pipe consists of one or more pipe segments, each comprising a fluid volume with distinct pressure and temperature states. These states evolve dynamically during

simulation and depend on the initial values specified in the block dialog box. The initial state inputs can be scalar values or vectors with sizes consistent with the number of pipe segments. A scalar input means that the value specified applies to all pipe segments.

Block Variants

The pipe inlets can be at different elevations. The elevation gain from port **A** to port **B** can be constant or variable. To specify a variable elevation gain, change the block variant to `Variable elevation`. You can change variants by right-clicking the block and selecting **Simscape > Block choices**. Revert to the `Constant elevation` variant to model a pipe with a constant elevation gain between the ports.

Pipe Geometry

The pipe diameter is fixed for rigid pipe walls and variable for flexible pipe walls. A rigid pipe can have a noncircular cross section with the hydraulic diameter taken as the pipe diameter:

$$D = D_H,$$

where:

- D is the pipe diameter.
- D_H hydraulic diameter specified in the block dialog box.

A flexible pipe is assumed circular in cross section. The pipe wall can expand and contract in the radial direction. The rate of expansion or contraction is given by

$$\frac{dD}{dt} = \frac{K_p (p - p_{Atm})}{\tau} - \frac{D - D_{Nom}}{\tau},$$

where:

- K_p is the static pressure-diameter compliance.
- p is the internal pipe pressure.
- p_{Atm} is the environment pressure external to the pipe.
- D_{Nom} is the nominal, or specified, pipe diameter.
- τ is the characteristic response time for pipe diameter changes.

The static pressure-diameter compliance is defined as

$$K_p = \frac{D_0}{E} \left(\frac{D_{Ext,0}^2 + D_0^2}{D_{Ext,0}^2 - D_0^2} + \nu \right),$$

where:

- D_0 is the initial internal diameter.
- $D_{Ext,0}$ is the initial external diameter.
- E is the pipe wall elasticity modulus.
- ν is the pipe wall Poisson ratio.

Mass Balance

The mass balance in the pipe 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 pipe through ports A and B.

If the **Fluid dynamic compressibility** parameter is set to `On` and the **Pipe wall specification** parameter is set to `Rigid`, the mass balance equation becomes

$$\dot{m}_A + \dot{m}_B = \rho V \left(\frac{1}{\beta} \frac{dp}{dt} - \alpha \frac{dT}{dt} \right),$$

where:

- V is the pipe volume.
- α is the isobaric thermal expansion coefficient.
- β is the isothermal bulk modulus.
- p is the thermal liquid pressure.
- T is the thermal liquid temperature.

If the **Fluid dynamic compressibility** parameter is set to `On` and the **Pipe wall specification** parameter is set to `Flexible`, the mass balance equation becomes

$$\dot{m}_A + \dot{m}_B = \rho V \left(\frac{1}{\beta} \frac{dp}{dt} - \alpha \frac{dT}{dt} \right) + \rho \frac{dV}{dt}.$$

Momentum Balance

The momentum balance in the pipe depends on the dynamic compressibility and fluid inertia settings. If the **Fluid dynamic compressibility** parameter is set to `On` and the **Fluid inertia** parameter is set to `Off`, the momentum conservation equation in the half pipe adjacent to port A is

$$(p_A - p) = \Delta p_{Loss,A} + \frac{\rho g \Delta z}{2},$$

while in the half pipe adjacent to port B it is

$$(p_B - p)S = \Delta p_{Loss,B} - \frac{\rho g \Delta z}{2}.$$

In the equations:

- p_A and p_B are the pressures at ports A and B.
- S is the cross-sectional area of the pipe.
- $\Delta p_{Loss,A}$ and $\Delta p_{Loss,B}$ are the pressure losses due to viscous friction in the port-A and port-B half pipes.
- ρ is the mass density in the pipe volume.

If the **Fluid dynamic compressibility** and **Fluid inertia** parameters are both set to `On`, the momentum conservation equation in the half pipe adjacent to port A becomes

$$(p_A - p) = \Delta p_{Loss,A} + \frac{\rho g \Delta z}{2} + \frac{\dot{m}_A L}{2S},$$

while in the half pipe adjacent to port B it becomes

$$(p_B - p) = \Delta p_{Loss,B} - \frac{\rho g \Delta z}{2} + \frac{\dot{m}_B L}{2S}.$$

In the equations:

- L is the pipe length.

If the **Fluid dynamic compressibility** parameter is set to `Off`, the block computes the mass balance between ports A and B directly as

$$p_A - p_B = \Delta p_{Loss,A} - \Delta p_{Loss,B} + \rho g \Delta z.$$

Viscous Friction Losses

The form of the pressure loss terms depends on the flow regime in the pipe. For laminar flows, the pressure loss in the half pipe adjacent to port A is

$$\Delta p_{Loss,A} = \frac{\dot{m}_A \mu_A \lambda L_{Eff}}{4D^2 \rho_A S},$$

while in the half pipe adjacent to port B it is

$$\Delta p_{Loss,B} = \frac{\dot{m}_B \mu_B \lambda L_{Eff}}{4D^2 \rho_B S},$$

where:

- μ_A and μ_B are the fluid dynamic viscosities at ports A and B.
- λ is the pipe shape factor.
- L_{Eff} is the effective pipe length, including the aggregate length of all pipe flow resistances.

For turbulent flows, the pressure loss in the half pipe adjacent to port A is

$$\Delta p_{Loss,A} = \frac{f_{Turb,A} L_{Eff} \dot{m}_A |\dot{m}_A|}{4D \rho_A S^2},$$

while in the half pipe adjacent to port B it is

$$\Delta p_{Loss,B} = \frac{f_{Turb,B} L_{Eff} \dot{m}_B |\dot{m}_B|}{4D \rho_B S^2},$$

where:

- $f_{Turb,A}$ and $f_{Turb,B}$ are the Darcy friction factors for turbulent flows.

If the **Viscous friction parameterization** parameter is set to `Correlation`, the block computes the Darcy friction factor through analytical expressions. For the half pipe adjacent to port A, the Darcy friction factor is

$$f_{Turb,A} = \frac{1}{\left[-1.8 \log_{10} \left(\frac{6.9}{\text{Re}_A} + \left(\frac{r}{3.7D} \right)^{1.11} \right) \right]^2},$$

while for the half pipe adjacent to port B it is

$$f_{Turb,B} = \frac{1}{\left[-1.8 \log_{10} \left(\frac{6.9}{Re_B} + \left(\frac{r}{3.7D} \right)^{1.11} \right) \right]^2},$$

where:

- Re_A and Re_B are the Reynolds numbers at ports A and B.
- r is the internal surface absolute roughness.

If the **Viscous friction parameterization** parameter is set to Tabulated data – Darcy friction factor vs. Reynolds number and the flow is turbulent, the block obtains the Darcy friction factor through tabular data specified in terms of the Reynolds number. If the flow is laminar, the block computes the friction factor is computed using the expression

$$f_{Lam} = \frac{\lambda}{Re_{Lam}}.$$

The block smooths the transition between laminar and turbulent regimes based on the Reynolds number at port A.

Energy Balance

The energy conservation equation for the pipe volume is

$$\frac{de}{dt} = \phi_A + \phi_B + \phi_H - \dot{m}g\Delta z,$$

where:

- e is the total energy in of the fluid in the pipe:
 $e = \rho u V$
- Φ_A and Φ_B are the energy flow rates into the pipe through the pipe inlets.
- Φ_H is the heat flow rate into the pipe through the pipe wall.
- \dot{m} is the average mass flow rate through the pipe:

$$\dot{m} = \frac{\dot{m}_A - \dot{m}_B}{2}$$

The heat flow rate through port H is given by

$$\phi_H = \frac{h_A + h_B}{2} S_{Wall} (T_H - T),$$

where:

- h_A and h_B are the liquid-wall heat transfer coefficients at the pipe inlets.
- T_H is the pipe wall temperature given by port H.
- S_{Wall} is the pipe wall surface area.

If the **Fluid dynamic compressibility** parameter is set to On and the **Pipe wall specification** parameter is set to Rigid,

$$\frac{de}{dt} = \rho V \left[\frac{du}{dp} \frac{dp}{dt} + \frac{du}{dT} \frac{dT}{dt} \right].$$

If the **Fluid dynamic compressibility** parameter is set to On and the **Pipe wall specification** parameter is set to Flexible,

$$\frac{de}{dt} = \rho V \left[\frac{du}{dp} \frac{dp}{dt} + \frac{du}{dT} \frac{dT}{dt} \right] + \rho \left(u + \frac{p}{\rho} \right) \frac{dV}{dt}.$$

Heat Flow Rate

Heat flow calculations between the thermal liquid and the pipe wall are based on the convective heat transfer coefficient. For the half pipe adjacent to port A, the heat transfer coefficient is

$$h_A = \frac{Nu_A k_A}{D},$$

while in the half pipe adjacent to port B it is

$$h_B = \frac{Nu_B k_B}{D},$$

where:

- h_A and h_B are the heat transfer coefficients at ports A and B.
- Nu_A and Nu_B are the Nusselt numbers at ports A and B.
- k_A and k_B are the fluid thermal conductivities at ports A and B.

The block can obtain the Nusselt numbers through empirical correlations for tubes or through tabulated data. If the **Heat transfer parameterization** parameter is set to

Correlation, the laminar-regime Nusselt number is a specified constant. The turbulent-regime Nusselt number is computed using the Gnielinski equation. For the half pipe adjacent to port A, this equation reads

$$Nu = \frac{\left(f_{Turb,A} / 8 \right) (Re_A - 1000) Pr_A}{1 + 12.7 \left(f_{Turb,A} / 8 \right)^{1/2} (Pr_A^{2/3} - 1)},$$

while for the half pipe adjacent to port B it reads

$$Nu = \frac{\left(f_{Turb,B} / 8 \right) (Re_B - 1000) Pr_B}{1 + 12.7 \left(f_{Turb,B} / 8 \right)^{1/2} (Pr_B^{2/3} - 1)}.$$

If the **Heat transfer parameterization** parameter is set to Tabulated data – Colburn factor vs. Reynolds number, the block obtains the Nusselt number in the half pipe adjacent to port A through the equation

$$Nu_A = j(Re_A) Re_A Pr_A^{1/3},$$

and in the half pipe adjacent to port B through the equation

$$Nu_B = j(Re_B) Re_B Pr_B^{1/3},$$

where:

- $j(Re_A)$ and $j(Re_B)$ are the Colburn factors at ports A and B specified as lookup tables in terms of the Reynolds number.

If the **Heat transfer parameterization** parameter is set to Tabulated data – Nusselt number vs. Reynolds number & Prandtl number, the block obtains the Nusselt number in the half pipe adjacent to port A through the equation

$$Nu_A = Nu(Re_A, Pr_A),$$

and in the half pipe adjacent to port B through the equation

$$Nu_B = Nu(Re_B, Pr_B).$$

Assumptions and Limitations

- The flow in the pipe is fully developed.
- Flexible pipes are circular in cross-section and expand only in the radial direction.

Parameters

Configuration Tab

Fluid dynamic 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.

Fluid inertia

Option to model the effects of fluid inertia in the pipe. Fluid inertia imparts on the fluid a resistance to changes in mass flow rate. This parameter is visible only when **Fluid dynamic compressibility** is set to `On`.

Number of segments

Number of pipe segments to include in the pipeline. This parameter is visible only when **Fluid dynamic compressibility** is set to `On`. The default value is 1.

Pipe length

Distance between pipe inlets A and B. The default value is 5 m.

Nominal cross-sectional area

Cross-sectional area of an equivalent circular pipe. This area is assumed constant throughout the length of the pipe. The default value is 0.01 m^2 .

Pipe wall specification

Option to model the pipe wall as rigid or flexible. Select `Flexible` to account for the expansion and contraction of the pipe wall due to fluid pressure. This parameter is visible only when **Fluid dynamic compressibility** is set to `On`. The default setting is `Rigid`.

Hydraulic diameter (rigid pipe)

Diameter of a pipe equivalent in cross-sectional area but circular in cross-sectional shape. This parameter is visible only when **Pipe wall specification** is set to `Rigid`. The default value is 0.1128 m.

Static pressure-diameter compliance

Proportionality constant between the radial strain of the pipe and pressure. The default value is $1.2e-6$ m/MPa. This parameter is visible only when **Pipe wall specification** is set to `Flexible`.

Viscoelastic process time constant

Characteristic time for a unit pipe deformation to occur. The default value is 0.01 s. This parameter is visible only when **Pipe wall specification** is set to `Flexible`.

Elevation gain from port A to port B

Change in elevation from port A to port B. This value enables the block to compute the elevation head of the pipe. The elevation gain can be positive or negative. The default value is 0, corresponding to a horizontal pipe.

This parameter is active when the block variant is set to `Constant elevation`. To view or change the active block variant, right-click the block and select **Simscape > Block choices**.

Gravitational acceleration

Value of the gravitational constant. The default value is 9.81 m/s².

Viscous Friction Tab

Viscous friction parameterization

Method of computing the viscous friction factor in laminar and turbulent flows. Select `Correlation` to automatically compute the viscous friction factor from semi-empirical correlations. Select `Tabulated data – Darcy friction factor vs. Reynolds number` to directly provide the viscous friction factor as a 1-way lookup table.

Aggregate equivalent length of local resistances

Pressure loss due to local resistances such as bends, inlets, and fittings, expressed as the equivalent length of these resistances. The default value is 1 m. This parameter is visible only when **Viscous friction parameterization** is set to `Correlation`.

Internal surface absolute roughness

Average height of all surface defects on the internal surface of the pipe. This parameter enables the calculation of the friction factor in the turbulent flow regime. The default value is $1.5e-5$ m. This parameter is visible only when **Viscous friction parameterization** is set to `Correlation`.

Reynolds number vector for Darcy friction factor

Vector of Reynolds numbers at which to specify the Darcy friction factor data. The block uses the Reynolds and Darcy friction factor vectors to construct a 1-D lookup table. The default vector is a 12-element array ranging from 400 to $1e+8$. This parameter is visible only when **Viscous friction parameterization** is set to `Tabulate data - Darcy friction factor vs. Reynolds number`.

Darcy friction factor vector

Vector of Darcy friction factors at the Reynolds numbers specified in the **Reynolds number vector for Darcy friction factor** parameter. The default vector is a 12-element array ranging from 0.264 to 0.0214. This parameter is visible only when **Viscous friction parameterization** is set to `Tabulate data - Darcy friction factor vs. Reynolds number`.

Shape factor for laminar flow viscous friction

Proportionality constant between the inverse of the Reynolds number and the Darcy friction factor in the laminar flow regime. This parameter encodes the pipe cross-sectional shape in the calculation of laminar friction losses. The default value, corresponding to a circular pipe cross-section, is 64. This parameter is visible only when **Viscous friction parameterization** is set to `Correlation`.

Laminar flow upper Reynolds number limit

Reynolds number separating the laminar and transitional flow regimes. The flow is laminar below this number and transitional above it. The default value is $2e+3$.

Turbulent flow lower Reynolds number limit

Reynolds number separating the transitional and turbulent flow regimes. The flow is transitional below this number and turbulent above it. The default value is $4e+3$.

Heat Transfer Tab

Heat transfer parameterization

Method of obtaining the Nusselt number in laminar and turbulent flows for use in heat transfer calculations. Select `Correlation` to automatically compute the

Nusselt number from semi-empirical correlations. Select `Tabulated data - Colburn factor vs. Reynolds number` to compute the Nusselt number from Colburn factor data that you provide. Select `Tabulated data - Nusselt number vs. Reynolds number & Prandtl number` to directly provide the Nusselt number as a 2-way lookup table.

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 in laminar flows. The parameter value changes with the pipe cross-sectional area and thermal boundary conditions, e.g., constant temperature or constant heat flux at the pipe wall. The default value, corresponding to a circular pipe cross section, is 3.66. This parameter is visible only when **Heat transfer parameterization** is set to `Correlation`.

Reynolds number vector for Colburn factor

Vector of Reynolds numbers at which to specify the Colburn factor 1-D lookup table. The default vector is `[100, 150, 1000]`. This parameter is visible only when **Heat transfer parameterization** is set to `Tabulated data - Colburn factor vs. Reynolds number`.

Colburn factor vector

Vector of Colburn factors at the Reynolds numbers provided in the **Reynolds number vector for Colburn factor** vector. The default vector is `[0.019, 0.013, 0.002]`. This parameter is visible only when **Heat transfer parameterization** is set to `Tabulated data - Colburn factor vs. Reynolds number`.

Reynolds number vector for Nusselt number

M-element vector of Reynolds numbers at which to specify the Nusselt number 2-D lookup table. The default vector is `[100, 150, 1000]`. This parameter is visible only when **Heat transfer parameterization** is set to `Tabulated data - Nusselt number vs. Reynolds number & Prandtl number`.

Prandtl number vector for Nusselt number

N-element vector of Prandtl numbers at which to specify the Nusselt number 2-D lookup table. The default vector is `[1, 10]`. This parameter is visible only when **Heat transfer parameterization** is set to `Tabulated data - Nusselt number vs. Reynolds number & Prandtl number`.

Nusselt number table, Nu(Re,Pr)

M×N matrix with the Nusselt numbers at the Reynolds and Prandtl numbers specified. The default matrix is [3.72, 4.21, ; 3.75, 4.44; 4.21, 7.15]. This parameter is visible only when **Heat transfer parameterization** is set to Tabulated data – Nusselt number vs. Reynolds number & Prandtl number.

Initial Conditions Tab

Initial liquid temperature

Initial temperature specified as a scalar, 2-element vector, or n-element vector:

- If the parameter input is a scalar, the initial temperature specified applies to all pipe segments.
- If the parameter is a 2-element vector, the initial temperature varies linearly between the two values across the pipe segments.
- If the parameter is an n-element vector, each value gives the temperature of a pipe segment. The vector size must match the specified number of pipe segments.

The default value, corresponding to room temperature, is 293.15 K.

Initial liquid pressure

Initial liquid pressure specified as a scalar, 2-element vector, or n-element vector:

- If the parameter input is a scalar, the initial pressure specified applies to all pipe segments.
- If the parameter is a 2-element vector, the initial pressure varies linearly between the two values across the pipe segments.
- If the parameter is an n-element vector, each value gives the pressure of a pipe segment. The vector size must match the specified number of pipe segments.

The default value, corresponding to atmospheric pressure, is 0.101325 Pa.

Ports

- **A** — Thermal liquid port representing pipe inlet A.
- **B** — Thermal liquid port representing pipe inlet B.

- **H** — Thermal conserving port for modeling heat transfer between the pipe and the environment.
- **EI** — Physical signal input port for the elevation gain between the ports. This port is active when the block variant is set to `Variable elevation`. To change the active block variant, right-click the block and select **Simscape** > **Block choices**.

See Also

Double-Acting Actuator (TL)

Introduced in R2016a

Pneumo-Hydraulic Actuator

Double-acting cylinder operated by hydraulic and pneumatic power

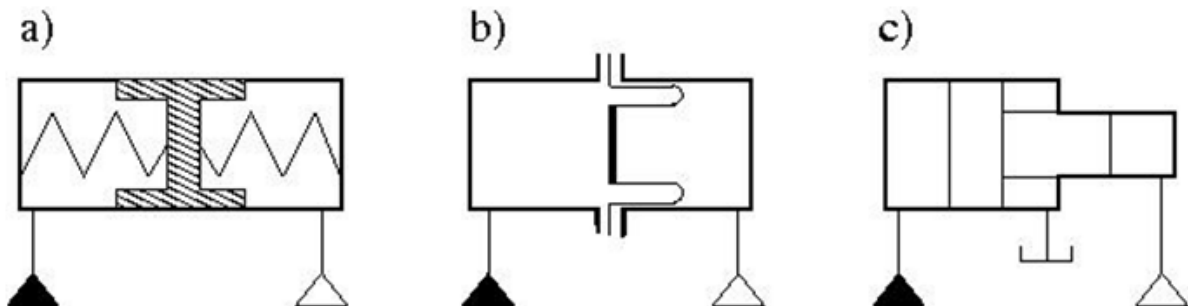


Library

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:

$$q_H = A_H \cdot v$$

$$F_H = A_H \cdot p$$

$$F_P = A_P \cdot p$$

$$F_{HS} = \begin{cases} K \cdot (x - stroke) + D \cdot v & \text{for } x > stroke \\ K \cdot x + D \cdot v & \text{for } x < 0 \\ 0 & \text{for } 0 \leq x \leq stroke \end{cases}$$

$$F_H = F_P + F_{HS} + F_L$$

$$v = \frac{dx}{dt}$$

$$V_P = V_0 + A_P \cdot (stroke - x)$$

$$G = \frac{V_P}{RT} \left(\frac{dp}{dt} - \frac{p}{T} \frac{dT}{dt} \right) - \frac{A_P}{RT} p \cdot v$$

$$Q_P = \frac{c_v \cdot V_P}{R} \frac{dp}{dt} - \frac{c_p \cdot A_P}{R} p \cdot v + Q_{HE}$$

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
<i>stroke</i>	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.

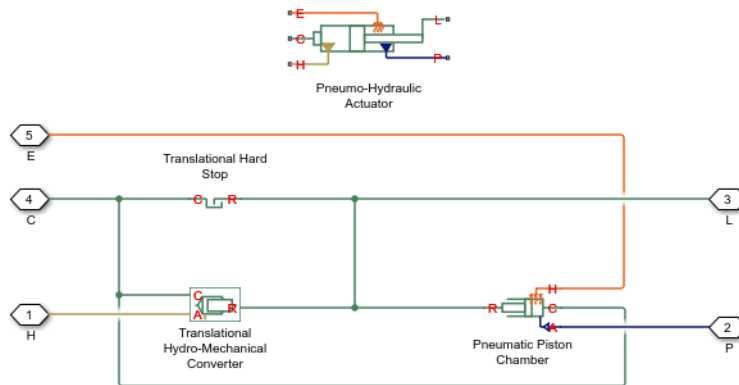
Port P is the pneumatic conserving port associated with the pneumatic side of the actuator. Port H is the hydraulic conserving port associated with the hydraulic inlet.

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 1-655
- “Pneumatic Side Tab” on page 1-656
- “Hard Stop Tab” on page 1-657

Hydraulic Side Tab

Hydraulic side piston area

Effective piston area on the hydraulic side. The default value is $2e-3 \text{ m}^2$.

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 $1e-4 \text{ m}^3$.

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 $1e-3 \text{ m}^2$.

Pneumatic side dead volume

Gas volume in pneumatic chamber that remains in the chamber after the rod is fully retracted. The default value is $1e-4 \text{ m}^3$.

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 $1e6 \text{ N/m}$.

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

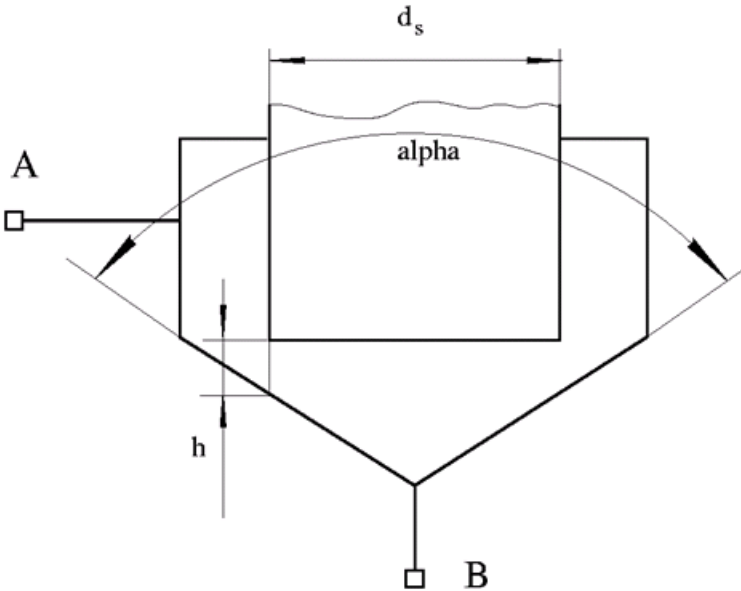


Library

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:

$$q = C_D \cdot A(h) \sqrt{\frac{2}{\rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

$$p = p_A - p_B$$

$$h = x_0 + x$$

$$h_{\max} = d_s \frac{\sqrt{1 + \cos \frac{\alpha}{2}} - 1}{\sin \alpha}$$

$$A(h) = \begin{cases} A_{leak} & \text{for } h \leq 0 \\ \pi \cdot \left(d_s + h \cdot \sin \frac{\alpha}{2} \cdot \cos \frac{\alpha}{2} \right) \cdot h \cdot \sin \frac{\alpha}{2} + A_{leak} & \text{for } 0 < h < h_{\max} \\ A_{\max} + A_{leak} & \text{for } h \geq h_{\max} \end{cases}$$

$$A_{\max} = \frac{\pi d_s^2}{4}$$

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:

$$p_{\text{cr}} = (p_{\text{avg}} + p_{\text{atm}})(1 - B_{\text{lam}})$$

$$p_{\text{avg}} = (p_A + p_B)/2$$

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:

$$p_{\text{cr}} = \frac{\rho}{2} \left(\frac{\text{Re}_{\text{cr}} \cdot \nu}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

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

$p = p_A - p_B$. 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 $1e-12 \text{ m}^2$.

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

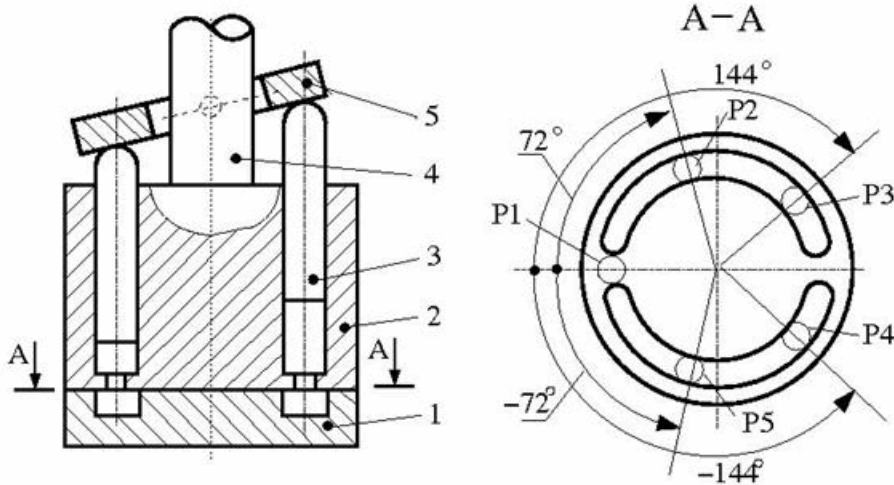


Library

Pumps and Motors

Description

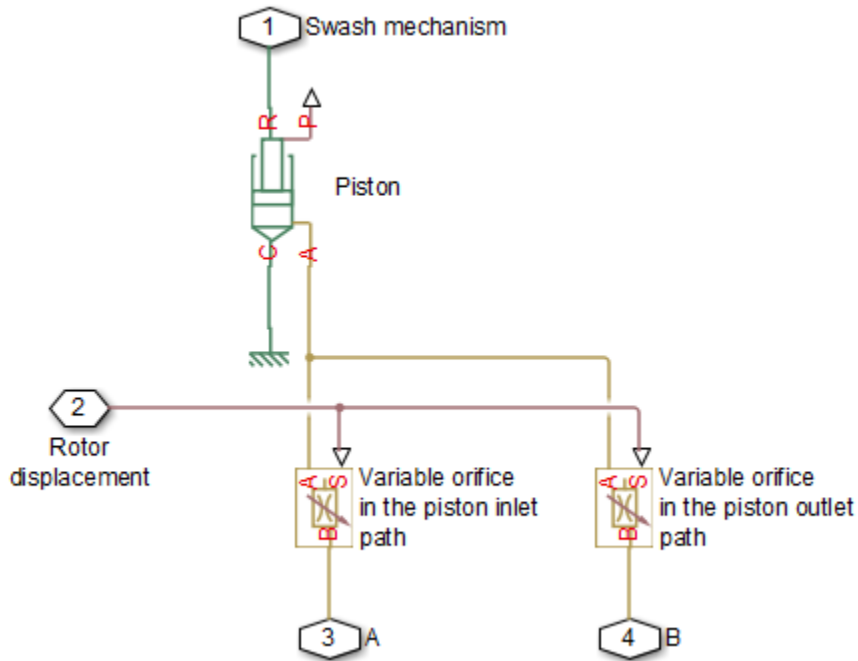
Porting plate is a key element of axial-piston machines. Its objective is to provide communication between pistons and pump ports during the rotor, or cylinder block, rotation. The porting plate is equipped with two crescent-shaped slots, one of which is connected to the intake port while the other is 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:

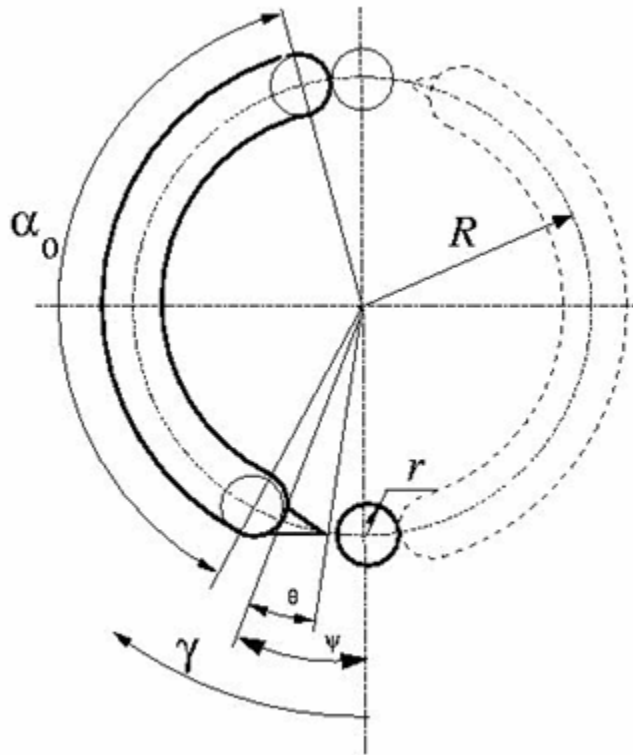
- 1 Porting plate
- 2 Rotor
- 3 Piston
- 4 Driving shaft
- 5 Swash plate

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, $r / R \leq \psi \leq 3 r / R$, 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

$$\psi + r / R \leq \gamma \leq \pi - 2 \cdot r / R$$

where γ is the rotation angle.

The orifice area is computed with the following equations:

$$A = A_{leak} \quad \text{for } \gamma_6 < \gamma \leq \gamma_1$$

$$A = G \cdot (\gamma - \gamma_1) + A_{leak} \quad \text{for } \gamma_1 < \gamma \leq \gamma_2$$

$$A = A_{tr} + r^2 (\beta - \sin \beta) + A_{leak} \quad \text{for } \gamma_2 < \gamma \leq \gamma_3$$

$$A = A_{tr} \cdot (1 - 100 \cdot (\gamma - \gamma_3)) + \pi \cdot r^2 + A_{leak} \quad \text{for } \gamma_3 < \gamma \leq \gamma_4$$

$$A = \pi \cdot r^2 + A_{leak} \quad \text{for } \gamma_4 < \gamma \leq \gamma_5$$

$$A = r^2 (\beta - \sin \beta) + A_{leak} \quad \text{for } \gamma_5 < \gamma \leq \gamma_6$$

$$\beta = \begin{cases} 2 \cdot \alpha \cos(R / r \cdot \sin(0.5 \cdot (\gamma_3 - \gamma))) & \text{for } \gamma_2 < \gamma \leq \gamma_3 \\ 2 \cdot \alpha \cos(R / r \cdot \sin(0.5 \cdot (2 \cdot r / R + \gamma - \pi))) & \text{for } \gamma_5 < \gamma \leq \gamma_6 \end{cases}$$

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:

$$q = C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

$$p = p_A - p_B$$

$$p_{cr} = \frac{\rho}{2} \left(\frac{Re_{cr} \cdot \nu}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

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 $r/R \leq \psi \leq 3r/R$, 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 $1e-9$ m².

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.

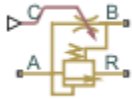
See Also

[Angle Sensor](#) | [Swash Plate](#)

Introduced in R2011a

Pressure-Compensated 3-Way Flow Control Valve

Hydraulic pressure compensating 3-way valve

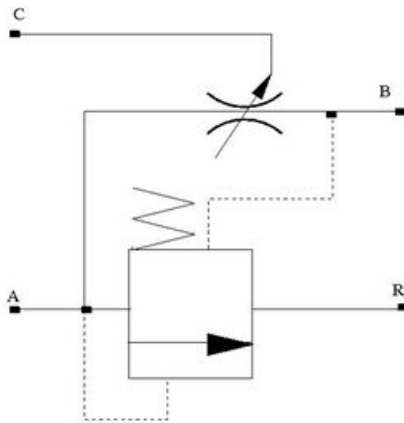


Library

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:

$$A_{pc} = \begin{cases} A_{leak} & \text{for } p < p_{set} \\ (p - p_{set}) \frac{A_{max_pc}}{p_{reg}} & \text{for } p_{set} \leq p \leq (p_{set} + p_{reg}) \\ A_{max_pc} & \text{for } p > (p_{set} + p_{reg}) \end{cases}$$

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:

$$q = C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

$$A = \begin{cases} A_{pc} & \text{for pressure compensator} \\ A_{or} & \text{for variable orifice} \end{cases}$$

$$A_{pc} = \begin{cases} h \cdot A_{\max} / h_{\max} + A_{leak} & \text{for } h > 0 \\ A_{leak} & \text{for } h \leq 0 \end{cases}$$

$$h = x_0 + x$$

$$p = \begin{cases} p_A - p_R & \text{for pressure compensator} \\ p_A - p_B & \text{for variable orifice} \end{cases}$$

$$p_{cr} = \frac{\rho}{2} \left(\frac{Re_{cr} \cdot v}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

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 1-677
- “Pressure Compensator Tab” on page 1-680

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 $1e-4 \text{ m}^2$. 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 $1e-9 \text{ m}^2$.

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 $6e5 \text{ Pa}$.

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 $0.5e5 \text{ Pa}$.

Pressure compensator maximum area

The area of a fully opened pressure compensator. The parameter value must be greater than zero. The default value is $1e-4 \text{ m}^2$.

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 $1e-9 \text{ m}^2$.

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

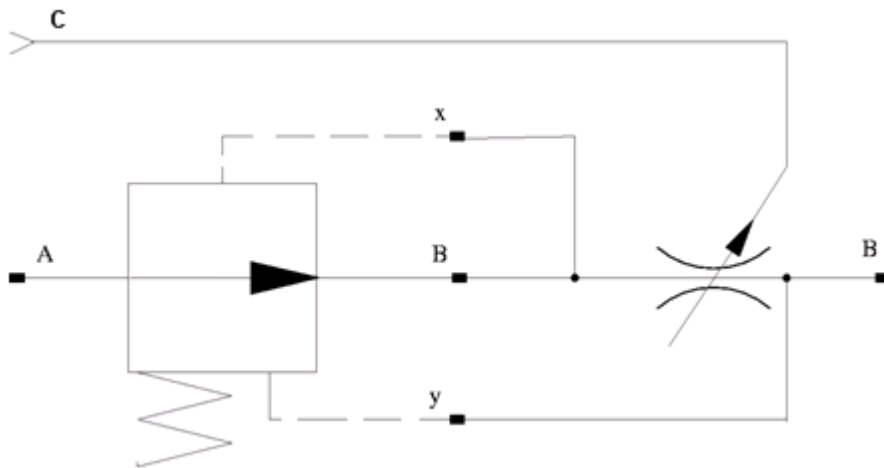


Library

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

$p = p_A - p_B$. 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 $5e-5 \text{ m}^2$. 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 $5e-4 \text{ m}$. 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 $1e-12 \text{ m}^2$.

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

Hydraulic pressure compensating valve

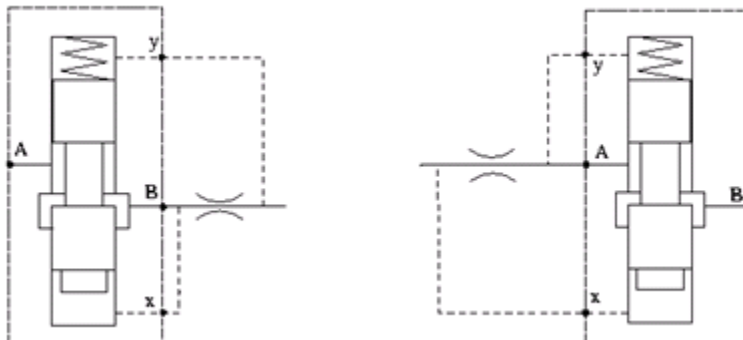


Library

Pressure Control Valves

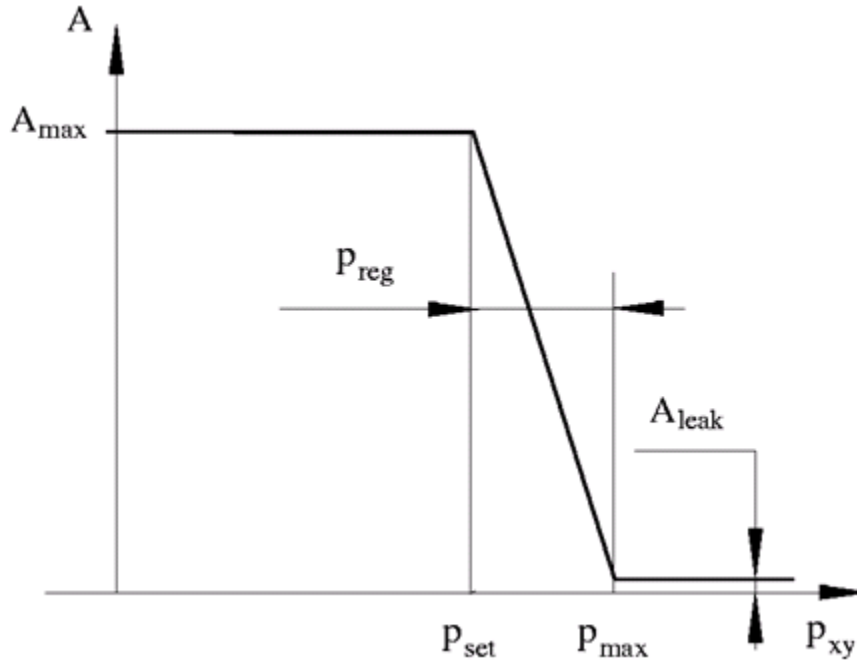
Description

The Pressure Compensator block represents a hydraulic pressure compensating valve, or pressure compensator. Pressure compensators are used to maintain preset pressure differential across a hydraulic component to minimize the influence of pressure variation on a flow rate passing through the component. The following illustration shows typical applications of a pressure compensator, where it is used in combination with the orifice installed downstream (left figure) or upstream (right figure). The compensator can be also used in combination with metering pumps, flow dividers, and so on.



The block is implemented as a data-sheet-based model, based on parameters usually provided in the manufacturer's catalogs or data sheets.

Pressure compensator is a normally open valve. Its opening is proportional to pressure difference between ports X and Y and the spring force. The following illustration shows typical relationship between the valve passage area A and the pressure difference p_{xy} .



The orifice remains fully open until the pressure difference is lower than valve preset pressure determined by the spring preload. When the preset pressure is reached, the valve control member is forced off its stop and starts closing the orifice, thus trying to maintain pressure differential at preset level. Any further increase in the pressure difference causes the control member to close the orifice even more, until the point when the orifice is fully closed. The pressure increase that is necessary to close the valve is referred to as regulation range, or pressure compensator static error, and usually is provided in manufacturer's catalog or data sheets.

The main parameters of the block are the valve maximum area and regulation range. In addition, you need to specify the leakage area of the valve. 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.

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:

$$q = C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

$$p = p_A - p_B$$

$$A(p) = \begin{cases} A_{\max} & \text{for } p_{xy} \leq p_{set} \\ A_{\max} - k \cdot (p_{xy} - p_{set}) & \text{for } p_{set} < p_{xy} < p_{\max} \\ A_{leak} & \text{for } p_{xy} \geq p_{\max} \end{cases}$$

$$k = \frac{A_{\max} - A_{leak}}{p_{reg}}$$

$$p_{xy} = p_x - p_y$$

where

q	Flow rate
p	Pressure differential across the valve

p_{xy}	Pressure differential across valve control terminals
p_A, p_B	Gauge pressures at the valve main terminals
p_x, p_y	Gauge pressures at the valve control terminals
p_{set}	Valve preset pressure
p_{max}	Pressure needed to fully close the orifice
p_{reg}	Regulation range
A	Instantaneous orifice passage area
$A(p)$	Pressure-dependent orifice passage area
A_{init}	Initial open area of the valve
A_{max}	Orifice maximum area
A_{leak}	Closed orifice leakage area
C_D	Flow discharge coefficient
ρ	Fluid density
τ	Time constant for the first order response of the valve opening
t	Time
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:

$$p_{cr} = \frac{\rho}{2} \left(\frac{Re_{cr} \cdot v}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

where

D_H	Instantaneous orifice hydraulic diameter
v	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

$p = p_A - p_B$. The control pressure differential is measured as $p_{xy} = p_x - p_y$, and it creates a force acting against the spring preload.

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.
- Flow consumption associated with the spool motion is neglected.

Parameters

Maximum passage area

Valve passage maximum cross-sectional area. The default value is $1e-4 \text{ m}^2$.

Valve pressure setting

Pressure difference that must be maintained across an element connected to ports X and Y. At this pressure the valve orifice starts to close. The default value is $3e6 \text{ Pa}$.

Valve regulation range

Pressure increase over the preset level needed to fully close the valve. Must be less than 0.2 of the **Valve pressure setting** parameter value. The default value is $1.5e5 \text{ Pa}$.

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.

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 $1e-12$ m².

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 $1e-12 \text{ m}^2$.

Restricted Parameters

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

- **Opening dynamics**
- **Laminar transition specification**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Opening dynamics** and **Laminar transition specification** 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 valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

X

Hydraulic conserving port associated with the pressure control terminal that opens the orifice.

Y

Hydraulic conserving port associated with the pressure control terminal that closes the orifice.

See Also

Pressure Reducing Valve | Pressure Relief Valve

Introduced in R2006a

Pressure Reducing 3-Way Valve

Pressure reducing/relieving valve

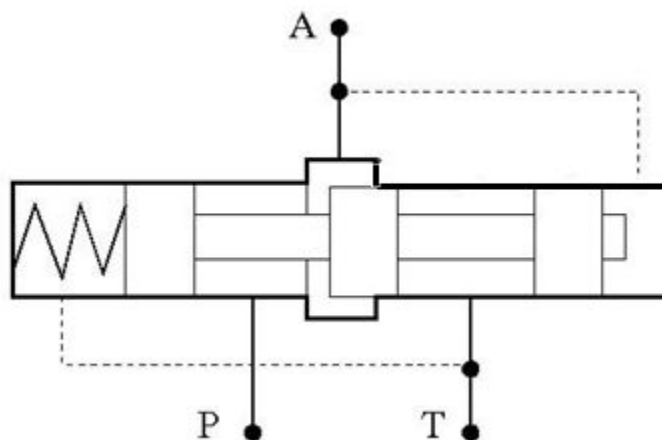


Library

Pressure Control Valves

Description

The Pressure Reducing 3-Way Valve block represents a hydraulic 3-way valve that is also known as a pressure reducing/relieving valve. The valve reduces inlet pressure to a preset value, similar to a conventional pressure reducing valve, but, additionally, starts working as a pressure-relief valve if the pressure continues to rise.



Initially, orifice P-A is fully open. It remains fully open while the outlet pressure at port A is lower than the valve preset pressure. When the preset pressure is reached, the valve control member is forced off its stop and starts closing the orifice, thus trying to maintain outlet pressure at the preset level. Any further increase in the outlet pressure causes the control member to close the orifice even more, until the orifice is fully closed. The pressure increase needed to close the valve is referred to as regulation range, and is generally provided in the catalogs, along with the valve maximum area. In a conventional pressure reducing valve, outlet pressure is no longer under control after orifice P-A is closed. The pressure reducing 3-way valve provides an additional relieving function by diverting some flow from the outlet to a tank through an additional orifice A-T.

The valve has the following area-pressure differential relationship:

$$A_{\text{red}} = A_{\text{med}} - (A_{\text{max}} - A_{\text{med}}) \cdot \tanh(k \cdot (p - p_{\text{red_med}}) / (p_{\text{red_max}} - p_{\text{red_med}}))$$

$$A_{\text{rel}} = A_{\text{med}} + (A_{\text{max}} - A_{\text{med}}) \cdot \tanh(k \cdot (p - p_{\text{rel_med}}) / (p_{\text{rel_max}} - p_{\text{rel_med}}))$$

$$A_{\text{med}} = (A_{\text{max}} + A_{\text{leak}}) / 2$$

$$p_{\text{red_max}} = p_{\text{set}} + p_{\text{reg}}$$

$$p_{\text{red_med}} = (p_{\text{set}} + p_{\text{reg}}) / 2$$

$$p_{\text{rel_set}} = p_{\text{set}} + p_{\text{reg}} + p_{\text{tr}}$$

$$p_{\text{rel_max}} = p_{\text{rel_set}} + p_{\text{reg}}$$

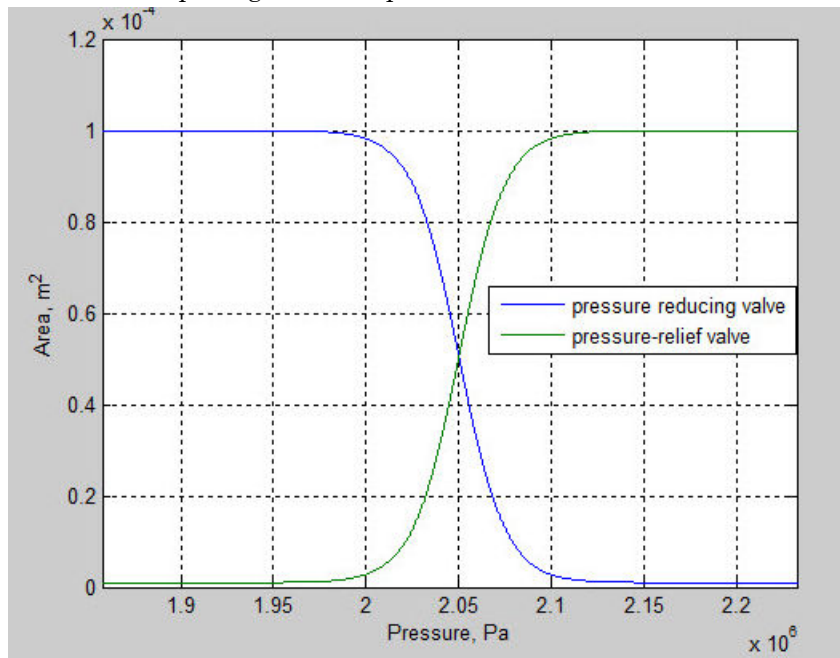
$$p_{\text{rel_med}} = (p_{\text{rel_set}} + p_{\text{reg}}) / 2$$

where

A_{max}	Maximum opening area of both the reducing and relieving valves
A_{red}	Pressure reducing (orifice P-A) valve opening area
A_{rel}	Pressure-relief (orifice A-T) valve opening area
A_{leak}	Leakage area (the area that remains open even after the orifice is completely closed)
p	Pressure drop across the valve, $p_A - p_T$
p_{set}	Preset pressure differential
p_{reg}	Regulation range

p_{tr}	Transition pressure (the pressure increment above the pressure of the fully closed reducing valve and setting pressure of the pressure-relief valve)
k	Valve opening adjustment coefficient

The distinctive feature of a pressure reducing 3-way valve is a sharp change of orifice openings. To avoid computational problems, the openings are approximated using the hyperbolic tangent function. The following figure shows an example of the relationship between the opening area and pressure.



In the example, the setting pressure is set to $20e5$ Pa, and the regulation range is set to $1e5$ Pa. The valve opening adjustment coefficient is set to 2. The higher the value of the coefficient, the closer the transition is to a linear relationship. The transition is close to the experimental data at k in the range of 3 to 4.

For both orifices, 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 areas have been determined, the block computes the flow rate for both orifices according to the following equations:

$$q = C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

$$A = \begin{cases} A_{red} & \text{for orifice P-A} \\ A_{rel} & \text{for orifice A-T} \end{cases}$$

$$p = \begin{cases} p_P - p_A & \text{for orifice P-A} \\ p_A - p_T & \text{for orifice A-T} \end{cases}$$

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
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:

$$p_{cr} = \frac{\rho}{2} \left(\frac{\text{Re}_{cr} \cdot v}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

where

D_H	Instantaneous orifice hydraulic diameter
v	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 instantaneous orifice passage area A in the flow equations above is then determined as follows:

$$A = \begin{cases} A_{red_dyn} & \text{for orifice P-A} \\ A_{rel_dyn} & \text{for orifice A-T} \end{cases}$$

$$A_{red_dyn}(t = 0) = A_{red_init}$$

$$\frac{dA_{red_dyn}}{dt} = \frac{A_{red} - A_{red_dyn}}{\tau}$$

$$A_{rel_dyn}(t = 0) = A_{rel_init}$$

$$\frac{dA_{rel_dyn}}{dt} = \frac{A_{rel} - A_{rel_dyn}}{\tau}$$

where

A_{red_dyn}	Instantaneous open area for pressure reducing valve (orifice P-A) with opening dynamics
----------------	---

A_{rel_dyn}	Instantaneous open area for pressure-relief valve (orifice A-T) with opening dynamics
A_{red_init}	Initial open area for pressure reducing valve (orifice P-A)
A_{rel_init}	Initial open area for pressure-relief valve (orifice A-T)
τ	Time constant for the first order response of the valve opening
t	Time

Connections P, A, and T are the conserving hydraulic ports associated with the valve inlet, outlet, and return, respectively. The block positive direction is from port P to port A and from port A to port T.

Assumptions and Limitations

Fluid inertia, valve inertia, friction, and hydraulic forces are not taken into account.

Parameters

Valve maximum area

The area of a fully opened orifice. Both orifices are assumed to have the same maximum area. The parameter value must be greater than zero. The default value is $1e-4 \text{ m}^2$.

Reducing valve pressure setting

Preset pressure level, at which the orifice P-A of the valve starts to close. The default value is $6e5 \text{ Pa}$.

Valve regulation range

Pressure increase over the preset level needed to fully close the pressure reducing valve. The lower the value of the range, the higher the valve sensitivity. The default value is $0.3e5 \text{ Pa}$.

Transition pressure

Pressure increment above the pressure of the fully closed reducing valve needed to reach the pressure at which the pressure-relief valve starts opening. The transition pressure must be greater than or equal to zero. The default value is $2e5 \text{ Pa}$.

Valve flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. The value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is 0.6.

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 **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 **Valve critical Reynolds number** parameter.

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 **Valve laminar transition specification** parameter is set to `Pressure ratio`.

Valve 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 **Valve laminar transition specification** parameter is set to `Reynolds number`.

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 $1e-9 \text{ m}^2$.

Valve opening adjustment coefficient

The coefficient controls how close the hyperbolic tangent function approximates the linear relationship between the orifice area and control pressure. See the block description for more information. The default value is 1.

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 reducing valve area

The initial opening area of the reducing valve. This parameter is available only if **Opening dynamics** is set to `Include valve opening dynamics`. The default value is $1e-9 \text{ m}^2$.

Initial relief valve area

The initial opening area of the relief valve. This parameter is available only if **Opening dynamics** is set to `Include valve opening dynamics`. The default value is $1e-9 \text{ m}^2$.

Restricted Parameters

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

- **Opening dynamics**
- **Valve laminar transition specification**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Opening dynamics** and **Valve laminar transition specification** 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:

P

Hydraulic conserving port associated with the valve inlet.

A

Hydraulic conserving port associated with the valve outlet.

T

Hydraulic conserving port that connects with the tank.

See Also

Pressure Reducing Valve

Introduced in R2013b

Pressure Reducing Valve

Pressure control valve maintaining reduced pressure in portion of system

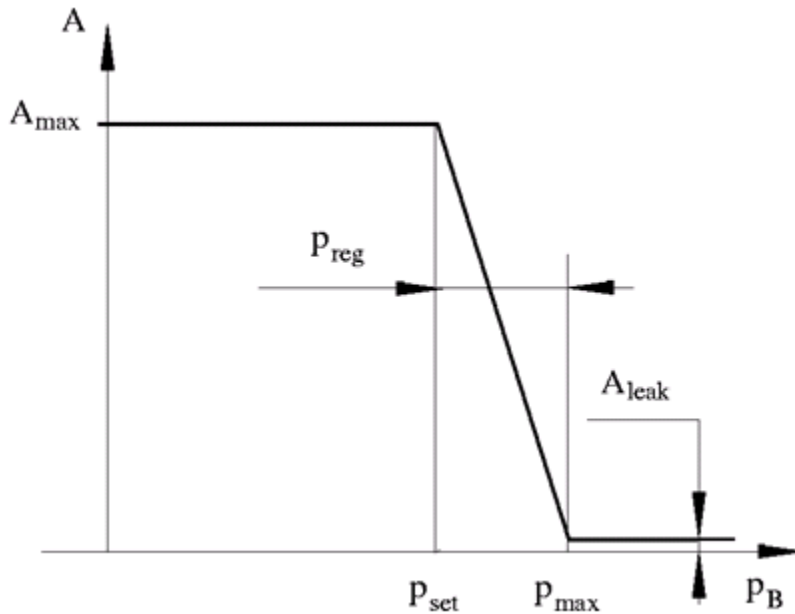


Library

Pressure Control Valves

Description

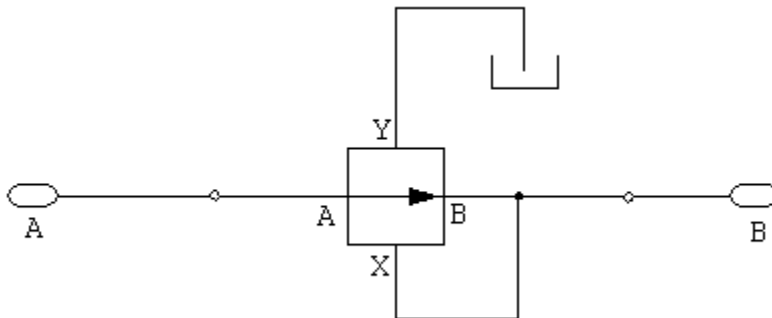
The Pressure Reducing Valve block represents a hydraulic pressure-reducing valve as a data-sheet-based model. Pressure-reducing valves are used to maintain reduced pressure in a portion of a system. The following figure shows the typical dependency between the valve passage area A and the pressure p_B downstream from the valve.



The pressure-reducing valve is a normally open valve and it remains fully open while outlet pressure is lower than the valve preset pressure. When the preset pressure is reached, the valve control member (spool, ball, poppet, etc.) is forced off its stop and starts closing the orifice, thus trying to maintain outlet pressure at preset level. Any further increase in the outlet pressure causes the control member to close the orifice even more until the point when the orifice is fully closed. The pressure increase that is necessary to close the valve is referred to as regulation range, and is generally provided in the catalogs, along with the valve maximum area and regulation range are the 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.

The block is built as a structural model based on the Pressure Compensator block, as shown in the following schematic.



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

$$p = p_A - p_B.$$

Parameters

Maximum passage area

Valve passage maximum cross-sectional area. The default value is $1e-4 \text{ m}^2$.

Valve pressure setting

Preset pressure level, at which the orifice of the valve starts to close. The default value is $5e6 \text{ Pa}$.

Valve regulation range

Pressure increase over the preset level needed to fully close the valve. Must be less than 0.2 of the **Valve pressure setting** parameter value. The default value is $5e5 \text{ Pa}$.

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. 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 $1e-12m^2$.

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 $1e-12 \text{ m}^2$.

Restricted Parameters

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

- **Opening dynamics**
- **Laminar transition specification**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Opening dynamics** and **Laminar transition specification** 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 valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

Examples

The Pump with Pressure-Reducing Valve and Two Loads example illustrates the use of the Pressure Reducing Valve block in hydraulic systems. The pressure reducing valve is set to 20×10^5 Pa and maintains this pressure downstream, as long as the upstream pressure is higher than this setting.

See Also

Pressure Compensator | Pressure Relief Valve

Introduced in R2006a

Pressure Reducing Valve (TL)

Pressure control valve for maintaining reduced pressure in fluid network portion



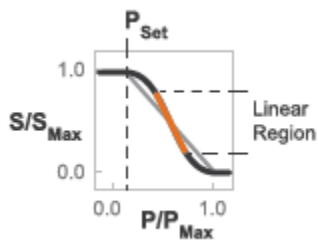
Library

Thermal Liquid/Valves/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:

$$PR_{Loss} = \frac{\sqrt{1 - (S_R/S)^2 (1 - C_d^2)} - C_d (S_R/S)}{\sqrt{1 - (S_R/S)^2 (1 - C_d^2)} + C_d (S_R/S)}.$$

The valve opening area is computed as

$$S_R = \begin{cases} S_{Max}, & p_{control} \leq p_{set} \\ S_{Max} (1 - \lambda_L) + S_{Linear} \lambda_L, & p_{control} < p_{Min} + \Delta p_{smooth} \\ S_{Linear}, & p_{control} \leq p_{Max} - \Delta p_{smooth}, \\ S_{Linear} (1 - \lambda_R) + S_{Leak} \lambda_R, & p_{control} < p_{Max} \\ S_{Leak}, & p_{control} \geq p_{Max} \end{cases}$$

where:

- S_{Leak} is the valve leakage area.
- S_{Linear} is the linear valve opening area:

$$S_{Linear} = \left(\frac{S_{Leak} - S_{Max}}{p_{Max} - p_{set}} \right) (p_{control} - p_{set}) + S_{Max}$$

- 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:

$$\bar{p}_L = \frac{p_{control} - p_{set}}{(p_{set} + \Delta p_{smooth}) - p_{set}}$$

and

$$\bar{p}_R = \frac{p_B - (p_{max} - \Delta p_{smooth})}{p_{max} - (p_{max} - \Delta p_{smooth})}$$

The critical mass flow rate is

$$\dot{m}_{cr} = Re_{cr} \mu_{Avg} \sqrt{\frac{\pi}{4} S_R}.$$

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

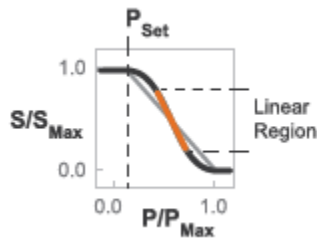
Aggregate area of all fluid leaks in the valve. The leakage area helps to prevent numerical issues due to isolated fluid network sections. For numerical robustness, set this parameter to a nonzero value. The default value is $1e-12$.

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

Pressure control valve maintaining preset pressure in system

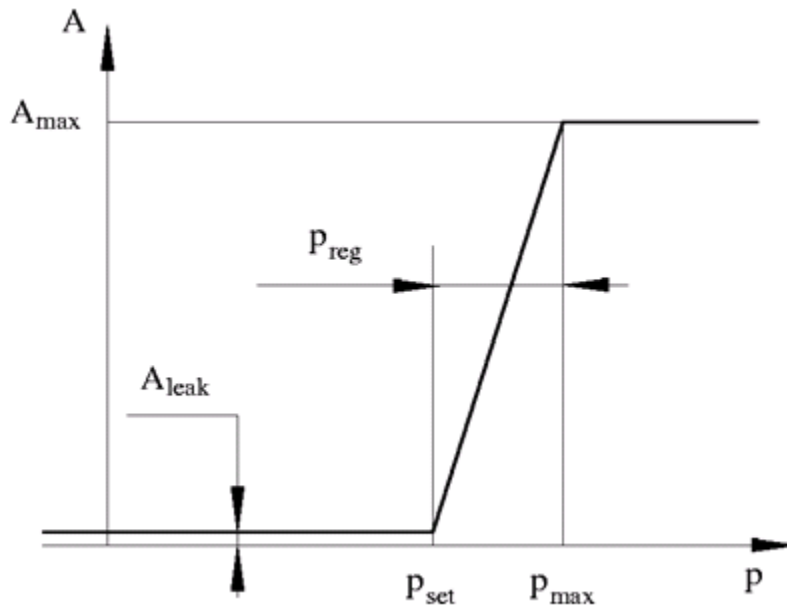


Library

Pressure Control Valves

Description

The Pressure Relief Valve block represents a hydraulic pressure relief valve as a data-sheet-based model. The following figure shows the typical dependency between the valve passage area A and the pressure differential p across the valve.



The valve remains closed while pressure at the valve inlet is lower than the valve preset pressure. When the preset 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. Some fluid is diverted to a tank through this orifice, thus reducing the pressure at the inlet. If this flow rate is not enough and pressure continues to rise, the area is further increased until the control member reaches its maximum. At this moment, the maximum flow rate is passing through the valve. The value of a maximum flow rate and the pressure increase over the preset level to pass this flow rate are generally provided in the catalogs. The pressure increase over the preset level is frequently referred to as valve steady state error, or regulation range. The valve maximum area and regulation range are the 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:

$$q = C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

$$p = p_A - p_B$$

$$A(p) = \begin{cases} A_{leak} & \text{for } p \leq p_{set} \\ A_{leak} + k \cdot (p - p_{set}) & \text{for } p_{set} < p < p_{max} \\ A_{max} & \text{for } p \geq p_{max} \end{cases}$$

$$k = \frac{A_{max} - A_{leak}}{p_{reg}}$$

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_{reg}	Regulation range
p_{set}	Valve preset pressure
p_{max}	Valve pressure at maximum opening
ρ	Fluid density
τ	Time constant for the first order response of the valve opening
t	Time
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:

$$p_{\text{cr}} = (p_{\text{avg}} + p_{\text{atm}})(1 - B_{\text{lam}})$$

$$p_{\text{avg}} = (p_A + p_B)/2$$

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:

$$p_{\text{cr}} = \frac{\rho}{2} \left(\frac{\text{Re}_{\text{cr}} \cdot v}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

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

$$P = P_A - P_B.$$

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 $1e-4 \text{ m}^2$.

Valve pressure setting

Preset pressure level, at which the orifice of the valve starts to open. The default value is $50e5 \text{ Pa}$.

Valve regulation range

Pressure increase over the preset level needed to fully open the valve. MathWorks recommends using values less than 0.2 of the **Valve pressure setting** parameter value. The default value is $5e5 \text{ Pa}$.

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. 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 $1e-12m^2$.

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 $1e-12 \text{ m}^2$.

Restricted Parameters

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

- **Opening dynamics**
- **Laminar transition specification**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Opening dynamics** and **Laminar transition specification** 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 valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

Examples

The Closed-Loop Pump Control with Flexible Drive Shaft example illustrates the use of the Pressure Relief Valve block in hydraulic systems. The valve is set to $75e5$ Pa and starts diverting fluid to tank as soon as the pressure at its inlet reaches this value.

See Also

[Pressure Compensator](#) | [Pressure Reducing Valve](#)

Introduced in R2006a

Pressure Relief Valve (TL)

Pressure control valve for maintaining preset pressure in fluid network



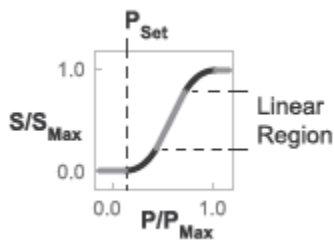
Library

Thermal Liquid/Valves/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:

$$PR_{Loss} = \frac{\sqrt{1 - (S_R/S)^2 (1 - C_d^2)} - C_d (S_R/S)}{\sqrt{1 - (S_R/S)^2 (1 - C_d^2)} + C_d (S_R/S)}.$$

The valve opening area is computed as

$$S_R = \begin{cases} S_{Leak}, & p_{control} \leq p_{set} \\ S_{Leak} (1 - \lambda_L) + S_{Linear} \lambda_L, & p_{control} < p_{Min} + \Delta p_{smooth} \\ S_{Linear}, & p_{control} \leq p_{Max} - \Delta p_{smooth}, \\ S_{Linear} (1 - \lambda_R) + S_{Max} \lambda_R, & p_{control} < p_{Max} \\ S_{Max}, & p_{control} \geq p_{Max} \end{cases}$$

where:

- S_{Leak} is the valve leakage area.
- S_{Linear} is the linear valve opening area:

$$S_{Linear} = \left(\frac{S_{Max} - S_{Leak}}{p_{Max} - p_{set}} \right) (p_{control} - p_{set}) + S_{Leak}$$

- 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:

$$\bar{p}_L = \frac{P_{control} - P_{set}}{(p_{set} + \Delta p_{smooth}) - P_{set}}$$

and

$$\bar{p}_R = \frac{p_{control} - (p_{max} - \Delta p_{smooth})}{p_{max} - (p_{max} - \Delta p_{smooth})}$$

The critical mass flow rate is

$$\dot{m}_{cr} = Re_{cr} \mu_{Avg} \sqrt{\frac{\pi}{4}} S_R.$$

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

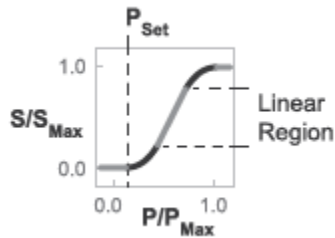
Aggregate area of all fluid leaks in the valve. The leakage area helps to prevent numerical issues due to isolated fluid network sections. For numerical robustness, set this parameter to a nonzero value. The default value is $1e-12$.

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



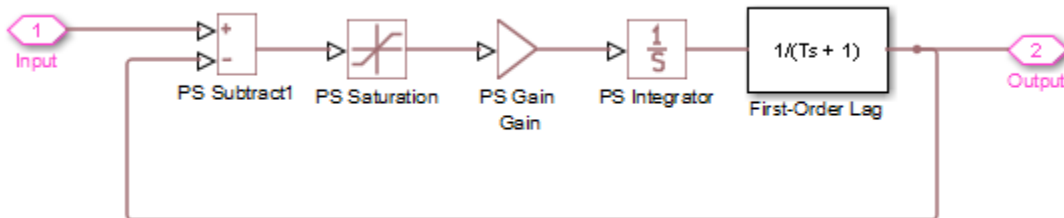
Library

Valve Actuators

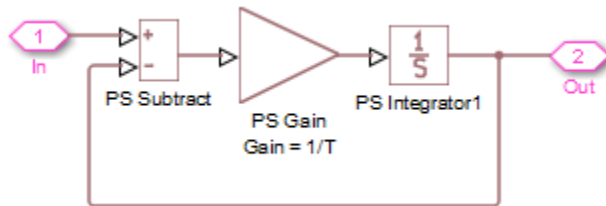
Description

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,

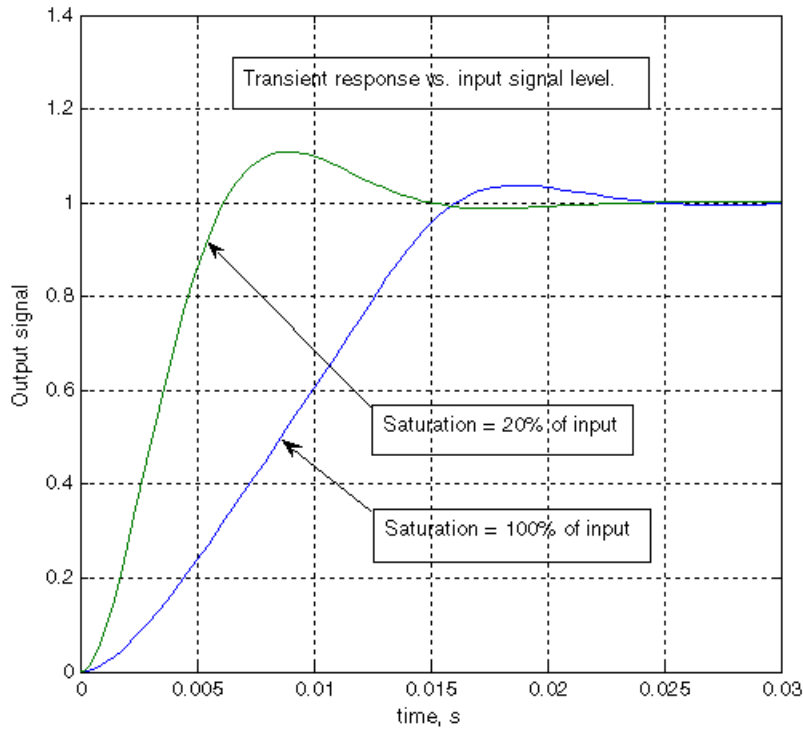


and is also built of the Physical Signals blocks: PS Subtract, PS Gain, and PS Integrator. The transfer function of the lag is

$$H(s) = \frac{1}{Ts + 1}$$

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 $1/\text{Time constant}$. 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

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 1-744). 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:

$$p = f \frac{(L + L_{eq})}{D_H} \frac{\rho}{2A^2} q \cdot |q| + \rho \cdot g(z_B - z_A)$$

$$f = \begin{cases} K_s / Re & \text{for } Re \leq Re_L \\ f_L + \frac{f_T - f_L}{Re_T - Re_L} (Re - Re_L) & \text{for } Re_L < Re < Re_T \\ \frac{1}{\left(-1.8 \log_{10} \left(\frac{6.9}{Re} + \left(\frac{r / D_H}{3.7} \right)^{1.11} \right) \right)^2} & \text{for } Re \geq Re_T \end{cases}$$

$$Re = \frac{q \cdot D_H}{A \cdot v}$$

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 1-741
- “Vertical Position Tab” on page 1-742
- “Variables Tab” on page 1-743

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 $1e-4 \text{ m}^2$.

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 1-744]. 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/s².

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 1-750). 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:

$$p = f \frac{(L + L_{eq})}{D_H} \frac{\rho}{2A^2} q \cdot |q| + \rho \cdot g(z_B - z_A)$$

$$f = \begin{cases} K_s / Re & \text{for } Re \leq Re_L \\ f_L + \frac{f_T - f_L}{Re_T - Re_L} (Re - Re_L) & \text{for } Re_L < Re < Re_T \\ \frac{1}{\left(-1.8 \log_{10} \left(\frac{6.9}{Re} + \left(\frac{r / D_H}{3.7} \right)^{1.11} \right) \right)^2} & \text{for } Re \geq Re_T \end{cases}$$

$$Re = \frac{q \cdot D_H}{A \cdot v}$$

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 1-747
- “Variables Tab” on page 1-748

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 $1e-4 \text{ m}^2$.

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 1-750]. 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 1.5×10^{-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.

e1_A

Physical signal input port that controls pipe elevation at port A.

e1_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

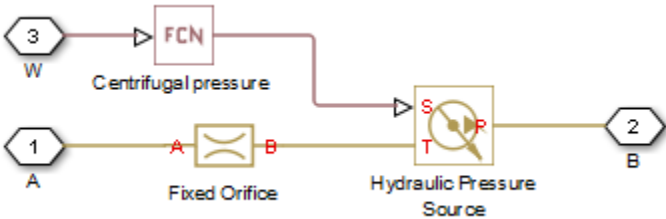


Library

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 $1e-4 \text{ m}^2$.

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 1-210 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

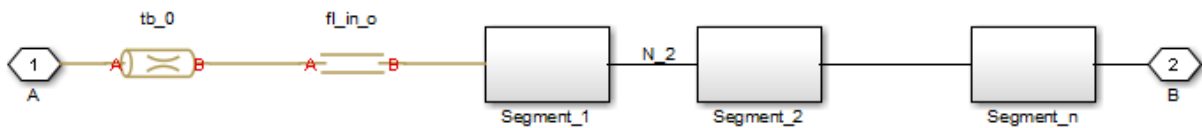


Library

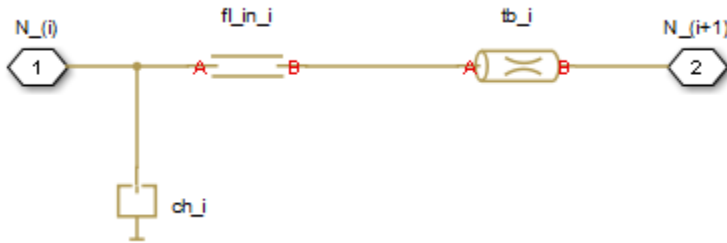
Pipelines

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, \dots, 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 $nn_0, nn_1, nn_2, \dots, nn_n$. The Constant Volume Hydraulic Chamber blocks are named ch_1, ch_2, \dots, ch_n , Hydraulic Resistive Tube blocks are named $tb_0, tb_1, tb_2, \dots, tb_n$, and Fluid Inertia blocks are named $fl_in_0, fl_in_1, fl_in_2, \dots, fl_in_n$.

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 m³/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

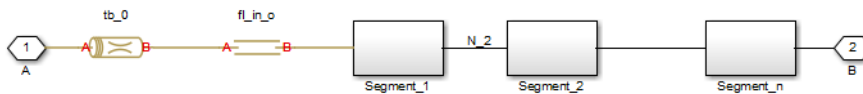


Library

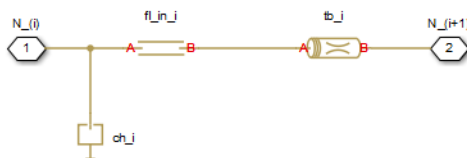
Low-Pressure Blocks

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, \dots, 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 $nn_0, nn_1, nn_2, \dots, nn_n$. The Constant Volume Hydraulic Chamber blocks are named ch_1, ch_2, \dots, ch_n , Resistive Pipe LP blocks are named $tb_0, tb_1, tb_2, \dots, tb_n$, and Fluid Inertia blocks are named $fl_in_0, fl_in_1, fl_in_2, \dots, fl_in_n$.

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 $p = p_A - p_B$.

Basic Assumptions and Limitations

Flow is assumed to be fully developed along the pipe length.

Parameters

- “Basic Parameters Tab” on page 1-764
- “Wall Compliance Tab” on page 1-765
- “Vertical Position Tab” on page 1-766

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 1.5×10^{-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 `0m`.

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 `0m`.

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/s².

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

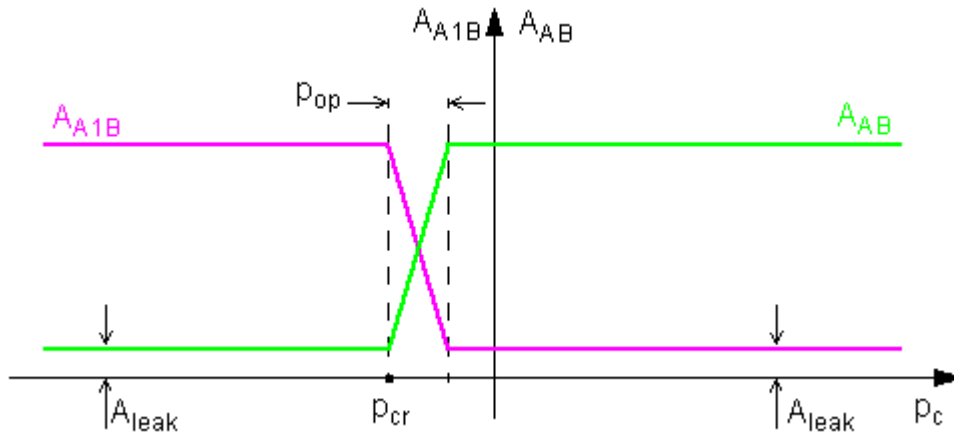
Description

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 ($p_{cr} \leq 0$).

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:

$$q_{AB} = C_D \cdot A_{AB} \sqrt{\frac{2}{\rho}} \cdot \frac{p_{AB}}{(p_{AB}^2 + p_{cr}^2)^{1/4}}$$

$$q_{A1B} = C_D \cdot A_{A1B} \sqrt{\frac{2}{\rho}} \cdot \frac{p_{A1B}}{(p_{A1B}^2 + p_{cr}^2)^{1/4}}$$

$$A_{AB} = \begin{cases} A_{leak} & \text{for } p_c \leq p_{crack} \\ A_{leak} + k \cdot (p_c - p_{crack}) & \text{for } p_{crack} < p_c < p_{crack} + p_{op} \\ A_{max} & \text{for } p_c \geq p_{crack} + p_{op} \end{cases}$$

$$A_{A1B} = \begin{cases} A_{leak} & \text{for } p_c \geq p_{crack} + p_{op} \\ A_{max} - k \cdot (p_c - p_{crack}) & \text{for } p_{crack} < p_c < p_{crack} + p_{op} \\ A_{max} & \text{for } p_c \leq p_{crack} \end{cases}$$

$$k = \frac{A_{max} - A_{leak}}{p_{op}}$$

$$p_c = p_A - p_{A1}$$

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:

$$p_{crAB} = \frac{\rho}{2} \left(\frac{Re_{cr} \cdot v}{C_D \cdot D_{HAB}} \right)^2$$

$$p_{crA1B} = \frac{\rho}{2} \left(\frac{Re_{cr} \cdot v}{C_D \cdot D_{HA1B}} \right)^2$$

$$D_{HAB} = \sqrt{\frac{4A_{AB}}{\pi}}$$

$$D_{HA1B} = \sqrt{\frac{4A_{A1B}}{\pi}}$$

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 $1e-4 \text{ m}^2$.

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 $1e-12$ m².

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 $1e-4$ m².

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

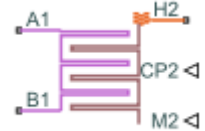
Check Valve | Pilot-Operated Check Valve

Introduced in R2006b

Simple Heat Exchanger (G)

Simple model of a heat exchanger between a gas network and a general fluid

Library: Fluid Network Interfaces / Heat Exchangers



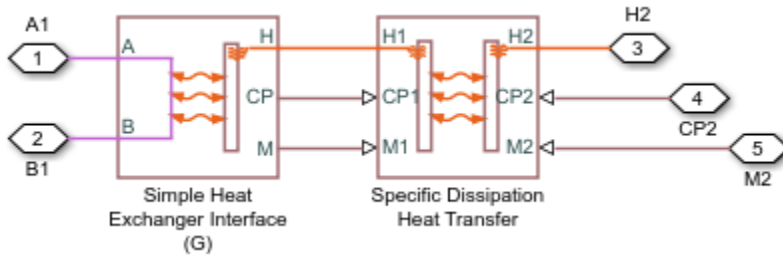
Description

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, $\text{SD}(\text{mdot1}, \text{mdot2})$ — 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 m³ (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 m² (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

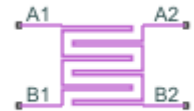
Simple Heat Exchanger Interface (G) | Specific Dissipation Heat Transfer

Introduced in R2017b

Simple Heat Exchanger (G-G)

Simple model of a heat exchanger between two gas networks

Library: Fluid Network Interfaces / Heat Exchangers



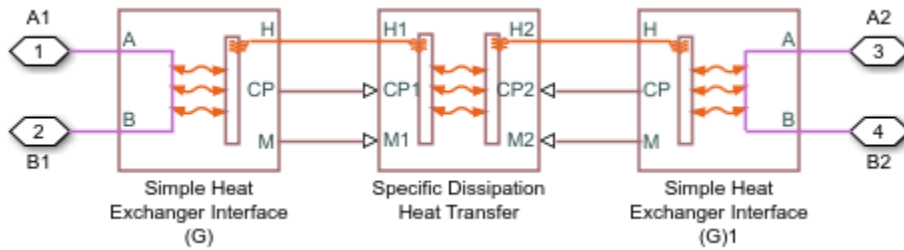
Description

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, \dot{m}_{1} — 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, \dot{m}_{2} — 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, $SD(\dot{m}_{1}, \dot{m}_{2})$ — 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 m³ (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 m² (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

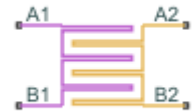
Simple Heat Exchanger Interface (G) | Specific Dissipation Heat Transfer

Introduced in R2017b

Simple Heat Exchanger (G-TL)

Simple model of a heat exchanger between a gas circuit and a thermal liquid network

Library: Fluid Network Interfaces / Heat Exchangers



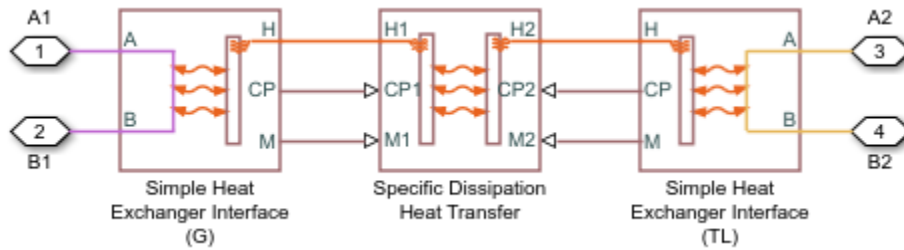
Description

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, $\text{SD}(\text{mdot1}, \text{mdot2})$ — 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 m³ (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 inlets0.01 m² (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 exchanger0.01 m³ (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 A2 and B2 — Flow area at the thermal liquid inlets
0.01 m² (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

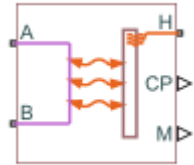
Simple Heat Exchanger Interface (G) | Simple Heat Exchanger Interface (TL) | Specific Dissipation Heat Transfer

Introduced in R2017b

Simple Heat Exchanger Interface (G)

Thermal interface between a gas and its surroundings

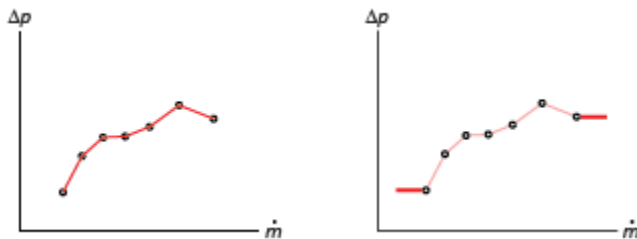
Library: Fluid Network Interfaces / Heat Exchangers /
Fundamental Components



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.

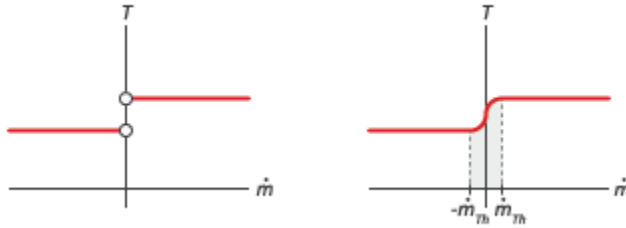
The pressure drop is calculated as a function of mass flow rate from tabulated data specified at some reference pressure and temperature. The calculation is based on linear interpolation if the mass flow rate is within the bounds of the tabulated data and on nearest-neighbor extrapolation otherwise. 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:

$$\frac{\partial M}{\partial p} \frac{dp}{dt} + \frac{\partial M}{\partial T} \frac{dT}{dt} = \dot{m}_A + \dot{m}_B,$$

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:

$$\frac{\partial E}{\partial p} \frac{dp}{dt} + \frac{\partial E}{\partial T} \frac{dT}{dt} = \phi_A + \phi_B + Q_H,$$

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:

$$p_A - p_B = \Delta p_A - \Delta p_B,$$

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 m³ (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 m² (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/m³ (default) | scalar with units of mass/volume

Density of the internal volume of gas at the start of simulation.

See Also

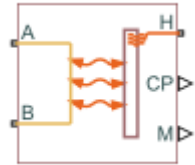
Simple Heat Exchanger (G) | Simple Heat Exchanger Interface (TL) | Specific Dissipation Heat Transfer

Introduced in R2017b

Simple Heat Exchanger Interface (TL)

Thermal interface between a thermal liquid and its surroundings

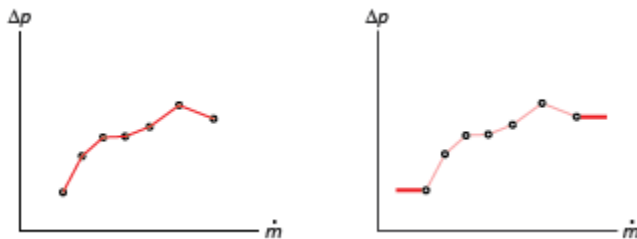
Library: Fluid Network Interfaces / Heat Exchangers /
Fundamental Components



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.

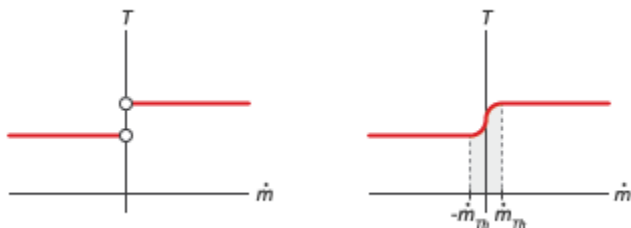
The pressure drop is calculated as a function of mass flow rate from tabulated data specified at some reference pressure and temperature. The calculation is based on linear interpolation if the mass flow rate is within the bounds of the tabulated data and on nearest-neighbor extrapolation otherwise. 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:

$$\dot{m}_A + \dot{m}_B = \begin{cases} \left(\frac{dp}{dt} \frac{1}{\beta} - \frac{dT}{dt} \alpha \right) \rho V, & \text{if **Thermal Liquid dynamic compressibility** is On} \\ 0, & \text{otherwise} \end{cases}$$

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:

$$\frac{\partial E}{\partial p} \frac{dp}{dt} + \frac{\partial E}{\partial T} \frac{dT}{dt} = \phi_A + \phi_B + Q_H,$$

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:

$$p_A - p_B = \Delta p_A - \Delta p_B,$$

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:

$$\Delta p_* = \Delta p(\dot{m}_*) \frac{\rho_R}{\rho_*},$$

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 :

$$\rho_{*,\text{smooth}} = \rho_* \left(\frac{1 + \alpha}{2} \right) + \rho \left(\frac{1 - \alpha}{2} \right),$$

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 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:

$$\dot{m}_{\text{avg}} = \frac{\dot{m}_A + \dot{m}_B}{2}$$

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.

T — 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 m³ (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 m² (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.

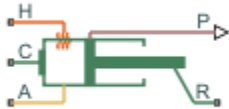
See Also

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



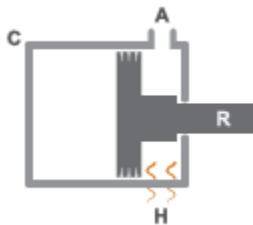
Library

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

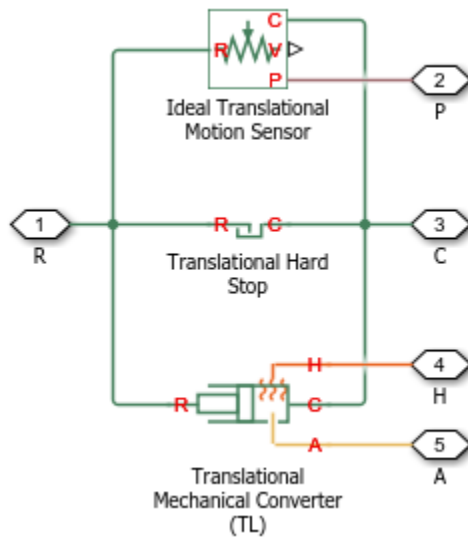
The direction of the piston motion depends on the mechanical orientation setting in the block dialog box. If the mechanical orientation is positive, then a positive gauge pressure at port A yields a positive piston translation relative to the actuator case. The direction of motion reverses for a negative 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 $1\text{e-}5 \text{ m}^3$.

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



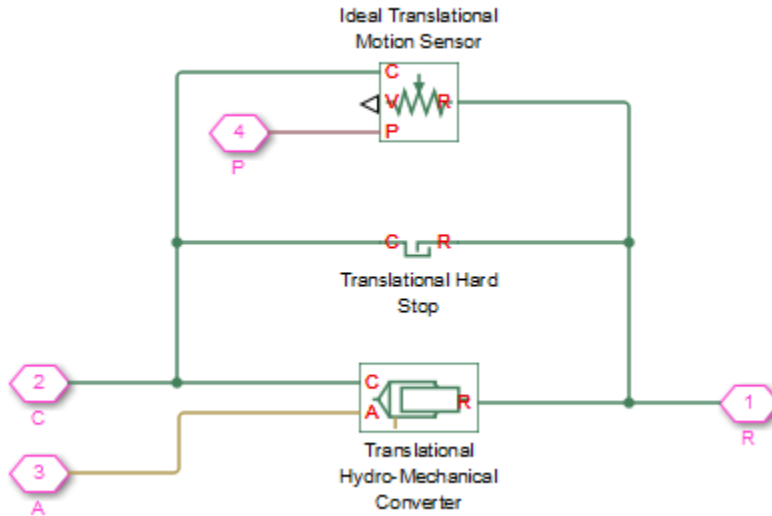
Library

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 $1e-4 \text{ m}^3$.

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 $1e6 \text{ N/m}$.

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 $5e-4 \text{ m}^2$.

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



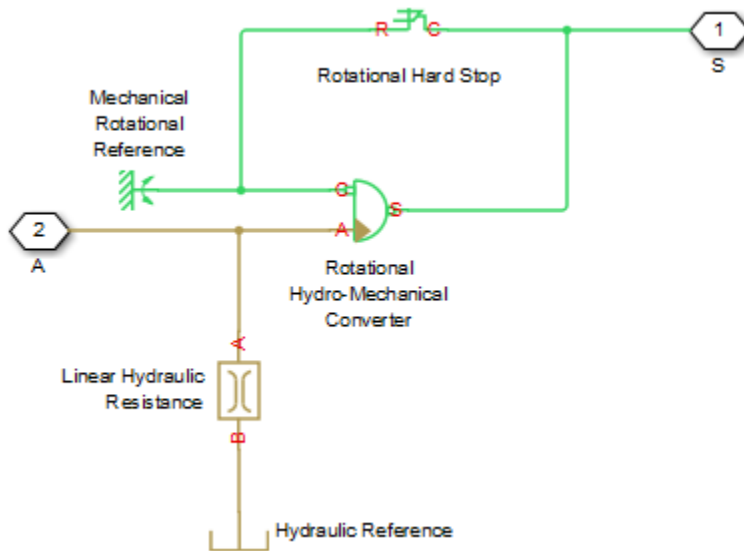
Library

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 $4.5 \times 10^{-5} \text{ m}^3/\text{rad}$.

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 $1 \times 10^{-4} \text{ m}^3$.

Leak coefficient

Leak coefficient for the Linear Hydraulic Resistance block. The default value is $1 \times 10^{-14} \text{ (m}^3/\text{s)/Pa}$.

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 $1 \times 10^6 \text{ N}\cdot\text{m}/\text{rad}$.

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 $150 \text{ N}\cdot\text{m}/(\text{rad}/\text{s})$.

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

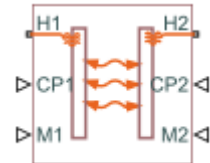
[Linear Hydraulic Resistance](#) | [Rotational Hard Stop](#) | [Rotational Hydro-Mechanical Converter](#)

Introduced in R2007a

Specific Dissipation Heat Transfer

Simple heat transfer model between two general fluids

Library: 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:

$$Q = SD(T_{1,\text{in}} - T_{2,\text{in}}),$$

where $T_{*,in}$ 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:

$$SD = SD(\dot{m}_1, \dot{m}_2),$$

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:

$$SD_{max} = \min(C_1, C_2),$$

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:

$$Q = \begin{cases} SD_{max} (T_{1,in} - T_{2,in}), & \text{if } SD > SD_{max} \\ SD (T_{1,in} - T_{2,in}), & \text{otherwise} \end{cases},$$

A warning is issued whenever the heat flow rate exceeds the maximum value,

$SD_{max} (T_{1,in} - T_{2,in})$, 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, $\dot{m}2$ — 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, $SD(\dot{m}1, \dot{m}2)$ — 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,

$$A = \begin{cases} b \cdot \sqrt{x^2 + \delta^2} & \text{for } x > 0 \\ b \cdot \delta & \text{for } x \leq 0 \end{cases}$$

For round holes,

$$A = \begin{cases} A_{\min} + Z \frac{d^2}{8} (\alpha - \sin \alpha) & \text{for } d \geq x \geq 0 \\ A_{\min} & \text{for } x < 0 \\ A_{\min} + A_{\max} & \text{for } x > d \end{cases}$$

$$\alpha = 2 \cdot \arccos \left(1 - 2 \frac{x}{d} \right)$$

$$A_{\min} = Z \cdot d \cdot \delta$$

$$A_{\max} = \pi d^2 / 4$$

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

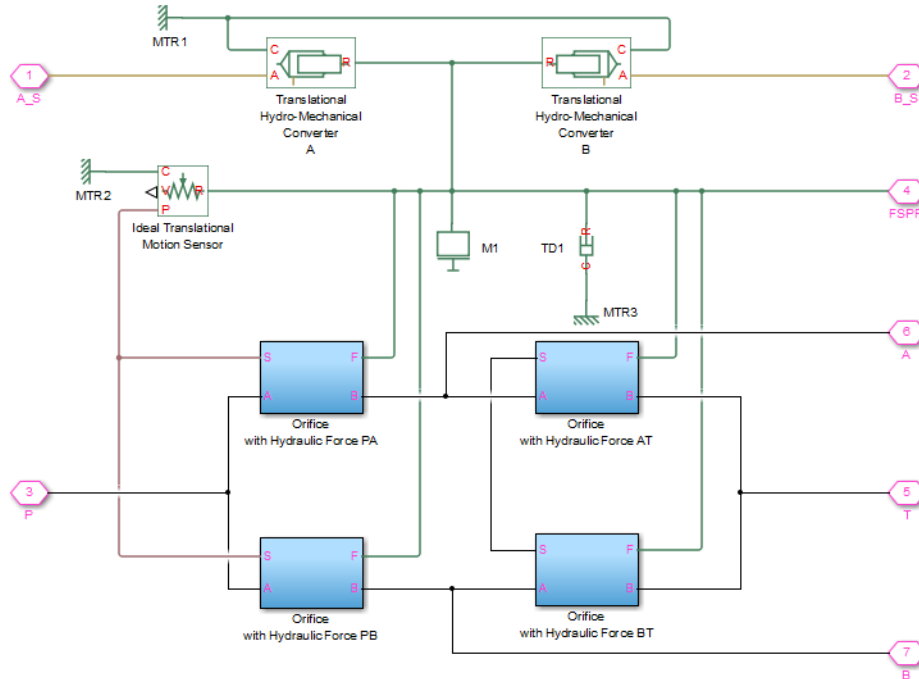
Physical signal port that provides the spool displacement.

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.

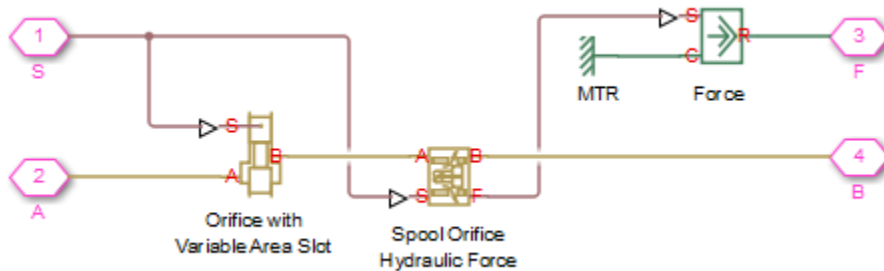


The spool (mass M1, viscous friction TD1) is shifted by the servo-actuator simulated by two Translational Hydro-Mechanical Converter blocks. Connections A_S and B_S are hydraulic ports for applying pilot control pressure.

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

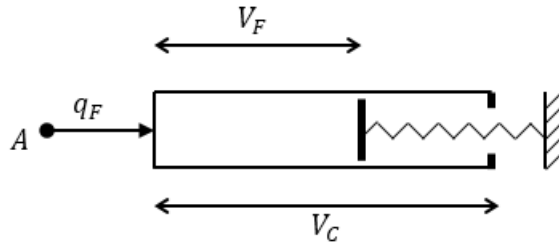
This block models a spring-loaded hydraulic fluid accumulator. The accumulator consists of a preloaded spring and a fluid chamber. The fluid chamber is connected to a hydraulic system.

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:

$$p_{spr} - p_{pr} = K_{spr} V_F$$

$$p_F = p_{spr} + p_{HS}$$

$$K_{spr} = \frac{p_{max} - p_{pr}}{V_C}$$

$$p_{HS} = \begin{cases} K_S (V_F - V_C) + K_d q_F^+ (V_F - V_C) & \text{if } V_F \geq V_C \\ K_S V_F - K_d q_F^- V_F & \text{if } V_F \leq 0 \\ 0 & \text{otherwise} \end{cases}$$

$$q_F^+ = \begin{cases} q_F & \text{if } q_F \geq 0 \\ 0 & \text{otherwise} \end{cases}$$

$$q_F^- = \begin{cases} q_F & \text{if } q_F \leq 0 \\ 0 & \text{otherwise} \end{cases}$$

where

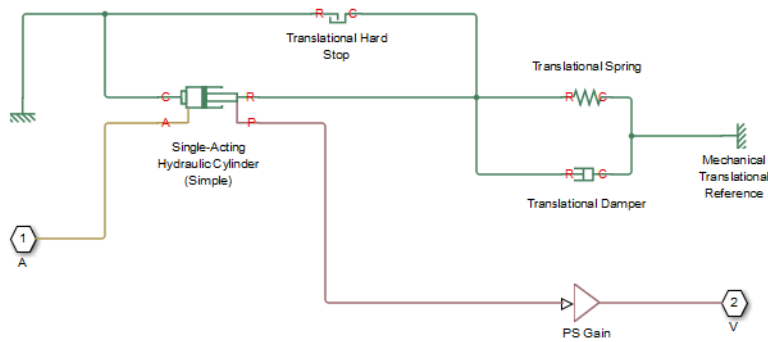
V_F	Volume of fluid in the accumulator
V_{init}	Initial volume of fluid in the accumulator
V_C	Fluid chamber capacity
p_F	Pressure at the accumulator inlet (gauge)
p_{pr}	Preload pressure (gauge)
K_{spr}	Spring gain coefficient
p_{max}	Pressure needed to fully fill the accumulator
p_{spr}	Pressure developed 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 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 1-844
- “Variables Tab” on page 1-845

Parameters Tab

Fluid chamber capacity

Amount of fluid that the accumulator can hold. The default value is $8e-3 \text{ m}^3$.

Preload pressure (gauge)

Spring pressure (gauge) when the fluid chamber is empty. The default value is $10e5 \text{ Pa}$.

Pressure at full capacity (gauge)

Spring pressure (gauge) when the fluid chamber is at capacity. The default value is $30e5 \text{ Pa}$.

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/m³.

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/m⁶.

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 1-852):

$$K_{SE} = K_{cor} \left(1 - \frac{A_S}{A_L} \right)^2$$

$$K_{SC} = K_{cor} \cdot 0.5 \left(1 - \frac{A_S}{A_L} \right)^{0.75}$$

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

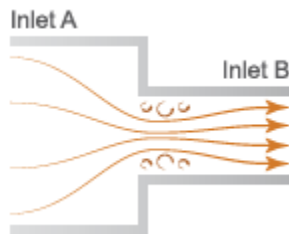


Library

Thermal Liquid/Pipes & Fittings

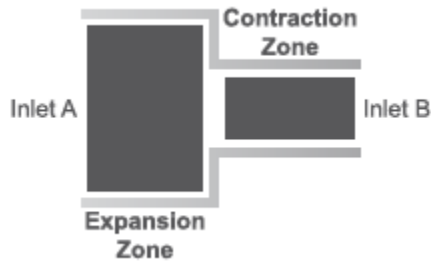
Description

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

$$\lambda = 3\bar{m}^2 - 2\bar{m}^3,$$

and

$$\bar{m}_{Cr} = Re_{Cr} \sqrt{\frac{\pi}{4} S_B \mu}.$$

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 $2e-2 \text{ m}^2$.

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

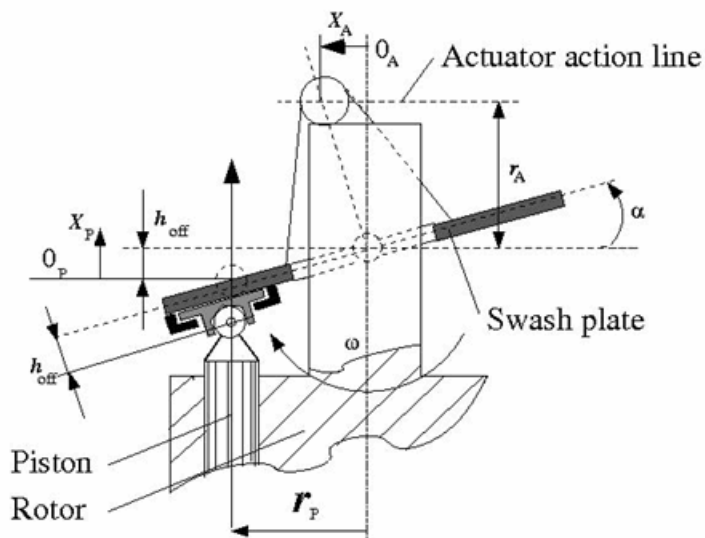


Library

Pumps and Motors

Description

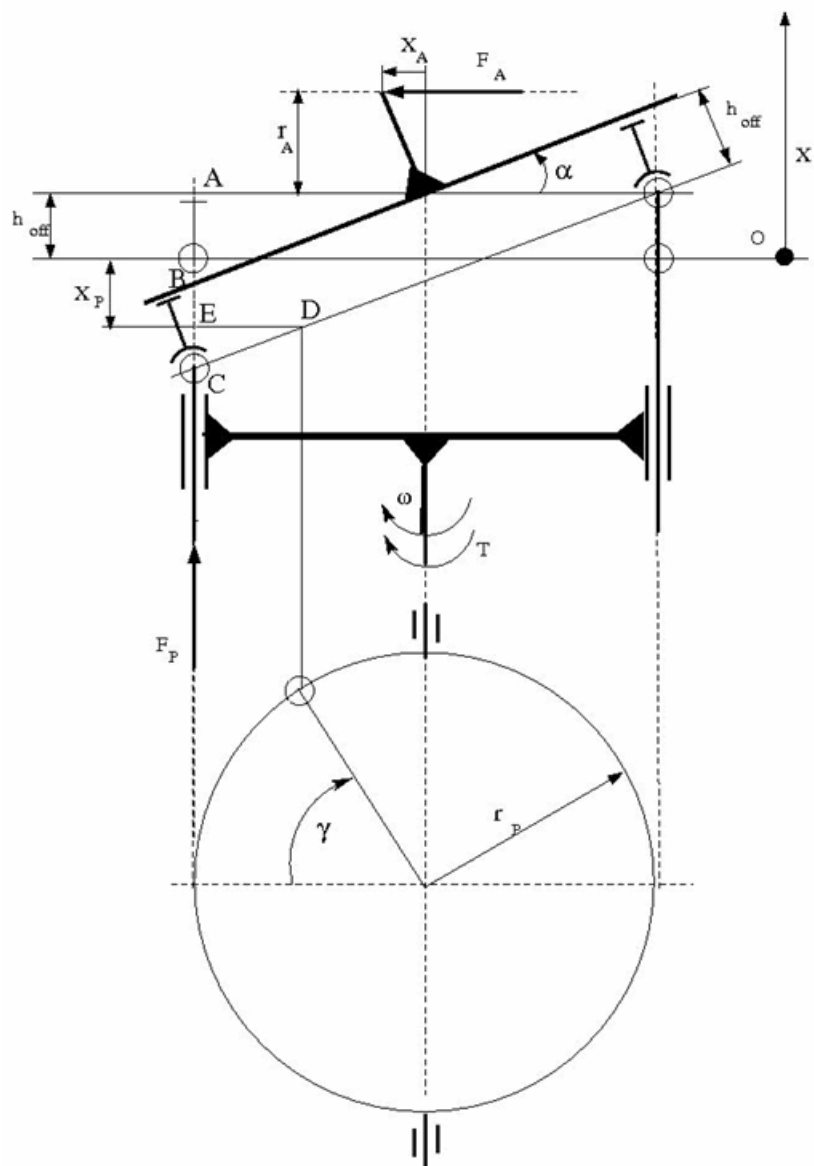
The Swash Plate block is a model of a swash plate mechanism used in axial-piston hydraulic pumps and motors to convert translational motion of a piston into rotational motion of a drive shaft. The mechanism is a combination of a cylinder block or rotor, with the piston fitted inside, and an angled swash 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 follows the plate, which causes the piston to reciprocate.

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.



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:

$$x_P = -(AB - h_{off} + BC + DE \cdot tg\alpha)$$

$$tg\alpha = \frac{x_A}{r_A}$$

which, after minor rearrangement, yields:

$$x_P = -\frac{r_P}{r_A} x_A \cos(\gamma + \beta) - h_{off} \left(\sqrt{1 + \frac{x_A^2}{r_A^2}} - 1 \right)$$

$$\gamma = \omega \cdot t$$

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:

$$F_P \delta x_P + F_A \delta x_A + (T - T_{res}) \delta \gamma = 0$$

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” on page 1-862 and substituted into “Equation 1-12” on page 1-863. 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:

$$T_{res} = \mu \cdot r_P \cdot F_P$$

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

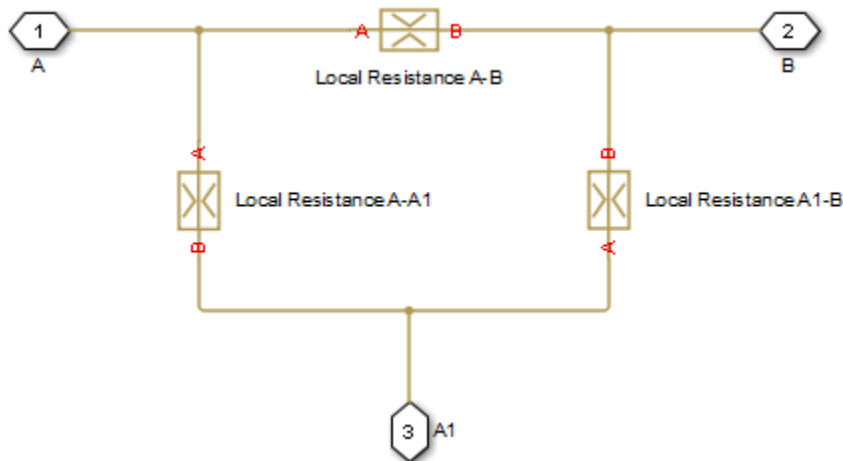


Library

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

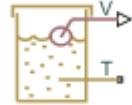
Elbow | Gradual Area Change | Local Resistance | Pipe Bend | Sudden Area Change

Introduced in R2006b

Tank

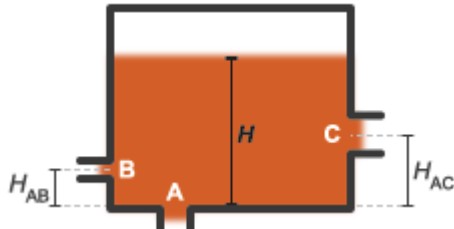
Pressurized fluid container with variable fluid level

Library: 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:

$$q_i = A_i \sqrt{\frac{2}{K_i \rho}} \frac{\Delta p_i}{(\Delta p_i^2 + p_{Cr,i}^2)^{1/4}},$$

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:

$$p_{Cr,i} = K_i \frac{\rho}{2} \left(\frac{Re_{Cr} v}{d_i} \right),$$

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:

$$\Delta p_i = p_{port,i} - (p_{press} + p_{elev,i}),$$

where:

- $p_{port,i}$ is the pressure at the port denoted by i .
- p_{press} is the specified tank **Pressurization** value.
- $p_{elev,i}$ 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 m² (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 m³ (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.

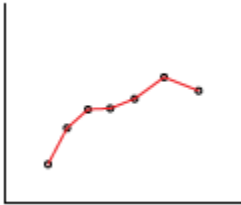
Dependencies

This parameter is active when the **Tank volume parameterization** parameter is set to Tabulated data – volume vs. level.

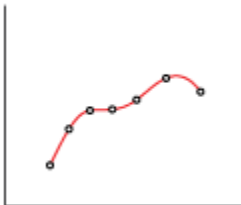
Interpolation method — Method of computing values inside the lookup table range

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.



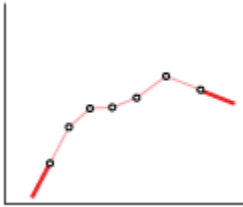
Dependencies

This parameter is active when the `Tank volume parameterization` parameter is set to `Tabulated data - Volume vs. level`.

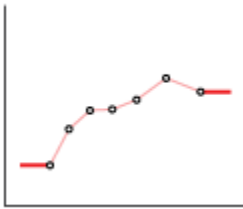
Extrapolation method — Method of computing values outside of the lookup table range

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/s² (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.

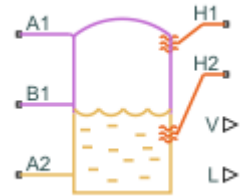
See Also

Constant Head Tank | Reservoir

Tank (G-TL)

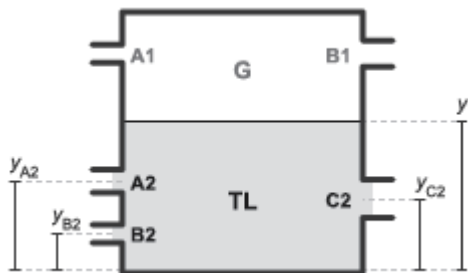
Pressurized tank with variable gas and thermal liquid volumes

Library: 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:

$$V_T = V_L + V_G,$$

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:

$$\dot{V}_G = -\dot{V}_L.$$

The time rate of change of the thermal liquid volume is calculated by differentiating the expression:

$$M_L = \rho_L V_L,$$

where M is mass and ρ is density. The differentiation gives the mass flow rate into the thermal liquid volume:

$$\dot{M}_L = V_L \dot{\rho}_L + \dot{V}_L \rho_L,$$

The time rate of change of the thermal liquid density is:

$$\dot{\rho}_L = \frac{\rho_L}{\beta_L} \dot{p}_L - \alpha_L \rho_L \dot{T}_L,$$

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:

$$\dot{V}_G = -\dot{V}_L \approx V_L \left(\frac{\dot{p}_L}{\beta_L} - \alpha_L \dot{T}_L \right) - \frac{\dot{M}_L}{\rho_L}$$

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:

$$\dot{M}_L = \sum_{i=A2, B2, C2} \dot{m}_i,$$

where \dot{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:

$$\dot{M}_L = V_L \left(\frac{\rho_L}{\beta_L} \dot{p}_G - \alpha_L \rho_L \dot{T}_L \right) + \dot{V}_L \rho_L,$$

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:

$$\dot{M}_G = \sum_{i=A1, B1} \dot{m}_i,$$

where \dot{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:

$$\dot{M}_G = \left. \frac{dM}{dp} \right|_G \dot{p}_G + \left. \frac{dM}{dT} \right|_G \dot{T}_G + \dot{V}_G \rho_G,$$

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 :

$$\left. \frac{dM}{dp} \right|_G \dot{p}_G + \left. \frac{dM}{dT} \right|_G \dot{T}_G + \left[V_L \left(\frac{\dot{p}_G}{\beta_L} - \alpha_L \dot{T}_L \right) - \frac{\dot{M}_L}{\rho_L} \right] \rho_G = \sum_{i=A1, B1} \dot{m}_i.$$

Rearranging terms gives the final expression for the mass balance in the gas volume:

$$\dot{p}_G \left(\left. \frac{dM}{dp} \right|_G + \frac{\rho_G V_L}{\beta_L} \right) + \left(\dot{T}_G \left. \frac{dM}{dT} \right|_G - \dot{T}_L \rho_G V_L \alpha_L \right) = \sum_{i=A1, B1} \dot{m}_i + \frac{\rho_G}{\rho_L} \sum_{i=A2, B2, C2} \dot{m}_i,$$

where \dot{M}_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:

$$\dot{p}_G \left(\left. \frac{dU}{dp} \right|_G + \rho_G h_G \frac{V_L}{\beta_L} \right) + \left(\dot{T}_G \left. \frac{dU}{dT} \right|_G - \dot{T}_L \rho_G h_G V_L \alpha_L \right) = Q_{H1} + \sum_{i=A1, B1} \phi_i + \frac{\rho_G}{\rho_L} h_G \sum_{i=A2, B2, C2} \dot{m}_i,$$

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:

$$\dot{p}_L \left. \frac{dU}{dp} \right|_L + \dot{T}_L \left. \frac{dU}{dT} \right|_L = Q_{H2} + \sum_{i=A2, B2, C2} \phi_i + \sum_{i=A2, B2, C2} \dot{m}_i g (y(i) - y) - h_L \sum_{i=A2, B2, C2} \dot{m}_i,$$

where the pressure derivative is:

$$\left. \frac{dU}{dp} \right|_L = -T_L \alpha_L V_L,$$

and the temperature derivative is:

$$\left. \frac{dU}{dT} \right|_L = c_{p,L} \rho_L V_L,$$

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:

$$p_{A1} = p_{B1} = p_G.$$

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 ($p_L = p_G$). Including the dynamic pressures ($p_{i,dyn}$) at the inlets:

$$p_i + p_{i,dyn} = p_G + \rho_L (y - y_i) g,$$

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:

$$p_{i,dyn} = \begin{cases} \frac{1}{2} \rho_L v_i^2, & \text{if } \dot{m}_i < 0 \\ 0, & \text{if } \dot{m}_i \geq 0 \end{cases}$$

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 m³ (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 m² (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] m³ (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] m² (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 m² (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] m² (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] m² (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/s² (default) | scalar number with units of length/time²

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/m³ (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 m³ (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.

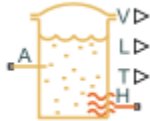
See Also

Tank | Tank (TL)

Introduced in R2017b

Tank (TL)

Thermal liquid container with variable fluid volume



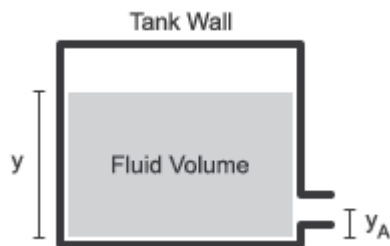
Library

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

$$p_A + p_{dyn} = p_{Ref} + \rho g(y - y_A),$$

where:

- p_A is the fluid pressure at inlet A.
- p_{Ref} is the constant tank pressurization.
- p_{dyn} is the dynamic pressure:

$$p_{\text{dyn}} = \begin{cases} 0, & \dot{m}_A \geq 0 \\ \frac{\dot{m}_A^2}{2\rho_A S_A^2}, & \dot{m}_A < 0 \end{cases}$$

- 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

$$m(C_p - h\alpha)\dot{T} = \Phi_A - \dot{m}_A h + Q,$$

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 0.01 m^2 .

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 $5\text{e}+3 \text{ kg}$.

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 (TL)

Flow control valve with temperature-based actuation



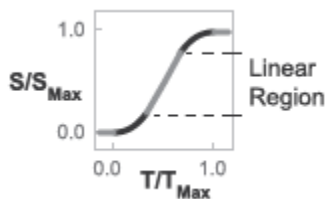
Library

Thermal Liquid/Valves/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

$$S_{Linear} = \left(\frac{S_{End} - S_{Start}}{T_{Range}} \right) (T_{Sensor} - T_{Activation}) + S_{Start},$$

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.
- $T_{Activation}$ 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:

$$\frac{d}{dt}(T_{Sensor}) = \frac{T_{Avg} - T_{Sensor}}{\tau},$$

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

$$\lambda_L = 3\bar{T}_L^2 - 2\bar{T}_L^3$$

and

$$\lambda_R = 3\bar{T}_R^2 - 2\bar{T}_R^3$$

where:

$$\bar{T}_L = \frac{T_{Sensor} - T_{Activation}}{\Delta T_{smooth}}$$

and

$$\bar{T}_R = \frac{T_{Sensor} - (T_{Activation} + T_{Range} - \Delta T_{smooth})}{\Delta T_{smooth}}$$

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

$$S_R = \begin{cases} S_{Start}, & T_{Sensor} \leq T_{Activation} \\ S_{Start} (1 - \lambda_L) + S_{Linear} \lambda_L, & T_{Sensor} < T_{Activation} + \Delta T_{smooth} \\ S_{Linear}, & T_{Sensor} < T_{Activation} + T_{Range} - \Delta T_{smooth}, \\ S_{Linear} (1 - \lambda_R) + S_{End} \lambda_R, & T_{Sensor} < T_{Activation} + T_{Range} \\ S_{End}, & T_{Sensor} \geq T_{Activation} + T_{Range} \end{cases}$$

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:

$$\dot{m}_{cr} = Re_{cr} \mu_{Avg} \sqrt{\frac{\pi}{4} S_R}.$$
- 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:

$$PR_{Loss} = \frac{\sqrt{1 - (S_R/S)^2 (1 - C_d^2)} - C_d (S_R/S)}{\sqrt{1 - (S_R/S)^2 (1 - C_d^2)} + C_d (S_R/S)}$$

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

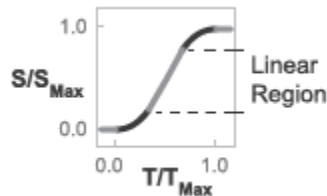
Valve flow area in the fully open position. The default value is $1e-4 \text{ m}^2$.

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 $1e-12 \text{ m}^2$.

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

Valve Actuator

Simplified model of valve driver



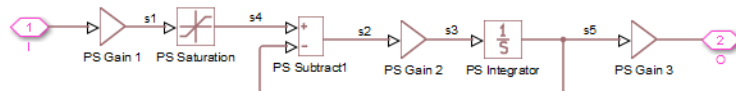
Library

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 \pm valve stroke range. The gain of the PS Gain 2 block is $1/\mathbf{Time\ constant}$, which makes it possible to reproduce the first-order lag by combining this block with the PS Subtract and PS Integrator blocks.

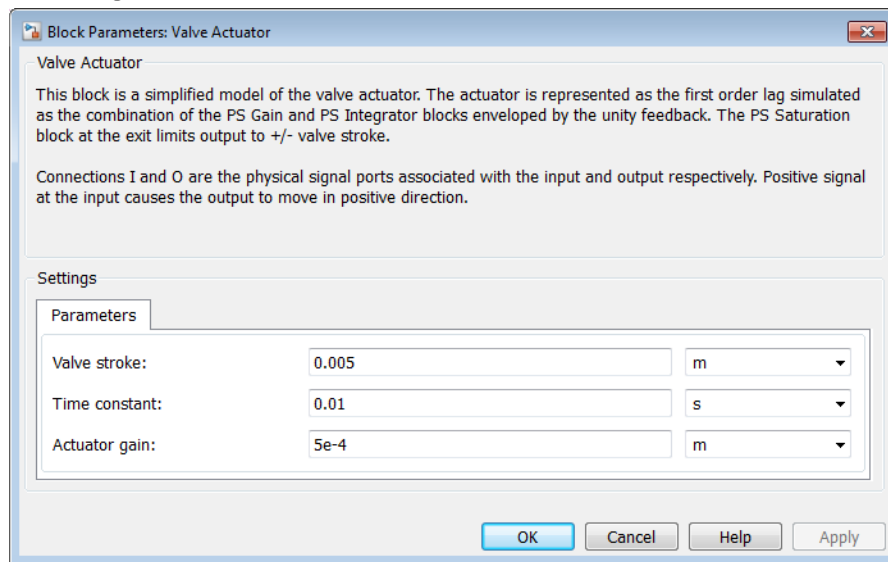
The transfer function of the lag is

$$H(s) = \frac{1}{Ts + 1}$$

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.

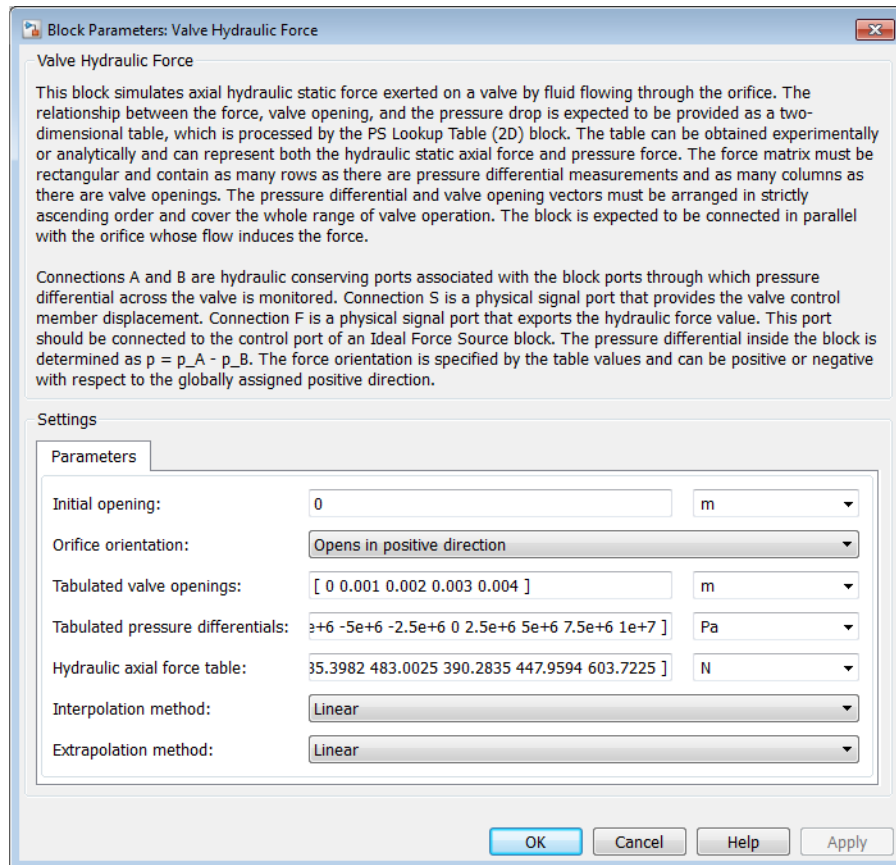
Connections A and B are hydraulic conserving ports that should be connected to the valve block ports in such a way as to monitor the pressure differential across the valve. Connection S is a physical signal port that provides the valve control member displacement. Connection F is a physical signal port 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 $p = p_A - p_B$. 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

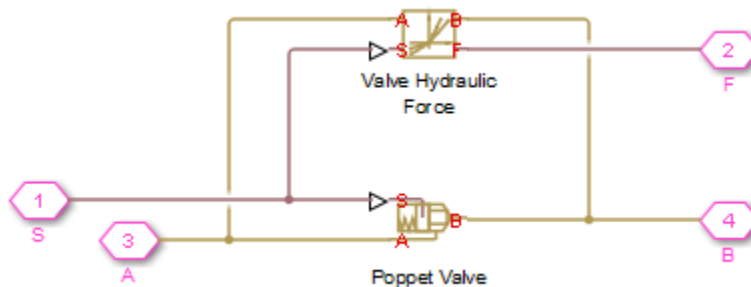
Physical signal port that provides the valve control member displacement.

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)

Orifice with variable cross-sectional area specified through physical signal input



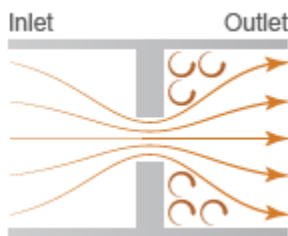
Library

Thermal Liquid/Orifices

Description

The Variable Area Orifice (TL) block models the pressure drop due to an orifice with a variable cross-sectional area. Physical signal input port S provides the displacement of the orifice control member. The block computes the orifice area from the control member displacement through a smoothed linear model or from tabular data.

The orifice consists of a contraction followed by a sudden expansion in flow area. The contraction causes the fluid to accelerate and its pressure to drop. The expansion recovers the lost pressure though only in part, as the flow separates from the wall, losing momentum in the process. You can ignore this pressure recovery by setting the **Pressure recovery** parameter to `Off`.



The orifice opening behavior depends on the opening orientation specified in the block dialog box. If the **Opening orientation** parameter is set to `Positive`, a positive control member displacement increases the orifice area. If the **Opening orientation** parameter is set to `Negative`, a positive displacement decreases the orifice area.

The orifice is adiabatic. It does not exchange heat with the environment. This block provides a building block for the n -way thermal liquid directional valves.

Orifice Area

The orifice area calculation depends on the parameterization selected in the block dialog box. If the **Orifice area parameterization** setting is `Tabulated data – area vs. opening`, the block obtains the orifice area from tabular data specified in terms of the control member opening, or position:

$$S_R = S_R(l),$$

where:

- S_R is the orifice cross-sectional area.
- l is the control member position.

The control member position is a function of the control member displacement provided through physical signal port S:

$$l = l_0 + \varepsilon d_S,$$

where:

- l_0 is the control member offset from the zero position.
- ε is an integer denoting the orifice orientation—1 if positive and -1 if negative.
- d_S is the control member displacement specified through physical signal port S.

If the **Orifice area parameterization** setting is `Linear area-opening relationship`, the block computes the orifice area directly from orifice geometry parameters. The orifice area calculation uses a linear function of the control member position as a starting point:

$$S_{Linear} = \left(\frac{S_{Max}}{l_{Max}} \right) l,$$

where:

- S_{Linear} is the orifice area in the linear orifice-opening range.
- S_{Max} is the maximum orifice area.
- l_{Max} is the maximum control member displacement.

This linear expression introduces undesirable discontinuities at the fully open and fully closed positions. The block eliminates these discontinuities through smoothing expressions given by the piecewise function:

$$S_R = \begin{cases} S_{Leak}, & l \leq l_{Min} \\ S_{Leak} (1 - \lambda_L) + S_{Linear} \lambda_L, & l < l_{Min} + \Delta l_{smooth} \\ S_{Linear}, & l \leq l_{Max} - \Delta l_{smooth}, \\ S_{Linear} (1 - \lambda_R) + S_{Max} \lambda_R, & l < l_{Max} \\ S_{Max}, & l \geq l_{Max} \end{cases}$$

where:

- S_{Leak} is the leakage area between the orifice inlets.
- l_{Min} is the minimum control member position:

$$l_{Min} = l_{Max} \left(\frac{S_{Leak}}{S_{Max}} \right)$$

- Δl_{smooth} is the portion of the linear orifice area function $S_R(l)$, to smooth:

$$\Delta l_{smooth} = f_{smooth} \frac{l_{max} - l_{Min}}{2},$$

- f_{smooth} is a smoothing factor from 0 through 1. This value is the fraction of the S_R function to smooth.
- λ_L and λ_R are the cubic polynomial smoothing functions

$$\lambda_L = 3\overline{\Delta L}_L^2 - 2\overline{\Delta L}_L^3,$$

and

$$\lambda_R = 3\overline{\Delta L}_R^2 - 2\overline{\Delta L}_R^3,$$

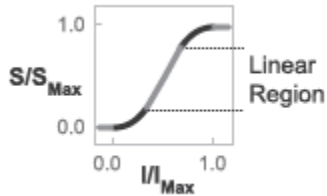
where

$$\overline{\Delta L}_L = \frac{l - l_{Min}}{\Delta l_{smooth}}$$

and

$$\overline{\Delta L}_R = \frac{l - (l_{Max} - \Delta l_{smooth})}{\Delta l_{smooth}}.$$

A smoothing factor of 0 corresponds to no smoothing anywhere in the S_R range. The orifice area reduces to the linear expression given by S_{Linear} in the open interval $]l_{Min}, l_{Max}[$. A value of 1 corresponds to full smoothing in the entire S_{Linear} range. The orifice area becomes the piecewise nonlinear function given by S_R .



Orifice Area Smoothing

Mass Balance

The mass balance equation in the orifice is

$$\dot{m}_A + \dot{m}_B = 0,$$

where:

- \dot{m}_A and \dot{m}_B are the mass flow rates into the orifice through ports A and B.

Momentum Balance

The momentum balance equation in the orifice is

$$p_A - p_B = \frac{\dot{m} \sqrt{\dot{m}^2 + \dot{m}_{cr}^2}}{2\rho C_d^2 S_R^2} \left[1 - \left(\frac{S_R}{S} \right)^2 \right] PR_{Loss},$$

where:

- p_A and p_B are the thermal liquid pressures at ports A and B.
- \dot{m} is the average mass flow rate through the orifice.
- \dot{m}_{cr} is the maximum flow rate in the laminar flow regime:

$$\dot{m}_{cr} = Re_{cr} \sqrt{\frac{\pi}{4}} A \mu$$

- ρ is the mass density in the orifice.
- C_d is the discharge coefficient.
- S is the cross-sectional area of the adjacent pipe segments.
- μ is the average dynamic viscosity in the orifice.
- PR_{Loss} is the pressure loss ratio [1 on page 1-921]:

$$PR_{Loss} = \begin{cases} \frac{\sqrt{1 - (S_R/S)^2 (1 - C_d^2)} - C_d (S_R/S)}{\sqrt{1 - (S_R/S)^2 (1 - C_d^2)} + C_d (S_R/S)}, & \text{if Pressure recovery is 'On'} \\ 1, & \text{if Pressure recovery is 'Off'} \end{cases}$$

Energy Balance

The energy conservation equation in the orifice gives

$$\phi_A + \phi_B = 0,$$

where:

- ϕ_A and ϕ_B are the energy flow rates into the orifice through ports A and B.

Parameters

Parameters Tab

Orifice area parameterization

Orifice area calculation approach. Select `Tabulated data – Area vs. opening` provide the orifice area explicitly through tabular data. Select `Linear area-opening relationship` to compute the orifice area from the control member displacement through a smoothed linear expression. The default parameterization is `Linear area-opening relationship`.

Maximum control displacement

Control member displacement in the fully open state. The block saturates the input physical signal S at this value. This parameter appears only when **Orifice area parameterization** is set to `Linear area-opening relationship`. The default value is 0.005 m.

Maximum orifice area

Orifice area in the fully open state. This parameter appears only when **Orifice area parameterization** is set to `Linear area-opening relationship`. The default value is $1e-4 \text{ m}^2$.

Leakage area

Minimum orifice area associated with fluid leakage between port A and port B. The default value is $1e-10$. This parameter appears only when **Orifice area parameterization** is set to `Linear area-opening relationship`.

Smoothing factor

Fraction of the displacement-area curve to smooth at the minimum and maximum displacement points. Smoothing eliminates discontinuities that reduce accuracy and slow down simulation speed. The smoothing factor must be between 0 and 1.

Enter a value of 0 to apply no smoothing to the displacement-area curve. Enter a value of 1 to smooth the entire length of the curve. The default value is 0.01. This parameter appears only when **Orifice area parameterization** is set to `Linear area-opening relationship`.

Control displacement vector

Vector with the control displacements for the displacement-area lookup table. These are the displacements at which you specify the orifice areas in the **Orifice area vector** parameter. This parameter appears only when **Orifice area parameterization** is set to `Tabulated data - Area vs. opening`. The default vector is a five-element array ranging from $1.0e-9$ to 0.00034356 .

Orifice area vector

Vector with the orifice areas for the displacement-area lookup table. The areas must correspond to the control member displacements specified in the **Control displacement vector** parameter. This parameter appears only when **Orifice area parameterization** is set to `Tabulated data - Area vs. opening`. The default vector is a five-element array ranging from -0.002 to 0.015 .

Opening orientation

Orientation of the orifice control member. If the opening orientation is `Positive`, a positive control member displacement opens the orifice. If the opening orientation is negative, a positive control member displacement closes the orifice. The default setting is `Positive`.

Control member offset

Control member offset from the fully closed position at a zero displacement. The default value is 0 m.

Cross-sectional area at ports A and B

Flow area at ports A and B. This area is the same for both ports. The default value is 0.01 m^2 .

Characteristic longitudinal length

Distance between the orifice inlets. This parameter provides a measure of the orifice longitudinal scale. The default value is 0.1 m.

Pressure recovery

Pressure recovery calculation approach. Select On to account for the pressure recovery in the orifice expansion zone. The pressure recovery depends on the orifice and pipe diameters. The default setting is On.

Discharge coefficient

Ratio of the actual mass flow rate through the orifice to its ideal, or theoretical, value. The discharge coefficient accounts for the effects of orifice geometry on the mass flow rate. The value must be between 0 and 1. The default value is 0.7.

Critical Reynolds number

Reynolds number at which flow transitions between laminar and turbulent regimes. The 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 port representing inlet A
- B — Thermal liquid port representing inlet B
- S — Physical signal input port for the control member position

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

Double-Acting Actuator (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:

$$q = C_D \cdot A(h) \sqrt{\frac{2}{\rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

$$p = p_A - p_B$$

$$h = x_0 + x \cdot or$$

$$A(h) = \begin{cases} h \cdot A_{\max} / h_{\max} + A_{leak} & \text{for } h > 0 \\ A_{leak} & \text{for } h \leq 0 \end{cases}$$

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:

$$p_{cr} = \frac{\rho}{2} \left(\frac{Re_{cr} \cdot \nu}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

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

$p = p_A - p_B$. 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 $5e-5 \text{ m}^2$. 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 $5e-4 \text{ m}$. 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 $1e-12$ m².

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

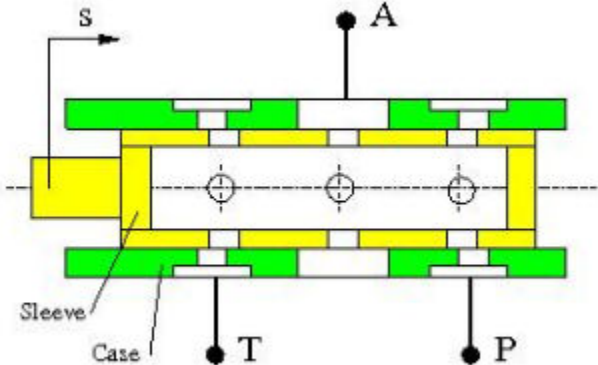


Library

Orifices

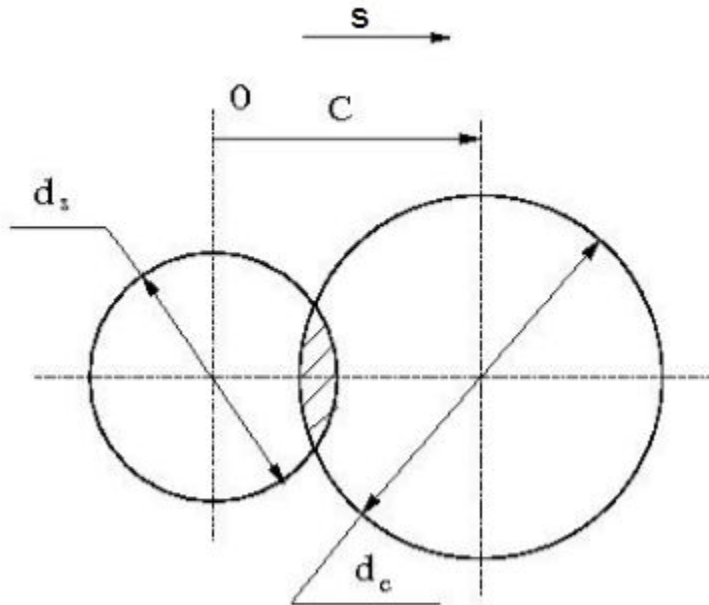
Description

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:

$$A = \begin{cases} A_{leak} & \text{for } c \geq (r + R) \\ A_{leak} + r^2 \cdot a \cos\left(\frac{c^2 + r^2 - R^2}{2cr}\right) + R^2 \cdot a \cos\left(\frac{c^2 - r^2 + R^2}{2cr}\right) & \text{for } c < (r + R) \\ -\frac{1}{2} \sqrt{(-c+r+R)(c+R-r)(c+r-R)(c+r+R)} & \end{cases}$$

$$q = z \cdot C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p}{(p^2 + p_{cr}^2)^{1/4}}$$

$$p = p_A - p_B$$

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:

$$p_{cr} = \frac{\pi}{4A} \left(\frac{Re_{cr}}{C_D} \right)^2 \cdot \rho \cdot \frac{v^2}{2}$$

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 $1e-12 \text{ m}^2$.

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 Hydraulic Machine

Variable-displacement reversible hydraulic machine with regime-dependable efficiency



Library

Pumps and Motors

Description

The Variable-Displacement Hydraulic Machine block represents a variable-displacement hydraulic machine of any type as a data-sheet-based model. The model accounts for the power flow direction and simulates the machine in both the motor and pump mode. The efficiency of the machine is variable, and you can set it in accordance with experimental data provided in the catalog or data sheet.

The machine displacement is controlled by the signal provided through the physical signal port C. The machine efficiency is simulated by implementing regime-dependable leakage and friction torque based on the experimentally established correlations between the machine efficiencies and pressure, angular velocity, and displacement.

With respect to the relationship between the control signal and the displacement, two block parameterization options are available:

- By the maximum displacement and stroke — The displacement is assumed to be linearly dependent on the control member position.
- By table-specified relationship between the control member position and the machine displacement — The displacement is determined by one-dimensional table lookup based on the control member position. You have a choice of two interpolation methods and two extrapolation methods.

The variable-displacement machine is represented with the following equations:

$$q = D \cdot \omega - k_m \cdot q_L$$

$$T = D \cdot p + k_m \cdot T_{fr}$$

$$D = \begin{cases} \frac{D_{\max}}{x_{\max}} \cdot x \\ D(x) \end{cases}$$

$$p = p_A - p_B$$

where

q	Machine flow rate
p	Pressure differential across the machine
p_A, p_B	Gauge pressures at the block terminals
D	Machine instantaneous displacement
D_{\max}	Machine maximum displacement
x	Control member displacement
x_{\max}	Control member maximum stroke
T	Torque at the machine shaft
ω	Machine shaft angular velocity
q_L	Leakage flow
T_{fr}	Friction torque
k_m	Machine type coefficient. $k_m = 1$ for the pump, $k_m = -1$ for the motor.

The key parameters that determine machine efficiency are its leakage and friction on the shaft. In the block, these parameters are specified with experimentally-based correlations similar to [1 on page 1-225]

$$q_L = D \cdot \omega \cdot k_{L1} \left(\frac{p}{p_{nom}} \right)^{k_{LP}} \left(\frac{D}{D_{\max}} \right)^{k_{LD}} \left(\frac{\omega}{\omega_{nom}} \right)^{k_{L\omega}}$$

$$T_{fr} = D \cdot p \cdot k_{F1} \left(\frac{p}{p_{nom}} \right)^{k_{FP}} \left(\frac{D}{D_{max}} \right)^{k_{FD}} \left(\frac{\omega}{\omega_{nom}} \right)^{k_{F\omega}}$$

where

p_{nom}	Nominal pressure
ω_{nom}	Nominal angular velocity
k_{L1}	Leakage proportionality coefficient
k_{F1}	Friction proportionality coefficient
$k_{LP}, k_{LD},$ $k_{L\omega}, k_{FP},$ $k_{FD}, k_{F\omega}$	Approximating coefficients

The approximating coefficients are determined from the efficiency plots, usually provided by the machine manufacturer. With the leakage known, the pump volumetric efficiency can be expressed as

$$\eta_{vp} = \frac{D \cdot \omega - q_L}{D \cdot \omega} = 1 - k_{L1} \left(\frac{p}{p_{nom}} \right)^{k_{LP}} \left(\frac{D}{D_{max}} \right)^{k_{LD}} \left(\frac{\omega}{\omega_{nom}} \right)^{k_{L\omega}}$$

For a motor, the expression looks like the following

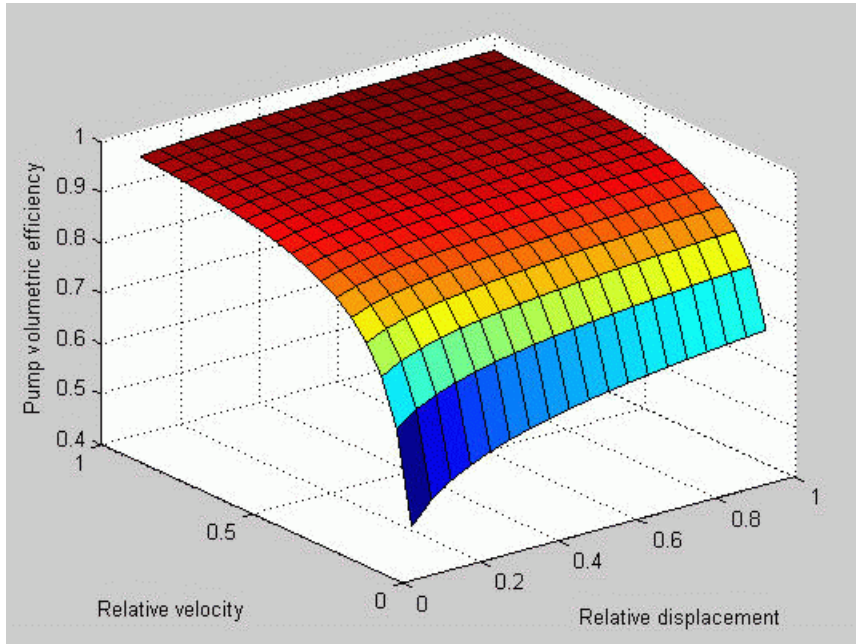
$$\eta_{vm} = \frac{D \cdot \omega}{D \cdot \omega + q_L} = \frac{1}{1 + k_{L1} \left(\frac{p}{p_{nom}} \right)^{k_{LP}} \left(\frac{D}{D_{max}} \right)^{k_{LD}} \left(\frac{\omega}{\omega_{nom}} \right)^{k_{L\omega}}}$$

The mechanical efficiency is based on the known friction torque

$$\eta_{mp} = \frac{D \cdot p}{D \cdot p + T_{fr}} = \frac{1}{1 + k_{F1} \left(\frac{p}{p_{nom}} \right)^{k_{FP}} \left(\frac{D}{D_{max}} \right)^{k_{FD}} \left(\frac{\omega}{\omega_{nom}} \right)^{k_{F\omega}}}$$

$$\eta_{mm} = \frac{D \cdot p - T_{fr}}{D \cdot p} = 1 - k_{F1} \left(\frac{p}{p_{nom}} \right)^{k_{FP}} \left(\frac{D}{D_{max}} \right)^{k_{FD}} \left(\frac{\omega}{\omega_{nom}} \right)^{k_{F\omega}}$$

The curve-fitting procedure is based on the comparison of the efficiency, determined with one of the above expressions, and the experimental data $\eta_{\text{exp}} = f(p, D, \omega)$, an example of which is shown in the following plot.



The procedure can be performed with the Optimization Toolbox™ software. For instance, the pump volumetric efficiency approximating coefficients can be found by solving the following problem:

$$\min_x F(x)$$

$$x = [k_{L1}, k_{LP}, k_{LD}, k_{L\omega}]$$

$$F(x) = \sum_i \sum_j \sum_k \left(\eta_{\text{exp}}(p_i, D_j, \omega_k) - \left(1 - k_{L1} \left(\frac{p_i}{p_{\text{nom}}} \right)^{k_{LP}} \left(\frac{D_j}{D_{\text{max}}} \right)^{k_{LD}} \left(\frac{\omega_k}{\omega_{\text{nom}}} \right)^{k_{L\omega}} \right) \right)^2$$

where

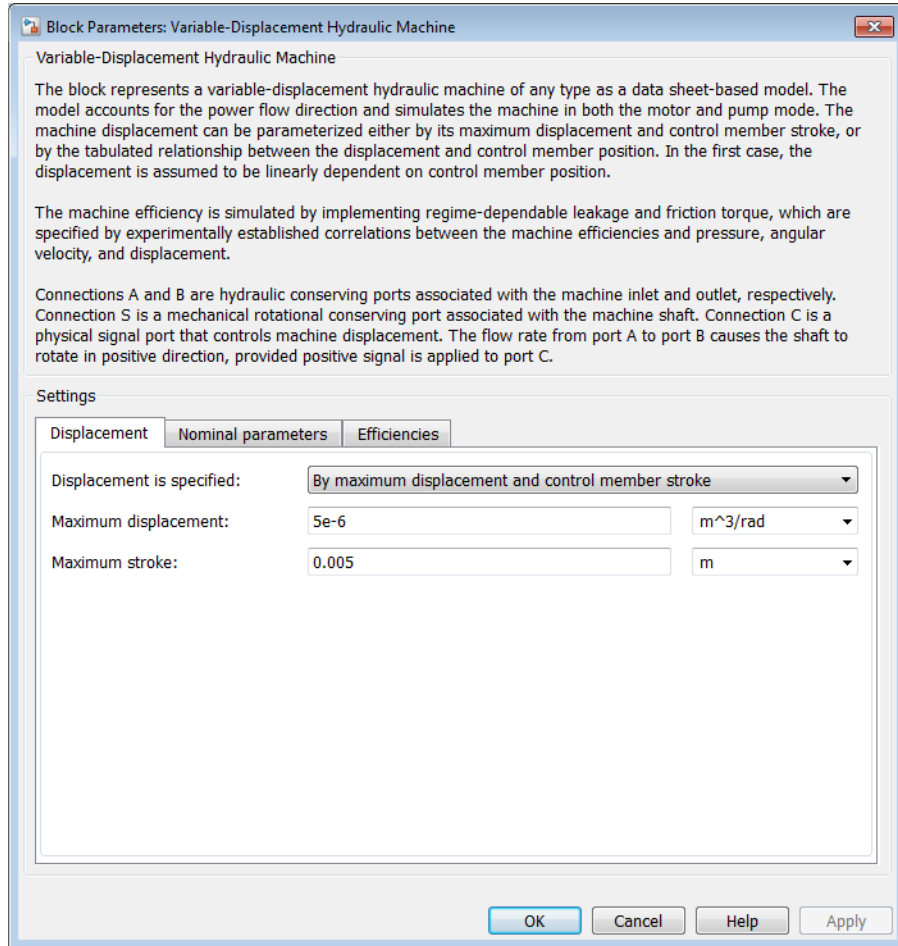
i	Number of experimental pressure points, from 1 to n
j	Number of experimental displacement points, from 1 to m
k	Number of experimental angular velocity points, from 1 to l

Connections A and B are hydraulic conserving ports associated with the machine inlet and outlet, respectively. Connection S is a mechanical rotational conserving port associated with the machine shaft. Connection C is a physical signal port that controls machine displacement. The flow rate from port A to port B causes the shaft to rotate in positive direction, provided positive signal is applied to port C.

Basic Assumptions and Limitations

- Fluid compressibility is neglected.
- No inertia on the machine shaft is considered.
- The model is applicable only for fluid and fluid temperature at which the approximating coefficients have been determined.
- Exercise extreme caution to not exceed the limits within which the approximating coefficients have been determined. The extrapolation could result in large errors.

Dialog Box and Parameters



- “Displacement Tab” on page 1-943
- “Nominal Parameters Tab” on page 1-944
- “Efficiencies Tab” on page 1-945

Displacement Tab

Displacement is specified

Select one of the following block parameterization options:

- By maximum displacement and control member stroke — Provide values for maximum machine displacement and maximum stroke. The displacement is assumed to be linearly dependent on the control member position. This is the default method.
- By displacement vs. control member position table — Provide tabulated data of machine displacements and control member positions. The displacement is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

Maximum displacement

Machine maximum displacement. The default value is $5e-6 \text{ m}^3/\text{rad}$.

Maximum stroke

Maximum control member stroke. The default value is 0.005 m. This parameter is used if displacement is specified as By maximum displacement and control member stroke.

Pump displacements table

Specify the vector of machine displacements as a one-dimensional array. The machine displacements vector must be of the same size as the control member positions vector. The default values, in m^3/rad , are $[-5e-06 \ -3e-06 \ 0 \ 3e-06 \ 5e-06]$. This parameter is used if displacement is specified as By displacement vs. control member position table.

Control member positions table

Specify the vector of input values for control member position 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 $[-0.0075 \ -0.0025 \ 0 \ 0.0025 \ 0.0075]$. This parameter is used if displacement is specified as By displacement vs. control member position 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 displacement is specified as `By displacement vs. control member position 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 displacement is specified as `By displacement vs. control member position table`.

Nominal Parameters Tab

Nominal pressure

Nominal pressure differential across the machine. The default value is $1e7$ Pa.

Nominal angular velocity

Nominal angular velocity of the output shaft. The default value is 188 rad/s.

Shaft velocity at peak friction

The friction torque on the machine shaft ideally should be introduced as $T_f \text{sign}(\omega)$. To avoid discontinuity at $\omega \rightarrow 0$, the friction is defined as $T_f \tanh(4\omega / \omega_{\max})$, where ω_{\max} is a small velocity, representing the shaft velocity at peak friction, at which $\tanh(4\omega / \omega_{\max})$ is equal to 0.999. The default value of ω_{\max} is 0.01 rad/s.

Efficiencies Tab

Volumetric efficiency proportionality coefficient

Approximating coefficient k_{LI} in the block description preceding. The default value is 0.05.

Volumetric efficiency pressure coefficient

Approximating coefficient k_{LP} in the block description preceding. The default value is 0.65.

Volumetric efficiency angular velocity coefficient

Approximating coefficient $k_{L\omega}$ in the block description preceding. The default value is -0.2.

Volumetric efficiency displacement coefficient

Approximating coefficient k_{LD} in the block description preceding. The default value is -0.8.

Mechanical efficiency proportionality coefficient

Approximating coefficient k_{FI} in the block description preceding. The default value is 0.06.

Mechanical efficiency pressure coefficient

Approximating coefficient k_{FP} in the block description preceding. The default value is -0.65.

Mechanical efficiency angular velocity coefficient

Approximating coefficient $k_{F\omega}$ in the block description preceding. The default value is 0.2.

Mechanical efficiency displacement coefficient

Approximating coefficient k_{FD} in the block description preceding. The default value is -0.75.

Restricted Parameters

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

- **Displacement is specified**
- **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 **Displacement is specified** parameter at the time the model entered Restricted mode.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the machine inlet.

B

Hydraulic conserving port associated with the machine outlet.

C

Physical signal port that controls machine displacement.

S

Mechanical rotational conserving port associated with the machine shaft.

References

[1] C.R. Cornell, *Dynamic Simulation of a Hydrostatically Propelled Vehicle*, SAE paper 811253, 1981, p. 22

See Also

Variable-Displacement Motor | Variable-Displacement Pump

Introduced in R2008b

Variable-Displacement Hydraulic Machine (External Efficiencies)

Variable-displacement reversible hydraulic machine with externally specified volumetric and mechanical efficiencies



Library

Pumps and Motors

Description

The Variable-Displacement Hydraulic Machine (External Efficiencies) block represents a reversible, variable positive-displacement hydraulic machine with volumetric and mechanical efficiencies provided externally to the block through physical signal ports EV and EM, respectively. The block can work as a pump or a motor, depending on the direction of mechanical and hydraulic power flow.

The machine displacement is controlled externally through a physical signal port, C.

The model offers two options to specify the relationship between the control member position and the machine displacement:

- By the maximum displacement and stroke — The displacement is assumed to be linearly dependent on the control member position.
- By table-specified relationship between the control member position and the machine displacement — The displacement is determined by one-dimensional table lookup based on the control member position. You have a choice of three interpolation methods and two extrapolation methods.

The machine switches between pump and motor modes at the moment the mechanical power changes sign:

- If $P_{\text{mech}} > 0$ — Pump mode.
- If $P_{\text{mech}} \leq 0$ — Motor mode.

To ensure continuity at the instance when the machine switches its operational mode, the block implements a transition region around this point, where the transition from one regime to another is smoothed by a simple third-order polynomial transition function. The transition region is defined by the power threshold value:

- If $P_{\text{mech}} \geq P_{\text{threshold}}$ — Pump mode.
- If $P_{\text{mech}} \leq -P_{\text{threshold}}$ — Motor mode.
- If $-P_{\text{threshold}} < P_{\text{mech}} < P_{\text{threshold}}$ — Transition region.

The variable-displacement machine is represented with the following equations:

$$q = D \cdot \omega \cdot K_v$$

$$\tau = -D \cdot p \cdot K_m$$

$$D = \begin{cases} \frac{D_{\text{max}}}{x_{\text{max}}} \cdot x \\ D(x) \end{cases}$$

$$K_v = \begin{cases} \eta_v & \text{pump mode} \\ \frac{1}{\eta_v} & \text{motor mode} \\ \frac{1}{\eta_v} + (3P_{\text{norm}}^2 - 2P_{\text{norm}}^3) \cdot \left(\eta_v - \frac{1}{\eta_v} \right) & \text{transition region} \end{cases}$$

$$K_m = \begin{cases} \frac{1}{\eta_m} & \text{pump mode} \\ \eta_m & \text{motor mode} \\ \eta_m + (3P_{\text{norm}}^2 - 2P_{\text{norm}}^3) \cdot \left(\frac{1}{\eta_m} - \eta_m \right) & \text{transition region} \end{cases}$$

$$P_{mech} = \tau \cdot \omega$$

$$P_{hyd} = p \cdot q$$

$$p = p_A - p_B$$

$$P_{norm} = \frac{P_{mech} - (-P_{threshold})}{P_{threshold} - (-P_{threshold})}$$

where

q	Machine flow rate
p	Pressure differential across the machine
p_A, p_B	Gauge pressures at the block terminals
ω	Machine shaft angular velocity
D	Machine instantaneous displacement
D_{max}	Machine maximum displacement
x	Control member displacement
x_{max}	Control member maximum stroke
τ	Machine shaft torque
K_v	Machine flow rate proportionality coefficient
K_m	Machine torque proportionality coefficient
η_v	Volumetric efficiency
η_m	Mechanical efficiency
P_{mech}	Mechanical power
P_{hyd}	Hydraulic power
$P_{threshold}$	Power threshold
P_{norm}	Normalized power

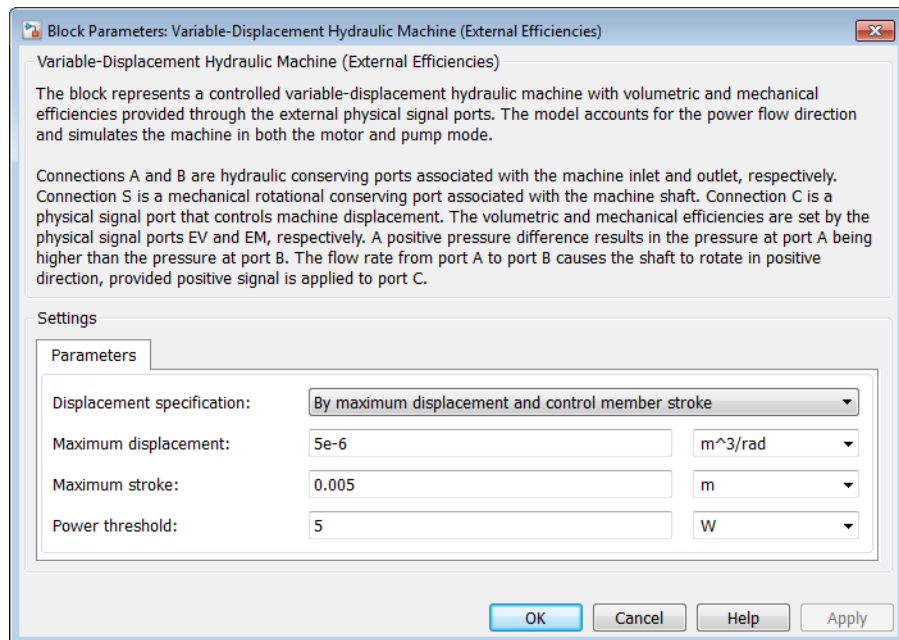
Connections A and B are hydraulic conserving ports associated with the machine inlet and outlet, respectively. Connection S is a mechanical rotational conserving port associated with the machine shaft. Connection C is a physical signal port that controls machine displacement. Connections EV and EM are physical signal ports for specifying volumetric and mechanical efficiencies, respectively. The flow rate from port A to port B

causes the shaft to rotate in the positive direction, provided a positive signal is applied to port C.

Basic Assumptions and Limitations

- Fluid compressibility is neglected.
- Inertia on the machine shaft is not considered.
- To ensure the continuity of the flow rate and torque at the instant the machine changes its mode between pump and motor, a small region (power threshold) is defined, which helps a smooth transition between the two modes.

Dialog Box and Parameters



Displacement specification

Select one of the following block parameterization options:

- By maximum displacement and control member stroke — Provide values for maximum machine displacement and maximum stroke. The displacement is assumed to be linearly dependent on the control member position. This is the default.
- By displacement vs. control member position table — Provide tabulated data of machine displacements and control member positions. The displacement is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.

Maximum displacement

Machine maximum displacement. The default value is $5e-6 \text{ m}^3/\text{rad}$.

Maximum stroke

Maximum control member stroke. The default value is 0.005 m . This parameter is used if displacement is specified as By maximum displacement and control member stroke.

Machine displacements table

Specify the vector of machine displacements as a one-dimensional array. The machine displacements vector must be of the same size as the control member positions vector. The default values, in m^3/rad , are $[-5e-06 \ -3e-06 \ 0 \ 3e-06 \ 5e-06]$. This parameter is used if displacement is specified as By displacement vs. control member position table.

Control member positions table

Specify the vector of input values for control member position 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 cubic or spline interpolation. The default values, in meters, are $[-0.0075 \ -0.0025 \ 0 \ 0.0025 \ 0.0075]$. This parameter is used if displacement is specified as By displacement vs. control member position 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 — Uses a linear interpolation function.
- Cubic — Uses the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP).

- `Spline` — Uses the cubic spline interpolation algorithm.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if displacement is specified as `By displacement vs. control member position 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:

- `From last 2 points` — Extrapolates using the linear method (regardless of the interpolation method specified), based on the last two output values at the appropriate end of the range. That is, the block uses the first and second specified output values if the input value is below the specified range, and the two last specified output values if the input value is above the specified range.
- `From last point` — Uses the last specified output value at the appropriate end of the range. That is, the block uses the last specified output value for all input values greater than the last specified input argument, and the first specified output value for all input values less than the first specified input argument.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if displacement is specified as `By displacement vs. control member position table`.

Power threshold

The power threshold is introduced to avoid discontinuity at the instance the machine switches between pump and motor modes of operation. In the absence of this transition region, the flow and torque change their values instantly, which may result in convergence problems due to discontinuity. A small region in the vicinity of zero power allows avoiding discontinuity and improving convergence. The default value is 5 W.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the machine inlet.

B

Hydraulic conserving port associated with the machine outlet.

C

Physical signal port that controls machine displacement.

EV

Physical signal port associated with the volumetric efficiency input.

EM

Physical signal port associated with the mechanical efficiency input.

S

Mechanical rotational conserving port associated with the machine shaft.

See Also

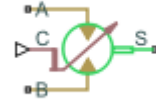
Variable-Displacement Hydraulic Machine | Variable-Displacement Motor | Variable-Displacement Pump

Introduced in R2015a

Variable-Displacement Motor

Variable-displacement bidirectional hydraulic motor

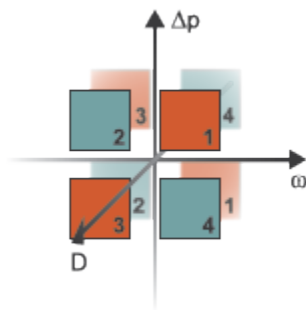
Library: Hydraulics (Isothermal) / Pumps & Motors



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 1-963.

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

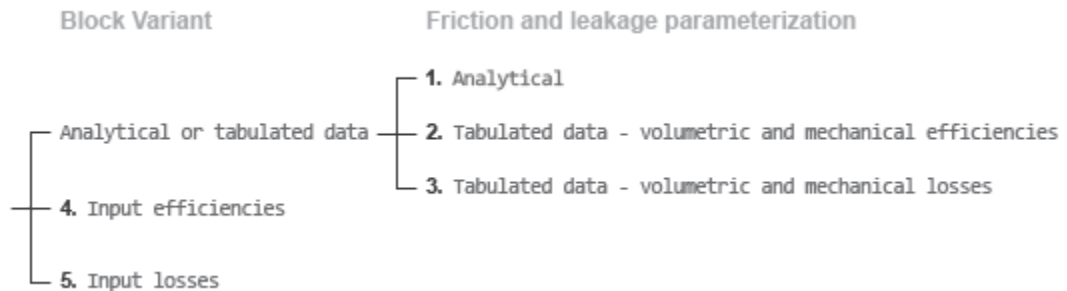
instantaneous volumetric displacement of the component (D). The “Operation Modes” on page 1-954 figure maps the modes to the octants of a Δp - ω - D chart. The modes are labeled 1–4:

- 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.

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:

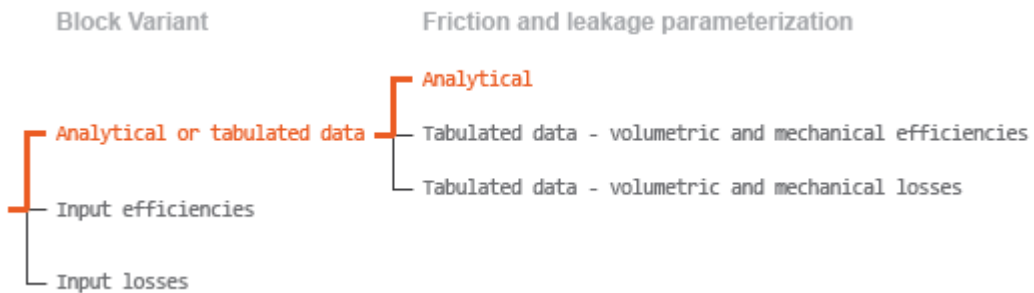
- D is the instantaneous fluid displacement determined from the physical signal input specified at port **C** or port **D**.
- D_{Max} is the specified value of the **Maximum displacement** block parameter.

- $D_{\text{Threshold}}$ 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.

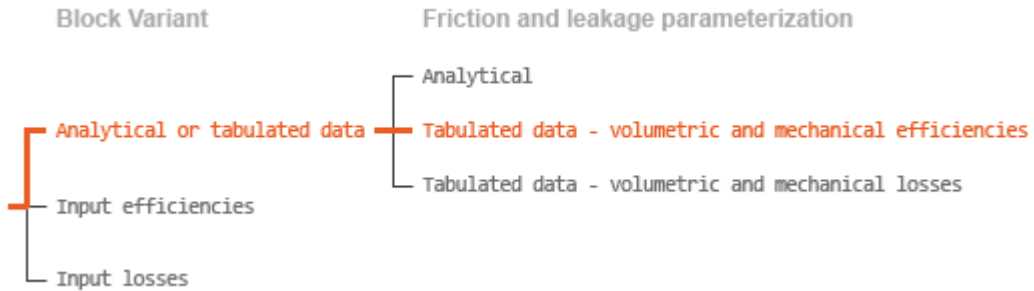
- K_{TP} is the specified value of the **Friction torque vs pressure drop coefficient** block parameter.
- τ_0 is the specified value of the **No-load torque** block parameter.
- $\omega_{Threshold}$ is the threshold angular velocity for the motor-pump transition. The threshold angular velocity is an 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

$$K_{HP} = \frac{v_{Nom}}{\rho v} \frac{\rho_{Nom} \omega_{Nom} D}{\Delta p_{Nom}} \left(\frac{1}{\eta_{v,Nom}} - 1 \right),$$

where:

- v_{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.
- v 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.
- $\eta_{v,Nom}$ 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

$$\alpha = \tanh\left(\frac{4\Delta p}{\Delta p_{\text{Threshold}}}\right) \tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right) \tanh\left(\frac{4D}{D_{\text{Threshold}}}\right),$$

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

$$q_{\text{Leak, Motor}} = (1 - \eta_v) q,$$

in motor mode and through the equation

$$q_{\text{Leak, Pump}} = -(1 - \eta_v) q_{\text{Ideal}}.$$

in pump mode, where:

- η_v is the volumetric efficiency obtained through interpolation or extrapolation of the **Volumetric efficiency table, $e_v(\text{dp}, \mathbf{w}, \mathbf{D})$** parameter data.

Similarly, the friction torque is computed from efficiency tabulated data through the equation

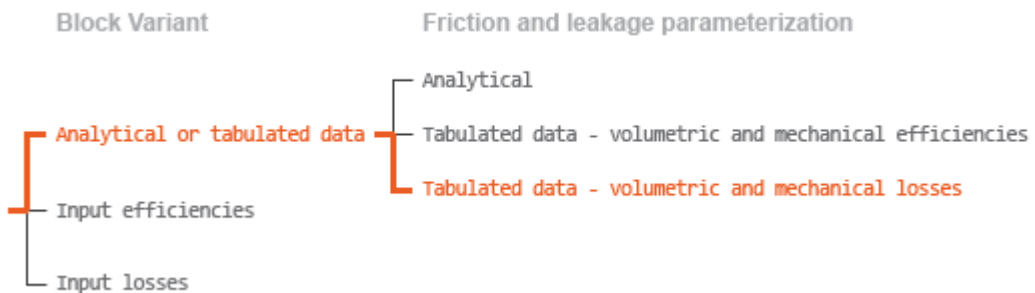
$$\tau_{\text{Friction, Motor}} = (1 - \eta_m) \tau_{\text{Ideal}},$$

in motor mode and through the equation

$$\tau_{\text{Friction, Pump}} = -(1 - \eta_m) \tau.$$

in pump mode, where:

- η_m is the mechanical efficiency obtained through interpolation or extrapolation of the **Mechanical efficiency table, $e_m(\text{dp}, \mathbf{w}, \mathbf{D})$** parameter data.



Case 3: Loss Tabulated Data

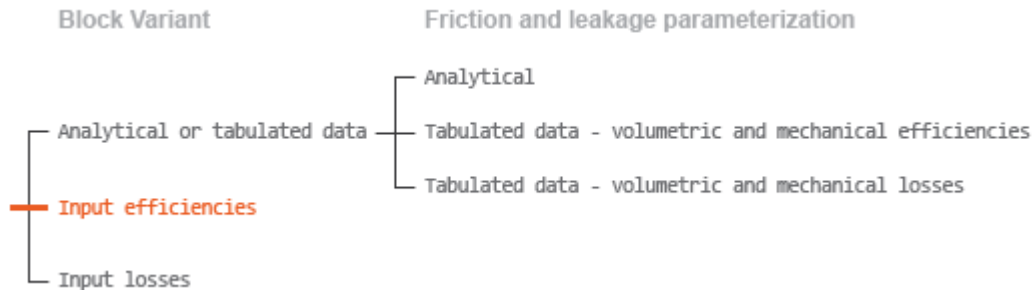
If the active block variant is Analytical or tabulated data and the **Leakage and friction parameterization** parameter is set to Tabulated data – volumetric and mechanical losses, the leakage flow rate equation is

$$q_{\text{Leak}} = q_{\text{Leak}}(\Delta p, \omega, D_{\text{Sat}}).$$

and the friction torque equation is

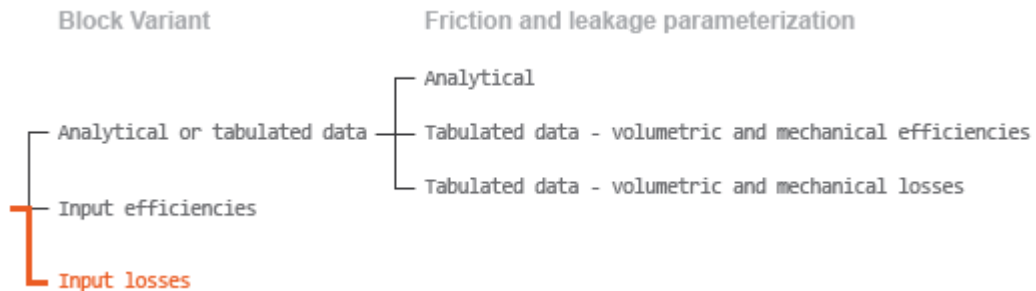
$$\tau_{\text{Friction}} = \tau_{\text{Friction}}(\Delta p, \omega, D_{\text{Sat}}),$$

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, cm³/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, $N\cdot m$

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 1-955.

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 cm³/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] cm³/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 data 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 data 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/m³ (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 cm³/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 cm³/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 cm³/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`.

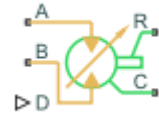
See Also

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

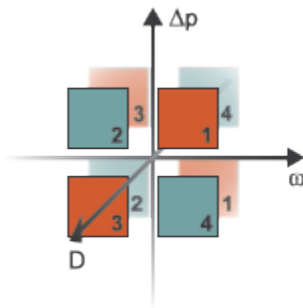
Library: Thermal Liquid / Pumps & Motors



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

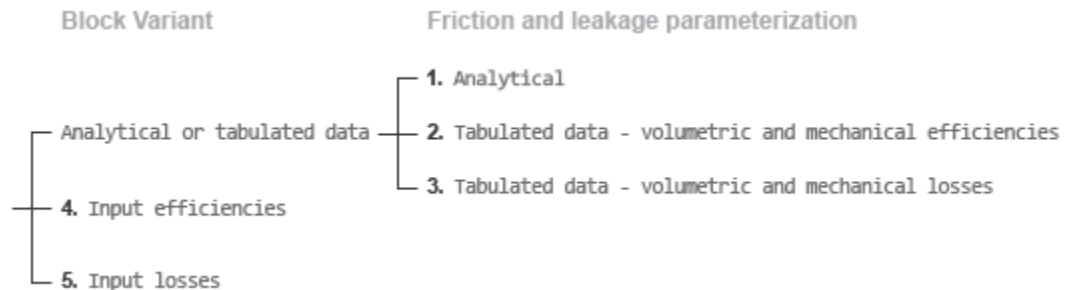
page 1-981 figure maps the modes to the octants of a Δp - ω - D chart. The modes are labeled 1–4:

- 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

$$\dot{m} = \dot{m}_{\text{Ideal}} + \dot{m}_{\text{Leak}},$$

where:

- \dot{m} is the actual mass flow rate.
- \dot{m}_{Ideal} is the ideal mass flow rate.
- \dot{m}_{Leak} is the internal leakage mass flow rate.

The torque generated at the motor is

$$\tau = \tau_{\text{Ideal}} - \tau_{\text{Friction}},$$

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.

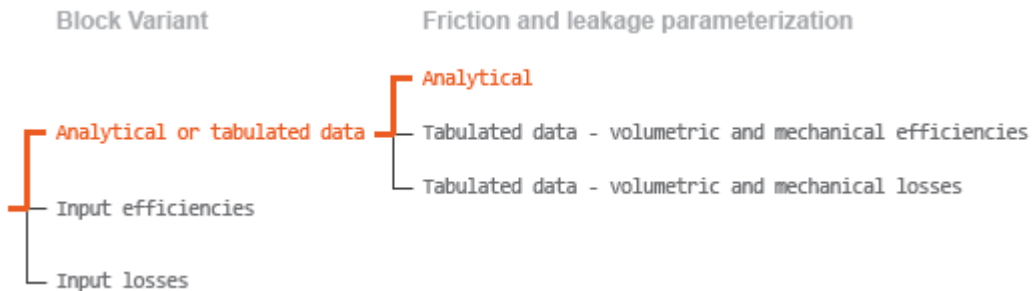
The saturation displacement is defined as:

$$D_{\text{Sat}} = \begin{cases} \sqrt{D^2 + D_{\text{Threshold}}^2}, & D \geq 0 \\ -\sqrt{D^2 + D_{\text{Threshold}}^2}, & D < 0 \end{cases}$$

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

$$\dot{m}_{\text{Leak}} = K_{\text{HP}} \Delta p,$$

and the friction torque is

$$\tau_{\text{Friction}} = \left(\tau_0 + K_{\text{TP}} |\Delta p| \frac{|D_{\text{Sat}}|}{D_{\text{Nom}}} \tanh \frac{4\omega}{(5e-5)\omega_{\text{Nom}}} \right),$$

where:

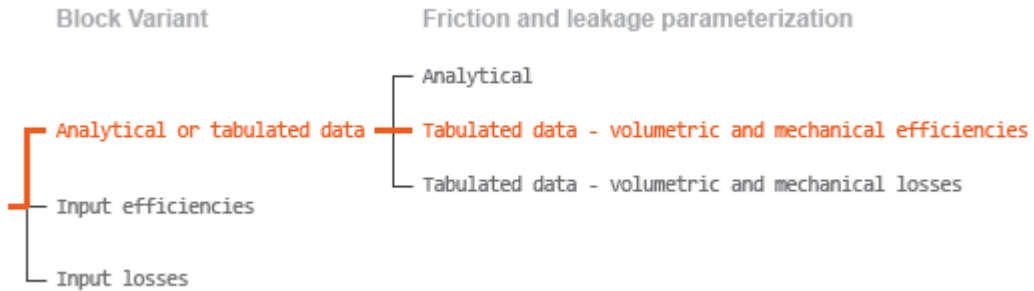
- K_{HP} is the Hagen-Poiseuille coefficient for laminar pipe flows. This coefficient is computed from the specified nominal parameters.
- K_{TP} is the specified value of the **Friction torque vs pressure drop coefficient** block parameter.
- D_{Nom} is the specified value of the **Nominal Displacement** 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

$$K_{\text{HP}} = \frac{D_{\text{Sat}} \omega_{\text{Nom}} \mu_{\text{Nom}} \left(\frac{1}{\eta_{\text{v,Nom}}} - 1 \right)}{\Delta p_{\text{Nom}}},$$

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

$$\alpha = \tanh\left(\frac{4\Delta p}{\Delta p_{\text{Threshold}}}\right) \tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right) \tanh\left(\frac{4D}{D_{\text{Threshold}}}\right),$$

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 figure), the leakage flow rate is:

$$\dot{m}_{\text{Leak, Motor}} = (1 - \eta_v) \dot{m},$$

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:

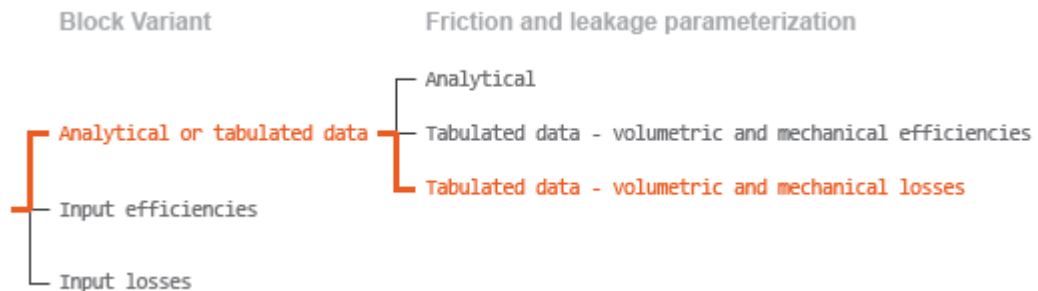
$$\dot{m}_{\text{Leak, Pump}} = -(1 - \eta_v) \dot{m}_{\text{Ideal}}.$$

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):

$$\tau_{\text{Friction, Motor}} = (1 - \eta_m) \tau_{\text{Ideal}},$$

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):

$$\tau_{\text{Friction, Pump}} = -(1 - \eta_m) \tau.$$



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:

$$q_{\text{Leak}} = q_{\text{Leak}}(\Delta p, \omega, D_{\text{Sat}}).$$

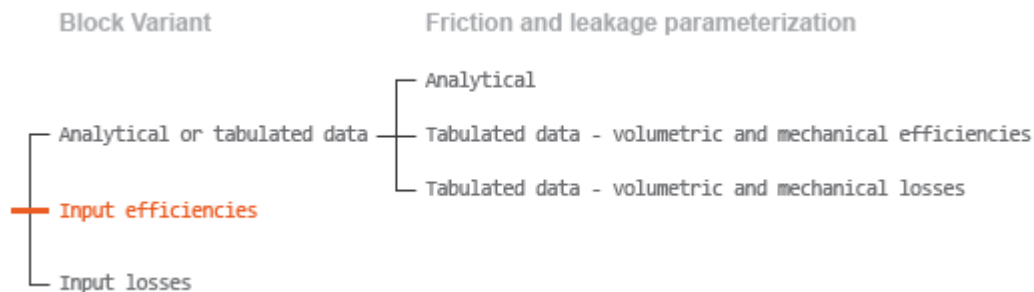
The mass flow rate due to leakage is calculated from the volumetric flow rate:

$$\dot{m}_{\text{Leak}} = \rho q_{\text{Leak}}.$$

The friction torque is similarly specified in tabulated form:

$$\tau_{\text{Friction}} = \tau_{\text{Friction}}(\Delta p, \omega, D_{\text{Sat}}),$$

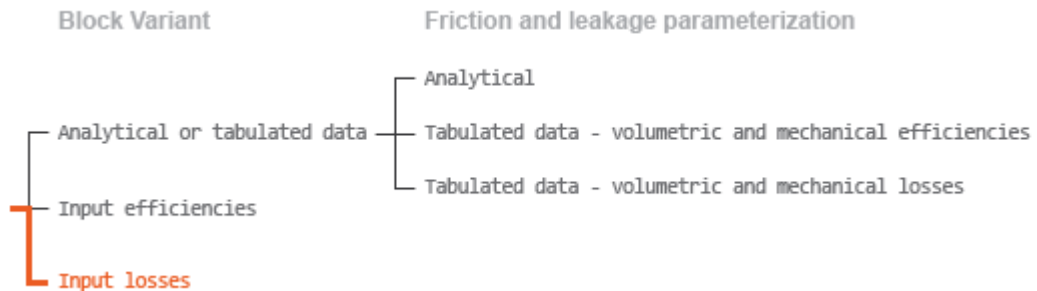
where $q_{\text{Leak}}(\Delta p, \omega)$ and $\tau_{\text{Friction}}(\Delta p, \omega)$ are the volumetric and mechanical losses, obtained through interpolation or extrapolation of the tabulated data specified via the **Volumetric loss table** and **Mechanical loss table** block parameters.



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, $N\cdot m$

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 1-955.

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 cm³/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 cm³/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 m² (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 $(\omega, \Delta p, D)$ 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 $(\omega, \Delta p, D)$ 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, $e_v(dp, \omega, D)$ — 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 $(\omega, \Delta p, D)$ 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 $(\omega, \Delta p, D)$ 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 cm³/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 $(\omega, \Delta p, D)$ 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 $(\omega, \Delta p, D)$ 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 $(\omega, \Delta p, D)$ 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

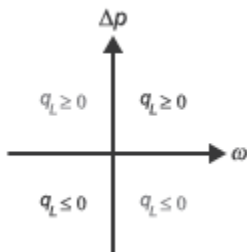
M-by-*N*-by-*L* matrix with units of volume/time

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 $(\omega, \Delta p, D)$ 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.

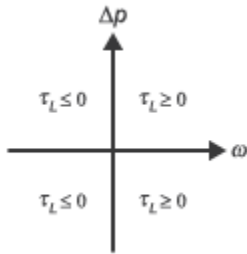
Mechanical loss table, torque_loss(dp,w,D) — Friction torques at the specified pressure drops, angular velocities, and displacements

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 $(\omega, \Delta p, D)$ 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 cm³/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 m² (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 cm³/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 m² (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



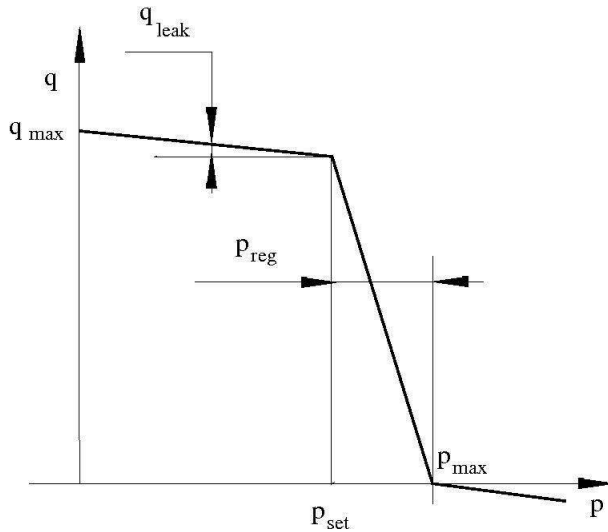
Library

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 preset 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:

$$q = D \cdot \omega - k_{leak} \cdot p$$

$$T = D \cdot p / \eta_{mech}$$

$$D = \begin{cases} D_{\max} & \text{for } p \leq p_{set} \\ D_{\max} - K(p - p_{set}) & \text{for } p_{set} < p < p_{\max} \\ 0 & \text{for } p \geq p_{\max} \end{cases}$$

$$p_{\max} = p_{set} + p_{reg}$$

$$K = D_{\max} / (p_{\max} - p_{set})$$

$$k_{leak} = \frac{k_{HP}}{v \cdot \rho}$$

$$k_{HP} = \frac{D_{\max} \cdot \omega_{nom} (1 - \eta_V) \cdot v_{nom} \cdot \rho_{nom}}{p_{nom}}$$

$$p = p_P - p_T$$

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, $\mu = \nu\rho$

The leakage flow at $p = p_{nom}$ and $\nu = \nu_{nom}$ can be determined from the catalog data

$$q_{leak} = D_{max} \cdot \omega_{nom} (1 - \eta_V)$$

which provides the formula to determine the Hagen-Poiseuille coefficient

$$k_{HP} = \frac{D_{max} \cdot \omega_{nom} (1 - \eta_V) \cdot \nu_{nom} \cdot \rho_{nom}}{p_{nom}}$$

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

Block Parameters: Variable-Displacement Pressure-Compensated Pump

Variable-Displacement Pressure-Compensated Pump

This block represents a variable-displacement, pressure-compensated hydraulic pump of any type as a data sheet-based model. The key parameters required to parameterize the block are the pump maximum displacement, volumetric and total efficiencies, regulation range, nominal pressure, and angular velocity.

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 delivers flow to port P as its driving shaft rotates in the globally assigned positive direction.

Settings

Parameters

Maximum displacement:	5e-6	m ³ /rad
Setting pressure:	100e5	Pa
Pressure regulation range:	6e5	Pa
Volumetric efficiency:	0.85	
Total efficiency:	0.75	
Nominal pressure:	100e5	Pa
Nominal angular velocity:	188	rad/s
Nominal kinematic viscosity:	18	cSt
Nominal fluid density:	950	kg/m ³

OK Cancel Help Apply

Maximum displacement

Pump displacement. The default value is $5e-6$ m³/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

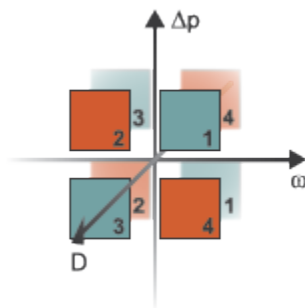
Library: Hydraulics (Isothermal) / Pumps & Motors



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 1-1022.

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

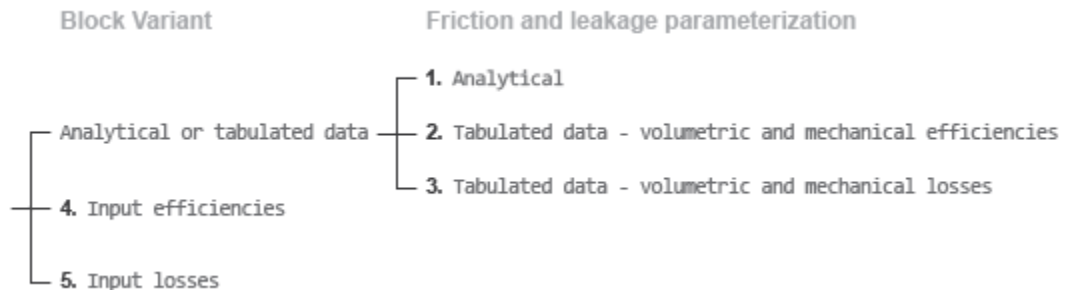
instantaneous displacement of the component (D). The “Operation Modes” on page 1-1013 figure maps the modes to the octants of a Δp - ω - D chart. The modes are labeled 1–4:

- 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`,

$$D_{\text{Sat}} = \begin{cases} \text{sign}(D) \bullet D_{\text{Max}}, & |D| \geq D_{\text{Max}} \\ \sqrt{D^2 + D_{\text{Threshold}}^2}, & D \geq 0 \\ -\sqrt{D^2 + D_{\text{Threshold}}^2}, & D < 0 \end{cases},$$

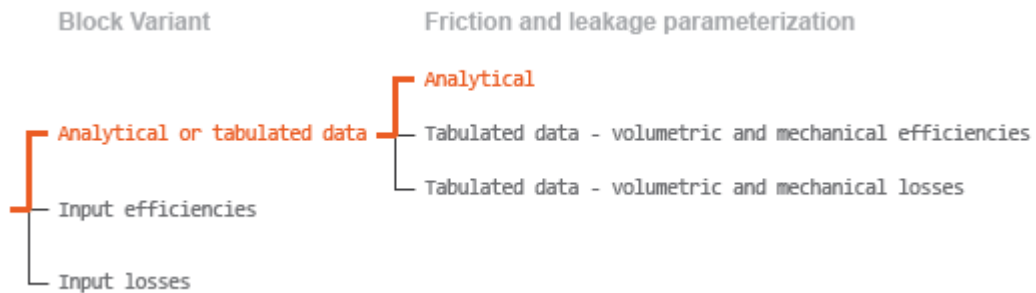
where:

- 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

$$q_{Leak} = K_{HP}\Delta p,$$

and the friction torque is

$$\tau_{Friction} = \left(\tau_0 + K_{TP} \left| \frac{D_{Sat}}{D_{Max}} \right| |\Delta p| \right) \tanh \left(\frac{4\omega}{\omega_{Thresh}} \right),$$

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.

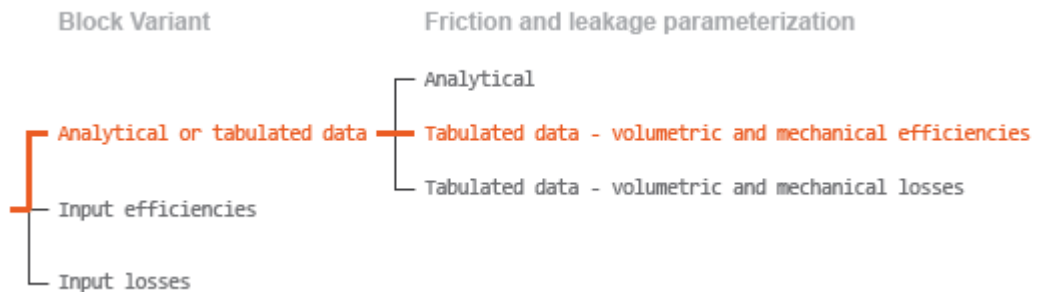
- $\omega_{\text{Threshold}}$ is the threshold angular velocity for the motor-pump transition. The threshold angular velocity is an internally set fraction of the specified value of the **Nominal shaft angular velocity** block parameter.

The Hagen-Poiseuille coefficient is determined from nominal fluid and component parameters through the equation

$$K_{\text{HP}} = \frac{v_{\text{Nom}}}{\rho v} \frac{\rho_{\text{Nom}} \omega_{\text{Nom}} D}{\Delta p_{\text{Nom}}} (1 - \eta_{v,\text{Nom}}),$$

where:

- v_{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.
- v 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.
- $\eta_{v,\text{Nom}}$ 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

$$\alpha = \tanh\left(\frac{4\Delta p}{\Delta p_{\text{Threshold}}}\right) \tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right) \tanh\left(\frac{4D}{D_{\text{Threshold}}}\right),$$

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

$$q_{\text{Leak,Pump}} = (1 - \eta_v) q_{\text{Ideal}},$$

in pump mode and through the equation

$$q_{\text{Leak,Motor}} = -(1 - \eta_v) q,$$

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

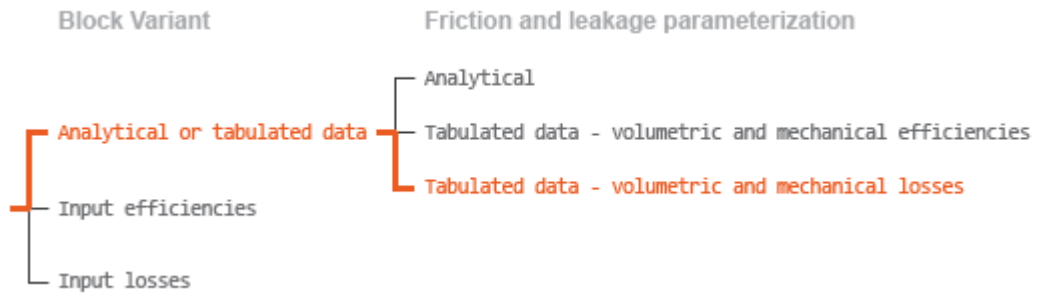
$$\tau_{\text{Friction,Pump}} = (1 - \eta_m) \tau,$$

in pump mode and through the equation

$$\tau_{\text{Friction,Motor}} = -(1 - \eta_m) \tau_{\text{Ideal}},$$

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

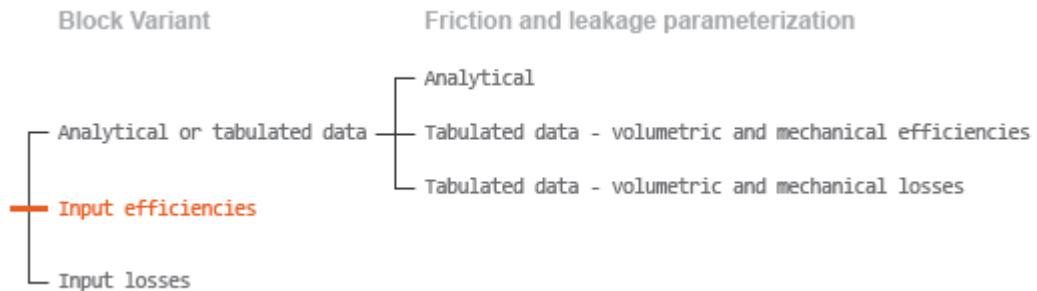
If the active block variant is **Analytical or tabulated data** and the **Leakage and friction parameterization** parameter is set to **Tabulated data – volumetric and mechanical losses**, the leakage flow rate equation is

$$q_{\text{Leak}} = q_{\text{Leak}}(\Delta p, \omega, D_{\text{Sat}}).$$

and the friction torque equation is

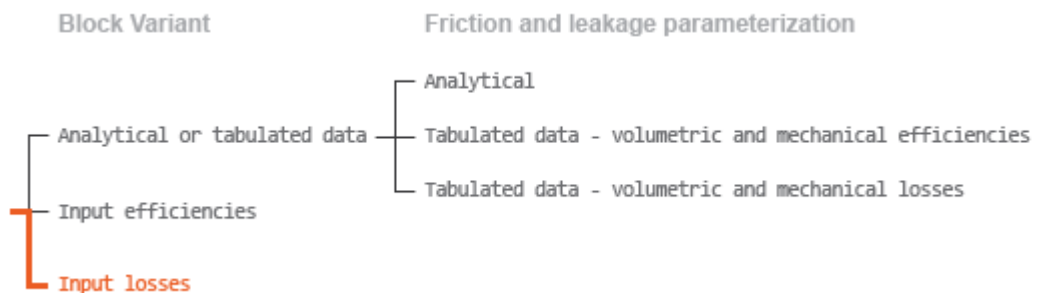
$$\tau_{\text{Friction}} = \tau_{\text{Friction}}(\Delta p, \omega, D_{\text{Sat}}),$$

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, cm³/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, $N\cdot m$

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 **τ — 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 1-1014.

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 cm³/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] cm³/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 data 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 data 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/m3` (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 cm³/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 cm³/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 cm³/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`.

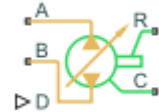
See Also

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

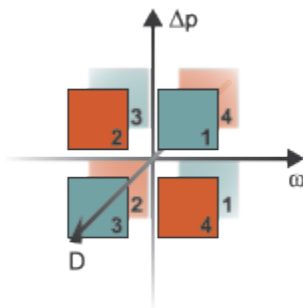
Library: Thermal Liquid / Pumps & Motors



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

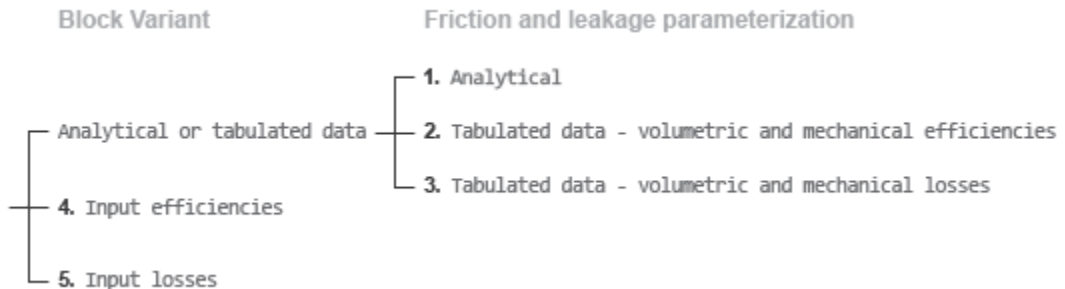
the instantaneous displacement input at port D (D). The “Operation Modes” on page 1-1013 figure maps the modes to the octants of a Δp - ω - D chart. The modes are labeled 1–4:

- 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

$$\dot{m} = \dot{m}_{\text{Ideal}} - \dot{m}_{\text{Leak}},$$

where:

- \dot{m} is the actual mass flow rate.
- \dot{m}_{Ideal} is the ideal mass flow rate.
- \dot{m}_{Leak} is the internal leakage mass flow rate.

The driving torque required to power the pump is

$$\tau = \tau_{\text{Ideal}} + \tau_{\text{Friction}},$$

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

$$\dot{m}_{\text{Ideal}} = \rho D_{\text{Sat}} \omega,$$

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.

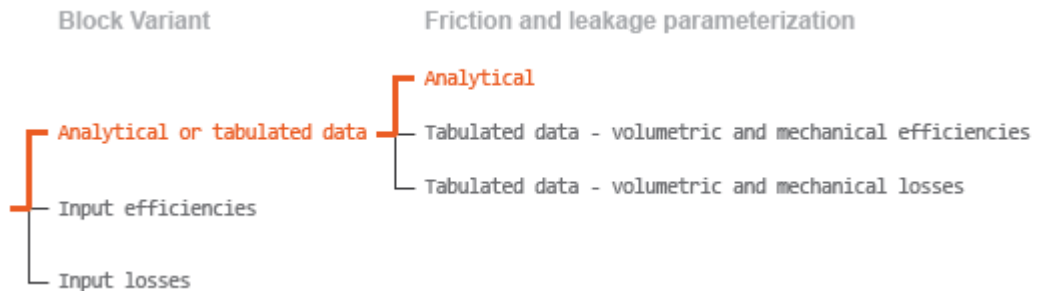
The saturation displacement is defined as:

$$D_{\text{Sat}} = \begin{cases} \sqrt{D^2 + D_{\text{Threshold}}^2}, & D \geq 0 \\ -\sqrt{D^2 + D_{\text{Threshold}}^2}, & D < 0 \end{cases}$$

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

$$\dot{m}_{\text{Leak}} = K_{\text{HP}} \Delta p,$$

and the friction torque is

$$\tau_{\text{Friction}} = \left(\tau_0 + K_{\text{TP}} |\Delta p| \frac{|D_{\text{Sat}}|}{D_{\text{Nom}}} \tanh \frac{4\omega}{(5e - 5)\omega_{\text{Nom}}} \right),$$

where:

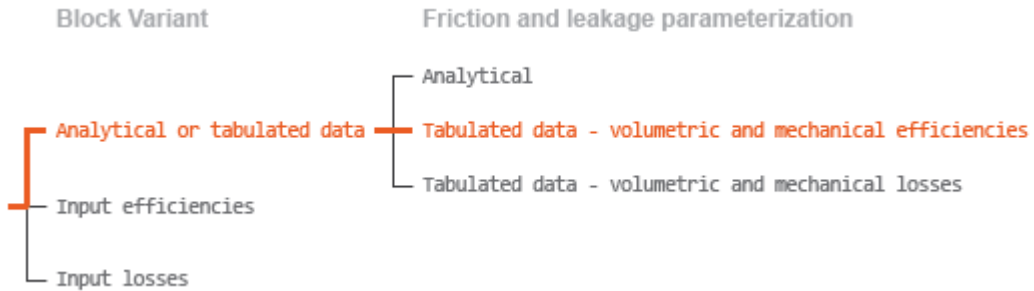
- K_{HP} is the Hagen-Poiseuille coefficient for laminar pipe flows. This coefficient is computed from the specified nominal parameters.
- K_{TP} is the specified value of the **Friction torque vs pressure drop coefficient** block parameter.
- D_{Nom} is the specified value of the **Nominal Displacement** 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

$$K_{\text{HP}} = \frac{D_{\text{Nom}} \omega_{\text{Nom}} \mu_{\text{Nom}} (1 - \eta_{\text{v,Nom}})}{\Delta p_{\text{Nom}}},$$

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

$$\alpha = \tanh\left(\frac{4\Delta p}{\Delta p_{\text{Threshold}}}\right) \tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right) \tanh\left(\frac{4D}{D_{\text{Threshold}}}\right),$$

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 figure), the leakage flow rate is:

$$\dot{m}_{\text{Leak,Pump}} = (1 - \eta_v) \dot{m}_{\text{Ideal}},$$

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:

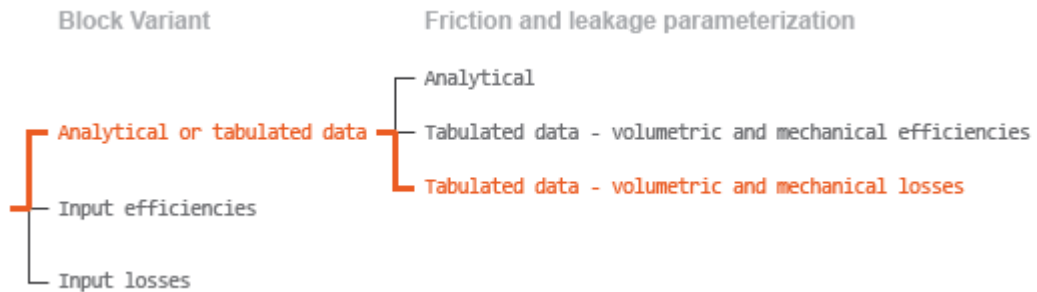
$$\dot{m}_{\text{Leak,Motor}} = -(1 - \eta_v) \dot{m}.$$

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):

$$\tau_{\text{Friction,Pump}} = (1 - \eta_m) \tau,$$

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):

$$\tau_{\text{Friction,Motor}} = -(1 - \eta_m) \tau_{\text{Ideal}}.$$



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:

$$q_{\text{Leak}} = q_{\text{Leak}}(\Delta p, \omega, D_{\text{Sat}}).$$

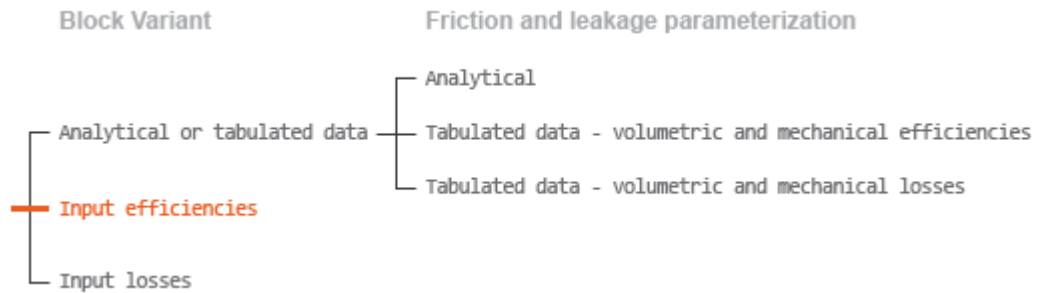
The mass flow rate due to leakage is calculated from the volumetric flow rate:

$$\dot{m}_{\text{Leak}} = \rho q_{\text{Leak}}.$$

The friction torque is similarly specified in tabulated form:

$$\tau_{\text{Friction}} = \tau_{\text{Friction}}(\Delta p, \omega, D_{\text{Sat}}),$$

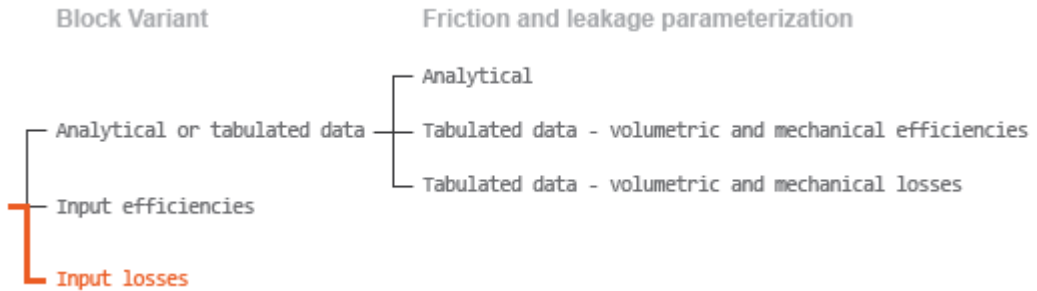
where $q_{\text{Leak}}(\Delta p, \omega)$ and $\tau_{\text{Friction}}(\Delta p, \omega)$ are the volumetric and mechanical losses, obtained through interpolation or extrapolation of the tabulated data specified via the **Volumetric loss table** and **Mechanical loss table** block parameters.



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, $N\cdot m$

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 1-1014.

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 cm³/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 cm³/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 m² (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 $(\omega, \Delta p, D)$ 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 $(\omega, \Delta p, D)$ 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 $(\omega, \Delta p, D)$ 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, $e_v(dp, \omega, D)$ — 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 $(\omega, \Delta p, D)$ 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 $(\omega, \Delta p, D)$ 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 cm³/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, Δp — 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 $(\omega, \Delta p, D)$ 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, ω — 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 $(\omega, \Delta p, D)$ 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 $(\omega, \Delta p, D)$ 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, $q_{\text{loss}}(\Delta p, \omega, D)$ — Internal leakage flow rates at the specified pressure gains, angular velocities, and displacements

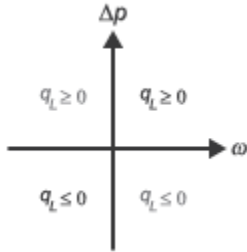
M -by- N -by- L matrix with units of volume/time

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, Δp** parameter.
- N is the number of vector elements in the **Shaft angular velocity vector for losses, ω** 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 $(\omega, \Delta p, D)$ 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.

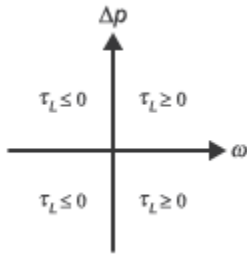
Mechanical loss table, torque_loss(dp,w,D) — Friction torques at the specified pressure gains, angular velocities, and displacements

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 $(\omega, \Delta p, D)$ 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 cm³/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 m² (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 modes0.5 cm³/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 outlet0.01 m² (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

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.

Glossary

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.