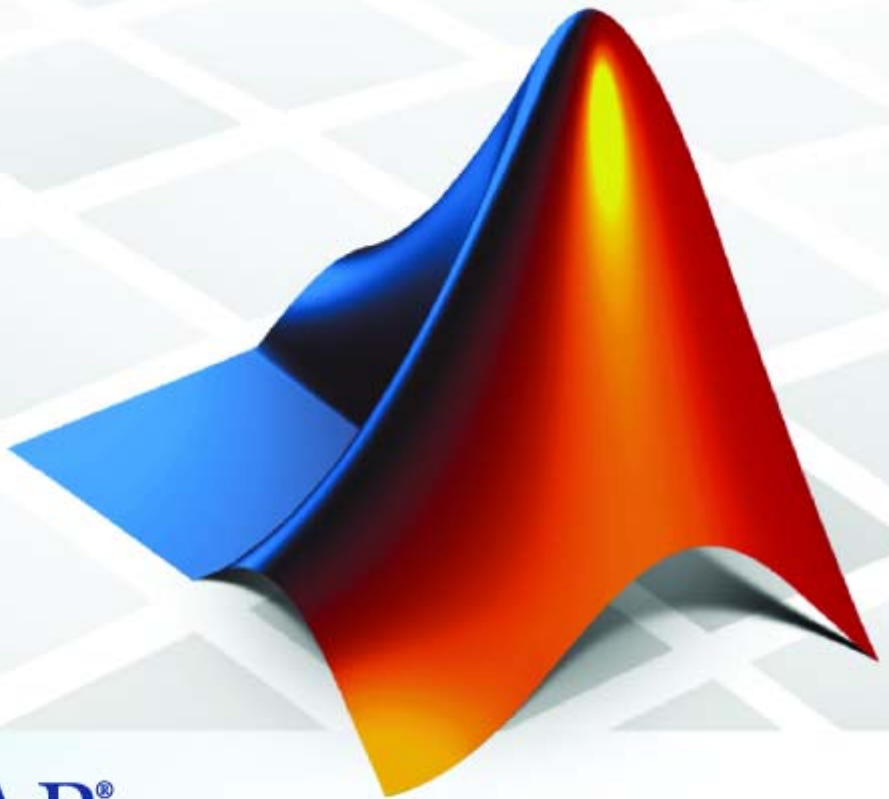


SimHydraulics® 1

Reference



MATLAB®
& **SIMULINK®**

How to Contact The MathWorks



www.mathworks.com
comp.soft-sys.matlab
www.mathworks.com/contact_TS.html

Web
Newsgroup
Technical Support



suggest@mathworks.com
bugs@mathworks.com
doc@mathworks.com
service@mathworks.com
info@mathworks.com

Product enhancement suggestions
Bug reports
Documentation error reports
Order status, license renewals, passcodes
Sales, pricing, and general information



508-647-7000 (Phone)



508-647-7001 (Fax)



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

For contact information about worldwide offices, see the MathWorks Web site.

SimHydraulics Reference

© COPYRIGHT 2006–2007 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, Simulink, Stateflow, Handle Graphics, Real-Time Workshop, SimBiology, SimHydraulics, SimEvents, and xPC TargetBox are registered trademarks and The MathWorks, the L-shaped membrane logo, Embedded MATLAB, and PolySpace are trademarks of The MathWorks, Inc.

Other product or brand names are trademarks or registered trademarks of their respective holders.

Patents

The 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
September 2006 Online only
March 2007 Online only
September 2007 Online only

New for Version 1.0 (Release 2006a+)
Revised for Version 1.1 (Release 2006b)
Revised for Version 1.2 (Release 2007a)
Revised for Version 1.2.1 (Release 2007b)

Blocks — By Category

1

Accumulators	1-2
Hydraulic Cylinders	1-3
Hydraulic Utilities	1-4
Local Hydraulic Resistances	1-5
Orifices	1-6
Pipelines	1-7
Pumps and Motors	1-8
Valves	1-9
Directional Valves	1-9
Flow Control Valves	1-9
Pressure Control Valves	1-10
Valve Actuators	1-10
Valve Forces	1-11

Blocks — Alphabetical List

2

Index

Blocks — By Category

Accumulators (p. 1-2)

Hydraulic Cylinders (p. 1-3)

Hydraulic Utilities (p. 1-4)

Local Hydraulic Resistances (p. 1-5)

Orifices (p. 1-6)

Pipelines (p. 1-7)

Pumps and Motors (p. 1-8)

Valves (p. 1-9)

Hydraulic accumulators

Hydraulic cylinders

Environment blocks, such as hydraulic fluid

Various local hydraulic resistances

Hydraulic orifices, to be used as valve building blocks

Hydraulic pipelines

Hydraulic pumps and motors

Hydraulic valves

Accumulators

Gas-Charged Accumulator

Simulate hydraulic accumulator with gas as compressible medium

Spring-Loaded Accumulator

Simulate hydraulic accumulator with spring used for energy storage

Hydraulic Cylinders

Cylinder Friction	Simulate friction in hydraulic cylinders
Double-Acting Hydraulic Cylinder	Simulate hydraulic actuator exerting force in both directions
Double-Acting Rotary Actuator	Simulate double-acting hydraulic rotary actuator
Single-Acting Hydraulic Cylinder	Simulate hydraulic actuator exerting force in one direction
Single-Acting Rotary Actuator	Simulate single-acting hydraulic rotary actuator

Hydraulic Utilities

Hydraulic Fluid

Set working fluid properties by selecting from list of predefined fluids

Reservoir

Simulate pressurized hydraulic reservoir

Local Hydraulic Resistances

Elbow	Simulate hydraulic resistance in elbow
Gradual Area Change	Simulate gradual enlargement or contraction
Local Resistance	Simulate all kinds of hydraulic resistances specified by loss coefficient
Pipe Bend	Simulate hydraulic resistance in pipe bend
Sudden Area Change	Simulate sudden enlargement or contraction
T-junction	Simulate hydraulic resistance of T-junction in pipe

Orifices

Annular Orifice	Simulate hydraulic variable orifice created by circular tube and round insert
Fixed Orifice	Simulate hydraulic orifice with constant cross-sectional area
Orifice with Variable Area Round Holes	Simulate hydraulic variable orifice shaped as set of round holes drilled in sleeve
Orifice with Variable Area Slot	Simulate hydraulic variable orifice shaped as rectangular slot
Variable Orifice	Simulate generic hydraulic variable orifice

Pipelines

Hydraulic Pipeline

Simulate hydraulic pipeline with resistive and fluid compressibility properties

Segmented Pipeline

Simulate hydraulic pipeline with resistive, fluid inertia, and fluid compressibility properties

Pumps and Motors

Centrifugal Pump

Simulate centrifugal pump

Fixed-Displacement Pump

Simulate fixed-displacement hydraulic pump

Hydraulic Motor

Simulate fixed-displacement hydraulic motor

Variable-Displacement Motor

Simulate variable-displacement reversible hydraulic motor

Variable-Displacement Pressure-Compensated Pump

Simulate hydraulic pump maintaining preset pressure at outlet by regulating its flow delivery

Variable-Displacement Pump

Simulate variable-displacement reversible hydraulic pump

Valves

Directional Valves (p. 1-9)	Hydraulic directional valves
Flow Control Valves (p. 1-9)	Hydraulic flow control valves
Pressure Control Valves (p. 1-10)	Hydraulic pressure control valves
Valve Actuators (p. 1-10)	Actuators for driving directional valves
Valve Forces (p. 1-11)	Blocks that simulate hydraulic forces exerted on valves

Directional Valves

2-Way Directional Valve	Simulate hydraulic continuous 2-way directional valve
3-Way Directional Valve	Simulate hydraulic continuous 3-way directional valve
4-Way Directional Valve	Simulate hydraulic continuous 4-way directional valve
Check Valve	Simulate hydraulic valve that allows flow in one direction only
Pilot-Operated Check Valve	Simulate hydraulic check valve that allows flow in one direction, but can be disabled by pilot pressure
Shuttle Valve	Simulate hydraulic valve that allows flow in one direction only

Flow Control Valves

Ball Valve	Simulate hydraulic ball valve
Needle Valve	Simulate hydraulic needle valve

Poppet Valve	Simulate hydraulic poppet valve
Pressure-Compensated Flow Control Valve	Simulate hydraulic pressure compensating valve

Pressure Control Valves

Pressure Compensator	Simulate hydraulic pressure compensating valve
Pressure Reducing Valve	Simulate pressure control valve maintaining reduced pressure in portion of system
Pressure Relief Valve	Simulate pressure control valve maintaining preset pressure in system

Valve Actuators

2-Position Valve Actuator	Simulate actuator for two-position valves
3-Position Valve Actuator	Simulate actuator for three-position valves
Hydraulic Double-Acting Valve Actuator	Simulate double-acting hydraulic valve actuator
Hydraulic Single-Acting Valve Actuator	Simulate single-acting hydraulic valve actuator
Proportional and Servo-Valve Actuator	Simulate continuous valve driver with output proportional to input signal

Valve Forces

Spool Orifice Hydraulic Force

Simulate axial hydraulic force exerted on spool

Valve Hydraulic Force

Simulate axial hydraulic static force exerted on valve

Blocks — Alphabetical List

2-Position Valve Actuator

Purpose Simulate actuator for two-position valves

Library Valve Actuators

Description

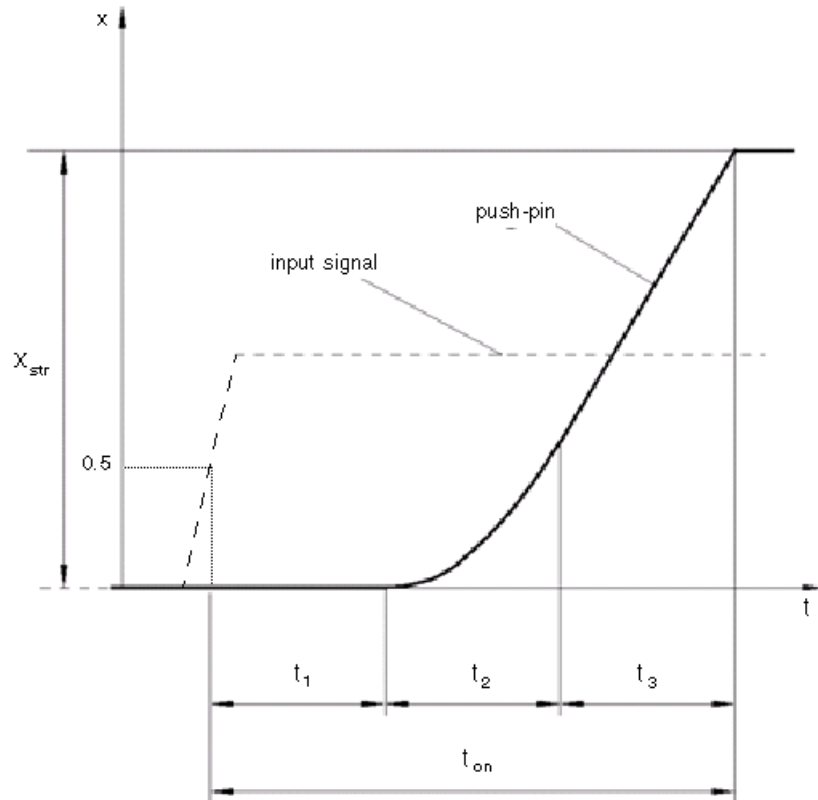


The 2-Position Valve Actuator block represents an actuator that you can use with directional valves to control their position. This actuator can drive a two-position valve. The block is developed as a data-sheet-based model and all its parameters are generally provided in catalogs or data sheets. The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

2-Position Valve Actuator



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 be removed, which causes the push-pin to retract. The retract motion follows exactly the same profile but “stretches” over

2-Position Valve Actuator

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.

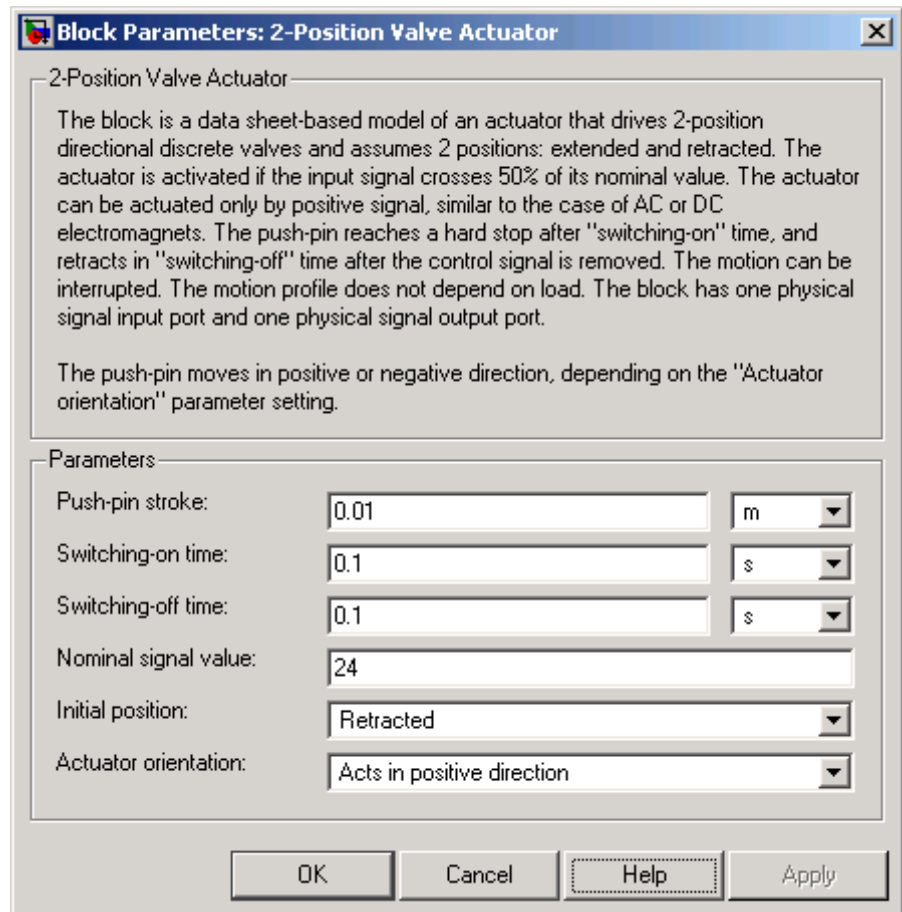
The push-pin is actuated only by positive signal, similar to the AC or DC electromagnets. The direction of push-pin motion is controlled by the **Actuator orientation** parameter, which can have one of two values: Acts in positive direction or Acts in negative direction.

Basic Assumptions and Limitations

The model is based on the following assumption:

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

Dialog Box and 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.

2-Position Valve Actuator

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.

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 demo (sh_2_pos_valve_actuator), 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 Hydraulic Circuit with Single-Acting Cylinder demo (sh_circuit_sa_cylinder), 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

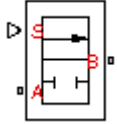
- 3-Position Valve Actuator
- Hydraulic Double-Acting Valve Actuator
- Hydraulic Single-Acting Valve Actuator
- Proportional and Servo-Valve Actuator

2-Way Directional Valve

Purpose Simulate 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 (P) and outlet port (A), 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 failure of computation.

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 monotonically 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

2-Way Directional Valve

is determined as $p = p_A - p_B$. Positive signal at the physical signal port S opens the valve.

Basic Assumptions and Limitations

The model is based on the following assumptions:

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

Dialog Box and Parameters

Block Parameters: 2-Way Directional Valve

2-Way Directional Valve

The block simulates a 2-way directional valve as a data sheet-based model. To parameterize the block, 3 options are available: (1) by maximum area and control member stroke, (2) by the table of valve area vs. control member displacement, and (3) by the pressure-flow rate characteristics. The lookup table block is used in the second and third cases for interpolation and extrapolation. 3 methods of interpolation and 2 methods of extrapolation are provided to choose from.

Connections A and B are hydraulic conserving ports associated with the valve inlet and outlet, respectively. Connection S is a physical signal port. The block positive direction is from port A to port B. Positive signal at port S opens the valve.

Parameters

Model parameterization:	<input type="text" value="By maximum area and opening"/>	
Valve passage maximum area:	<input type="text" value="5e-05"/>	<input type="text" value="m^2"/>
Valve maximum opening:	<input type="text" value="0.005"/>	<input type="text" value="m"/>
Flow discharge coefficient:	<input type="text" value="0.7"/>	
Initial opening:	<input type="text" value="0"/>	<input type="text" value="m"/>
Critical Reynolds number:	<input type="text" value="12"/>	
Leakage area:	<input type="text" value="1e-12"/>	<input type="text" value="m^2"/>

OK Cancel Help Apply

2-Way Directional Valve

Block Parameters: 2-Way Directional Valve

2-Way Directional Valve

The block simulates a 2-way directional valve as a data sheet-based model. To parameterize the block, 3 options are available: (1) by maximum area and control member stroke, (2) by the table of valve area vs. control member displacement, and (3) by the pressure-flow rate characteristics. The lookup table block is used in the second and third cases for interpolation and extrapolation. 3 methods of interpolation and 2 methods of extrapolation are provided to choose from.

Connections A and B are hydraulic conserving ports associated with the valve inlet and outlet, respectively. Connection S is a physical signal port. The block positive direction is from port A to port B. Positive signal at port S opens the valve.

Parameters

Model parameterization: By area vs. opening table

Tabulated valve openings: [-0.002 0 0.005 0.015] m

Tabulated valve passage area: [1e-12 4e-12 1e-05 1.02e-05] m²

Interpolation method: Linear

Extrapolation method: From last 2 points

Flow discharge coefficient: 0.7

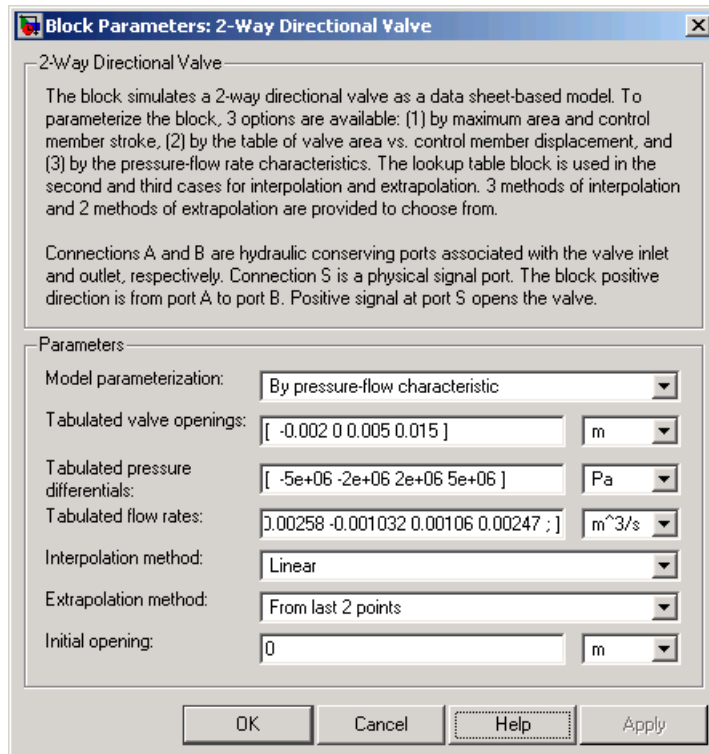
Initial opening: 0 m

Critical Reynolds number: 12

Leakage area: 1e-12 m²

OK Cancel Help Apply

2-Way Directional Valve



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 three 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 three 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 tabulated 1-by- m array. The input values vector must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. The default values, in meters, are $[-2e-3, 0, 5e-3, 15e-3]$. 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 tabulated 1-by- m array. The valve passage area vector

2-Way Directional Valve

must be the same size as the valve 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.

Tabulated pressure differentials

Specify the vector of input values for pressure differentials as a tabulated 1-by-n array. The input values vector must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. The default values, in Pa, are $[-5e6, -2e6, 2e6, 5e6]$. This parameter is used if **Model parameterization** is set to By pressure-flow characteristic.

Tabulated flow rates

Specify the output values for flow rates as a tabulated m-by-n matrix, defining the function values at the input grid vertices. 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-12,  -0.6e-13,   0.6e13,    0.1e12; ...  
  -0.8e-7, -0.4e-7,   0.4e7,     0.8e7; ...  
  -2.5e-3, -1.03e-3,  1.e-3,    2.4e-3; ...  
  -2.58e-3, -1.032e-3, 1.06e-3,  2.47e-3]
```

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** — For one-dimensional table lookup (By area vs. opening table), uses a linear interpolation function. For two-dimensional table lookup (By pressure-flow characteristic), uses a bilinear interpolation algorithm,

which is an extension of linear interpolation for functions in two variables.

- **Cubic** — For one-dimensional table lookup (By area vs. opening table), uses the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP). For two-dimensional table lookup (By pressure-flow characteristic), uses the bicubic interpolation algorithm.
- **Spline** — For one-dimensional table lookup (By area vs. opening table), uses the cubic spline interpolation algorithm. For two-dimensional table lookup (By pressure-flow characteristic), uses the bicubic spline interpolation algorithm.

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:

- **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.

2-Way Directional Valve

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.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

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. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 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

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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 Hydraulic Closed-Loop Circuit with 2-Way Valve demo (`sh_closed_loop_circuit_2_way_valve`), 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

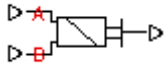
- 3-Way Directional Valve
- 4-Way Directional Valve

3-Position Valve Actuator

Purpose Simulate actuator for three-position valves

Library Valve Actuators

Description

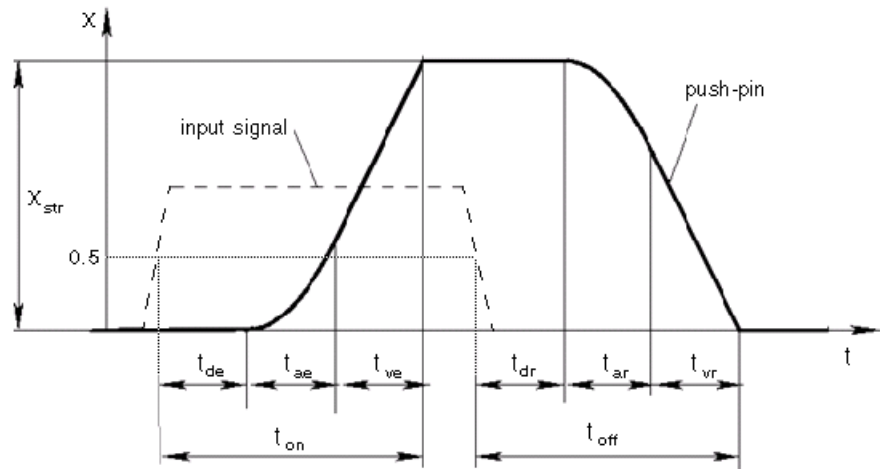


The 3-Position Valve Actuator block represents an actuator that you can use with directional valves to control their position. This actuator can drive a three-position valve. The block is developed as a data-sheet-based model and all its parameters are generally provided in catalogs or data sheets. The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

3-Position Valve Actuator



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

3-Position Valve Actuator

is interrupted, the switch-on or switch-off times are proportionally decreased depending on the instantaneous push-pin position.

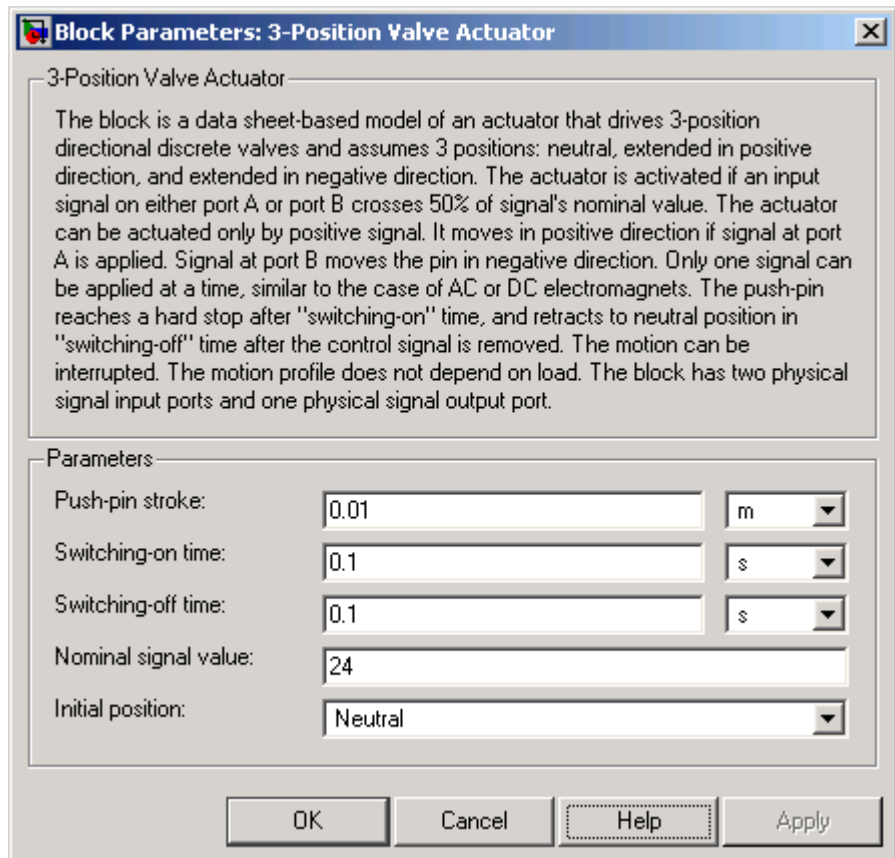
Only positive signals activate the actuator. In other words, negative signals at ports A and B have no effect on the actuator, which is similar to the behavior of electromagnetically controlled 3-position directional valves.

Basic Assumptions and Limitations

The model is based on the following assumption:

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

Dialog Box and 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.

3-Position Valve Actuator

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`.

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 demo (`sh_3_pos_valve_actuator`), 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 demo.

See Also

2-Position Valve Actuator

Hydraulic Double-Acting Valve Actuator

Hydraulic Single-Acting Valve Actuator

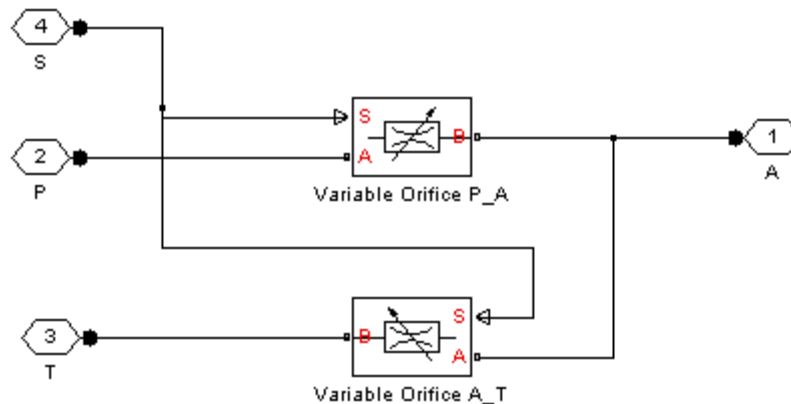
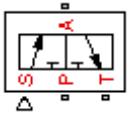
Proportional and Servo-Valve Actuator

3-Way Directional Valve

Purpose Simulate hydraulic continuous 3-way directional valve

Library Directional Valves

Description The 3-Way Directional Valve block represents a continuous, symmetrical, 3-way directional valve. The fluid flow is pumped in the valve through the inlet line and is distributed between an outside pressure line (usually connected to a single-acting actuator) and the return line. The block has three hydraulic connections, corresponding to inlet port (P), actuator port (A), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of two Variable Orifice blocks, connected as shown in the following diagram.



One Variable Orifice block, called `orifice_PA`, is installed in the P-A path. The second Variable Orifice block, called `orifice_AT`, is installed in the A-T path. Both 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 `orifice_PA` and closes `orifice_AT`. As a result, the openings of the orifices are computed as follows:

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

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

where

h_{PA}	Orifice opening for the orifice_PA block
h_{AT}	Orifice opening for the orifice_AT block
h_{PA0}	Initial opening for the orifice_PA block
h_{AT0}	Initial opening for the orifice_AT block
x	Control member displacement from initial position

The valve simulated by the 3-Way Directional Valve block is assumed to be symmetrical. This means that both orifices are of the same shape and size and are parameterized with the same method. You can choose one of the following block 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

3-Way Directional Valve

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 failure of computation.

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 monotonically increasing. The vertices can be nonuniformly spaced. You have a choice of three interpolation methods and two extrapolation methods.

If you need to simulate a nonsymmetrical 3-way valve (i.e., with different orifices), use any of the variable orifice blocks from the Building Blocks library (such as Orifice with Variable Area Round Holes, Orifice with Variable Area Slot, or Variable Orifice) and connect them the same way as the Variable Orifice blocks in the schematic diagram of this 3-Way Directional Valve block.

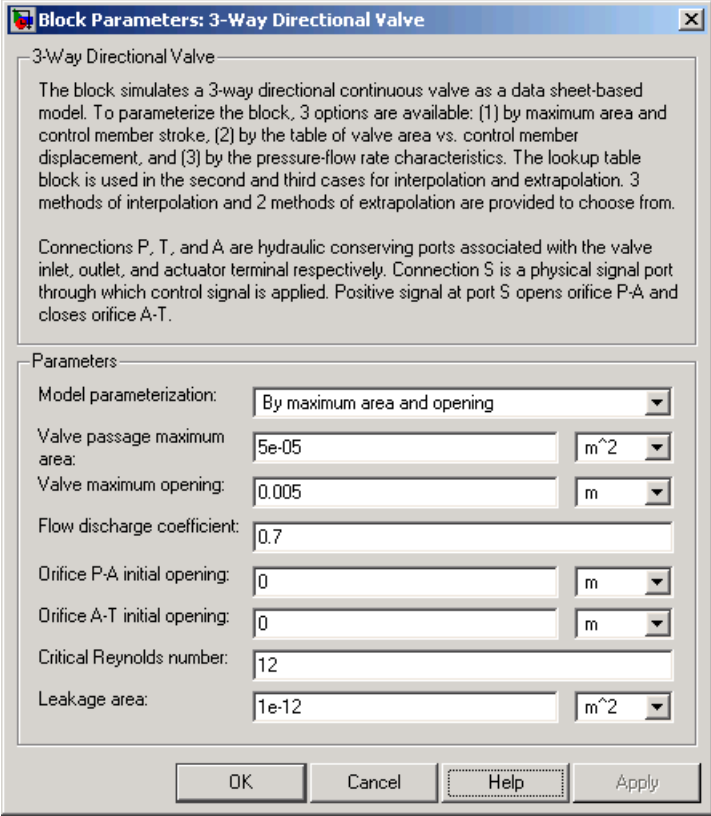
Positive signal at the physical signal port S opens the orifice in the P-A path and closes the orifice in the A-T path. The directionality of nested blocks is clear from the schematic diagram.

Basic Assumptions and Limitations

The model is based on the following assumptions:

- Fluid inertia is not taken into account.
- Spool loading, such as inertia, spring, hydraulic forces, and so on, is not taken into account.
- Only symmetrical configuration of the valve is considered. In other words, both orifices are assumed to have the same shape and size.

Dialog Box and Parameters



Block Parameters: 3-Way Directional Valve

3-Way Directional Valve

The block simulates a 3-way directional continuous valve as a data sheet-based model. To parameterize the block, 3 options are available: (1) by maximum area and control member stroke, (2) by the table of valve area vs. control member displacement, and (3) by the pressure-flow rate characteristics. The lookup table block is used in the second and third cases for interpolation and extrapolation. 3 methods of interpolation and 2 methods of extrapolation are provided to choose from.

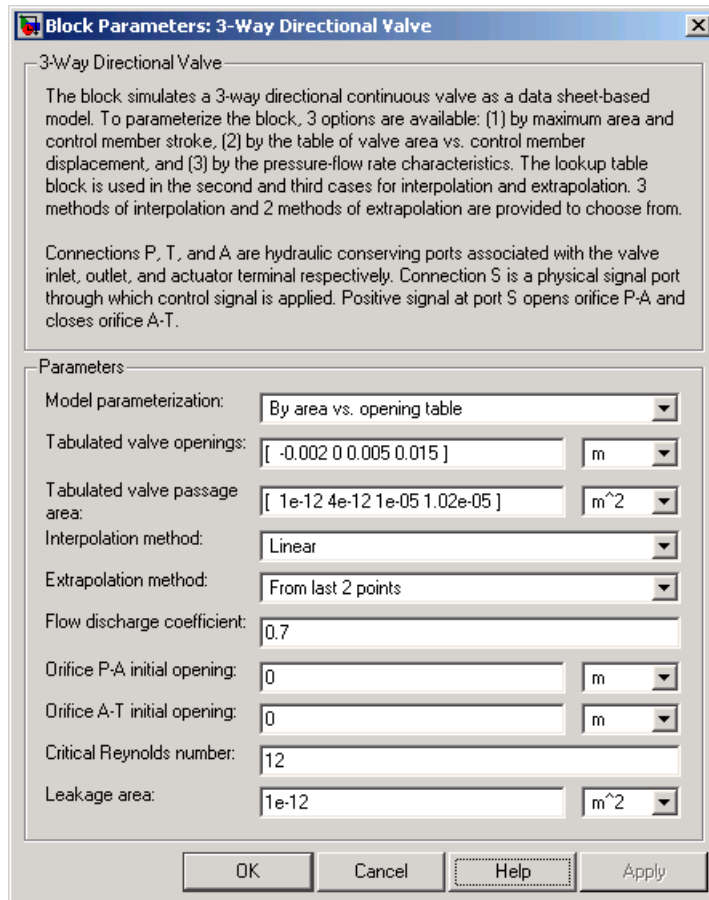
Connections P, T, and A are hydraulic conserving ports associated with the valve inlet, outlet, and actuator terminal respectively. Connection S is a physical signal port through which control signal is applied. Positive signal at port S opens orifice P-A and closes orifice A-T.

Parameters

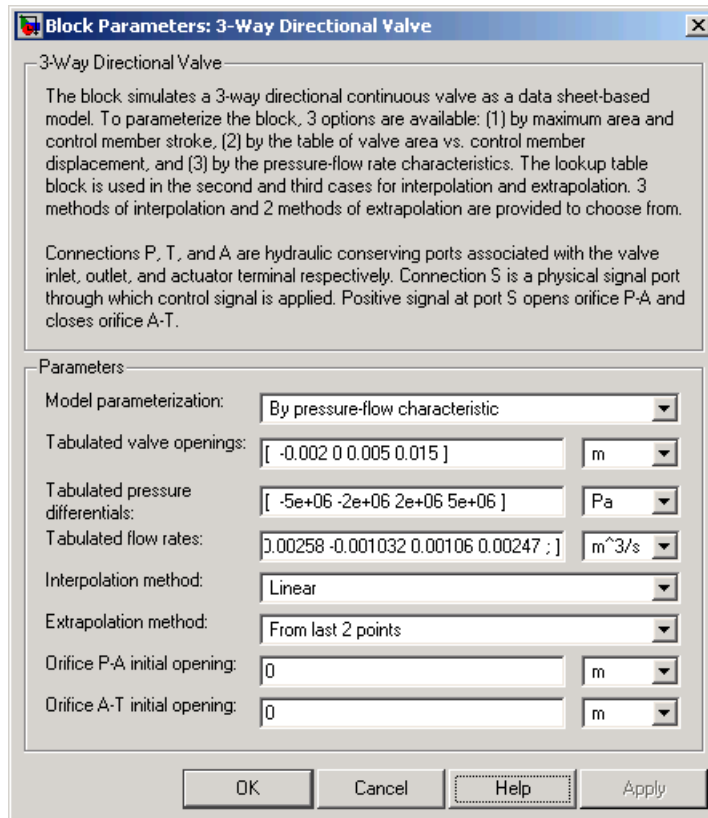
Model parameterization:	By maximum area and opening	
Valve passage maximum area:	5e-05	m ²
Valve maximum opening:	0.005	m
Flow discharge coefficient:	0.7	
Orifice P-A initial opening:	0	m
Orifice A-T initial opening:	0	m
Critical Reynolds number:	12	
Leakage area:	1e-12	m ²

OK Cancel Help Apply

3-Way Directional Valve



3-Way Directional Valve



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.

3-Way Directional Valve

- **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 three 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 three 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 tabulated 1-by- m array. The input values vector must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. The default values, in meters, are $[-2e-3, 0, 5e-3, 15e-3]$. 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 tabulated 1-by-m array. The valve passage area vector must be the same size as the valve 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.

Tabulated pressure differentials

Specify the vector of input values for pressure differentials as a tabulated 1-by-n array. The input values vector must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. The default values, in Pa, are [-5e6, -2e6, 2e6, 5e6]. This parameter is used if **Model parameterization** is set to By pressure-flow characteristic.

Tabulated flow rates

Specify the output values for flow rates as a tabulated m-by-n matrix, defining the function values at the input grid vertices. 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-12,  -0.6e-13,   0.6e13,    0.1e12; ...  
  -0.8e-7, -0.4e-7,   0.4e7,     0.8e7; ...  
  -2.5e-3, -1.03e-3,  1.e-3,     2.4e-3; ...  
  -2.58e-3, -1.032e-3, 1.06e-3,  2.47e-3]
```

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** — For one-dimensional table lookup (By area vs. opening table), uses a linear interpolation function.

3-Way Directional Valve

For two-dimensional table lookup (By pressure-flow characteristic), uses a bilinear interpolation algorithm, which is an extension of linear interpolation for functions in two variables.

- Cubic — For one-dimensional table lookup (By area vs. opening table), uses the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP). For two-dimensional table lookup (By pressure-flow characteristic), uses the bicubic interpolation algorithm.
- Spline — For one-dimensional table lookup (By area vs. opening table), uses the cubic spline interpolation algorithm. For two-dimensional table lookup (By pressure-flow characteristic), uses the bicubic spline interpolation algorithm.

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:

- 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) 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.

Orifice P-A initial opening

Initial opening for the orifice in the P-A path. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is 0.

Orifice A-T initial opening

Initial opening for the orifice in the A-T path. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is 0.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

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. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is $1e-12 \text{ m}^2$.

3-Way Directional Valve

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

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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 pressure supply line inlet.

T

Hydraulic conserving port associated with the return line connection.

A

Hydraulic conserving port associated with the actuator connection port.

S

Physical signal port to control spool displacement.

Examples

The 3-Way Directional Valve block is demonstrated in the Hydraulic Circuit with 3-Way Valve and Differential Cylinder demo (`sh_circuit_3_way_valve_diff_cylinder`), where it is used to switch between a conventional and differential connection of the cylinder.

See Also

2-Way Directional Valve

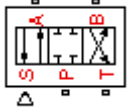
4-Way Directional Valve

4-Way Directional Valve

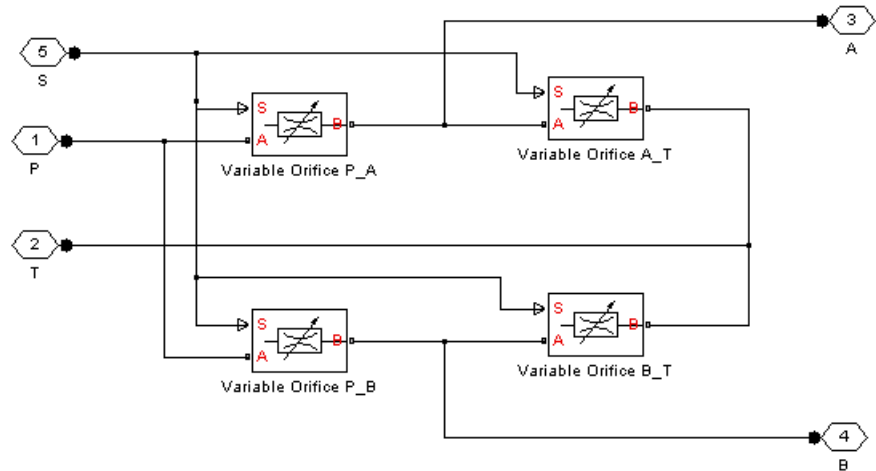
Purpose Simulate hydraulic continuous 4-way directional valve

Library Directional Valves

Description



The 4-Way Directional Valve block represents a continuous, symmetrical, 4-way directional valve. The fluid flow is pumped in the valve through the inlet line and is distributed between two outside pressure lines (usually connected to a double-acting actuator) and the return line. 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.



The Variable Orifice blocks are installed as follows: orifice_PA is in the P-A path, orifice_PB is in the P-B path, orifice_AT is in the A-T path, and orifice_BT is in the B-T path. 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 orifice_PA and

orifice_BT and closes orifice_PB and orifice_AT. As a result, the openings of the orifices are computed as follows:

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

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

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

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

where

h_{PA}	Orifice opening for the orifice_PA block
h_{PB}	Orifice opening for the orifice_PB block
h_{AT}	Orifice opening for the orifice_AT block
h_{BT}	Orifice opening for the orifice_BT block
h_{PA0}	Initial opening for the orifice_PA block
h_{PB0}	Initial opening for the orifice_PB block
h_{AT0}	Initial opening for the orifice_AT block
h_{BT0}	Initial opening for the orifice_BT block
x	Control member displacement from initial position

The valve simulated by the 4-Way Directional Valve block is assumed to be symmetrical. In other words, all four orifices are of the same shape and size and are parameterized with the same method. You can choose one of the following block 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.

4-Way Directional Valve

- 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 failure of computation.

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 monotonically increasing. The vertices can be nonuniformly spaced. You have a choice of three interpolation methods and two extrapolation methods.

If you need to simulate a nonsymmetrical 4-way valve (i.e., with different orifices), use any of the variable orifice blocks from the Building Blocks library (such as Orifice with Variable Area Round Holes, Orifice with Variable Area Slot, or Variable Orifice) and connect them the same way as the Variable Orifice blocks in the schematic diagram of this 4-Way Directional Valve block.

Positive signal at the physical signal port S opens the orifices in the P-A and B-T paths and closes the orifices in the P-B and A-T paths. The directionality of nested blocks is clear from the schematic diagram.

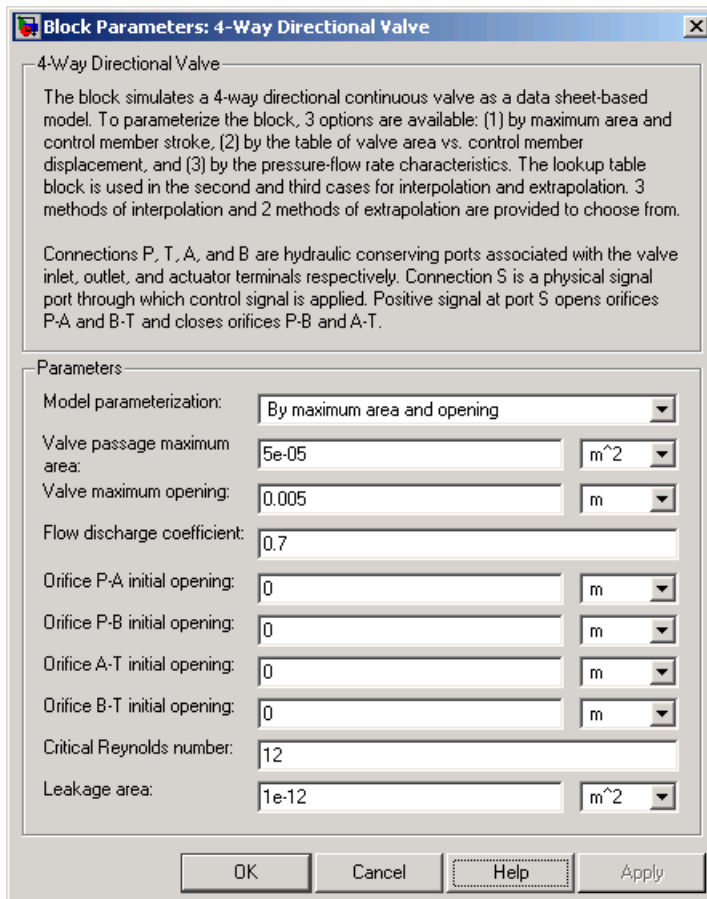
Basic Assumptions and Limitations

The model is based on the following assumptions:

- Fluid inertia is not taken into account.
- Spool loading, such as inertia, spring, hydraulic forces, and so on, is not taken into account.
- Only symmetrical configuration of the valve is considered. In other words, all four orifices are assumed to have the same shape and size.

4-Way Directional Valve

Dialog Box and Parameters



4-Way Directional Valve

Block Parameters: 4-Way Directional Valve

4-Way Directional Valve

The block simulates a 4-way directional continuous valve as a data sheet-based model. To parameterize the block, 3 options are available: (1) by maximum area and control member stroke, (2) by the table of valve area vs. control member displacement, and (3) by the pressure-flow rate characteristics. The lookup table block is used in the second and third cases for interpolation and extrapolation. 3 methods of interpolation and 2 methods of extrapolation are provided to choose from.

Connections P, T, A, and B are hydraulic conserving ports associated with the valve inlet, outlet, and actuator terminals respectively. Connection S is a physical signal port through which control signal is applied. Positive signal at port S opens orifices P-A and B-T and closes orifices P-B and A-T.

Parameters

Model parameterization: By area vs. opening table

Tabulated valve openings: [-0.002 0 0.005 0.015] m

Tabulated valve passage area: [1e-12 4e-12 1e-05 1.02e-05] m²

Interpolation method: Linear

Extrapolation method: From last 2 points

Flow discharge coefficient: 0.7

Orifice P-A initial opening: 0 m

Orifice P-B initial opening: 0 m

Orifice A-T initial opening: 0 m

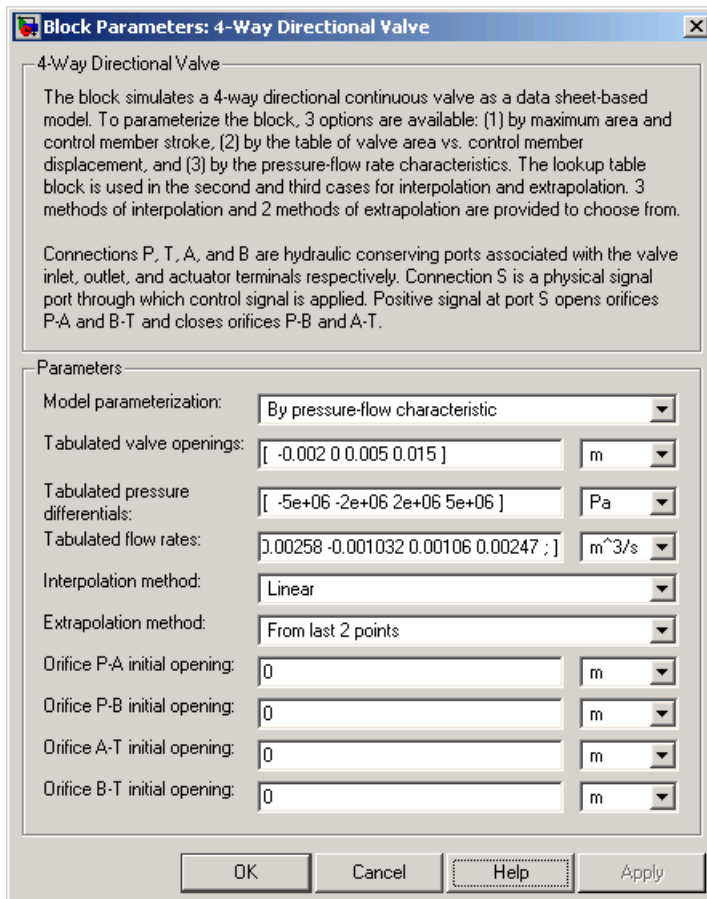
Orifice B-T initial opening: 0 m

Critical Reynolds number: 12

Leakage area: 1e-12 m²

OK Cancel Help Apply

4-Way Directional Valve



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 three 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 three 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 tabulated 1-by-m array. The input values vector must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. The default values, in meters, are $[-2e-3, 0, 5e-3, 15e-3]$. 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

4-Way Directional Valve

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 tabulated 1-by-m array. The valve passage area vector must be the same size as the valve 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.

Tabulated pressure differentials

Specify the vector of input values for pressure differentials as a tabulated 1-by-n array. The input values vector must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. The default values, in Pa, are [-5e6, -2e6, 2e6, 5e6]. This parameter is used if **Model parameterization** is set to By pressure-flow characteristic.

Tabulated flow rates

Specify the output values for flow rates as a tabulated m-by-n matrix, defining the function values at the input grid vertices. 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-12,  -0.6e-13,   0.6e13,    0.1e12; ...  
  -0.8e-7,  -0.4e-7,    0.4e7,     0.8e7; ...  
  -2.5e-3,  -1.03e-3,   1.e-3,     2.4e-3; ...  
  -2.58e-3, -1.032e-3,  1.06e-3,   2.47e-3]
```

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**—For one-dimensional table lookup (By area vs. opening table), uses a linear interpolation function. For two-dimensional table lookup (By pressure-flow characteristic), uses a bilinear interpolation algorithm, which is an extension of linear interpolation for functions in two variables.
- **Cubic**—For one-dimensional table lookup (By area vs. opening table), uses the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP). For two-dimensional table lookup (By pressure-flow characteristic), uses the bicubic interpolation algorithm.
- **Spline**—For one-dimensional table lookup (By area vs. opening table), uses the cubic spline interpolation algorithm. For two-dimensional table lookup (By pressure-flow characteristic), uses the bicubic spline interpolation algorithm.

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:

- **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

4-Way Directional Valve

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) 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.

Orifice P-A initial opening

Initial opening for the orifice in the P-A path. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is 0.

Orifice P-B initial opening

Initial opening for the orifice in the P-B path. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is 0.

Orifice A-T initial opening

Initial opening for the orifice in the A-T path. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is 0.

Orifice B-T initial opening

Initial opening for the orifice in the B-T path. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is 0.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

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. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 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

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

4-Way Directional Valve

- 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.

Examples

The 3-Way Directional Valve block in the Closed-Loop Circuit with 4-Way Valve demo (`sh_closed_loop_circuit_4_way_valve_cust_cyl`) is an open-center, symmetrical valve controlling a double-acting cylinder.

See Also

- 2-Way Directional Valve
- 3-Way Directional Valve

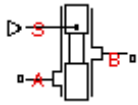
Purpose

Simulate hydraulic variable orifice created by circular tube and round insert

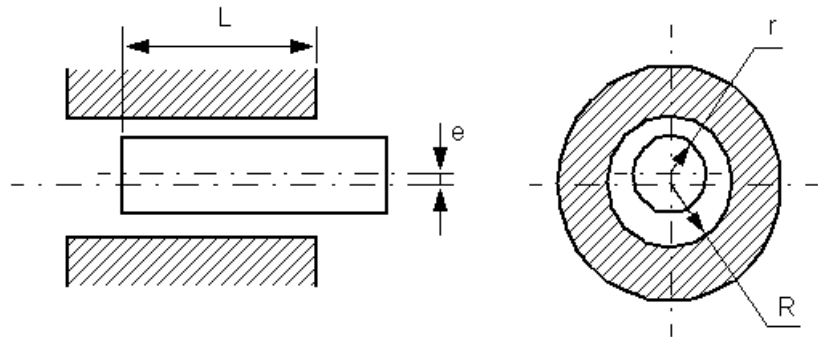
Library

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]):

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

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

where

Annular Orifice

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.

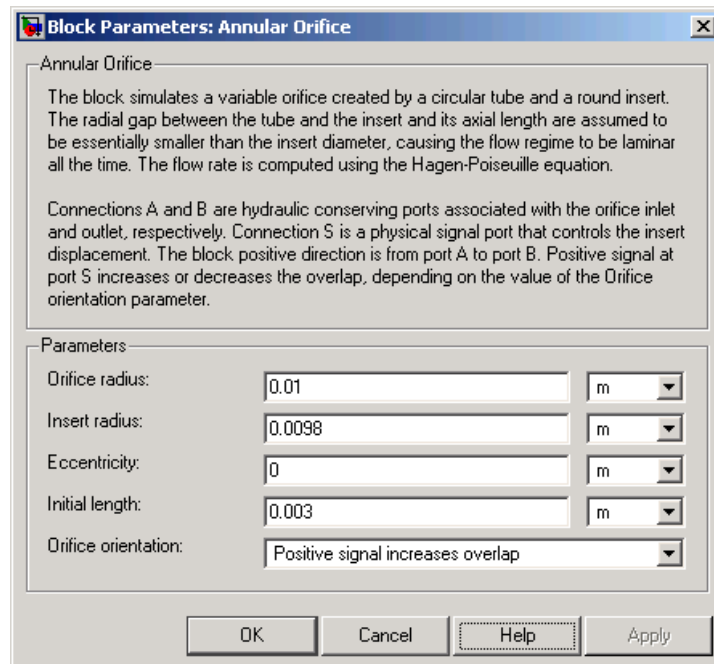
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as $p = p_A - p_B$. Positive signal at the physical signal port S increases or decreases the overlap, depending on the value of the parameter **Orifice orientation**.

Basic Assumptions and Limitations

The model is based on the following assumption:

- Fluid inertia is not taken into account.

Dialog Box and 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.

Annular Orifice

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

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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 Orifice

Fixed Orifice

Orifice with Variable Area Round Holes

Orifice with Variable Area Slot

Variable Area Orifice

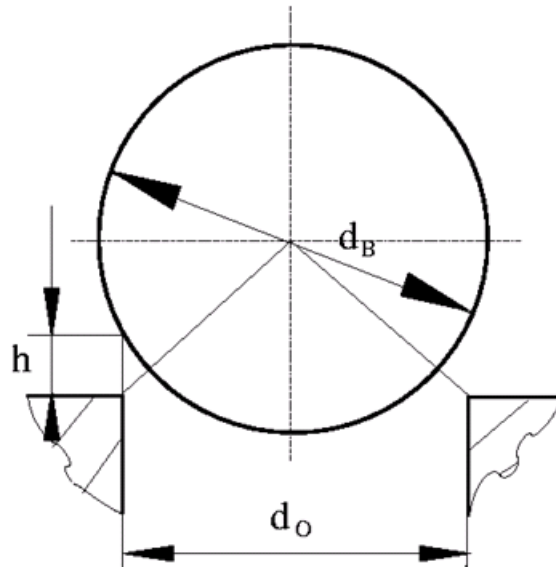
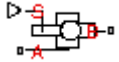
Variable Orifice

Ball Valve

Purpose Simulate hydraulic ball valve

Library Flow Control Valves

Description The Ball Valve block models a variable orifice created by a spherical ball and a round sharp-edged orifice.



The flow rate through the valve is proportional to the valve opening and to the pressure differential across the valve. 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 flow rate is determined according to the following equations:

$$q = \begin{cases} C_D \cdot A \sqrt{\frac{2}{\rho} |p| \cdot \text{sign}(p)} & \text{for } Re \geq Re_{cr} \\ 2C_{DL} \cdot A \frac{D_H}{v \cdot \rho} p & \text{for } Re < Re_{cr} \end{cases}$$

$$h = x_0 + x$$

$$A(h) = \begin{cases} A_{leak} & \text{for } h \leq 0 \\ \pi \cdot r_O \left(1 - \frac{r_B}{D^2}\right) \cdot D & \text{for } 0 < h < h_{max} \\ A_{max} + A_{leak} & \text{for } h \geq h_{max} \end{cases}$$

$$D = \sqrt{\left(\sqrt{r_B^2 - r_O^2} + h\right)^2 + r_O^2}$$

$$p = p_A - p_B$$

$$Re = \frac{q \cdot D_H}{A(h) \cdot v}$$

$$C_{DL} = \left(\frac{C_D}{\sqrt{Re_{cr}}}\right)^2$$

$$D_H = \sqrt{\frac{4A(h)}{\pi}}$$

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

where

Ball Valve

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	Ball displacement from initial position
h	Valve opening
d_o	Orifice diameter
r_o	Orifice radius
d_B	Ball diameter
r_B	Ball radius
ρ	Fluid density
D_H	Valve instantaneous hydraulic diameter
ν	Fluid kinematic viscosity
A_{leak}	Closed valve leakage area
A_{max}	Maximum valve open area

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

The model is based on the following assumptions:

- Fluid inertia is not taken into account.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.

- The flow passage area is assumed to be equal to the side surface of the frustum of the cone located between the ball center and the orifice edge.

Dialog Box and Parameters

Block Parameters: Ball Valve

Ball Valve

The block models a ball valve created by a spherical ball and a round sharp-edged orifice. The flow rate through the valve is proportional to the orifice opening and to the pressure differential across the valve. The model accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number and comparing its value with the critical Reynolds number.

Connections A and B are conserving hydraulic ports associated with the valve inlet and outlet, respectively. Connection S is a physical signal port. The block positive direction is from port A to port B.

Parameters

Valve ball diameter: 0.01 m

Orifice diameter: 0.005 m

Initial opening: 0 m

Flow discharge coefficient: 0.65

Critical Reynolds number: 10

Leakage area: 1e-12 m²

OK Cancel Help Apply

Valve ball diameter

The diameter of the valve ball. It must be greater than the orifice diameter. The default value is 0.01 m.

Orifice diameter

The diameter of the orifice of the valve. The default value is 0.005 m.

Ball Valve

Initial opening

The initial opening of the valve. Its 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.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 10.

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. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is $1e-12 \text{ m}^2$.

Global Parameters

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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

Needle Valve
Poppet Valve
Pressure-Compensated Flow Control Valve

Centrifugal Pump

Purpose Simulate centrifugal pump

Library Pumps and Motors

Description



The Centrifugal Pump block represents a centrifugal pump of any type as a data-sheet-based model. The pump is parameterized with the polynomial whose coefficients are determined analytically or experimentally, depending on the data available. The relationship between pump characteristics and angular velocity is determined from the affinity laws. The model can be used if the shaft rotates in positive direction only.

The centrifugal pump is simulated with the following equation:

$$p = k \cdot p_E - p_{HL} - p_D \quad (2-1)$$

where

p Pressure differential across the pump

k Correlation 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 value

The Euler pressure, p_E , is determined with the Euler equation for centrifugal machines [1, 2] based on known pump dimensions. For an existing pump, the Euler pressure can be approximated with the equation

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

where

- ρ_{ref} Fluid density
- c_0, c_1 Approximating coefficients. They can be determined either analytically from the Euler equation [1, 2] or experimentally.
- q Pump volumetric delivery

The pressure loss due to hydraulic losses in the pump passages, p_{HL} , is approximated with the equation

$$p_{HL} = \rho_{ref} \cdot c_2 \cdot q^2$$

where

- ρ_{ref} Fluid density
- c_2 Approximating coefficient
- q Pump volumetric delivery

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

$$p_D = \rho_{ref} \cdot c_3 (q_D - q)$$

where

- ρ_{ref} Fluid density
- c_3 Approximating coefficient
- q Pump volumetric delivery
- q_D Pump design delivery (nominal delivery)

Centrifugal Pump

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 a different velocity, the affinity laws are used. First, the new reference delivery is computed with the expression

$$q_{ref} = q \frac{\omega_{ref}}{\omega} \quad (2-2)$$

where q and ω are the instantaneous values of the pump delivery and angular velocity. Then the pressure differential across the pump at a different angular velocity and density is determined with the formula

$$p = p_{ref} \cdot \left(\frac{\omega}{\omega_{ref}} \right)^2 \cdot \frac{\rho}{\rho_{ref}}$$

where p_{ref} is the pressure differential computed with Equation 2-1 at pump delivery determined according to Equation 2-2.

The pump efficiency is assumed to be the same as it is at the reference parameters. It is computed with the following equations:

$$\eta = \frac{N_{ref.hydr}}{N_{ref.br}}$$

$$N_{ref.hydr} = p_{ref} \cdot q_{ref}$$

$$N_{ref.br} = p_{Eref} \cdot q_{ref} + N_{mech.loss}$$

where

η Pump efficiency

$N_{ref.hydr}$ Power of the flow at the pump's outlet

p_{ref}	Pressure differential across the pump at delivery $q = q_{ref}$
q_{ref}	Pump reference delivery
p_{Eref}	Euler pressure at reference parameters
$N_{ref.br}$	Mechanical brake power at the pump's driving shaft
$N_{mech.loss}$	Power of mechanical losses in the pump drive train

Assuming that the efficiency remains the same at similar regimes, the torque at the driving shaft is determined from the following equation:

$$T = \frac{N_{ref.br}}{\omega_{ref}} \cdot \left(\frac{\omega}{\omega_{ref}} \right)^2 \cdot \frac{\rho}{\rho_{ref}}$$

The hydraulic power at the pump outlet is computed with the equation

$$N_{hyd} = p \cdot q$$

where p and q are the current values of the pump pressure differential and delivery, respectively.

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.

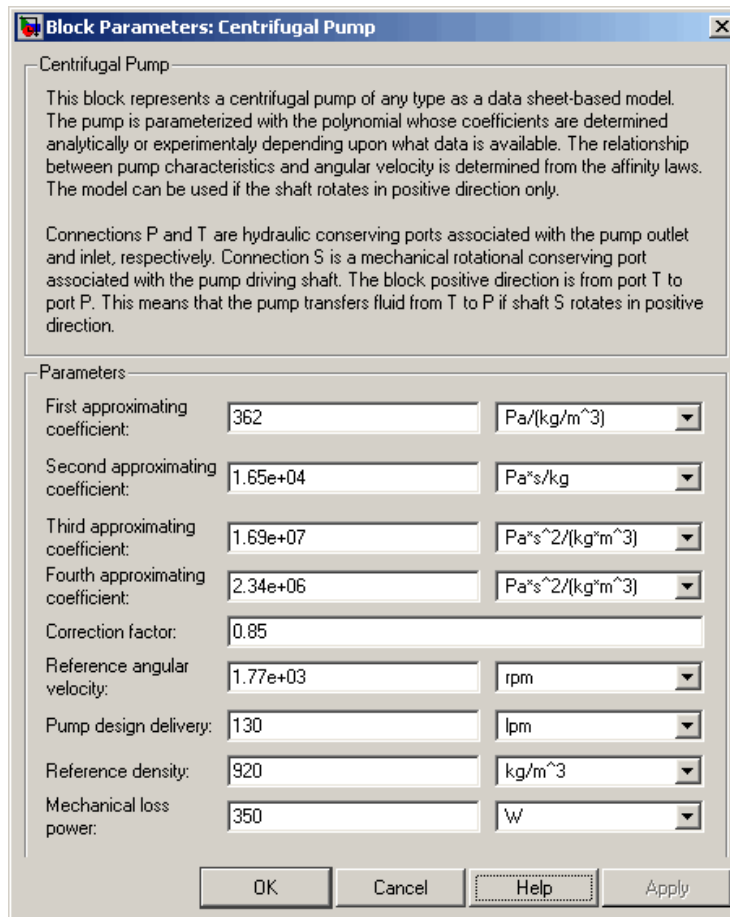
Basic Assumptions and Limitations

The model is based on the following assumptions:

- Fluid compressibility is neglected.
- The pump rotates in positive direction only.
- No reverse flow through the pump is allowed.
- The pump efficiency remains the same at similar regimes.

Centrifugal Pump

Dialog Box and Parameters



Centrifugal Pump

This block represents a centrifugal pump of any type as a data sheet-based model. The pump is parameterized with the polynomial whose coefficients are determined analytically or experimentally depending upon what data is available. The relationship between pump characteristics and angular velocity is determined from the affinity laws. The model can be used if the shaft rotates in positive direction only.

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 if shaft S rotates in positive direction.

Parameters

First approximating coefficient:	362	Pa/(kg/m ³)
Second approximating coefficient:	1.65e+04	Pa*s/kg
Third approximating coefficient:	1.69e+07	Pa*s ² /(kg*m ³)
Fourth approximating coefficient:	2.34e+06	Pa*s ² /(kg*m ³)
Correction factor:	0.85	
Reference angular velocity:	1.77e+03	rpm
Pump design delivery:	130	lpm
Reference density:	920	kg/m ³
Mechanical loss power:	350	W

OK Cancel Help Apply

First approximating coefficient

Approximating coefficient c_0 in the block description preceding. The default value is 362 Pa/(kg/m³).

Second approximating coefficient

Approximating coefficient c_1 in the block description preceding. The default value is 1.65e4 Pa*s/kg.

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 $1.69e7 \text{ Pa}\cdot\text{s}^2/(\text{kg}\cdot\text{m}^3)$.

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 $2.34e6 \text{ Pa}\cdot\text{s}^2/(\text{kg}\cdot\text{m}^3)$.

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 0.85.

Reference angular velocity

Angular velocity of the driving shaft, at which the pump characteristics are determined. The default value is 1.77e3 rpm.

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.

Reference density

Fluid density at which the pump characteristics are determined. The default value is $920 \text{ kg}/\text{m}^3$.

Mechanical loss power

Power of mechanical loss in the pump drive train at reference parameters. The default value is 350 W.

Global Parameters

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Centrifugal Pump

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, *Centrifugal Pump Clinic*, Marcel Decker, NY, 1981

See Also

Fixed-Displacement Pump

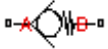
Variable-Displacement Pressure-Compensated Pump

Variable-Displacement Pump

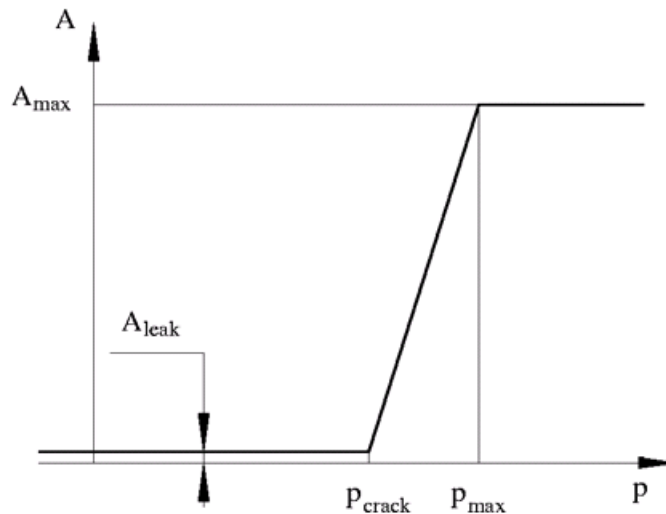
Purpose Simulate hydraulic valve that allows flow in one direction only

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

Check 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. Theoretically, the parameter can be set to zero, but it is not recommended.

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 flow rate is determined according to the following equations:

$$q = \begin{cases} C_D \cdot A \sqrt{\frac{2}{\rho} |p| \cdot \text{sign}(p)} & \text{for } Re \geq Re_{cr} \\ 2C_{DL} \cdot A \frac{D_H}{\nu \cdot \rho} p & \text{for } Re < Re_{cr} \end{cases}$$

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

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

$$p = p_A - p_B$$

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

$$C_{DL} = \left(\frac{C_D}{\sqrt{\text{Re}_{cr}}} \right)^2$$

$$D_H = \sqrt{\frac{4A(p)}{\pi}}$$

where

q	Flow rate through the valve
p	Pressure differential across the valve
p_A, p_B	Gauge pressures at the block terminals
C_D	Flow discharge coefficient
$A(p)$	Instantaneous orifice passage area
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
D_H	Instantaneous orifice hydraulic diameter
ρ	Fluid density
ν	Fluid kinematic viscosity

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 $p = p_A - p_B$.

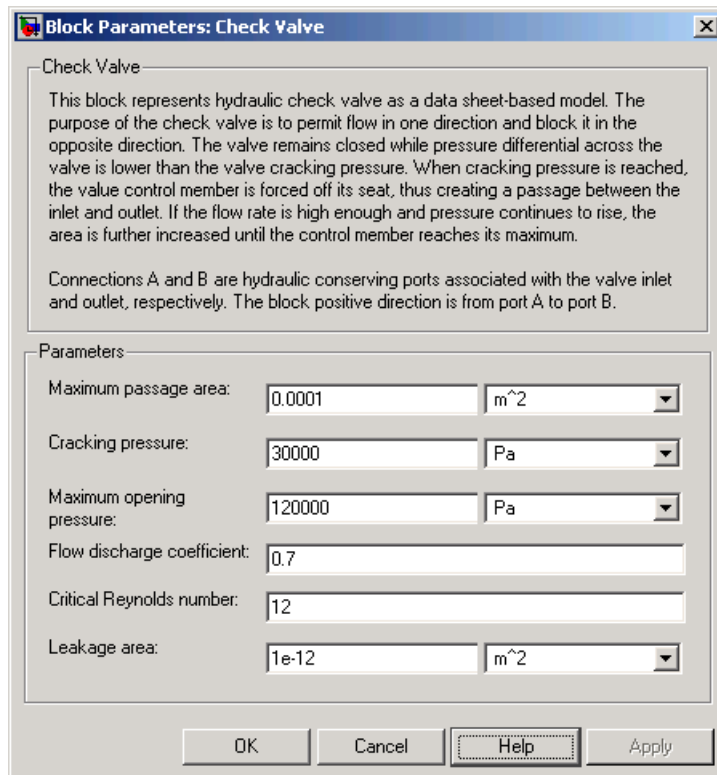
Check Valve

Basic Assumptions and Limitations

The model is based on the following assumptions:

- Valve opening is linearly proportional to the pressure differential.
- No loading on the valve, such as inertia, friction, spring, and so on, is considered.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.

Dialog Box and Parameters



Block Parameters: Check Valve

Check Valve

This block represents 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 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 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.

Connections A and B are hydraulic conserving ports associated with the valve inlet and outlet, respectively. The block positive direction is from port A to port B.

Parameters

Maximum passage area:	0.0001	m ²
Cracking pressure:	30000	Pa
Maximum opening pressure:	120000	Pa
Flow discharge coefficient:	0.7	
Critical Reynolds number:	12	
Leakage area:	1e-12	m ²

OK Cancel Help Apply

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.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

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. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is $1e-12 \text{ m}^2$.

Check Valve

Global Parameters

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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 Graetz Flow Control Circuit demo (`sh_Graetz_circuit`) 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

Purpose Simulate 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

Cylinder Friction

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:

$$F = K \cdot v$$

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

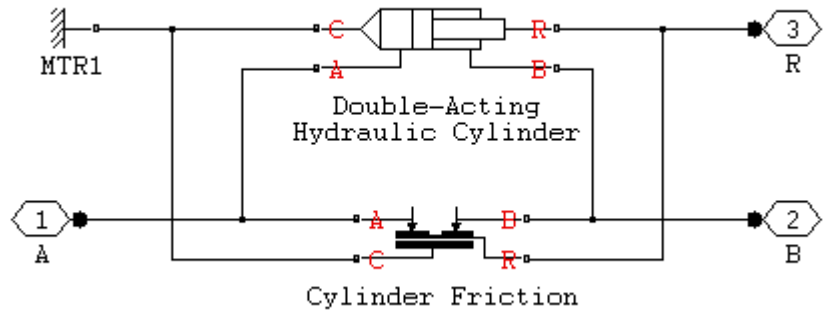
where

K Proportionality coefficient

v_{th} Velocity threshold

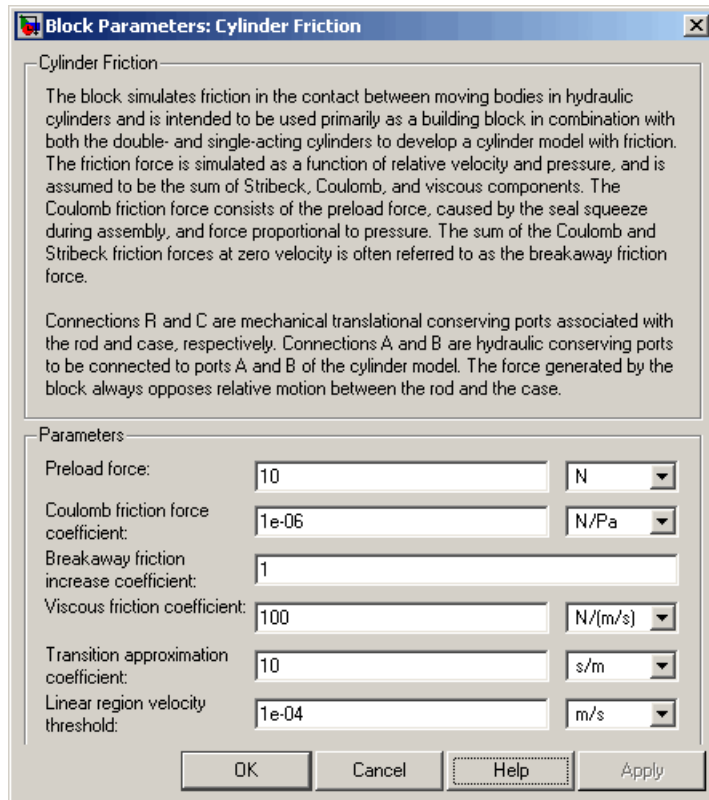
Connections R and C are mechanical translational conserving ports associated with the rod and case, respectively. Connections A and B are hydraulic conserving ports to be connected to ports A and B of the cylinder model, as shown in the following illustration. The force generated by the block always opposes relative motion between the rod and the case.

Cylinder Friction



Cylinder Friction

Dialog Box and Parameters



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. The 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.

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.

Cylinder Friction

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

Double-Acting Hydraulic Cylinder

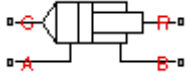
Purpose

Simulate hydraulic actuator exerting force in both directions

Library

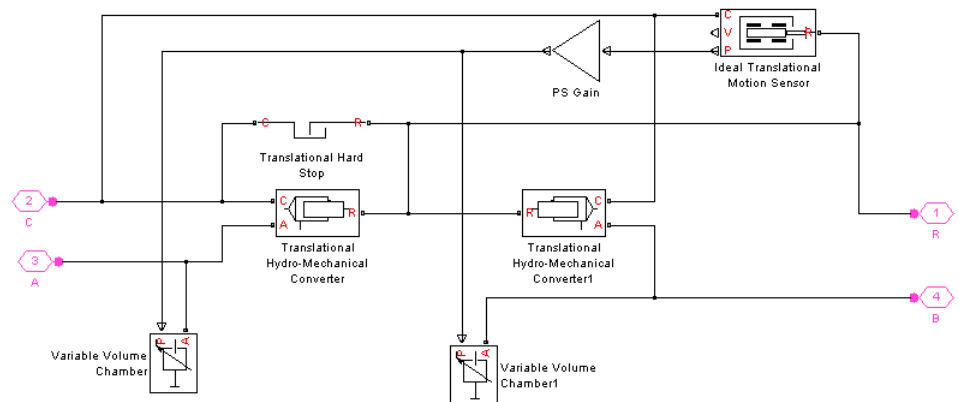
Hydraulic Cylinders

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 other SimHydraulics 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 chamber A and port B is connected to chamber B.

The energy through hydraulic port A or B is directed to the appropriate Translational Hydro-Mechanical Converter block and Variable Volume Chamber block. The converter transforms hydraulic energy into mechanical energy, while the chamber accounts for the fluid

Double-Acting Hydraulic Cylinder

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 Ideal Translational Motion Sensor block in the schematic is introduced to determine an instantaneous piston position, which is necessary for the Variable Volume Chamber blocks.

The block directionality is adjustable and can be controlled with the **Cylinder orientation** parameter.

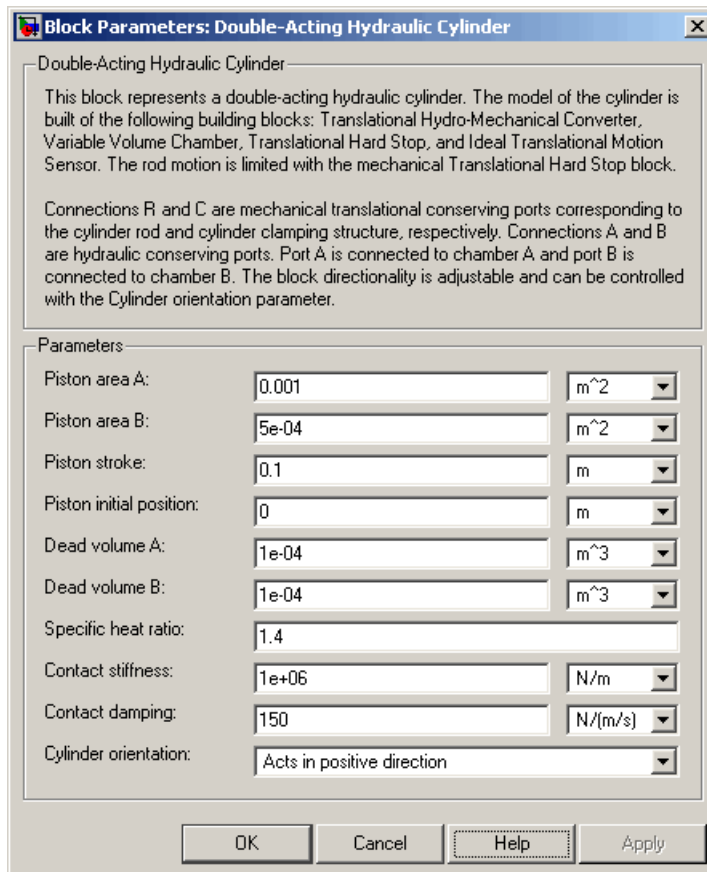
Basic Assumptions and Limitations

The model is based on the following assumptions:

- 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.

Double-Acting Hydraulic Cylinder

Dialog Box and Parameters



Block Parameters: Double-Acting Hydraulic Cylinder

Double-Acting Hydraulic Cylinder

This block represents a double-acting hydraulic cylinder. The model of the cylinder is built of the following building blocks: Translational Hydro-Mechanical Converter, Variable Volume Chamber, Translational Hard Stop, and Ideal Translational Motion Sensor. The rod motion is limited with the mechanical Translational Hard Stop block.

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. The block directionality is adjustable and can be controlled with the Cylinder orientation parameter.

Parameters

Piston area A:	0.001	m ²
Piston area B:	5e-04	m ²
Piston stroke:	0.1	m
Piston initial position:	0	m
Dead volume A:	1e-04	m ³
Dead volume B:	1e-04	m ³
Specific heat ratio:	1.4	
Contact stiffness:	1e+06	N/m
Contact damping:	150	N/(m/s)
Cylinder orientation:	Acts in positive direction	

OK Cancel Help Apply

Piston area A

Chamber A effective piston area. The default value is 0.001 m².

Piston area B

Chamber B effective piston area. The default value is 5e-5 m².

Piston stroke

Piston maximum travel between caps. The default value is 0.1 m.

Double-Acting Hydraulic Cylinder

Piston initial position

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.

Dead volume A

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

Dead volume B

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

Specific heat ratio

Gas-specific heat ratio for the Variable Volume Chamber blocks. 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 $1 \text{e}6 \text{ N/m}$.

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, The MathWorks recommends that you assign a nonzero value to this parameter. The default value is 150 N*s/m .

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

Fluid bulk modulus

The parameter is determined by the type of working fluid selected for the system under design. 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.

Double-Acting Hydraulic Cylinder

Examples

The Double-Acting Hydraulic Cylinder with Flexible Clamping demo (`sh_cylinder_da_flexible_clamping`) 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.

See Also

Single-Acting Hydraulic Cylinder
Ideal Translational Motion Sensor
Translational Hard Stop
Translational Hydro-Mechanical Converter
Variable Volume Chamber

Double-Acting Rotary Actuator

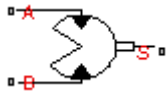
Purpose

Simulate double-acting hydraulic rotary actuator

Library

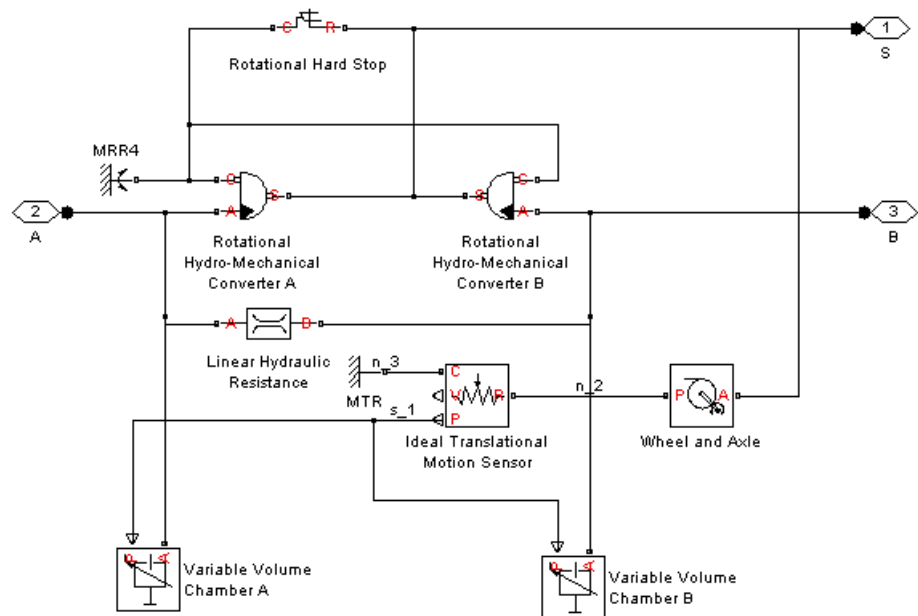
Hydraulic Cylinders

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.



Double-Acting Rotary Actuator

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.
Rotational Hydro-Mechanical Converter B	Converts hydraulics energy into mechanical rotational energy when fluid is pumped into actuator chamber B.
Rotational Hard Stop	Imposes limits on shaft rotation.
Linear Hydraulic Resistance	Accounts for leakages.
Variable Volume Chamber A	Accounts for fluid compressibility in actuator chamber A.
Variable Volume Chamber B	Accounts for fluid compressibility in actuator chamber B.
Ideal Translational Motion Sensor	Determines an instantaneous shaft position, which is necessary for the Variable Volume Chamber block.
Wheel and Axle	Converts shaft rotation into translational motion to provide input to the Ideal Translational Motion Sensor block

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.

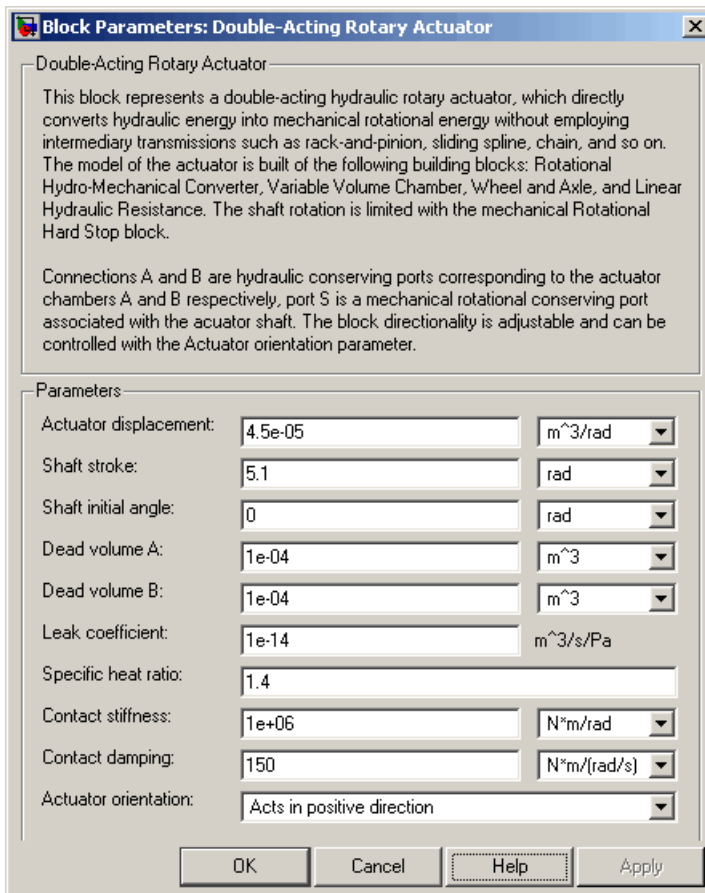
Double-Acting Rotary Actuator

Basic Assumptions and Limitations

The model is based on the following assumption:

- 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.

Dialog Box and Parameters



Block Parameters: Double-Acting Rotary Actuator

Double-Acting Rotary Actuator

This block represents 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. The model of the actuator is built of the following building blocks: Rotational Hydro-Mechanical Converter, Variable Volume Chamber, Wheel and Axle, and Linear Hydraulic Resistance. The shaft rotation is limited with the mechanical Rotational Hard Stop block.

Connections A and B are hydraulic conserving ports corresponding to the actuator chambers A and B respectively, port 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.

Parameters

Actuator displacement:	4.5e-05	m ³ /rad
Shaft stroke:	5.1	rad
Shaft initial angle:	0	rad
Dead volume A:	1e-04	m ³
Dead volume B:	1e-04	m ³
Leak coefficient:	1e-14	m ³ /s/Pa
Specific heat ratio:	1.4	
Contact stiffness:	1e+06	N*m/rad
Contact damping:	150	N*m/(rad/s)
Actuator orientation:	Acts in positive direction	

OK Cancel Help Apply

Double-Acting Rotary Actuator

Actuator displacement

Effective displacement of the actuator. The default value is $4.5e-5 \text{ m}^3/\text{rad}$.

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

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

Leak coefficient

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

Specific heat ratio

Gas-specific heat ratio for the Variable Volume Chamber block. The default value is 1.4.

Contact stiffness

Specifies the elastic property of colliding bodies for the Rotational Hard Stop block. The greater the value of the parameter, the less the bodies penetrate into each other, the more rigid the impact becomes. Lesser value of the parameter makes contact softer, but generally improves convergence and computational efficiency. The default value is $1e6 \text{ N}\cdot\text{m}/\text{rad}$.

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, The MathWorks recommends that you assign a nonzero value to this parameter. The default value is 150 N*m/(rad/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 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

Fluid bulk modulus

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

Double-Acting Rotary Actuator

- 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

- Ideal Translational Motion Sensor
- Linear Hydraulic Resistance
- Rotational Hard Stop
- Rotational Hydro-Mechanical Converter
- Variable Volume Chamber
- Wheel and Axle

Purpose Simulate 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], 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.

Elbow

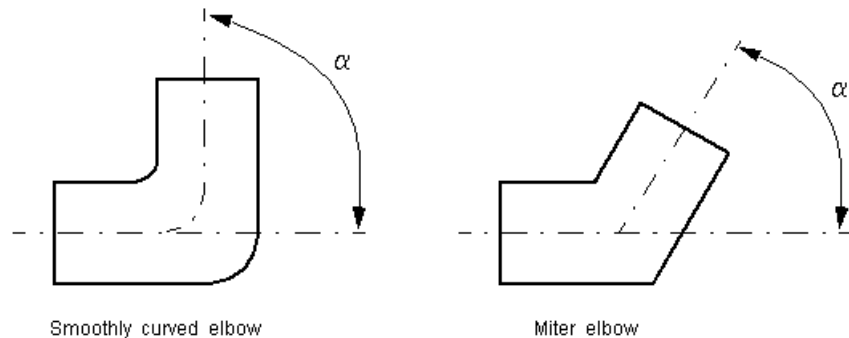
The core data for the pressure loss coefficient computation is the table-specified relationship between the friction factor f_T and the internal diameter for clean commercial steel pipes, with flow in the zone of complete turbulence (see [1], p. A-26). For smoothly curved, standard 90° elbows, the pressure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

The model is based on the following assumptions:

- Fluid inertia is not taken into account.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.
- The elbow is assumed to be made of a clean commercial steel pipe.

Elbow

Dialog Box and Parameters

Block Parameters: Elbow

Elbow

The block represents an elbow as a local hydraulic resistance. The pressure loss in the elbow is characterized by the table-specified pressure loss coefficient provided in the Crane Technical Paper. Two types of elbows are considered: smoothly curved standard elbows and sharp-edged, miter elbows. The block covers elbows in the 5-100 mm and 0-90 deg range.

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

Elbow internal diameter: m

Elbow angle: deg

Elbow type:

Critical Reynolds number:

OK Cancel Help Apply

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.

Elbow type

The parameter can have one of two values: Smoothly curved elbow or Miter bend. The default value is Smoothly curved elbow.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 80.

Restricted Parameters

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

- **Elbow type**

All other block parameters are available for modification.

Global Parameters

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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

Elbow

See Also

Gradual Area Change

Local Resistance

Pipe Bend

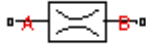
Sudden Area Change

T-junction

Purpose Simulate 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 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 flow rate is determined according to the following equations:

$$q = \begin{cases} C_D \cdot A \sqrt{\frac{2}{\rho} |p| \cdot \text{sign}(p)} & \text{for } Re \geq Re_{cr} \\ 2C_{DL} \cdot A \frac{D_H}{\nu \cdot \rho} p & \text{for } Re < Re_{cr} \end{cases}$$

$$p = p_A - p_B$$

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

$$C_{DL} = \left(\frac{C_D}{\sqrt{Re_{cr}}} \right)^2$$

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

where

q Flow rate

p Pressure differential

p_A, p_B Gauge pressures at the block terminals

Fixed Orifice

C_D	Flow discharge coefficient
A	Orifice passage area
D_H	Orifice hydraulic diameter
ρ	Fluid density
ν	Fluid kinematic viscosity

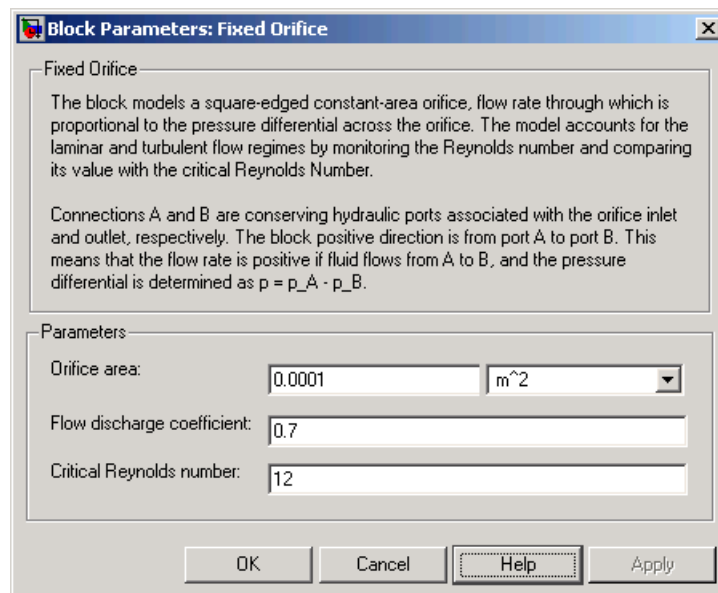
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as $p = p_A - p_B$.

Basic Assumptions and Limitations

The model is based on the following assumptions:

- Fluid inertia is not taken into account.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.

Dialog Box and Parameters



Orifice area

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

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.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12, which corresponds to a round orifice in thin material with sharp edges.

Fixed Orifice

Global Parameters

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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 Orifice

Orifice with Variable Area Round Holes

Orifice with Variable Area Slot

Variable Area Orifice

Variable Orifice

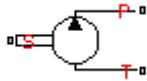
Purpose

Simulate fixed-displacement hydraulic pump

Library

Pumps and Motors

Description



The Fixed-Displacement Pump block represents a positive, fixed-displacement pump of any type as a data-sheet-based model. The key parameters required for this block are pump displacement, volumetric and total efficiencies, nominal pressure, and angular velocity. All these parameters are generally provided in the data sheets or catalogs. The fixed-displacement pump is represented with the following equations:

$$q = D \cdot \omega - k_{leak} \cdot p$$

$$T = D \cdot p / \eta_{mech}$$

$$k_{leak} = k_{HP} / v \cdot \rho$$

$$k_{HP} = \frac{D \cdot \omega_{nom} (1 - \eta_V) \cdot v_{nom} \cdot \rho}{P_{nom}}$$

$$p = p_P - p_T$$

where

q	Pump delivery
p	Pressure differential across the pump
p_P, p_T	Gauge pressures at the block terminals
T	Torque at the pump driving shaft
D	Pump displacement
ω	Pump angular velocity
k_{leak}	Leakage coefficient

Fixed-Displacement Pump

k_{HP}	Hagen-Poiseuille coefficient
η_V	Pump volumetric efficiency
η_{mech}	Pump mechanical efficiency
ν	Fluid kinematic viscosity
ρ	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

$$q_{leak} = D\omega_{nom}(1 - \eta_V)$$

which provides the formula to determine the Hagen-Poiseuille coefficient

$$k_{HP} = \frac{D\omega_{nom}(1 - \eta_V) \cdot \nu_{nom} \cdot \rho}{p_{nom}}$$

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 $p = p_P - p_T$.

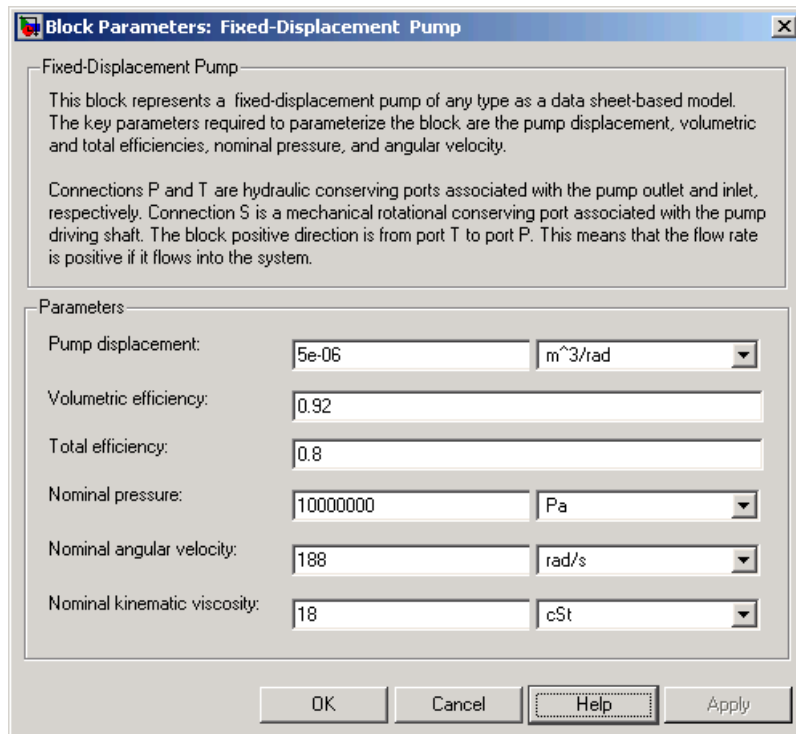
Basic Assumptions and Limitations

The model is based on the following assumptions:

- 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.

Fixed-Displacement Pump

Dialog Box and Parameters



Pump displacement

Pump displacement. The default value is $5e-6 \text{ m}^3/\text{rad}$.

Volumetric efficiency

Pump volumetric efficiency specified at nominal pressure, angular velocity, and fluid viscosity. The default value is 0.92.

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 0.8.

Nominal pressure

Pressure differential across the pump, at which both the volumetric and total efficiencies are specified. The default value is 1×10^7 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.

Global Parameters

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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 Power Unit with Fixed-Displacement Pump demo (sh_power_unit_fxd_dsp1_pump) contains a fixed-displacement pump, which is driven by a motor through a compliant transmission, a pressure-relief valve, and a variable orifice, which simulates system fluid consumption. The motor model is represented as an Ideal Angular Velocity Source block, which rotates the shaft at 188 rad/s at zero torque. The load on the shaft decreases the velocity with a slip coefficient of 1.2

Fixed-Displacement Pump

(rad/s)/Nm. The load on the driving shaft is measured with the torque sensor. The shaft between the motor and the pump is assumed to be compliant and simulated with rotational spring and damper.

The simulation starts with the variable orifice open, which results in a low system pressure and the maximum flow rate going to the system. The orifice starts closing at 0.5 s, and is closed completely at 3 s. The output pressure builds up until it reaches the pressure setting of the relief valve (75e5 Pa), and is maintained at this level by the valve. At 3 s, the variable orifice starts opening, thus returning the system to its initial state.

See Also

Centrifugal Pump

Variable-Displacement Pressure-Compensated Pump

Variable-Displacement Pump

Purpose Simulate 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 connected to a hydraulic system. The chambers are separated by a bladder, piston, or any kind of elastic diaphragm.

If the fluid pressure at the accumulator inlet becomes higher than the precharge pressure, fluid enters the accumulator chamber and compresses the gas, thus storing hydraulic energy. A drop in the fluid pressure at the inlet forces the stored fluid back into the system.

Normally, pressure in the gas chamber is equal to that of the fluid chamber. But if pressure at the accumulator inlet (p) drops below the accumulator's precharge value (p_{pr}), the gas chamber gets isolated from the system with the inlet valve. In this case, pressure in the gas chamber remains constant and equal to the precharge value, while pressure at the inlet depends on pressure in the system to which the accumulator is connected. If pressure at the inlet builds up to the precharge value or higher, the chambers start interacting again. The accumulator is described with the following equations:

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

$$V_F = \begin{cases} 0 & \text{for } p_{inl} \leq p_{pr} \\ V_A \cdot \left(1 - \left(\frac{p_{pr}}{p} \right)^{\frac{1}{k}} \right) & \text{for } p_{inl} > p_{pr} \end{cases}$$

where

Gas-Charged Accumulator

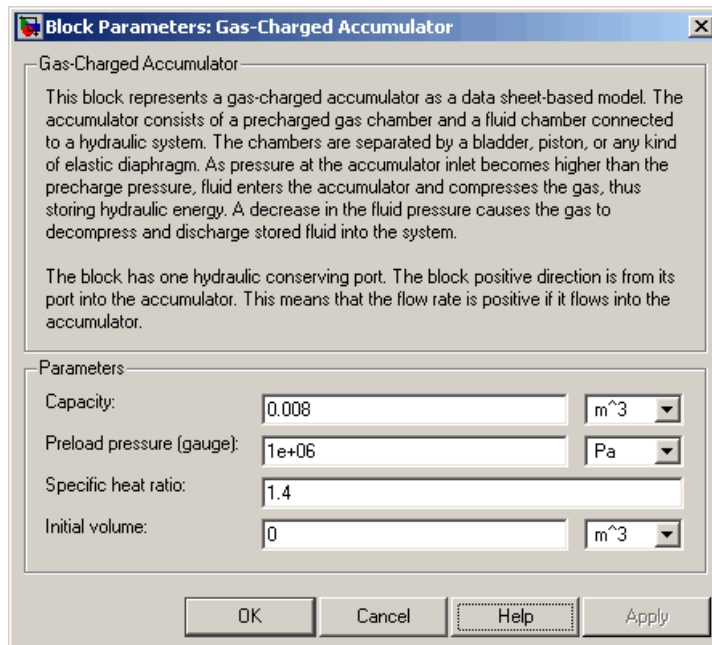
V_F	Fluid volume
V_A	Accumulator capacity
p	Inlet gauge pressure
p_{pr}	Precharge pressure
k	Specific heat ratio
q	Volumetric flow rate
t	Time

Basic Assumptions and Limitations

The model is based on the following assumptions:

- The gas compression is determined on the basis of the thermodynamics of ideal gases.
- The process is assumed to be polytropic.
- No loading on the separator, such as inertia, friction, and so on, is considered.
- Fluid compressibility is not taken into account.

Dialog Box and Parameters



Capacity

Accumulator capacity. The default value is 0.008 m³.

Preload pressure (gauge)

Precharge gauge pressure. The default value is 1e6 Pa.

Specific heat ratio

Specific heat ratio (adiabatic index). No units. The default value is 1.4. To account for heat exchange, you can set it within a range between 1 (isothermal process) and 1.4 (adiabatic process).

Initial volume

Initial volume of fluid in the accumulator. 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 "Computing Initial Conditions". The default value is 0.

Gas-Charged Accumulator

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.

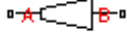
See Also

Spring-Loaded Accumulator

Purpose Simulate 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] and [2]):

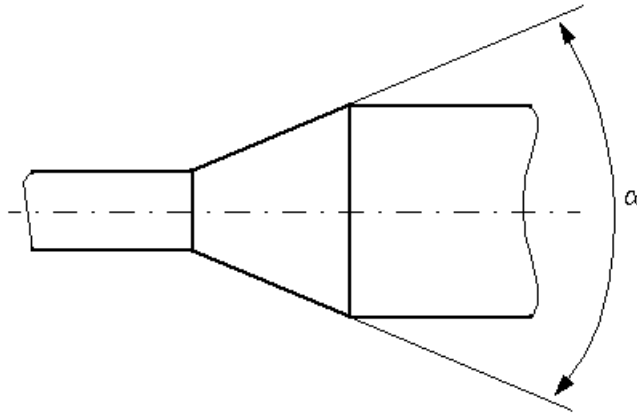
$$K_{GE} = \begin{cases} K_{cor} \left(1 - \frac{A_s}{A_L}\right)^2 \cdot 2.6 \sin \frac{\alpha}{2} & \text{for } 0 < \alpha \leq 45^\circ \\ K_{cor} \left(1 - \frac{A_s}{A_L}\right)^2 & \text{for } 45^\circ < \alpha < 180^\circ \end{cases}$$

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

where

Gradual Area Change

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 three 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 $p = p_A - p_B$.

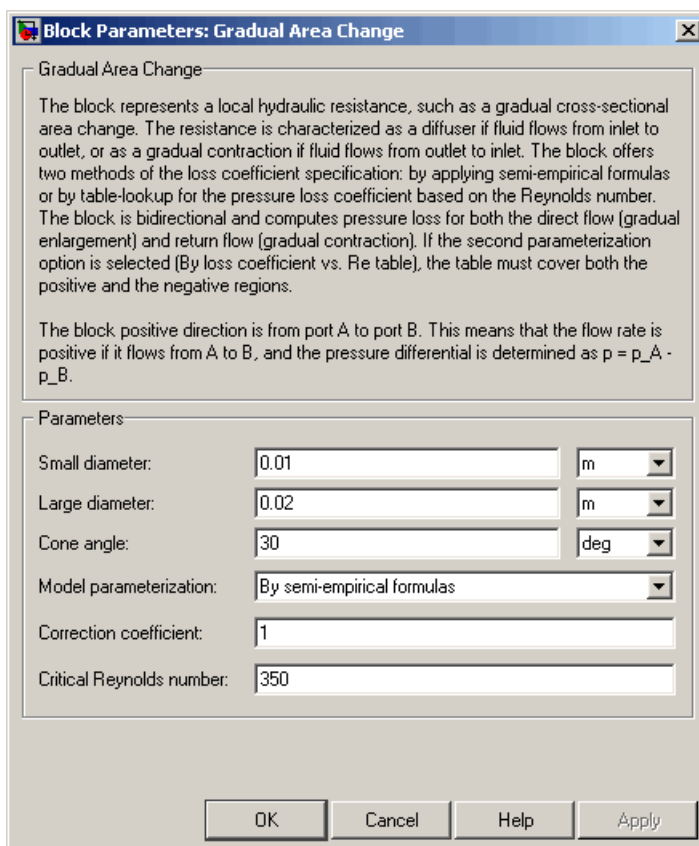
Basic Assumptions and Limitations

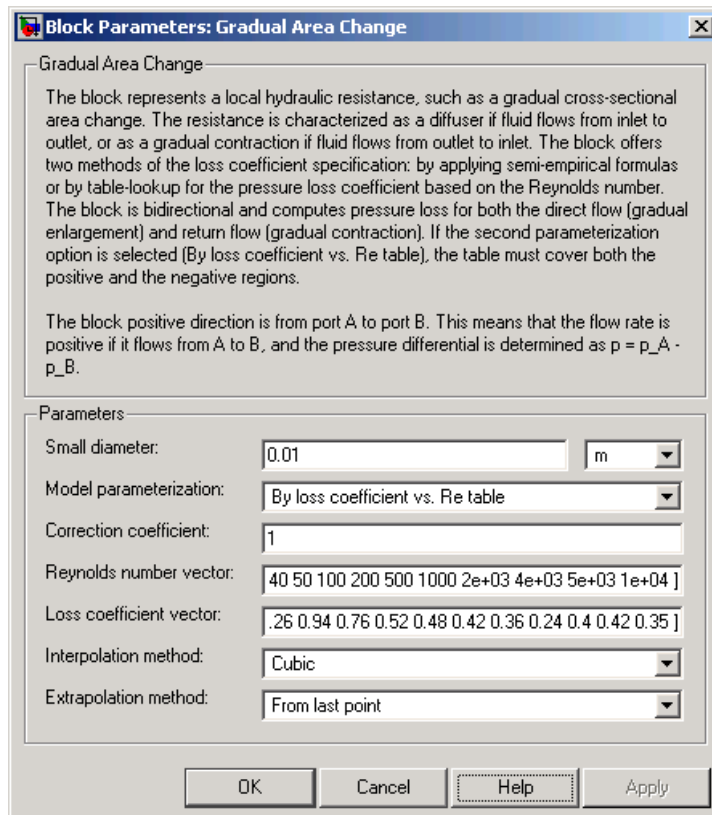
The model is based on the following assumption:

- Fluid inertia is not taken into account.
- If you select parameterization by semi-empirical formulas, the transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.
- If you select parameterization by the table-specified relationship $K=f(Re)$, the flow is assumed to be turbulent.

Gradual Area Change

Dialog Box and Parameters





Small diameter

Resistance small diameter. The default value is 0.01 m.

Large diameter

Resistance large diameter. The default value is 0.02 m. This parameter is used if **Model parameterization** is set to By semi-empirical formulas.

Gradual Area Change

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.

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.
- By loss coefficient vs. Re table — 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 three interpolation methods and two extrapolation methods. The table must cover both the positive and the negative flow regions.

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.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 350. This parameter is used if **Model parameterization** is set to By semi-empirical formulas.

Reynolds number vector

Specify the vector of input values for Reynolds numbers as a tabulated 1-by- m array. The input values vector must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. 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 output values for the loss coefficient as a tabulated 1-by-m array. The loss coefficient vector must be 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 — Uses a linear interpolation function.
- Cubic — Uses the Piecewise Cubic Hermite Interpolation Polinomial (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 **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:

- 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.

Gradual Area Change

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 **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

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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

Hydraulic Double-Acting Valve Actuator

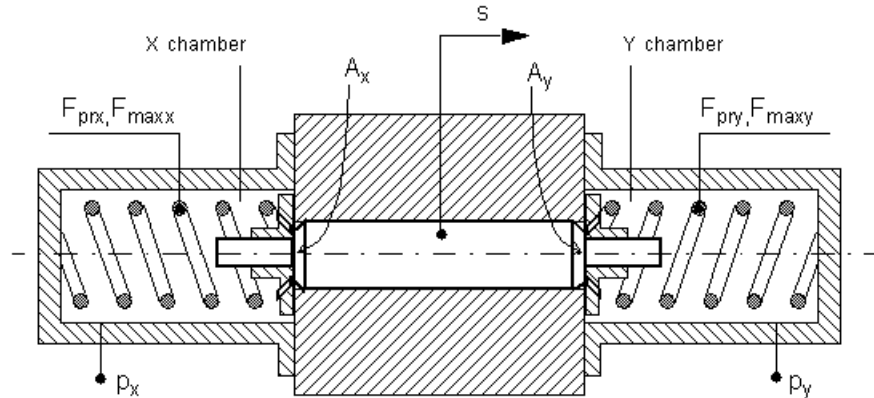
Purpose Simulate double-acting hydraulic valve actuator

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

Hydraulic Double-Acting Valve Actuator

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:

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

$$L_x = \frac{str_x}{F_{\max x} - F_{prx}}$$

$$L_y = \frac{str_y}{F_{\max y} - F_{pry}}$$

If $F \geq 0$,

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

If $F < 0$,

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

where

- F Force acting on the valve
- s Piston displacement
- p_x Pressure in the actuator X chamber
- p_y Pressure in the actuator Y chamber
- A_x Valve face area in the X chamber

Hydraulic Double-Acting Valve Actuator

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 model is based on the following assumptions:

- 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.

Hydraulic Double-Acting Valve Actuator

Dialog Box and Parameters

Block Parameters: Hydraulic Double-Acting Valve Actuator

Hydraulic Double-Acting Valve Actuator

This block represents a double-acting hydraulic valve actuator. Use it 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.

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.

Parameters

Piston area at port X:	<input type="text" value="2e-04"/>	<input type="text" value="m^2"/>
Piston area at port Y:	<input type="text" value="2e-04"/>	<input type="text" value="m^2"/>
Preload force at port X:	<input type="text" value="0"/>	<input type="text" value="N"/>
Preload force at port Y:	<input type="text" value="0"/>	<input type="text" value="N"/>
Spring maximum force at port X:	<input type="text" value="50"/>	<input type="text" value="N"/>
Spring maximum force at port Y:	<input type="text" value="50"/>	<input type="text" value="N"/>
Piston stroke at port X:	<input type="text" value="0.005"/>	<input type="text" value="m"/>
Piston stroke at port Y:	<input type="text" value="0.005"/>	<input type="text" value="m"/>
Actuator orientation:	<input type="text" value="Acts in positive direction"/>	

OK Cancel Help Apply

Hydraulic Double-Acting Valve Actuator

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.

Hydraulic Double-Acting Valve Actuator

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

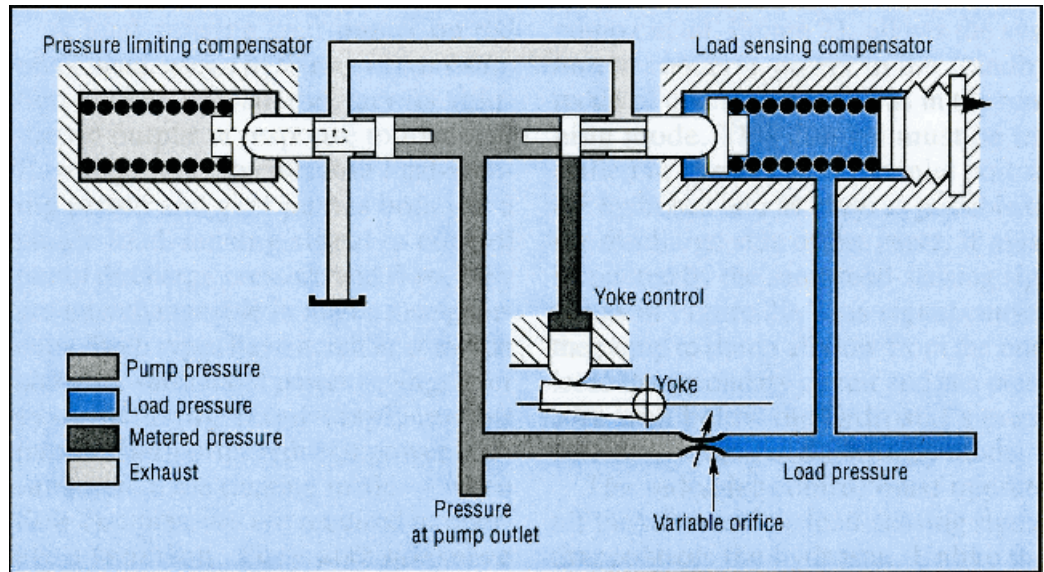
The block has the following ports:

- 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

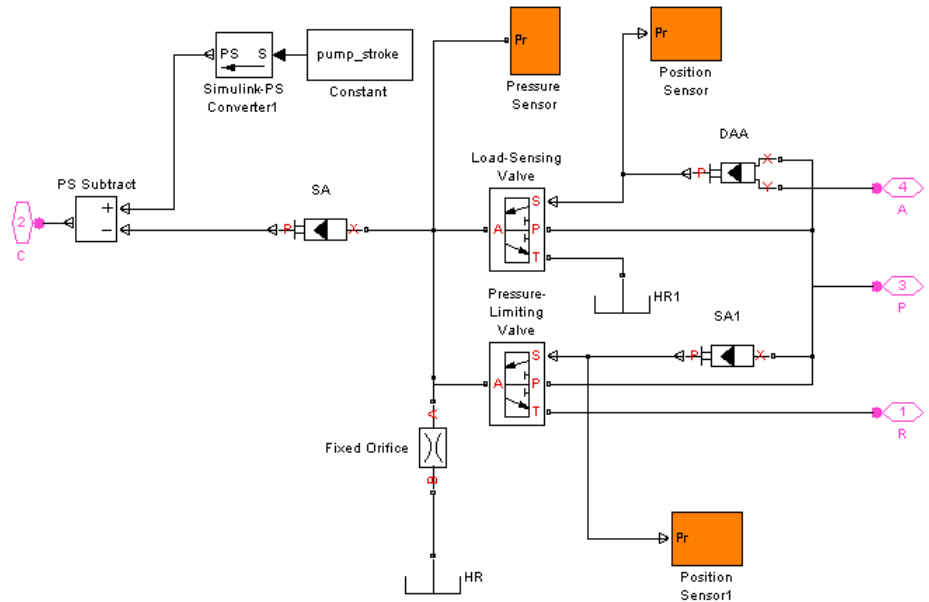
The following illustration shows a typical control unit of a variable-displacement pump that provides load sensing and pressure limiting (see [1]). In the unit, the load-sensing compensator varies the pump displacement to maintain a preset pressure differential across the variable orifice, while the pressure-limiting compensator does not allow the pump pressure to exceed the pressure limit.

Hydraulic Double-Acting Valve Actuator



The Hydraulic Actuator with Load-Sensing Variable-Displacement Pump demo (`sh_hydraulic_actuator_load_sensing_pump`) implements this type of control. The next illustration shows the schematic of the Load-Sensing and Pressure-Limiting Control block in the demo.

Hydraulic Double-Acting Valve Actuator



There are three hydraulic valve actuators in the model:

- SA1 — A single-acting actuator that controls the Pressure-Limiting Valve.
- SA — A single-acting valve actuator that acts on the pump displacement control device (yoke control).
- DAA — 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 demo model to see the parameter settings for the blocks.

References

- [1] F. Yeaple, *Fluid Power Design Handbook*, Marcel Dekker, Inc., 1995

Hydraulic Double-Acting Valve Actuator

See Also

2-Position Valve Actuator

3-Position Valve Actuator

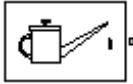
Hydraulic Single-Acting Valve Actuator

Proportional and Servo-Valve Actuator

Purpose Set working fluid properties 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 Chamber or Variable Volume 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:

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

where

E_l Pure liquid bulk modulus

p_a Atmospheric pressure

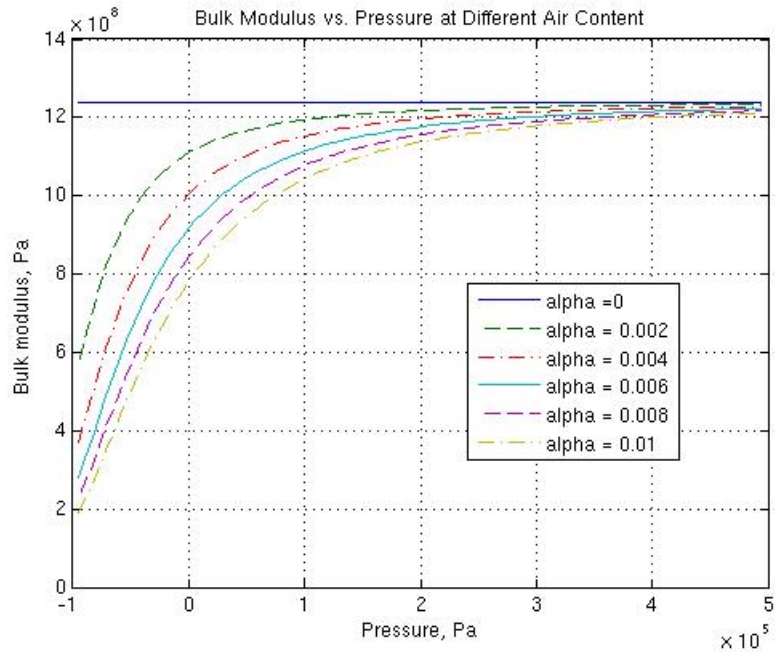
α Relative gas content at atmospheric pressure, $\alpha = V_g/V_L$

V_g Gas volume at atmospheric pressure

Hydraulic Fluid

- 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, etc., and as such cannot be accurately simulated in SimHydraulics. 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.

If it is known that cavitation is unlikely in the system under design, you can set the relative gas content in the fluid properties to zero, thus increasing the speed of computations.

The Hydraulic Fluid block offers a selection of predefined fluids. See “Examples” on page 2-133 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, SimHydraulics 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 a Hydraulic Fluid block or Custom Hydraulic Fluid block to be connected to it. Therefore, there must be as many Hydraulic Fluid blocks (or Custom Hydraulic Fluid blocks) as there are loops in the system.

Note 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 Skydrol LD-4 at 75°C and with a 0.005 ratio of entrapped air.

Hydraulic Fluid

Dialog Box and Parameters

Block Parameters: Hydraulic Fluid

Hydraulic Fluid
Set working fluid properties

Parameters

Hydraulic fluid: Skydrol LD-4

Relative amount of trapped air: 0.005

System temperature (C): 75

Fluid Properties

Density: 9.501537e+002 kg/m³

Viscosity: 5.501634e-006 cSt

Bulk modulus: 1.139178e+009 Pa

OK Cancel Help Apply

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 32 (ESSO UNIVIS N 32)
- Gasoline
- Diesel fuel
- Jet fuel
- Water-Glycol 60/40

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. The default value is 0.005.

System temperature

Fluid temperature (C). The default value is 75.

Restricted Parameters

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

- **Hydraulic fluid**

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:

Hydraulic Fluid

```
props = sh_stockfluidproperties
```

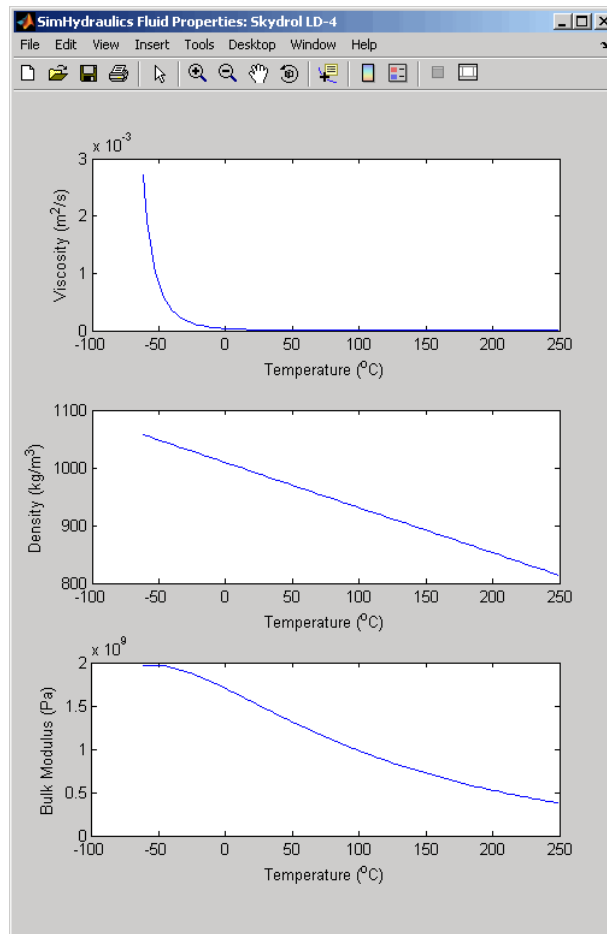
The system responds with a list of available fluids:

```
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

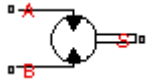
Custom Hydraulic Fluid

Hydraulic Motor

Purpose Simulate fixed-displacement hydraulic motor

Library Pumps and Motors

Description



The Hydraulic Motor block represents a positive, fixed-displacement hydraulic motor of any type as a data-sheet-based model. The key parameters required to parameterize the block are motor displacement, volumetric and total efficiencies, nominal pressure, and angular velocity. All these parameters are generally provided in the data sheets or catalogs. The motor is represented with the following equations:

$$q = D \cdot \omega + k_{leak} \cdot p$$

$$T = D \cdot p \cdot \eta_{mech}$$

$$k_{leak} = k_{HP} / v \cdot \rho$$

$$k_{HP} = \frac{D \cdot \omega_{nom} (1 - \eta_V) \cdot v_{nom} \cdot \rho}{P_{nom}}$$

$$p = p_A - p_B$$

where

- q Flow rate through the motor
- p Pressure differential across the motor
- p_A, p_B Gauge pressures at the block terminals
- T Torque at the motor output shaft
- D Motor displacement
- ω Output shaft angular velocity
- k_{leak} Leakage coefficient

k_{HP}	Hagen-Poiseuille coefficient
η_V	Motor volumetric efficiency
η_{mech}	Motor mechanical efficiency
ν	Fluid kinematic viscosity
ρ	Fluid density
p_{nom}	Motor nominal pressure
ω_{nom}	Motor 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 motor 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

$$q_{leak} = D\omega_{nom}(1 - \eta_V)$$

which provides the formula to determine the Hagen-Poiseuille coefficient

$$k_{HP} = \frac{D\omega_{nom}(1 - \eta_V) \cdot \nu_{nom} \cdot \rho}{p_{nom}}$$

Hydraulic Motor

The motor mechanical efficiency is not usually available in data sheets, therefore it is determined from the total and volumetric efficiency by assuming that the hydraulic efficiency is negligibly small

$$\eta_{mech} = \eta_{total} / \eta_V$$

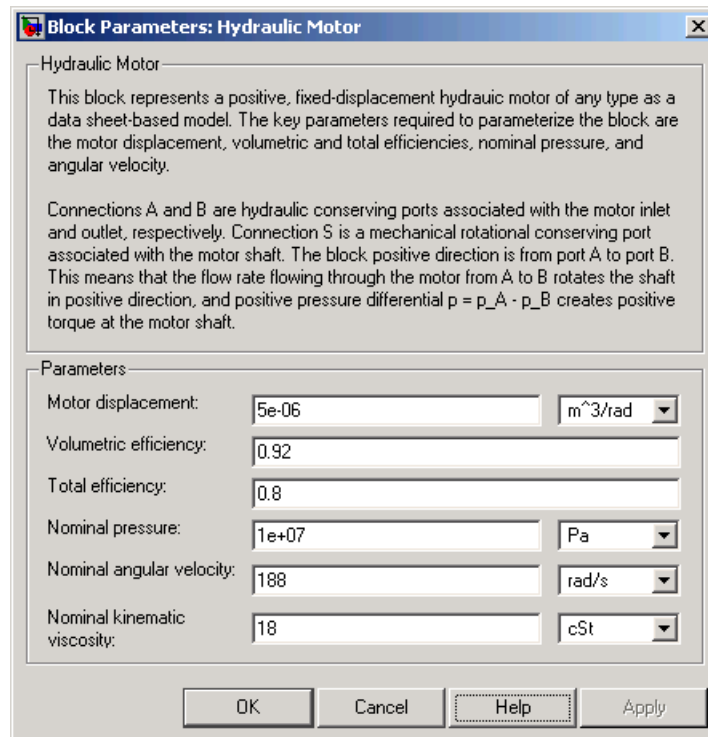
The block hydraulic positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and rotates the output shaft in the globally assigned positive direction. The pressure differential across the motor is determined as $p = p_A - p_B$, and positive pressure differential accelerates the shaft in the positive direction.

Basic Assumptions and Limitations

The model is based on the following assumptions:

- Fluid compressibility is neglected.
- No loading on the motor shaft, such as inertia, friction, spring, and so on, is considered.
- Leakage inside the motor is assumed to be linearly proportional to its pressure differential.

Dialog Box and Parameters



Motor displacement

Motor displacement. The default value is $5e-6 \text{ m}^3/\text{rad}$.

Volumetric efficiency

Motor volumetric efficiency specified at nominal pressure, angular velocity, and fluid viscosity. The default value is 0.92.

Total efficiency

Motor total efficiency, which is determined as a ratio between the mechanical power at the output shaft and hydraulic power at the motor inlet at nominal pressure, angular velocity, and fluid viscosity. The default value is 0.8.

Hydraulic Motor

Nominal pressure

Pressure differential across the motor, at which both the volumetric and total efficiencies are specified. The default value is 1×10^7 Pa.

Nominal angular velocity

Angular velocity of the output 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.

Global Parameters

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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 motor inlet.

B

Hydraulic conserving port associated with the motor outlet.

S

Mechanical rotational conserving port associated with the motor output shaft.

See Also

Variable-Displacement Motor

Purpose Simulate hydraulic pipeline with resistive and fluid compressibility properties

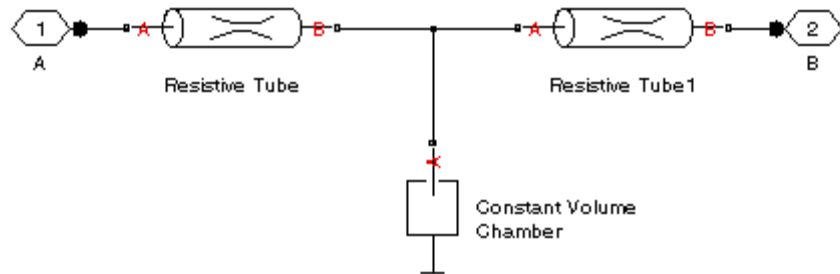
Library Pipelines

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 SimHydraulics building blocks and its schematic diagram is shown below.



The Resistive Tube blocks account for friction losses, while the Constant Volume Chamber block accounts for fluid compressibility. By using the block parameters, you can set the model to simulate pipeline with rigid or compliant 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 $p = p_A - p_B$.

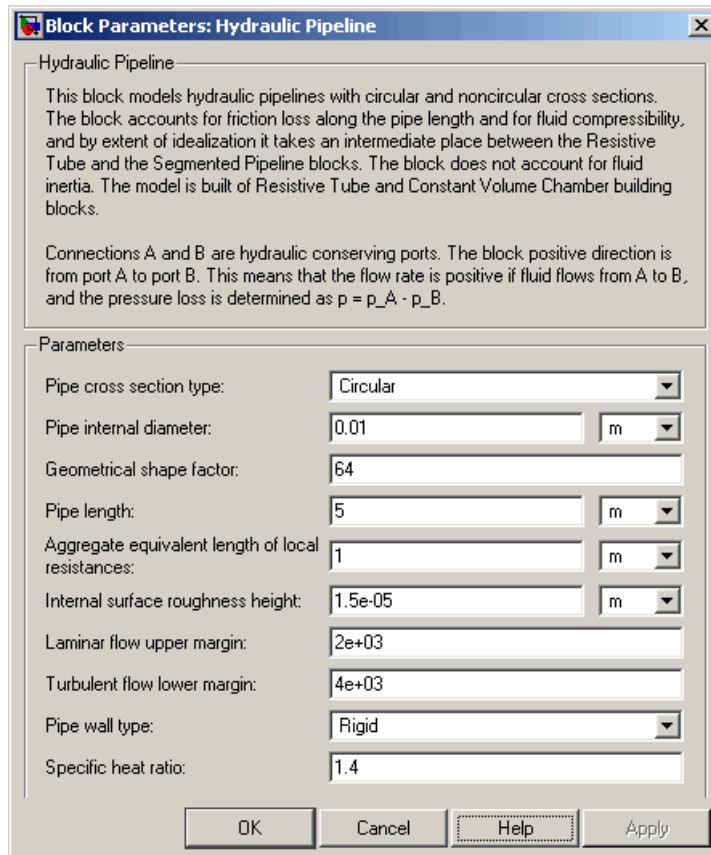
Hydraulic Pipeline

Basic Assumptions and Limitations

Dialog Box and Parameters

The model is based on the following assumptions:

- Flow is assumed to be fully developed along the pipe length.
- Fluid inertia is not taken into account.



Block Parameters: Hydraulic Pipeline

Hydraulic Pipeline

This block models hydraulic pipelines with circular and noncircular cross sections. The block accounts for friction loss along the pipe length and for fluid compressibility, and by extent of idealization it takes an intermediate place between the Resistive Tube and the Segmented Pipeline blocks. The block does not account for fluid inertia. The model is built of Resistive Tube and Constant Volume Chamber building blocks.

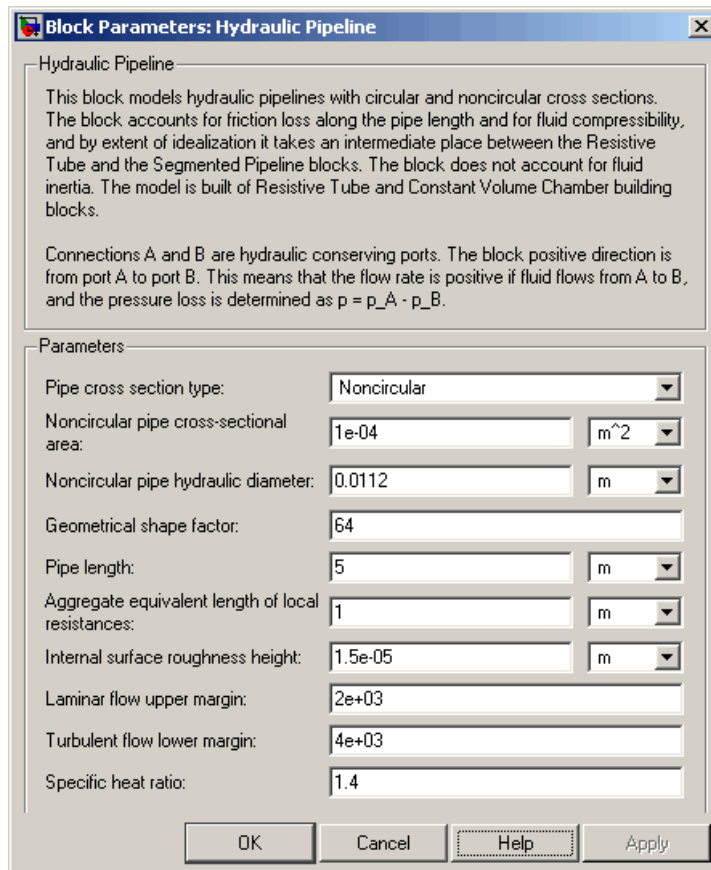
Connections A and B are hydraulic conserving ports. The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

Parameters

Pipe cross section type:	Circular
Pipe internal diameter:	0.01 m
Geometrical shape factor:	64
Pipe length:	5 m
Aggregate equivalent length of local resistances:	1 m
Internal surface roughness height:	1.5e-05 m
Laminar flow upper margin:	2e+03
Turbulent flow lower margin:	4e+03
Pipe wall type:	Flexible
Static pressure-diameter coefficient:	2e-12 m/Pa
Viscoelastic process time constant:	0.01 s
Specific heat ratio:	1.4

OK Cancel Help Apply

Hydraulic Pipeline



Pipe cross section type

The parameter can have one of two values: Circular or Noncircular. For a circular pipe, you need to specify its internal diameter. For a noncircular pipe, you need to 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 $1e-4$ m².

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

The parameter is used for computing friction factor at laminar flow and depends of the shape of the pipe cross section. For a pipe with noncircular cross section, you must 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 (see [1]). 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.

Pipe wall type

The parameter is available only for circular pipes and can have one of two values: **Rigid** or **Flexible**. If the parameter is set to **Rigid**, wall compliance is not taken into account, which can improve computational efficiency. The value **Flexible** is recommended for hoses and metal pipes where wall compliance can affect the system behavior. The default value is **Rigid**.

Static pressure-diameter coefficient

Coefficient that establishes relationship between the pressure and the internal diameter at steady-state conditions. This coefficient can be determined analytically for cylindrical metal pipes or experimentally for hoses. The parameter is used if the **Pipe wall type** parameter is set to **Flexible**. The default value is $2e-10$ m/Pa.

Viscoelastic process time constant

Time constant in the transfer function that relates pipe internal diameter to pressure variations. By using this parameter, the simulated elastic or viscoelastic process is approximated with the first-order lag. The value is determined experimentally or provided by the manufacturer. The parameter is used if the **Pipe wall type** parameter is set to **Flexible**. The default value is 0.008 s.

Specific heat ratio

Gas-specific heat ratio for the Constant Volume Chamber block.
The default value is 1.4.

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

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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

Hydraulic Pipeline

See Also

Linear Hydraulic Resistance

Resistive Tube

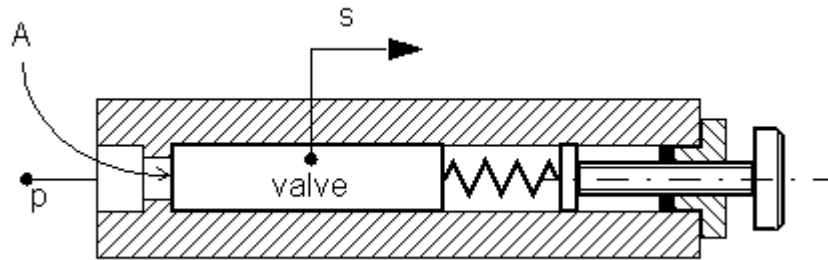
Segmented Pipeline

Hydraulic Single-Acting Valve Actuator

Purpose Simulate 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:

$$F = p \cdot A$$

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

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

Hydraulic Single-Acting Valve Actuator

where

p	Pressure applied to the piston
s	Piston displacement
A	Piston area
F	Instantaneous spring force
F_{pr}	Spring preload force
F_{max}	Spring force at maximum piston displacement
$stroke$	Piston stroke
or	Actuator orientation with respect to the globally assigned positive direction. If pressure applied at port X moves the piston in positive direction, or equals 1. If pressure applied at port X moves the piston in negative direction, or equals -1 .

Connection X is a hydraulic conserving port associated with the valve chamber. Connection P is a physical signal port whose output corresponds to piston displacement. Pressure applied at port X moves the piston in the positive or negative direction, depending on the value of the **Actuator orientation** parameter.

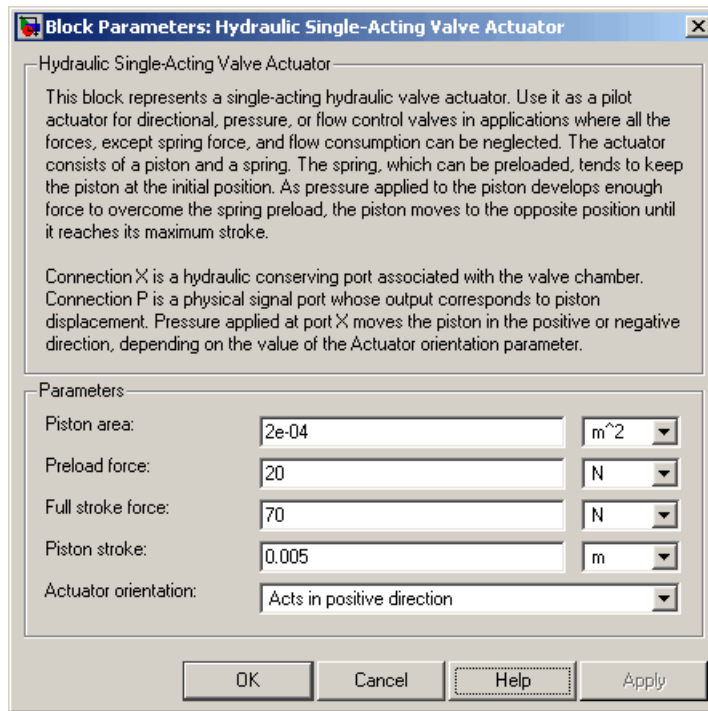
Basic Assumptions and Limitations

The model is based on the following assumptions:

- 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.

Hydraulic Single-Acting Valve Actuator

Dialog Box and 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

Hydraulic Single-Acting Valve Actuator

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.

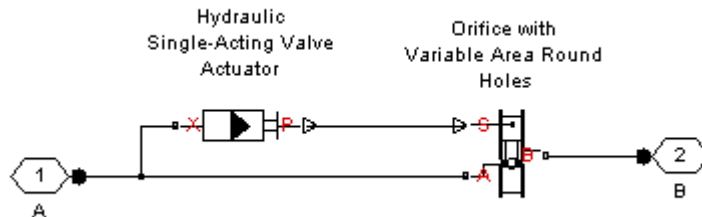
Ports

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.



Hydraulic Single-Acting Valve Actuator

See Also

2-Position Valve Actuator

3-Position Valve Actuator

Hydraulic Double-Acting Valve Actuator

Proportional and Servo-Valve Actuator

Local Resistance

Purpose Simulate all kinds of hydraulic resistances 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 loss is determined according to the following equations:

$$p = \begin{cases} K \frac{\rho}{2A^2} q |q| & \text{for } Re > Re_{cr} \\ K \cdot Re_{cr} \frac{v \cdot \rho}{2D_H \cdot A} q & \text{for } Re \leq Re_{cr} \end{cases}$$

$$p = p_A - p_B$$

$$K = \begin{cases} \text{const} \\ K(Re) \end{cases}$$

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

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

where

q Flow rate

ρ Pressure loss

p_A, p_B	Gauge pressures at the block terminals
K	Pressure loss coefficient, which can be specified either as a constant, or as a table-specified function of the Reynolds number
Re	Reynolds number
Re_{cr}	Reynolds number of the transition from laminar to turbulent flow
D_H	Orifice hydraulic diameter
A	Passage area
ρ	Fluid density
ν	Fluid kinematic viscosity

Two block parameterization options are available:

- 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 table-specified $K=f(Re)$ relationship — The pressure loss coefficient is specified as function of the Reynolds number. The flow regime is assumed to be turbulent all the time. It is your responsibility to provide the appropriate values in the $K=f(Re)$ table to ensure turbulent flow.

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.

Local Resistance

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

The model is based on the following assumption:

- Fluid inertia is not taken into account.
- If you select parameterization by semi-empirical formulas, the transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.
- If you select parameterization by the table-specified relationship $K=f(Re)$, the flow is assumed to be completely turbulent.

Dialog Box and Parameters

Block Parameters: Local Resistance

Local Resistance

The block represents a local hydraulic resistance such as a bend, elbow, fitting, filter, local change in flow cross-section, etc. The pressure loss caused by resistance is computed with the semi-empirical formula based on pressure loss coefficient, which is usually provided in catalogs, data sheets, or hydraulic textbooks. The resistance can also be specified by a table, in which the loss coefficient is tabulated vs. Reynolds number. The resistance can be symmetrical or asymmetrical. In asymmetrical resistances, the pressure loss coefficients are different for the direct and reverse flows. If the loss coefficient is specified by a table, the table must cover both the positive and the negative flow regions.

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

Resistance area:

Model parameterization:

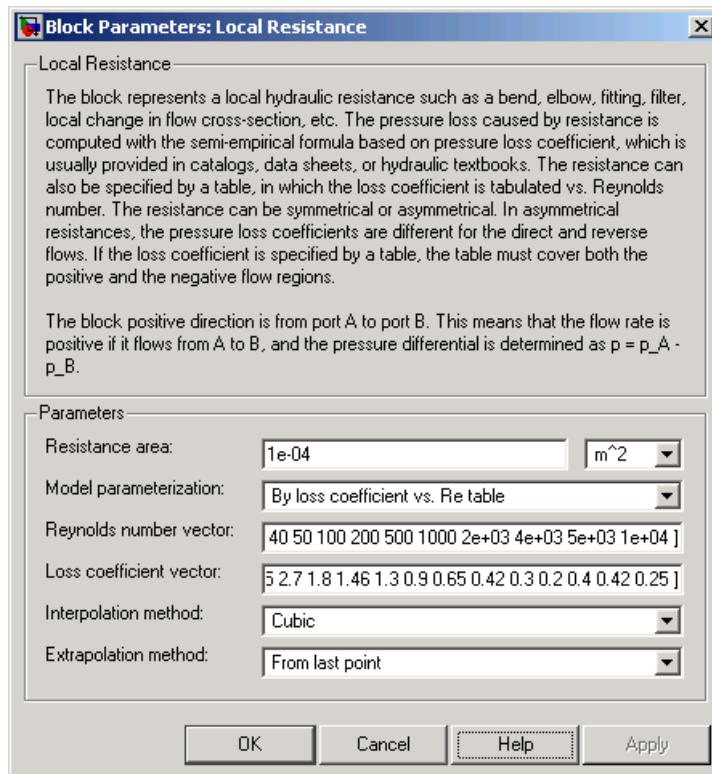
Pressure loss coefficient for direct flow:

Pressure loss coefficient for reverse flow:

Critical Reynolds number:

OK Cancel Help Apply

Local Resistance



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 three 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]). 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**.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 150. This parameter is used if **Model parameterization** is set to **By semi-empirical formulas**.

Reynolds number vector

Specify the vector of input values for Reynolds numbers as a tabulated 1-by-m array. The input values vector must be strictly monotonically increasing. The values can be non-uniformly spaced. You must provide at least three values. The default values are [-4000, -3000, -2000, -1000, -500, -200, -100, -50, -40, -30, -20, -15, -10, 10, 20, 30, 40, 50, 100, 200,

Local Resistance

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 output values for the loss coefficient as a tabulated 1-by-m array. The loss coefficient vector must be 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 — 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 **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:

- 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 **Model parameterization** is set to By loss coefficient vs. Re table.

Global Parameters

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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

Local Resistance

See Also

Elbow

Gradual Area Change

Pipe Bend

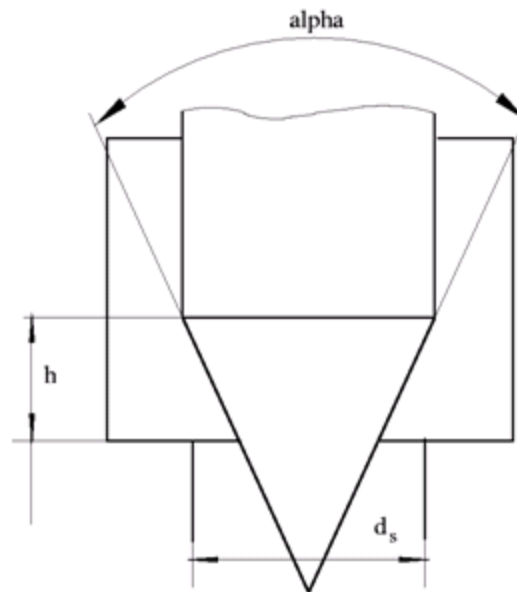
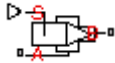
Sudden Area Change

T-junction

Purpose Simulate 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 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 flow rate is determined according to the following equations:

Needle Valve

$$q = \begin{cases} C_D \cdot A \sqrt{\frac{2}{\rho} |p| \cdot \text{sign}(p)} & \text{for } Re \geq Re_{cr} \\ 2C_{DL} \cdot A \frac{D_H}{v \cdot \rho} p & \text{for } Re < Re_{cr} \end{cases}$$

$$h = x_0 + x$$

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

$$p = p_A - p_B$$

$$Re = \frac{q \cdot D_H}{A(h) \cdot v}$$

$$C_{DL} = \left(\frac{C_D}{\sqrt{Re_{cr}}} \right)^2$$

$$D_H = \sqrt{\frac{4A(h)}{\pi}}$$

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

where

q Flow rate

ρ 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
d_s	Orifice diameter
α	Needle angle
ρ	Fluid density
D_H	Valve instantaneous hydraulic diameter
ν	Fluid kinematic viscosity
A_{leak}	Closed valve leakage area
A_{max}	Maximum valve open area

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.

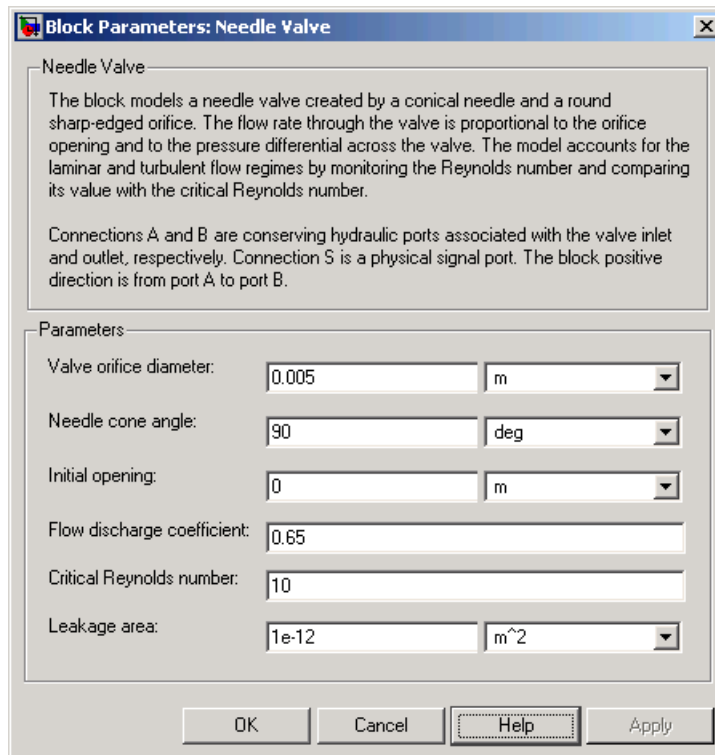
Basic Assumptions and Limitations

The model is based on the following assumptions:

- Fluid inertia is not taken into account.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.
- The flow passage area is assumed to be equal to the frustum side surface area.

Needle Valve

Dialog Box and 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.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 10.

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. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is $1e-12 \text{ m}^2$.

Global Parameters

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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.

Needle Valve

- B Hydraulic conserving port associated with the valve outlet.
- S Physical signal port to control spool displacement.

See Also

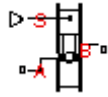
- Ball Valve
- Poppet Valve
- Pressure-Compensated Flow Control Valve

Orifice with Variable Area Round Holes

Purpose Simulate hydraulic variable orifice shaped as set of round holes drilled in sleeve

Library Orifices

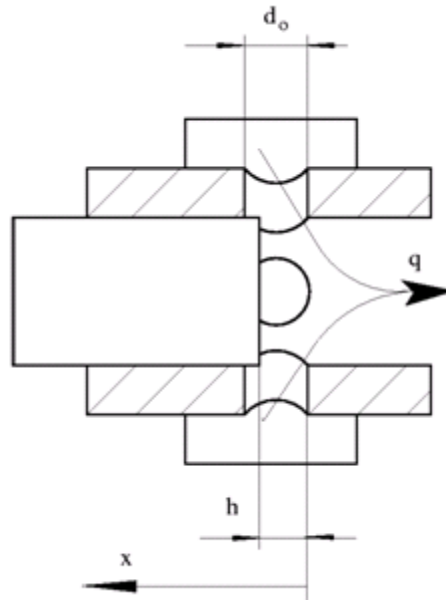
Description



The block models a variable orifice created by a cylindrical spool and a set of round holes drilled in the sleeve. All the holes are of the same diameter, evenly spread along the sleeve perimeter, and their center lines are located in the same plane. The flow rate through the orifice is proportional to the orifice opening and to the pressure differential across the orifice. The following schematic shows the cross section of an orifice with variable round holes, where

- q Flow rate
- h Orifice opening
- x Spool displacement from initial position
- d_0 Orifice hole diameter

Orifice with Variable Area Round Holes



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 flow rate is determined according to the following equations:

$$q = \begin{cases} C_D \cdot A \sqrt{\frac{2}{\rho} |p| \cdot \text{sign}(p)} & \text{for } Re \geq Re_{cr} \\ 2C_{DL} \cdot A \frac{D_H}{\nu \cdot \rho} p & \text{for } Re < Re_{cr} \end{cases}$$

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

Orifice with Variable Area Round Holes

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

$$p = p_A - p_B$$

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

$$C_{DL} = \left(\frac{C_D}{\sqrt{Re_{cr}}} \right)^2$$

$$D_H = \sqrt{\frac{4A(h)}{\pi}}$$

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

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
- d_0 Hole diameter
- z Number of holes
- x_0 Initial opening

Orifice with Variable Area Round Holes

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
D_H	Instantaneous orifice hydraulic diameter
ν	Fluid kinematic viscosity
A_{leak}	Closed orifice leakage area
A_{max}	Fully open orifice passage area

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

The model is based on the following assumptions:

- Fluid inertia is not taken into account.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.

Orifice with Variable Area Round Holes

Dialog Box and Parameters

Block Parameters: Orifice with Variable Area Round Holes

Orifice with Variable Area Round Holes

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.

Connections A and B are conserving hydraulic ports associated with the orifice inlet and outlet, respectively. Connection S is a physical signal port. The block positive direction is from port A to port B. Positive signal at port S opens or closes the orifice, depending on the value of the Orifice orientation parameter.

Parameters

Diameter of round holes: m

Number of round holes:

Flow discharge coefficient:

Initial opening: m

Orifice orientation:

Critical Reynolds number:

Leakage area: m²

OK Cancel Help Apply

Diameter of round holes

Diameter of the orifice holes. The default value is 5×10^{-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.

Orifice with Variable Area Round Holes

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`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 10.

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. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 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 with Variable Area Round Holes

- **Orifice orientation**

All other block parameters are available for modification.

Global Parameters

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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 Orifice

Fixed Orifice

Orifice with Variable Area Slot

Variable Area Orifice

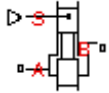
Variable Orifice

Orifice with Variable Area Slot

Purpose Simulate 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 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 flow rate is determined according to the following equations:

$$q = \begin{cases} C_D \cdot A \sqrt{\frac{2}{\rho} |p| \cdot \text{sign}(p)} & \text{for } Re \geq Re_{cr} \\ 2C_{DL} \cdot A \frac{D_H}{\nu \cdot \rho} p & \text{for } Re < Re_{cr} \end{cases}$$

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

$$A(h) = \begin{cases} b \cdot h + A_{leak} & \text{for } h > 0 \\ A_{leak} & \text{for } h \leq 0 \end{cases}$$

$$p = p_A - p_B$$

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

$$C_{DL} = \left(\frac{C_D}{\sqrt{Re_{cr}}} \right)^2$$

$$D_H = \sqrt{\frac{4A(h)}{\pi}}$$

Orifice with Variable Area Slot

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
D_H	Instantaneous orifice hydraulic diameter
ν	Fluid kinematic viscosity
A_{leak}	Closed orifice leakage area

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**.

Orifice with Variable Area Slot

Basic Assumptions and Limitations

The model is based on the following assumptions:

- Fluid inertia is not taken into account.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.

Dialog Box and Parameters

Block Parameters: Orifice with Variable Area Slot

Orifice with Variable Area Slot

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.

Connections A and B are conserving hydraulic ports associated with the orifice inlet and outlet, respectively. Connection S is a physical signal port. The block positive direction is from port A to port B. Positive signal at port S opens or closes the orifice, depending on the value of the Orifice orientation parameter.

Parameters

Orifice width: 0.01 m

Flow discharge coefficient: 0.7

Initial opening: 0 m

Orifice orientation: Opens in positive direction

Critical Reynolds number: 12

Leakage area: 1e-12 m²

OK Cancel Help Apply

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.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

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. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is $1e-12 \text{ m}^2$.

Orifice with Variable Area Slot

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

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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 Orifice

Fixed Orifice

Orifice with Variable Area Round Holes

Variable Area Orifice

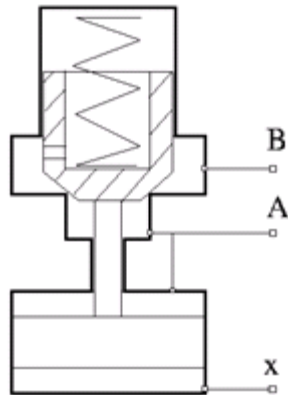
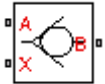
Variable Orifice

Pilot-Operated Check Valve

Purpose Simulate 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. Theoretically, the parameter can be set to zero, but it is not recommended.

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 flow rate is determined according to the following equations:

Pilot-Operated Check Valve

$$q = \begin{cases} C_D \cdot A \sqrt{\frac{2}{\rho} |p| \cdot \text{sign}(p)} & \text{for } Re \geq Re_{cr} \\ 2C_{DL} \cdot A \frac{D_H}{v \cdot \rho} p & \text{for } Re < Re_{cr} \end{cases}$$

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

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

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

$$p = p_A - p_B$$

$$Re = \frac{q \cdot D_H}{A(p) \cdot v}$$

$$C_{DL} = \left(\frac{C_D}{\sqrt{Re_{cr}}} \right)^2$$

$$D_H = \sqrt{\frac{4A(p)}{\pi}}$$

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, $k_p = p_X/p_A$
k	Valve gain coefficient
C_D	Flow discharge coefficient
$A(\rho)$	Instantaneous orifice passage area
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
D_H	Instantaneous orifice hydraulic diameter
ρ	Fluid density
ν	Fluid kinematic viscosity

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 $p = p_A - p_B$.

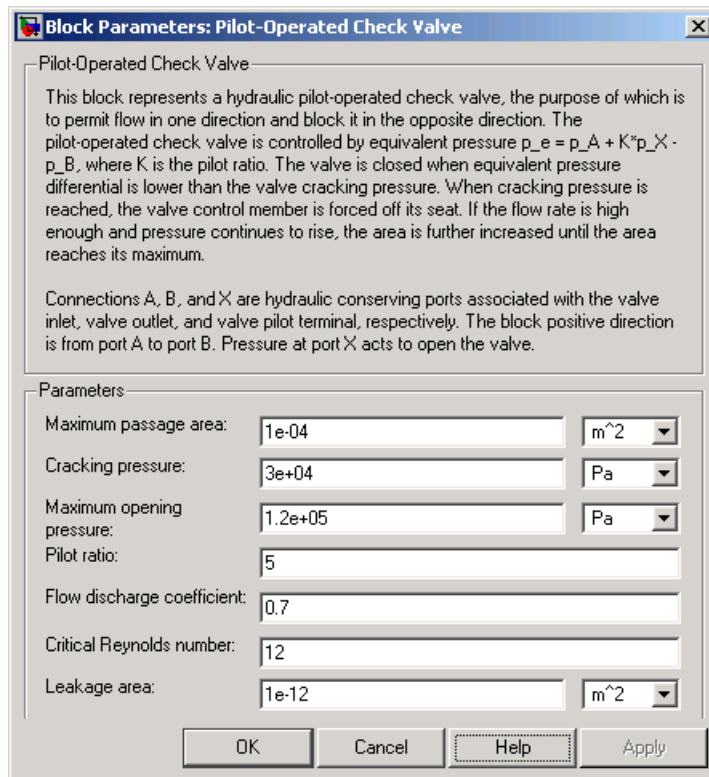
Basic Assumptions and Limitations

The model is based on the following assumptions:

- 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.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.

Pilot-Operated Check Valve

Dialog Box and 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.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

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. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is $1e-12 \text{ m}^2$.

Global Parameters**Fluid density**

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

Pilot-Operated Check Valve

- 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

Purpose Simulate 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 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], 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.

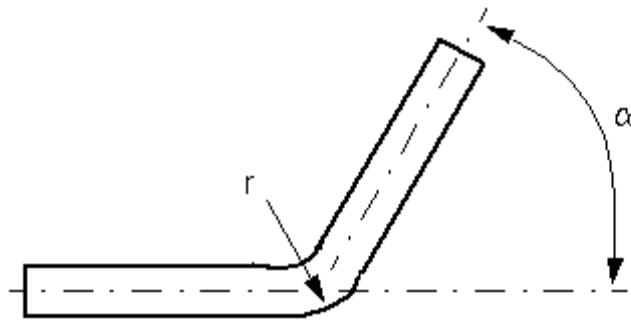
Pipe Bend

The pressure loss coefficient is determined according to the table provided in [1], p. A-29:

$$K = f(r, d, \alpha)$$

where

- d Pipe internal diameter
- r Curvature radius ($d \leq r \leq 20d$)
- α Bend angle in degrees ($0 \leq \alpha \leq 180$)



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

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

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 Resistive Tube block.

Basic Assumptions and Limitations

The model is based on the following assumptions:

- Fluid inertia, fluid compressibility, and wall compliance are not taken into account.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.
- The bend is assumed to be made of a clean commercial steel pipe.

Pipe Bend

Dialog Box and Parameters

Block Parameters: Pipe Bend

Pipe Bend

The block represents a local hydraulic resistance, such as a pipe bend. The pressure loss in the bend is assumed to consist of (1) the loss in the straight pipe and (2) the loss due to curvature. The loss in a straight pipe is simulated with the 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.

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:

Pipe diameter:	<input type="text" value="0.01"/>	<input type="text" value="m"/>
Bend radius:	<input type="text" value="0.04"/>	<input type="text" value="m"/>
Bend angle:	<input type="text" value="90"/>	<input type="text" value="deg"/>
Internal surface roughness height:	<input type="text" value="1.5e-05"/>	<input type="text" value="m"/>
Critical Reynolds number:	<input type="text" value="350"/>	

OK Cancel Help Apply

Pipe diameter

The internal diameter of the pipe. The default value is 0.01 m.

Bend radius

The radius of the bend. The default value is 0.04 m.

Bend angle

The angle of the bend. The value must be in the range between 0 and 180 degrees. The default value is 90 deg.

Internal surface roughness height

Roughness height on the pipe internal surface. The parameter is typically provided in data sheets or manufacturer's catalogs. The default value is $1.5e-5$ m, which corresponds to drawn tubing.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The 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 geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 350.

Global Parameters

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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

Pipe Bend

See Also

Elbow

Gradual Area Change

Local Resistance

Resistive Tube

Sudden Area Change

T-junction

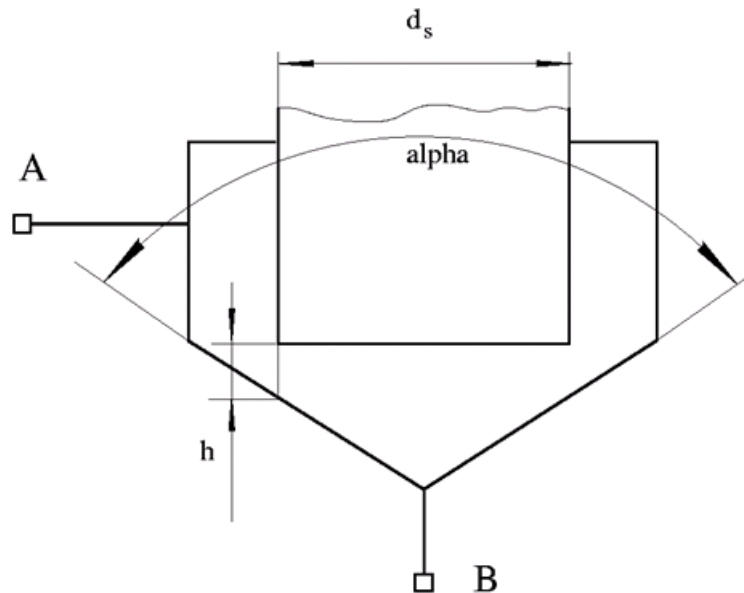
Purpose Simulate hydraulic poppet valve

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 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 flow rate is determined according to the following equations:

Poppet Valve

$$q = \begin{cases} C_D \cdot A \sqrt{\frac{2}{\rho} |p| \cdot \text{sign}(p)} & \text{for } Re \geq Re_{cr} \\ 2C_{DL} \cdot A \frac{D_H}{v \cdot \rho} p & \text{for } Re < Re_{cr} \end{cases}$$

$$h = x_0 + x$$

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

$$p = p_A - p_B$$

$$Re = \frac{q \cdot D_H}{A(h) \cdot v}$$

$$C_{DL} = \left(\frac{C_D}{\sqrt{Re_{cr}}} \right)^2$$

$$D_H = \sqrt{\frac{4A(h)}{\pi}}$$

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

where

q Flow rate

ρ 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
D_H	Valve instantaneous hydraulic diameter
ν	Fluid kinematic viscosity
A_{leak}	Closed valve leakage area
A_{max}	Maximum valve open area

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.

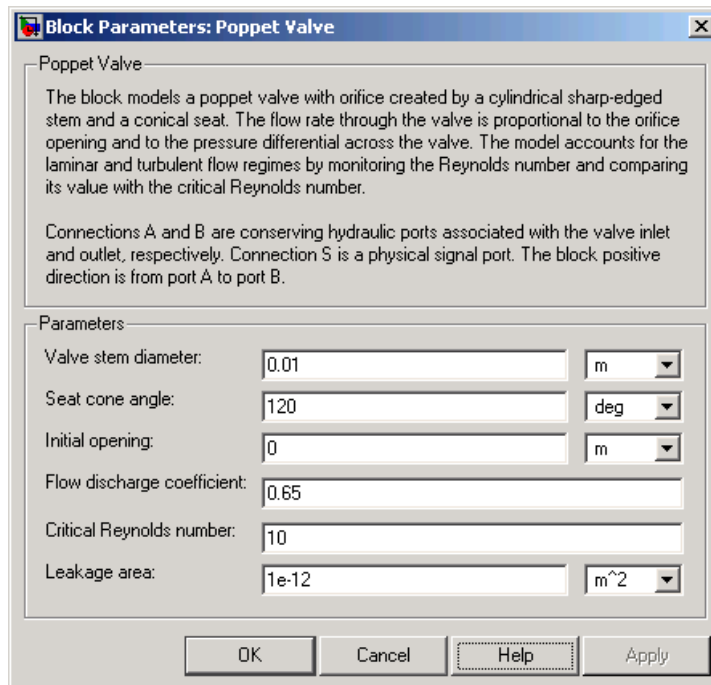
Basic Assumptions and Limitations

The model is based on the following assumptions:

- Fluid inertia is not taken into account.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.
- The flow passage area is assumed to be equal to the frustum side surface area.

Poppet Valve

Dialog Box and 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.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 10.

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. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is $1e-12 \text{ m}^2$.

Global Parameters

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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.

Poppet Valve

S

Physical signal port to control spool displacement.

See Also

Ball Valve

Needle Valve

Pressure-Compensated Flow Control Valve

Pressure-Compensated Flow Control Valve

Purpose

Simulate hydraulic pressure compensating valve

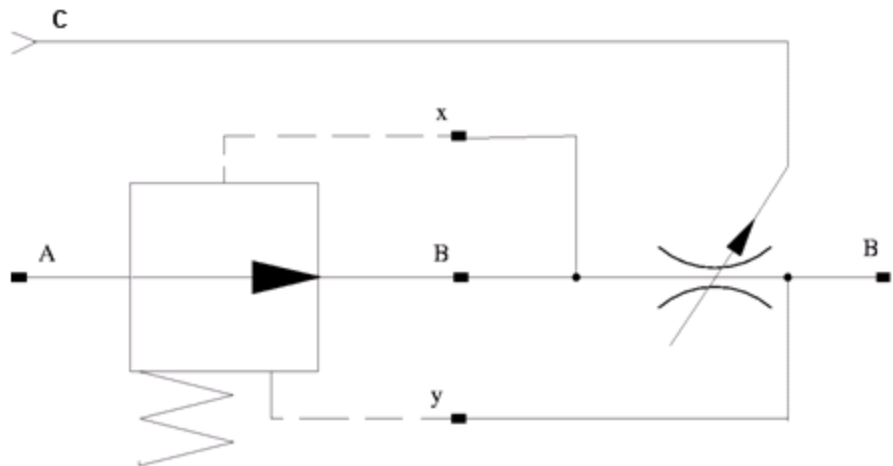
Library

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 $A=A(h)$.

Pressure-Compensated Flow Control Valve

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.

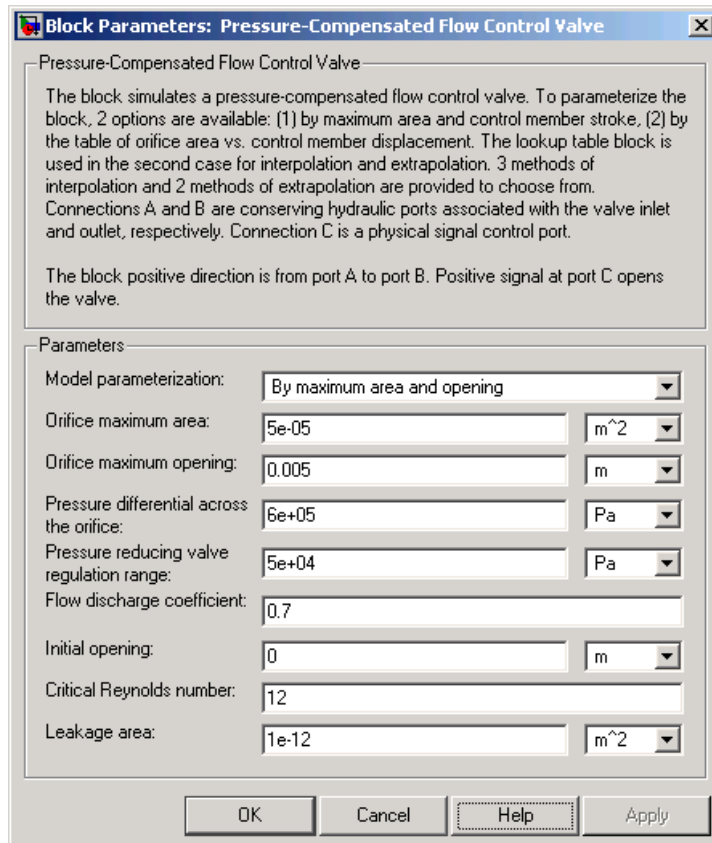
Assumptions and Limitations

The model is based on the following assumption:

- Fluid inertia is not taken into account.

Pressure-Compensated Flow Control Valve

Dialog Box and Parameters



Block Parameters: Pressure-Compensated Flow Control Valve

Pressure-Compensated Flow Control Valve

The block simulates a pressure-compensated flow control valve. To parameterize the block, 2 options are available: (1) by maximum area and control member stroke, (2) by the table of orifice area vs. control member displacement. The lookup table block is used in the second case for interpolation and extrapolation. 3 methods of interpolation and 2 methods of extrapolation are provided to choose from. Connections A and B are conserving hydraulic ports associated with the valve inlet and outlet, respectively. Connection C is a physical signal control port.

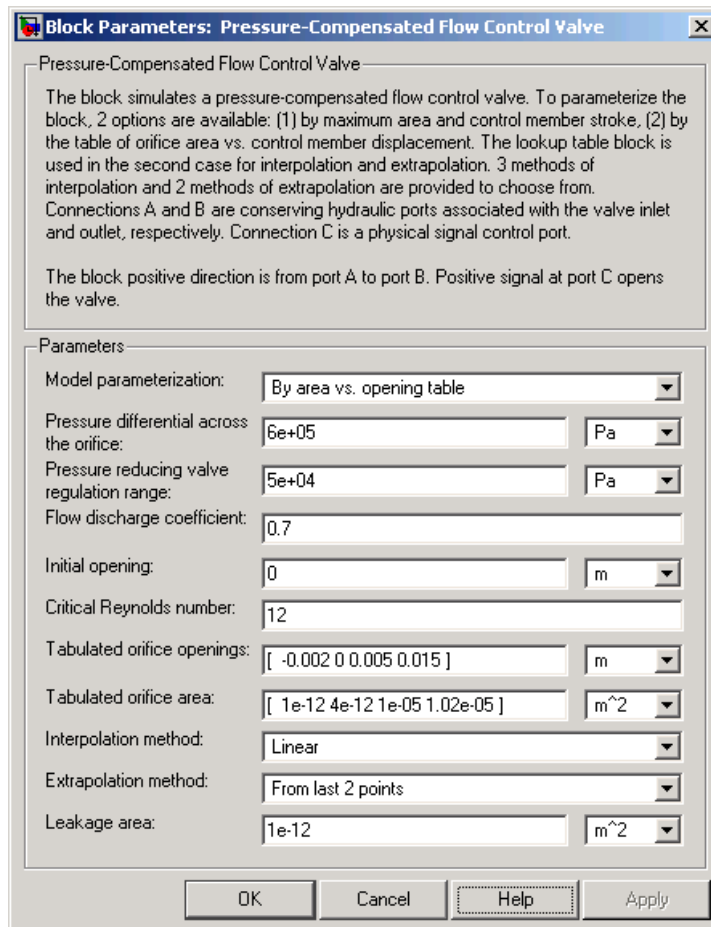
The block positive direction is from port A to port B. Positive signal at port C opens the valve.

Parameters

Model parameterization:	By maximum area and opening	
Orifice maximum area:	5e-05	m ²
Orifice maximum opening:	0.005	m
Pressure differential across the orifice:	6e+05	Pa
Pressure reducing valve regulation range:	5e+04	Pa
Flow discharge coefficient:	0.7	
Initial opening:	0	m
Critical Reynolds number:	12	
Leakage area:	1e-12	m ²

OK Cancel Help Apply

Pressure-Compensated Flow Control Valve



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

Pressure-Compensated Flow Control Valve

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 three 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 tabulated 1-by- m array. The input values vector must be strictly monotonically increasing. The values can be non-uniformly spaced. You must provide at least three values. 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 output values for orifice area as a tabulated 1-by- m array. The orifice area vector must be 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

Pressure-Compensated Flow Control Valve

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 — 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.

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:

- 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.

Pressure-Compensated Flow Control Valve

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. 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.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

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. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is $1e-12$ m².

Pressure-Compensated Flow Control Valve

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

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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
Needle Valve

Pressure-Compensated Flow Control Valve

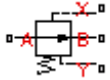
Poppet Valve

Pressure Compensator

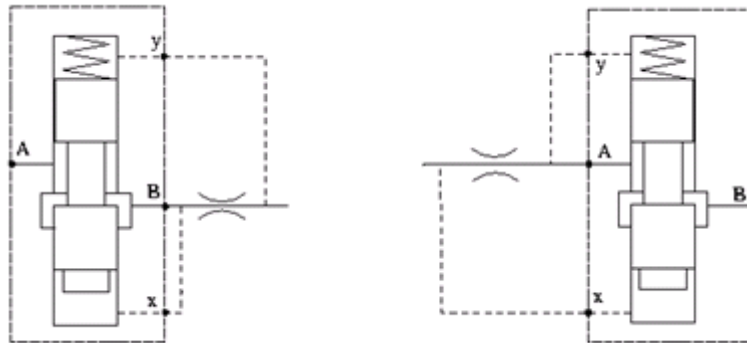
Purpose Simulate hydraulic pressure compensating valve

Library Pressure Control Valves

Description



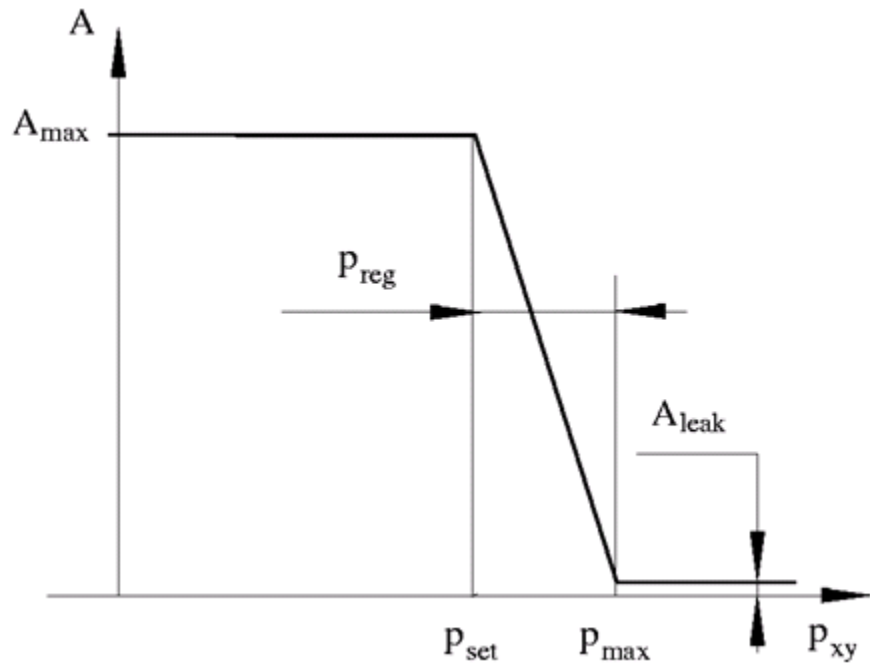
The Pressure Compensator block represents a hydraulic pressure compensating valve, or pressure compensator. Pressure compensators are used to maintain preset pressure differential across a hydraulic component to minimize the influence of pressure variation on a flow rate passing through the component. 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} .

Pressure Compensator



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

Pressure Compensator

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 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 flow rate is computed according to the following equations:

$$q = \begin{cases} C_D \cdot A \sqrt{\frac{2}{\rho} |p| \cdot \text{sign}(p)} & \text{for } Re \geq Re_{cr} \\ 2C_{DL} \cdot A \frac{D_H}{v \cdot \rho} p & \text{for } Re < Re_{cr} \end{cases}$$

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

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

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

$$p = p_A - p_B$$

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

$$Re = \frac{q \cdot D_H}{A(h) \cdot v}$$

$$C_{DL} = \left(\frac{C_D}{\sqrt{\text{Re}_{cr}}} \right)^2$$

$$D_H = \sqrt{\frac{4A(h)}{\pi}}$$

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
$A(h)$	Instantaneous orifice passage area
A_{max}	Orifice maximum area
C_D	Flow discharge coefficient
ρ	Fluid density
D_H	Instantaneous orifice hydraulic diameter
ν	Fluid kinematic viscosity
A_{leak}	Closed orifice leakage area

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

Pressure Compensator

measured as $P_{xy} = P_x - P_y$, and it creates a force acting against the spring preload.

Assumptions and Limitations

The model is based on the following assumptions:

- 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.

Dialog Box and Parameters

Block Parameters: Pressure Compensator

Pressure Compensator

The block represents a hydraulic pressure compensator. This kind of valve is used to maintain preset pressure differential across a hydraulic component to minimize the influence of pressure variation on the flow rate passing through the component.

Connections A and B are hydraulic conserving ports associated with the valve inlet and outlet, respectively. Connections X and Y are hydraulic conserving ports associated with the valve control ports. The pressure differential that controls the valve is determined as $p_{XY} = p_X - p_Y$. The block positive direction is from port A to port B.

Parameters

Maximum passage area:	<input type="text" value="1e-04"/>	<input type="text" value="m^2"/>
Valve pressure setting:	<input type="text" value="3e+06"/>	<input type="text" value="Pa"/>
Valve regulation range:	<input type="text" value="1.5e+05"/>	<input type="text" value="Pa"/>
Flow discharge coefficient:	<input type="text" value="0.7"/>	
Critical Reynolds number:	<input type="text" value="12"/>	
Leakage area:	<input type="text" value="1e-12"/>	<input type="text" value="m^2"/>

OK Cancel Help Apply

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 $3e6$ 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 $1.5e5$ 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.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

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. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is $1e-12$ m².

Global Parameters

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Pressure Compensator

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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

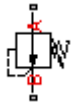
Pressure Relief Valve

Pressure Reducing Valve

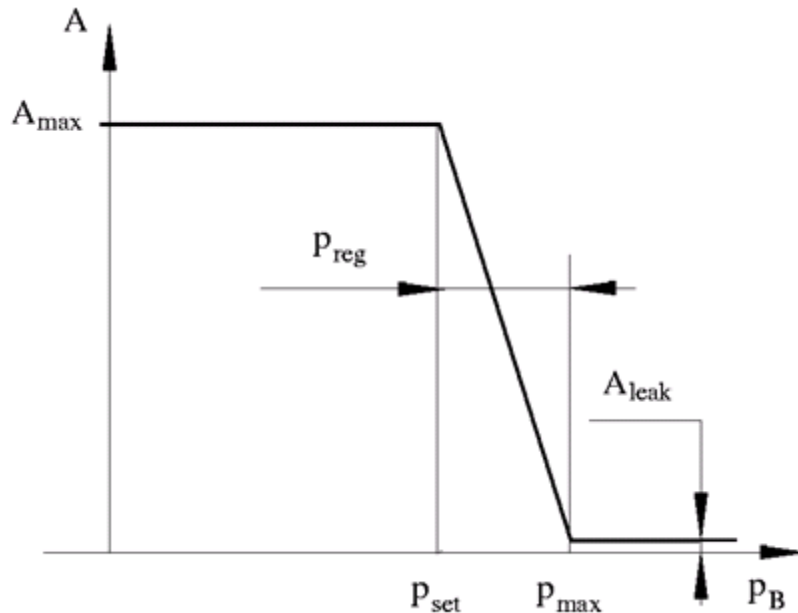
Purpose Simulate pressure control valve maintaining reduced pressure in portion of system

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.



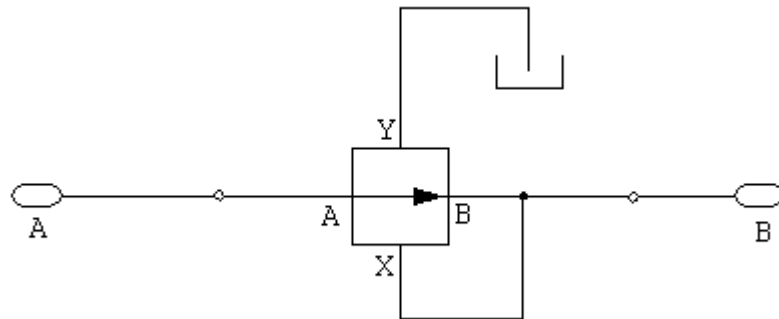
The pressure-reducing valve is a normally open valve and it remains fully open while outlet pressure is lower than the valve preset pressure. When the preset pressure is reached, the valve control member (spool, ball, poppet, etc.) is forced off its stop and starts closing the orifice, thus

Pressure Reducing Valve

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. The valve maximum area and regulation range are the key parameters of the block.

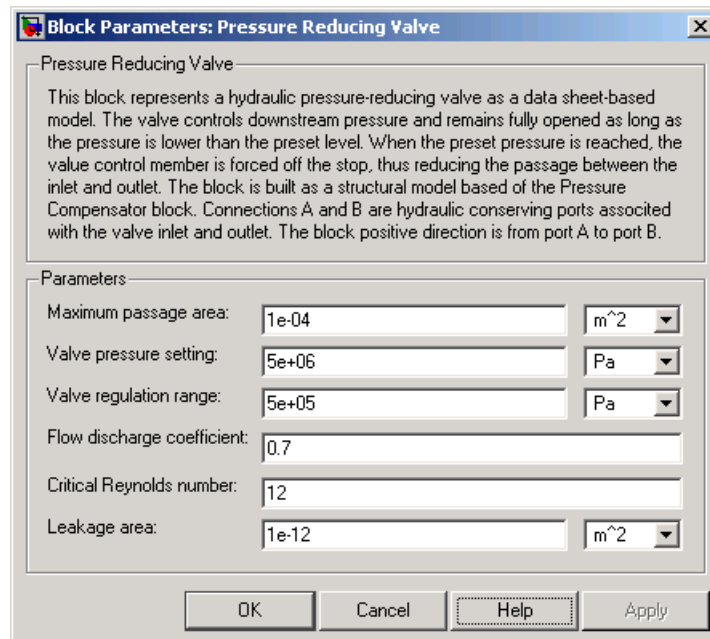
In addition to the maximum area, the leakage area is also required to characterize the valve. The main purpose of the parameter is not to account for possible leakage, even though this is also important, but to maintain numerical integrity of the circuit by preventing a portion of the system from getting 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. Theoretically, the parameter can be set to zero, but it is not recommended.

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 $p = p_A - p_B$.

Dialog Box and 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

Pressure Reducing Valve

usually is provided in textbooks or manufacturer data sheets. The default value is 0.7.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

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. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is $1e-12m^2$.

Global Parameters

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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 Power Unit with Pressure Reducing Valve demo (sh_power_unit_pressure_red_valve) 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

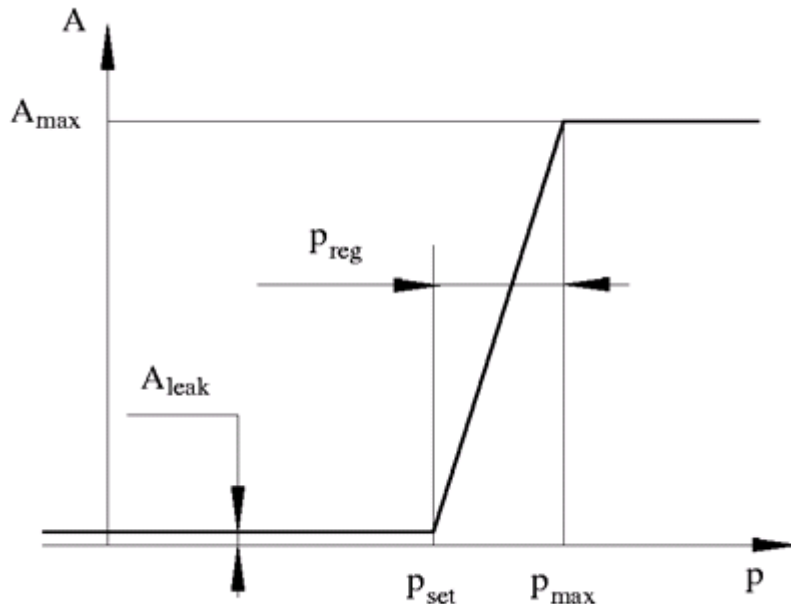
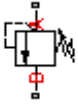
Pressure Compensator
Pressure Relief Valve

Pressure Relief Valve

Purpose Simulate 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. Theoretically, the parameter can be set to zero, but it is not recommended.

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 flow rate is determined according to the following equations:

$$q = \begin{cases} C_D \cdot A \sqrt{\frac{2}{\rho} |p| \cdot \text{sign}(p)} & \text{for } Re \geq Re_{cr} \\ 2C_{DL} \cdot A \frac{DH}{v \cdot \rho} p & \text{for } Re < Re_{cr} \end{cases}$$

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

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

Pressure Relief Valve

$$p = p_A - p_B$$

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

$$C_{DL} = \left(\frac{C_D}{\sqrt{\text{Re}_{cr}}} \right)^2$$

$$D_H = \sqrt{\frac{4A(p)}{\pi}}$$

where

q	Flow rate through the valve
p	Pressure differential across the valve
p_A, p_B	Gauge pressures at the block terminals
C_D	Flow discharge coefficient
$A(p)$	Instantaneous orifice passage area
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
D_H	Instantaneous orifice hydraulic diameter
ρ	Fluid density
ν	Fluid kinematic viscosity

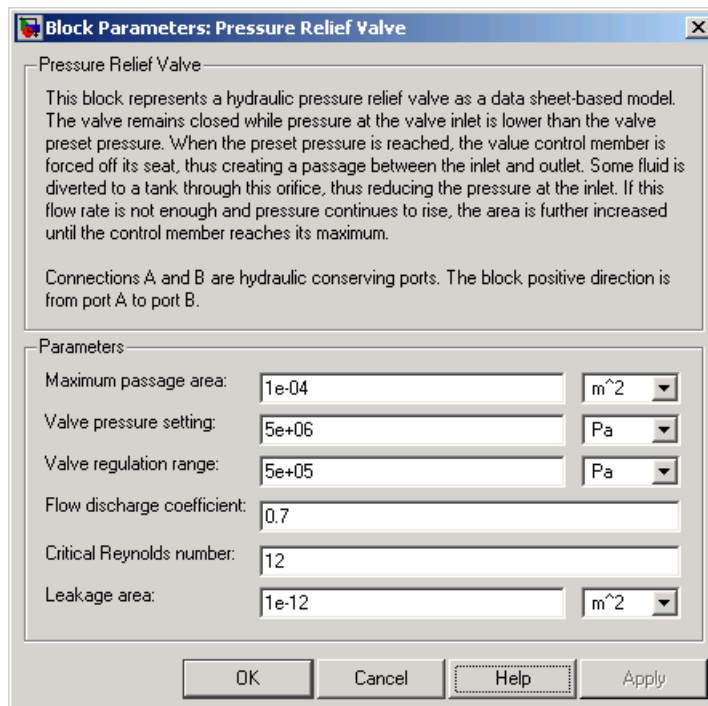
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as $p = p_A - p_B$.

Basic Assumptions and Limitations

The model is based on the following assumptions:

- Valve opening is linearly proportional to the pressure differential.
- No loading on the valve, such as inertia, friction, spring, and so on, is considered.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.

Dialog Box and Parameters



Block Parameters: Pressure Relief Valve

Pressure Relief Valve

This block represents a hydraulic pressure relief valve as a data sheet-based model. 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 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.

Connections A and B are hydraulic conserving ports. The block positive direction is from port A to port B.

Parameters

Maximum passage area:	<input type="text" value="1e-04"/>	<input type="text" value="m^2"/>
Valve pressure setting:	<input type="text" value="5e+06"/>	<input type="text" value="Pa"/>
Valve regulation range:	<input type="text" value="5e+05"/>	<input type="text" value="Pa"/>
Flow discharge coefficient:	<input type="text" value="0.7"/>	
Critical Reynolds number:	<input type="text" value="12"/>	
Leakage area:	<input type="text" value="1e-12"/>	<input type="text" value="m^2"/>

OK Cancel Help Apply

Pressure Relief Valve

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. 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.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

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. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is $1e-12 \text{ m}^2$.

Global Parameters

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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 Power Unit with Fixed-Displacement Pump demo (sh_power_unit_fxd_dsp1_pump) 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

Proportional and Servo-Valve Actuator

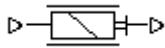
Purpose

Simulate continuous valve driver with output proportional to input signal

Library

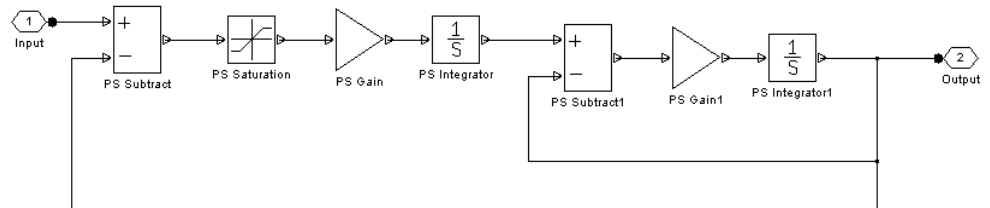
Valve Actuators

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.

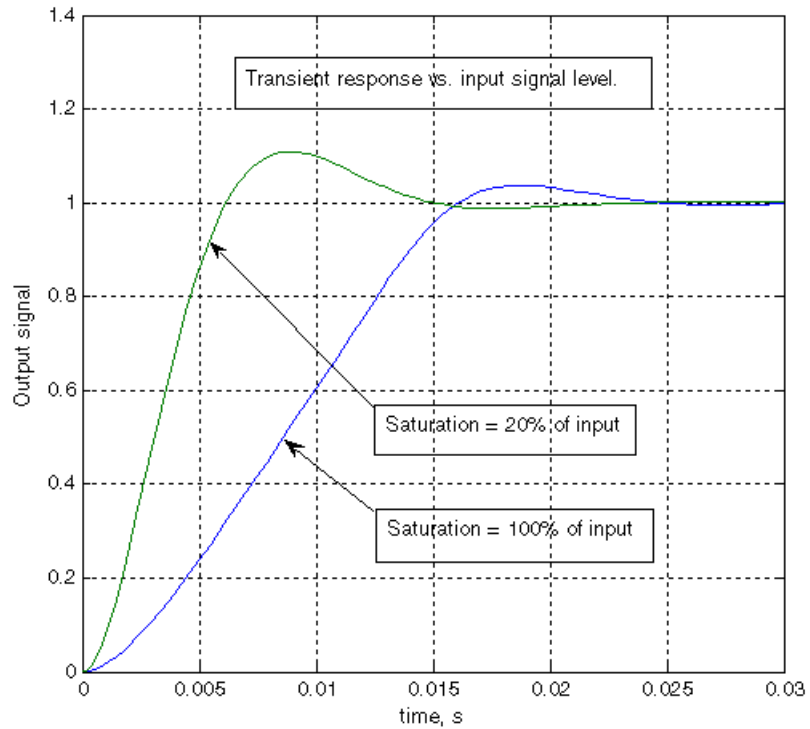
The Proportional and Servo-Valve Actuator block is built using the blocks from the Physical Signals library. Both the input and the output of the block are physical signals. The block diagram of the model is shown in the following figure.



The model consists of the first-order lag, PS Integrator, PS Saturation block, and the PS Subtract block that closes the feedback. The configuration is found to be suitable to simulate behavior of servo-valves and high-quality proportional valves.

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. The response that corresponds to 100% of the input signal is considerably slower than that with the 20% saturation.

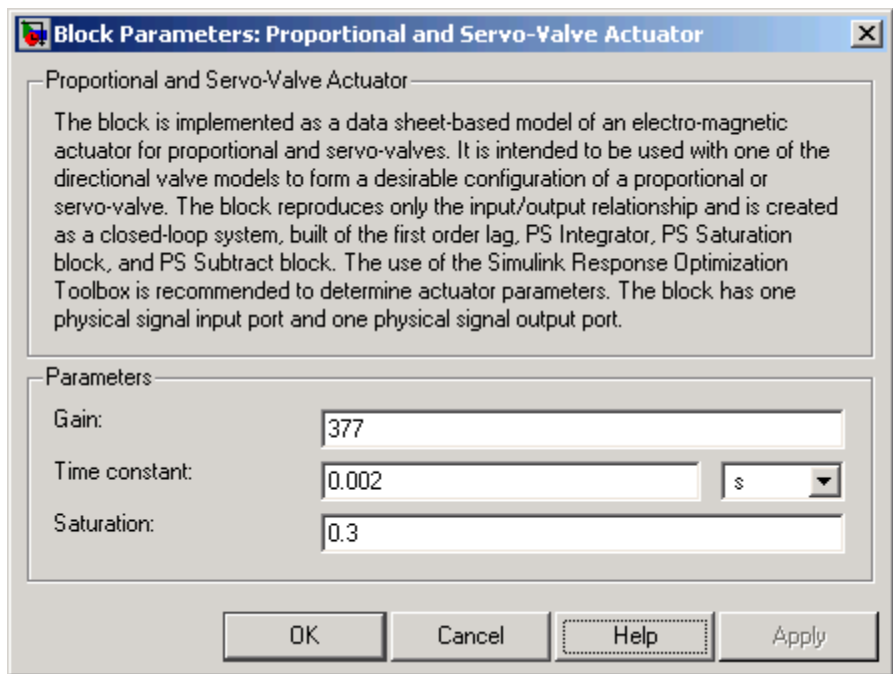
Proportional and Servo-Valve Actuator



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 Simulink Response Optimization Toolbox.

Proportional and Servo-Valve Actuator

Dialog Box and Parameters



Gain

Gain of the first-order lag. The default value is 377.

Time constant

Time constant of the first-order lag. The default value is 0.002 s.

Saturation

Saturation level of the 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.

Proportional and Servo-Valve Actuator

Examples

The Closed-Loop Electrohydraulic Actuator with Proportional Valve demo (`sh_closed_loop_actuator`) 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

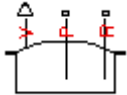
Hydraulic Single-Acting Valve Actuator

Reservoir

Purpose Simulate pressurized hydraulic reservoir

Library Hydraulic Utilities

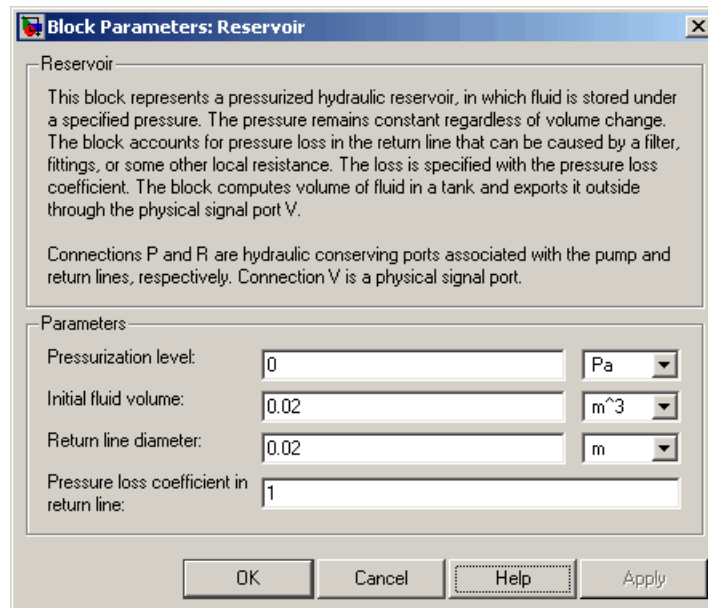
Description



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.

Dialog Box and 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 m³.

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. The default value is 1.

Ports

The block has the following ports:

P

Hydraulic conserving port associated with the pump line.

Reservoir

- R Hydraulic conserving port associated with the return line.
- V Physical signal port that output the volume of fluid in the tank.

See Also Hydraulic Reference

Segmented Pipeline

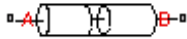
Purpose

Simulate hydraulic pipeline with resistive, fluid inertia, and fluid compressibility properties

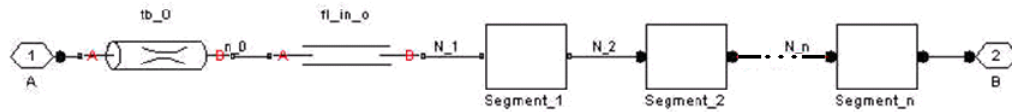
Library

Pipelines

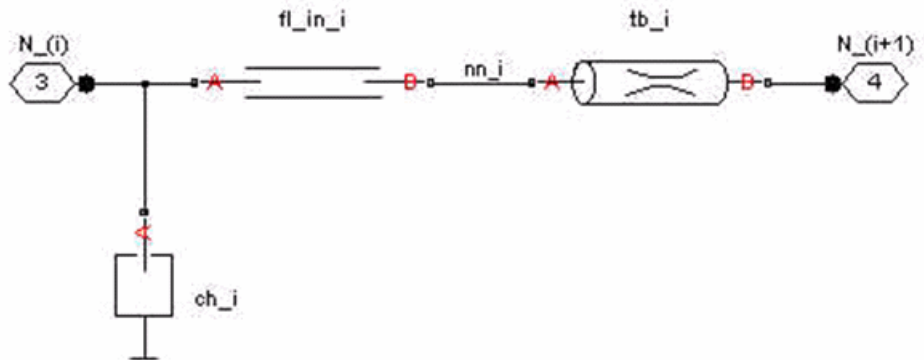
Description



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

Segmented Pipeline

The model contains as many Constant 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 Chamber block is placed between two branches, each consisting of a Resistive Tube block and a Fluid Inertia block. Every 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 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 Tube and Fluid Inertia blocks are assigned names $nn_0, nn_1, nn_2, \dots, nn_n$. The Constant Chamber blocks are named ch_1, ch_2, \dots, ch_n , 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 is the block parameter. In determining the number of segments needed, you have to find a compromise between the accuracy and computational burden for a particular application. It is practically impossible to determine analytically how many elements are necessary to get the results with a specified accuracy. The golden rule is to use as many elements as possible based on computational considerations, and an experimental assessment is perhaps the only

reliable way to make any conclusions. As an approximate estimate, you can use the following formula:

$$N > \frac{4L}{\pi \cdot c} \omega$$

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 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 $p = p_A - p_B$.

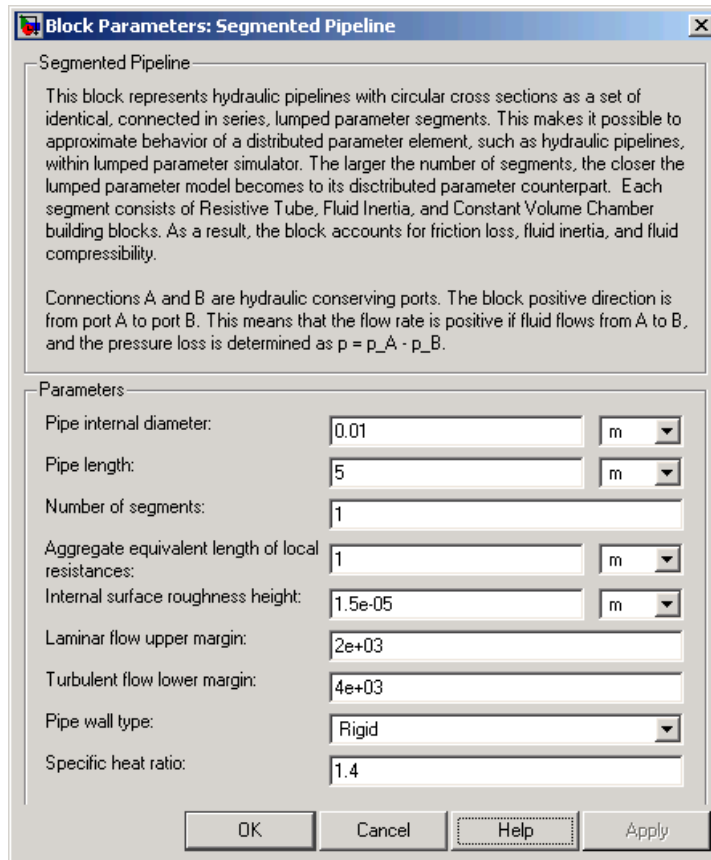
Segmented Pipeline

Basic Assumptions and Limitations

Dialog Box and Parameters

The model is based on the following assumption:

- Flow is assumed to be fully developed along the pipe length.



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.

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. 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 Compliant. If the parameter is set to Rigid, wall compliance is not taken into account, which can improve computational efficiency. The value Compliant is recommended for hoses and metal pipes where wall compliance can affect the system behavior. The default value is Rigid.

Segmented Pipeline

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 Compliant, 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 Chamber block. The default value is 1.4.

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.

Global Parameters

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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

Linear Hydraulic Resistance

Resistive Tube

Shuttle Valve

Purpose Simulate hydraulic valve that allows flow in one direction only

Library Directional Valves

Description

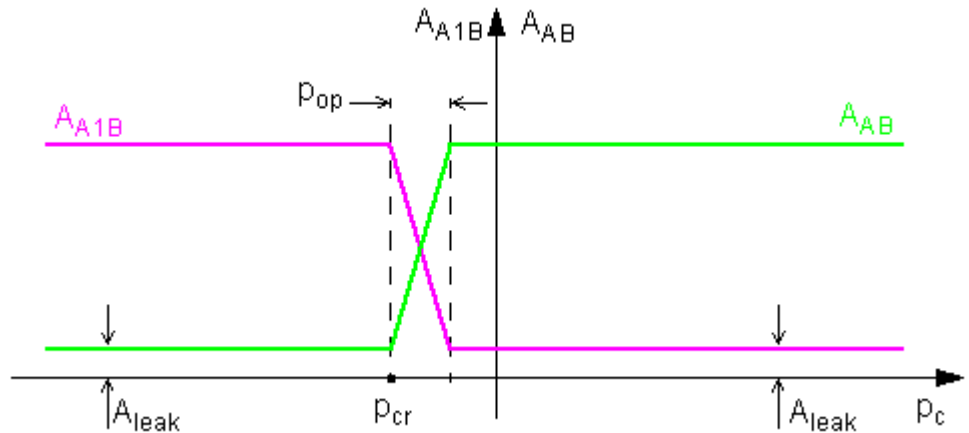


The Shuttle Valve block represents a hydraulic shuttle valve as a data-sheet-based model. The valve has two inlet ports (A and A1) and one outlet port (B). The valve is controlled by pressure differential

$p_c = p_A - p_{A1}$. The valve permits flow either between ports A and B or between ports A1 and B, depending on the pressure differential p_c . Initially, path A-B is assumed to be opened. To open path A1-B (and close A-B at the same time), pressure differential must be less than the valve cracking pressure ($p_{cr} <= 0$).

When cracking pressure is reached, the valve control member (spool, ball, poppet, etc.) is forced off its seat and moves to the opposite seat, thus opening one passage and closing the other. If the flow rate is high enough and pressure continues to change, the control member continues to move until it reaches its extreme position. At this moment, one of the valve passage areas is at its maximum. The valve maximum area and the cracking and maximum pressures are generally provided in the catalogs and are the three key parameters of the block.

The relationship between the A-B, A1-B path openings and control pressure p_c is shown in the following illustration.



In addition to the maximum area, the leakage area is also required to characterize the valve. The main purpose of the parameter is not to account for possible leakage, even though this is also important, but to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Theoretically, the parameter can be set to zero, but it is not recommended.

The model accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number for each orifice (Re_{AB}, Re_{A1B}) and comparing its value with the critical Reynolds number (Re_{cr}). The flow rate through each of the orifices is determined according to the following equations:

$$q_{AB} = \begin{cases} C_D \cdot A_{AB} \sqrt{\frac{2}{\rho} |p_{AB}| \cdot \text{sign}(p_{AB})} & \text{for } Re_{AB} \geq Re_{cr} \\ 2C_{DL} \cdot A_{AB} \frac{D_{HAB}}{v \cdot \rho} p_{AB} & \text{for } Re_{AB} < Re_{cr} \end{cases}$$

Shuttle Valve

$$q_{A1B} = \begin{cases} C_D \cdot A_{A1B} \sqrt{\frac{2}{\rho} |p_{A1B}| \cdot \text{sign}(p_{A1B})} & \text{for } Re_{A1B} \geq Re_{cr} \\ 2C_{DL} \cdot A_{A1B} \frac{D_{HA1B}}{v \cdot \rho} p_{A1B} & \text{for } Re_{A1B} < Re_{cr} \end{cases}$$

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

$$A_{A1B} = \begin{cases} A_{leak} & \text{for } p_{A1B} \geq p_{cr} + p_{op} \\ A_{max} - k \cdot (p_{A1B} - p_{cr}) & \text{for } p_{cr} < p_{A1B} < p_{cr} + p_{op} \\ A_{max} & \text{for } p_{A1B} \leq p_{cr} \end{cases}$$

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

$$p_{AB} = p_A - p_B$$

$$p_{A1B} = p_{A1} - p_B$$

$$Re_{AB} = \frac{q_{AB} \cdot D_{HAB}}{A_{AB} \cdot v}$$

$$Re_{A1B} = \frac{q_{A1B} \cdot D_{HA1B}}{A_{A1B} \cdot v}$$

$$C_{DL} = \left(\frac{C_D}{\sqrt{Re_{cr}}} \right)^2$$

$$D_{HAB} = \sqrt{\frac{4A_{AB}}{\pi}}$$

$$D_{HA1B} = \sqrt{\frac{4A_{A1B}}{\pi}}$$

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
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_{cr}	Valve cracking pressure differential
p_{opx}	Pressure differential needed to fully shift the valve
D_{HAB}, D_{HA1B}	Instantaneous orifice hydraulic diameters
ρ	Fluid density
ν	Fluid kinematic viscosity

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

The model is based on the following assumptions:

- Valve opening is linearly proportional to the pressure differential.
- No loading on the valve, such as inertia, friction, spring, and so on, is considered.

Shuttle Valve

- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.

Dialog Box and Parameters

Block Parameters: Shuttle Valve

Shuttle Valve

This block represents 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 ports A1 and B depending on pressure differential P_c . Initially, path A-B is assumed to be opened. To open A1-B path (and close A-B at the same time), pressure differential must be greater than the valve cracking pressure ($P_{cr} <= 0$).

Connections A, A1 and B are hydraulic conserving ports associated with the valve inlets and outlet, respectively. The block positive direction is from port A to port B and from port A1 to port B. Control pressure is determined as $P_c = P_a - P_{a1}$.

Parameters

Maximum passage area:	<input type="text" value="1e-04"/>	<input type="text" value="m^2"/>
Cracking pressure:	<input type="text" value="-1e+04"/>	<input type="text" value="Pa"/>
Opening pressure:	<input type="text" value="1e+04"/>	<input type="text" value="Pa"/>
Flow discharge coefficient:	<input type="text" value="0.7"/>	
Critical Reynolds number:	<input type="text" value="12"/>	
Leakage area:	<input type="text" value="1e-12"/>	<input type="text" value="m^2"/>

OK Cancel Help Apply

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 $-1e4 \text{ Pa}$.

Opening pressure

Pressure differential across the valve needed to shift the valve from one extreme position to another. The default value is $1e4 \text{ Pa}$.

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.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

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. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is $1e-12 \text{ m}^2$.

Global Parameters

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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.

Shuttle Valve

- 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

Single-Acting Hydraulic Cylinder

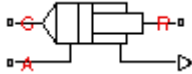
Purpose

Simulate hydraulic actuator exerting force in one direction

Library

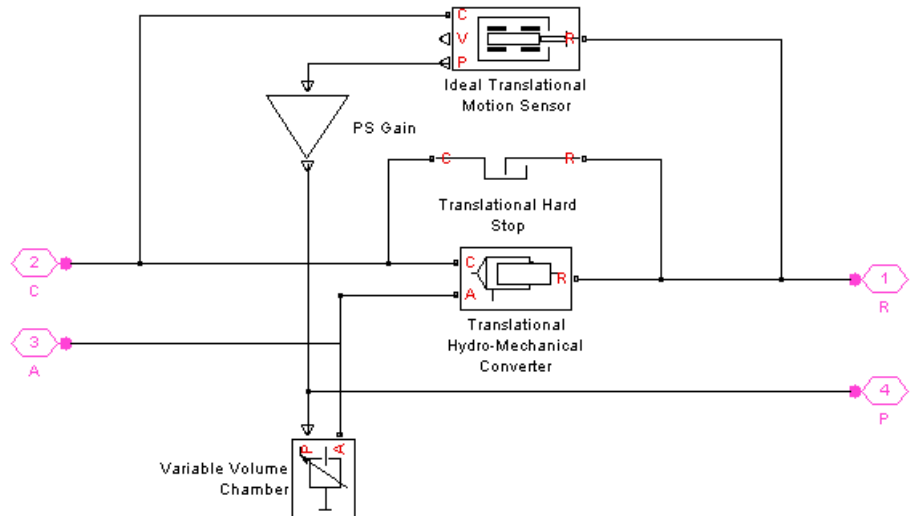
Hydraulic Cylinders

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 opposite direction.

The model of the cylinder is built of other SimHydraulics 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

Single-Acting Hydraulic Cylinder

with the cylinder inlet. The physical signal output port provides rod displacement.

The energy through port A is directed to the Translational Hydro-Mechanical Converter block and the Variable Volume Chamber block. The converter transforms hydraulic energy into mechanical energy, while the chamber 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 Ideal Translational Motion Sensor block in the schematic is introduced to determine an instantaneous piston position, which is necessary for the Variable Volume Chamber block.

The block directionality is adjustable and can be controlled with the **Cylinder orientation** parameter.

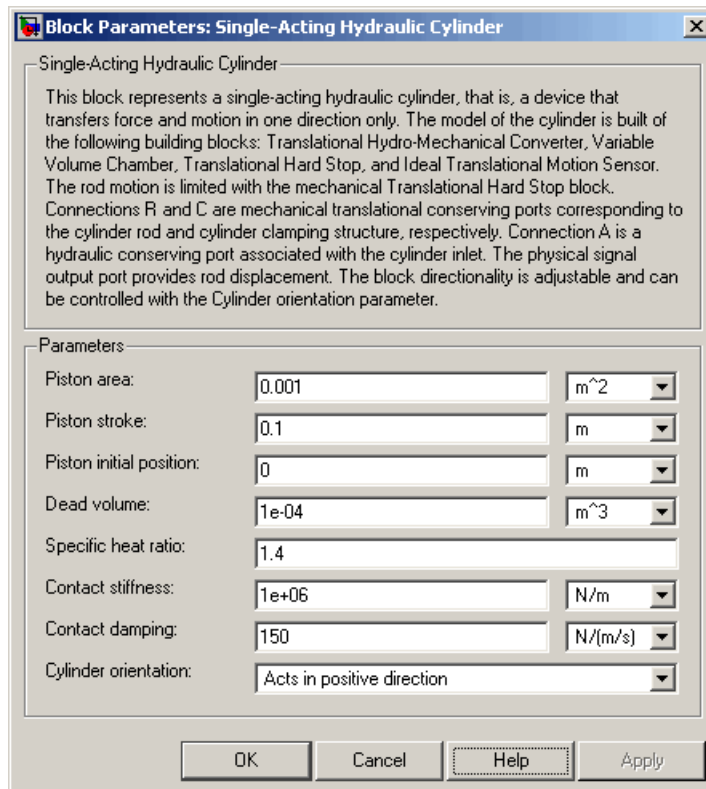
Basic Assumptions and Limitations

The model is based on the following assumptions:

- 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.

Single-Acting Hydraulic Cylinder

Dialog Box and Parameters



Piston area

Effective piston area. The default value is 0.001 m².

Piston stroke

Piston maximum travel between caps. The default value is 0.1 m.

Piston initial position

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.

Single-Acting Hydraulic Cylinder

Dead volume

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

Specific heat ratio

Gas-specific heat ratio for the Variable Volume 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, The MathWorks recommends that you assign a nonzero value to this parameter. The default value is 150 N*s/m .

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

Fluid bulk modulus

The parameter is determined by the type of working fluid selected for the system under design. 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.

The block also has a physical signal output port, which outputs rod displacement.

See Also

Double-Acting Hydraulic Cylinder

Ideal Translational Motion Sensor

Translational Hard Stop

Translational Hydro-Mechanical Converter

Variable Volume Chamber

Single-Acting Rotary Actuator

Purpose Simulate single-acting hydraulic rotary actuator

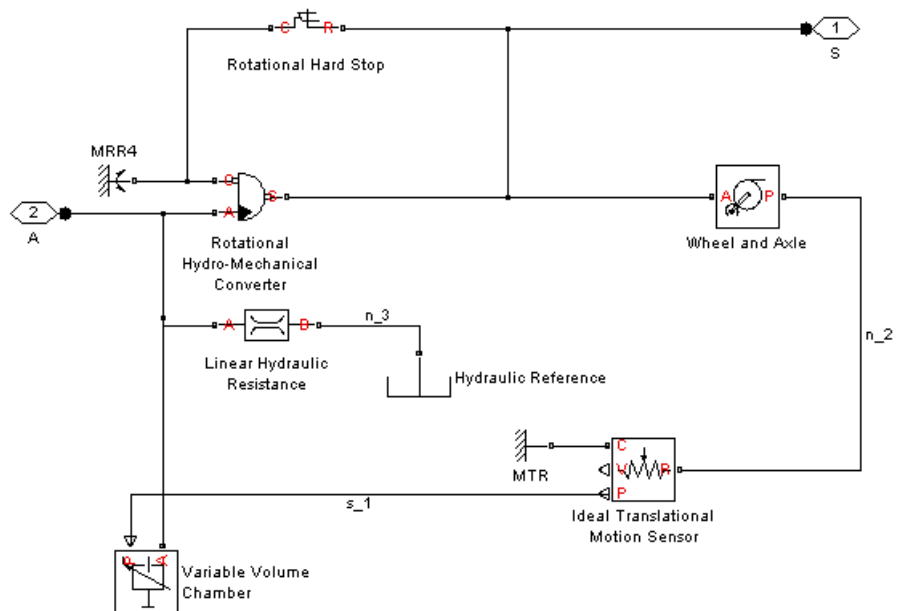
Library Hydraulic Cylinders

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 opposite direction.

The model of the actuator is built of Simscape Foundation library blocks. The schematic diagram of the model is shown below.



Single-Acting Rotary Actuator

The blocks in the diagram perform the following functions:

Rotational Hydro-Mechanical Converter	Converts hydraulics energy into mechanical rotational energy and vice versa.
Rotational Hard Stop	Imposes limits on shaft rotation.
Linear Hydraulic Resistance	Accounts for leakages.
Variable Volume Chamber	Accounts for fluid compressibility.
Ideal Translational Motion Sensor	Determines an instantaneous shaft position, which is necessary for the Variable Volume Chamber block.
Wheel and Axle	Converts shaft rotation into translational motion to provide input to the Ideal Translational Motion Sensor block

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

The model is based on the following assumption:

- 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.

Single-Acting Rotary Actuator

Dialog Box and Parameters

Block Parameters: Single-Acting Rotary Actuator

Single-Acting Rotary Actuator

This block represents 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. The model of the actuator is built of the following building blocks: Rotational Hydro-Mechanical Converter, Variable Volume Chamber, Wheel and Axle, and Linear Hydraulic Resistance. The shaft rotation is limited with the mechanical Rotational Hard Stop block.

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.

Parameters

Actuator displacement:	4.5e-05	m ³ /rad
Shaft stroke:	5.1	rad
Shaft initial angle:	0	rad
Dead volume:	1e-04	m ³
Leak coefficient:	1e-14	m ³ /s/Pa
Specific heat ratio:	1.4	
Contact stiffness:	1e+06	N*m/rad
Contact damping:	150	N*m/(rad/s)
Actuator orientation:	Acts in positive direction	

OK Cancel Help Apply

Actuator displacement

Effective displacement of the actuator. The default value is $4.5e-5 \text{ m}^3/\text{rad}$.

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

Leak coefficient

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

Specific heat ratio

Gas-specific heat ratio for the Variable Volume Chamber block. The default value is 1.4.

Contact stiffness

Specifies the elastic property of colliding bodies for the Rotational Hard Stop block. The greater the value of the parameter, the less the bodies penetrate into each other, the more rigid the impact becomes. Lesser value of the parameter makes contact softer, but generally improves convergence and computational efficiency. The default value is $1e6 \text{ N}\cdot\text{m/rad}$.

Contact damping

Specifies dissipating property of colliding bodies for the Rotational Hard Stop block. At zero damping, the impact is close to an absolutely elastic one. The greater the value of the parameter, the more energy dissipates during an interaction. Keep in mind that damping affects slider motion as long as the slider is in contact with the stop, including the period when slider is pulled back from the contact. For computational efficiency and convergence reasons, The MathWorks recommends that you assign a nonzero value to this parameter. The default value is $150 \text{ N}\cdot\text{m}/(\text{rad/s})$.

Single-Acting Rotary Actuator

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

Fluid bulk modulus

The parameter is determined by the type of working fluid selected for the system under design. 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

Ideal Translational Motion Sensor
Linear Hydraulic Resistance
Rotational Hard Stop
Rotational Hydro-Mechanical Converter

Single-Acting Rotary Actuator

Variable Volume Chamber

Wheel and Axle

Spool Orifice Hydraulic Force

Purpose Simulate 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 orifice is supposed to be rectangular with the width considerably larger than the radial clearance between the spool and the sleeve.



The force is simulated according to the following equations:

$$F = \rho \frac{q^2}{A} \cos \theta \cdot or$$

$$\theta = 0.3663 + 0.8373(1 - \exp(-x/1.848))$$

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

$$A = \begin{cases} b \cdot \sqrt{x^2 + \delta^2} & \text{for } x > 0 \\ b \cdot \delta & \text{for } x \leq 0 \end{cases}$$

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

- 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.

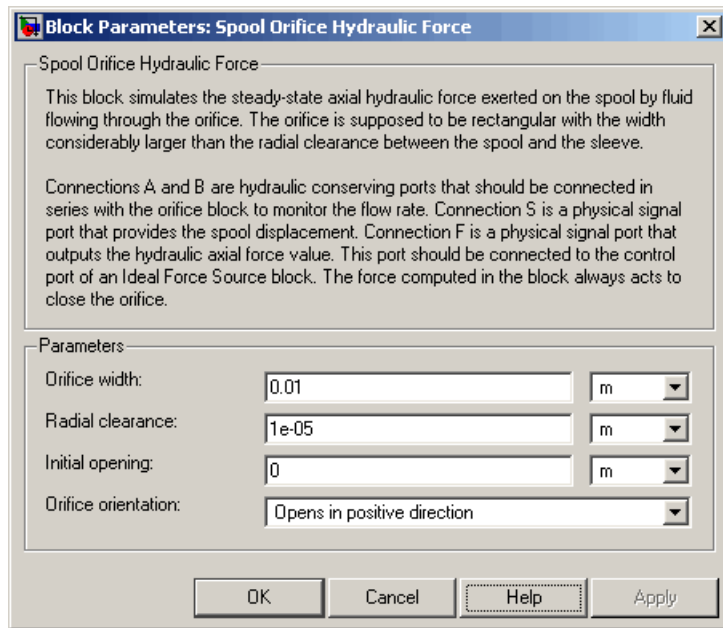
Basic Assumptions and Limitations

The model is based on the following assumptions:

- The transient effects are assumed to be negligible.
- The jet angle approximation is based on the Richard von Mises equation.
- The block can be used with rectangular orifices whose width is considerably larger than the axial opening.

Spool Orifice Hydraulic Force

Dialog Box and Parameters



Orifice width

Orifice width. The parameter must be greater than zero. The default value is 0.01 m.

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 parameter:

- **Orifice orientation**

All other block parameters are available for modification.

Ports

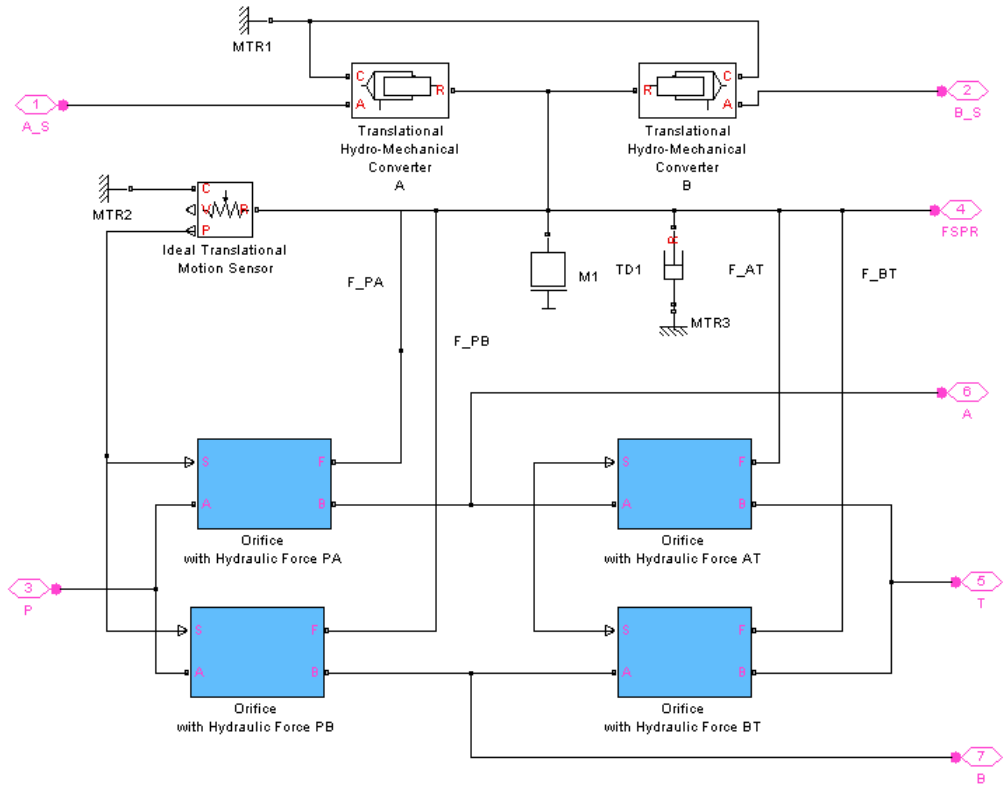
The block has the following ports:

- 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.

Spool Orifice Hydraulic Force



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.

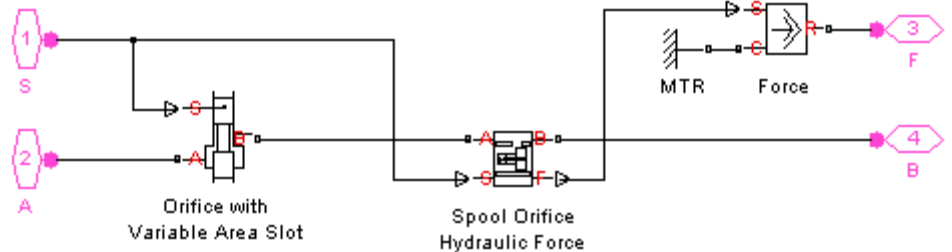
Four variable orifices are represented by subsystems:

- Orifice with Hydraulic Force PA
- Orifice with Hydraulic Force PB
- Orifice with Hydraulic Force AT

Spool Orifice Hydraulic Force

- 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 Hydraulic System with Servo-Valve demo (`sh_hydraulic_system_with_servo_valve`).

See Also

Valve Hydraulic Force

Spring-Loaded Accumulator

Purpose Simulate hydraulic accumulator with spring used for energy storage

Library Accumulators

Description



This block represents a spring-loaded accumulator, where fluid entering the accumulator compresses the spring, thus storing hydraulic energy. Since the spring compression increases as fluid enters the chamber and decreases as the accumulator is discharged, the pressure is not constant. The spring is preloaded. Therefore, fluid starts entering the chamber only after the inlet pressure crosses over this threshold. The accumulator is described with the following equations:

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

$$V_F = \begin{cases} 0 & \text{for } p \leq p_{pr} \\ k(p - p_{pr}) & \text{for } p_{pr} < p < p_{max} \\ V_{max} & \text{for } p \geq p_{max} \end{cases}$$

$$k = \frac{V_{max}}{p_{max} - p_{pr}}$$

where

- p Pressure at the accumulator inlet
- q Flow rate into accumulator
- V_{max} Accumulator capacity (maximum volume)
- V_F Instantaneous volume of fluid in the accumulator
- p_{pr} Preload pressure
- p_{max} Pressure needed to fully fill the accumulator

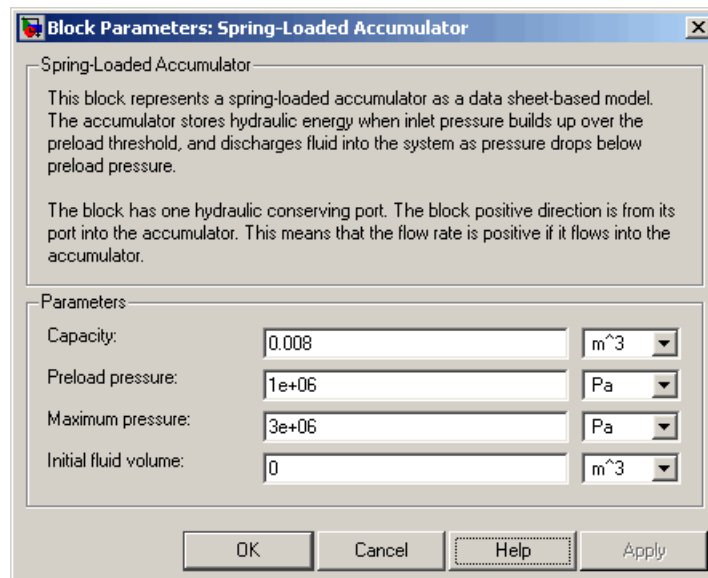
The block positive direction is from port A into the accumulator. This means that the flow rate is positive if it flows into the accumulator.

Basic Assumptions and Limitations

The model is based on the following assumptions:

- The spring has linear characteristics.
- No loading on the separator, such as inertia, friction, and so on, is considered.
- Fluid compressibility is not taken into account.

Dialog Box and Parameters



Capacity

Accumulator volumetric capacity. The default value is 0.008 m³.

Preload pressure

Pressure at which fluid starts entering the chamber. The default value is 1e6 Pa.

Maximum pressure

Pressure at which the accumulator is fully charged. The default value is 3e6 Pa.

Spring-Loaded Accumulator

Initial fluid volume

Initial volume of fluid in the accumulator. 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 "Computing Initial Conditions". The default value is 0.

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

Purpose Simulate 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]):

$$K_{SE} = K_{cor} \left(1 - \frac{A_S}{A_L} \right)^2$$

$$K_{SC} = K_{cor} \cdot 0.5 \left(1 - \frac{A_S}{A_L} \right)^{0.75}$$

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

Sudden Area Change

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 three 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

The model is based on the following assumption:

- Fluid inertia is not taken into account.
- If you select parameterization by semi-empirical formulas, the transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.
- If you select parameterization by the table-specified relationship $K=f(Re)$, the flow is assumed to be turbulent.

Dialog Box and Parameters

Block Parameters: Sudden Area Change

Sudden Area Change

The block represents a local hydraulic resistance, such as a sudden cross-sectional area change. The resistance is characterized as a sudden enlargement if fluid flows from inlet to outlet, or as a sudden contraction if fluid flows from outlet to inlet. The block offers two methods of the loss coefficient specification: by applying semi-empirical formulas or by table-lookup for the pressure loss coefficient based on the Reynolds number. The block is bidirectional and computes pressure loss for both the direct flow (sudden enlargement) and return flow (sudden contraction). If the second parameterization option is selected (By loss coefficient vs. Re table), the table must cover both the positive and negative Reynolds number regions.

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

Small diameter: m

Large diameter: m

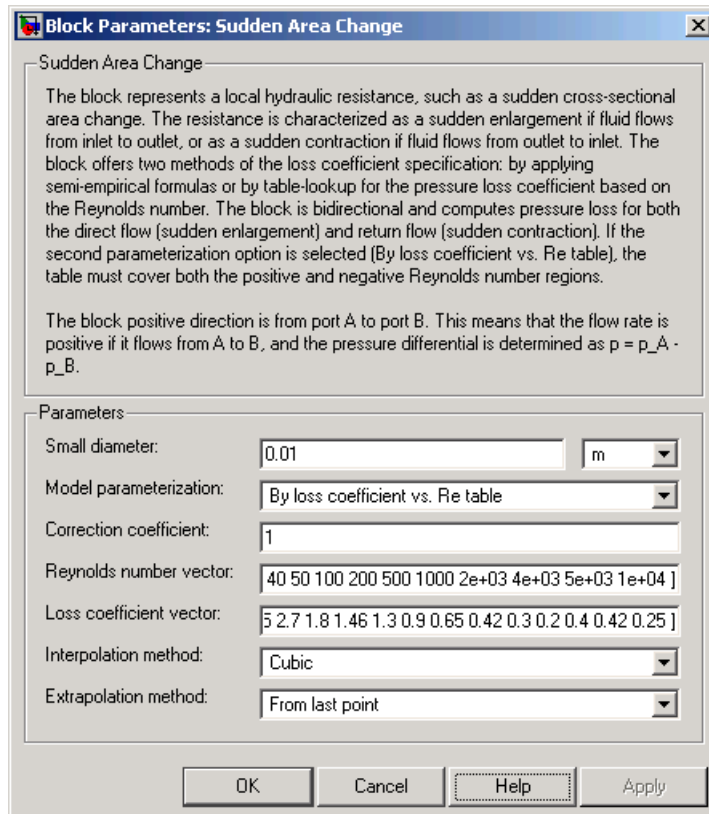
Model parameterization:

Correction coefficient:

Critical Reynolds number:

OK Cancel Help Apply

Sudden Area Change



Small diameter

Resistance small diameter. The default value is 0.01 m.

Large diameter

Resistance large diameter. The default value is 0.02 m. This parameter is used if **Model parameterization** is set to By semi-empirical formulas.

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.
- By loss coefficient vs. Re table — 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 three interpolation methods and two extrapolation methods. The table must cover both the positive and the negative flow regions.

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.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 110. This parameter is used if **Model parameterization** is set to By semi-empirical formulas.

Reynolds number vector

Specify the vector of input values for Reynolds numbers as a tabulated 1-by-m array. The input values vector must be strictly monotonically increasing. The values can be non-uniformly spaced. You must provide at least three values. 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.

Sudden Area Change

Loss coefficient vector

Specify the vector of output values for the loss coefficient as a tabulated 1-by-m array. The loss coefficient vector must be 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 — 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 **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:

- 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 **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

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

Sudden Area Change

- 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

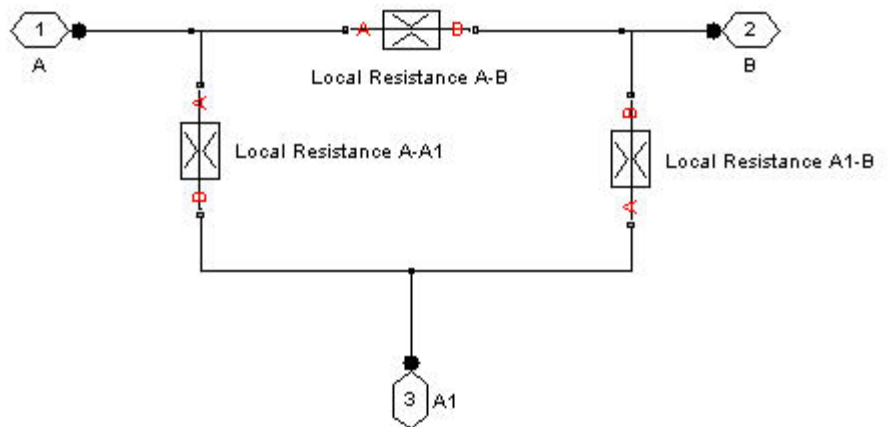
Purpose Simulate hydraulic resistance of T-junction in pipe

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.

T-junction

Basic Assumptions and Limitations

The model is based on the following assumption:

- Fluid inertia is not taken into account.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.

Dialog Box and Parameters

Block Parameters: T-junction

T-junction

The block represents a T-junction (Wye connection) consisting of a main run and a branch merging to the main run at a specified angle. The junction as a hydraulic resistance is specified by six pressure loss coefficients that characterize pressure-flow rate relationship for every possible connection for both the direct and the reverse flow.

The block positive direction is from port A to port B, from port A to port A1, and from port A1 to port B.

Parameters

Main pipe diameter:	<input type="text" value="0.01"/>	m
Branch pipe diameter:	<input type="text" value="0.01"/>	m
A-B pressure loss coefficient:	<input type="text" value="1.12"/>	
B-A pressure loss coefficient:	<input type="text" value="1.12"/>	
A-A1 pressure loss coefficient:	<input type="text" value="1.36"/>	
A1-A pressure loss coefficient:	<input type="text" value="1.65"/>	
A1-B pressure loss coefficient:	<input type="text" value="1.6"/>	
B-A1 pressure loss coefficient:	<input type="text" value="1.8"/>	
Critical Reynolds number:	<input type="text" value="120"/>	

OK Cancel Help Apply

Main pipe diameter

The internal pipe diameter of the main run. The default value is 0.01 m.

T-junction

Branch pipe diameter

The internal pipe diameter of the branch. The default value is 0.01 m.

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.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 120.

Global Parameters

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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

Valve Hydraulic Force

Purpose Simulate 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.

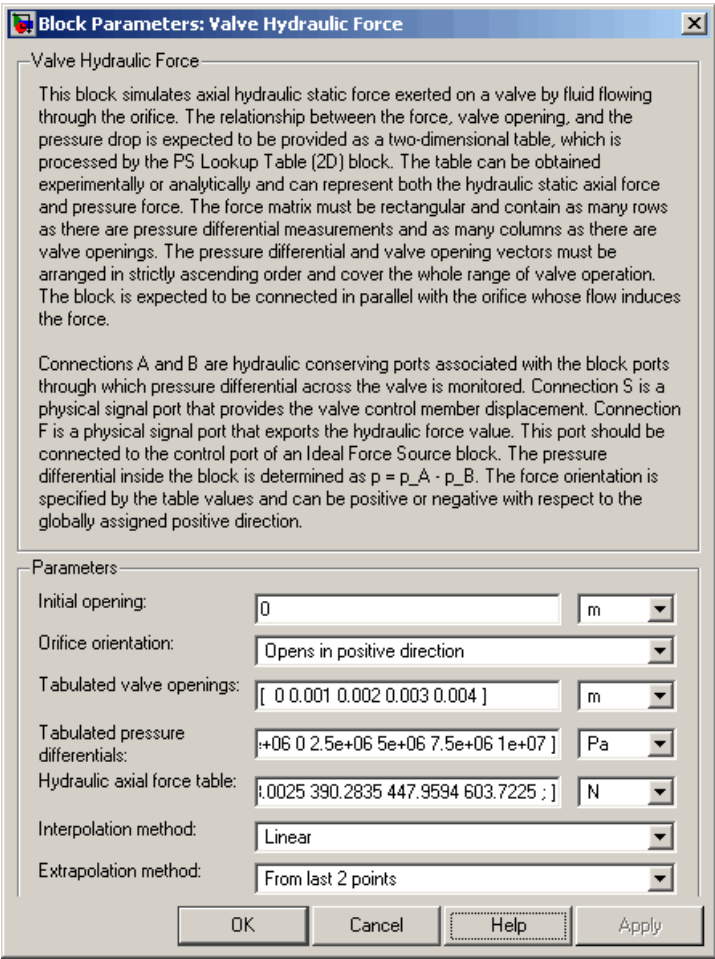
Connections A and B are hydraulic conserving ports that should be connected to the valve block ports in such a way as to monitor the pressure differential across the valve. Connection S is a physical signal port that provides the valve control member displacement. Connection F is a physical signal port 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

The model is based on the following assumption:

- 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.

Valve Hydraulic Force

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 tabulated 1-by-n array. The input values vector must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. 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 tabulated 1-by-m array. The input values vector must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. The default values, in Pa, are `[-100e5, -75e5, -50e5, -25e5, 0, 25e5, 50e5, 75e5, 100e5]`.

Hydraulic axial force table

Specify the output values for the hydraulic axial force as a tabulated m-by-n matrix, defining the function values at the input grid vertices. 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** — Uses a bilinear interpolation algorithm, which is an extension of linear interpolation for functions in two variables.
- **Cubic** — Uses the bicubic interpolation algorithm.
- **Spline** — Uses the bicubic spline interpolation algorithm.

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:

- **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.

Valve Hydraulic Force

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

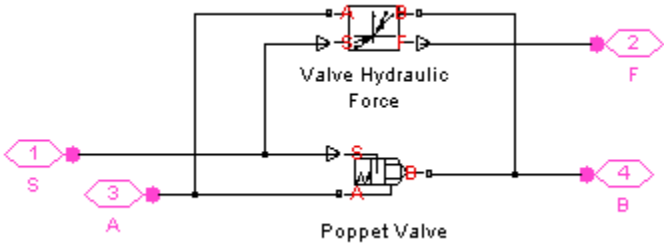
The block has the following ports:

- 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.

Valve Hydraulic Force



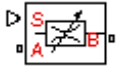
See Also Spool Orifice Hydraulic Force

Variable Orifice

Purpose Simulate 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 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}). After the

area has been determined, the flow rate is computed according to the following equations:

$$q = \begin{cases} C_D \cdot A \sqrt{\frac{2}{\rho} |p| \cdot \text{sign}(p)} & \text{for } Re \geq Re_{cr} \\ 2C_{DL} \cdot A \frac{D_H}{v \cdot \rho} p & \text{for } Re < Re_{cr} \end{cases}$$

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

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

$$p = p_A - p_B$$

$$Re = \frac{q \cdot D_H}{A(h) \cdot v}$$

$$C_{DL} = \left(\frac{C_D}{\sqrt{Re_{cr}}} \right)^2$$

$$D_H = \sqrt{\frac{4A(h)}{\pi}}$$

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

Variable Orifice

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
D_H	Instantaneous orifice hydraulic diameter
ν	Fluid kinematic viscosity
A_{leak}	Closed orifice leakage area

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 monotonically increasing. The vertices can be non-uniformly 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 or closes the orifice depending on the value of the orifice orientation indicator.

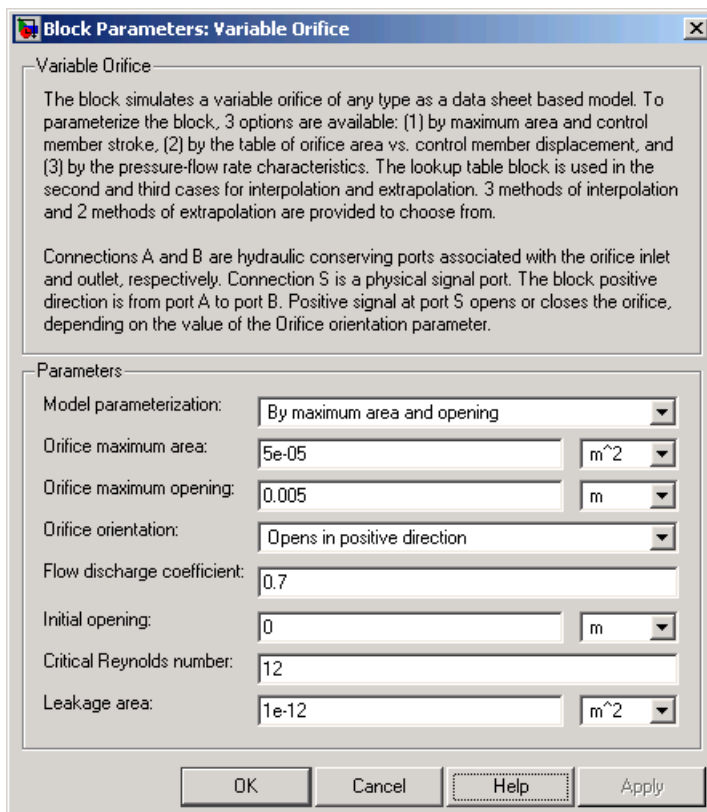
Basic Assumptions and Limitations

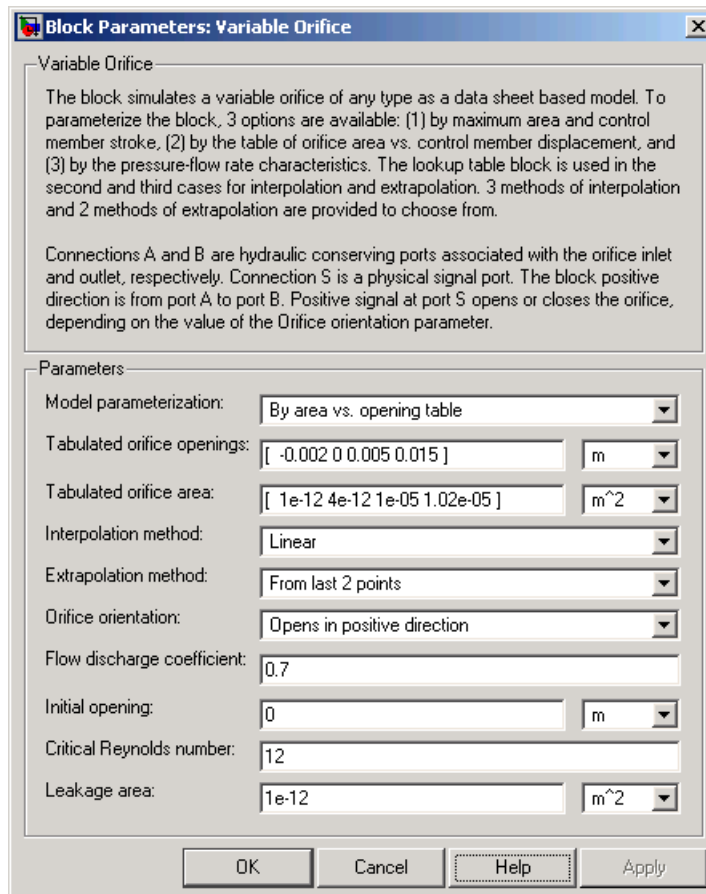
The model is based on the following assumptions:

- Fluid inertia is not taken into account.
- 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.

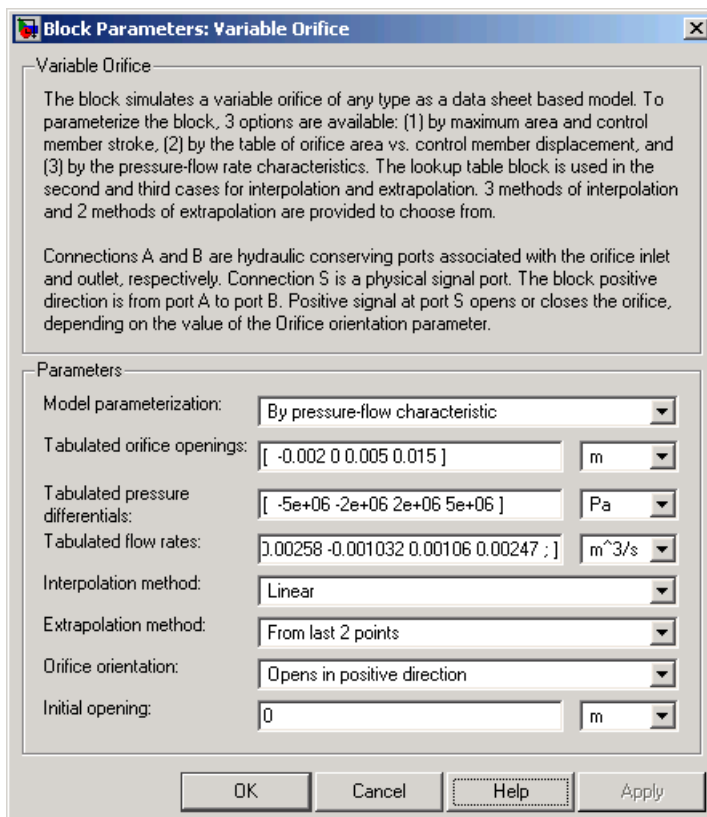
Variable Orifice

Dialog Box and Parameters





Variable Orifice



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 three 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 three 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 tabulated 1-by-m array. The input values vector must be strictly monotonically increasing. The values can be non-uniformly spaced. You must provide at least three values. The default values, in meters, are $[-2e-3, 0, 5e-3, 15e-3]$. 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.

Variable Orifice

Tabulated orifice area

Specify the vector of output values for orifice area as a tabulated 1-by-m array. The orifice area vector must be 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.

Tabulated pressure differentials

Specify the vector of input values for pressure differentials as a tabulated 1-by-n array. The input values vector must be strictly monotonically increasing. The values can be non-uniformly spaced. You must provide at least three values. The default values, in Pa, are $[-5e6, -2e6, 2e6, 5e6]$. This parameter is used if **Model parameterization** is set to By pressure-flow characteristic.

Tabulated flow rates

Specify the output values for flow rates as a tabulated m-by-n matrix, defining the function values at the input grid vertices. 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-12,  -0.6e-13,   0.6e13,    0.1e12; ...  
  -0.8e-7, -0.4e-7,   0.4e7,     0.8e7; ...  
  -2.5e-3, -1.03e-3,  1.e-3,     2.4e-3; ...  
  -2.58e-3, -1.032e-3, 1.06e-3,  2.47e-3]
```

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** — For one-dimensional table lookup (By area vs. opening table), uses a linear interpolation function. For two-dimensional table lookup (By pressure-flow characteristic), uses a bilinear interpolation algorithm, which is an extension of linear interpolation for functions in two variables.
- **Cubic** — For one-dimensional table lookup (By area vs. opening table), uses the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP). For two-dimensional table lookup (By pressure-flow characteristic), uses the bicubic interpolation algorithm.
- **Spline** — For one-dimensional table lookup (By area vs. opening table), uses the cubic spline interpolation algorithm. For two-dimensional table lookup (By pressure-flow characteristic), uses the bicubic spline interpolation algorithm.

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:

- **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

Variable Orifice

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) 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.

Critical Reynolds number

The maximum Reynolds number for laminar flow. 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 orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

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. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 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**
- **Orifice orientation**
- **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

Fluid density

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

Variable Orifice

- 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 demo (`sh_hydraulic_flapper_nozzle_amplifier`) illustrates the use of the Variable Orifice block in hydraulic systems.

See Also

- Annular Orifice
- Constant Area Orifice
- Fixed Orifice
- Orifice with Variable Area Round Holes
- Orifice with Variable Area Slot
- PS Lookup Table (1D)
- PS Lookup Table (2D)
- Variable Area Orifice

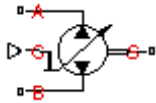
Purpose

Simulate variable-displacement reversible hydraulic motor

Library

Pumps and Motors

Description



The Variable-Displacement Motor block represents a variable-displacement reversible motor of any type as a data-sheet-based model. The motor displacement is controlled by the signal provided through the physical signal port C. The motor efficiency is determined based on volumetric and total efficiencies, nominal pressure, and nominal angular velocity. All these parameters are generally provided in the data sheets or catalogs.

Two block parameterization options are available:

- By the motor 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 motor 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 variable-displacement motor is represented with the following equations:

$$q = D \cdot \omega - k_{leak} \cdot p$$

$$T = D \cdot p \cdot \eta_{mech}$$

$$D = \begin{cases} \frac{D_{max}}{x_{max}} \cdot x \\ D(x) \end{cases}$$

$$k_{leak} = k_{HP} / v \cdot \rho$$

Variable-Displacement Motor

$$k_{HP} = \frac{D \cdot \omega_{nom} (1 - \eta_V) \cdot v_{nom} \cdot \rho}{p_{nom}}$$

$$p = p_A - p_B$$

where

q	Motor flow rate
p	Pressure differential across the motor
p_A, p_B	Gauge pressures at the block terminals
D	Motor instantaneous displacement
D_{max}	Motor maximum displacement
x	Control member displacement
x_{max}	Control member maximum stroke
T	Torque at the motor output shaft
ω	Output shaft angular velocity
k_{leak}	Leakage coefficient
k_{HP}	Hagen-Poiseuille coefficient
η_V	Motor volumetric efficiency
η_{mech}	Motor mechanical efficiency
ν	Fluid kinematic viscosity
ρ	Fluid density
p_{nom}	Motor nominal pressure
ω_{nom}	Motor 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

$$q_{leak} = D\omega_{nom} (1 - \eta_V)$$

which provides the formula to determine the Hagen-Poiseuille coefficient

$$k_{HP} = \frac{D\omega_{nom} (1 - \eta_V) \cdot \nu_{nom} \cdot \rho}{p_{nom}}$$

The motor 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 A to port B. This means that the motor rotates its shaft in the globally assigned positive direction if the fluid flows from port A to port B and a positive signal is applied to port C.

Variable-Displacement Motor

Basic Assumptions and Limitations

The model is based on the following assumptions:

- Fluid compressibility is neglected.
- No loading on the motor shaft, such as inertia, friction, spring, and so on, is considered.
- Leakage inside the motor is assumed to be linearly proportional to its pressure differential.

Variable-Displacement Motor

Dialog Box and Parameters

Block Parameters: Variable-Displacement Motor

Variable-Displacement Motor

This block represents a variable-displacement reversible motor of any type as a data sheet-based model. The model can be parameterized either by the motor maximum displacement and control member stroke or by the tabulated relationship between motor displacement and control member position. In the first case, the displacement is assumed to be linearly dependant on control member position. The motor efficiency is determined on a basis of volumetric and total efficiencies, nominal pressure, and nominal angular velocity.

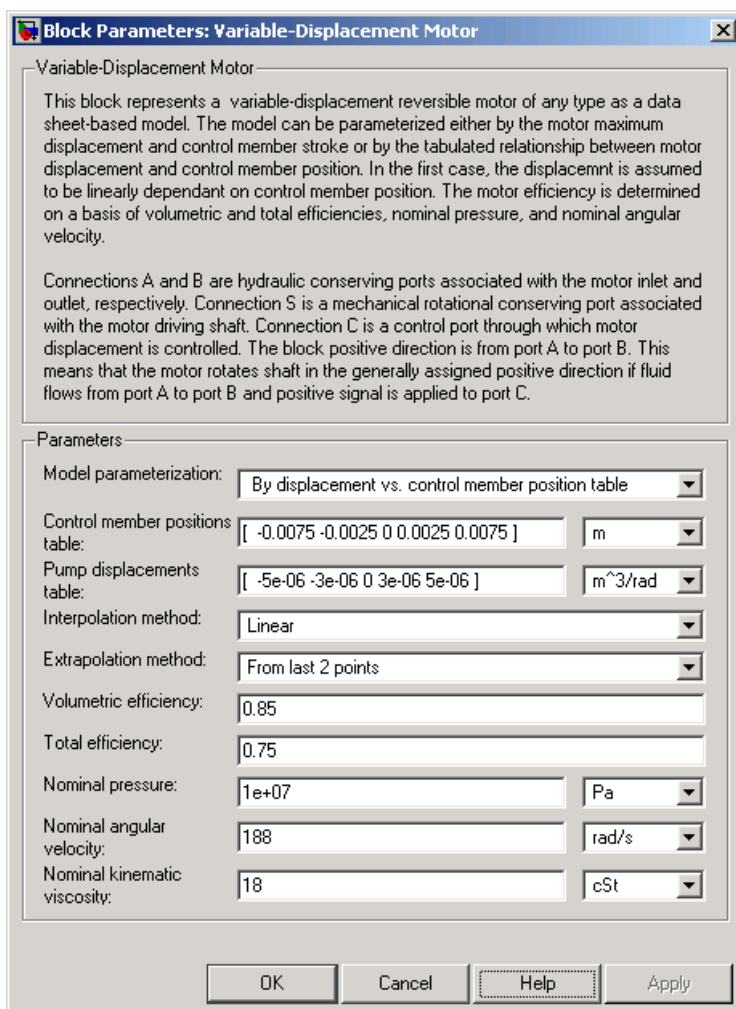
Connections A and B are hydraulic conserving ports associated with the motor inlet and outlet, respectively. Connection S is a mechanical rotational conserving port associated with the motor driving shaft. Connection C is a control port through which motor displacement is controlled. The block positive direction is from port A to port B. This means that the motor rotates shaft in the generally assigned positive direction if fluid flows from port A to port B and positive signal is applied to port C.

Parameters

Model parameterization:	By maximum displacement and control member stroke	
Maximum displacement:	5e-06	m ³ /rad
Maximum stroke:	0.005	m
Volumetric efficiency:	0.85	
Total efficiency:	0.75	
Nominal pressure:	1e+07	Pa
Nominal angular velocity:	188	rad/s
Nominal kinematic viscosity:	18	cSt

OK Cancel Help Apply

Variable-Displacement Motor



Model parameterization

Select one of the following block parameterization options:

- By maximum displacement and control member stroke
— Provide values for maximum motor 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 motor 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

Motor maximum displacement. The default value is $5e-6$ m³/rad. This parameter is used if **Model parameterization** is set to By maximum displacement and control member stroke.

Maximum stroke

Maximum control member stroke. The default value is 0.005 m. This parameter is used if **Model parameterization** is set to By maximum displacement and control member stroke.

Control member positions table

Specify the vector of input values for control member position as a tabulated 1-by-m array. The input values vector must be strictly monotonically increasing. The values can be non-uniformly spaced. You must provide at least three values. The default values, in meters, are [-0.0075 -0.0025 0 0.0025 0.0075]. This parameter is used if **Model parameterization** is set to By displacement vs. control member position table.

Pump displacements table

Specify the vector of output values for the motor displacement as a tabulated 1-by-m array. The motor displacements vector must be the same size as the control member positions vector. The default values, in m³/rad, are [-5e-06 -3e-06 0 3e-06 5e-06]. This parameter is used if **Model parameterization** is set to By displacement vs. control member position table.

Variable-Displacement Motor

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 **Model parameterization** is set to 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 **Model parameterization** is set to By displacement vs. control member position table.

Volumetric efficiency

Motor volumetric efficiency specified at nominal pressure, angular velocity, and fluid viscosity. The default value is 0.85.

Total efficiency

Motor total efficiency, which is determined as a ratio between the hydraulic power at the motor inlet and mechanical power at the output shaft at nominal pressure, angular velocity, and fluid viscosity. The default value is 0.75.

Nominal pressure

Pressure differential across the motor, at which both the volumetric and total efficiencies are specified. The default value is 1×10^7 Pa.

Nominal angular velocity

Angular velocity of the output 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.

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.

Variable-Displacement Motor

Global Parameters

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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 motor inlet.

B

Hydraulic conserving port associated with the motor outlet.

C

Physical signal port that controls motor displacement.

S

Mechanical rotational conserving port associated with the motor output shaft.

See Also

Hydraulic Motor

Variable-Displacement Pressure-Compensated Pump

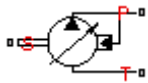
Purpose

Simulate hydraulic pump maintaining preset pressure at outlet by regulating its flow delivery

Library

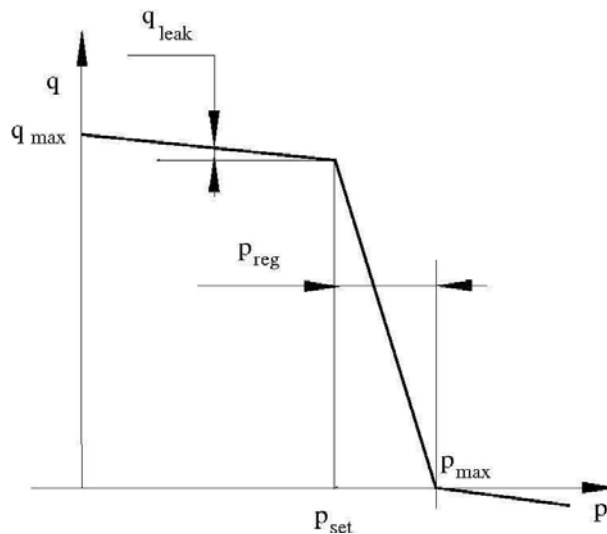
Pumps and Motors

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

Variable-Displacement Pressure-Compensated Pump

the pressure setting has been reached, the output flow is regulated to maintain preset pressure by changing the pump's displacement. The displacement can be changed from its maximum value down to zero, depending upon system flow requirements. The pressure range between the preset pressure and the maximum pressure, at which the displacement is zero, is referred to as regulation range. The smaller the range, the higher the accuracy at which preset pressure is maintained. The range size also affects the pump stability, and decreasing the range generally causes stability to decrease.

The variable-displacement, pressure-compensated pump is represented with the following equations:

$$q = D \cdot \omega - k_{leak} \cdot p$$

$$T = D \cdot p / \eta_{mech}$$

$$D = \begin{cases} D_{max} & \text{for } p \leq p_{set} \\ D_{max} - K(p - p_{set}) & \text{for } p_{set} < p < p_{max} \\ 0 & \text{for } p \geq p_{max} \end{cases}$$

$$p_{max} = p_{set} + p_{reg}$$

$$K = D_{max} / (p_{max} - p_{set})$$

$$k_{leak} = k_{HP} / v \cdot \rho$$

$$k_{HP} = \frac{D \cdot \omega_{nom} (1 - \eta_V) \cdot v_{nom} \cdot \rho}{p_{nom}}$$

$$p = p_P - p_T$$

where

Variable-Displacement Pressure-Compensated Pump

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
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

Variable-Displacement Pressure-Compensated Pump

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

$$q_{leak} = D\omega_{nom}(1 - \eta_V)$$

which provides the formula to determine the Hagen-Poiseuille coefficient

$$k_{HP} = \frac{D\omega_{nom}(1 - \eta_V) \cdot \nu_{nom} \cdot \rho}{p_{nom}}$$

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 $p = p_P - p_T$.

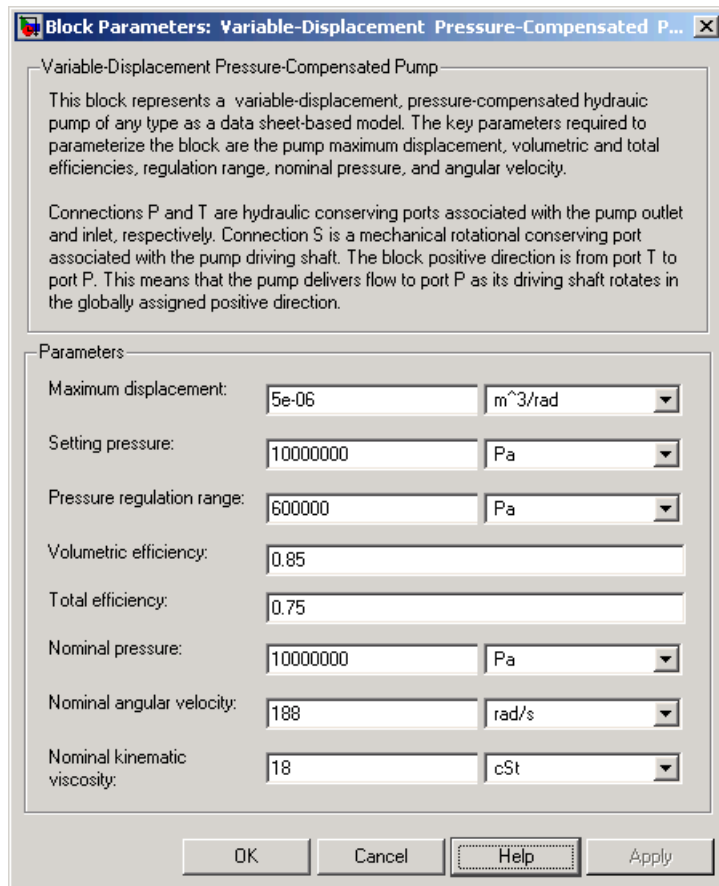
Basic Assumptions and Limitations

The model is based on the following assumptions:

- 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.

Variable-Displacement Pressure-Compensated Pump

Dialog Box and Parameters



Block Parameters: Variable-Displacement Pressure-Compensated P...

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.

Parameters

Maximum displacement:	5e-06	m ³ /rad
Setting pressure:	10000000	Pa
Pressure regulation range:	600000	Pa
Volumetric efficiency:	0.85	
Total efficiency:	0.75	
Nominal pressure:	10000000	Pa
Nominal angular velocity:	188	rad/s
Nominal kinematic viscosity:	18	cSt

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.

Variable-Displacement Pressure-Compensated Pump

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 1×10^7 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.

Global Parameters

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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.

Variable-Displacement Pressure-Compensated Pump

Examples

The Closed-Loop Electrohydraulic Actuator with Proportional Valve demo (sh_closed_loop_actuator) illustrates the use of the Variable-Displacement Pressure-Compensated Pump block in hydraulic systems.

See Also

Centrifugal Pump

Fixed-Displacement Pump

Variable-Displacement Pump

Variable-Displacement Pump

Purpose Simulate variable-displacement reversible hydraulic pump

Library Pumps and Motors

Description



The Variable-Displacement Pump block represents a variable-displacement reversible pump of any type as a data-sheet-based model. The pump delivery is proportional to the control signal provided through the physical signal port C. The pump efficiency is determined based on volumetric and total efficiencies, nominal pressure, and angular velocity. All these parameters are generally provided in the data sheets or catalogs.

Two block parameterization options are available:

- By the pump 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 pump 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 variable-displacement pump is represented with the following equations:

$$q = D \cdot \omega - k_{leak} \cdot p$$

$$T = D \cdot p / \eta_{mech}$$

$$D = \begin{cases} D_{max} \cdot x \\ x_{max} \\ D(x) \end{cases}$$

$$k_{leak} = k_{HP} / v \cdot \rho$$

Variable-Displacement Pump

$$k_{HP} = \frac{D \cdot \omega_{nom} (1 - \eta_V) \cdot v_{nom} \cdot \rho}{P_{nom}}$$

$$p = p_P - p_T$$

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
x	Control member displacement
x_{max}	Control member maximum stroke
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
p_{nom}	Pump nominal pressure
ω_{nom}	Pump nominal angular velocity
ν_{nom}	Nominal fluid kinematic viscosity

Variable-Displacement Pump

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

$$q_{leak} = D\omega_{nom} (1 - \eta_V)$$

which provides the formula to determine the Hagen-Poiseuille coefficient

$$k_{HP} = \frac{D\omega_{nom} (1 - \eta_V) \cdot \nu_{nom} \cdot \rho}{p_{nom}}$$

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

$$\eta_{mech} = \eta_{total} / \eta_V$$

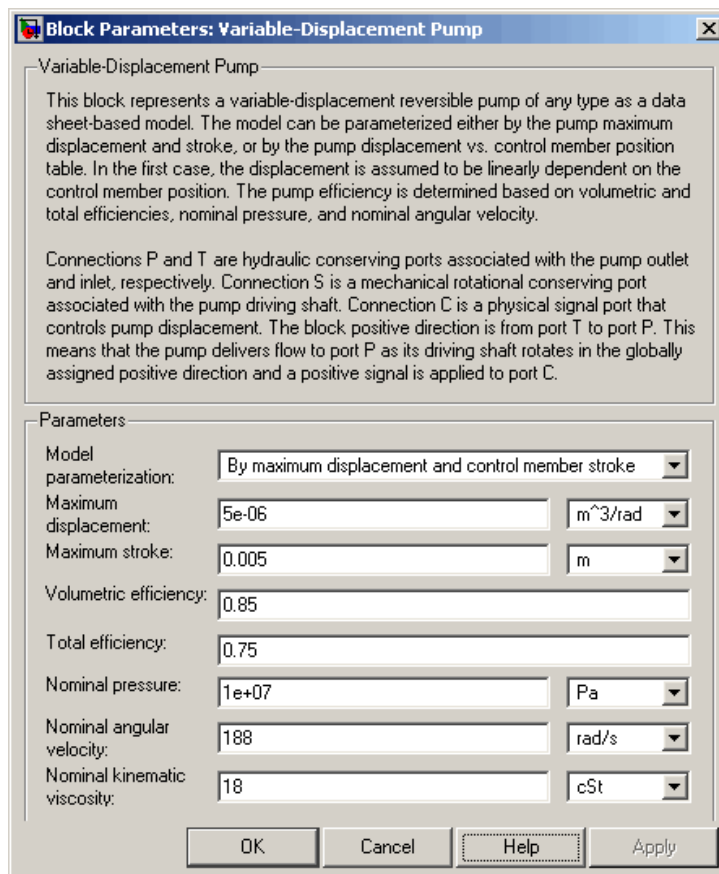
The block positive direction is from port T to port P. This means that the pump transfers fluid from T to P as its driving shaft S rotates in the globally assigned positive direction and a positive signal is applied to port C.

Basic Assumptions and Limitations

The model is based on the following assumptions:

- 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 Pump

Variable-Displacement Pump

This block represents a variable-displacement reversible pump of any type as a data sheet-based model. The model can be parameterized either by the pump maximum displacement and stroke, or by the pump displacement vs. control member position table. In the first case, the displacement is assumed to be linearly dependent on the control member position. The pump efficiency is determined based on volumetric and total efficiencies, nominal pressure, and nominal angular velocity.

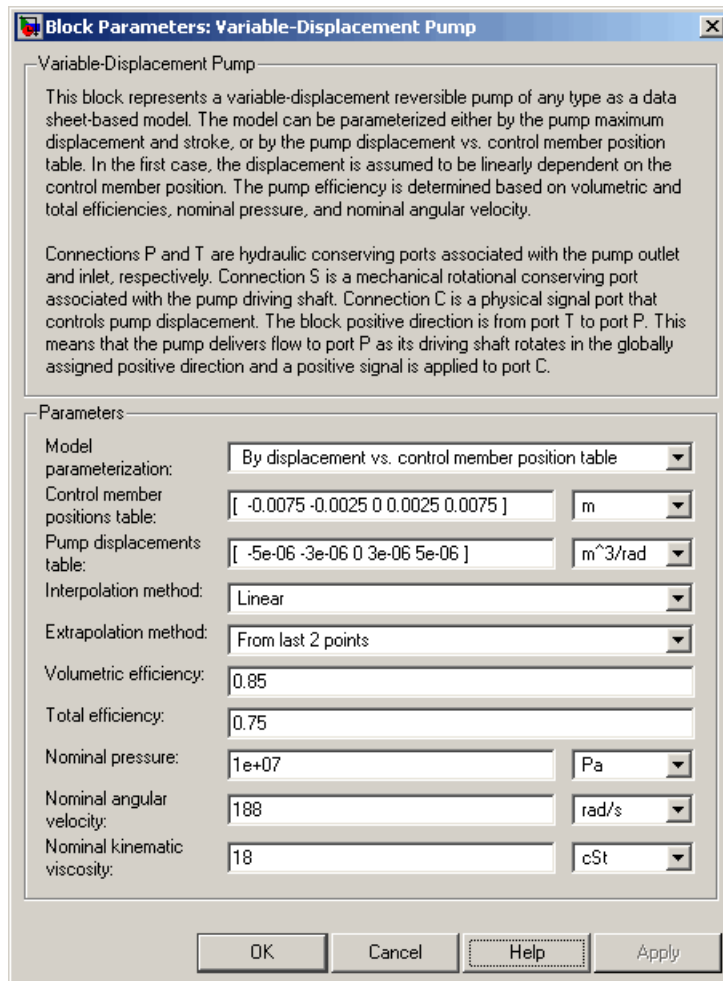
Connections P and T are hydraulic conserving ports associated with the pump outlet and inlet, respectively. Connection S is a mechanical rotational conserving port associated with the pump driving shaft. Connection C is a physical signal port that controls pump displacement. The block positive direction 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 and a positive signal is applied to port C.

Parameters

Model parameterization:	By maximum displacement and control member stroke	
Maximum displacement:	5e-06	m ³ /rad
Maximum stroke:	0.005	m
Volumetric efficiency:	0.85	
Total efficiency:	0.75	
Nominal pressure:	1e+07	Pa
Nominal angular velocity:	188	rad/s
Nominal kinematic viscosity:	18	cSt

OK Cancel Help Apply

Variable-Displacement Pump



Model parameterization

Select one of the following block parameterization options:

- By maximum displacement and control member stroke — Provide values for maximum pump displacement and maximum

control member 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 pump 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

Pump maximum displacement. The default value is $5e-6$ m^3/rad . This parameter is used if **Model parameterization** is set to By maximum displacement and control member stroke.

Maximum stroke

Maximum control member stroke. The default value is 0.005 m. This parameter is used if **Model parameterization** is set to By maximum displacement and control member stroke.

Control member positions table

Specify the vector of input values for control member position as a tabulated 1-by-m array. The input values vector must be strictly monotonically increasing. The values can be non-uniformly spaced. You must provide at least three values. The default values, in meters, are [-0.0075 -0.0025 0 0.0025 0.0075]. This parameter is used if **Model parameterization** is set to By displacement vs. control member position table.

Pump displacements table

Specify the vector of output values for the pump displacement as a tabulated 1-by-m array. The pump displacements vector must be 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 **Model parameterization** is set to By displacement vs. control member position table.

Variable-Displacement Pump

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 **Model parameterization** is set to 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 **Model parameterization** is set to By displacement vs. control member position table.

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 $1 \text{e}7 \text{ Pa}$.

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.

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.

Variable-Displacement Pump

Global Parameters

Fluid kinematic viscosity

The parameter is determined by the type of working fluid selected for the system under design. 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.

C

Physical signal port that controls pump displacement.

S

Mechanical rotational conserving port associated with the pump driving shaft.

See Also

Centrifugal Pump

Fixed-Displacement Pump

Variable-Displacement Pressure-Compensated Pump

2-Position Valve Actuator block 2-2
2-Way Directional Valve block 2-8
3-Position Valve Actuator block 2-18
3-Way Directional Valve block 2-24
4-Way Directional Valve block 2-36

A

accumulators
 gas-charged 2-107
 spring-loaded 2-266
Annular Orifice block 2-49

B

Ball Valve block 2-54

C

Centrifugal Pump block 2-60
Check Valve block 2-67
Cylinder Friction block 2-73

D

Double-Acting Hydraulic Cylinder block 2-79
Double-Acting Rotary Actuator block 2-85

E

Elbow block 2-91

F

Fixed Orifice block 2-97
Fixed-Displacement Pump block 2-101

G

Gas-Charged Accumulator block 2-107
Gradual Area Change block 2-111

H

Hydraulic Double-Acting Valve Actuator
 block 2-120
Hydraulic Fluid block 2-129
Hydraulic Motor block 2-136
Hydraulic Pipeline block 2-141
Hydraulic Single-Acting Valve Actuator
 block 2-149

L

Local Resistance block 2-154

N

Needle Valve block 2-163

O

Orifice with Variable Area Round Holes
 block 2-169
Orifice with Variable Area Slot block 2-176

P

Pilot-Operated Check Valve block 2-182
Pipe Bend block 2-189
Poppet Valve block 2-195
Pressure Compensator block 2-210
Pressure Reducing Valve block 2-217
Pressure Relief Valve block 2-222
Pressure-Compensated Flow Control Valve
 block 2-201
Proportional and Servo-Valve Actuator
 block 2-228

R

Reservoir block 2-232

S

Segmented Pipeline block 2-235
Shuttle Valve block 2-242
Single-Acting Hydraulic Cylinder block 2-249
Single-Acting Rotary Actuator block 2-254
Spool Orifice Hydraulic Force block 2-260
Spring-Loaded Accumulator block 2-266
Sudden Area Change block 2-269

T

T-junction block 2-277

V

Valve Hydraulic Force block 2-282
Variable Orifice block 2-288
Variable-Displacement Motor block 2-301
Variable-Displacement Pressure-Compensated
Pump block 2-311
Variable-Displacement Pump block 2-318