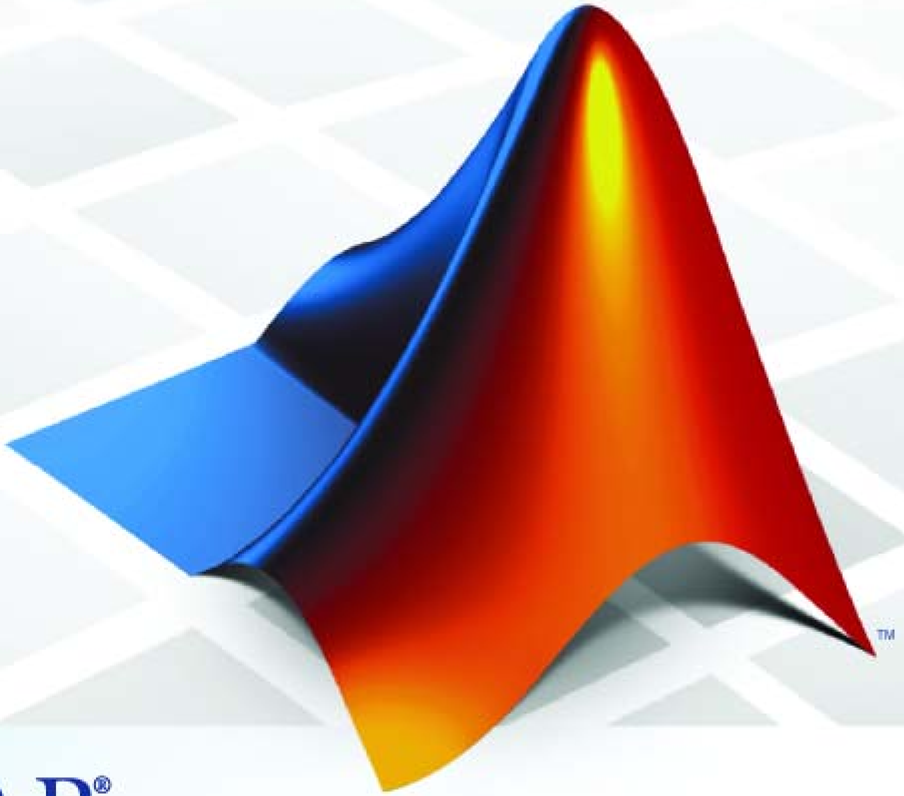
SimHydraulics® 1

Reference



MATLAB®
& SIMULINK®

The MathWorks™
Accelerating the pace of engineering and science

How to Contact The MathWorks

`www.mathworks.com`	Web
`comp.soft-sys.matlab`	Newsgroup
`www.mathworks.com/contact_TS.html`	Technical Support
`suggest@mathworks.com`	Product enhancement suggestions
`bugs@mathworks.com`	Bug reports
`doc@mathworks.com`	Documentation error reports
`service@mathworks.com`	Order status, license renewals, passcodes
`info@mathworks.com`	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 2006–2008 by The MathWorks, Inc.

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

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

Trademarks

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

Patents

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	New for Version 1.0 (Release 2006a+)
September 2006	Online only	Revised for Version 1.1 (Release 2006b)
March 2007	Online only	Revised for Version 1.2 (Release 2007a)
September 2007	Online only	Revised for Version 1.2.1 (Release 2007b)
March 2008	Online only	Revised for Version 1.3 (Release 2008a)
October 2008	Online only	Revised for Version 1.4 (Release 2008b)

Contents

1 Block Reference

2 Blocks — Alphabetical List

Index

Block Reference

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 Hydraulic Machine	Simulate variable-displacement reversible hydraulic machine with regime-dependable efficiency
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
Cartridge Valve Insert	Simulate hydraulic cartridge valve insert
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 Cartridge Valve Actuator	Simulate double-acting hydraulic actuator for cartridge 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

2

Blocks — Alphabetical List

2-Position Valve Actuator

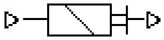
Purpose

Simulate actuator for two-position valves

Library

Valve Actuators

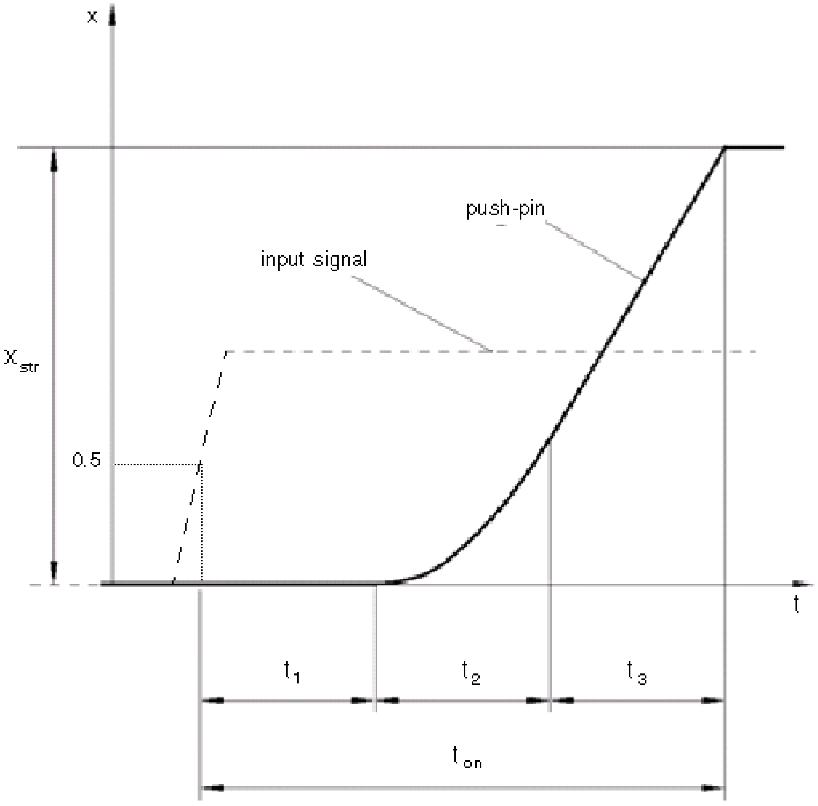
Description



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:



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 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.

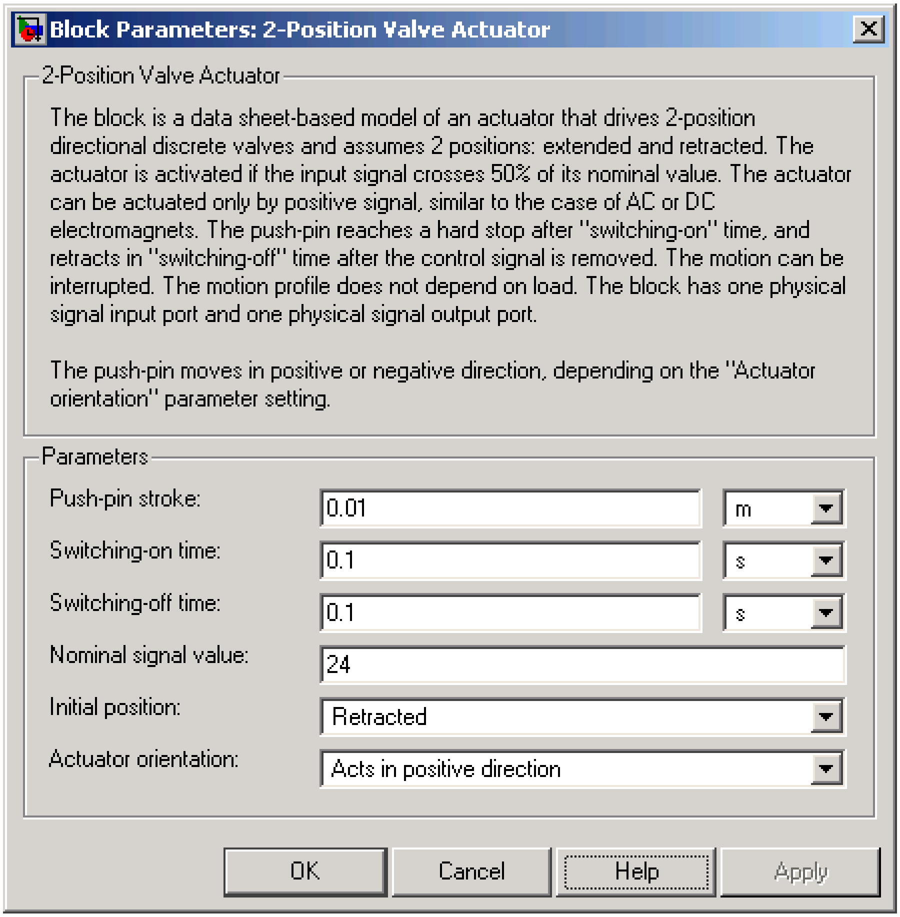
The push-pin is actuated only by positive signal, similar to the AC or DC electromagnets. The direction of push-pin motion is controlled by the **Actuator orientation** parameter, which can have one of two values: `Acts in positive direction` or `Acts in negative direction`.

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



2-Position Valve Actuator

The block is a data sheet-based model of an actuator that drives 2-position directional discrete valves and assumes 2 positions: extended and retracted. The actuator is activated if the input signal crosses 50% of its nominal value. The actuator can be actuated only by positive signal, similar to the case of AC or DC electromagnets. The push-pin reaches a hard stop after "switching-on" time, and retracts in "switching-off" time after the control signal is removed. The motion can be interrupted. The motion profile does not depend on load. The block has one physical signal input port and one physical signal output port.

The push-pin moves in positive or negative direction, depending on the "Actuator orientation" parameter setting.

Parameters

Push-pin stroke:	0.01	m
Switching-on time:	0.1	s
Switching-off time:	0.1	s
Nominal signal value:	24	
Initial position:	Retracted	
Actuator orientation:	Acts in positive direction	

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

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

Switching-off time

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

Nominal signal value

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

Initial position

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

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

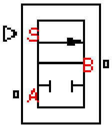
Purpose

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.

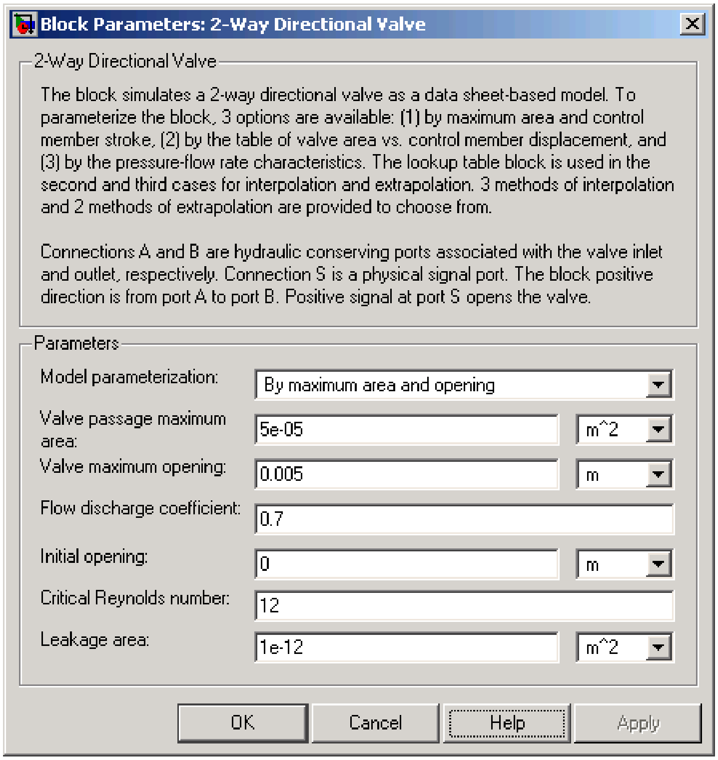
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as $p = p_A - p_B$. Positive signal at the physical signal port `S` opens the valve.

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: By maximum area and opening

Valve passage maximum area: 5e-05 m^2

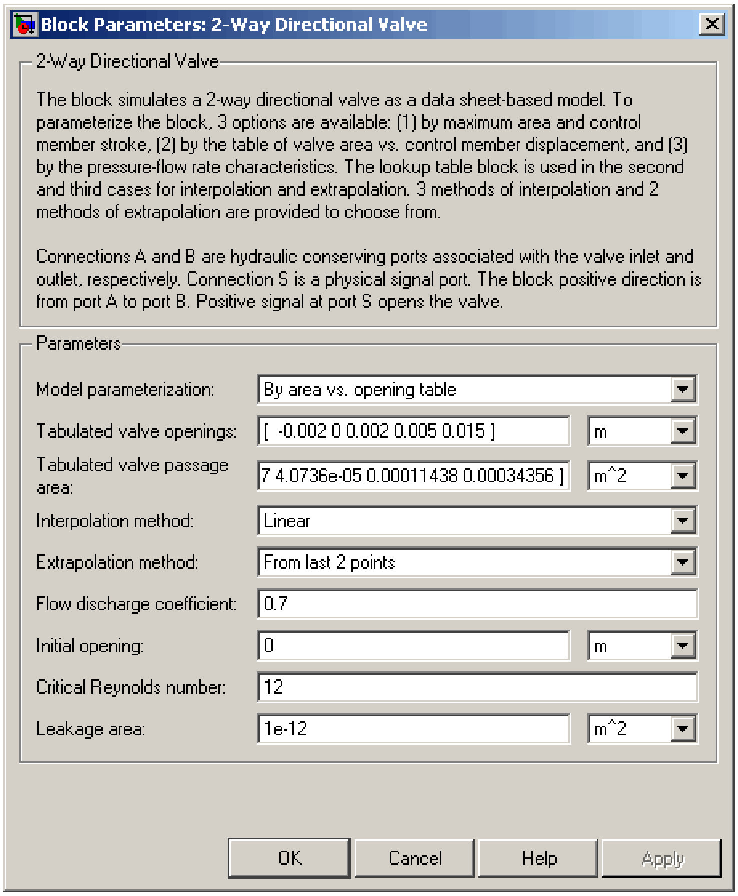
Valve maximum opening: 0.005 m

Flow discharge coefficient: 0.7

Initial opening: 0 m

Critical Reynolds number: 12

Leakage area: 1e-12 m^2



Block Parameters: 2-Way Directional Valve

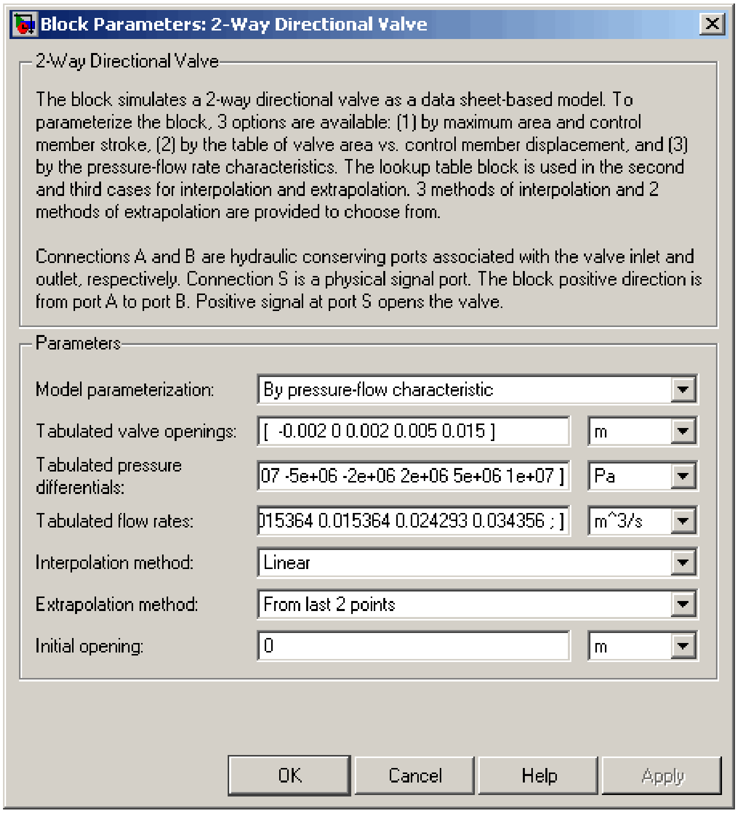
2-Way Directional Valve

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 0.002 0.005 0.015] m
Tabulated valve passage area: 7 4.0736e-05 0.00011438 0.00034356] m^2
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 m^2



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 pressure-flow characteristic

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

Tabulated pressure differentials: 07 -5e+06 -2e+06 2e+06 5e+06 1e+07] Pa

Tabulated flow rates: 015364 0.015364 0.024293 0.034356 ;] m^3/s

Interpolation method: Linear

Extrapolation method: From last 2 points

Initial opening: 0 m

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` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve maximum opening

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

Tabulated valve openings

Specify the vector of input values for valve openings as a 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 `[-0.002 0 0.002 0.005 0.015]`. If **Model parameterization** is set to `By area vs. opening table`, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to `By pressure-flow characteristic`, the **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Tabulated valve passage area

Specify the vector of output values for valve passage area as a 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-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Model parameterization** is set to `By area vs.  opening table`.

Tabulated pressure differentials

Specify the vector of input values for pressure differentials as a 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 `[-1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]`. This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Tabulated flow rates

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

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

Interpolation method

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

- `Linear` — 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 Polinomial (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`.

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` 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

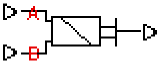
Purpose

Simulate actuator for three-position valves

Library

Valve Actuators

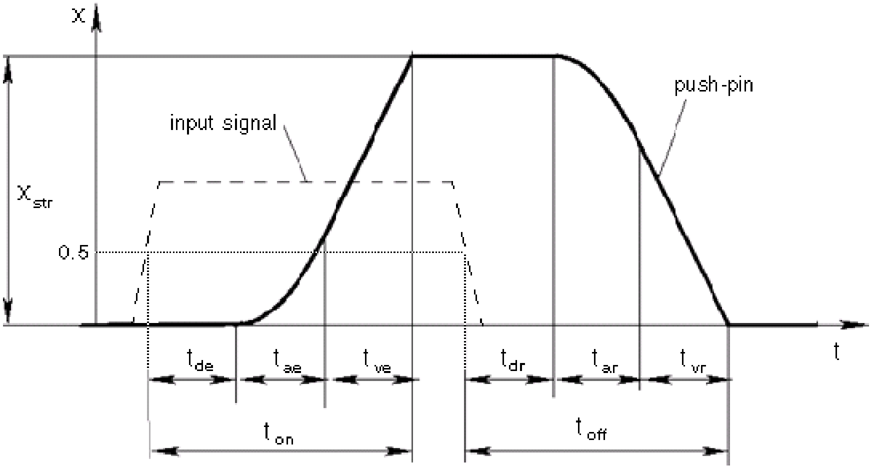
Description



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:



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

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

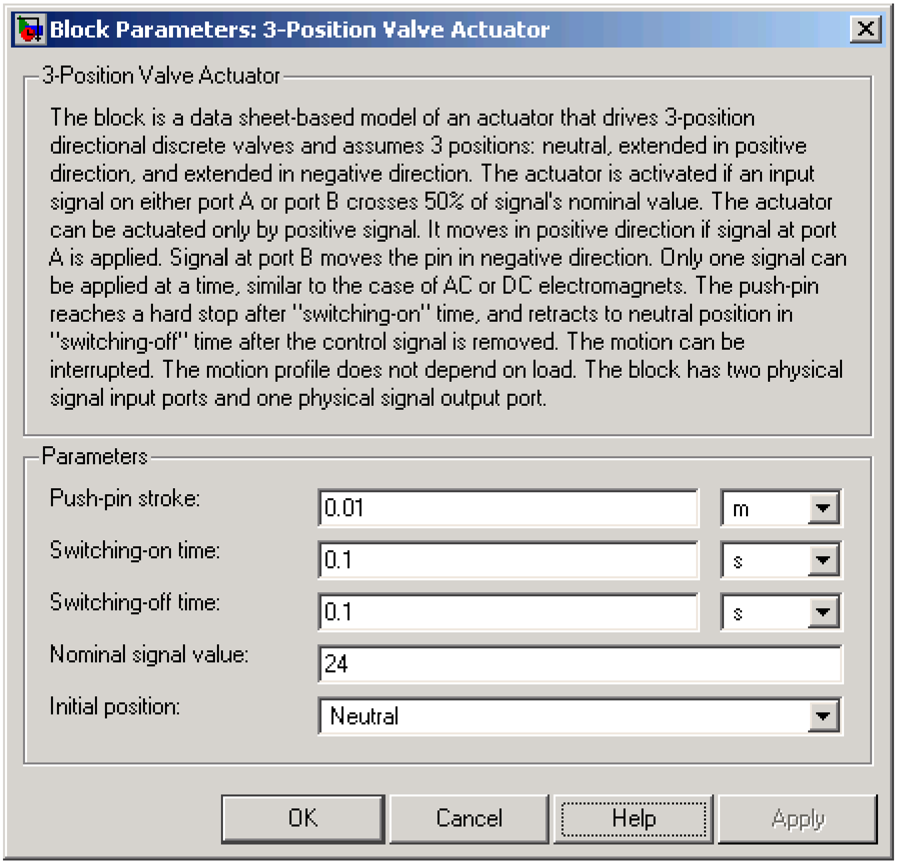
Only positive signals activate the actuator. In other words, negative signals at ports A and B have no effect on the actuator, which is similar to the behavior of electromagnetically controlled 3-position directional valves.

Basic Assumptions and Limitations

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



3-Position Valve Actuator

The block is a data sheet-based model of an actuator that drives 3-position directional discrete valves and assumes 3 positions: neutral, extended in positive direction, and extended in negative direction. The actuator is activated if an input signal on either port A or port B crosses 50% of signal's nominal value. The actuator can be actuated only by positive signal. It moves in positive direction if signal at port A is applied. Signal at port B moves the pin in negative direction. Only one signal can be applied at a time, similar to the case of AC or DC electromagnets. The push-pin reaches a hard stop after "switching-on" time, and retracts to neutral position in "switching-off" time after the control signal is removed. The motion can be interrupted. The motion profile does not depend on load. The block has two physical signal input ports and one physical signal output port.

Parameters

Push-pin stroke: 0.01 m

Switching-on time: 0.1 s

Switching-off time: 0.1 s

Nominal signal value: 24

Initial position: Neutral

Push-pin stroke

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

Switching-on time

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

Switching-off time

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

Nominal signal value

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

Initial position

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

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

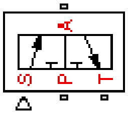
Hydraulic Single-Acting Valve Actuator

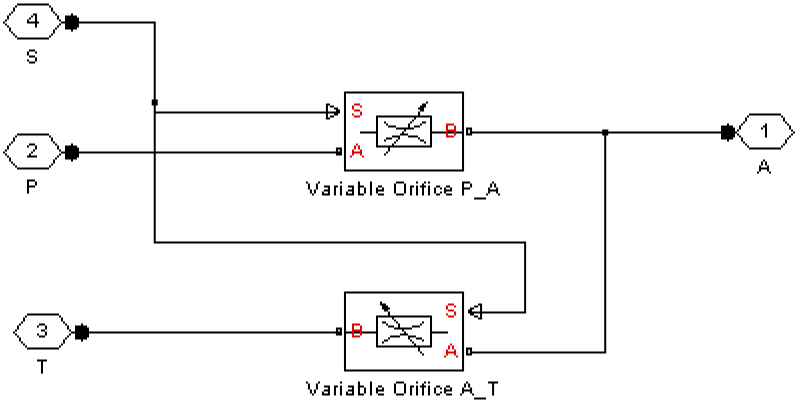
Proportional and Servo-Valve Actuator

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 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.

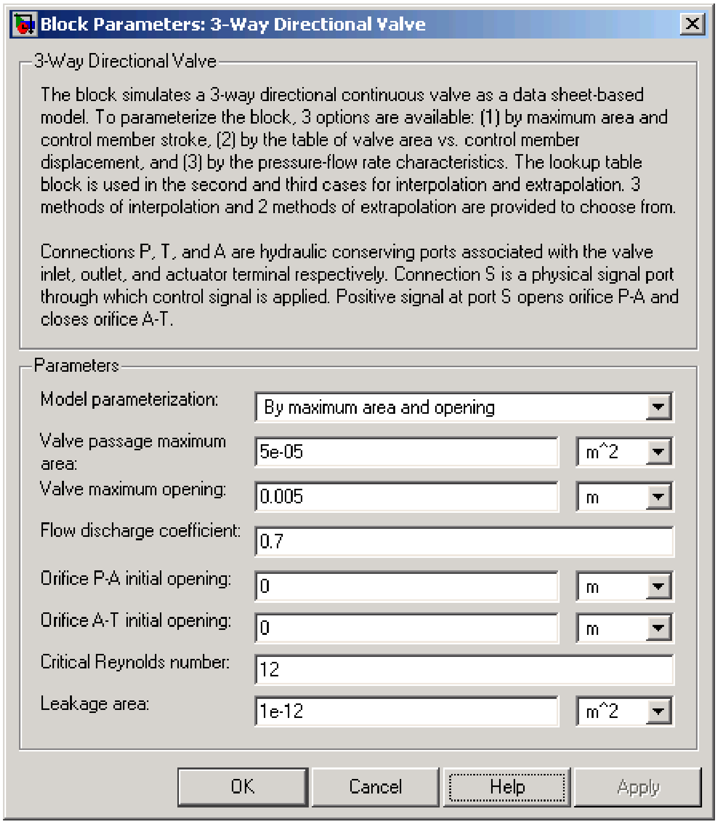
Positive signal at the physical signal port `S` opens the orifice in the `P-A` path and closes the orifice in the `A-T` path. The directionality of nested blocks is clear from the schematic diagram.

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

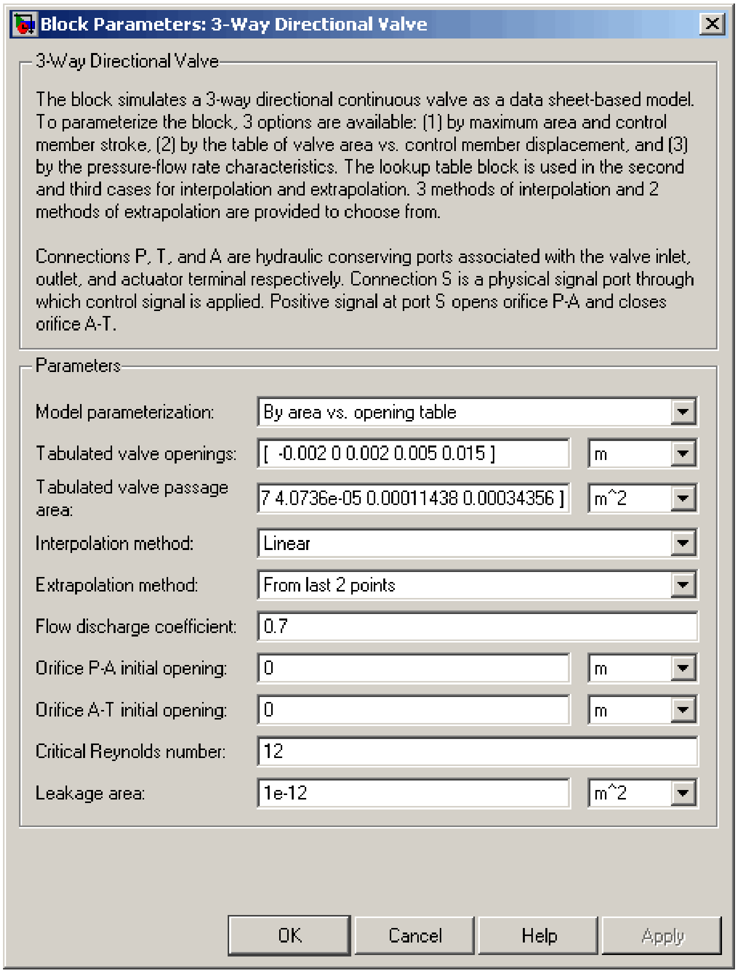
3-Way Directional Valve

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



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 area vs. opening table

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

Tabulated valve passage area: 7 4.0736e-05 0.00011438 0.00034356] m^2

Interpolation method: Linear

Extrapolation method: From last 2 points

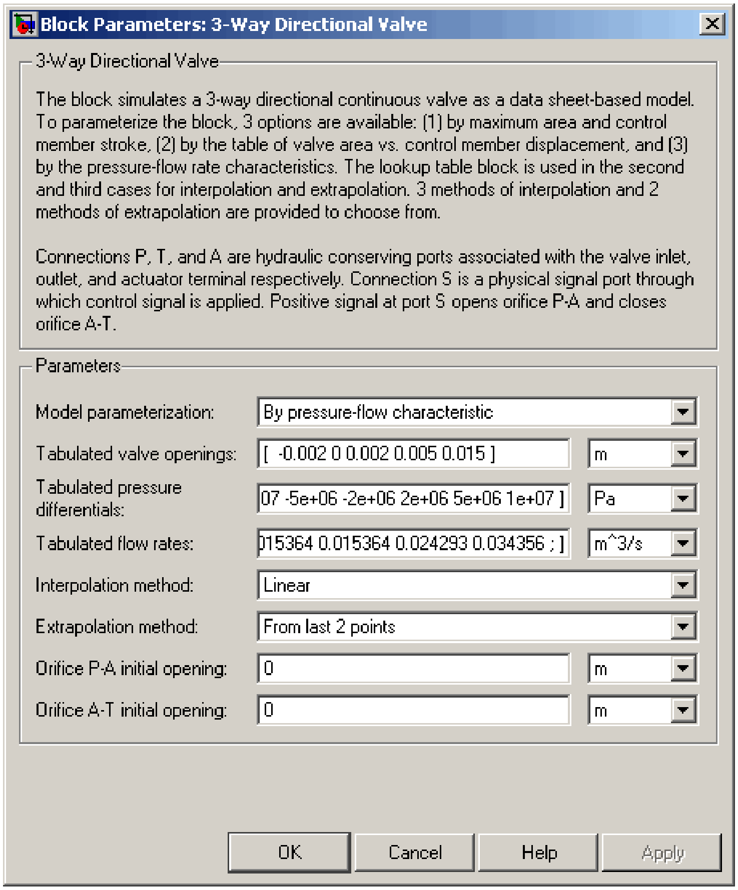
Flow discharge coefficient: 0.7

Orifice P-A initial opening: 0 m

Orifice A-T initial opening: 0 m

Critical Reynolds number: 12

Leakage area: 1e-12 m^2



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 pressure-flow characteristic

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

Tabulated pressure differentials: 07 -5e+06 -2e+06 2e+06 5e+06 1e+07] Pa

Tabulated flow rates: 015364 0.015364 0.024293 0.034356 ;] m^3/s

Interpolation method: Linear

Extrapolation method: From last 2 points

Orifice P-A initial opening: 0 m

Orifice A-T initial opening: 0 m

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` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve maximum opening

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

Tabulated valve openings

Specify the vector of input values for valve openings as a 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 `[-0.002 0 0.002 0.005 0.015]`. If **Model parameterization** is set to `By area vs.  opening table`, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to `By pressure-flow characteristic`, the **Tabulated valve openings** values will

be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Tabulated valve passage area

Specify the vector of output values for valve passage area as a 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-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Model parameterization** is set to `By area vs. opening table`.

Tabulated pressure differentials

Specify the vector of input values for pressure differentials as a 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 `[-1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]`. This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Tabulated flow rates

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

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

Interpolation method

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

- `Linear` — 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 Polinomial (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 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:

`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

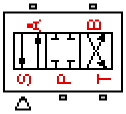
Purpose

Simulate hydraulic continuous 4-way directional valve

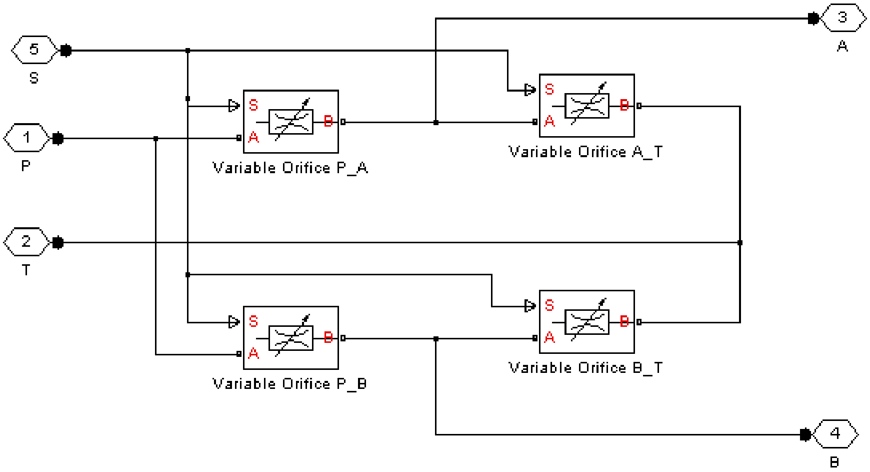
Library

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.

- `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:

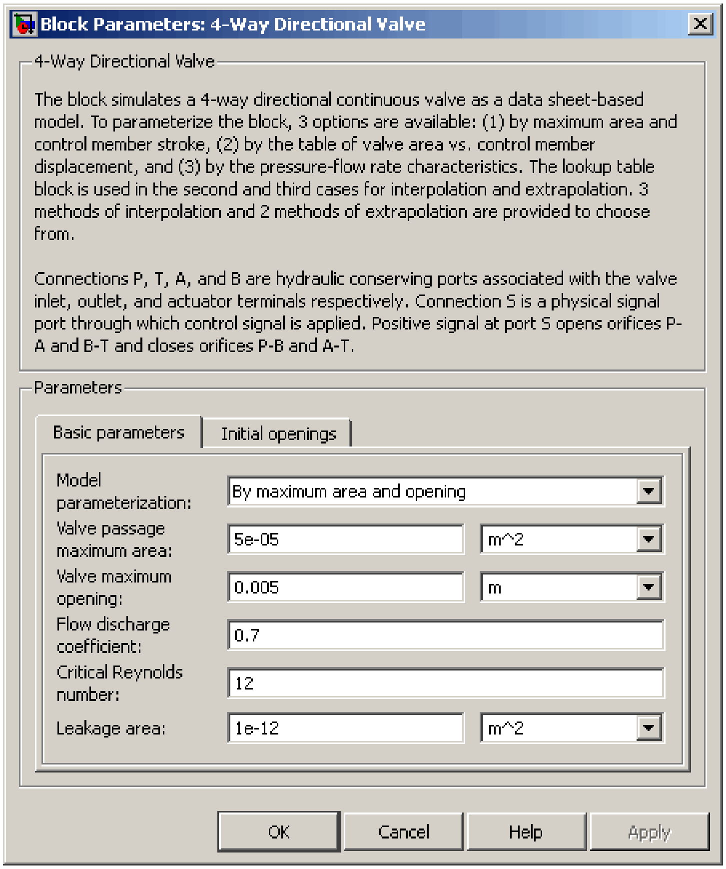
- Fluid inertia is not taken into account.
- Spool loading, such as inertia, spring, hydraulic forces, and so on, is not taken into account.
- Only symmetrical configuration of the valve is considered. In other words, all four orifices are assumed to have the same shape and size.

Dialog Box and Parameters

The block dialog box contains two tabs:

- “Basic Parameters” on page 2-41
- “Initial Openings” on page 2-48

Basic Parameters



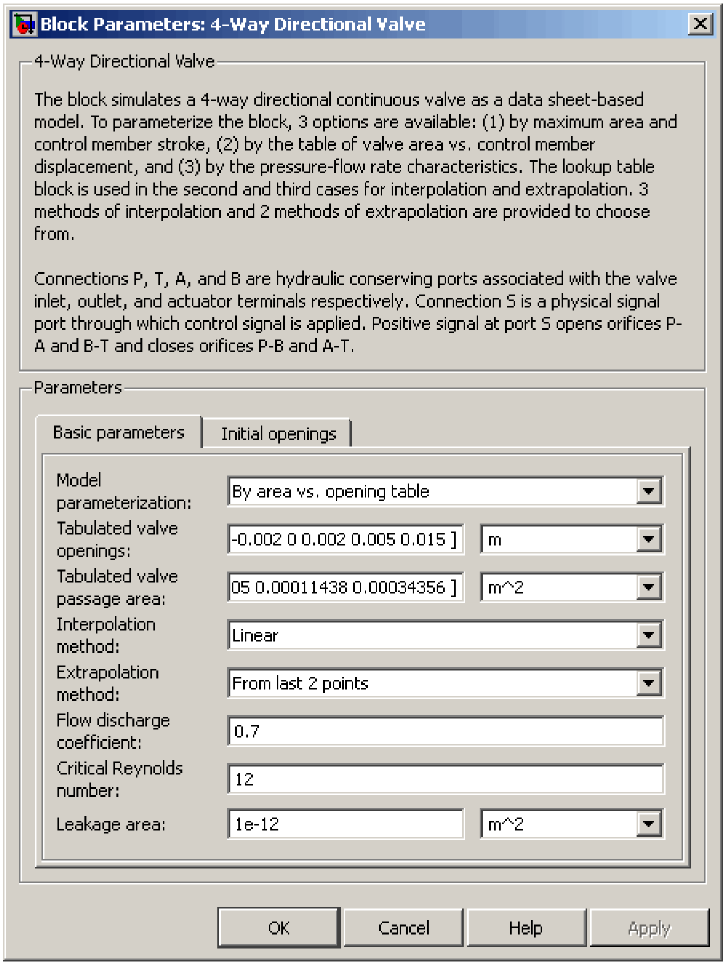
4-Way Directional Valve

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 maximum area and opening	
Valve passage maximum area:	5e-05	m^2
Valve maximum opening:	0.005	m
Flow discharge coefficient:	0.7	
Critical Reynolds number:	12	
Leakage area:	1e-12	m^2



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 0.002 0.005 0.015] m

Tabulated valve passage area: 05 0.00011438 0.00034356] m^2

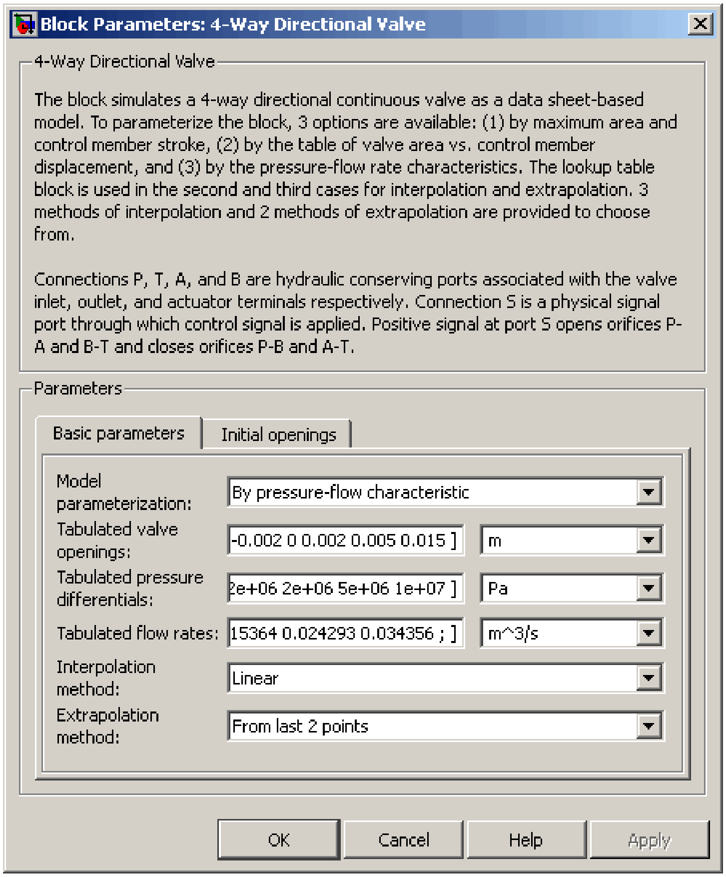
Interpolation method: Linear

Extrapolation method: From last 2 points

Flow discharge coefficient: 0.7

Critical Reynolds number: 12

Leakage area: 1e-12 m^2



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 pressure-flow characteristic

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

Tabulated pressure differentials: 2e+06 2e+06 5e+06 1e+07] Pa

Tabulated flow rates: 15364 0.024293 0.034356 ;] m^3/s

Interpolation method: Linear

Extrapolation method: From last 2 points

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` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve maximum opening

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

Tabulated valve openings

Specify the vector of input values for valve openings as a 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 `[-0.002 0 0.002 0.005 0.015]`. If **Model parameterization** is set to `By area vs.  opening table`, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to `By pressure-flow characteristic`, the **Tabulated valve openings** values will

be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Tabulated valve passage area

Specify the vector of output values for valve passage area as a 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-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Model parameterization** is set to `By area vs. opening table`.

Tabulated pressure differentials

Specify the vector of input values for pressure differentials as a 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 `[-1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]`. This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Tabulated flow rates

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

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

Interpolation method

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

- `Linear` — 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 Polinomial (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`.

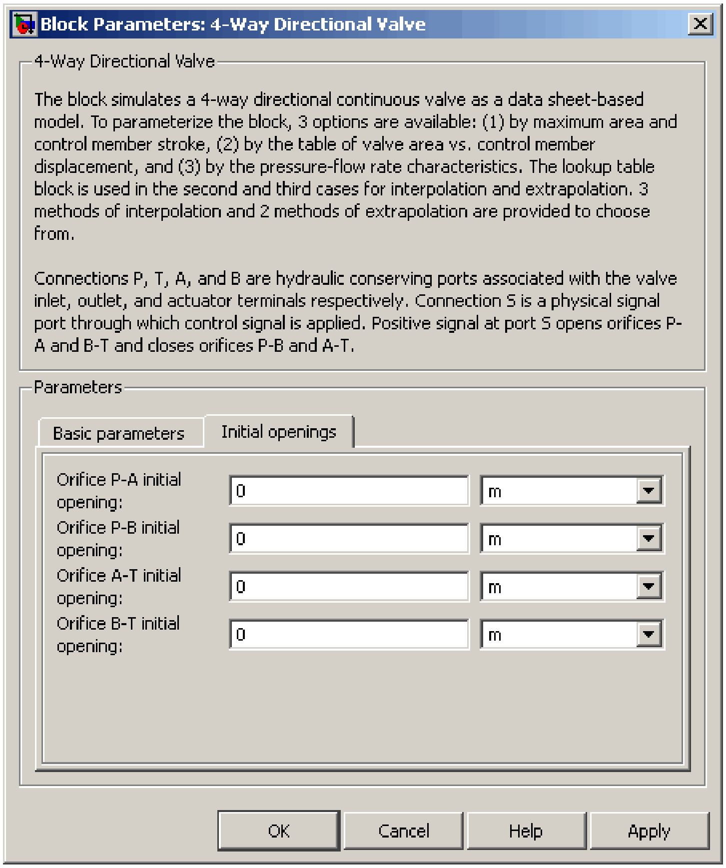
Critical Reynolds number

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` m^2.

Initial Openings



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

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

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`.

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.

B
: Hydraulic conserving port associated with the actuator connection port.

S
: Physical signal port to control spool displacement.

Examples

The 4-Way Directional Valve block in the Closed-Loop Circuit with 4-Way Valve and Custom Cylinder 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

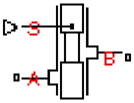
Purpose

Simulate hydraulic variable orifice created by circular tube and round insert

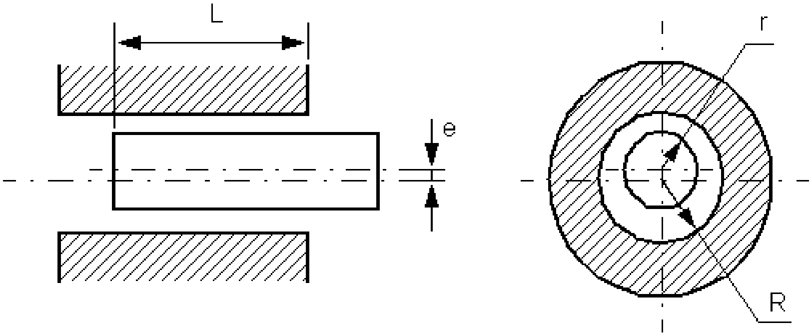
Library

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]):

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

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

where

q	Flow rate
p	Pressure differential
R	Orifice radius
r	Insert radius
L	Overlap length
ε	Eccentricity ratio
e	Eccentricity
ρ	Fluid density
ν	Fluid kinematic viscosity

Use this block to simulate leakage path in plungers, valves, and cylinders.

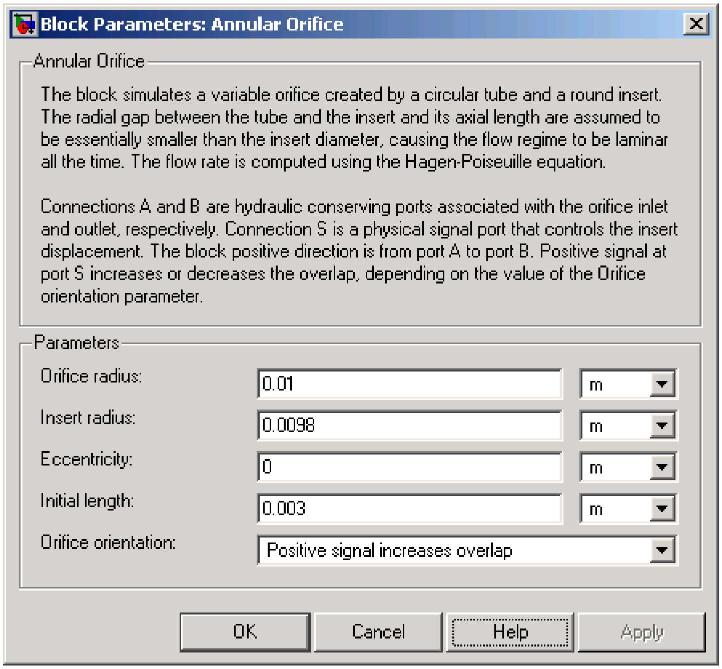
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as $p = p_A - p_B$. Positive signal at the physical signal port `S` increases or decreases the overlap, depending on the value of the parameter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

Basic Assumptions and Limitations

The model is based on the following assumption:

- Fluid inertia is not taken into account.

Dialog Box and Parameters



Block Parameters: Annular Orifice

Annular Orifice

The block simulates a variable orifice created by a circular tube and a round insert. 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. The flow rate is computed using the Hagen-Poiseuille equation.

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

Parameters

Orifice radius: 0.01 m

Insert radius: 0.0098 m

Eccentricity: 0 m

Initial length: 0.003 m

Orifice orientation: Positive signal increases overlap

Orifice radius

The radius of the tube. The default value is `0.01` m.

Insert radius

The radius of the insert. The default value is `0.0098` m.

Eccentricity

The distance between the central axes of the insert and the tube. The parameter can be a positive value, smaller than the difference between the radius of the tube and the radius of the insert, or equal to zero for coaxial configuration. The default value is `0`.

Initial length

Initial overlap between the tube and the insert. The parameter must be positive. The value of initial length does not depend on the orifice orientation. The default value is `0.003` m.

Orifice orientation
The parameter is introduced to specify the effect of the control signal on the orifice overlap. The parameter can be set to one of two options: `Positive signal increases overlap` or `Negative signal increases overlap`. The default value is `Positive signal increases overlap`.

Restricted Parameters

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

- **Orifice orientation**

All other block parameters are available for modification.

Global Parameters

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

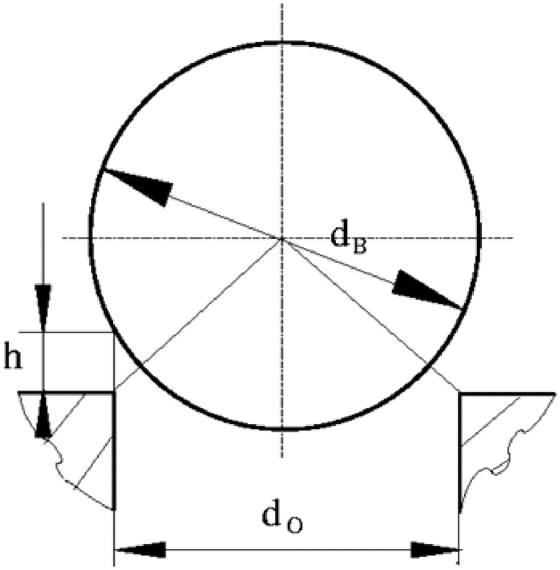
Simulate hydraulic ball valve

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:

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

$$h = x_0 + x$$

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

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

$$p = p_A - p_B$$

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

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

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

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

where

q	Flow rate
p	Pressure differential
p_A, p_B	Gauge pressures at the block terminals
C_D	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
x_0	Initial opening
x	Ball displacement from initial position
h	Valve opening
d_0	Orifice diameter
r_0	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.

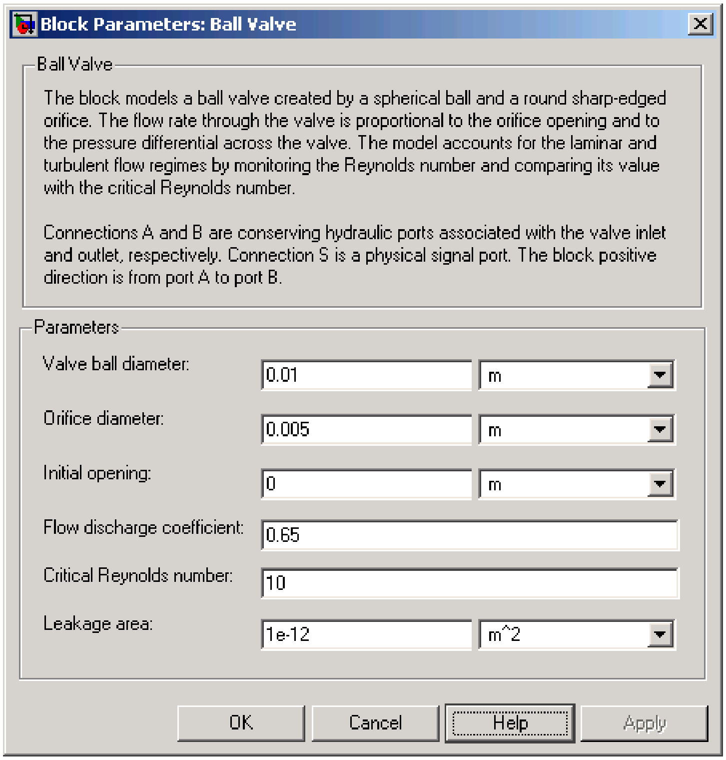
Basic Assumptions and Limitations

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



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.

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	m^2

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.

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` 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

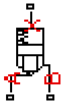
Pressure-Compensated Flow Control Valve

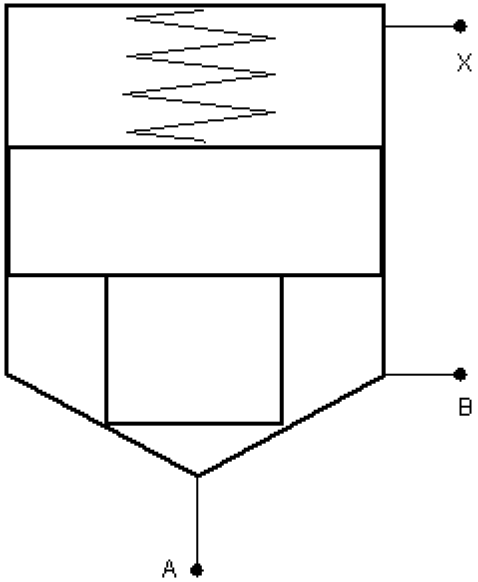
Cartridge Valve Insert

Purpose Simulate hydraulic cartridge valve insert

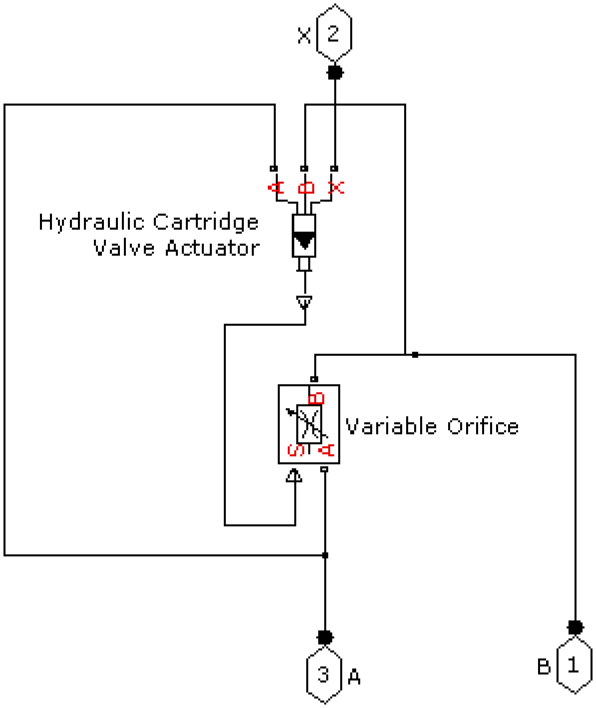
Library Directional Valves

Description The Cartridge Valve Insert block represents an insert of a hydraulic cartridge valve consisting of a poppet interacting with the seat. The poppet position is determined by pressures at ports A, B, and X and force of the spring. A schematic diagram of the cartridge valve insert is shown in the following illustration.





The Cartridge Valve Insert block is a structural model consisting of a Hydraulic Cartridge Valve Actuator block and a Variable Orifice block, as shown in the next illustration.



Pressures at port A and port B tend to open the valve, while pressure at the control port X, together with the spring, acts to close it. The model does not account for flow rates caused by poppet displacement and any loading on the poppet, such as inertia and friction. The valve remains closed as long as the aggregate pressure force is lower than the spring preload force. The poppet is forced off its seat as the preload force is reached and moves up proportionally to pressure increase until it passes the full stroke. Hydraulic properties of the gap between the poppet and the seat are simulated with the Variable Orifice block.

Connections A, B, and X are hydraulic conserving ports associated with the valve inlet, valve outlet, and valve control terminal, respectively.

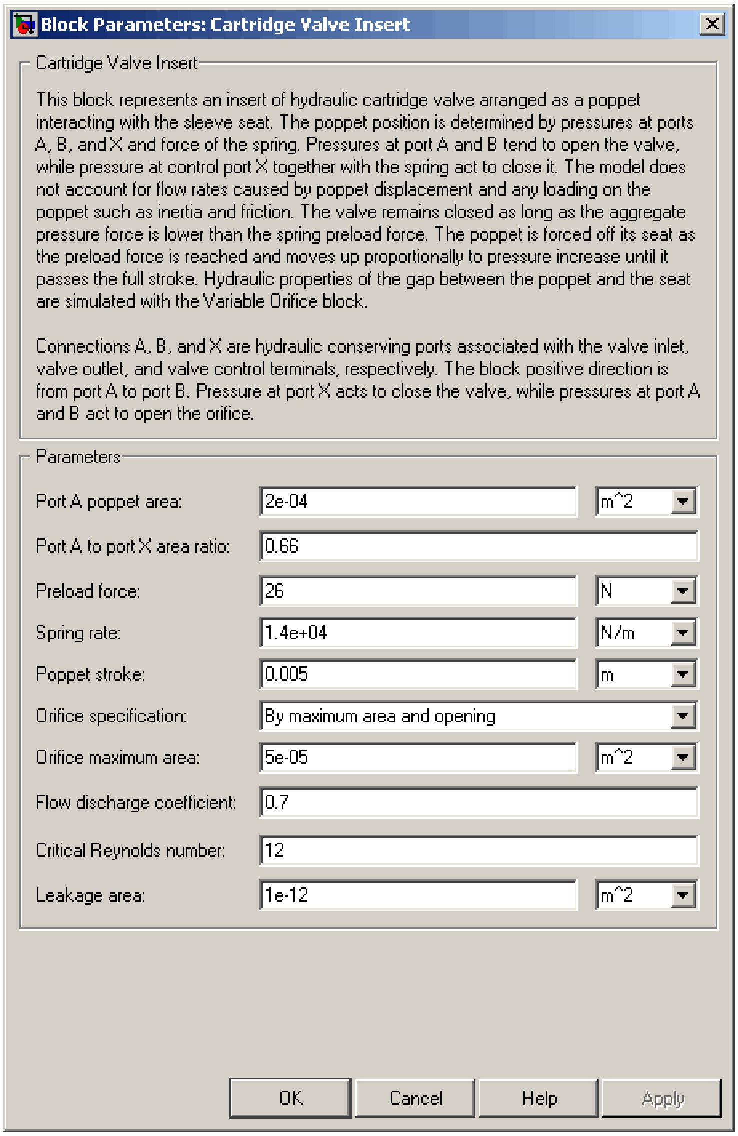
The block positive direction is from port A to port B. Pressure at port X acts to close the valve, while pressures at port A and port B act to open the orifice.

Basic Assumptions and Limitations

The model is based on the following assumptions:

- Valve opening is linearly proportional to the pressure differential.
- No loading on the poppet, such as inertia or friction, is considered.
- The model does not account for flow rates caused by poppet displacement.
- For orifices specified by the passage area (the first two parameterization options), the transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re = Re_{cr}$.
- For orifices specified by pressure-flow characteristics (the third parameterization option), the model does not explicitly account for the flow regime or leakage flow rate because the tabulated data is assumed to account for these characteristics.

Dialog Box and Parameters



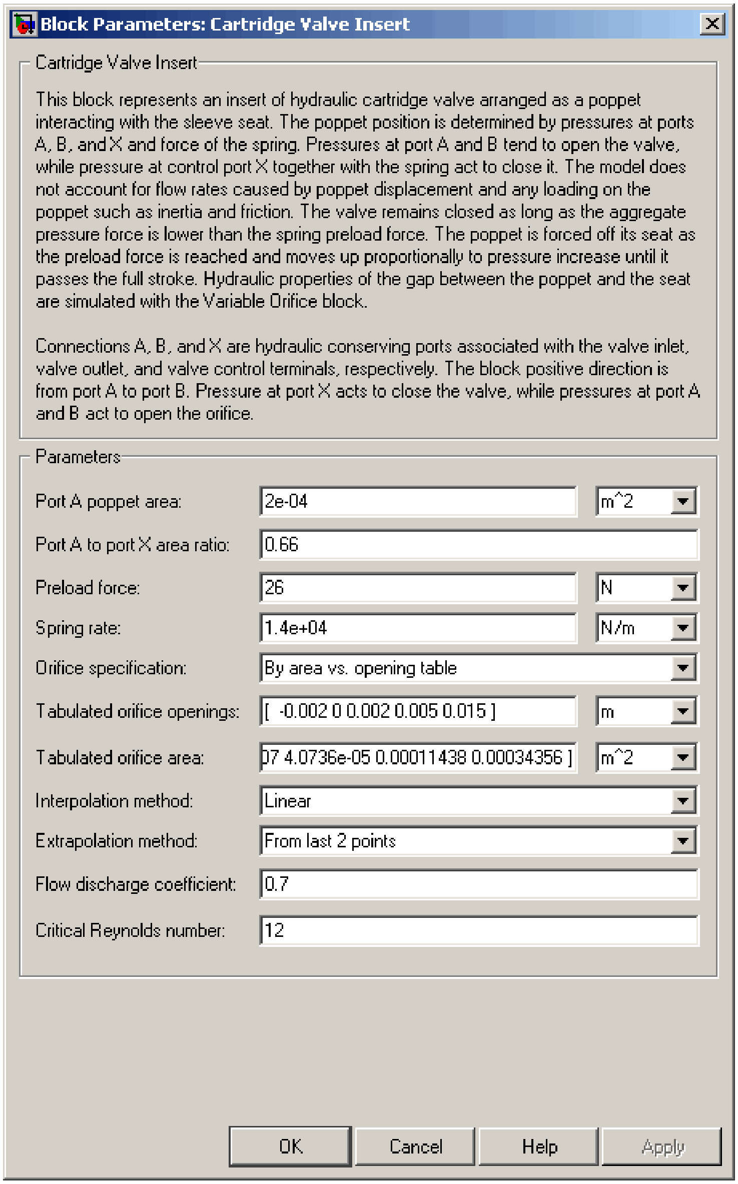
Cartridge Valve Insert

This block represents an insert of hydraulic cartridge valve arranged as a poppet interacting with the sleeve seat. The poppet position is determined by pressures at ports A, B, and X and force of the spring. Pressures at port A and B tend to open the valve, while pressure at control port X together with the spring act to close it. The model does not account for flow rates caused by poppet displacement and any loading on the poppet such as inertia and friction. The valve remains closed as long as the aggregate pressure force is lower than the spring preload force. The poppet is forced off its seat as the preload force is reached and moves up proportionally to pressure increase until it passes the full stroke. Hydraulic properties of the gap between the poppet and the seat are simulated with the Variable Orifice block.

Connections A, B, and X are hydraulic conserving ports associated with the valve inlet, valve outlet, and valve control terminals, respectively. The block positive direction is from port A to port B. Pressure at port X acts to close the valve, while pressures at port A and B act to open the orifice.

Parameters

Port A poppet area:	2e-04	m^2
Port A to port X area ratio:	0.66	
Preload force:	26	N
Spring rate:	1.4e+04	N/m
Poppet stroke:	0.005	m
Orifice specification:	By maximum area and opening	
Orifice maximum area:	5e-05	m^2
Flow discharge coefficient:	0.7	
Critical Reynolds number:	12	
Leakage area:	1e-12	m^2



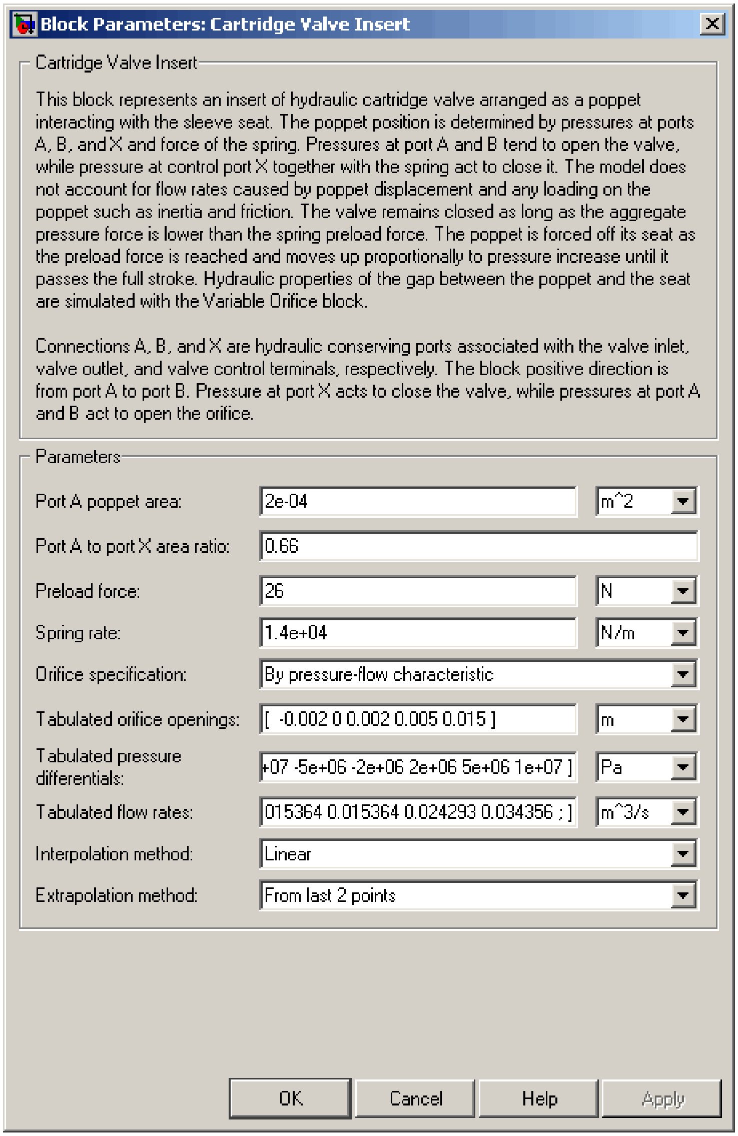
Cartridge Valve Insert

This block represents an insert of hydraulic cartridge valve arranged as a poppet interacting with the sleeve seat. The poppet position is determined by pressures at ports A, B, and X and force of the spring. Pressures at port A and B tend to open the valve, while pressure at control port X together with the spring act to close it. The model does not account for flow rates caused by poppet displacement and any loading on the poppet such as inertia and friction. The valve remains closed as long as the aggregate pressure force is lower than the spring preload force. The poppet is forced off its seat as the preload force is reached and moves up proportionally to pressure increase until it passes the full stroke. Hydraulic properties of the gap between the poppet and the seat are simulated with the Variable Orifice block.

Connections A, B, and X are hydraulic conserving ports associated with the valve inlet, valve outlet, and valve control terminals, respectively. The block positive direction is from port A to port B. Pressure at port X acts to close the valve, while pressures at port A and B act to open the orifice.

Parameters

Port A poppet area:	2e-04	m^2
Port A to port X area ratio:	0.66	
Preload force:	26	N
Spring rate:	1.4e+04	N/m
Orifice specification:	By area vs. opening table	
Tabulated orifice openings:	[-0.002 0 0.002 0.005 0.015]	m
Tabulated orifice area:	07 4.0736e-05 0.00011438 0.00034356]	m^2
Interpolation method:	Linear	
Extrapolation method:	From last 2 points	
Flow discharge coefficient:	0.7	
Critical Reynolds number:	12	



Block Parameters: Cartridge Valve Insert

Cartridge Valve Insert

This block represents an insert of hydraulic cartridge valve arranged as a poppet interacting with the sleeve seat. The poppet position is determined by pressures at ports A, B, and X and force of the spring. Pressures at port A and B tend to open the valve, while pressure at control port X together with the spring act to close it. The model does not account for flow rates caused by poppet displacement and any loading on the poppet such as inertia and friction. The valve remains closed as long as the aggregate pressure force is lower than the spring preload force. The poppet is forced off its seat as the preload force is reached and moves up proportionally to pressure increase until it passes the full stroke. Hydraulic properties of the gap between the poppet and the seat are simulated with the Variable Orifice block.

Connections A, B, and X are hydraulic conserving ports associated with the valve inlet, valve outlet, and valve control terminals, respectively. The block positive direction is from port A to port B. Pressure at port X acts to close the valve, while pressures at port A and B act to open the orifice.

Parameters

Port A poppet area:	2e-04	m^2
Port A to port X area ratio:	0.66	
Preload force:	26	N
Spring rate:	1.4e+04	N/m
Orifice specification:	By pressure-flow characteristic	
Tabulated orifice openings:	[-0.002 0 0.002 0.005 0.015]	m
Tabulated pressure differentials:	+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]	Pa
Tabulated flow rates:	015364 0.015364 0.024293 0.034356 ;]	m^3/s
Interpolation method:	Linear	
Extrapolation method:	From last 2 points	

Port A poppet area

Effective poppet area at port A. The parameter value must be greater than zero. The default value is `2e-4` m^2.

Port A to port X area ratio

Ratio between poppet areas at port A and port X. The parameter value must be greater than zero. The default value is `0.66`.

Preload force

Spring preload force. The default value is `26` N.

Spring rate

Spring rate. The default value is `1.4e4` N/m.

Poppet stroke

Maximum poppet stroke. The parameter value must be greater than or equal to zero. The default value is `5e-3` m. This parameter is used if **Orifice specification** is set to `By maximum area and opening`.

Orifice specification

Select one of the following methods for specifying the hydraulic properties of the gap between the poppet and the seat:

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

For more information on these options, see the Variable Orifice block reference page.

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

Tabulated orifice openings
Specify the vector of input values for orifice openings as a 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 `[-0.002 0 0.002 0.005 0.015]`. If **Orifice specification** is set to `By area vs. opening table`, the **Tabulated orifice openings** values will be used together with **Tabulated orifice area** values for one-dimensional table lookup. If **Orifice specification** is set to `By pressure-flow characteristic`, the **Tabulated orifice openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Tabulated orifice area
Specify the vector of 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-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Orifice specification** is set to `By area vs. opening table`.

Tabulated pressure differentials
Specify the 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 `[-1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]`. This parameter is used if **Orifice specification** is set to `By pressure-flow characteristic`.

Tabulated flow rates

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

This parameter is used if **Orifice specification** is set to `By pressure-flow characteristic`.

Interpolation method

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

- `Linear` — 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 Polinomial (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 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` m^2.

Restricted Parameters

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

- **Orifice specification**
- **Interpolation method**
- **Extrapolation method**

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

Global Parameters

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.

X

Hydraulic conserving port associated with the valve control terminal.

See Also

Check Valve

Hydraulic Cartridge Valve Actuator

Pilot-Operated Check Valve

Centrifugal Pump

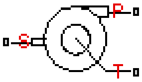
Purpose

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. Depending on data listed in the manufacturer's catalogs or data sheets for your particular pump, you can choose one of the following model parameterization options:

- `By approximating polynomial` — Provide values for the polynomial coefficients. These values can be determined analytically or experimentally, depending on the data available. This is the default method.
- `By two 1D characteristics:  P-Q and N-Q` — Provide tabulated data of pressure differential and brake power versus pump delivery characteristics. The pressure differential and brake power are determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.
- `By two 2D characteristics:  P-Q-W and N-Q-W` — Provide tabulated data of pressure differential and brake power versus pump delivery characteristics at different angular velocities. The pressure differential and brake power are determined by two-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.

These parameterization options are further described in greater detail:

- "Parameterizing the Pump by Approximating Polynomial" on page 2-75
- "Parameterizing the Pump by Pressure Differential and Brake Power Versus Pump Delivery" on page 2-79
- "Parameterizing the Pump by Pressure Differential and Brake Power Versus Pump Delivery at Different Angular Velocities " on page 2-80

Connections P and T are hydraulic conserving ports associated with the pump outlet and inlet, respectively. Connection S is a mechanical rotational conserving port associated with the pump driving shaft. The block positive direction is from port T to port P. This means that the pump transfers fluid from T to P as its driving shaft S rotates in the globally assigned positive direction.

Note The model is developed only for positive, nonzero shaft speeds. In other words, the pump driving shaft must rotate in positive direction only, without stopping.

Parameterizing the Pump by Approximating Polynomial

If you set the **Model parameterization** parameter to `By approximating polynomial`, the pump is parameterized with the polynomial whose coefficients are determined, analytically or experimentally, for a specific angular velocity depending on the data available. The pump characteristics at other angular velocities are determined from the affinity laws.

The approximating polynomial is derived from the Euler pulse moment equation [1, 2], which for a known pump can be represented as the following:

$$p = k \bullet p_E - p_{HL} - p_D$$

where

p	Pressure differential across the pump
k	Correction factor. The factor is introduced to account for dimensional fluctuations, blade incongruity, blade volumes, fluid internal friction, and so on. The factor should be set to 1 if the approximating coefficients are determined experimentally.
p_E	Euler pressure

p_{HL}	Pressure loss due to hydraulic losses in the pump passages
p_D	Pressure loss caused by deviations of the pump delivery from its nominal 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, operating at constant angular velocity and specific fluid, the Euler pressure can be approximated with the equation

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

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_P	Pump volumetric delivery

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

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

where

ρ_{ref}	Fluid density
c_2	Approximating coefficient
q_P	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

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

where

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

The resulting approximating polynomial takes the form:

$$p = \rho_{ref} \left(k(c_0 - c_1 q) - c_2 q^2 - c_3 \left(q_D - q_P\right)^2 \right) \tag{2-1}$$

The pump characteristics, approximated with four coefficients c_0, c_1, c_2, and c_3, are determined for a specific fluid and a specific angular velocity of the pump's driving shaft. These two parameters correspond, respectively, to the **Reference density** and **Reference angular velocity** parameters in the block dialog box. To apply the characteristics for another velocity ω or density ρ, the affinity laws are used. First, the new reference delivery is computed with the expression

$$q_{ref} = q \frac{\omega_{ref}}{\omega} \tag{2-2}$$

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:

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

$$N_{ref.hyd} = p_{ref} \bullet q_{ref}$$

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

where

η	Pump efficiency
$N_{ref.hyd}$	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:

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

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

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

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

Parameterizing the Pump by Pressure Differential and Brake Power Versus Pump Delivery

If you set the **Model parameterization** parameter to `By two 1D characteristics:  P-Q and N-Q`, the pump characteristics are computed by using two one-dimensional table lookups: for the pressure differential based on the pump delivery and for the pump brake power based on the pump delivery. Both characteristics are specified at the same angular velocity ω_{ref} (**Reference angular velocity**) and the same fluid density ρ_{ref} (**Reference density**).

To compute pressure differential at another angular velocity, affinity laws are used, similar to the first parameterization option. First, the new reference delivery q_{ref} is computed with the expression

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

where q is the current pump delivery. Then the pressure differential across the pump at current angular velocity ω and density ρ is computed as

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

where p_{ref} is the pressure differential determined from the P-Q characteristic at pump delivery q_{ref}.

Brake power is determined with the equation

$$N = N_{ref} \cdot \left(\frac{\omega}{\omega_{ref}} \right)^3 \cdot \frac{\rho}{\rho_{ref}}$$

where N_{ref} is the reference brake power obtained from the N-Q characteristic at pump delivery q_{ref}.

The torque at the pump driving shaft is computed with the equation $T = N / \omega$.

Parameterizing the Pump by Pressure Differential and Brake Power Versus Pump Delivery at Different Angular Velocities

If you set the **Model parameterization** parameter to `By two 2D characteristics: P-Q-W and N-Q-W`, the pump characteristics are read out from two two-dimensional table lookups: for the pressure differential based on the pump delivery and angular velocity and for the pump brake power based on the pump delivery and angular velocity.

Both the pressure differential and brake power are scaled if fluid density ρ is different from the reference density ρ_{ref}, at which characteristics have been obtained

$$p = p_{ref} \cdot \frac{\rho}{\rho_{ref}}$$

$$N = N_{ref} \cdot \frac{\rho}{\rho_{ref}}$$

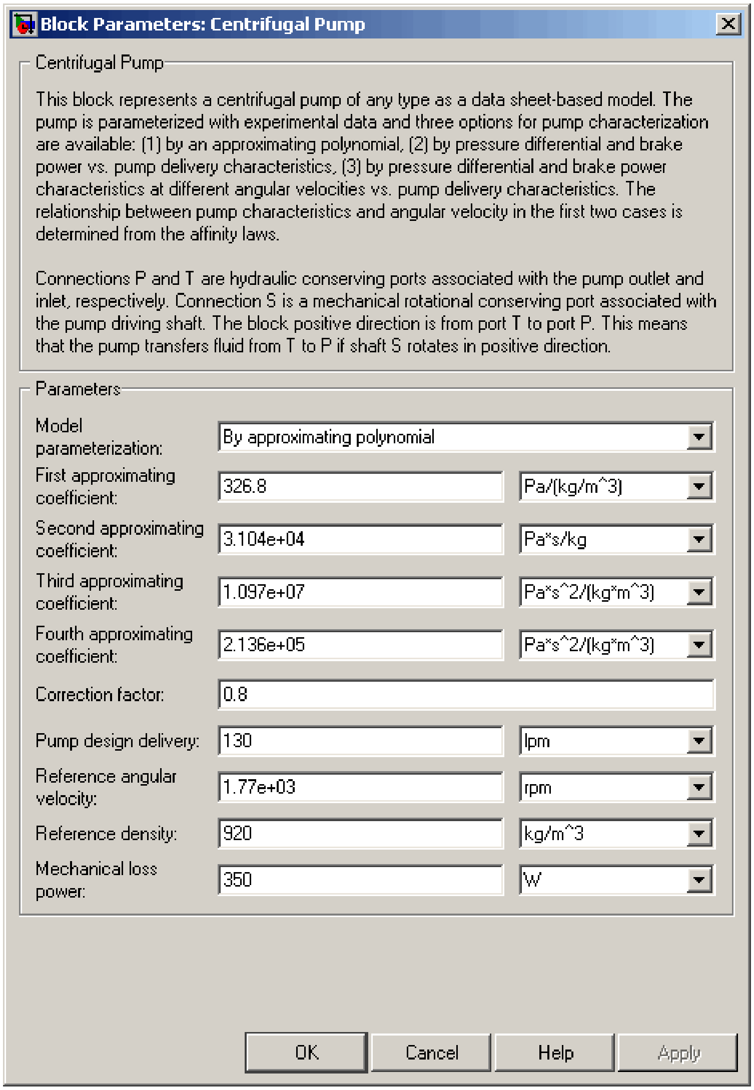
where p_{ref} and N_{ref} are the pressure differential and brake power obtained from the plots.

Basic Assumptions and Limitations

The model is based on the following assumptions:

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

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 experimental data and three options for pump characterization are available: (1) by an approximating polynomial, (2) by pressure differential and brake power vs. pump delivery characteristics, (3) by pressure differential and brake power characteristics at different angular velocities vs. pump delivery characteristics. The relationship between pump characteristics and angular velocity in the first two cases is determined from the affinity laws.

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

Model parameterization: By approximating polynomial

First approximating coefficient: 326.8 Pa/(kg/m^3)

Second approximating coefficient: 3.104e+04 Pa*s/kg

Third approximating coefficient: 1.097e+07 Pa*s^2/(kg*m^3)

Fourth approximating coefficient: 2.136e+05 Pa*s^2/(kg*m^3)

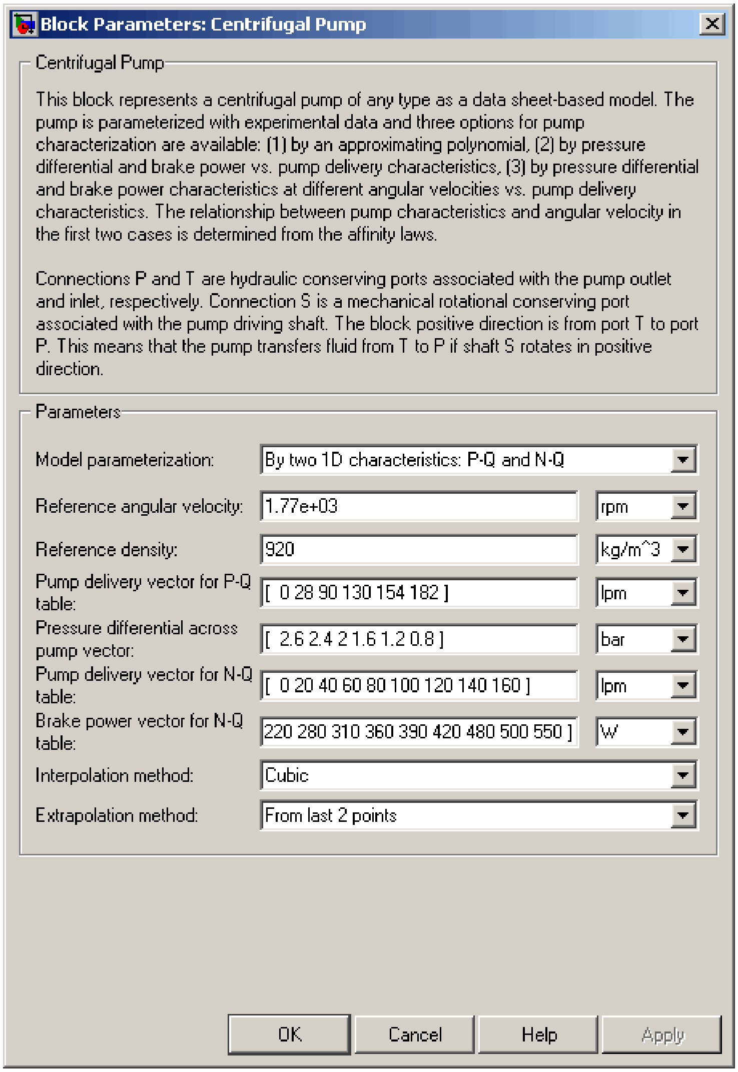
Correction factor: 0.8

Pump design delivery: 130 lpm

Reference angular velocity: 1.77e+03 rpm

Reference density: 920 kg/m^3

Mechanical loss power: 350 W



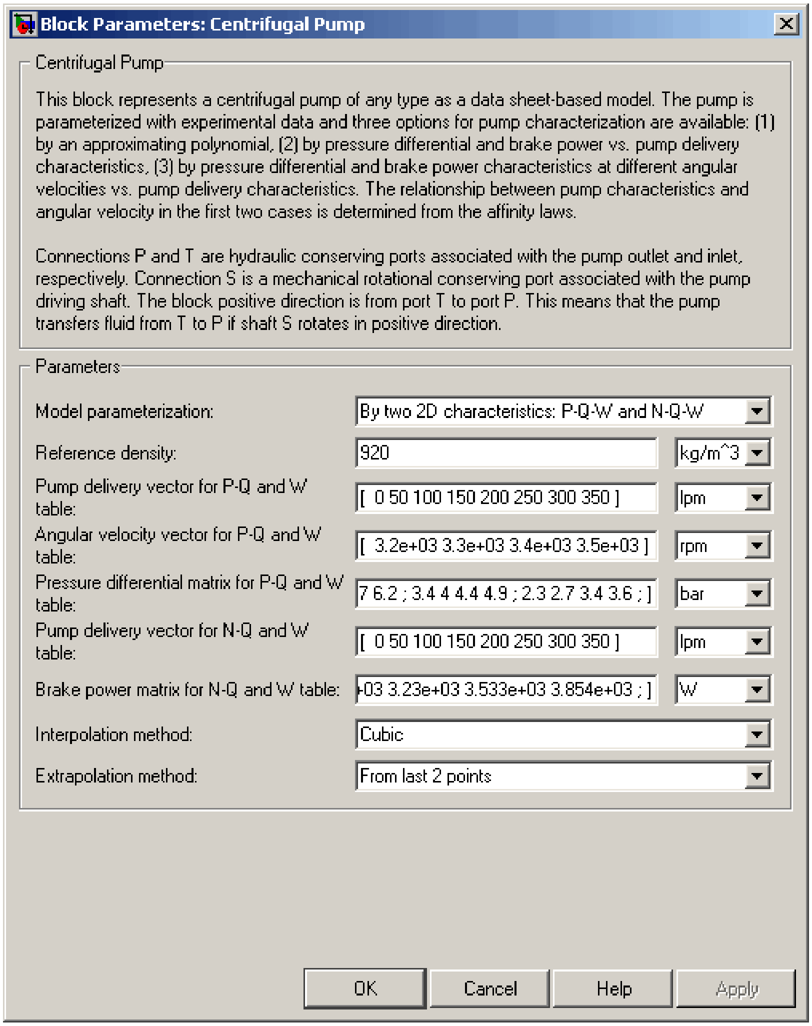
Centrifugal Pump

This block represents a centrifugal pump of any type as a data sheet-based model. The pump is parameterized with experimental data and three options for pump characterization are available: (1) by an approximating polynomial, (2) by pressure differential and brake power vs. pump delivery characteristics, (3) by pressure differential and brake power characteristics at different angular velocities vs. pump delivery characteristics. The relationship between pump characteristics and angular velocity in the first two cases is determined from the affinity laws.

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

Model parameterization:	By two 1D characteristics: P-Q and N-Q	
Reference angular velocity:	1.77e+03	rpm
Reference density:	920	kg/m^3
Pump delivery vector for P-Q table:	[0 28 90 130 154 182]	lpm
Pressure differential across pump vector:	[2.6 2.4 2 1.6 1.2 0.8]	bar
Pump delivery vector for N-Q table:	[0 20 40 60 80 100 120 140 160]	lpm
Brake power vector for N-Q table:	220 280 310 360 390 420 480 500 550]	W
Interpolation method:	Cubic	
Extrapolation method:	From last 2 points	



Block Parameters: Centrifugal Pump

Centrifugal Pump

This block represents a centrifugal pump of any type as a data sheet-based model. The pump is parameterized with experimental data and three options for pump characterization are available: (1) by an approximating polynomial, (2) by pressure differential and brake power vs. pump delivery characteristics, (3) by pressure differential and brake power characteristics at different angular velocities vs. pump delivery characteristics. The relationship between pump characteristics and angular velocity in the first two cases is determined from the affinity laws.

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

Model parameterization:	By two 2D characteristics: P-Q-W and N-Q-W	
Reference density:	920	kg/m^3
Pump delivery vector for P-Q and W table:	[0 50 100 150 200 250 300 350]	lpm
Angular velocity vector for P-Q and W table:	[3.2e+03 3.3e+03 3.4e+03 3.5e+03]	rpm
Pressure differential matrix for P-Q and W table:	7 6.2 ; 3.4 4 4.4 4.9 ; 2.3 2.7 3.4 3.6 ;]	bar
Pump delivery vector for N-Q and W table:	[0 50 100 150 200 250 300 350]	lpm
Brake power matrix for N-Q and W table:	+03 3.23e+03 3.533e+03 3.854e+03 ;]	W
Interpolation method:	Cubic	
Extrapolation method:	From last 2 points	

Model parameterization

Select one of the following methods for specifying the pump parameters:

- `By approximating polynomial` — Provide values for the polynomial coefficients. These values can be determined analytically or experimentally, depending on the data available. The relationship between pump characteristics and angular velocity is determined from the affinity laws. This is the default method.
- `By two 1D characteristics: P-Q and N-Q` — Provide tabulated data of pressure differential and brake power versus pump delivery characteristics. The pressure differential and brake power are determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods. The relationship between pump characteristics and angular velocity is determined from the affinity laws.
- `By two 2D characteristics: P-Q-W and N-Q-W` — Provide tabulated data of pressure differential and brake power versus pump delivery characteristics at different angular velocities. The pressure differential and brake power are determined by two-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.

First approximating coefficient

Approximating coefficient c_0 in the block description preceding. The default value is `326.8` Pa/(kg/m^3). This parameter is used if **Model parameterization** is set to `By approximating polynomial`.

Second approximating coefficient

Approximating coefficient c_1 in the block description preceding. The default value is `3.104e4` Pa*s/kg. This parameter is used if **Model parameterization** is set to `By approximating polynomial`.

Third approximating coefficient

Approximating coefficient c_2 in the block description preceding. This coefficient accounts for hydraulic losses in the pump. The default value is `1.097e7` Pa*s^2/(kg*m^3). This parameter is used if **Model parameterization** is set to `By approximating polynomial`.

Fourth approximating coefficient
Approximating coefficient c_3 in the block description preceding. This coefficient accounts for additional hydraulic losses caused by deviation from the nominal delivery. The default value is `2.136e5` Pa*s^2/(kg*m^3). This parameter is used if **Model parameterization** is set to `By approximating polynomial`.

Correction factor
The factor, denoted as k in the block description preceding, accounts for dimensional fluctuations, blade incongruity, blade volumes, fluid internal friction, and other factors that decrease Euler theoretical pressure. The default value is `0.8`. This parameter is used if **Model parameterization** is set to `By approximating polynomial`.

Pump design delivery
The pump nominal delivery. The blades profile, pump inlet, and pump outlet are shaped for this particular delivery. Deviation from this delivery causes an increase in hydraulic losses. The default value is `130` lpm. This parameter is used if **Model parameterization** is set to `By approximating polynomial`.

Reference angular velocity
Angular velocity of the driving shaft, at which the pump characteristics are determined. The default value is `1.77e3` rpm. This parameter is used if **Model parameterization** is set to `By approximating polynomial` or `By two 1D characteristics: P-Q and N-Q`.

Reference density
Fluid density at which the pump characteristics are determined. The default value is `920` kg/m^3.

Mechanical loss power
Power of mechanical loss in the pump drive train at reference parameters. The default value is `350` W. This parameter is

used if **Model parameterization** is set to `By approximating polynomial`.

Pump delivery vector for P-Q table

Specify the vector of pump deliveries, as a tabulated 1-by-`n` array, to be used together with the vector of pressure differentials to specify the P-Q pump characteristic. The vector values must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. The default values, in lpm, are `[0 28 90 130 154 182]`. This parameter is used if **Model parameterization** is set to `By two 1D characteristics: P-Q and N-Q`.

Pressure differential across pump vector

Specify the vector of pressure differentials across the pump as a tabulated 1-by-`n` array. The vector will be used together with the pump deliveries vector to specify the P-Q pump characteristic. The vector must be of the same size as the pump deliveries vector for the P-Q table. The default values, in bar, are `[2.6 2.4 2 1.6 1.2 0.8]`. This parameter is used if **Model parameterization** is set to `By two 1D characteristics: P-Q and N-Q`.

Pump delivery vector for N-Q table

Specify the vector of pump deliveries, as a tabulated 1-by-`n` array, to be used together with the vector of the pump brake power to specify the N-Q pump characteristic. The vector values must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. The default values, in lpm, are `[0 20 40 60 80 100 120 140 160]`. This parameter is used if **Model parameterization** is set to `By two 1D characteristics: P-Q and N-Q`.

Brake power vector for N-Q table

Specify the vector of pump brake power as a tabulated 1-by-`n` array. The vector will be used together with the pump deliveries vector to specify the N-Q pump characteristic. The vector must be of the same size as the pump deliveries vector for the N-Q table. The default values, in W, are `[220 280 310 360 390 420 480`

`500 550]`. This parameter is used if **Model parameterization** is set to `By two 1D characteristics:  P-Q and N-Q`.

Pump delivery vector for P-Q and W table

Specify the vector of pump deliveries, as a tabulated 1-by-`m` array, to be used together with the vector of angular velocities and the pressure differential matrix to specify the pump P-Q-W characteristic. The vector values must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. The default values, in lpm, are `[0 50 100 150 200 250 300 350]`. This parameter is used if **Model parameterization** is set to `By two 2D characteristics: P-Q-W and N-Q-W`.

Angular velocity vector for P-Q and W table

Specify the vector of angular velocities, as a tabulated 1-by-`n` array, to be used for calculating both the pump P-Q-W and N-Q-W characteristics. The vector values must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. The default values, in rpm, are `[3.2e+03 3.3e+03 3.4e+03 3.5e+03]`. This parameter is used if **Model parameterization** is set to `By two 2D characteristics:  P-Q-W and N-Q-W`.

Pressure differential matrix for P-Q and W table

Specify the matrix of pressure differentials across pump, as a tabulated `m`-by-`n` matrix, defining the pump P-Q-W characteristic together with the pump delivery and angular velocity vectors. Each value in the matrix specifies pressure differential for a specific combination of pump delivery and angular velocity. The matrix size must match the dimensions defined by the pump delivery and angular velocity vectors. The default values, in bar, are:

```
[ 8.3 8.8 9.3 9.9 ;
  7.8 8.3 8.8 9.4 ;
  7.2 7.6 8.2 8.7 ;
  6.5 7 7.5 8 ;
```

```
5.6 6.1 6.6 7.1 ;
4.7 5.2 5.7 6.2 ;
3.4 4 4.4 4.9 ;
2.3 2.7 3.4 3.6 ; ]
```

This parameter is used if **Model parameterization** is set to `By two 2D characteristics: P-Q-W and N-Q-W`.

Pump delivery vector for N-Q and W table

Specify the vector of pump deliveries, as a tabulated 1-by-`m` array, to be used together with the vector of angular velocities and the brake power matrix to specify the pump N-Q-W characteristic. The vector values must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. The default values, in lpm, are `[0 50 100 150 200 250 300 350]`. This parameter is used if **Model parameterization** is set to `By two 2D characteristics: P-Q-W and N-Q-W`.

Brake power matrix for N-Q and W table

Specify the matrix of pump brake power, as a tabulated `m-by-n` matrix, defining the pump N-Q-W characteristic together with the pump delivery and angular velocity vectors. Each value in the matrix specifies brake power for a specific combination of pump delivery and angular velocity. The matrix size must match the dimensions defined by the pump delivery and angular velocity vectors. The default values, in W, are:

```
[ 1.223e+03 1.341e+03 1.467e+03 1.6e+03 ;
  1.414e+03 1.551e+03 1.696e+03 1.85e+03 ;
  1.636e+03 1.794e+03 1.962e+03 2.14e+03 ;
  1.941e+03 2.129e+03 2.326e+03 2.54e+03 ;
  2.224e+03 2.439e+03 2.66e+03 2.91e+03 ;
  2.453e+03 2.691e+03 2.947e+03 3.21e+03 ;
  2.757e+03 3.024e+03 3.307e+03 3.608e+03 ;
  2.945e+03 3.23e+03 3.533e+03 3.854e+03 ; ]
```

This parameter is used if **Model parameterization** is set to `By two 2D characteristics:  P-Q-W and N-Q-W`.

Interpolation method

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

- `Linear` — For one-dimensional table lookup (`By two 1D characteristics:  P-Q and N-Q`), uses a linear interpolation function. For two-dimensional table lookup (`By two 2D characteristics:  P-Q-W and N-Q-W`), uses a bilinear interpolation algorithm, which is an extension of linear interpolation for functions in two variables.
- `Cubic` — For one-dimensional table lookup (`By two 1D characteristics:  P-Q and N-Q`), uses the Piecewise Cubic Hermite Interpolation Polinomial (PCHIP). For two-dimensional table lookup (`By two 2D characteristics: P-Q-W and N-Q-W`), uses the bicubic interpolation algorithm.
- `Spline` — For one-dimensional table lookup (`By two 1D characteristics:  P-Q and N-Q`), uses the cubic spline interpolation algorithm. For two-dimensional table lookup (`By two 2D characteristics:  P-Q-W and N-Q-W`), uses the bicubic spline interpolation algorithm.

This parameter is used if **Model parameterization** is set to `By By two 1D characteristics:  P-Q and N-Q` or `By two By two 2D characteristics:  P-Q-W and N-Q-W`. For more information on interpolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Extrapolation method

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

- `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.

This parameter is used if **Model parameterization** is set to `By By two 1D characteristics:  P-Q and N-Q` or `By two By two 2D characteristics:  P-Q-W and N-Q-W`. For more information on extrapolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Restricted Parameters

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

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

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

Global Parameters

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.

Ports

The block has the following ports:

T

Hydraulic conserving port associated with the pump suction, or inlet.

P

Hydraulic conserving port associated with the pump outlet.

S

Mechanical rotational conserving port associated with the pump driving shaft.

References

[1] T.G. Hicks, T.W. Edwards, *Pump Application Engineering*, McGraw-Hill, NY, 1971

[2] I.J. Karassic, J.P. Messina, P. Cooper, C.C. Heald, *Pump Handbook*, Third edition, McGraw-Hill, NY, 2001

See Also

Fixed-Displacement Pump

Variable-Displacement Pressure-Compensated Pump

Variable-Displacement Pump

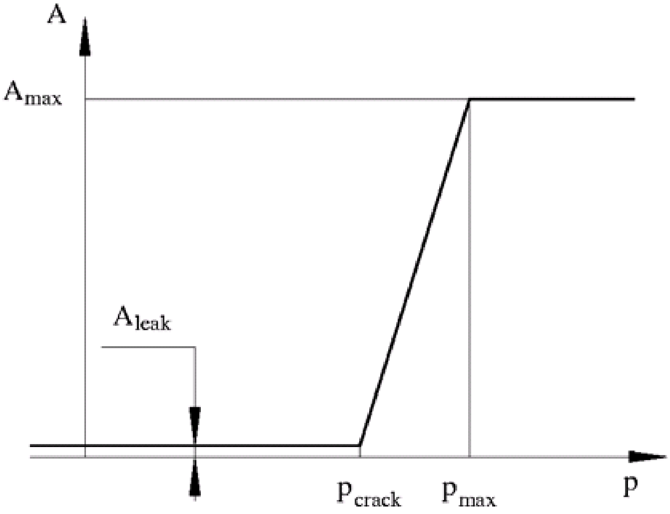
Check Valve

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 value 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:

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

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

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

$$p = p_A - p_B$$

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

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

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

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

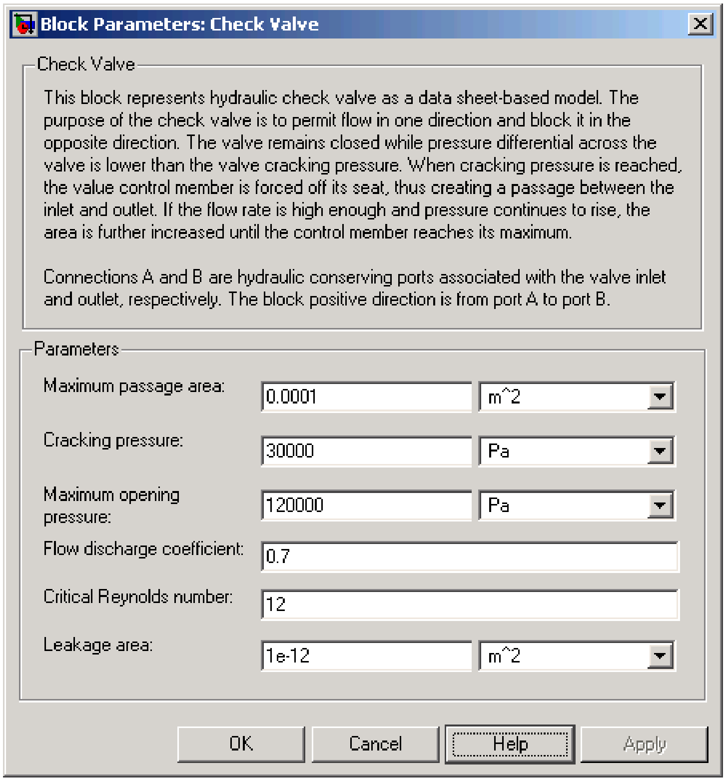
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as $p = p_A - p_B$.

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 value 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

Maximum passage area

Valve passage maximum cross-sectional area. The default value is `1e-4` m^2.

Cracking pressure

Pressure level at which the orifice of the valve starts to open. The default value is `3e4` Pa.

Maximum opening pressure

Pressure differential across the valve needed to fully open the valve. Its value must be higher than the cracking pressure. The default value is `1.2e5` Pa.

Flow discharge coefficient

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

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` 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 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

Cylinder Friction

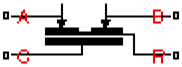
Purpose

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:

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

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

where

F	Friction force
F_C	Coulomb friction
F_{pr}	Preload force
f_{cfr}	Coulomb friction coefficient
p_A, p_B	Pressures in cylinder chambers
K_{brk}	Breakaway friction force increase coefficient
c_v	Transition coefficient

v	Relative velocity in the contact
f_{vfr}	Viscous friction coefficient

To avoid discontinuity at $v = 0$, a small region $|v| \leq v_{th}$ is introduced around zero velocity, where friction force is assumed to be linearly proportional to velocity:

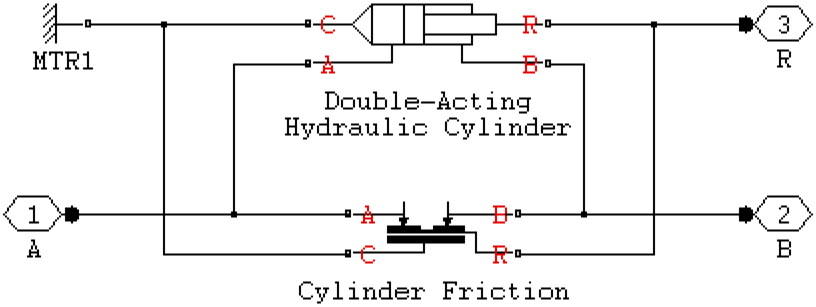
$$F = K \bullet v$$

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

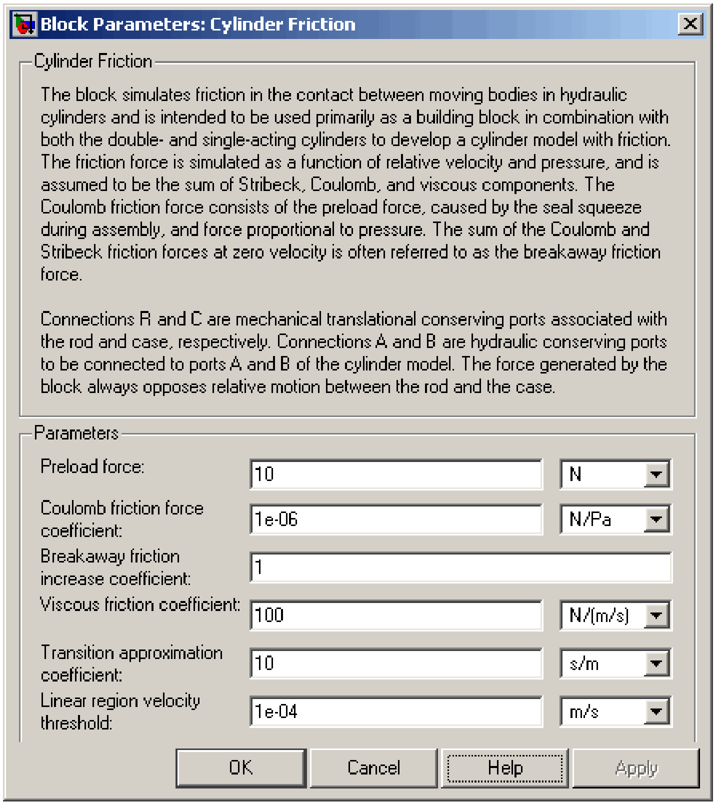
where

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.


MTR1
C
R
A
B
Double-Acting
Hydraulic Cylinder
3
R
1
A
A
D
C
R
2
B
Cylinder Friction

Dialog Box and Parameters



Cylinder Friction

The 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 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.

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. The force generated by the block always opposes relative motion between the rod and the case.

Parameters

Preload force: 10 N
Coulomb friction force coefficient: 1e-06 N/Pa
Breakaway friction increase coefficient: 1
Viscous friction coefficient: 100 N/(m/s)
Transition approximation coefficient: 10 s/m
Linear region velocity threshold: 1e-04 m/s

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 $c_v \sim= 4/v_{min,}$ 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.

`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

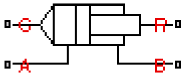
Purpose

Simulate hydraulic actuator exerting force in both directions

Library

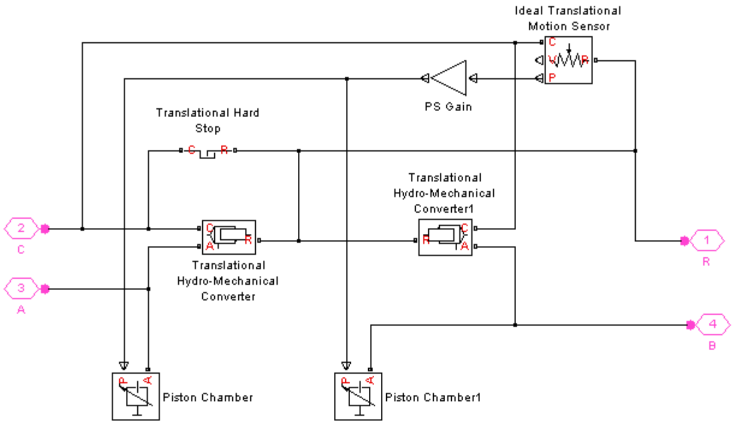
Hydraulic Cylinders

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



Connections R and C are mechanical translational conserving ports corresponding to the cylinder rod and cylinder clamping structure, respectively. Connections A and B are hydraulic conserving ports. Port A is connected to 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 Piston 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 Piston Chamber blocks.

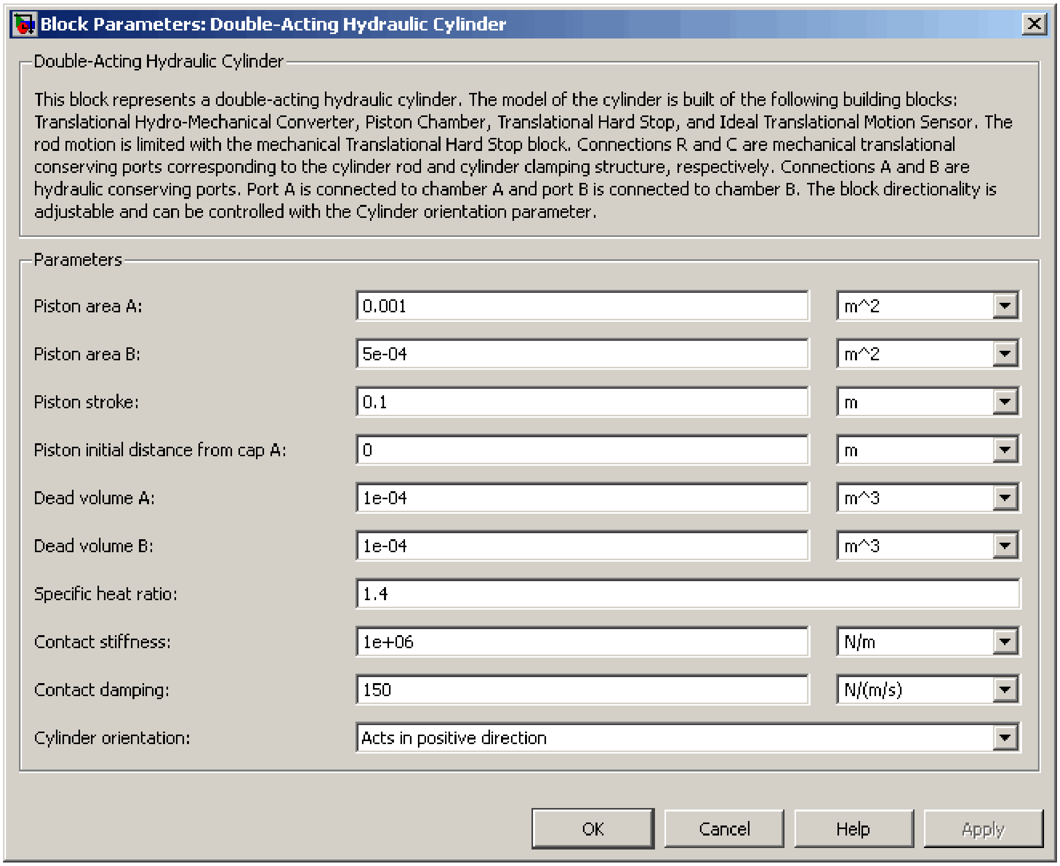
The block directionality is adjustable and can be controlled with the **Cylinder orientation** parameter.

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.

Dialog Box and Parameters



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, Piston 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 distance from cap A:	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	

Piston area A

Chamber A effective piston area. The default value is `0.001` m^2.

Piston area B

Chamber B effective piston area. The default value is `5e-5` m^2.

Piston stroke

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

Piston initial distance from cap A

The distance that the piston is extended at the beginning of simulation. You can set the piston position to any point within its stroke. The default value is `0`, which corresponds to the fully retracted position.

Dead volume A

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

Dead volume B

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

Specific heat ratio

Gas-specific heat ratio for the Piston 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 `1e6` N/m.

Contact damping

Specifies dissipating property of colliding bodies for the Translational Hard Stop block. At zero damping, the impact is close to an absolutely elastic one. The greater the value of the parameter, the more energy dissipates during an interaction. Keep in mind that damping affects slider motion as long as the slider is in contact with the stop, including the period when slider is pulled back from the contact. For computational efficiency and convergence reasons, 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.

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.

The Closed-Loop Circuit with 4-Way Valve and Custom Cylinder demo (`sh_closed_loop_circuit_4_way_valve_cust_cyl`) demonstrates the use of a 4-way valve in combination with a double-acting cylinder in a simple closed-loop actuator. The demo shows how to connect the blocks and set the initial orifice openings for the 4-way valve to model the forward and return strokes of the cylinder under load.

See Also

Single-Acting Hydraulic Cylinder

Ideal Translational Motion Sensor

Translational Hard Stop

Translational Hydro-Mechanical Converter

Piston Chamber

Double-Acting Rotary Actuator

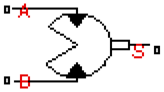
Purpose

Simulate double-acting hydraulic rotary actuator

Library

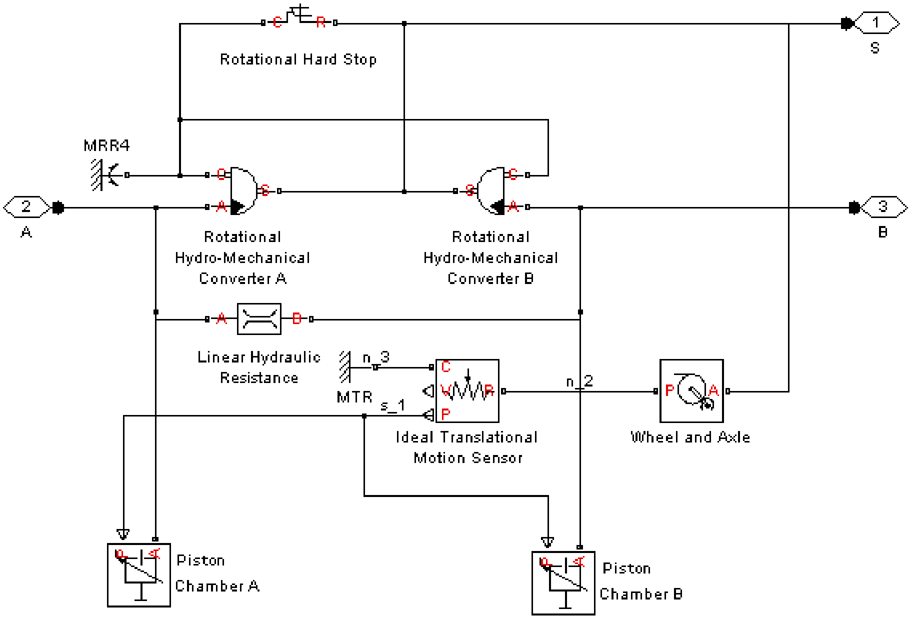
Hydraulic Cylinders

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.



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.
Piston Chamber A	Accounts for fluid compressibility in actuator chamber A.
Piston Chamber B	Accounts for fluid compressibility in actuator chamber B.
Ideal Translational Motion Sensor	Determines an instantaneous shaft position, which is necessary for the Piston 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.

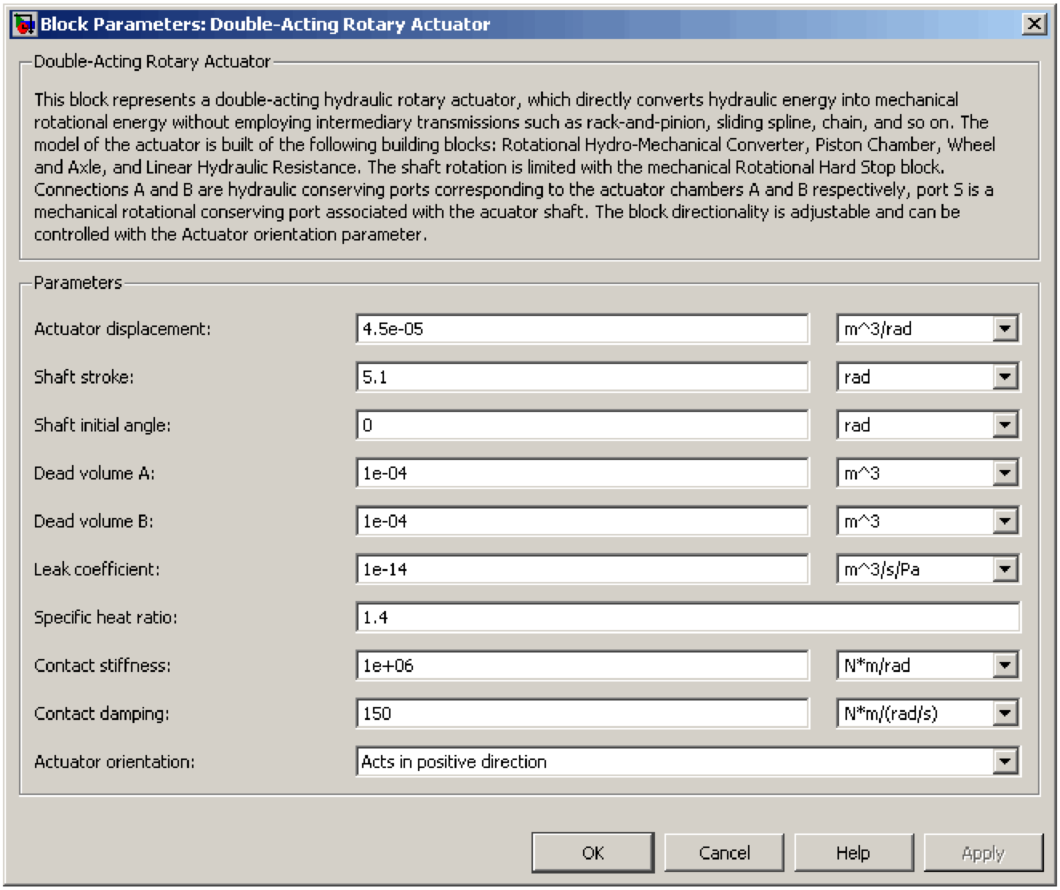
The block directionality is adjustable and can be controlled with the **Actuator orientation** parameter.

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, Piston 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 acuator 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	

Actuator displacement
Effective displacement of the actuator. The default value is `4.5e-5` m^3/rad.

Shaft stroke
Shaft maximum travel between stops. The default value is `5.1` rad.

Shaft initial angle
The position of the shaft at the beginning of simulation. You can set the shaft position to any angle within its stroke. The default value is `0`, which corresponds to the shaft position at the very beginning of the stroke.

Dead volume A
Fluid volume in chamber A that remains in the chamber when the shaft is positioned at the very beginning of the stroke. The default value is `1e-4` m^3.

Dead volume B
Fluid volume in chamber B that remains in the chamber when the shaft is positioned at the end of the stroke. The default value is `1e-4` m^3.

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

Specific heat ratio
Gas-specific heat ratio for the Piston Chamber block. The default value is `1.4`.

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

Contact damping
Specifies dissipating property of colliding bodies for the Rotational Hard Stop block. At zero damping, the impact is close to an absolutely elastic one. The greater the value of the parameter, the more energy dissipates during an interaction. Keep in mind that damping affects slider motion as long as the slider is in contact with the stop, including the period when slider is pulled back from the contact. For computational efficiency and convergence reasons, 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:

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

Piston Chamber

Wheel and Axle

Elbow

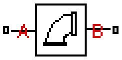
Purpose

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.

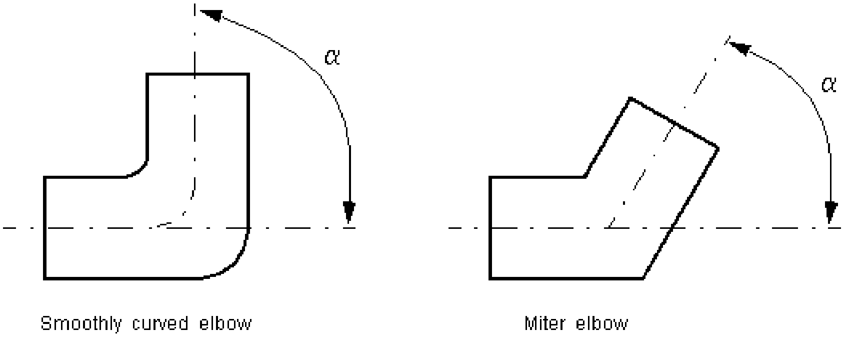
The core data for the pressure loss coefficient computation is the table-specified relationship between the friction factor f_T and the internal diameter for clean commercial steel pipes, with flow in the zone of complete turbulence (see [1], p. A-26). For smoothly curved, standard 90° elbows, the pressure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

$$K = 30 f_T$$

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 ($0 \le \alpha \le 90$).



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

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

For sharp-edged, miter bends the pressure loss coefficient is determined according to the table provided in [1], 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

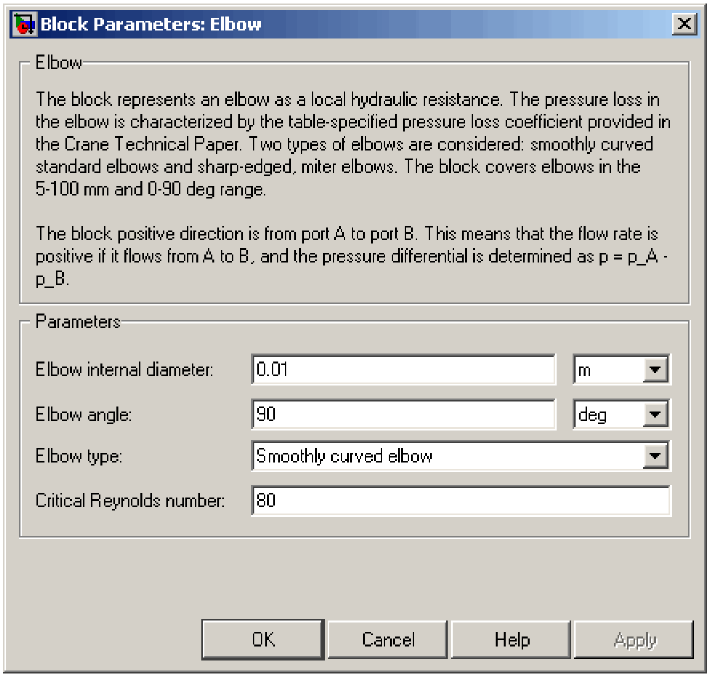
The formulas used in the Elbow block are very approximate, especially in the laminar and transient flow regions. For more accurate results, use the Local Resistance block with a table-specified `K=f(Re)` relationship.

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.

Dialog Box and Parameters



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: 0.01 m

Elbow angle: 90 deg

Elbow type: Smoothly curved elbow

Critical Reynolds number: 80

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

See Also

Gradual Area Change

Local Resistance

Pipe Bend

Sudden Area Change

T-junction

Fixed Orifice

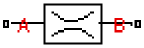
Purpose

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:

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

$$p = p_A - p_B$$

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

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

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

where

q	Flow rate
p	Pressure differential
$p_{A,}p_B$	Gauge pressures at the block terminals

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

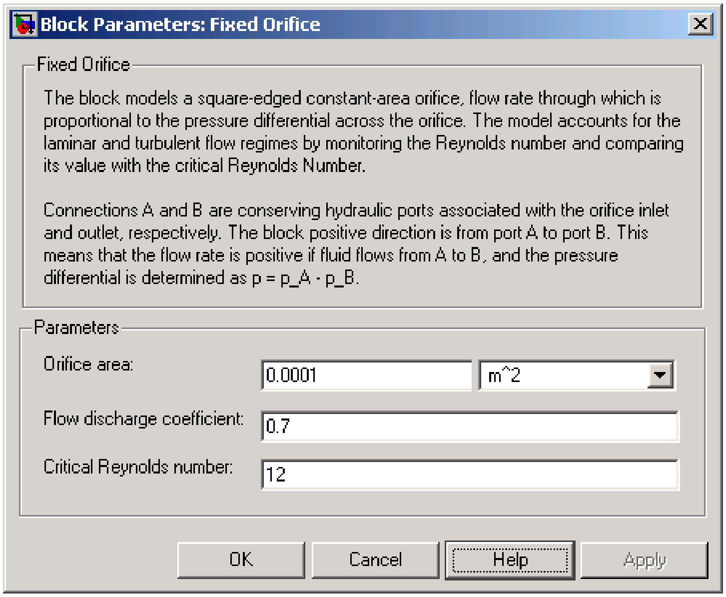
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as $p = p_A - p_B$.

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



Fixed Orifice

The block models a square-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 and comparing its value with the critical Reynolds Number.

Connections A and B are conserving hydraulic ports associated with the orifice inlet and outlet, respectively. The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure differential is determined as p = p_A - p_B.

Parameters

Orifice area: 0.0001 m^2

Flow discharge coefficient: 0.7

Critical Reynolds number: 12

Orifice area

Orifice passage area. The default value is `1e-4` m^2.

Flow discharge coefficient

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

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.

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

Fixed-Displacement Pump

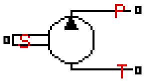
Purpose

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:

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

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

$$k_{leak} = k_{HP} / \nu \bullet \rho$$

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

$$p = p_P - p_T$$

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

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, $\mu = \nu \cdot \rho$

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

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

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

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

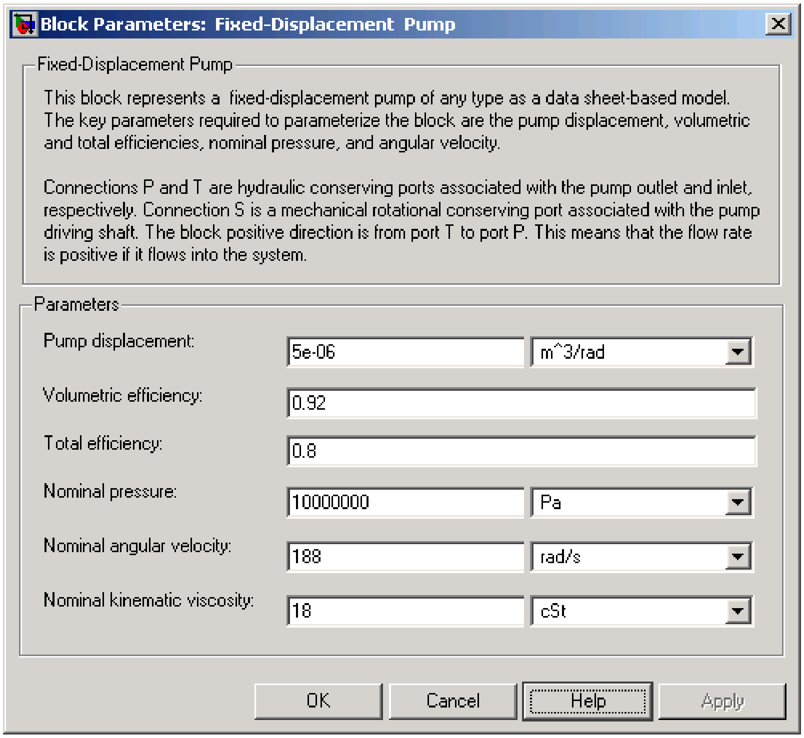
The block positive direction is from port T to port P. This means that the pump transfers fluid from T to P provided that the shaft S rotates in the positive direction. The pressure differential across the pump is determined as $p = p_P - p_T$.

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.

Dialog Box and Parameters



This block represents a fixed-displacement pump of any type as a data sheet-based model. The key parameters required to parameterize the block are the pump displacement, volumetric and total efficiencies, 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 flow rate is positive if it flows into the system.

Pump displacement: 5e-06 m^3/rad

Volumetric efficiency: 0.92

Total efficiency: 0.8

Nominal pressure: 10000000 Pa

Nominal angular velocity: 188 rad/s

Nominal kinematic viscosity: 18 cSt

Pump displacement

Pump displacement. The default value is `5e-6` m^3/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 `1e7` Pa.

Nominal angular velocity
Angular velocity of the driving shaft, at which both the volumetric and total efficiencies are specified. The default value is `188` rad/s.

Nominal kinematic viscosity
Working fluid kinematic viscosity, at which both the volumetric and total efficiencies are specified. The default value is `18` cSt.

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_dspl_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

(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

Gas-Charged Accumulator

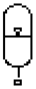
Purpose

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:

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

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

where

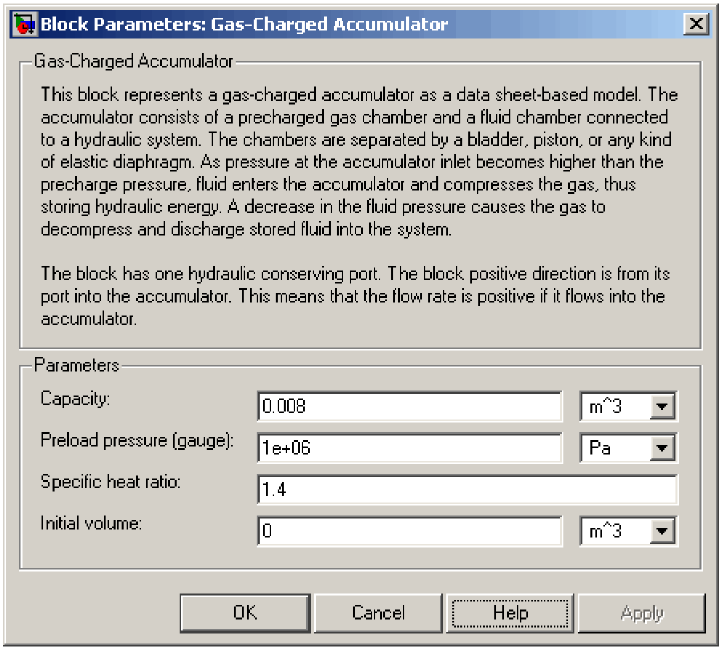
V_F	Fluid volume
V_A	Accumulator capacity
p	Inlet gauge pressure
p_{pr}	Precharge pressure
k	Specific heat ratio
q	Volumetric flow rate
t	Time

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



Gas-Charged Accumulator

This block represents a gas-charged accumulator as a data sheet-based model. 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. As pressure at the accumulator inlet becomes higher than the precharge pressure, fluid enters the accumulator and compresses the gas, thus storing hydraulic energy. A decrease in the fluid pressure causes the gas to decompress and discharge stored fluid into the system.

The block has one hydraulic conserving port. The block positive direction is from its port into the accumulator. This means that the flow rate is positive if it flows into the accumulator.

Parameters

Capacity: 0.008 m^3

Preload pressure (gauge): 1e+06 Pa

Specific heat ratio: 1.4

Initial volume: 0 m^3

Capacity

Accumulator capacity. The default value is `0.008` m^3.

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`.

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

Gradual Area Change

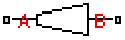
Purpose

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.

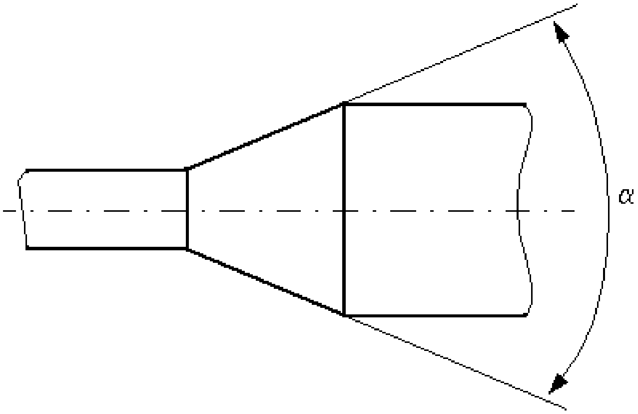
If you choose to apply the semi-empirical formulas, you provide geometric parameters of the resistance, and the pressure loss coefficient is determined according to the A.H. Gibson equations (see [1] and [2]):

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

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

where

K_{GE}	Pressure loss coefficient for the gradual enlargement, which takes place if fluid flows from inlet to outlet
K_{GC}	Pressure loss coefficient for the gradual contraction, which takes place if fluid flows from outlet to inlet
K_{cor}	Correction factor
A_S	Small area
A_L	Large area
α	Enclosed angle



If you choose to specify the pressure loss coefficient by a table, you have to provide a tabulated relationship between the loss coefficient and the Reynolds number. In this case, the loss coefficient is determined by one-dimensional table lookup. You have a choice of 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.

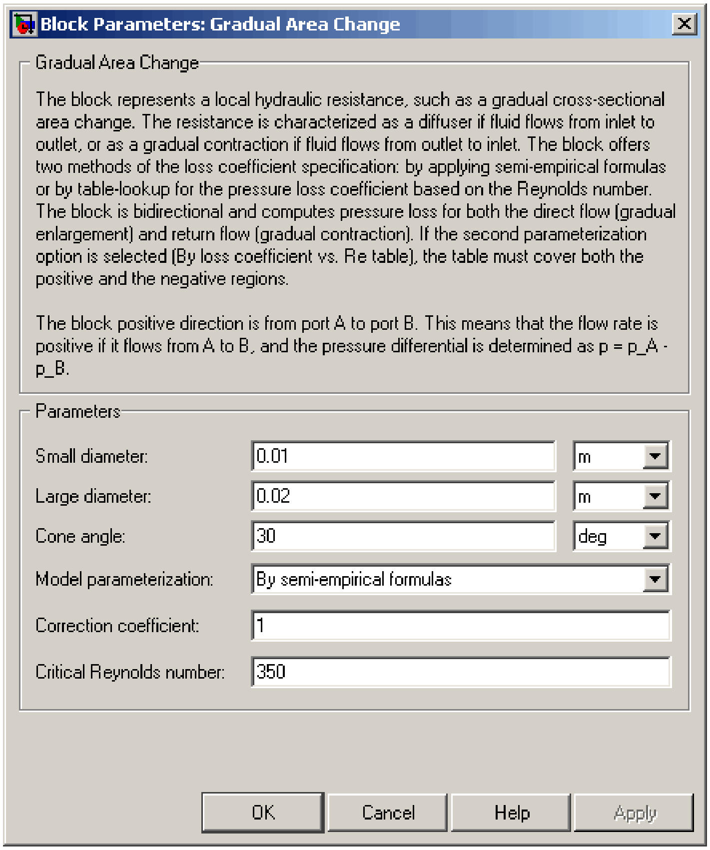
The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

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.

Dialog Box and Parameters



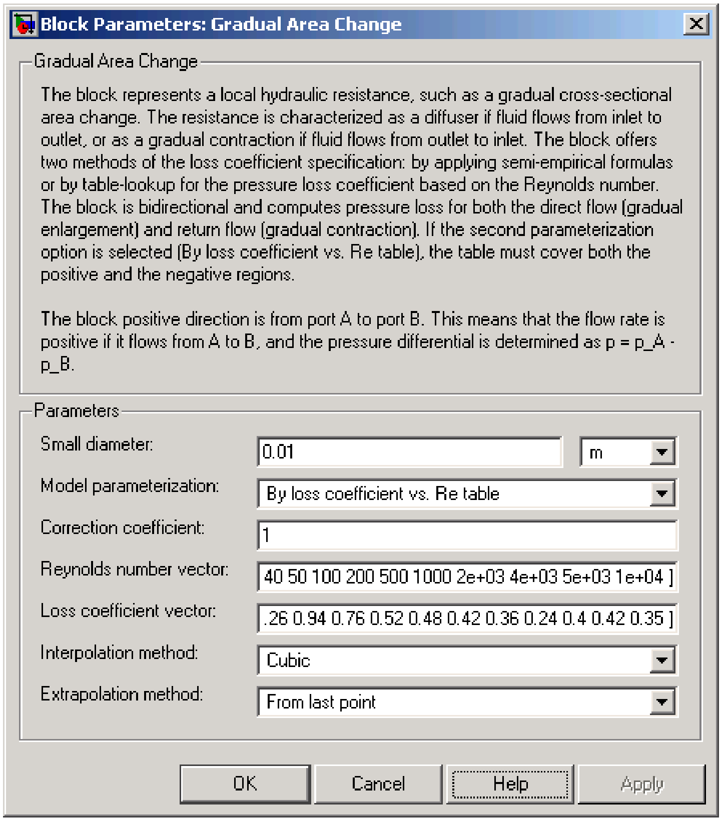
Gradual Area Change

The block represents a local hydraulic resistance, such as a gradual cross-sectional area change. The resistance is characterized as a diffuser if fluid flows from inlet to outlet, or as a gradual 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 (gradual enlargement) and return flow (gradual contraction). If the second parameterization option is selected (By loss coefficient vs. Re table), the table must cover both the positive and the negative 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:	0.01	m
Large diameter:	0.02	m
Cone angle:	30	deg
Model parameterization:	By semi-empirical formulas	
Correction coefficient:	1	
Critical Reynolds number:	350	



Block Parameters: Gradual Area Change

Gradual Area Change

The block represents a local hydraulic resistance, such as a gradual cross-sectional area change. The resistance is characterized as a diffuser if fluid flows from inlet to outlet, or as a gradual 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 (gradual enlargement) and return flow (gradual contraction). If the second parameterization option is selected (By loss coefficient vs. Re table), the table must cover both the positive and the negative 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: 0.01 m

Model parameterization: By loss coefficient vs. Re table

Correction coefficient: 1

Reynolds number vector: 40 50 100 200 500 1000 2e+03 4e+03 5e+03 1e+04]

Loss coefficient vector: .26 0.94 0.76 0.52 0.48 0.42 0.36 0.24 0.4 0.42 0.35]

Interpolation method: Cubic

Extrapolation method: From last point

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`.

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. 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

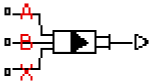
Purpose

Simulate double-acting hydraulic actuator for cartridge valves

Library

Valve Actuators

Description



Use the Hydraulic Cartridge Valve Actuator block as a pilot actuator for cartridge valves, as well as pilot-operated pressure and control valves in applications where all the forces, except spring and pressure forces, and flow consumption can be neglected. This block represents a double-acting hydraulic valve actuator driven by three pressures. The actuator drives a valve (spool, poppet, etc.) whose position depends on pressures at ports A, B, and X and the force of the spring. Pressures at ports A and B tend to open the valve, while pressure at control port X together with the spring force act to close it.

The valve remains closed as long as the aggregate pressure force is lower than the spring preload force. The poppet is forced off its seat as the preload force is reached and moves up proportionally to pressure increase until it passes the full stroke.

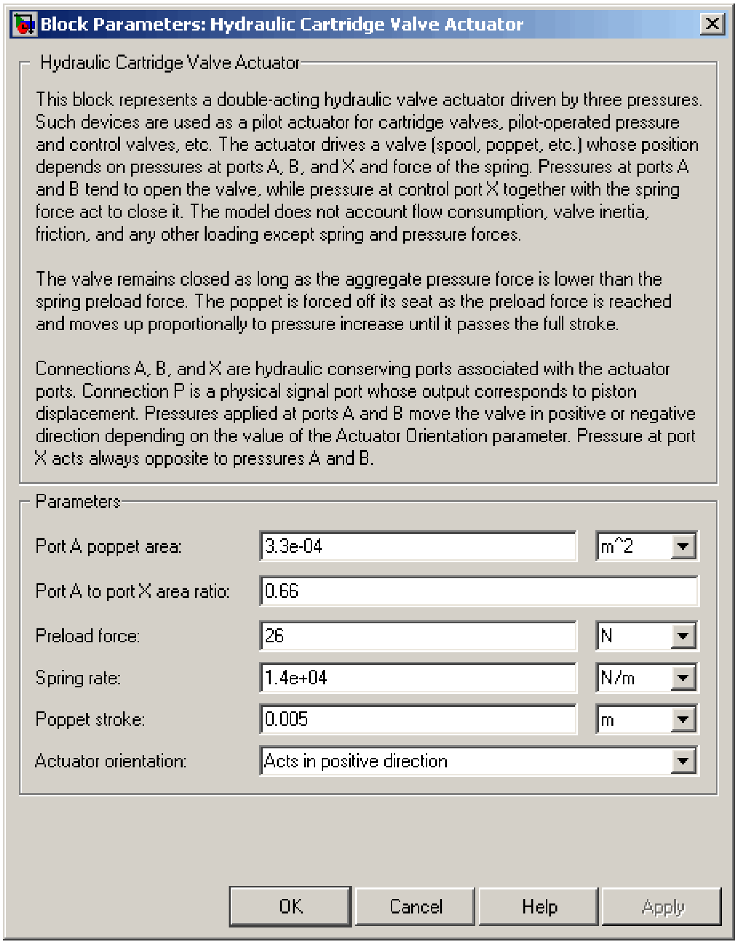
Connections A, B, and X are hydraulic conserving ports associated with the actuator ports. Connection P is a physical signal port whose output corresponds to piston displacement. Pressures applied at ports A and B move the piston in the positive or negative direction, depending on the value of the **Actuator orientation** parameter, with pressure at port X acting in the opposite direction.

Basic Assumptions and Limitations

The 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.

Dialog Box and Parameters



Hydraulic Cartridge Valve Actuator

This block represents a double-acting hydraulic valve actuator driven by three pressures. Such devices are used as a pilot actuator for cartridge valves, pilot-operated pressure and control valves, etc. The actuator drives a valve (spool, poppet, etc.) whose position depends on pressures at ports A, B, and X and force of the spring. Pressures at ports A and B tend to open the valve, while pressure at control port X together with the spring force act to close it. The model does not account flow consumption, valve inertia, friction, and any other loading except spring and pressure forces.

The valve remains closed as long as the aggregate pressure force is lower than the spring preload force. The poppet is forced off its seat as the preload force is reached and moves up proportionally to pressure increase until it passes the full stroke.

Connections A, B, and X are hydraulic conserving ports associated with the actuator ports. Connection P is a physical signal port whose output corresponds to piston displacement. Pressures applied at ports A and B move the valve in positive or negative direction depending on the value of the Actuator Orientation parameter. Pressure at port X acts always opposite to pressures A and B.

Parameters

Port A poppet area: 3.3e-04 m^2
Port A to port X area ratio: 0.66
Preload force: 26 N
Spring rate: 1.4e+04 N/m
Poppet stroke: 0.005 m
Actuator orientation: Acts in positive direction

Port A poppet area

Effective poppet area at port A. The parameter value must be greater than zero. The default value is `3.3e-4` m^2.

Port A to port X area ratio

Ratio between poppet areas at port A and port X. The parameter value must be greater than zero. The default value is `0.66`.

Preload force
Spring preload force. The default value is `26` N.

Spring rate
Spring rate. The default value is `1.4e4` N/m.

Poppet stroke
Maximum poppet stroke. The parameter value must be greater than or equal to zero. The default value is `5e-3` m.

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 pressures applied at ports A and B move the poppet in the negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

Restricted Parameters

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

- **Actuator orientation**

All other block parameters are available for modification.

Ports

The block has the following ports:

`A`
Hydraulic conserving port associated with the valve inlet.

`B`
Hydraulic conserving port associated with the valve outlet.

`X`
Hydraulic conserving port associated with the valve control terminal.

P

Physical signal port that outputs poppet displacement.

See Also

2-Position Valve Actuator

3-Position Valve Actuator

Cartridge Valve Insert

Hydraulic Double-Acting Valve Actuator

Hydraulic Single-Acting Valve Actuator

Proportional and Servo-Valve Actuator

Hydraulic Double-Acting Valve Actuator

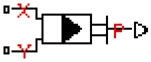
Purpose

Simulate double-acting hydraulic valve actuator

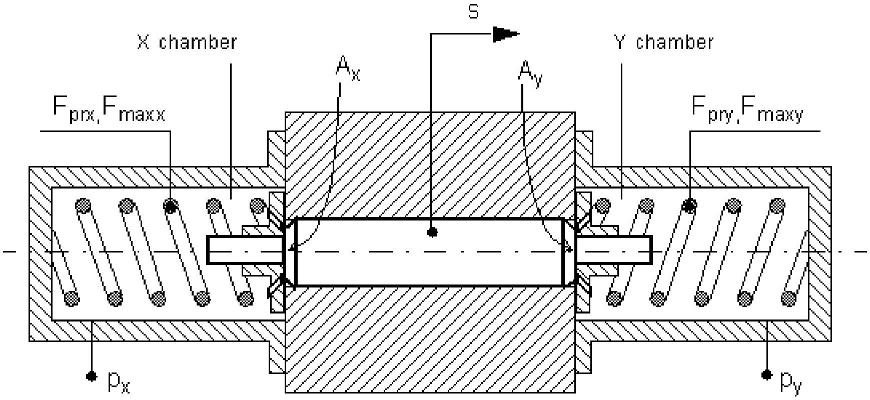
Library

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 applied at port Y shifts the valve in the *y*-direction, overcoming the spring located in the X chamber.

The actuator is simulated according to the following equations:

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

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

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

If $F >= 0$,

$$s = \begin{cases} 0 & \text{for } F <= F_{pry} \\ L_y \bullet \left(F - F_{pry}\right) \bullet or & \text{for } F_{pry} < F < F_{\max y} \\ str_y \bullet or & \text{for } F >= F_{\max y} \end{cases}$$

If $F < 0$,

$$s = \begin{cases} 0 & \text{for } |F| <= F_{prx} \\ -L_x \bullet \left(|F| - F_{prx}\right) \bullet or & \text{for } F_{prx} < |F| < F_{\max x} \\ -str_x \bullet or & \text{for } |F| >= F_{\max x} \end{cases}$$

where

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

A_y	Valve face area in the Y chamber
str_x	Valve stroke in *x*-direction
str_y	Valve stroke in *y*-direction
F_{prx}	Chamber X spring preload force
F_{maxx}	Chamber X spring maximum force
F_{pry}	Chamber Y spring preload force
F_{maxy}	Chamber Y spring maximum force
or	Actuator orientation with respect to the globally assigned positive direction. If pressure applied at port X moves the piston in positive direction, *or* equals 1. If pressure applied at port X moves the piston in negative direction, *or* equals –1.

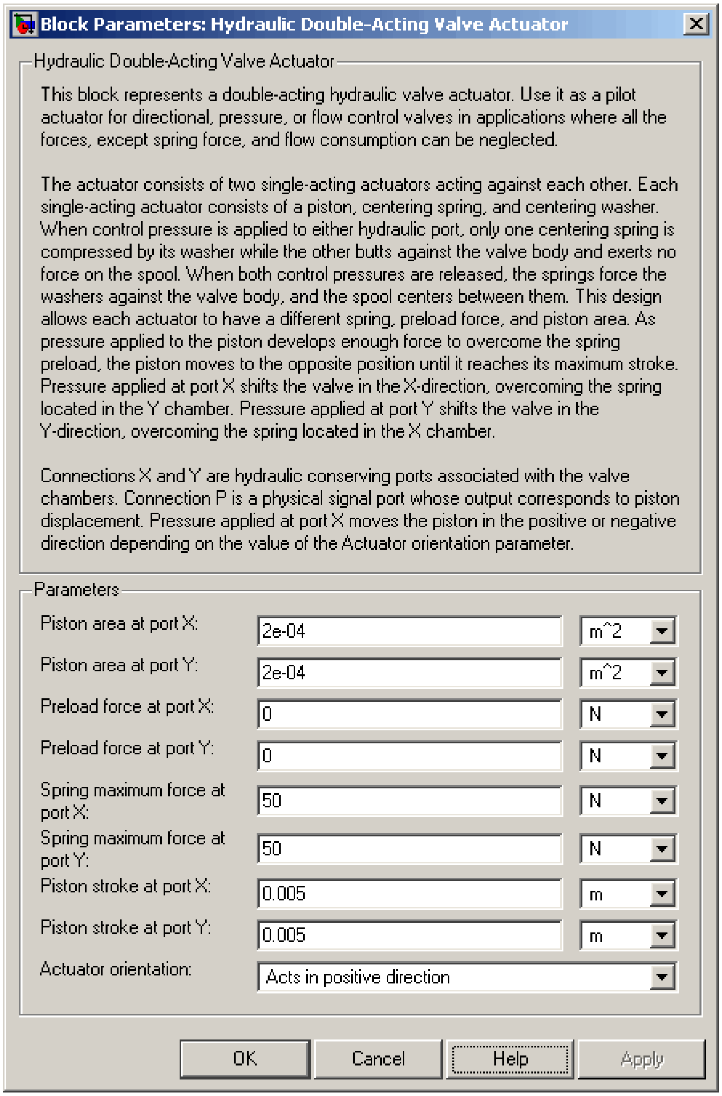
Connections X and Y are hydraulic conserving ports associated with the valve chambers. Connection P is a physical signal port whose output corresponds to piston displacement. Pressure applied at port X moves the piston in the positive or negative direction depending on the value of the **Actuator orientation**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

Basic Assumptions and Limitations

The 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.

Dialog Box and Parameters



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:	2e-04	m^2
Piston area at port Y:	2e-04	m^2
Preload force at port X:	0	N
Preload force at port Y:	0	N
Spring maximum force at port X:	50	N
Spring maximum force at port Y:	50	N
Piston stroke at port X:	0.005	m
Piston stroke at port Y:	0.005	m
Actuator orientation:	Acts in positive direction	

Piston area at port X
Effective piston area at port X. The parameter value must be greater than zero. The default value is `2e-4` m^2.

Piston area at port Y
Effective piston area at port Y. The parameter value must be greater than zero. The default value is `2e-4` m^2.

Preload force at port X
Spring preload force at port X. The default value is `0`.

Preload force at port Y
Spring preload force at port Y. The default value is `0`.

Spring maximum force at port X
Chamber X spring maximum force. The parameter value must be greater than the spring preload force. The default value is `50` N.

Spring maximum force at port Y
Chamber Y spring maximum force. The parameter value must be greater than the spring preload force. The default value is `50` N.

Piston stroke at port X
Piston stroke in chamber X. The parameter value must be greater than or equal to zero. The default value is `5e-3` m.

Piston stroke at port Y
Piston stroke in chamber Y. The parameter value must be greater than or equal to zero. The default value is `5e-3` m.

Actuator orientation
Specifies actuator orientation with respect to the globally assigned positive direction. The actuator can be installed in two different ways, depending upon whether it moves the piston in the positive or in the negative direction when pressure is applied at its inlet. If pressure applied at port X moves the piston in the negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

Restricted Parameters

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

- **Actuator orientation**

All other block parameters are available for modification.

Ports

The block has the following ports:

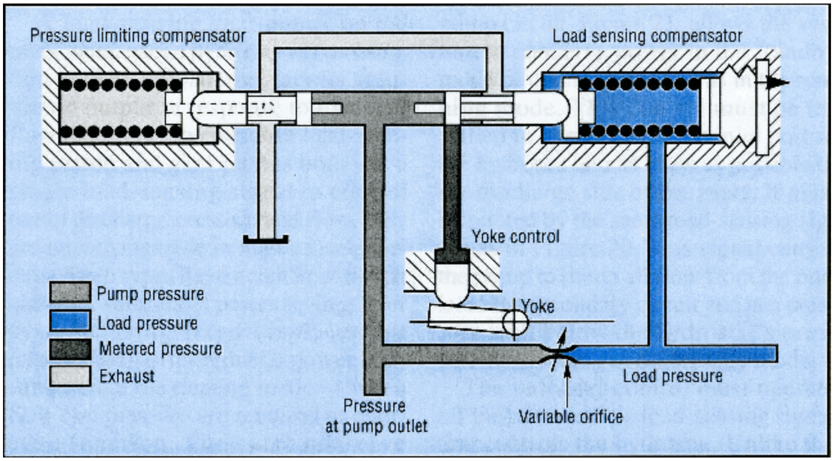
X
: Hydraulic conserving port associated with the valve X chamber.

Y
: Hydraulic conserving port associated with the valve Y chamber.

P
: Physical signal port that outputs piston displacement.

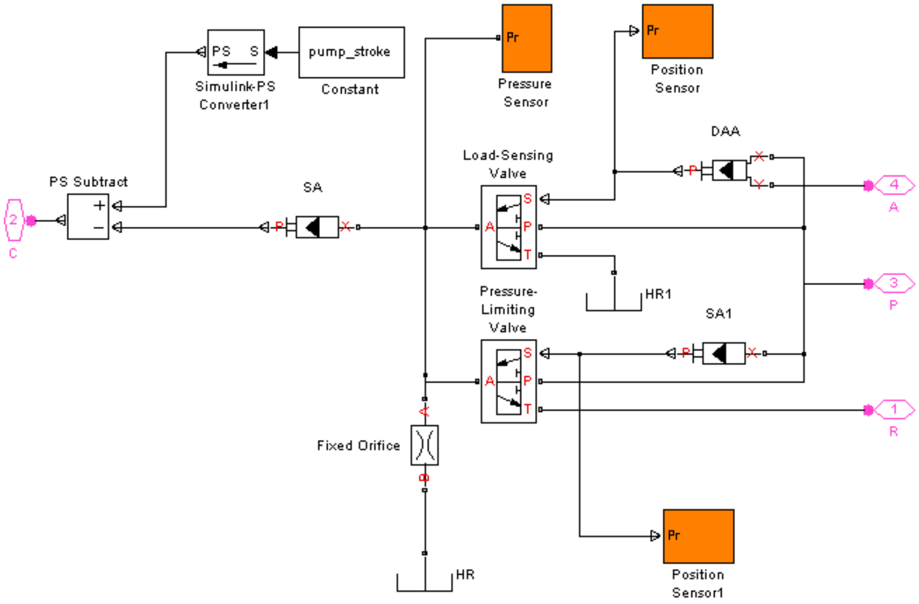
Examples

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



The Hydraulic 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. Yeapple, *Fluid Power Design Handbook*, Marcel Dekker, Inc., 1995

See Also

2-Position Valve Actuator

3-Position Valve Actuator

Hydraulic Single-Acting Valve Actuator

Proportional and Servo-Valve Actuator

Hydraulic Fluid

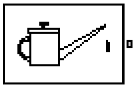
Purpose

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:

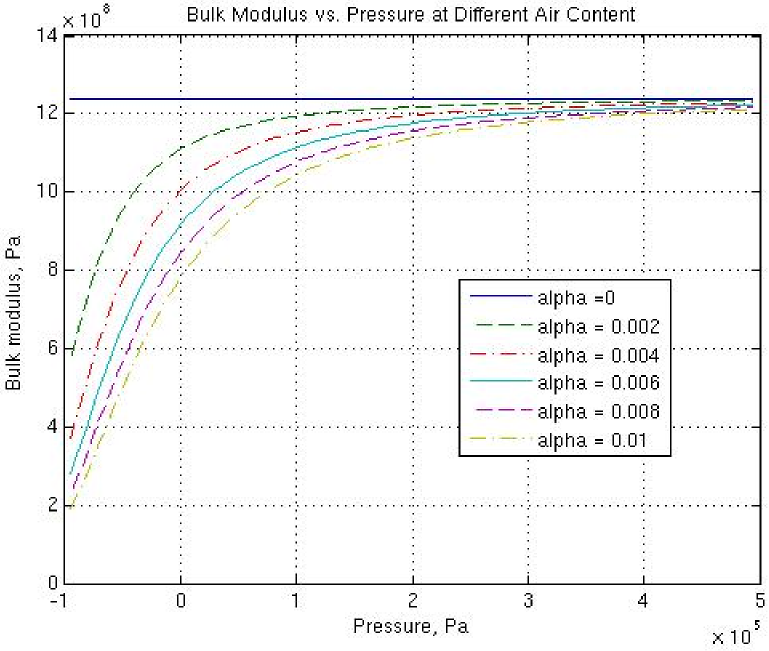
$$E = E_l \frac{1 + \alpha \left(\frac{p_a}{p_a + p} \right)^{1/n}}{1 + \alpha \frac{p_a^{1/n}}{n \bullet (p_a + p)^{\frac{n+1}{n}}} E_l}$$

where

E_1	Pure liquid bulk modulus
p_α	Atmospheric pressure
α	Relative gas content at atmospheric pressure, $\alpha = V_G/V_L$
V_G	Gas volume at atmospheric pressure

V_L	Volume of liquid
n	Gas-specific heat ratio

The main objective of representing fluid as a mixture of liquid and gas is to introduce an approximate model of cavitation, which takes place in a chamber if pressure drops below fluid vapor saturation level. As it is seen in the graph below, the bulk modulus of a mixture decreases at $p \to p_a$, thus considerably slowing down further pressure change. At high pressure, $p >> p_a$, a small amount of nondissolved gas has practically no effect on the system behavior.



Cavitation is an inherently thermodynamic process, requiring consideration of multiple-phase fluids, heat transfers, etc., and as such cannot be accurately simulated with SimHydraulics® software. But the simplified version implemented in the block is good enough to signal if pressure falls below dangerous level, and to prevent computation failure that normally occurs at negative pressures.

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.

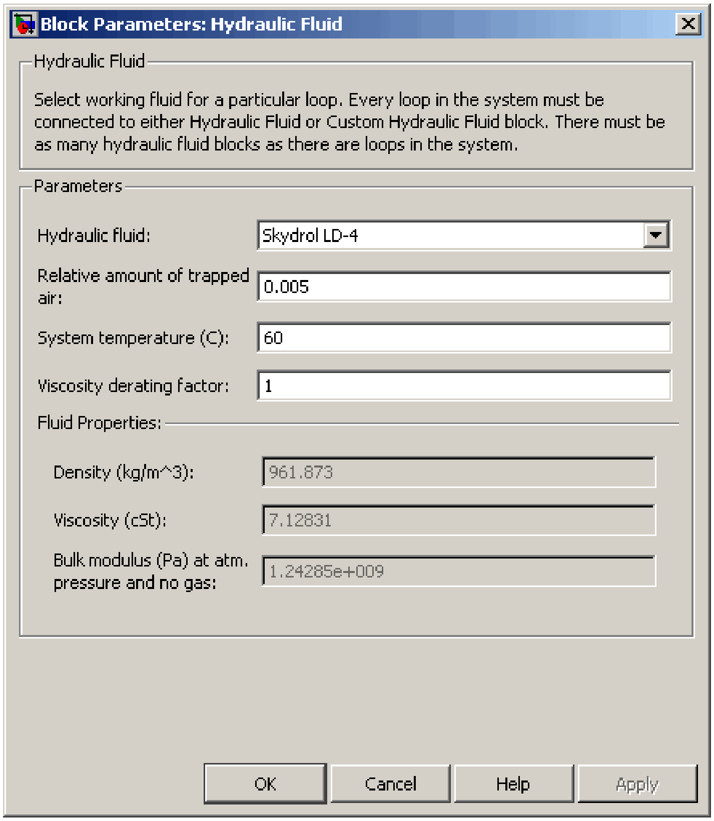
The Hydraulic Fluid block offers a selection of predefined fluids. See "Examples" on page 2-163 for how you can get information on the fluid properties used in the block. Once you select a fluid name, you can also specify the temperature of the fluid and the relative amount of entrained, nondissolved gas.

The Hydraulic Fluid block has one port. You can connect it to a hydraulic diagram by branching a connection line off the main line and connecting it to the port. When you connect the Hydraulic Fluid block to a hydraulic line, the software automatically identifies the hydraulic blocks connected to the particular loop and propagates the hydraulic fluid properties to all the hydraulic blocks in the loop.

Each topologically distinct hydraulic loop in a diagram requires 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 60°C and with a 0.005 ratio of entrapped air.

Dialog Box and Parameters



Block Parameters: Hydraulic Fluid

Hydraulic Fluid

Select working fluid for a particular loop. Every loop in the system must be connected to either Hydraulic Fluid or Custom Hydraulic Fluid block. There must be as many hydraulic fluid blocks as there are loops in the system.

Parameters

Hydraulic fluid: Skydrol LD-4

Relative amount of trapped air: 0.005

System temperature (C): 60

Viscosity derating factor: 1

Fluid Properties:

Density (kg/m^3): 961.873

Viscosity (cSt): 7.12831

Bulk modulus (Pa) at atm. pressure and no gas: 1.24285e+009

Hydraulic fluid

Hydraulic fluid type. Select one of the predefined fluids:

- `Skydrol LD-4` (default)
- `Skydrol 500B-4`
- `Skydrol-5`
- `HyJet-4A`
- `Fluid MIL-F-83282`

- `Fluid MIL-F-5606`
- `Fluid MIL-F-87257`
- `Oil-10W`
- `Oil-30W`
- `Oil-50W`
- `Oil SAE-30`
- `Oil SAE-50`
- `Transmission fluid ATF (Dexron III)`
- `ISO VG 22 (ESSO UNIVIS N 22)`
- `ISO VG 32 (ESSO UNIVIS N 32)`
- `ISO VG 46 (ESSO UNIVIS N 46)`
- `Brake fluid DOT3`
- `Brake fluid DOT4`
- `Brake fluid DOT5`
- `Gasoline`
- `Diesel fuel`
- `Jet fuel`
- `Water-Glycol 60/40`
- `Water`

Relative amount of trapped air

Amount of entrained, nondissolved gas in the fluid. The amount is specified as the ratio of gas volume at normal conditions to the fluid volume in the chamber. The default value is `0.005`.

System temperature

Fluid temperature (C). The default value is `60`.

Viscosity derating factor

Proportionality coefficient that you can use to adjust fluid viscosity, if needed. Specify a value between 0.5 and 1.5. The default value is `1`.

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:

   ```
   props = sh_stockfluidproperties
   ```

   The system responds with a list of available fluids:

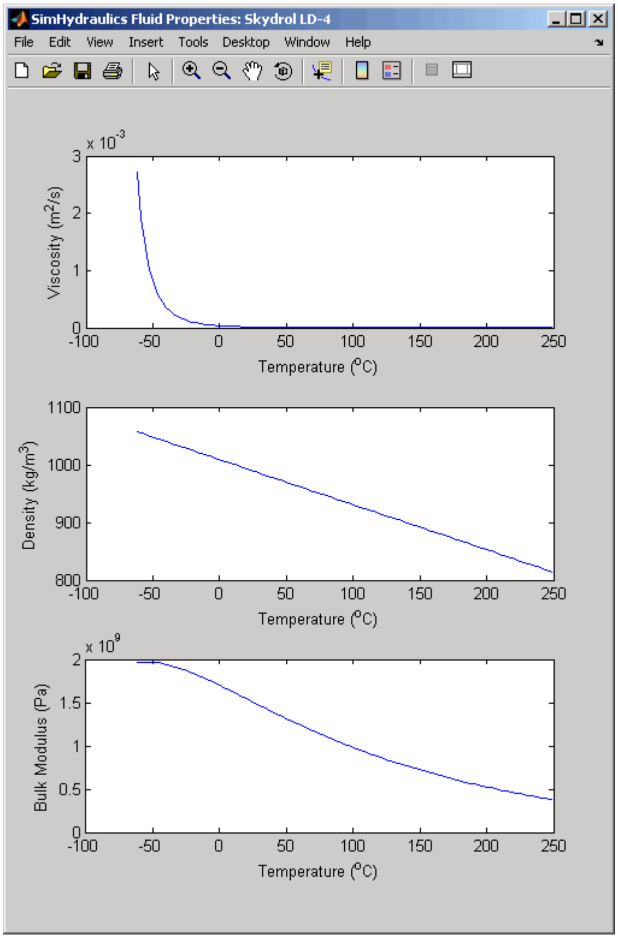
   ```
   props =

        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

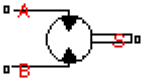
Purpose

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:

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

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

$$k_{leak} = k_{HP} / \nu \bullet \rho$$

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

$$p = p_A - p_B$$

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, $\mu = \nu \cdot \rho$

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 efficiency by assuming that the hydraulic efficiency is negligibly small

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

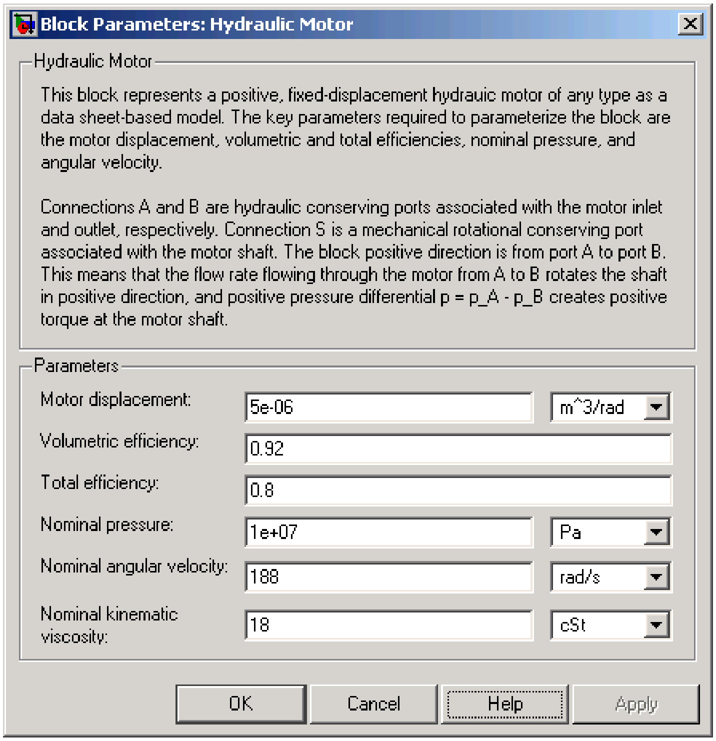
The block hydraulic positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and rotates the output shaft in the globally assigned positive direction. The pressure differential across the motor is determined as $p = p_A - p_B$, and positive pressure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



Hydraulic Motor

This 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 the motor displacement, volumetric and total efficiencies, nominal pressure, and 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 shaft. The block positive direction is from port A to port B. This means that the flow rate flowing through the motor from A to B rotates the shaft in positive direction, and positive pressure differential p = p_A - p_B creates positive torque at the motor shaft.

Parameters

Motor displacement: 5e-06 m^3/rad

Volumetric efficiency: 0.92

Total efficiency: 0.8

Nominal pressure: 1e+07 Pa

Nominal angular velocity: 188 rad/s

Nominal kinematic viscosity: 18 cSt

Motor displacement

Motor displacement. The default value is `5e-6` m^3/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`.

Nominal pressure

Pressure differential across the motor, at which both the volumetric and total efficiencies are specified. The default value is `1e7` 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

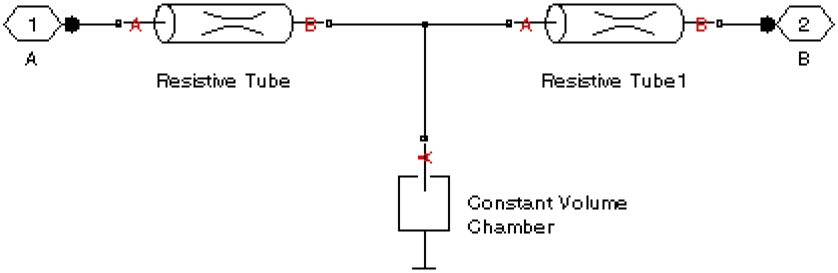
Library

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 Simscape Foundation library 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.

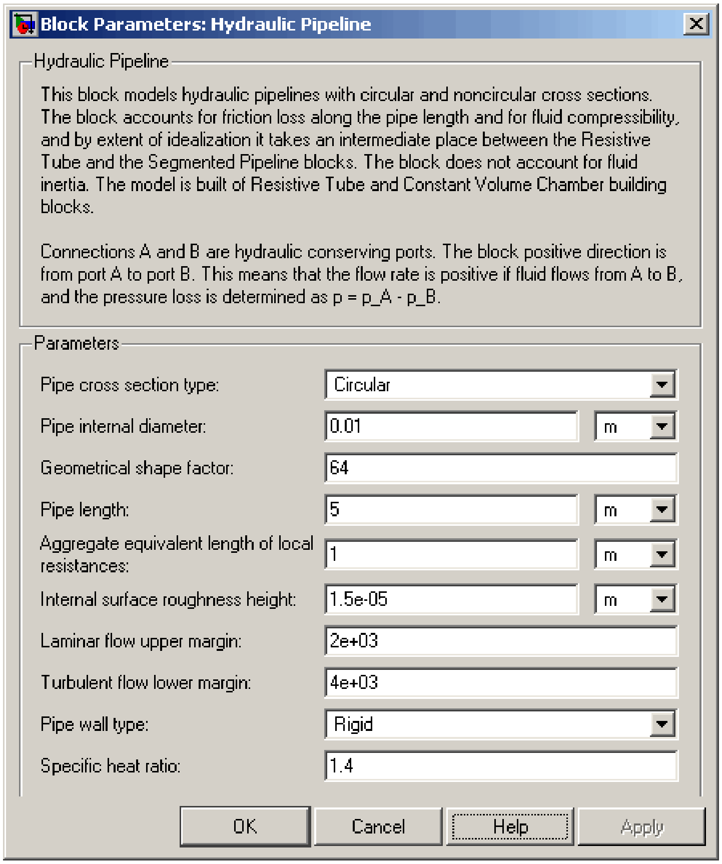
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

Basic Assumptions and Limitations

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.

Dialog Box and Parameters



Block Parameters: Hydraulic Pipeline

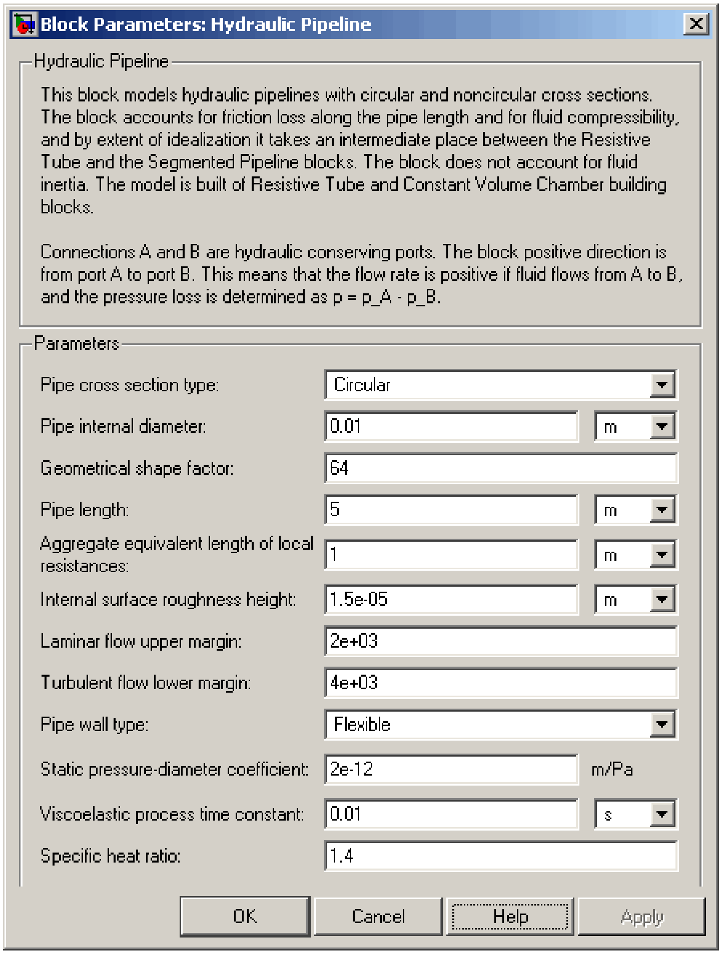
Hydraulic Pipeline

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:	Rigid	
Specific heat ratio:	1.4	



Block Parameters: Hydraulic Pipeline

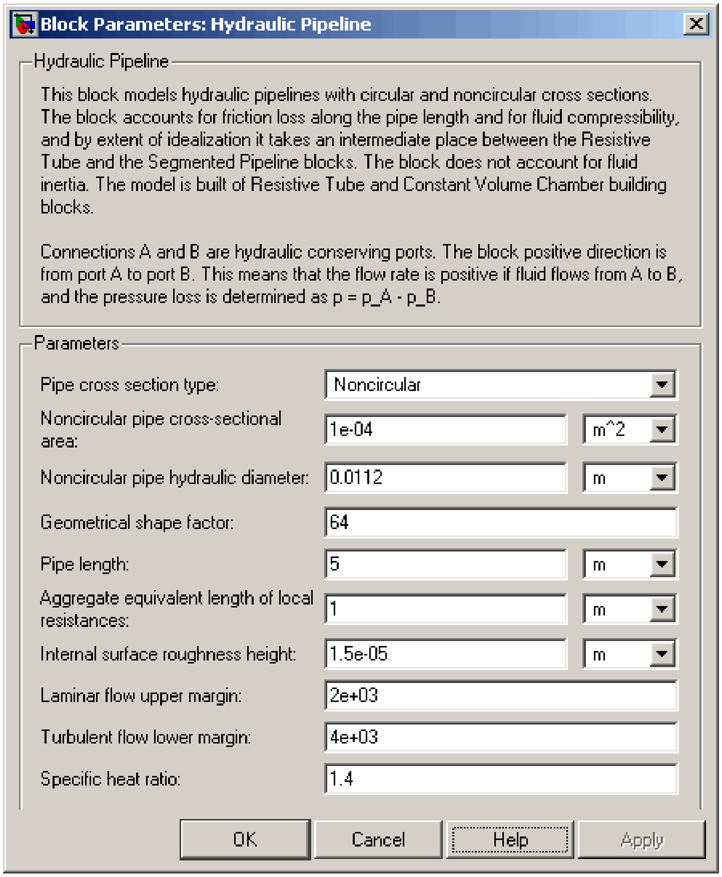
Hydraulic Pipeline

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	



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:	Noncircular	
Noncircular pipe cross-sectional area:	1e-04	m^2
Noncircular pipe hydraulic diameter:	0.0112	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	
Specific heat ratio:	1.4	

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` m^2.

Noncircular pipe hydraulic diameter

Hydraulic diameter of the pipe cross section. The parameter is used if **Pipe cross section type** is set to `Noncircular`. The default value is `0.0112` m.

Geometrical shape factor

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. This length is added to the geometrical pipe length only for hydraulic resistance computation. The fluid volume depends on pipe geometrical length only. The default value is `1` m.

Internal surface roughness height

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

Laminar flow upper margin

Specifies the Reynolds number at which the laminar flow regime is assumed to start converting into turbulent. Mathematically, this is the maximum Reynolds number at fully developed laminar flow. The default value is `2000`.

Turbulent flow lower margin

Specifies the Reynolds number at which the turbulent flow regime is assumed to be fully developed. Mathematically, this is the minimum Reynolds number at turbulent flow. The default value is `4000`.

Pipe wall type

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

Static pressure-diameter coefficient

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

Viscoelastic process time constant

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

Specific heat ratio
Gas-specific heat ratio for the Constant Volume 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

See Also

Linear Hydraulic Resistance

Resistive Tube

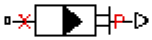
Segmented Pipeline

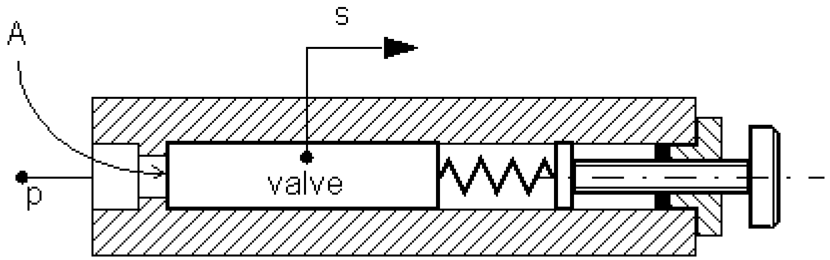
Hydraulic Single-Acting Valve Actuator

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:

$$F = p \bullet A$$

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

$$s = \begin{cases} 0 & \text{for } F <= F_{pr} \\ L \bullet \left(F - F_{pr}\right) \bullet or & \text{for } F_{pr} < F < F_{\max} \\ stroke \bullet or & \text{for } F >= F_{\max} \end{cases}$$

where

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

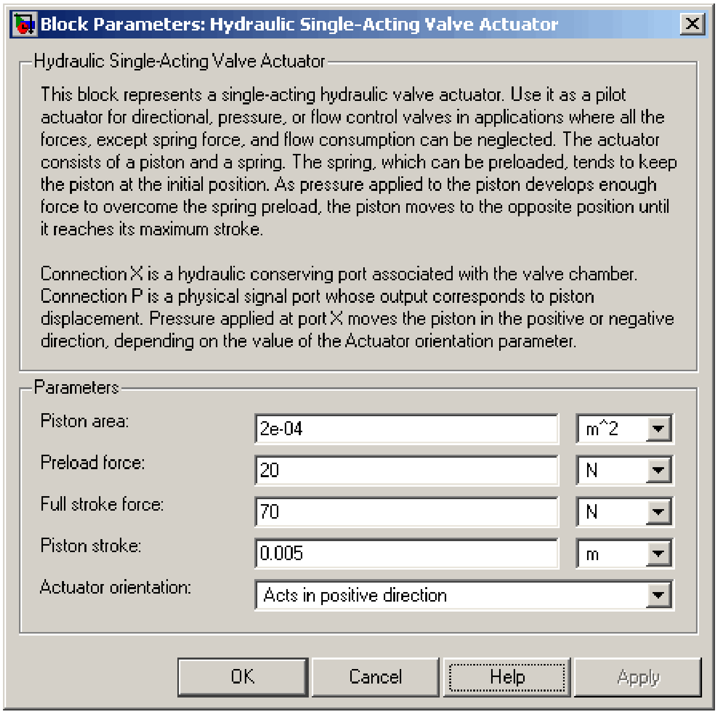
Connection X is a hydraulic conserving port associated with the valve chamber. Connection P is a physical signal port whose output corresponds to piston displacement. Pressure applied at port X moves the piston in the positive or negative direction, depending on the value of the **Actuator orientation**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

Basic Assumptions and Limitations

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.

Dialog Box and Parameters



Block Parameters: Hydraulic Single-Acting Valve Actuator

Hydraulic Single-Acting Valve Actuator

This block represents a single-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 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.

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.

Parameters

Piston area:	2e-04	m^2
Preload force:	20	N
Full stroke force:	70	N
Piston stroke:	0.005	m
Actuator orientation:	Acts in positive direction	

Piston area
: Effective piston area. The default value is `2e-4` m^2.

Preload force
: Spring preload force. The default value is `20` N.

Full stroke force
: Force necessary to move the piston to maximum stroke. The default value is `70` N.

Piston stroke
: Piston stroke. The default value is `5e-3` m.

Actuator orientation
: Specifies actuator orientation with respect to the globally assigned positive direction. The actuator can be installed in two different

ways, depending upon whether it moves the piston in the positive or in the negative direction when pressure is applied at its inlet. If pressure applied at port X moves the piston in the negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

Restricted Parameters

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

- **Actuator orientation**

All other block parameters are available for modification.

Ports

The block has the following ports:

`X`

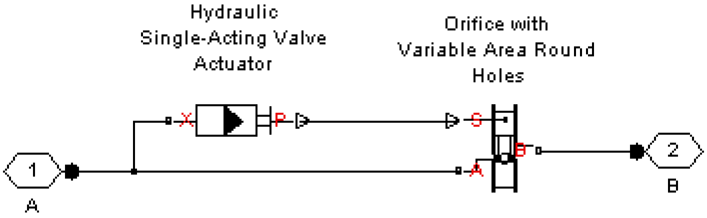
Hydraulic conserving port associated with the valve chamber.

`P`

Physical signal port that outputs piston displacement.

Examples

The following example shows a model of a pressure-relief valve built using the Hydraulic Single-Acting Valve Actuator and Orifice with Variable Area Round Holes blocks.



See Also

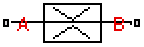
2-Position Valve Actuator
3-Position Valve Actuator
Hydraulic Double-Acting Valve Actuator
Proportional and Servo-Valve Actuator

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:

$$p = \begin{cases} K \dfrac{\rho}{2A^2} q\,|\,q\,| & \text{for } Re > Re_{\text{cr}} \\ K \bullet \mathrm{Re}_{cr} \dfrac{\nu \bullet \rho}{2D_H \bullet A} q & \text{for } Re <= Re_{\text{cr}} \end{cases}$$

$$p = p_A - p_B$$

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

$$\mathrm{Re} = \frac{q \bullet D_H}{A \bullet \nu}$$

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

where

q	Flow rate
p	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.

Connections A and B are conserving hydraulic ports associated with the block inlet and outlet, respectively.

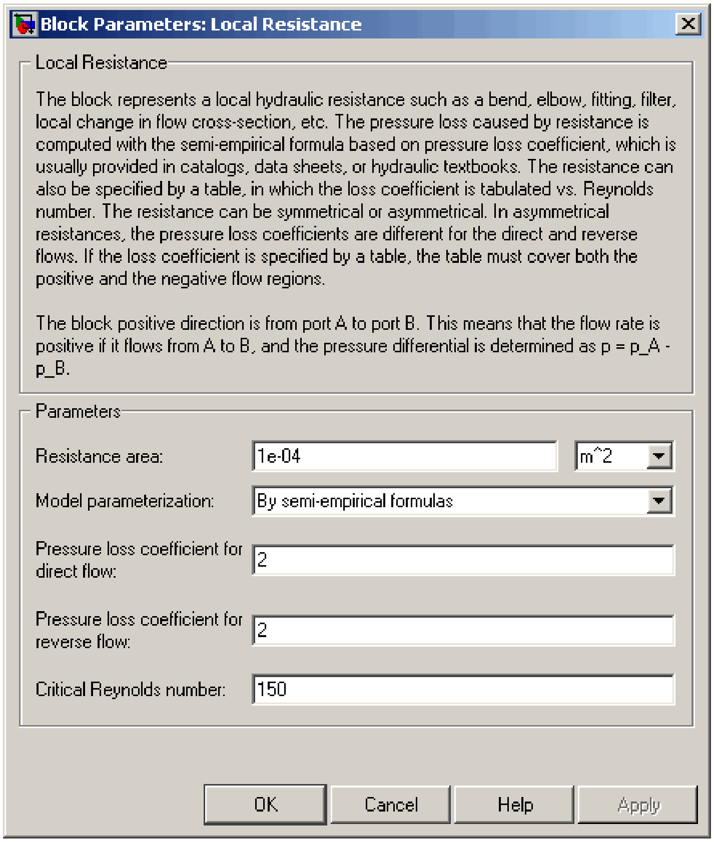
The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

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



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

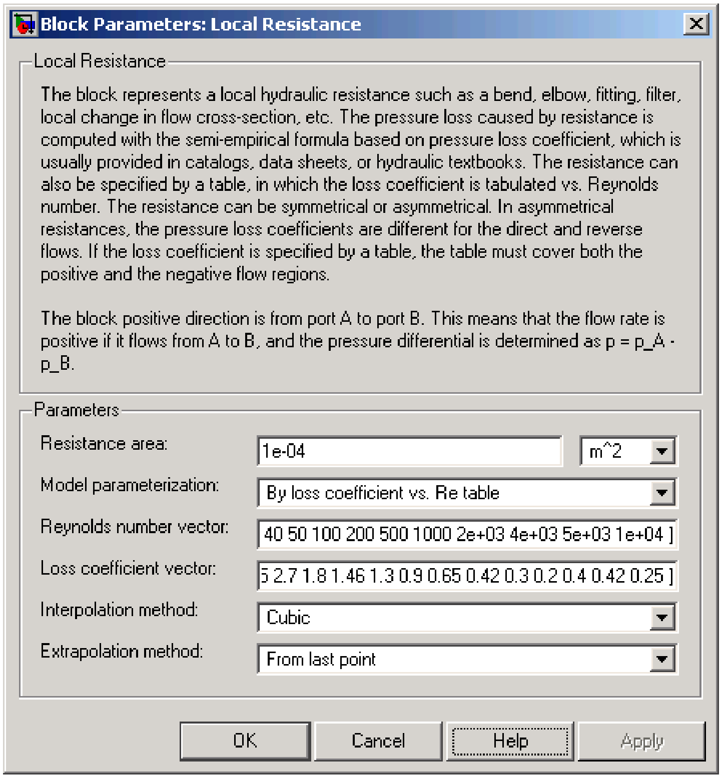
Resistance area: 1e-04 m^2

Model parameterization: By semi-empirical formulas

Pressure loss coefficient for direct flow: 2

Pressure loss coefficient for reverse flow: 2

Critical Reynolds number: 150



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: 1e-04 m^2

Model parameterization: By loss coefficient vs. Re table

Reynolds number vector: 40 50 100 200 500 1000 2e+03 4e+03 5e+03 1e+04]

Loss coefficient vector: 5 2.7 1.8 1.46 1.3 0.9 0.65 0.42 0.3 0.2 0.4 0.42 0.25]

Interpolation method: Cubic

Extrapolation method: From last point

Resistance area

The smallest passage area. The default value is `1e-4` m^2.

Model parameterization

Select one of the following methods for specifying the pressure loss coefficient:

- `By semi-empirical formulas` — Provide a scalar value for the pressure loss coefficient. For asymmetrical resistances, you have to provide separate coefficients for direct and reverse flow. This is the default method.

- `By loss coefficient vs. Re table` — Provide tabulated data of loss coefficients and corresponding Reynolds numbers. The loss coefficient is determined by one-dimensional table lookup. You have a choice of 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 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. 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

See Also

Elbow

Gradual Area Change

Pipe Bend

Sudden Area Change

T-junction

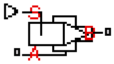
Purpose

Simulate hydraulic needle valve

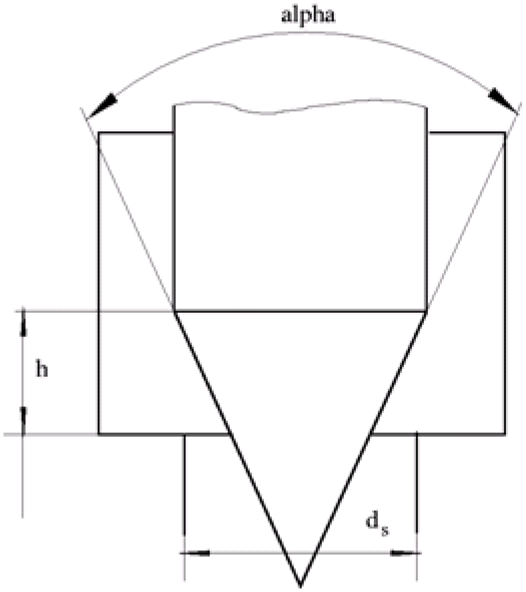
Library

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:

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

$$h = x_0 + x$$

$$h_{\max} = \frac{d_s}{\tan\left(\alpha/2\right)}$$

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

$$p = p_A - p_B$$

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

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

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

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

where

q	Flow rate
p	Pressure differential
p_A, p_B	Gauge pressures at the block terminals
C_D	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
x_0	Initial opening
x	Needle displacement from initial position
h	Valve opening
h_{max}	Maximum needle stroke
d_s	Orifice diameter
α	Needle angle
ρ	Fluid density
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.

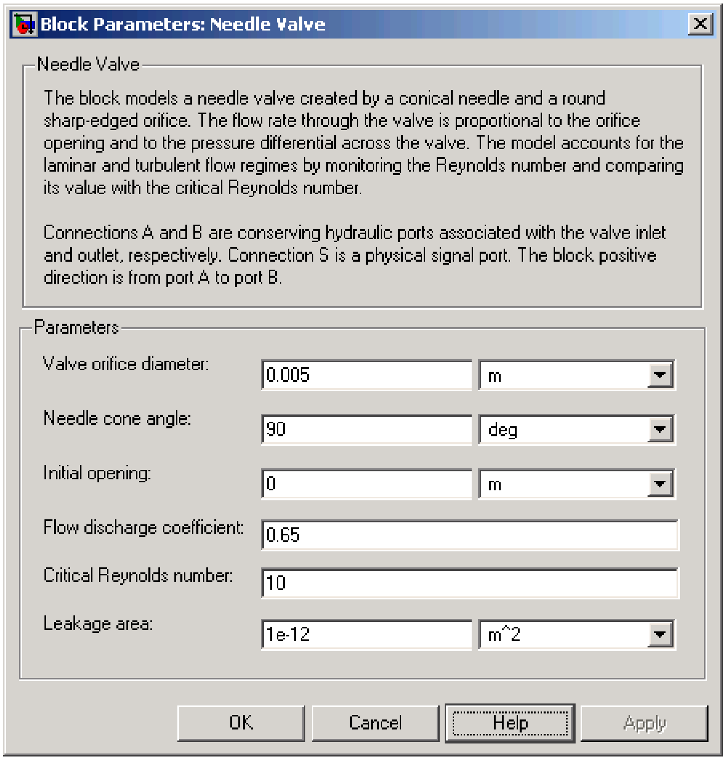
Basic Assumptions and Limitations

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.

Dialog Box and Parameters



Needle Valve

The block models a needle valve created by a conical needle 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 orifice diameter: 0.005 m

Needle cone angle: 90 deg

Initial opening: 0 m

Flow discharge coefficient: 0.65

Critical Reynolds number: 10

Leakage area: 1e-12 m^2

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` 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

Ball Valve

Poppet Valve

Pressure-Compensated Flow Control Valve

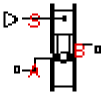
Purpose

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

Library

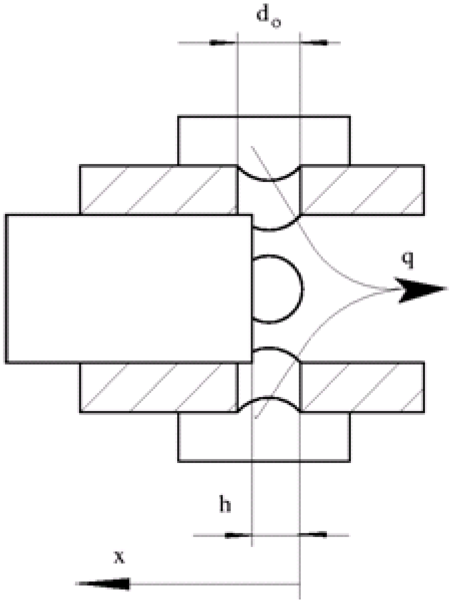
Orifices

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



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:

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

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

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

$$p = p_A - p_B$$

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

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

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

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

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

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

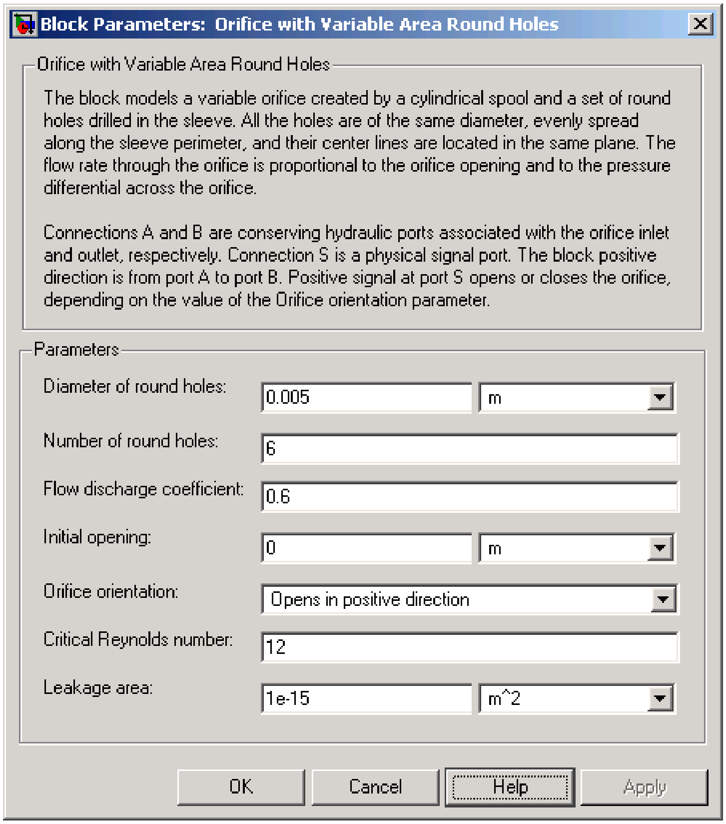
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as $p = p_A - p_B$. Positive signal at the physical signal port `S` opens or closes the orifice depending on the value of the parameter **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 $Re=Re_{cr}$.

Dialog Box and Parameters



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: 0.005 m

Number of round holes: 6

Flow discharge coefficient: 0.6

Initial opening: 0 m

Orifice orientation: Opens in positive direction

Critical Reynolds number: 12

Leakage area: 1e-15 m^2

Diameter of round holes

Diameter of the orifice holes. The default value is `5e-3` m.

Number of round holes

Number of holes. The default value is `6`.

Flow discharge coefficient

Semi-empirical parameter for orifice capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.6`.

Initial opening

Orifice initial opening. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The value of initial opening does not depend on the orifice orientation. The default value is `0`.

Orifice orientation

The parameter is introduced to specify the effect of the orifice control member motion on the valve opening. The parameter can be set to one of two options: `Opens in positive direction` or `Opens in negative direction`. The value `Opens in positive direction` specifies an orifice whose control member opens the valve when it is shifted in the globally assigned positive direction. The parameter is extremely useful for building a multi-orifice valve with all the orifices being controlled by the same spool. The default value is `Opens in positive direction`.

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` m^2.

Restricted Parameters

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

- **Orifice orientation**

All other block parameters are available for modification.

Global Parameters

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

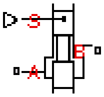
Purpose

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:

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

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

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

$$p = p_A - p_B$$

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

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

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

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
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

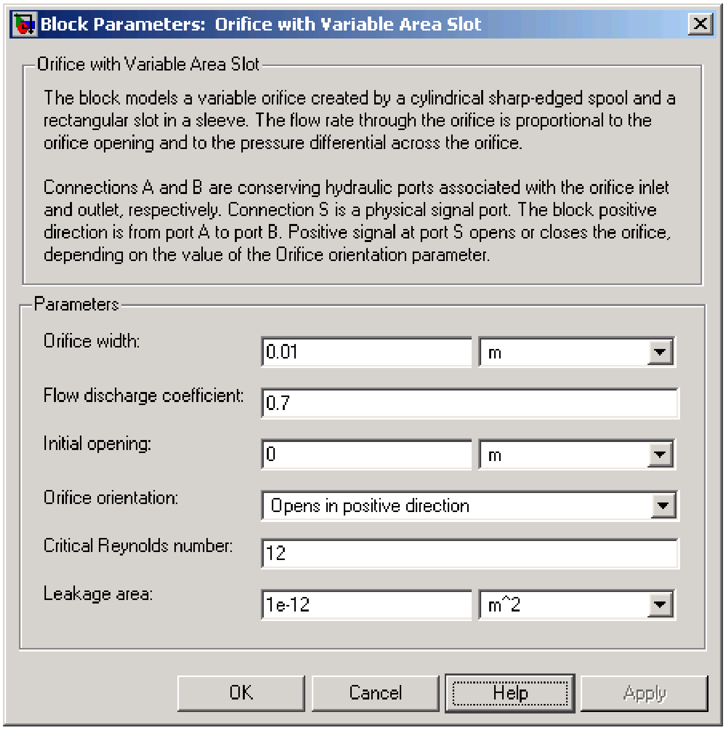
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as $p = p_A - p_B$. Positive signal at the physical signal port `S` opens or closes the orifice depending on the value of the parameter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

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 m^2

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` m^2.

Restricted Parameters

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

- **Orifice orientation**

All other block parameters are available for modification.

Global Parameters

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

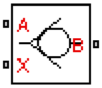
Purpose

Simulate hydraulic check valve that allows flow in one direction, but can be disabled by pilot pressure

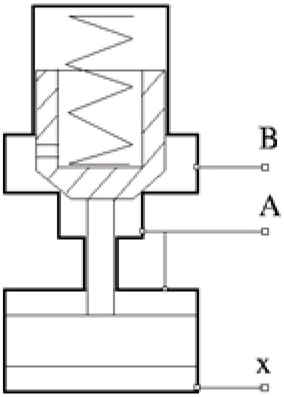
Library

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

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

where

$p_{A,} p_B$	Gauge pressures at the valve terminals
p_X	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

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

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 value 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:

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

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

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

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

$$p = p_A - p_B$$

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

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

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

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
$A(p)$	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

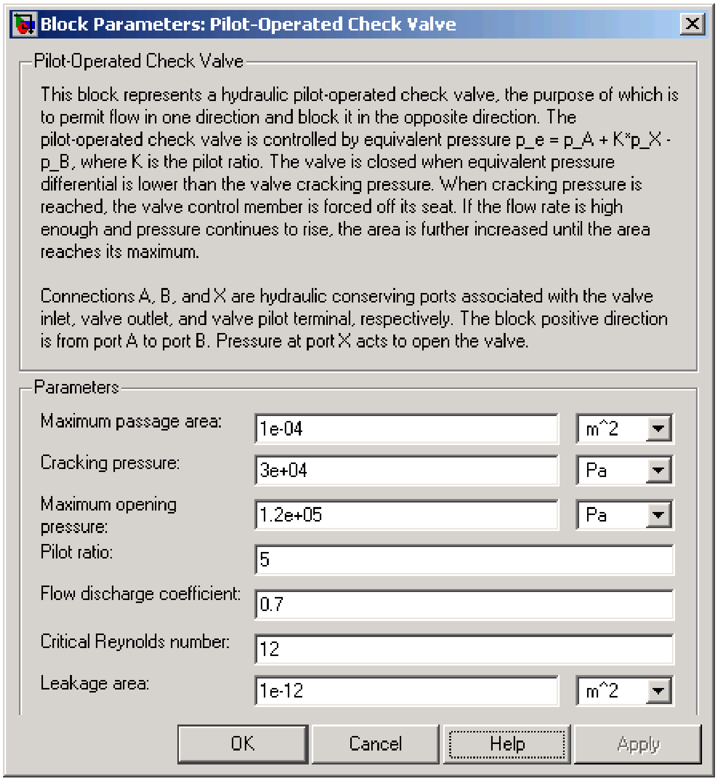
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as $p = p_A - p_B$.

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 $Re=Re_{cr}$.

Dialog Box and Parameters



This block represents a hydraulic pilot-operated check valve, the purpose of which is to permit flow in one direction and block it in the opposite direction. The pilot-operated check valve is controlled by equivalent pressure p_e = p_A + K*p_X - p_B, where K is the pilot ratio. The valve is closed when equivalent pressure differential is lower than the valve cracking pressure. When cracking pressure is reached, the valve control member is forced off its seat. If the flow rate is high enough and pressure continues to rise, the area is further increased until the area reaches its maximum.

Connections A, B, and X are hydraulic conserving ports associated with the valve inlet, valve outlet, and valve pilot terminal, respectively. The block positive direction is from port A to port B. Pressure at port X acts to open the valve.

Maximum passage area: 1e-04 m^2

Cracking pressure: 3e+04 Pa

Maximum opening pressure: 1.2e+05 Pa

Pilot ratio: 5

Flow discharge coefficient: 0.7

Critical Reynolds number: 12

Leakage area: 1e-12 m^2

Maximum passage area

Valve passage maximum cross-sectional area. The default value is `1e-4` m^2.

Cracking pressure

Pressure level at which the orifice of the valve starts to open. The default value is `3e4` Pa.

Maximum opening pressure

Pressure differential across the valve needed to fully open the valve. Its value must be higher than the cracking pressure. The default value is `1.2e5` Pa.

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` 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.

X

Hydraulic conserving port associated with the valve pilot terminal.

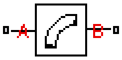
See Also

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

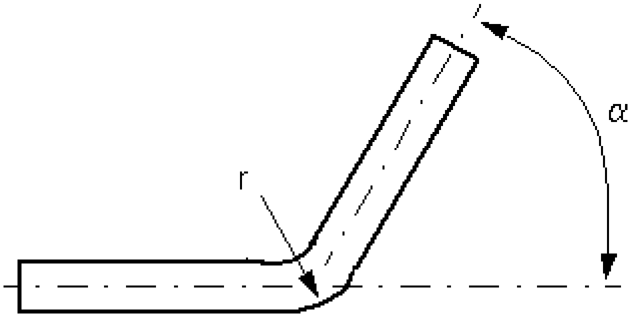
For laminar flow regime, the formula for pressure loss computation is modified, as described in the reference documentation for the Local Resistance block.

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 ($d \le r \le 20d$)
α	Bend angle in degrees ($0 \le \alpha \le 180$)



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

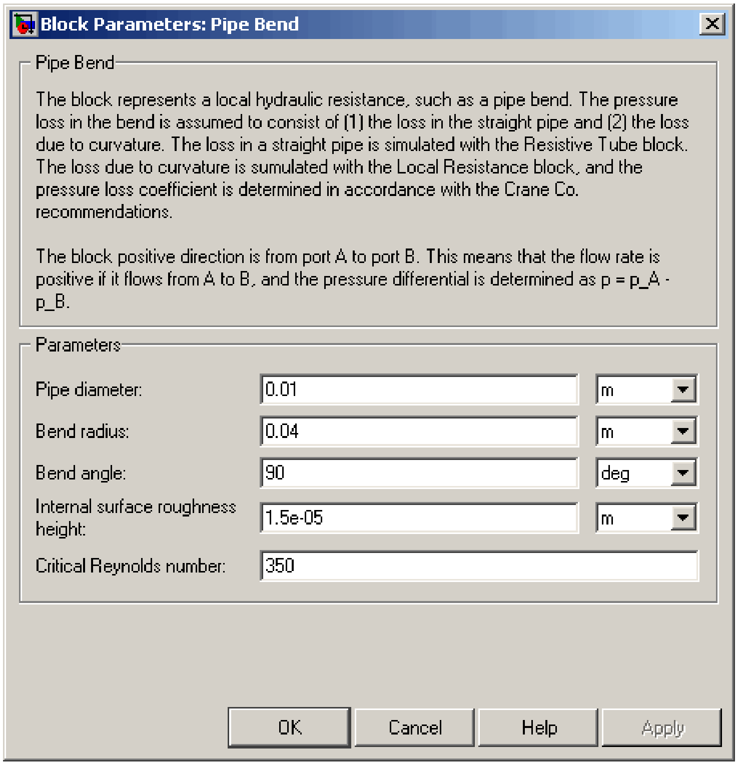
The formulas used in the Pipe Bend block are very approximate, especially in the laminar and transient flow regions. For more accurate results, use a combination of the Local Resistance block with a table-specified `K=f(Re)` relationship and the Resistive Tube block.

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.

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 sumulated 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:	0.01	m
Bend radius:	0.04	m
Bend angle:	90	deg
Internal surface roughness height:	1.5e-05	m
Critical Reynolds number:	350	

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

See Also

Elbow

Gradual Area Change

Local Resistance

Resistive Tube

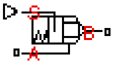
Sudden Area Change

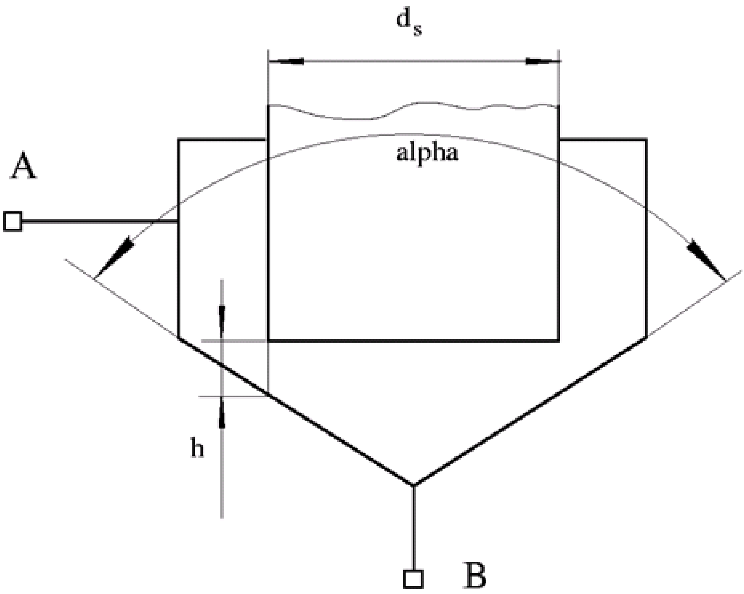
T-junction

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:

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

$$h = x_0 + x$$

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

$$p = p_A - p_B$$

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

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

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

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

where

q	Flow rate
p	Pressure differential
p_A, p_B	Gauge pressures at the block terminals

C_D	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
x_0	Initial opening
x	Stem displacement from initial position
h	Valve opening
h_{max}	Maximum valve opening. The passage area remains constant and equal to A_{max} after this.
d_s	Stem diameter
α	Cone angle
ρ	Fluid density
D_H	Valve instantaneous hydraulic diameter
ν	Fluid kinematic viscosity
A_{leak}	Closed valve leakage area
A_{max}	Maximum valve open area

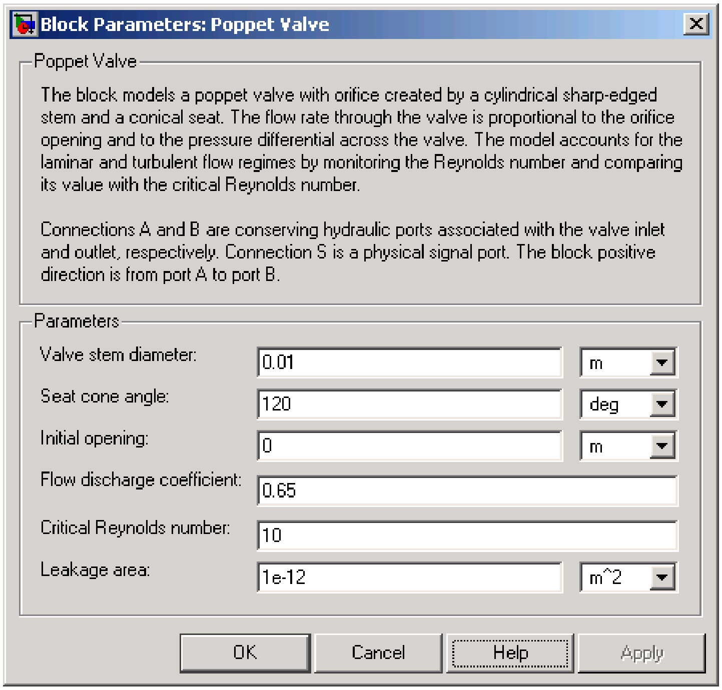
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as $p = p_A - p_B$. Positive signal at the physical signal port `S` opens the valve.

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.

Dialog Box and Parameters



Poppet Valve

The block models a poppet valve with orifice created by a cylindrical sharp-edged stem and a conical seat. 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 stem diameter:	0.01	m
Seat cone angle:	120	deg
Initial opening:	0	m
Flow discharge coefficient:	0.65	
Critical Reynolds number:	10	
Leakage area:	1e-12	m^2

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` 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

Ball Valve

Needle Valve

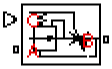
Pressure-Compensated Flow Control Valve

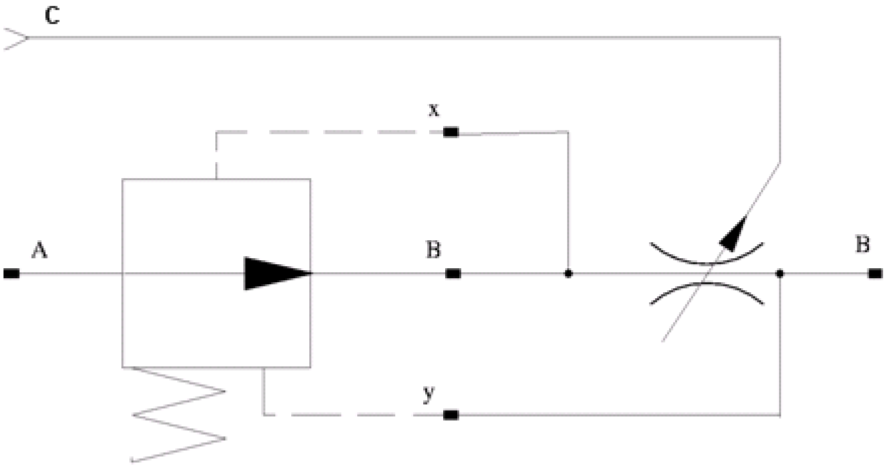
Pressure-Compensated Flow Control Valve

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)`*.

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.

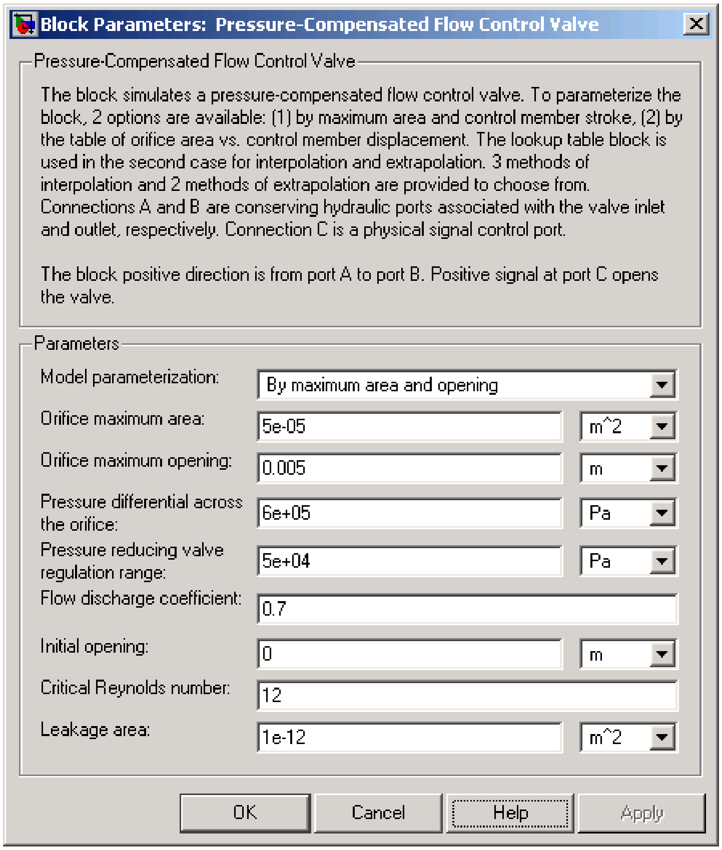
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as $p = p_A - p_B$. Positive signal at port C opens the valve.

Assumptions and Limitations

The model is based on the following assumption:

- Fluid inertia is not taken into account.

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

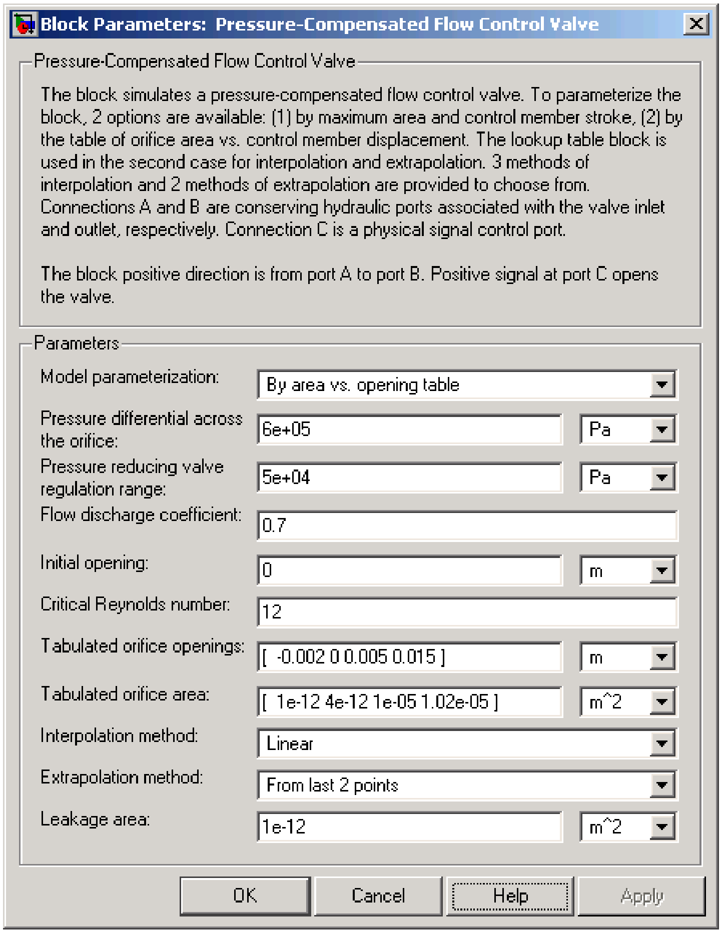
Pressure reducing valve regulation range: 5e+04 Pa

Flow discharge coefficient: 0.7

Initial opening: 0 m

Critical Reynolds number: 12

Leakage area: 1e-12 m^2



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 area vs. opening table
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
Tabulated orifice openings: [-0.002 0 0.005 0.015] m
Tabulated orifice area: [1e-12 4e-12 1e-05 1.02e-05] m^2
Interpolation method: Linear
Extrapolation method: From last 2 points
Leakage area: 1e-12 m^2

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.

Orifice maximum area

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

Orifice maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `5e-4` m. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Tabulated orifice openings

Specify the vector of input values for orifice openings as a 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]`. 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 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 Polinomial (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.

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

Pressure differential across the orifice

Pressure difference that must be maintained across the element by the pressure compensator. The default value is `6e5` Pa.

Pressure reducing valve regulation range

Pressure increase over the preset level needed to fully close the valve. Must be less than 0.2 of the **Pressure differential across the orifice** parameter value. The default value is `5e4` Pa.

Flow discharge coefficient

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

Initial opening

Orifice initial opening. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The value of initial opening does not depend on the orifice orientation. The default value is `0`.

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` 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.

`C`

Physical signal control port.

See Also

- Ball Valve
- Needle Valve
- Poppet Valve

Pressure Compensator

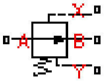
Purpose

Simulate hydraulic pressure compensating valve

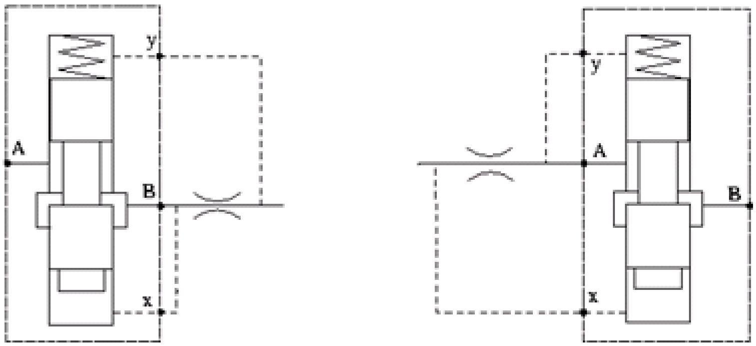
Library

Pressure Control Valves

Description

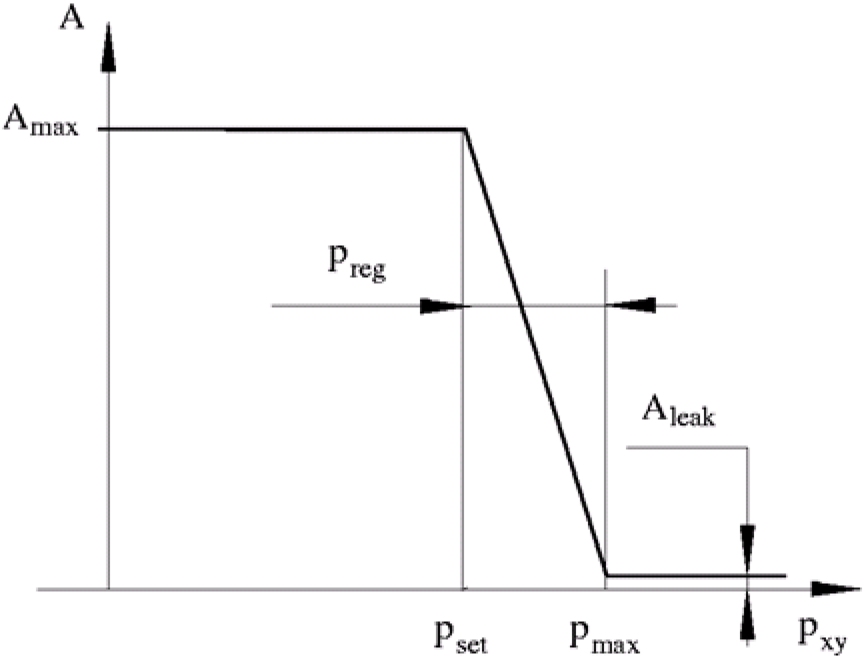


The Pressure Compensator block represents a hydraulic pressure compensating valve, or pressure compensator. Pressure compensators are used to maintain preset pressure differential across a hydraulic component to minimize the influence of pressure variation on a flow rate passing through the component.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



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 if fully closed. The pressure increase that is necessary to close the valve is referred to as regulation range, or pressure compensator static error, and usually is provided in manufacturer's catalog or data sheets.

The main parameters of the block are the valve maximum area and regulation range. In addition, you need to specify the leakage area of the valve. Physically, it represents a possible clearance in the closed valve, but the main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or "hanging" part of the system could affect computational efficiency and even cause failure of computation.

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:

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

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

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

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

$$p = p_A - p_B$$

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

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

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

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

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

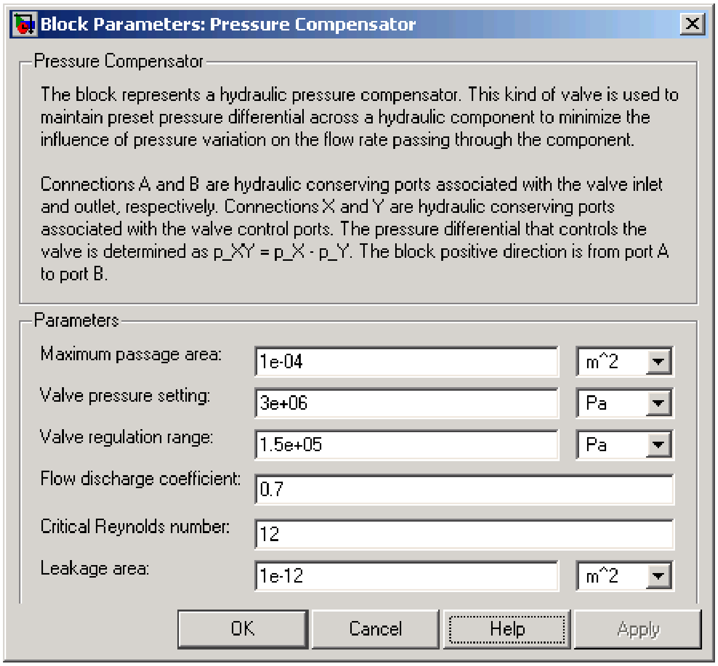
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as $p = p_A - p_B$. The control pressure differential is measured as $p_{xy} = p_x - p_y$, and it creates a force acting against the spring 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



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.

Maximum passage area: 1e-04 m^2

Valve pressure setting: 3e+06 Pa

Valve regulation range: 1.5e+05 Pa

Flow discharge coefficient: 0.7

Critical Reynolds number: 12

Leakage area: 1e-12 m^2

Maximum passage area

Valve passage maximum cross-sectional area. The default value is `1e-4` 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` 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.

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

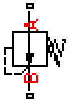
Purpose

Simulate pressure control valve maintaining reduced pressure in portion of system

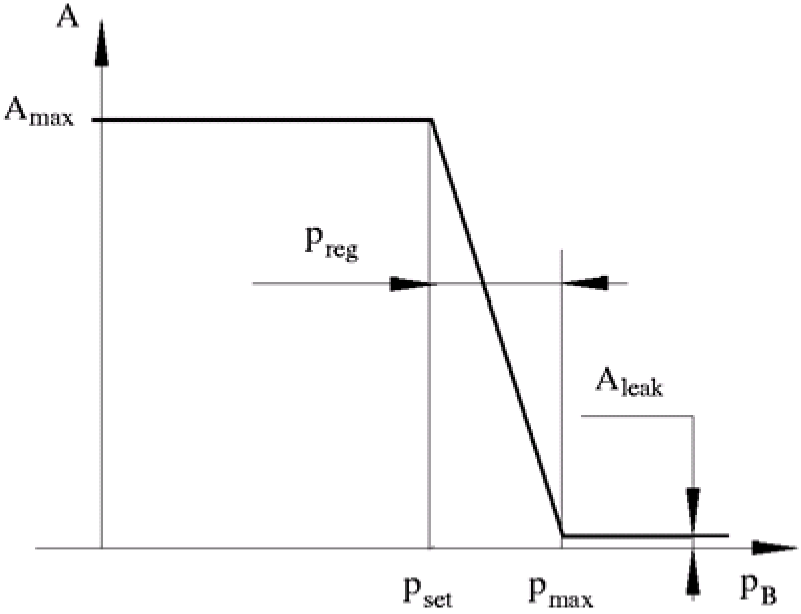
Library

Pressure Control Valves

Description



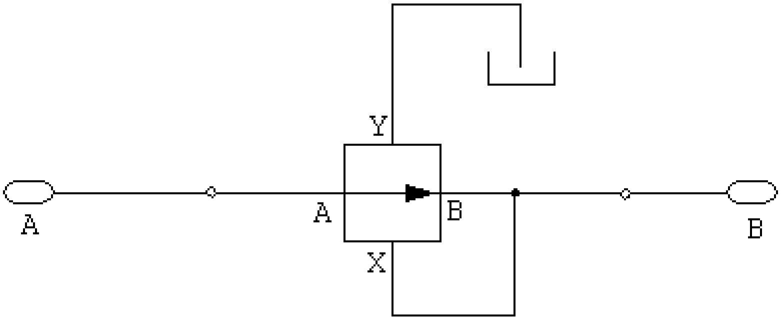
The Pressure Reducing Valve block represents a hydraulic pressure-reducing valve as a data-sheet-based model. Pressure-reducing valves are used to maintain reduced pressure in a portion of a system. The following figure shows the typical dependency between the valve passage area A and the pressure p_B 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 value control member (spool, ball, poppet, etc.) is forced off its stop and starts closing the orifice, thus trying to maintain outlet pressure at preset level. Any further increase in the outlet pressure causes the control member to close the orifice even more until the point when the orifice if 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.

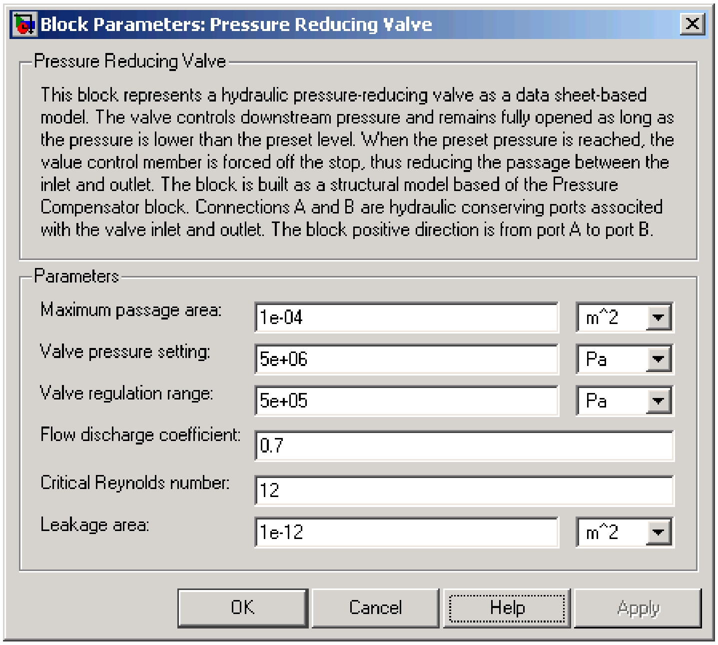
In addition to the maximum area, the leakage area is also required to characterize the valve. The main purpose of the parameter is not to account for possible leakage, even though this is also important, but to maintain numerical integrity of the circuit by preventing a portion of the system from getting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



Pressure Reducing Valve

This block represents a hydraulic pressure-reducing valve as a data sheet-based model. The valve controls downstream pressure and remains fully opened as long as the pressure is lower than the preset level. When the preset pressure is reached, the value control member is forced off the stop, thus reducing the passage between the inlet and outlet. The block is built as a structural model based of the Pressure Compensator block. Connections A and B are hydraulic conserving ports associted with the valve inlet and outlet. The block positive direction is from port A to port B.

Parameters

Maximum passage area: 1e-04 m^2

Valve pressure setting: 5e+06 Pa

Valve regulation range: 5e+05 Pa

Flow discharge coefficient: 0.7

Critical Reynolds number: 12

Leakage area: 1e-12 m^2

Maximum passage area
: Valve passage maximum cross-sectional area. The default value is `1e-4` m^2.

Valve pressure setting
: Preset pressure level, at which the orifice of the valve starts to close. The default value is `5e6` 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` 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-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 20e5 Pa and maintains this pressure downstream, as long as the upstream pressure is higher than this setting.

See Also

Pressure Compensator

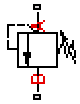
Pressure Relief Valve

Pressure Relief Valve

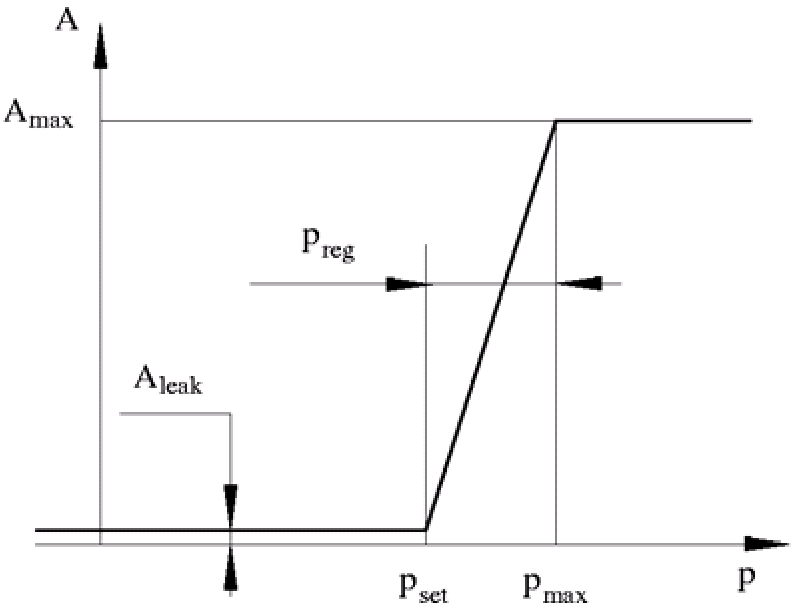
Purpose Simulate pressure control valve maintaining preset pressure in system

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 value 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:

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

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

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

$$p = p_A - p_B$$

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

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

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

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

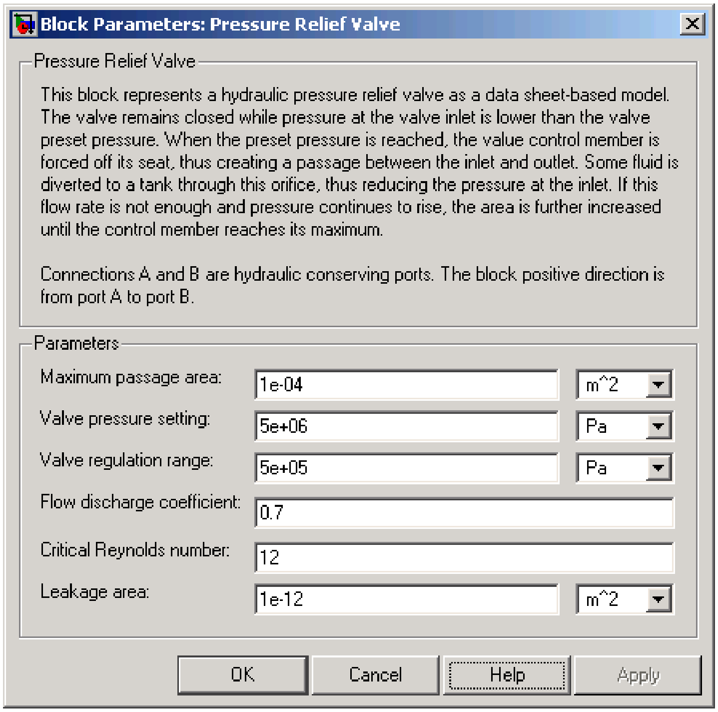
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as $p = p_A - p_B$.

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 value 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:	1e-04	m^2
Valve pressure setting:	5e+06	Pa
Valve regulation range:	5e+05	Pa
Flow discharge coefficient:	0.7	
Critical Reynolds number:	12	
Leakage area:	1e-12	m^2

Maximum passage area

Valve passage maximum cross-sectional area. The default value is `1e-4` m^2.

Valve pressure setting

Preset pressure level, at which the orifice of the valve starts to open. The default value is `50e5` 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` 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-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 Fixed-Displacement Pump demo (`sh_power_unit_fxd_dspl_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

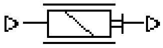
Purpose

Simulate continuous valve driver with output proportional to input signal

Library

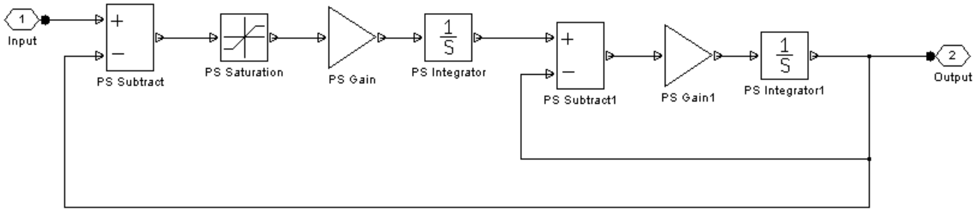
Valve Actuators

Description



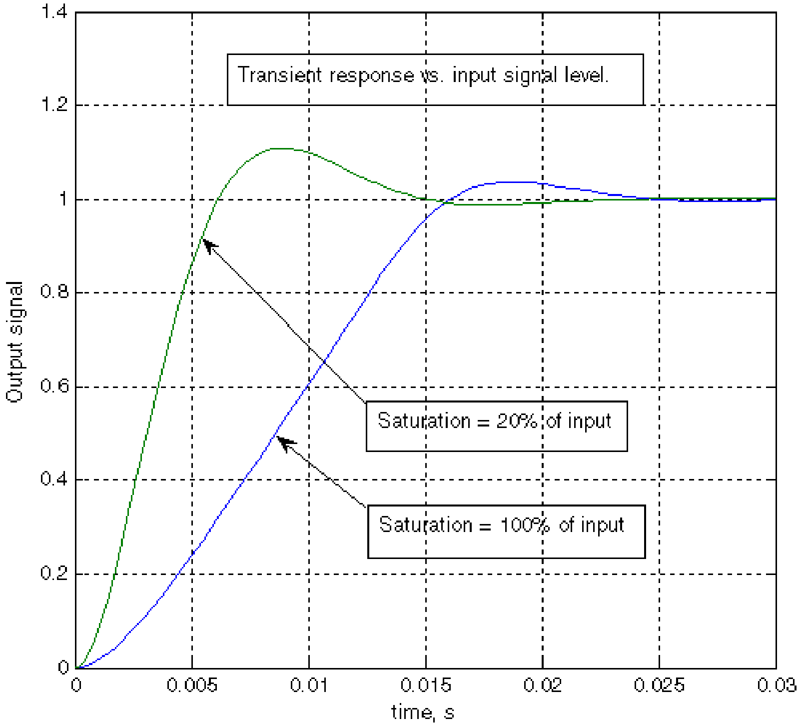
The Proportional and Servo-Valve Actuator block represents an electromagnetic actuator that is used in proportional and servo-valves to drive a spool or other working member. The block is intended to work with one of the directional valve models to form a desirable configuration of a proportional or 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.



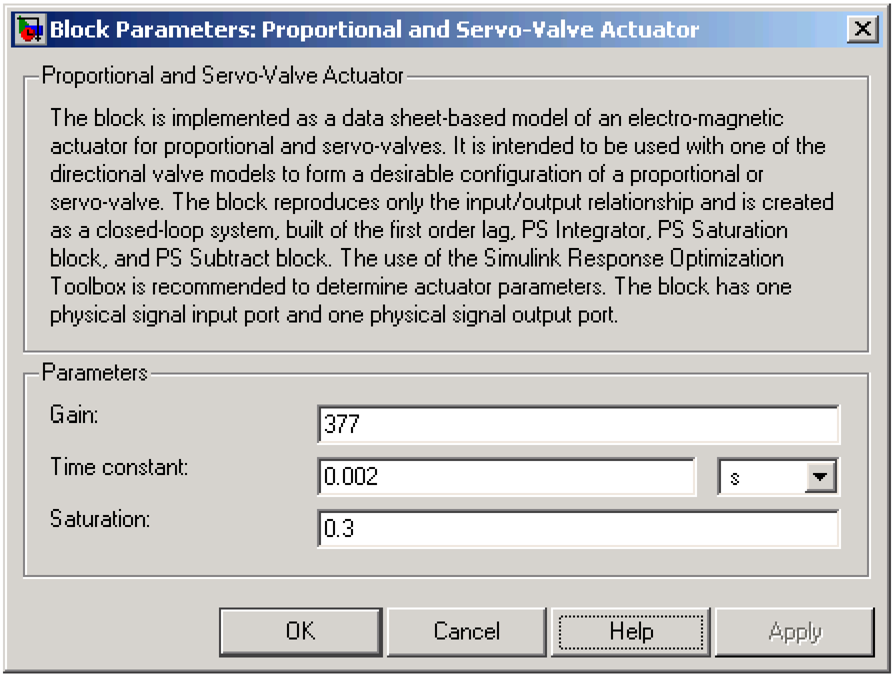
The model consists of the first-order lag, PS Integrator, PS Saturation block, and the PS Subtract block that closes the feedback. The configuration is found to be suitable to simulate behavior of servo-valves and high-quality proportional valves.

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 considerable slower than that with the 20% saturation.



You can adjust the block parameters, such as saturation, gain, and time constant, to make the transient responses close enough to those provided in the data sheet. The most effective way to adjust the parameters is to use the Optimization Toolbox™ software.

Dialog Box and Parameters



Proportional and Servo-Valve Actuator

The block is implemented as a data sheet-based model of an electro-magnetic actuator for proportional and servo-valves. It is intended to be used with one of the directional valve models to form a desirable configuration of a proportional or servo-valve. The block reproduces only the input/output relationship and is created as a closed-loop system, built of the first order lag, PS Integrator, PS Saturation block, and PS Subtract block. The use of the Simulink Response Optimization Toolbox is recommended to determine actuator parameters. The block has one physical signal input port and one physical signal output port.

Parameters

Gain: 377

Time constant: 0.002 s

Saturation: 0.3

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.

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

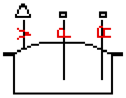
Hydraulic Single-Acting Valve Actuator

Reservoir

Purpose Simulate pressurized hydraulic reservoir

Library Hydraulic Utilities

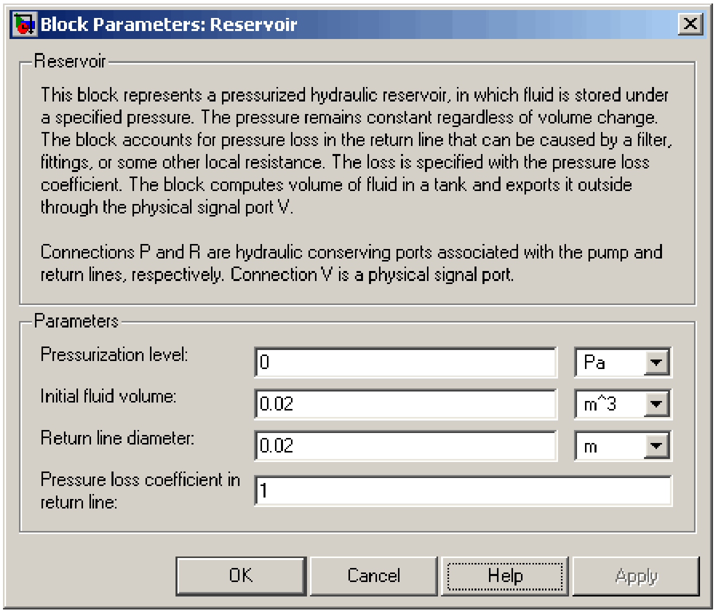
Description



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



Reservoir

This 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 volume of fluid in a tank and exports it outside through the physical signal port V.

Connections P and R are hydraulic conserving ports associated with the pump and return lines, respectively. Connection V is a physical signal port.

Parameters

Pressurization level: 0 Pa

Initial fluid volume: 0.02 m^3

Return line diameter: 0.02 m

Pressure loss coefficient in return line: 1

Pressurization level

The pressure inside the reservoir. The default value is 0.

Initial fluid volume

The initial volume of fluid in the tank. The default value is `0.02` m^3.

Return line diameter

The diameter of the return line. The default value is `0.02` m.

Pressure loss coefficient in return line

The value of the pressure loss coefficient, to account for pressure loss in the return line. The default value is `1`.

Ports

The block has the following ports:

`P`

Hydraulic conserving port associated with the pump line.

`R`

Hydraulic conserving port associated with the return line.

`V`

Physical signal port that output the volume of fluid in the tank.

See Also

Hydraulic Reference

Purpose

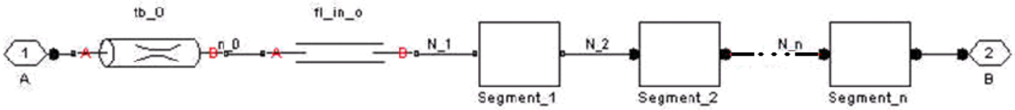
Simulate hydraulic pipeline with resistive, fluid inertia, and fluid compressibility properties

Library

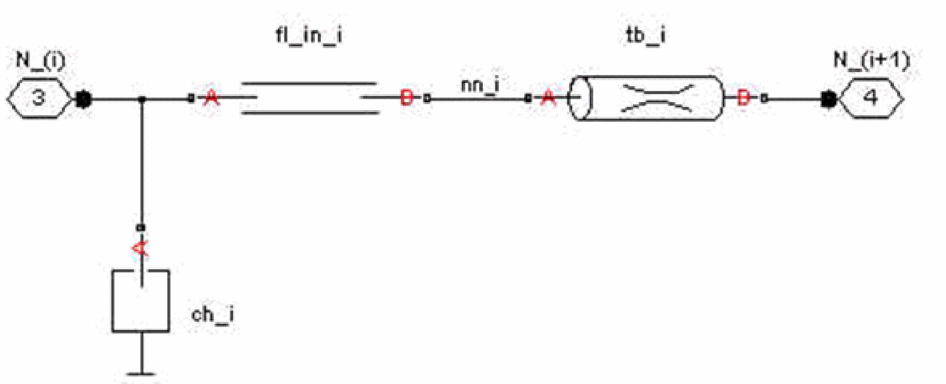
Pipelines

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

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`, ..., `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`, ..., `nn_n`. The Constant Chamber blocks are named `ch_1`, `ch_2`, ..., `ch_n`, Resistive Tube blocks are named `tb_0`, `tb_1`, `tb_2`, ..., `tb_n`, and Fluid Inertia blocks are named `fl_in_0`, `fl_in_1`, `fl_in_2`, ..., `fl_in_n`.

The number of segments 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.

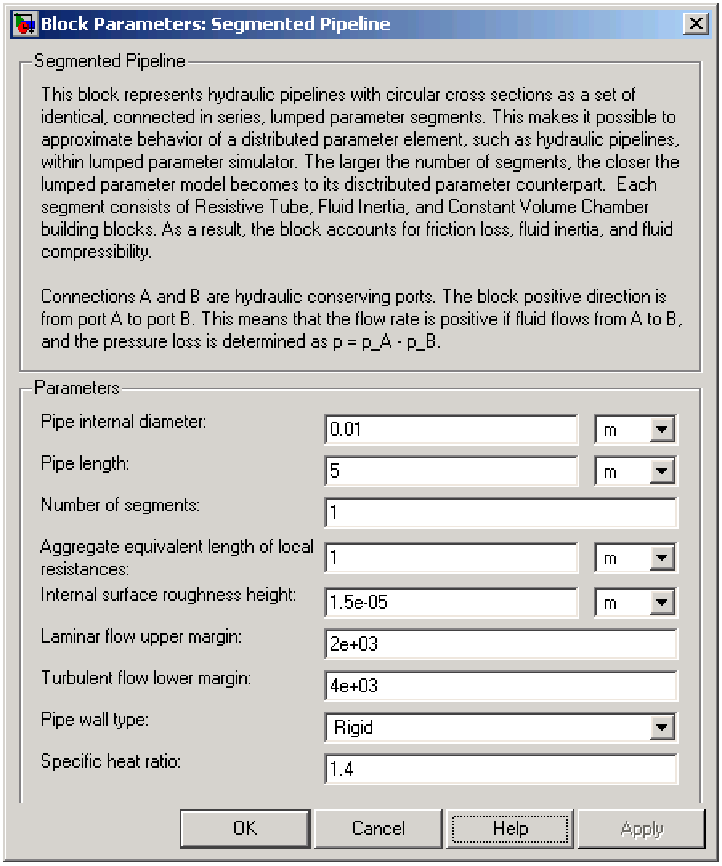
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

Basic Assumptions and Limitations

The model is based on the following assumption:

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

Dialog Box and Parameters



This block represents hydraulic pipelines with circular cross sections as a set of identical, connected in series, lumped parameter segments. This makes it possible to approximate behavior of a distributed parameter element, such as hydraulic pipelines, within lumped parameter simulator. The larger the number of segments, the closer the lumped parameter model becomes to its disctributed parameter counterpart. Each segment consists of Resistive Tube, Fluid Inertia, and Constant Volume Chamber building blocks. As a result, the block accounts for friction loss, fluid inertia, and fluid compressibility.

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.

Pipe internal diameter: 0.01 m

Pipe length: 5 m

Number of segments: 1

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: Rigid

Specific heat ratio: 1.4

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. This length is added to the geometrical pipe length only for hydraulic resistance computation. Both the fluid volume and fluid inertia are determined based on pipe geometrical length only. The default value is `1` m.

Internal surface roughness height

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

Laminar flow upper margin

Specifies the Reynolds number at which the laminar flow regime is assumed to start converting into turbulent. Mathematically, this is the maximum Reynolds number at fully developed laminar flow. The default value is `2000`.

Turbulent flow lower margin

Specifies the Reynolds number at which the turbulent flow regime is assumed to be fully developed. Mathematically, this is the minimum Reynolds number at turbulent flow. The default value is `4000`.

Pipe wall type

The parameter can have one of two values: `Rigid` or `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`.

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

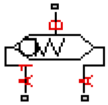
Resistive Tube

Shuttle Valve

Purpose Simulate hydraulic valve that allows flow in one direction only

Library Directional Valves

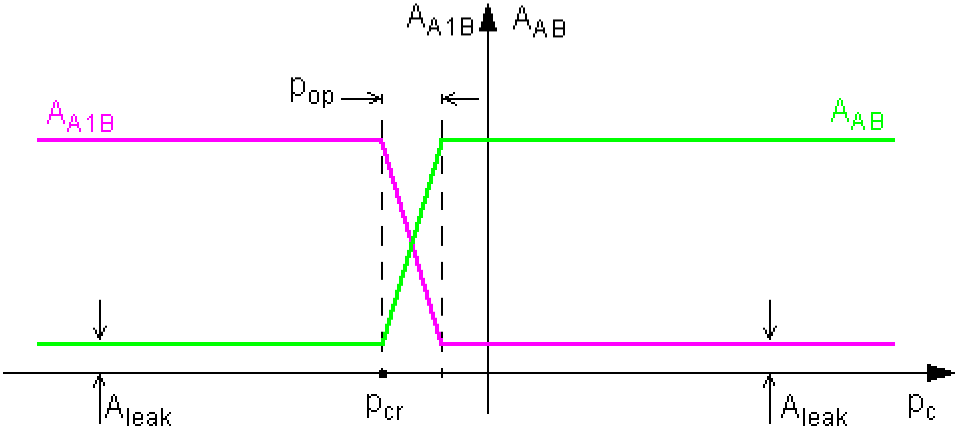
Description



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 value 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:

$$q_{AB} = \begin{cases} C_D \bullet A_{AB} \sqrt{\frac{2}{\rho} | \, p_{AB} \, |} \bullet sign(p_{AB}) & \text{for } Re_{AB} >= Re_{\text{cr}} \\ 2C_{DL} \bullet A_{AB} \dfrac{D_{HAB}}{\nu \bullet \rho} p_{AB} & \text{for } Re_{AB} < Re_{\text{cr}} \end{cases}$$

$$q_{A1B} = \begin{cases} C_D \bullet A_{A1B} \sqrt{\frac{2}{\rho} \left| p_{A1B} \right|} \bullet sign(p_{A1B}) & \text{for } Re_{A1B} >= Re_{\text{cr}} \\ 2C_{DL} \bullet A_{A1B} \frac{D_{HA1B}}{\nu \bullet \rho} p_{A1B} & \text{for } Re_{A1B} < Re_{\text{cr}} \end{cases}$$

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

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

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

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

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

$$\mathrm{Re}_{AB} = \frac{q_{AB} \bullet D_{HAB}}{A_{AB} \bullet \nu}$$

$$\mathrm{Re}_{A1B} = \frac{q_{A1B} \bullet D_{HA1B}}{A_{A1B} \bullet \nu}$$

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

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

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

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 $p_c = p_A - p_{A1}$.

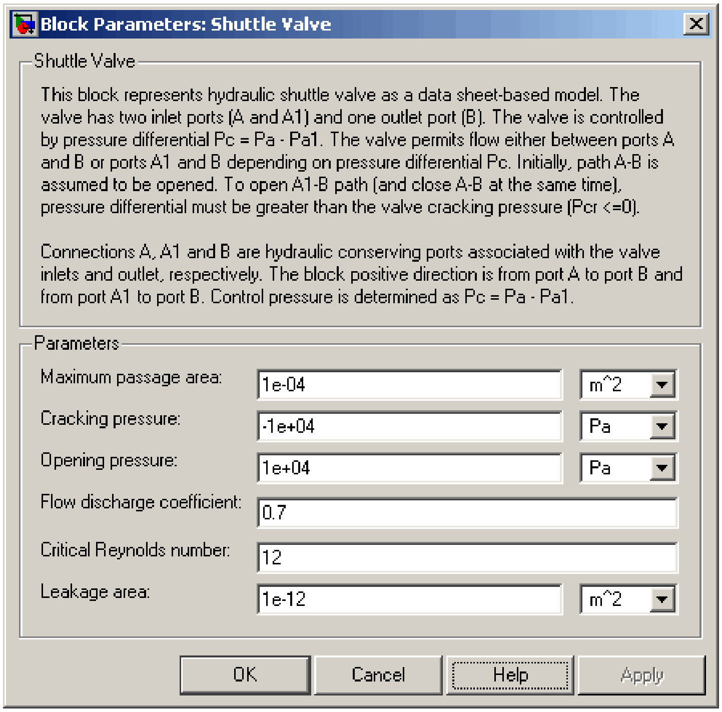
Basic Assumptions and Limitations

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.

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

Dialog Box and Parameters



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 Pc = Pa - Pa1. The valve permits flow either between ports A and B or ports A1 and B depending on pressure differential Pc. 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 (Pcr <=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 Pc = Pa - Pa1.

Parameters

Maximum passage area:	1e-04	m^2
Cracking pressure:	-1e+04	Pa
Opening pressure:	1e+04	Pa
Flow discharge coefficient:	0.7	
Critical Reynolds number:	12	
Leakage area:	1e-12	m^2

Maximum passage area

Valve passage maximum cross-sectional area. The default value is `1e-4` m^2.

Cracking pressure

Pressure differential level at which the orifice of the valve starts to open. The default value is `-1e4` Pa.

Opening pressure

Pressure differential across the valve needed to shift the valve from one extreme position to another. The default value is `1e4` Pa.

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

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` 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.

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

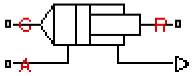
Purpose

Simulate hydraulic actuator exerting force in one direction

Library

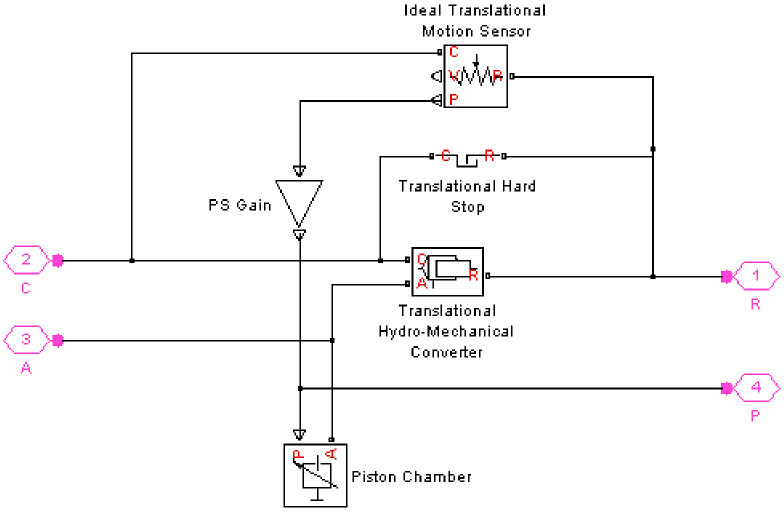
Hydraulic Cylinders

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



Connections R and C are mechanical translational conserving ports corresponding to the cylinder rod and cylinder clamping structure, respectively. Connection A is a hydraulic conserving port associated with the cylinder inlet. The physical signal output port provides rod displacement.

The energy through port A is directed to the Translational Hydro-Mechanical Converter block and the Piston 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 Piston Chamber block.

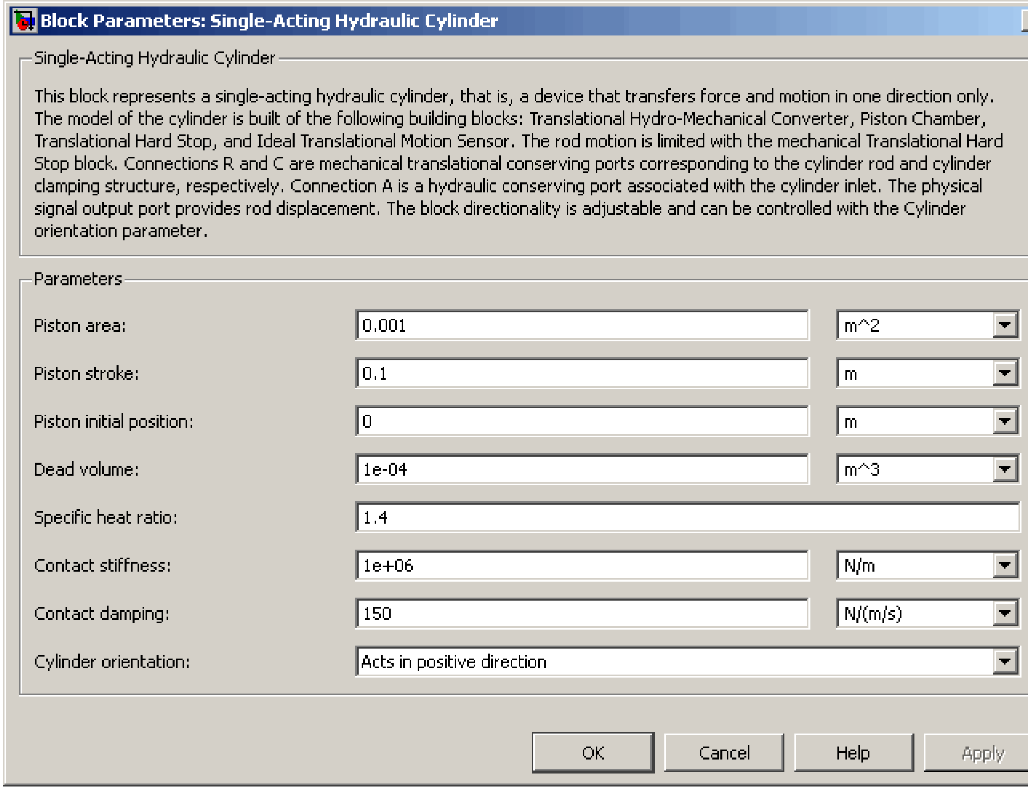
The block directionality is adjustable and can be controlled with the **Cylinder orientation** parameter.

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.

Dialog Box and Parameters



Single-Acting Hydraulic Cylinder

This block represents a single-acting hydraulic cylinder, that is, a device that transfers force and motion in one direction only. The model of the cylinder is built of the following building blocks: Translational Hydro-Mechanical Converter, Piston 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. Connection A is a hydraulic conserving port associated with the cylinder inlet. The physical signal output port provides rod displacement. The block directionality is adjustable and can be controlled with the Cylinder orientation parameter.

Parameters

Piston area:	0.001	m^2
Piston stroke:	0.1	m
Piston initial position:	0	m
Dead volume:	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	

Piston area

Effective piston area. The default value is `0.001` m^2.

Piston stroke

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

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

Fluid volume that remains in the chamber after the rod is fully retracted. The default value is `1e-4` m^3.

Specific heat ratio

Gas-specific heat ratio for the Piston Chamber block. The default value is `1.4`.

Contact stiffness

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

Contact damping

Specifies dissipating property of colliding bodies for the Translational Hard Stop block. At zero damping, the impact is close to an absolutely elastic one. The greater the value of the parameter, the more energy dissipates during an interaction. Keep in mind that damping affects slider motion as long as the slider is in contact with the stop, including the period when slider is pulled back from the contact. For computational efficiency and convergence reasons, 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

Piston Chamber

Single-Acting Rotary Actuator

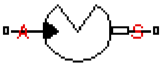
Purpose

Simulate single-acting hydraulic rotary actuator

Library

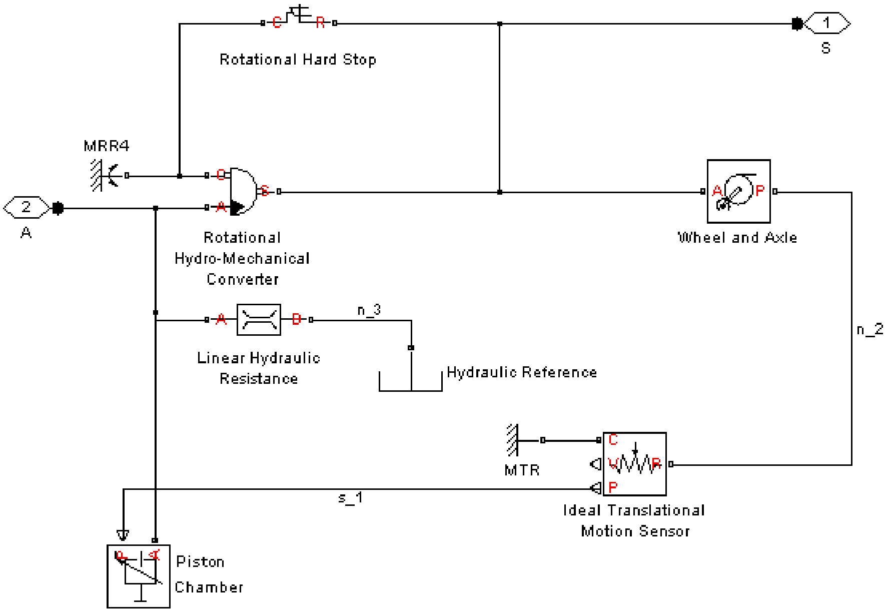
Hydraulic Cylinders

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.



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.
Piston Chamber	Accounts for fluid compressibility.
Ideal Translational Motion Sensor	Determines an instantaneous shaft position, which is necessary for the Piston 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.

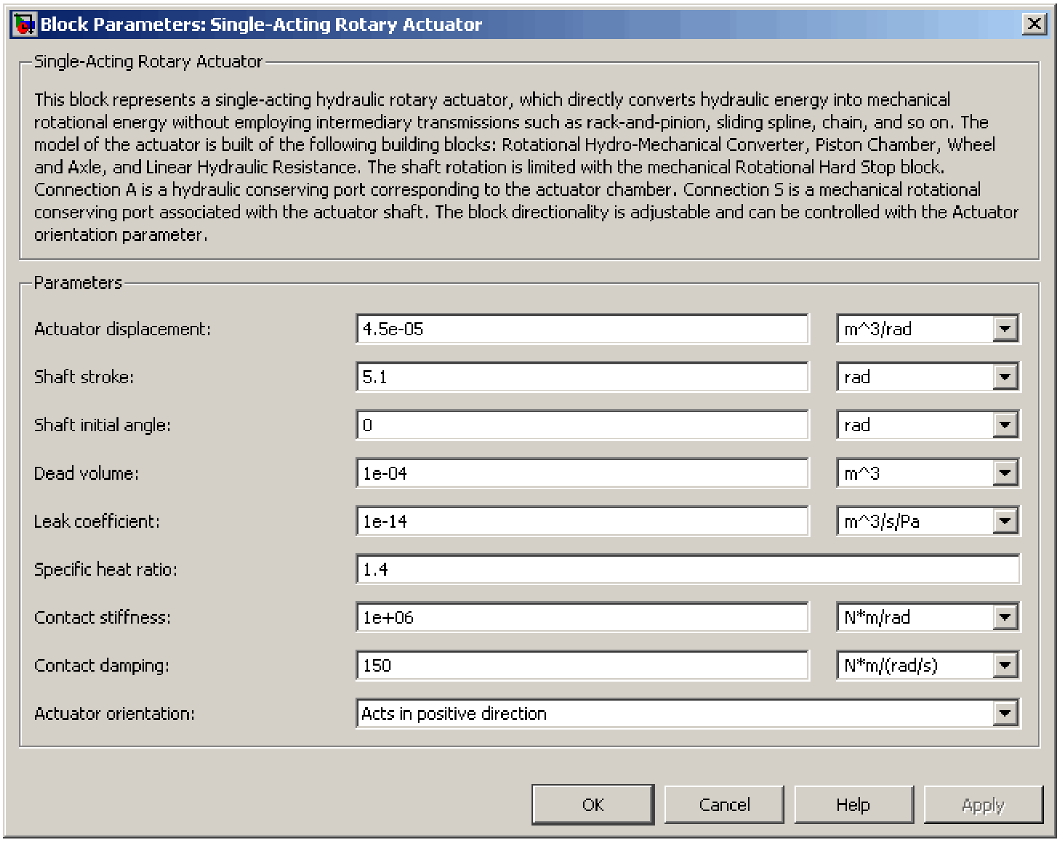
The block directionality is adjustable and can be controlled with the **Actuator orientation** parameter.

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.

Dialog Box and Parameters



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, Piston 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

Actuator displacement

Effective displacement of the actuator. The default value is `4.5e-5` m^3/rad.

Shaft stroke

Shaft maximum travel between stops. The default value is `5.1` rad.

Shaft initial angle

The position of the shaft at the beginning of simulation. You can set the shaft position to any angle within its stroke. The default value is `0`, which corresponds to the shaft position at the very beginning of the stroke.

Dead volume

Fluid volume that remains in the chamber when the shaft is positioned at the very beginning of the stroke. The default value is `1e-4` m^3.

Leak coefficient

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

Specific heat ratio

Gas-specific heat ratio for the Piston Chamber block. The default value is `1.4`.

Contact stiffness

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

Contact damping

Specifies dissipating property of colliding bodies for the Rotational Hard Stop block. At zero damping, the impact is close to an absolutely elastic one. The greater the value of the parameter, the more energy dissipates during an interaction. Keep in mind that damping affects slider motion as long as the slider is in contact with the stop, including the period when slider is pulled back from the contact. For computational efficiency and convergence reasons, 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:

`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

Piston Chamber

Wheel and Axle

Spool Orifice Hydraulic Force

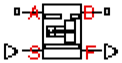
Purpose

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:

$$F = p\frac{q^2}{A}\cos\theta \bullet or$$

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

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

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

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.

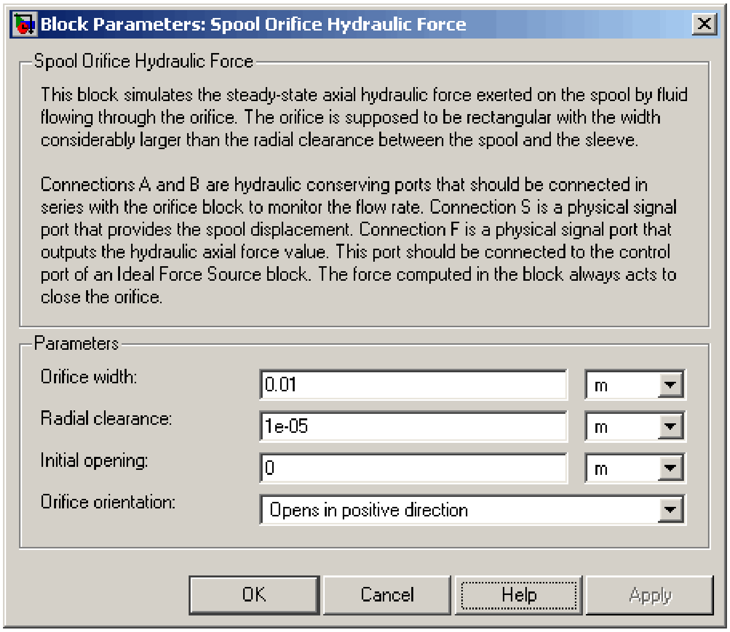
Connections A and B are hydraulic conserving ports that should be connected in series with the orifice block to monitor the flow rate. Connection S is a physical signal port that provides the spool displacement. Connection F is a physical signal port that outputs the hydraulic axial force value. This 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.

Dialog Box and Parameters



Spool Orifice Hydraulic Force

This 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.

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.

Parameters

Orifice width: 0.01 m

Radial clearance: 1e-05 m

Initial opening: 0 m

Orifice orientation: Opens in positive direction

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

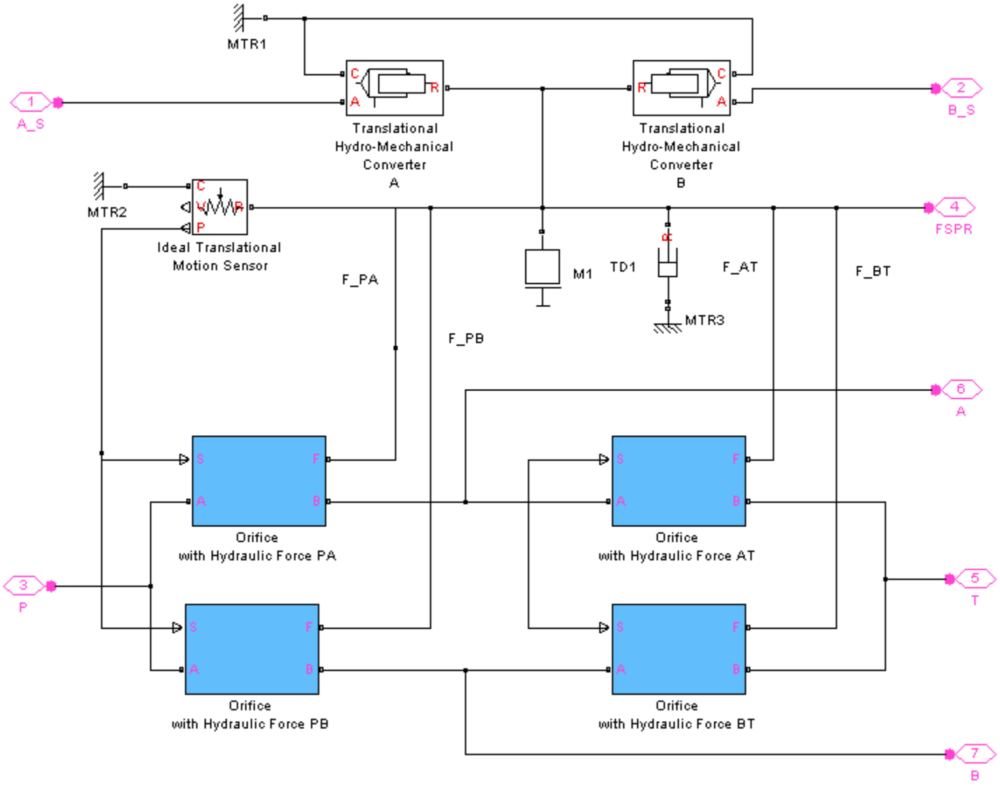
Physical signal port that provides the spool displacement.

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.



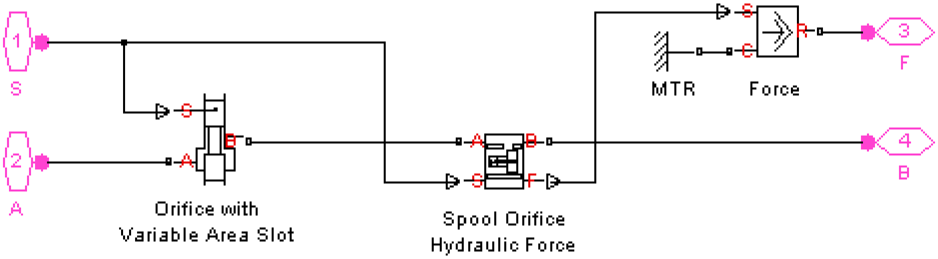
The spool (mass `M1`, viscous friction `TD1`) is shifted by the servo-actuator simulated by two Translational Hydro-Mechanical Converter blocks. Connections A_S and B_S are hydraulic ports for applying pilot control pressure.

Four variable orifices are represented by subsystems:

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

- Orifice with Hydraulic Force BT

The structure of a subsystem is shown in the following illustration.



It consists of an Orifice with Variable Area Slot block, which simulates hydraulic properties of the orifice, connected in series with a Spool Orifice Hydraulic Force block. The force value computed in the block is exported through its port F and passed to the Force block.

The forces on all four orifices (`F_PA`, `F_PB`, `F_AT`, `F_BT`) are applied to the valve spool as it is shown in the first schematic.

For more details and for parameter settings, see the Hydraulic System with Servo-Valve demo (`sh_hydraulic_system_with_servo_valve`).

See Also

Valve Hydraulic Force

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:

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

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

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

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

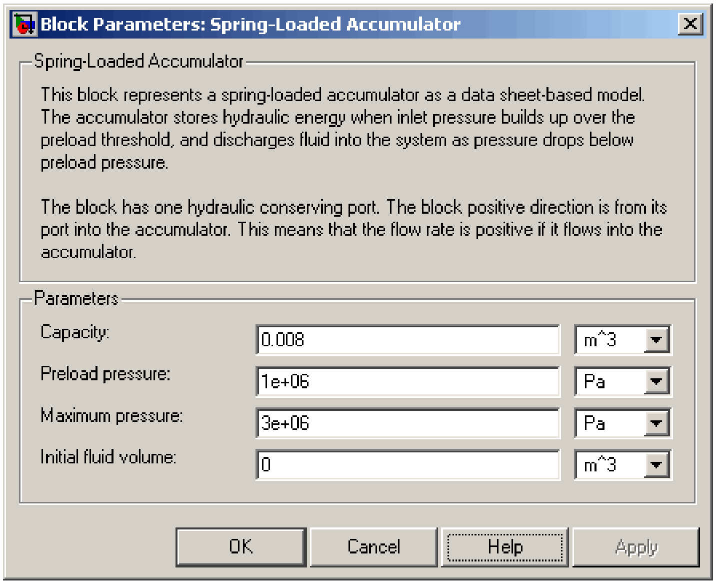
The block positive direction is from port A into the accumulator. This means that the flow rate is positive if it flows into the accumulator.

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



This block represents a spring-loaded accumulator as a data sheet-based model. The accumulator stores hydraulic energy when inlet pressure builds up over the preload threshold, and discharges fluid into the system as pressure drops below preload pressure.

The block has one hydraulic conserving port. The block positive direction is from its port into the accumulator. This means that the flow rate is positive if it flows into the accumulator.

Capacity: 0.008 m^3

Preload pressure: 1e+06 Pa

Maximum pressure: 3e+06 Pa

Initial fluid volume: 0 m^3

Capacity

Accumulator volumetric capacity. The default value is `0.008` m^3.

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.

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

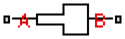
Gas-Charged Accumulator

Sudden Area Change

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

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.

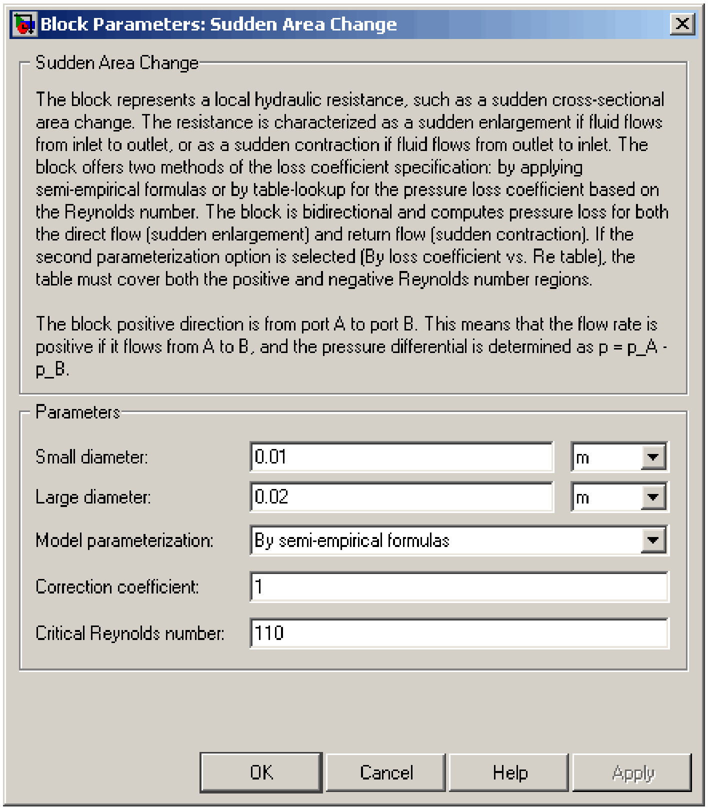
The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

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



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

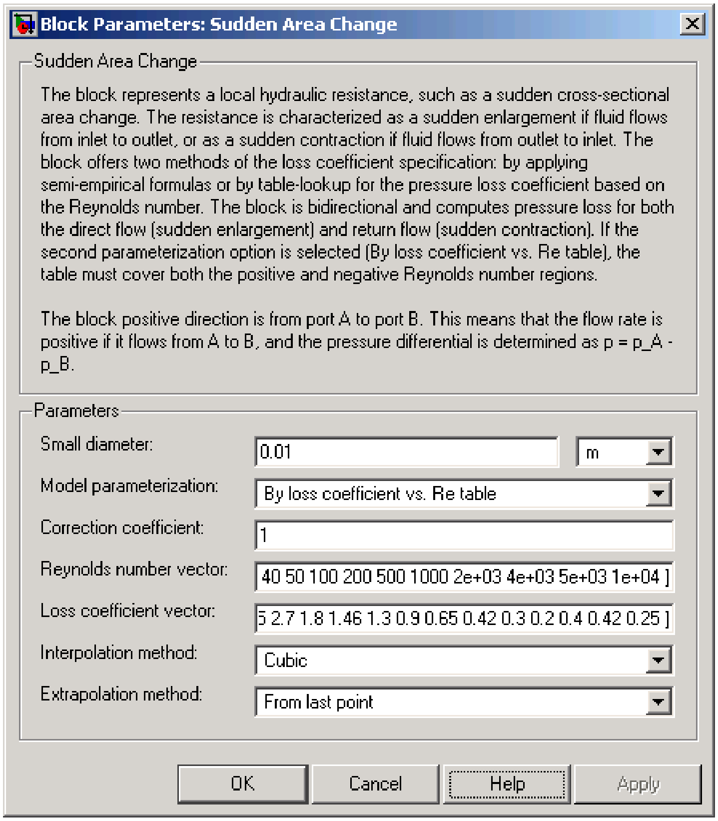
Small diameter: 0.01 m

Large diameter: 0.02 m

Model parameterization: By semi-empirical formulas

Correction coefficient: 1

Critical Reynolds number: 110



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: 0.01 m

Model parameterization: By loss coefficient vs. Re table

Correction coefficient: 1

Reynolds number vector: 40 50 100 200 500 1000 2e+03 4e+03 5e+03 1e+04]

Loss coefficient vector: 5 2.7 1.8 1.46 1.3 0.9 0.65 0.42 0.3 0.2 0.4 0.42 0.25]

Interpolation method: Cubic

Extrapolation method: From last point

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 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. 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] Idelchik, I.E., *Handbook of Hydraulic Resistance*, CRC Begell House, 1994

See Also

Elbow

Gradual Area Change

Local Resistance

Pipe Bend

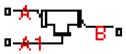
T-junction

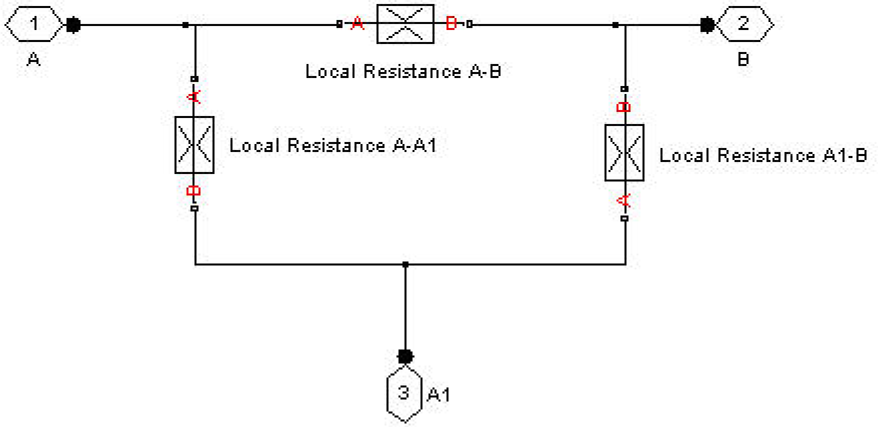
T-junction

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.

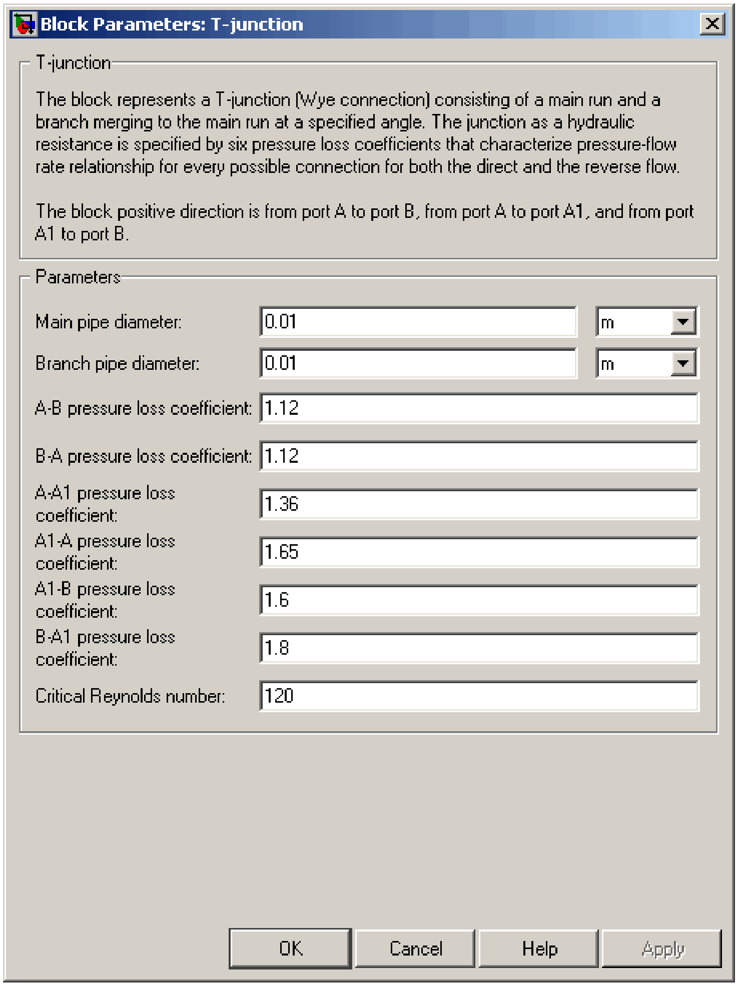
The block positive direction is from port A to port B, from port A to port A1, and from port A1 to port B.

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



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:	0.01	m
Branch pipe diameter:	0.01	m
A-B pressure loss coefficient:	1.12	
B-A pressure loss coefficient:	1.12	
A-A1 pressure loss coefficient:	1.36	
A1-A pressure loss coefficient:	1.65	
A1-B pressure loss coefficient:	1.6	
B-A1 pressure loss coefficient:	1.8	
Critical Reynolds number:	120	

Main pipe diameter

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

Branch pipe diameter

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

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

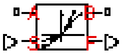
Purpose

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.

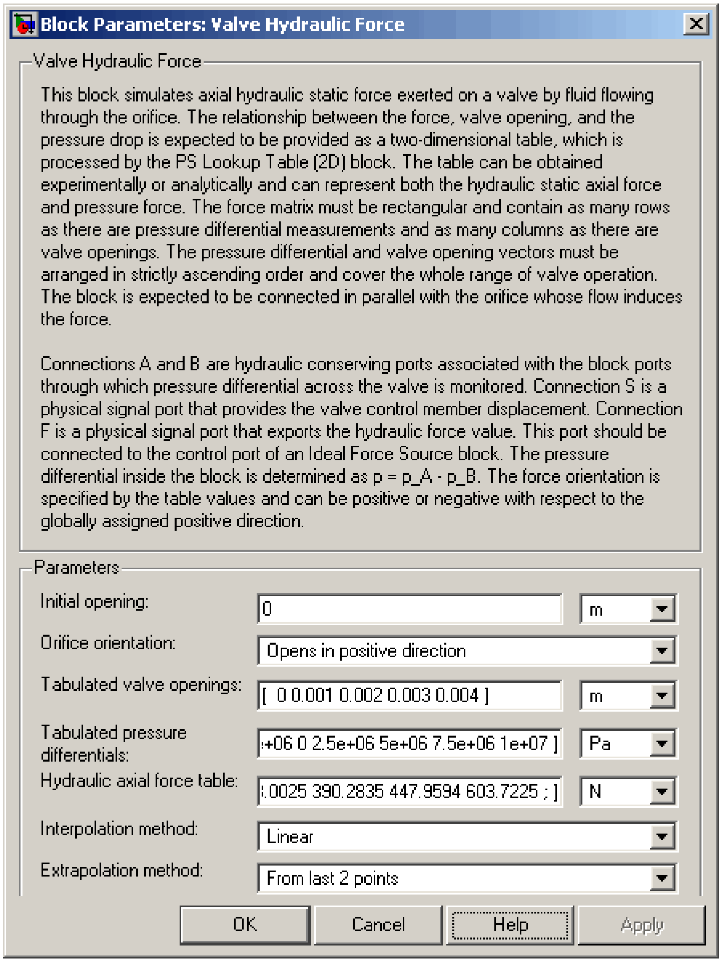
Connections A and B are hydraulic conserving ports that should be connected to the valve block ports in such a way as to monitor the pressure differential across the valve. Connection S is a physical signal port that provides the valve control member displacement. Connection F is a physical signal port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



Block Parameters: Valve Hydraulic Force

Valve Hydraulic Force

This block simulates axial hydraulic static force exerted on a valve by fluid flowing through the orifice. The relationship between the force, valve opening, and the pressure drop is expected to be 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 force. 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 valve opening vectors must be arranged in strictly ascending order and cover the whole range of valve operation. The block is expected to be connected in parallel with the orifice whose flow induces the force.

Connections A and B are hydraulic conserving ports associated with the block ports through which pressure differential across the valve is monitored. Connection S is a physical signal port that provides the valve control member displacement. Connection F is a physical signal port that exports the hydraulic 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.

Parameters

Initial opening: 0 m
Orifice orientation: Opens in positive direction
Tabulated valve openings: [0 0.001 0.002 0.003 0.004] m
Tabulated pressure differentials: +06 0 2.5e+06 5e+06 7.5e+06 1e+07] Pa
Hydraulic axial force table: .0025 390.2835 447.9594 603.7225 ;] N
Interpolation method: Linear
Extrapolation method: From last 2 points

Initial opening

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

Orifice orientation

The parameter is introduced to specify the effect of the valve opening on the valve force. The parameter can be set to one of two options: `Opens in positive direction` or `Opens in negative direction`. The value `Opens in positive direction` specifies an orifice that opens when the valve is shifted in the globally assigned positive direction. The default value is `Opens in positive direction`.

Tabulated valve openings

Specify the vector of input values for valve openings as a 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.

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

Restricted Parameters

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

- **Orifice orientation**
- **Interpolation method**
- **Extrapolation method**

All other block parameters are available for modification.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with a valve port.

B

Hydraulic conserving port associated with another valve port to monitor the pressure differential.

S

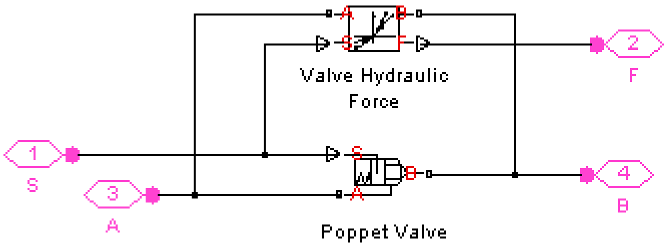
Physical signal port that provides the valve control member displacement.

F

Physical signal port that outputs hydraulic axial force.

Examples

The following example shows a model of a poppet valve built of a Poppet Valve block and a Valve Hydraulic Force block. The Valve Hydraulic Force block is connected in parallel and provides tabulated data to compute hydraulic force acting on the valve. The force value is exported through the F port.



See Also Spool Orifice Hydraulic Force

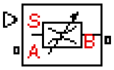
Purpose

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:

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

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

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

$$p = p_A - p_B$$

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

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

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

where

q	Flow rate
p	Pressure differential
p_A, p_B	Gauge pressures at the block terminals
C_D	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area

A_{max}	Orifice maximum area
h_{max}	Control member maximum displacement
x_0	Initial opening
x	Control member displacement from initial position
h	Orifice opening
or	Orifice orientation indicator. The variable assumes +1 value if the control member displacement in the globally assigned positive direction opens the orifice, and –1 if positive motion decreases the opening.
ρ	Fluid density
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 nonuniformly spaced. You have a choice of three interpolation methods and two extrapolation methods.

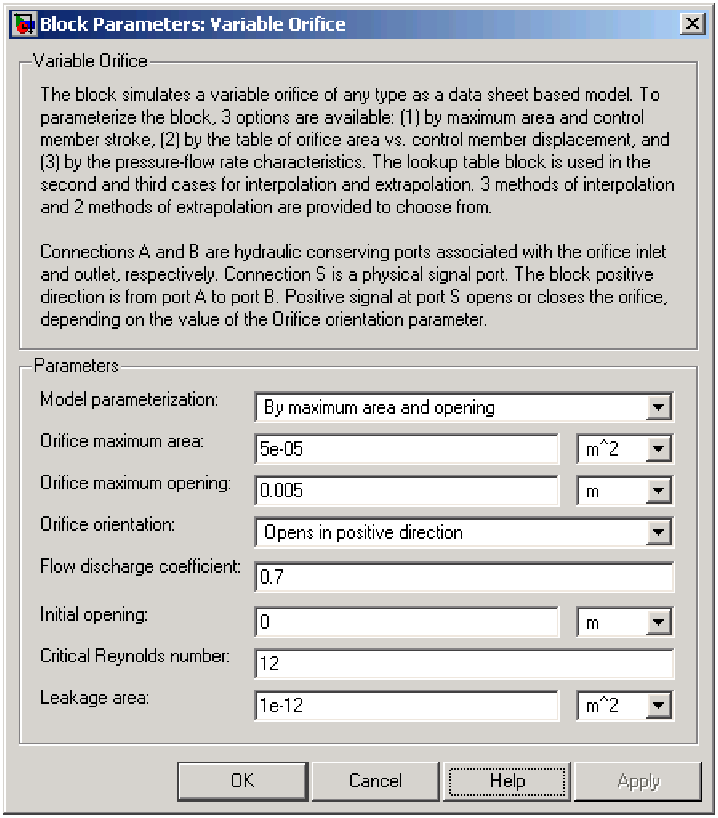
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as $p = p_A - p_B$. Positive signal at the physical signal port `S` opens or closes the orifice depending on the value of the orifice orientation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

Dialog Box and Parameters



Variable Orifice

The block simulates a variable orifice of any type 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 orifice 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 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

Model parameterization: By maximum area and opening

Orifice maximum area: 5e-05 m^2

Orifice maximum opening: 0.005 m

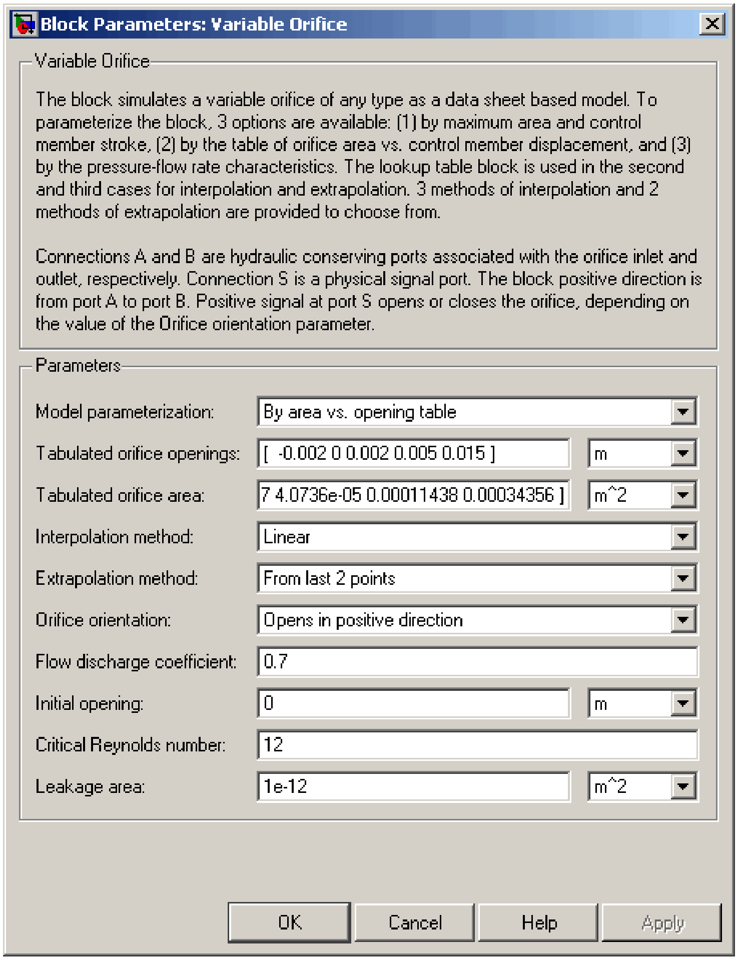
Orifice orientation: Opens in positive direction

Flow discharge coefficient: 0.7

Initial opening: 0 m

Critical Reynolds number: 12

Leakage area: 1e-12 m^2



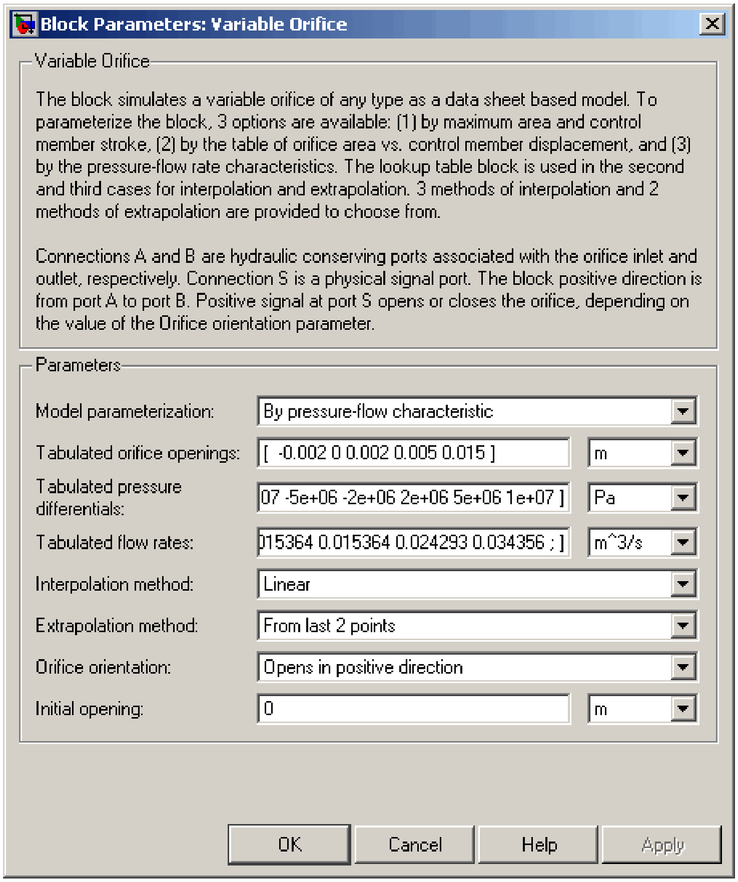
Variable Orifice

The block simulates a variable orifice of any type 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 orifice 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 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

Model parameterization:	By area vs. opening table	
Tabulated orifice openings:	[-0.002 0 0.002 0.005 0.015]	m
Tabulated orifice area:	7 4.0736e-05 0.00011438 0.00034356]	m^2
Interpolation method:	Linear	
Extrapolation method:	From last 2 points	
Orifice orientation:	Opens in positive direction	
Flow discharge coefficient:	0.7	
Initial opening:	0	m
Critical Reynolds number:	12	
Leakage area:	1e-12	m^2



Variable Orifice

The block simulates a variable orifice of any type 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 orifice 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 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

Model parameterization: By pressure-flow characteristic
Tabulated orifice openings: [-0.002 0 0.002 0.005 0.015] m
Tabulated pressure differentials: 07 -5e+06 -2e+06 2e+06 5e+06 1e+07] Pa
Tabulated flow rates: 015364 0.015364 0.024293 0.034356 ;] m^3/s
Interpolation method: Linear
Extrapolation method: From last 2 points
Orifice orientation: Opens in positive direction
Initial opening: 0 m

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` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Orifice maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `5e-4` m. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Tabulated orifice openings

Specify the vector of input values for orifice openings as a 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 `[-0.002 0 0.002 0.005 0.015]`. If **Model parameterization** is set to `By area vs. opening table`, the **Tabulated orifice openings** values will be used together with **Tabulated orifice area** values for one-dimensional table lookup. If **Model parameterization** is set to `By pressure-flow characteristic`, the **Tabulated orifice openings** values will

be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

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-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Model parameterization** is set to `By area vs.  opening table`.

Tabulated pressure differentials

Specify the vector of input values for pressure differentials as a 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 `[-1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]`. This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Tabulated flow rates

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

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

Interpolation method

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

- `Linear` — 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 Polinomial (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.

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` 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:

`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

Variable-Displacement Hydraulic Machine

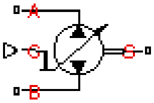
Purpose

Simulate variable-displacement reversible hydraulic machine with regime-dependable efficiency

Library

Pumps and Motors

Description



The Variable-Displacement Hydraulic Machine block represents a variable-displacement hydraulic machine of any type as a data-sheet-based model. The model accounts for the power flow direction and simulates the machine in both the motor and pump mode. The efficiency of the machine is variable, and you can set it in accordance with experimental data provided in the catalog or data sheet.

The machine displacement is controlled by the signal provided through the physical signal port C. The machine efficiency is simulated by implementing regime-dependable leakage and friction torque based on the experimentally established correlations between the machine efficiencies and pressure, angular velocity, and displacement.

With respect to the relationship between the control signal and the displacement, two block parameterization options are available:

- By the maximum displacement and stroke — The displacement is assumed to be linearly dependent on the control member position.
- By table-specified relationship between the control member position and the machine displacement — The displacement is determined by one-dimensional table lookup based on the control member position. You have a choice of three interpolation methods and two extrapolation methods.

The variable-displacement machine is represented with the following equations:

$$q = D \cdot \omega - k_m \cdot q_L$$

$$T = D \cdot p + k_m \cdot T_{fr}$$

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

$$p = p_A - p_B$$

where

q	Machine flow rate
p	Pressure differential across the machine
p_A, p_B	Gauge pressures at the block terminals
D	Machine instantaneous displacement
D_{max}	Machine maximum displacement
x	Control member displacement
x_{max}	Control member maximum stroke
T	Torque at the machine shaft
ω	Machine shaft angular velocity
q_L	Leakage flow
T_{fr}	Friction torque
k_m	Machine type coefficient. $k_m = 1$ for the pump, $k_m = -1$ for the motor.

The key parameters that determine machine efficiency are its leakage and friction on the shaft. In the block, these parameters are specified with experimentally-based correlations similar to [1]

$$q_L = D \cdot \omega \cdot k_{L1} \left(\frac{p}{p_{nom}} \right)^{k_{LP}} \left(\frac{D}{D_{\max}} \right)^{k_{LD}} \left(\frac{\omega}{\omega_{nom}} \right)^{k_{L\omega}}$$

$$T_{fr} = D \bullet p \bullet k_{F1} \left(\frac{p}{p_{nom}} \right)^{k_{FP}} \left(\frac{D}{D_{\max}} \right)^{k_{FD}} \left(\frac{\omega}{\omega_{nom}} \right)^{k_{F\omega}}$$

where

p_{nom}	Nominal pressure
ω_{nom}	Nominal angular velocity
k_{L1}	Leakage proportionality coefficient
k_{F1}	Friction proportionality coefficient
k_{LP}, k_{LD}, $k_{L\omega}$, k_{FP}, k_{FD}, $k_{F\omega}$	Approximating coefficients

The approximating coefficients are determined from the efficiency plots, usually provided by the machine manufacturer. With the leakage known, the pump volumetric efficiency can be expressed as

$$\eta_{vp} = \frac{D_{\omega} - q_L}{D_{\omega}} = 1 - k_{L1} \left(\frac{p}{p_{nom}} \right)^{k_{LP}} \left(\frac{D}{D_{\max}} \right)^{k_{LD}} \left(\frac{\omega}{\omega_{nom}} \right)^{k_{L\omega}}$$

For a motor, the expression looks like the following

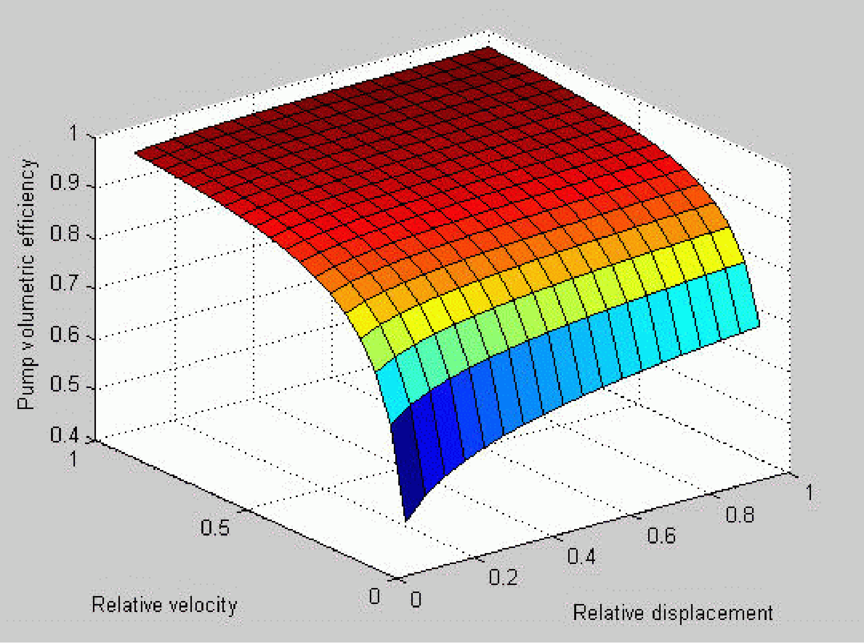
$$\eta_{vm} = \frac{D_{\omega}}{D_{\omega} + q_L} = \frac{1}{1 + k_{L1} \left(\frac{p}{p_{nom}} \right)^{k_{LP}} \left(\frac{D}{D_{\max}} \right)^{k_{LD}} \left(\frac{\omega}{\omega_{nom}} \right)^{k_{L\omega}}}$$

The mechanical efficiency is based on the known friction torque

$$\eta_{mp} = \frac{D_p}{D_p + T_{fr}} = \frac{1}{1 + k_{F1}\left(\frac{p}{p_{nom}}\right)^{k_{FP}}\left(\frac{D}{D_{\max}}\right)^{k_{FD}}\left(\frac{\omega}{\omega_{nom}}\right)^{k_{F\omega}}}$$

$$\eta_{mm} = \frac{D_p - T_{fr}}{D_p} = 1 - k_{F1}\left(\frac{p}{p_{nom}}\right)^{k_{FP}}\left(\frac{D}{D_{\max}}\right)^{k_{FD}}\left(\frac{\omega}{\omega_{nom}}\right)^{k_{F\omega}}$$

The curve-fitting procedure is based on the comparison of the efficiency, determined with one of the above expressions, and the experimental data $\eta_{\exp} = f(p, D, \omega)$, an example of which is shown in the following plot.



The procedure can be performed with the Optimization Toolbox software. For instance, the pump volumetric efficiency approximating coefficients can be found by solving the following problem:

$$\min_{x} F(x)$$

$$x = [k_{L1}, k_{LP}, k_{LD}, k_{L\omega}]$$

$$F(x) = \sum_i \sum_j \sum_k \left(\eta_{\exp}\left(p_i, D_j, \omega_k\right) - \left(1 - k_{L1} \left(\frac{p_i}{p_{nom}}\right)^{k_{LP}} \left(\frac{D_j}{D_{\max}}\right)^{k_{LD}} \left(\frac{\omega_k}{\omega_{nom}}\right)^{k_{L\omega}}\right)\right)^2$$

where

`i`	Number of experimental pressure points, from 1 to `n`
`j`	Number of experimental displacement points, from 1 to `m`
`k`	Number of experimental angular velocity points, from 1 to `l`

Connections A and B are hydraulic conserving ports associated with the machine inlet and outlet, respectively. Connection S is a mechanical rotational conserving port associated with the machine shaft. Connection C is a physical signal port that controls machine displacement. The flow rate from port A to port B causes the shaft to rotate in positive direction, provided positive signal is applied to port C.

Basic Assumptions and Limitations

The model is based on the following assumptions:

- Fluid compressibility is neglected.
- No inertia on the machine shaft is considered.
- The model is applicable only for fluid and fluid temperature at which the approximating coefficients have been determined.

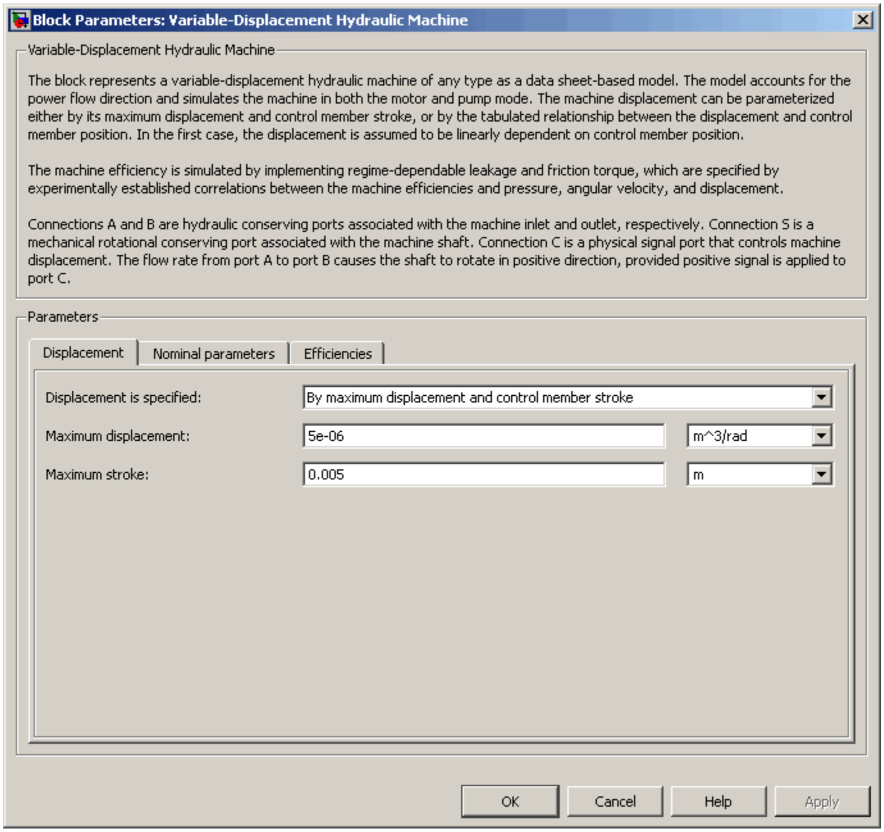
- Extreme caution must be exercised to not exceed the limits within which the approximating coefficients have been determined. The extrapolation could result in large errors.

Dialog Box and Parameters

The block dialog box contains three tabs:

- "Displacement" on page 2-336
- "Nominal Parameters" on page 2-340
- "Efficiencies" on page 2-341

Displacement



Variable-Displacement Hydraulic Machine

The block represents a variable-displacement hydraulic machine of any type as a data sheet-based model. The model accounts for the power flow direction and simulates the machine in both the motor and pump mode. The machine displacement can be parameterized either by its maximum displacement and control member stroke, or by the tabulated relationship between the displacement and control member position. In the first case, the displacement is assumed to be linearly dependent on control member position.

The machine efficiency is simulated by implementing regime-dependable leakage and friction torque, which are specified by experimentally established correlations between the machine efficiencies and pressure, angular velocity, and displacement.

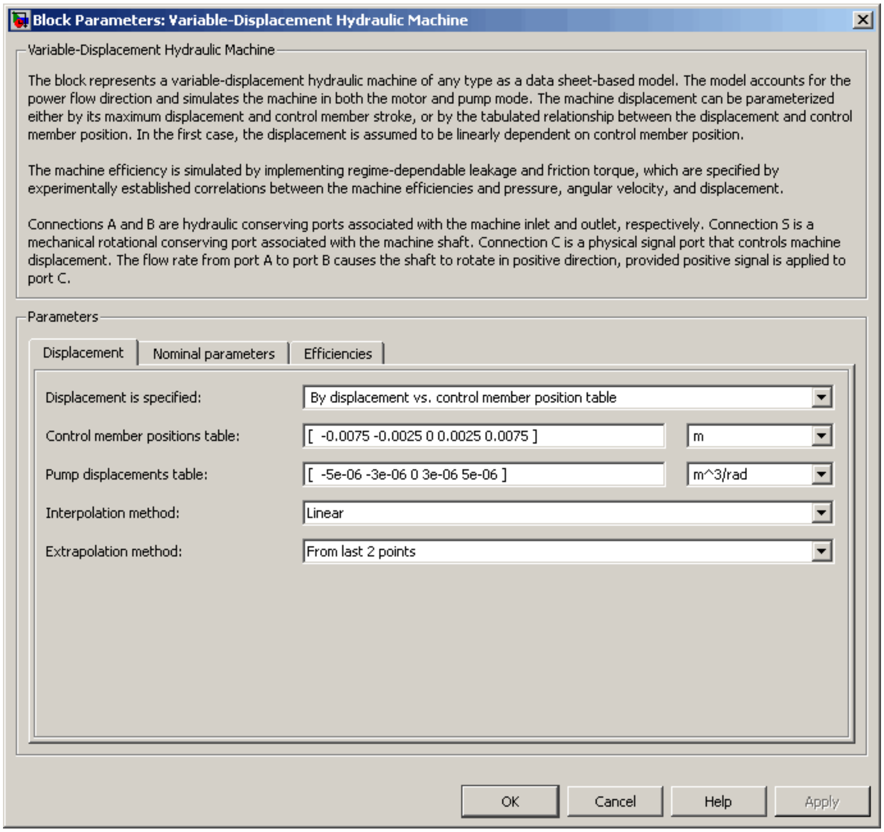
Connections A and B are hydraulic conserving ports associated with the machine inlet and outlet, respectively. Connection S is a mechanical rotational conserving port associated with the machine shaft. Connection C is a physical signal port that controls machine displacement. The flow rate from port A to port B causes the shaft to rotate in positive direction, provided positive signal is applied to port C.

Parameters

Displacement is specified: By maximum displacement and control member stroke

Maximum displacement: 5e-06 m^3/rad

Maximum stroke: 0.005 m



Block Parameters: Variable-Displacement Hydraulic Machine

Variable-Displacement Hydraulic Machine

The block represents a variable-displacement hydraulic machine of any type as a data sheet-based model. The model accounts for the power flow direction and simulates the machine in both the motor and pump mode. The machine displacement can be parameterized either by its maximum displacement and control member stroke, or by the tabulated relationship between the displacement and control member position. In the first case, the displacement is assumed to be linearly dependent on control member position.

The machine efficiency is simulated by implementing regime-dependable leakage and friction torque, which are specified by experimentally established correlations between the machine efficiencies and pressure, angular velocity, and displacement.

Connections A and B are hydraulic conserving ports associated with the machine inlet and outlet, respectively. Connection S is a mechanical rotational conserving port associated with the machine shaft. Connection C is a physical signal port that controls machine displacement. The flow rate from port A to port B causes the shaft to rotate in positive direction, provided positive signal is applied to port C.

Parameters

Displacement is specified: By displacement vs. control member position table

Control member positions table: [-0.0075 -0.0025 0 0.0025 0.0075] m

Pump displacements table: [-5e-06 -3e-06 0 3e-06 5e-06] m^3/rad

Interpolation method: Linear

Extrapolation method: From last 2 points

Displacement is specified

Select one of the following block parameterization options:

- `By maximum displacement and control member stroke` — Provide values for maximum machine displacement and maximum stroke. The displacement is assumed to be linearly dependent on the control member position. This is the default method.
- `By displacement vs. control member position table` — Provide tabulated data of machine displacements and

control member positions. The displacement is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.

Maximum displacement

Machine maximum displacement. The default value is `5e-6` m^3/rad. This parameter is used if displacement is specified as `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 displacement is specified as `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 nonuniformly 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 displacement is specified as `By displacement vs. control member position table`.

Pump displacements table

Specify the vector of output values for the machine displacement as a tabulated 1-by-`m` array. The machine 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 displacement is specified as `By displacement vs. control member position table`.

Interpolation method

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

- `Linear` — Uses a linear interpolation function.
- `Cubic` — Uses the Piecewise Cubic Hermite Interpolation 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 displacement is specified as `By displacement vs.  control member position table`.

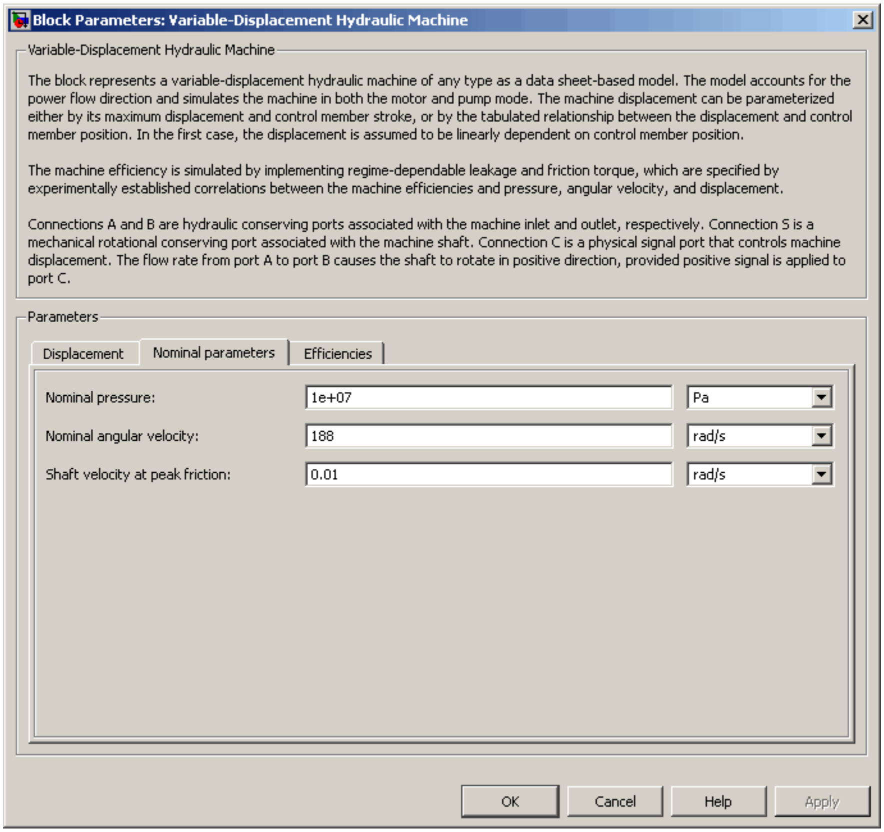
Extrapolation method

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

- `From last 2 points` — Extrapolates using the linear method (regardless of the interpolation method specified), based on the last two output values at the appropriate end of the range. That is, the block uses the first and second specified output values if the input value is below the specified range, and the two last specified output values if the input value is above the specified range.
- `From last point` — Uses the last specified output value at the appropriate end of the range. That is, the block uses the last specified output value for all input values greater than the last specified input argument, and the first specified output value for all input values less than the first specified input argument.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if displacement is specified as `By displacement vs.  control member position table`.

Nominal Parameters



Variable-Displacement Hydraulic Machine

The block represents a variable-displacement hydraulic machine of any type as a data sheet-based model. The model accounts for the power flow direction and simulates the machine in both the motor and pump mode. The machine displacement can be parameterized either by its maximum displacement and control member stroke, or by the tabulated relationship between the displacement and control member position. In the first case, the displacement is assumed to be linearly dependent on control member position.

The machine efficiency is simulated by implementing regime-dependable leakage and friction torque, which are specified by experimentally established correlations between the machine efficiencies and pressure, angular velocity, and displacement.

Connections A and B are hydraulic conserving ports associated with the machine inlet and outlet, respectively. Connection S is a mechanical rotational conserving port associated with the machine shaft. Connection C is a physical signal port that controls machine displacement. The flow rate from port A to port B causes the shaft to rotate in positive direction, provided positive signal is applied to port C.

Parameters

Nominal pressure: 1e+07 Pa

Nominal angular velocity: 188 rad/s

Shaft velocity at peak friction: 0.01 rad/s

Nominal pressure

Nominal pressure differential across the machine. The default value is `1e7` Pa.

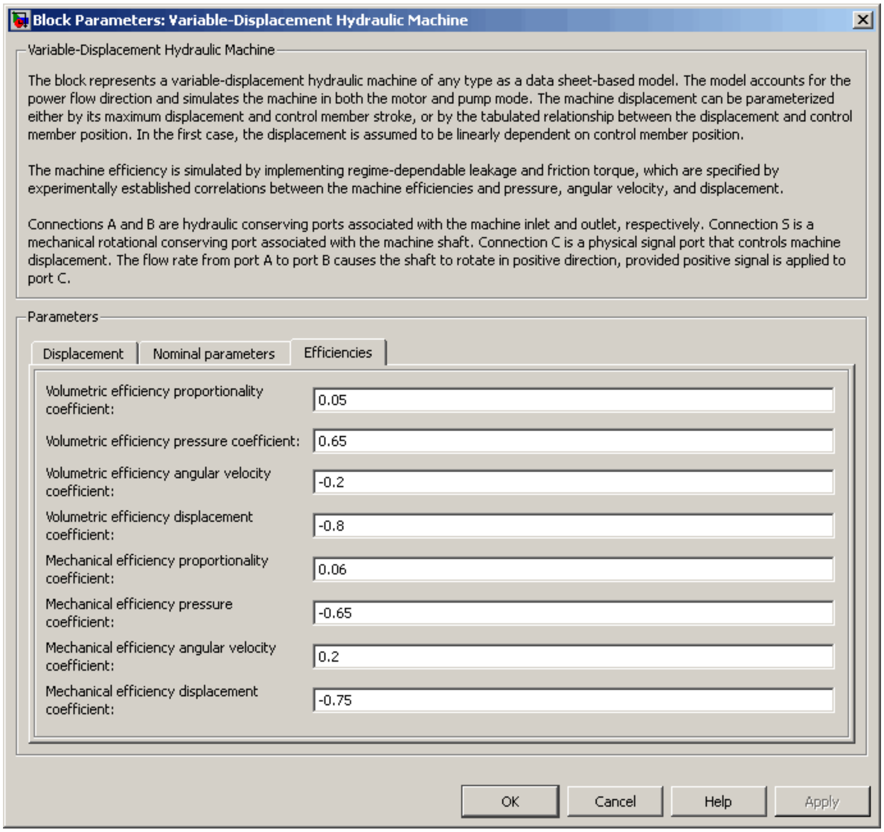
Nominal angular velocity

Nominal angular velocity of the output shaft. The default value is `188` rad/s.

Shaft velocity at peak friction

The friction torque on the machine shaft ideally should be introduced as T_{fr}`sign`(ω). To avoid discontinuity at ω –> 0, the friction is defined as T_{fr}`tanh`(4ω /ω_{max}), where ω_{max} is a small velocity, representing the shaft velocity at peak friction, at which `tanh`(4ω /ω_{max}) is equal to 0.999. The default value of ω_{max} is `0.01` rad/s.

Efficiencies



Variable-Displacement Hydraulic Machine

The block represents a variable-displacement hydraulic machine of any type as a data sheet-based model. The model accounts for the power flow direction and simulates the machine in both the motor and pump mode. The machine displacement can be parameterized either by its maximum displacement and control member stroke, or by the tabulated relationship between the displacement and control member position. In the first case, the displacement is assumed to be linearly dependent on control member position.

The machine efficiency is simulated by implementing regime-dependable leakage and friction torque, which are specified by experimentally established correlations between the machine efficiencies and pressure, angular velocity, and displacement.

Connections A and B are hydraulic conserving ports associated with the machine inlet and outlet, respectively. Connection S is a mechanical rotational conserving port associated with the machine shaft. Connection C is a physical signal port that controls machine displacement. The flow rate from port A to port B causes the shaft to rotate in positive direction, provided positive signal is applied to port C.

Volumetric efficiency proportionality coefficient: 0.05

Volumetric efficiency pressure coefficient: 0.65

Volumetric efficiency angular velocity coefficient: -0.2

Volumetric efficiency displacement coefficient: -0.8

Mechanical efficiency proportionality coefficient: 0.06

Mechanical efficiency pressure coefficient: -0.65

Mechanical efficiency angular velocity coefficient: 0.2

Mechanical efficiency displacement coefficient: -0.75

Volumetric efficiency proportionality coefficient
Approximating coefficient k_{L1} in the block description preceding. The default value is `0.05`.

Volumetric efficiency pressure coefficient
Approximating coefficient k_{LP} in the block description preceding. The default value is `0.65`.

Volumetric efficiency angular velocity coefficient
Approximating coefficient $k_{L\omega}$ in the block description preceding. The default value is `-0.2`.

Volumetric efficiency displacement coefficient
Approximating coefficient k_{LD} in the block description preceding. The default value is `-0.8`.

Mechanical efficiency proportionality coefficient
Approximating coefficient k_{F1} in the block description preceding. The default value is `0.06`.

Mechanical efficiency pressure coefficient
Approximating coefficient k_{FP} in the block description preceding. The default value is `-0.65`.

Mechanical efficiency angular velocity coefficient
Approximating coefficient $k_{F\omega}$ in the block description preceding. The default value is `0.2`.

Mechanical efficiency displacement coefficient
Approximating coefficient k_{FD} in the block description preceding. The default value is `-0.75`.

Restricted Parameters

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

- **Displacement is specified**
- **Interpolation method**
- **Extrapolation method**

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

Ports

The block has the following ports:

A
: Hydraulic conserving port associated with the machine inlet.

B
: Hydraulic conserving port associated with the machine outlet.

C
: Physical signal port that controls machine displacement.

S
: Mechanical rotational conserving port associated with the machine shaft.

References

[1] C.R. Cornell, *Dynamic Simulation of a Hydrostatically Propelled Vehicle*, SAE paper 811253, 1981, p. 22

See Also

Variable-Displacement Motor

Variable-Displacement Pump

Variable-Displacement Motor

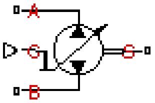
Purpose

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:

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

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

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

$$k_{leak} = k_{HP} / \nu \bullet \rho$$

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

$$p = p_A - p_B$$

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, $\mu = \nu \cdot \rho$

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

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

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

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

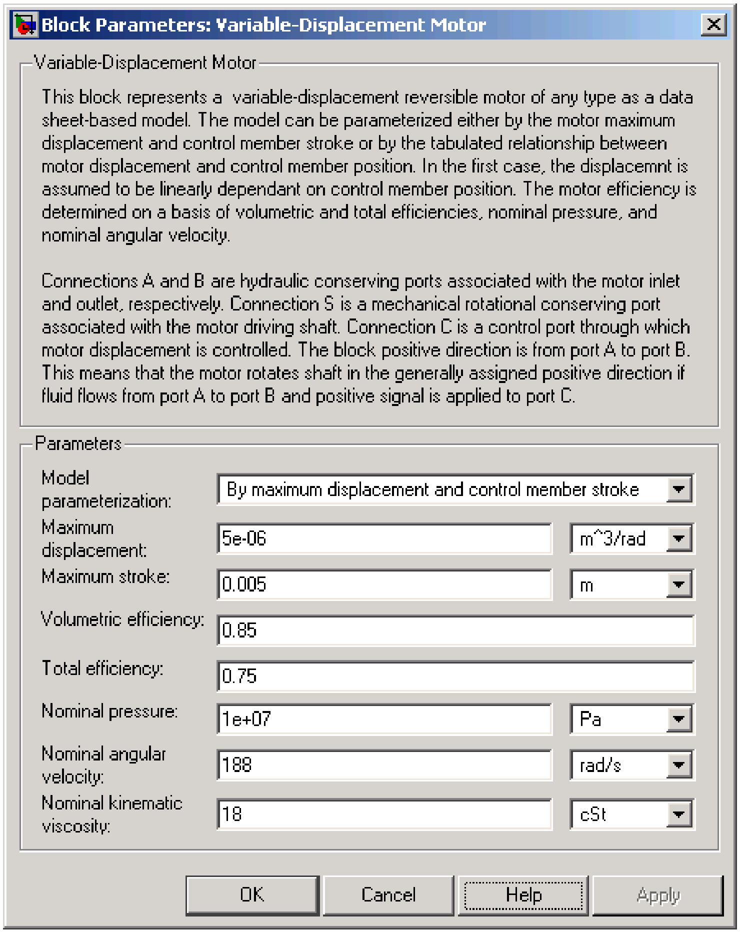
The block positive direction is from port A to port B. This means that the motor rotates its shaft in the globally assigned positive direction if the fluid flows from port A to port B and a positive signal is applied to port C.

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



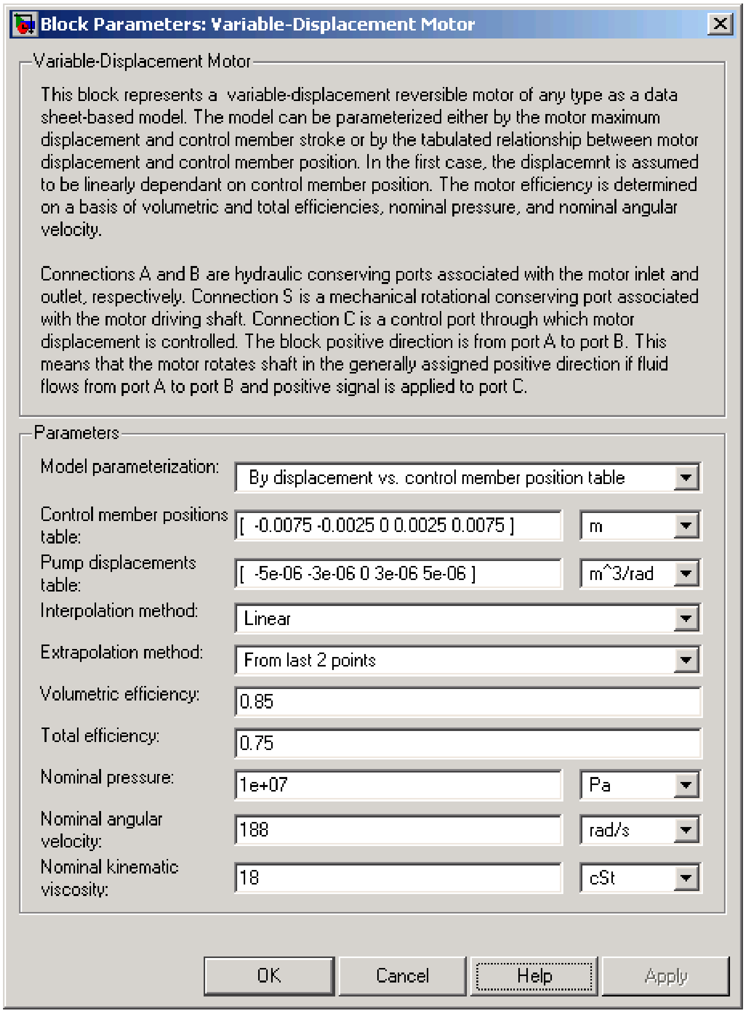
Variable-Displacement Motor

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 displacemnt 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



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 displacemnt 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 displacement vs. control member position table
Control member positions table: [-0.0075 -0.0025 0 0.0025 0.0075] m
Pump displacements table: [-5e-06 -3e-06 0 3e-06 5e-06] m^3/rad
Interpolation method: Linear
Extrapolation method: From last 2 points
Volumetric efficiency: 0.85
Total efficiency: 0.75
Nominal pressure: 1e+07 Pa
Nominal angular velocity: 188 rad/s
Nominal kinematic viscosity: 18 cSt

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` 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 nonuniformly 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 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`.

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 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 `1e7` 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.

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

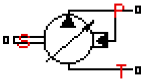
Purpose

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

Library

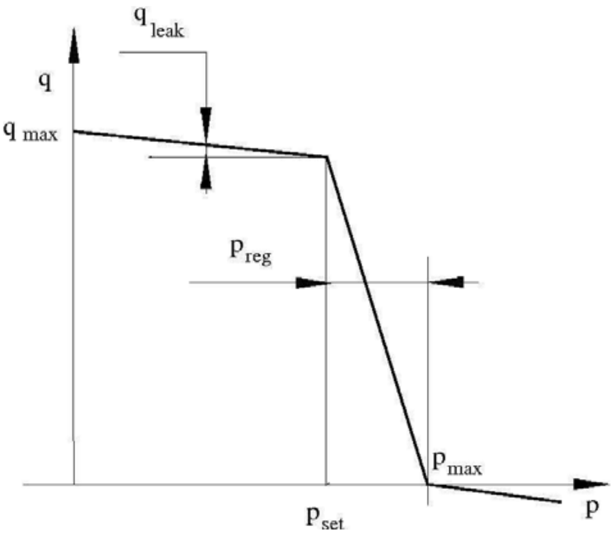
Pumps and Motors

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

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

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

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

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

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

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

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

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

$$p = p_P - p_T$$

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
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

q_{leak}	Leakage flow
d, l	Geometric parameters of the leakage path
μ	Fluid dynamic viscosity, $\mu = \nu \cdot \rho$

The leakage flow at $p = p_{nom}$ and $\nu = \nu_{nom}$ can be determined from the catalog data

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

which provides the formula to determine the Hagen-Poiseuille coefficient

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

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

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

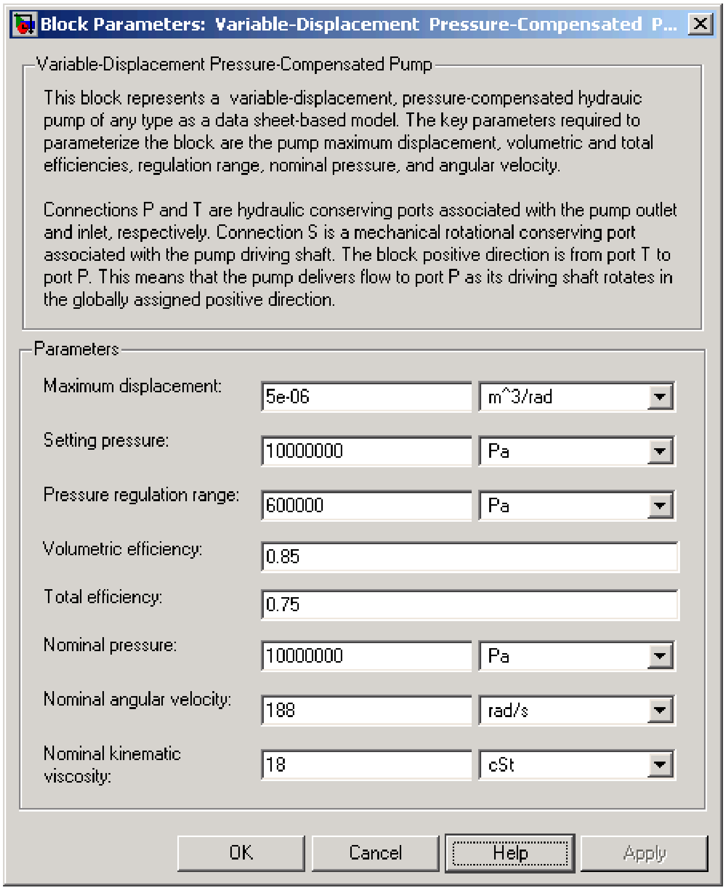
The block positive direction is from port T to port P. This means that the pump transfers fluid from T to P provided that the shaft S rotates in the positive direction. The pressure differential across the pump is determined as $p = p_P - p_T$.

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.

Dialog Box and Parameters



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

Maximum displacement

Pump displacement. The default value is `5e-6` m^3/rad.

Setting pressure

Pump pressure setting. The default value is `1e7` Pa.

Pressure regulation range

Pressure range required to change the pump displacement from its maximum to zero. The default value is `6e5` Pa.

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

Total efficiency
Pump total efficiency, which is determined as a ratio between the hydraulic power at the pump outlet and mechanical power at the driving shaft at nominal pressure, angular velocity, and fluid viscosity. The default value is `0.75`.

Nominal pressure
Pressure differential across the pump, at which both the volumetric and total efficiencies are specified. The default value is `1e7` Pa.

Nominal angular velocity
Angular velocity of the driving shaft, at which both the volumetric and total efficiencies are specified. The default value is `188` rad/s.

Nominal kinematic viscosity
Working fluid kinematic viscosity, at which both the volumetric and total efficiencies are specified. The default value is `18` cSt.

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 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

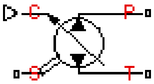
Purpose

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:

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

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

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

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

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

$$p = p_P - p_T$$

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

The leakage flow is determined based on the assumption that it is linearly proportional to the pressure differential across the pump and can be computed by using the Hagen-Poiseuille formula

$$p = \frac{128\mu l}{\pi d^4} q_{leak} = \frac{\mu}{k_{HP}} q_{leak}$$

where

q_{leak}	Leakage flow
d, l	Geometric parameters of the leakage path
μ	Fluid dynamic viscosity, $\mu = \nu \cdot \rho$

The leakage flow at $p = p_{nom}$ and $\nu = \nu_{nom}$ can be determined from the catalog data

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

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

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

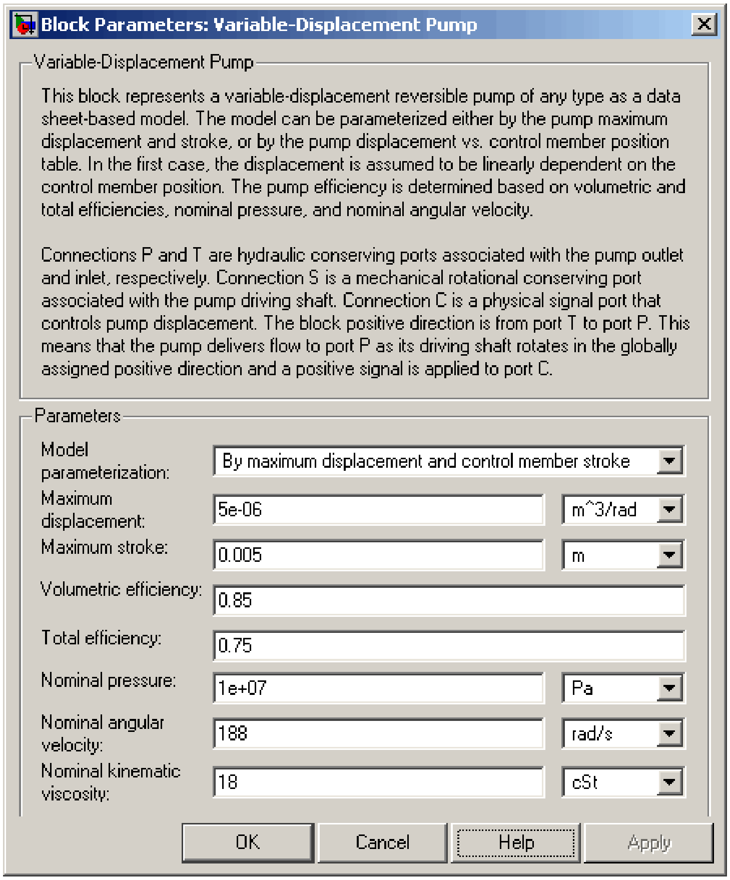
The block positive direction is from port T to port P. This means that the pump transfers fluid from T to P as its driving shaft S rotates in the globally assigned positive direction and a positive signal is applied to port C.

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

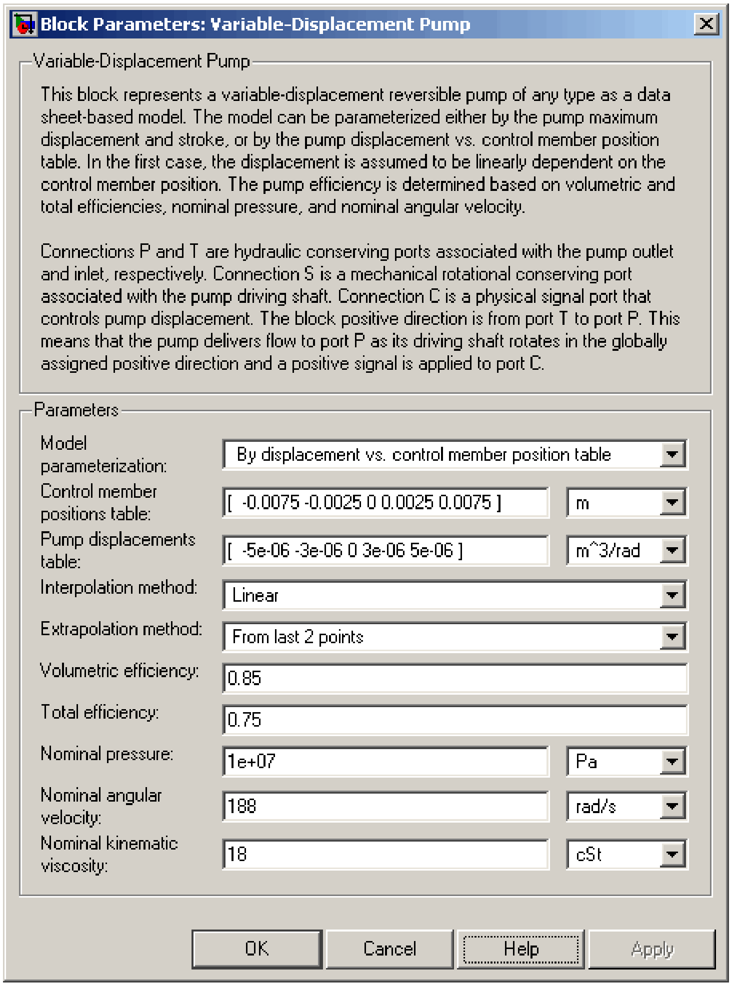
Variable-Displacement Pump

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



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 displacement vs. control member position table

Control member positions table: [-0.0075 -0.0025 0 0.0025 0.0075] m

Pump displacements table: [-5e-06 -3e-06 0 3e-06 5e-06] m^3/rad

Interpolation method: Linear

Extrapolation method: From last 2 points

Volumetric efficiency: 0.85

Total efficiency: 0.75

Nominal pressure: 1e+07 Pa

Nominal angular velocity: 188 rad/s

Nominal kinematic viscosity: 18 cSt

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 nonuniformly 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`.

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 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 `1e7` Pa.

Nominal angular velocity

Angular velocity of the driving shaft, at which both the volumetric and total efficiencies are specified. The default value is `188` rad/s.

Nominal kinematic viscosity

Working fluid kinematic viscosity, at which both the volumetric and total efficiencies are specified. The default value is `18` cSt.

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 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

Index