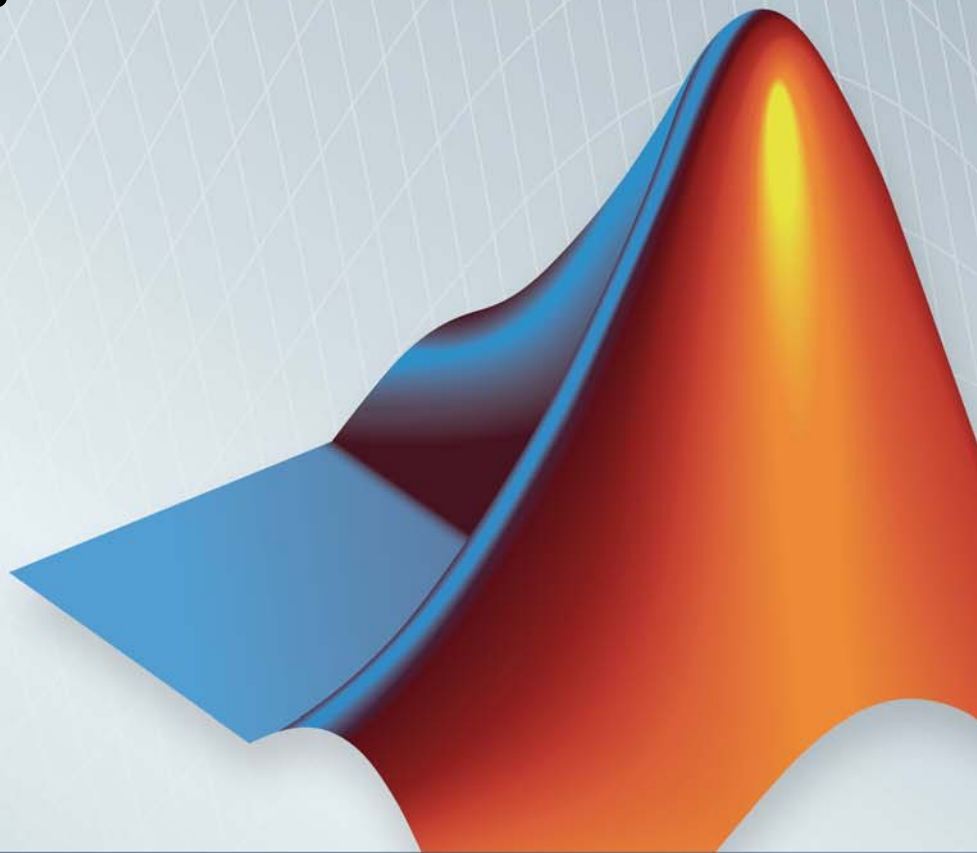


**SimHydraulics®**

Reference

**R2013b**



**MATLAB® & SIMULINK®**



## How to Contact MathWorks



[www.mathworks.com](http://www.mathworks.com) Web  
[comp.soft-sys.matlab](mailto:comp.soft-sys.matlab) Newsgroup  
[www.mathworks.com/contact\\_TS.html](http://www.mathworks.com/contact_TS.html) Technical Support



[suggest@mathworks.com](mailto:suggest@mathworks.com) Product enhancement suggestions  
[bugs@mathworks.com](mailto:bugs@mathworks.com) Bug reports  
[doc@mathworks.com](mailto:doc@mathworks.com) Documentation error reports  
[service@mathworks.com](mailto:service@mathworks.com) Order status, license renewals, passcodes  
[info@mathworks.com](mailto: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*<sup>®</sup> Reference

© COPYRIGHT 2006–2013 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](http://www.mathworks.com/trademarks) for a list of additional trademarks. Other product or brand names may be trademarks or registered trademarks of their respective holders.

### Patents

MathWorks products are protected by one or more U.S. patents. Please see [www.mathworks.com/patents](http://www.mathworks.com/patents) for more information.

### **Revision History**

March 2006	Online only	New for Version 1.0 (Release 2006a+)
September 2006	Online only	Revised for Version 1.1 (Release 2006b)
March 2007	Online only	Revised for Version 1.2 (Release 2007a)
September 2007	Online only	Revised for Version 1.2.1 (Release 2007b)
March 2008	Online only	Revised for Version 1.3 (Release 2008a)
October 2008	Online only	Revised for Version 1.4 (Release 2008b)
March 2009	Online only	Revised for Version 1.5 (Release 2009a)
September 2009	Online only	Revised for Version 1.6 (Release 2009b)
March 2010	Online only	Revised for Version 1.7 (Release 2010a)
September 2010	Online only	Revised for Version 1.8 (Release 2010b)
April 2011	Online only	Revised for Version 1.9 (Release 2011a)
September 2011	Online only	Revised for Version 1.10 (Release 2011b)
March 2012	Online only	Revised for Version 1.10.1 (Release 2012a)
September 2012	Online only	Revised for Version 1.11 (Release 2012b)
March 2013	Online only	Revised for Version 1.12 (Release 2013a)
September 2013	Online only	Revised for Version 1.13 (Release 2013b)



## Blocks — Alphabetical List

**1**

## Bibliography

**A**

## Glossary

## Index



# Blocks — Alphabetical List

---

# 2-Position Valve Actuator

---

**Purpose** Actuator for two-position valves

**Library** Valve Actuators

**Description**



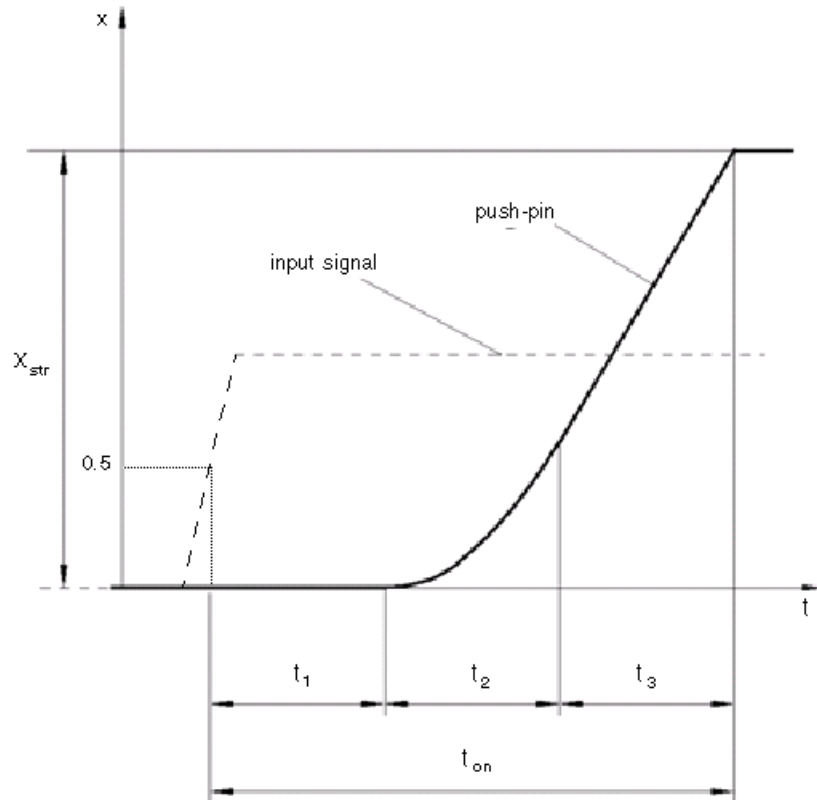
The 2-Position Valve Actuator block represents an actuator that you can use with directional valves to control their position. This actuator can drive a two-position valve. The block is developed as a data-sheet-based model and all its parameters are generally provided in catalogs or data sheets. The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

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

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



## 2-Position Valve Actuator



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

## 2-Position Valve Actuator

---

switch-off time. **Switching-on time** and **Switching-off time** are the block parameters.

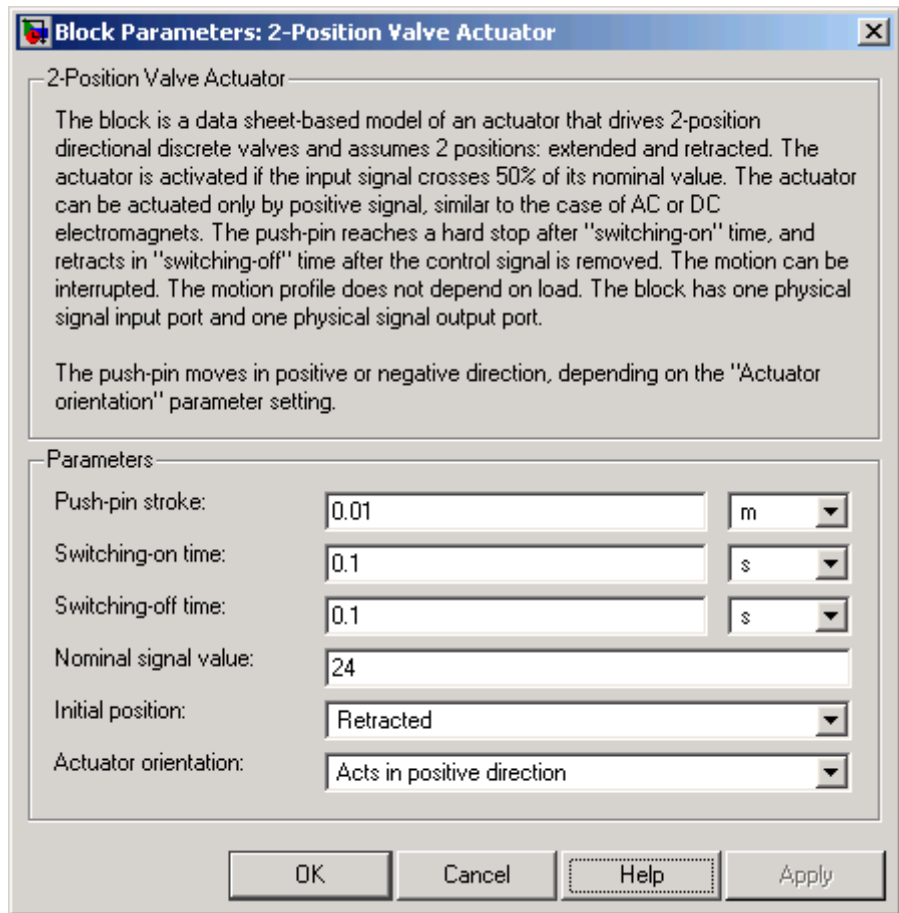
The transition in any direction can be interrupted at any time by changing the input signal. If motion is interrupted, the switch-on or switch-off times are proportionally decreased depending on the instantaneous push-pin position.

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

### **Basic Assumptions and Limitations**

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

## Dialog Box and Parameters



### Push-pin stroke

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

### Switching-on time

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

## 2-Position Valve Actuator

---

### Switching-off time

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

### Nominal signal value

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

### Initial position

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

In selecting the initial position, consider the following:

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

### Actuator orientation

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

## Restricted Parameters

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

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

All other block parameters are available for modification.

## Ports

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

## Examples

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

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

## See Also

3-Position Valve Actuator

Hydraulic Double-Acting Valve Actuator

Hydraulic Single-Acting Valve Actuator

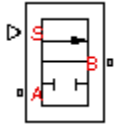
Proportional and Servo-Valve Actuator

# 2-Way Directional Valve

**Purpose** Hydraulic continuous 2-way directional valve

**Library** Directional Valves

## Description



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

$$h = x_0 + x$$

where

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

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

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

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

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

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential

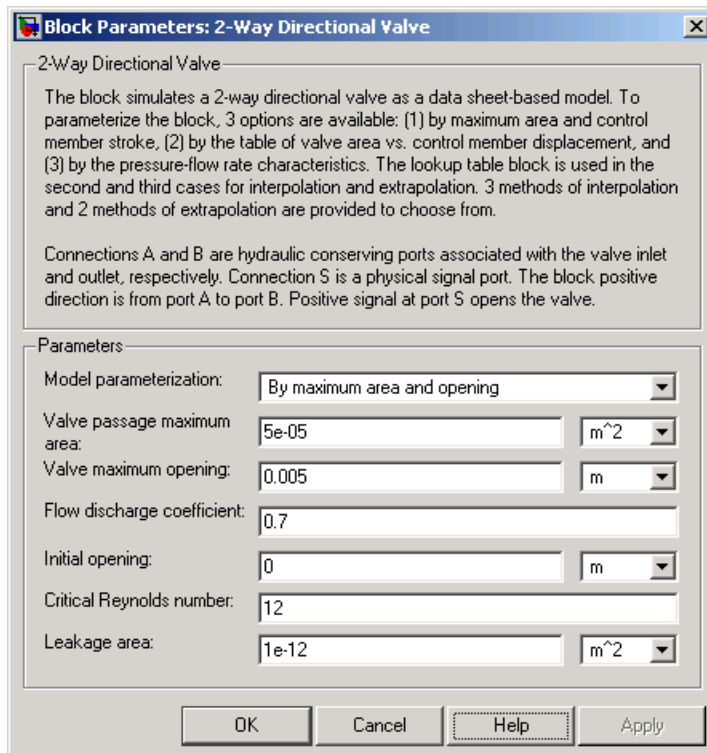
# 2-Way Directional Valve

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

## Basic Assumptions and Limitations

- 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





# 2-Way Directional Valve

**Block Parameters: 2-Way Directional Valve**

2-Way Directional Valve

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

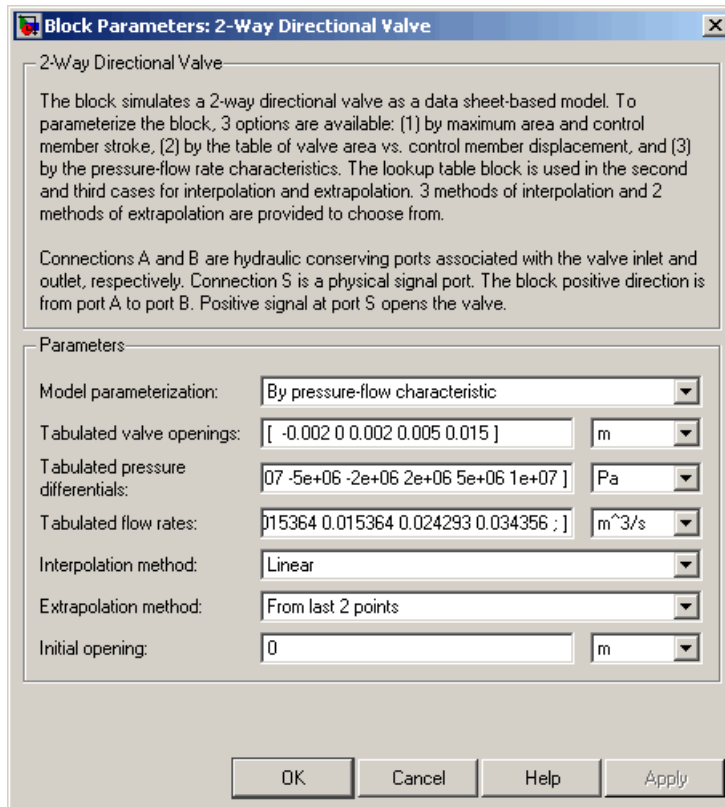
Connections A and B are hydraulic conserving ports associated with the valve inlet and outlet, respectively. Connection S is a physical signal port. The block positive direction is from port A to port B. Positive signal at port S opens the valve.

Parameters:

Model parameterization:	By area vs. opening table	
Tabulated valve openings:	[ -0.002 0 0.002 0.005 0.015 ]	m
Tabulated valve passage area:	[ 7 4.0736e-05 0.00011438 0.00034356 ]	m <sup>2</sup>
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 <sup>2</sup>

OK Cancel Help Apply

# 2-Way Directional Valve



## Model parameterization

Select one of the following methods for specifying the valve:

- **By maximum area and opening** — Provide values for the maximum valve passage area and the maximum valve opening. The passage area is linearly dependent on the control member displacement, that is, the valve is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.

- **By area vs. opening table** — Provide tabulated data of valve openings and corresponding valve passage areas. The passage area is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.
- **By pressure-flow characteristic** — Provide tabulated data of valve openings, pressure differentials, and corresponding flow rates. The flow rate is determined by two-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.

### **Valve passage maximum area**

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

### **Valve maximum opening**

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

### **Tabulated valve openings**

Specify the vector of input values for valve openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in meters, are  $[-0.002 \ 0 \ 0.002 \ 0.005 \ 0.015]$ . 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.

## 2-Way Directional Valve

---

### Tabulated valve passage area

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

### Tabulated pressure differentials

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

### Tabulated flow rates

Specify the flow rates as an  $m$ -by- $n$  matrix, where  $m$  is the number of valve openings and  $n$  is the number of pressure differentials. Each value in the matrix specifies flow rate taking place at a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in  $\text{m}^3/\text{s}$ , 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 Polynomial (PCHIP). For two-dimensional table lookup (By pressure-flow characteristic), uses the bicubic interpolation algorithm.
- **Spline** — For one-dimensional table lookup (By area vs. opening table), uses the cubic spline interpolation algorithm. For two-dimensional table lookup (By pressure-flow characteristic), uses the bicubic spline interpolation algorithm.

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

## Extrapolation method

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

- **From last 2 points** — Extrapolates using the linear method (regardless of the interpolation method specified), based on the last two output values at the appropriate end of the range. That is, the block uses the first and second specified output values if the input value is below the specified range, and the

## 2-Way Directional Valve

---

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 simulation to fail. Therefore, MathWorks

recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## Restricted Parameters

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

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

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

## Global Parameters

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

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

## 2-Way Directional Valve

---

### **Examples**

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

### **See Also**

3-Way Directional Valve

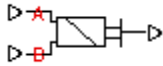
4-Way Directional Valve



**Purpose** Actuator for three-position valves

**Library** Valve Actuators

## Description

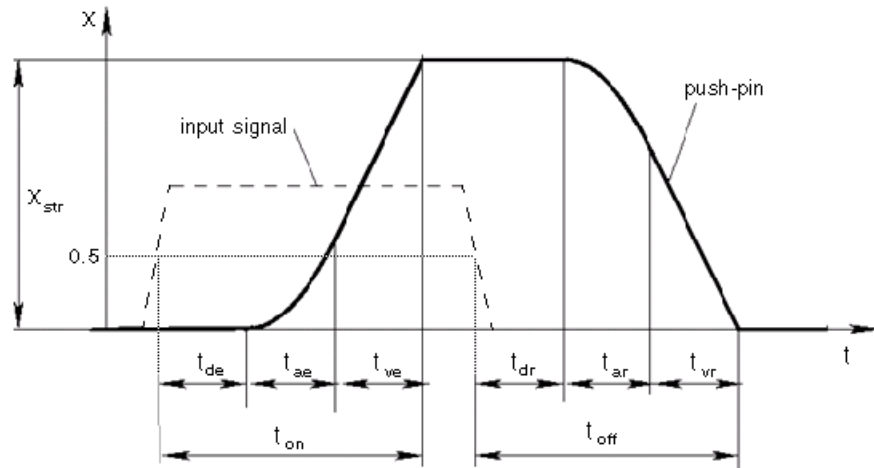


The 3-Position Valve Actuator block represents an actuator that you can use with directional valves to control their position. This actuator can drive a three-position valve. The block is developed as a data-sheet-based model and all its parameters are generally provided in catalogs or data sheets. The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

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

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

# 3-Position Valve Actuator



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

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

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

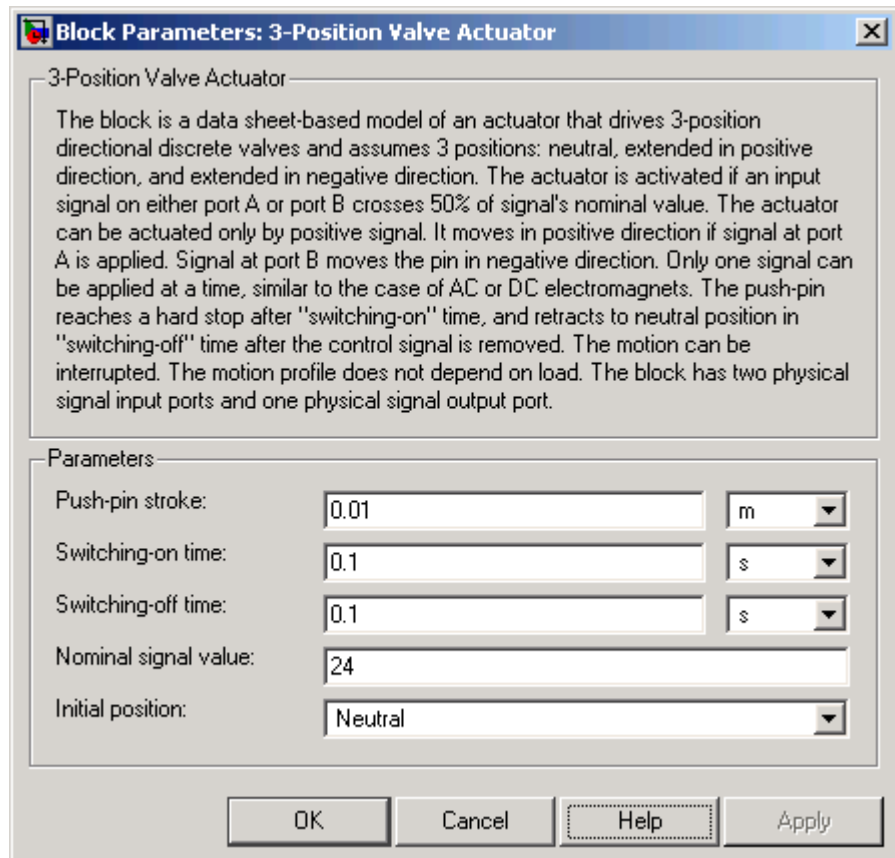
Only positive signals activate the actuator. In other words, negative signals at ports A and B have no effect on the actuator, which is similar to the behavior of electromagnetically controlled 3-position directional valves.

### **Basic Assumptions and Limitations**

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

# 3-Position Valve Actuator

## Dialog Box and Parameters



### Push-pin stroke

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

### Switching-on time

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

### Switching-off time

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

## Nominal signal value

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

## Initial position

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

In selecting the initial position, consider the following:

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

## Restricted Parameters

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

- **Initial position**

All other block parameters are available for modification.

## Ports

The block has the following ports:

A

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

## 3-Position Valve Actuator

---

B

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

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

### Examples

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

### See Also

2-Position Valve Actuator

Hydraulic Double-Acting Valve Actuator

Hydraulic Single-Acting Valve Actuator

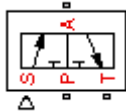
Proportional and Servo-Valve Actuator

# 3-Way Directional Valve

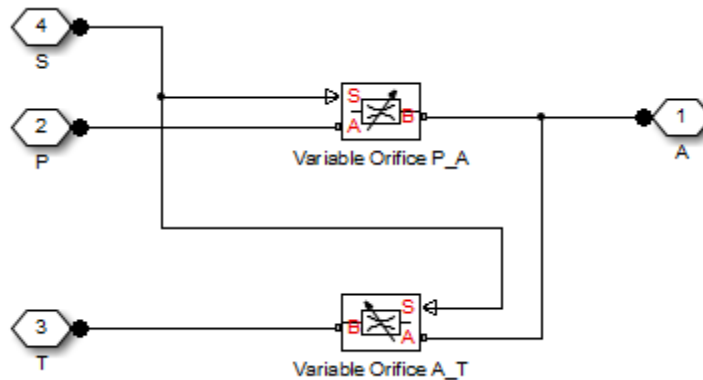
**Purpose** Hydraulic continuous 3-way directional valve

**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:

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

## 3-Way Directional Valve

---

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

where

$h_{PA}$	Orifice opening for the <code>orifice_PA</code> block
$h_{AT}$	Orifice opening for the <code>orifice_AT</code> block
$h_{PA0}$	Initial opening for the <code>orifice_PA</code> block
$h_{AT0}$	Initial opening for the <code>orifice_AT</code> 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 simulation to fail.

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

If you need to simulate a nonsymmetrical 3-way valve (i.e., with different orifices), use any of the variable orifice blocks from the Orifices 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.

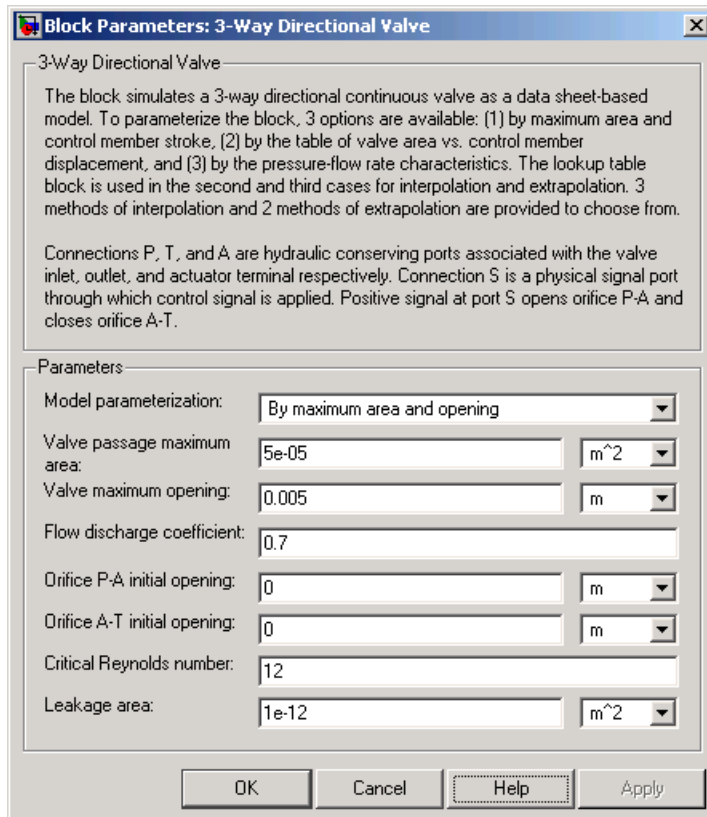
Positive signal at the physical signal port S opens the orifice in the P-A path and closes the orifice in the A-T path. The directionality of nested blocks is clear from the schematic diagram.

# 3-Way Directional Valve

## Basic Assumptions and Limitations

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



# 3-Way Directional Valve

**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:

Tabulated valve openings:

Tabulated valve passage area:

Interpolation method:

Extrapolation method:

Flow discharge coefficient:

Orifice P-A initial opening:

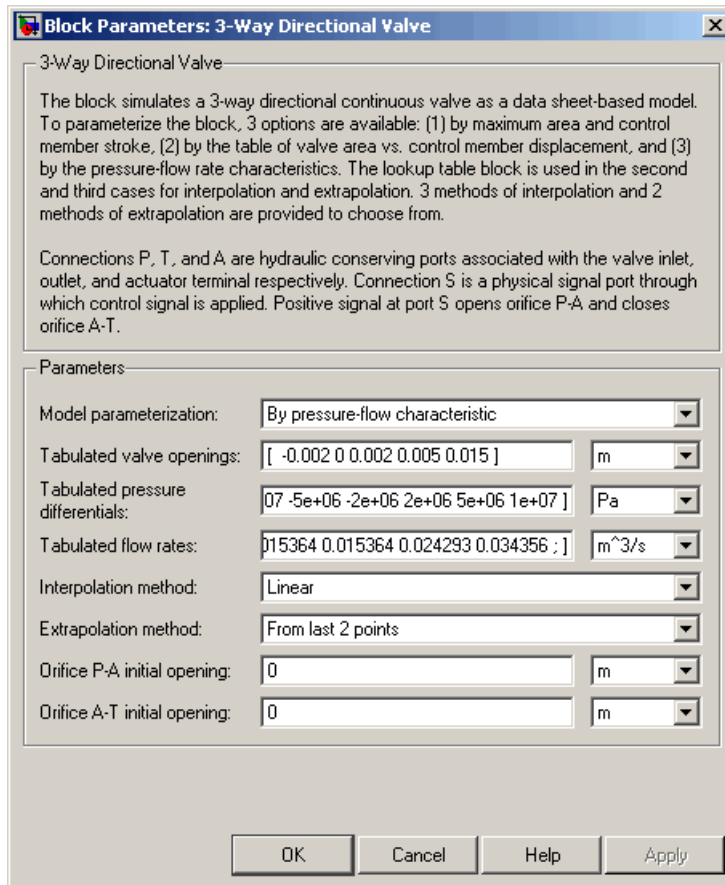
Orifice A-T initial opening:

Critical Reynolds number:

Leakage area:

OK Cancel Help Apply

# 3-Way Directional Valve



## Model parameterization

Select one of the following methods for specifying the valve:

- By maximum area and opening — Provide values for the maximum valve passage area and the maximum valve opening. The passage area is linearly dependent on the control member displacement, that is, the valve is closed at the initial position of the control member (zero displacement), and the maximum

opening takes place at the maximum displacement. This is the default method.

- **By area vs. opening table** — Provide tabulated data of valve openings and corresponding valve passage areas. The passage area is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.
- **By pressure-flow characteristic** — Provide tabulated data of valve openings, pressure differentials, and corresponding flow rates. The flow rate is determined by two-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.

### **Valve passage maximum area**

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

### **Valve maximum opening**

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

### **Tabulated valve openings**

Specify the vector of input values for valve openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in meters, are  $[-0.002 \ 0 \ 0.002 \ 0.005 \ 0.015]$ . 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**

## 3-Way Directional Valve

---

characteristic, the **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

### Tabulated valve passage area

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

### Tabulated pressure differentials

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

### Tabulated flow rates

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

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

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

## Interpolation method

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

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

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

## Extrapolation method

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

- **From last 2 points**—Extrapolates using the linear method (regardless of the interpolation method specified), based on the last two output values at the appropriate end of the range. That is, the block uses the first and second specified output

## 3-Way Directional Valve

---

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 simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## Restricted Parameters

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

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

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

## Global Parameters

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

# 3-Way Directional Valve

---

- P Hydraulic conserving port associated with the pressure supply line inlet.
- T Hydraulic conserving port associated with the return line connection.
- A Hydraulic conserving port associated with the actuator connection port.
- S Physical signal port to control spool displacement.

## Examples

You can see the 3-Way Directional Valve block in the Hydraulic Circuit with 3-Way Valve and Differential Cylinder example, 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

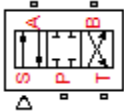
## Purpose

Hydraulic continuous 4-way directional valve

## Library

Directional Valves

## Description

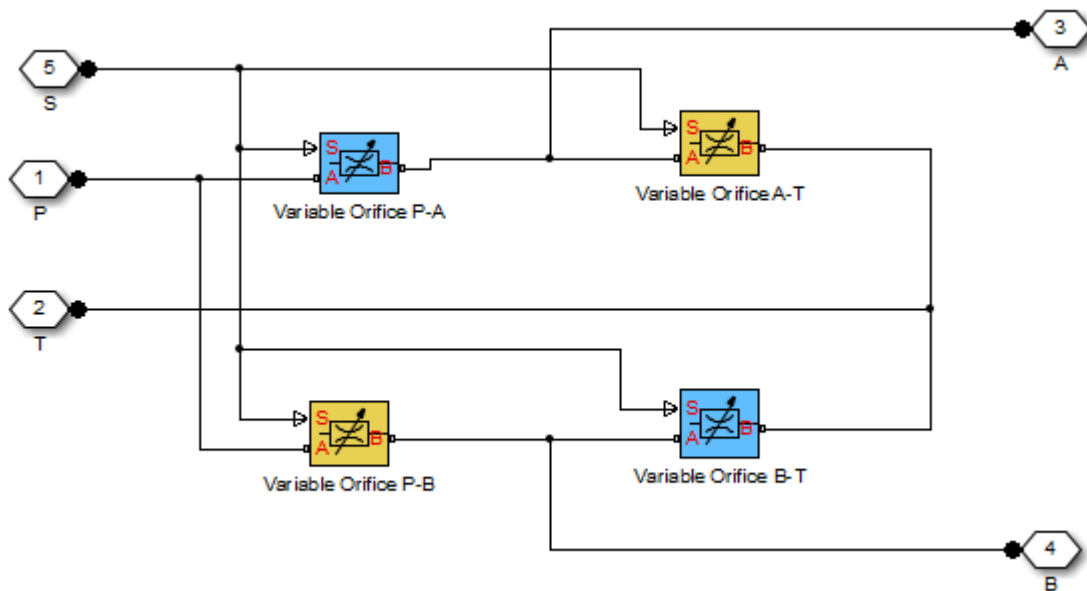


The 4-Way Directional Valve block represents a continuous 4-way directional valve. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position.

There are multiple configurations of 4-way directional valves, depending on the port connections in three distinctive valve positions: leftmost, neutral, and rightmost. This block lets you model the most popular configurations by changing the initial openings of the orifices, as shown in Basic 4-Way Directional Valve Configurations on page 1-39. Other SimHydraulics® blocks provide more 4-way and 6-way directional valve configurations. For more information, see “Modeling Directional Valves”.

The 4-Way Directional Valve block is built of four Variable Orifice blocks, connected as shown in the following diagram.

# 4-Way Directional Valve



The Variable Orifice blocks are installed as follows: orifice P-A is in the P-A path, orifice P-B is in the P-B path, orifice A-T is in the A-T path, and orifice B-T 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 the orifices colored blue in the block diagram (orifices P-A and B-T ) and closes the orifices colored yellow (orifices P-B and A-T). As a result, the openings of the orifices are computed as follows:

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

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

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

# 4-Way Directional Valve

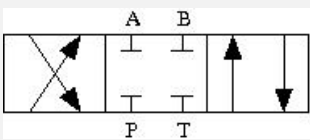
$$h_{BT} = h_{BT0} + x$$

where

- $h_{PA}$  Orifice opening for the Variable Orifice P-A block
- $h_{PB}$  Orifice opening for the Variable Orifice P-B block
- $h_{AT}$  Orifice opening for the Variable Orifice A-T block
- $h_{BT}$  Orifice opening for the Variable Orifice B-T block
- $h_{PA0}$  Initial opening for the Variable Orifice P-A block
- $h_{PB0}$  Initial opening for the Variable Orifice P-B block
- $h_{AT0}$  Initial opening for the Variable Orifice A-T block
- $h_{BT0}$  Initial opening for the Variable Orifice B-T block
- $x$  Control member displacement from initial position

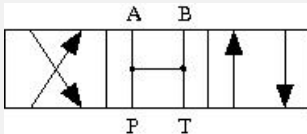
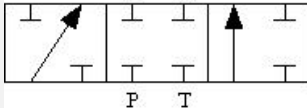
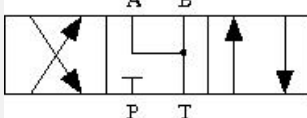
By default, all initial openings are set to zero. By adjusting their values, you can obtain 11 different configurations, as shown in the following table. To specify the initial openings of the orifices, use the “Initial Openings” on page 1-52 tab of the block dialog box.

## Basic 4-Way Directional Valve Configurations

No	Configuration	Initial Openings
1		<p>All four orifices are overlapped in neutral position:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening &lt; 0</li> <li>• Orifice P-B initial opening &lt; 0</li> <li>• Orifice A-T initial opening &lt; 0</li> </ul>

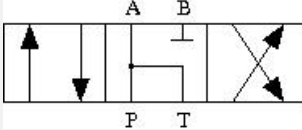
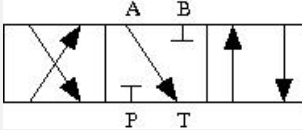
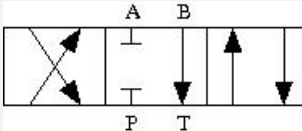
# 4-Way Directional Valve

## Basic 4-Way Directional Valve Configurations (Continued)

No	Configuration	Initial Openings
2		<ul style="list-style-type: none"> <li>• Orifice B-T initial opening &lt; 0</li> </ul> <p>All four orifices are open (underlapped) in neutral position:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening &gt; 0</li> <li>• Orifice P-B initial opening &gt; 0</li> <li>• Orifice A-T initial opening &gt; 0</li> <li>• Orifice B-T initial opening &gt; 0</li> </ul>
3		<p>Orifices P-A and P-B are overlapped. Orifices A-T and B-T are overlapped for more than valve stroke:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening &lt; 0</li> <li>• Orifice P-B initial opening &lt; 0</li> <li>• Orifice A-T initial opening &lt; - valve_stroke</li> <li>• Orifice B-T initial opening &lt; - valve_stroke</li> </ul>
4		<p>Orifices P-A and P-B are overlapped, while orifices A-T and B-T are open:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening &lt; 0</li> <li>• Orifice P-B initial opening &lt; 0</li> <li>• Orifice A-T initial opening &gt; 0</li> <li>• Orifice B-T initial opening &gt; 0</li> </ul>

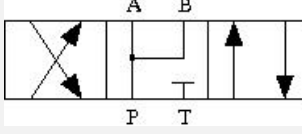
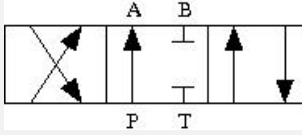
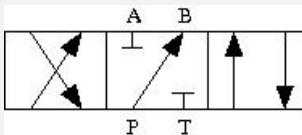
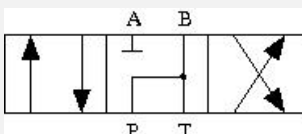
# 4-Way Directional Valve

## Basic 4-Way Directional Valve Configurations (Continued)

No	Configuration	Initial Openings
5		<p>Orifices P-A and A-T are open in neutral position, while orifices P-B and B-T are overlapped:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening &gt; 0</li> <li>• Orifice P-B initial opening &lt; 0</li> <li>• Orifice A-T initial opening &gt; 0</li> <li>• Orifice B-T initial opening &lt; 0</li> </ul>
6		<p>Orifice A-T is initially open, while all three remaining orifices are overlapped:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening &lt; 0</li> <li>• Orifice P-B initial opening &lt; 0</li> <li>• Orifice A-T initial opening &gt; 0</li> <li>• Orifice B-T initial opening &lt; 0</li> </ul>
7		<p>Orifice B-T is initially open, while all three remaining orifices are overlapped:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening &lt; 0</li> <li>• Orifice P-B initial opening &lt; 0</li> <li>• Orifice A-T initial opening &lt; 0</li> <li>• Orifice B-T initial opening &gt; 0</li> </ul>

# 4-Way Directional Valve

## Basic 4-Way Directional Valve Configurations (Continued)

No	Configuration	Initial Openings
8		<p>Orifices P-A and P-B are open, while orifices A-T and B-T are overlapped:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening &gt; 0</li> <li>• Orifice P-B initial opening &gt; 0</li> <li>• Orifice A-T initial opening &lt; 0</li> <li>• Orifice B-T initial opening &lt; 0</li> </ul>
9		<p>Orifice P-A is initially open, while all three remaining orifices are overlapped:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening &gt; 0</li> <li>• Orifice P-B initial opening &lt; 0</li> <li>• Orifice A-T initial opening &lt; 0</li> <li>• Orifice B-T initial opening &lt; 0</li> </ul>
10		<p>Orifice P-B is initially open, while all three remaining orifices are overlapped:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening &lt; 0</li> <li>• Orifice P-B initial opening &gt; 0</li> <li>• Orifice A-T initial opening &lt; 0</li> <li>• Orifice B-T initial opening &lt; 0</li> </ul>
11		<p>Orifices P-B and B-T are open, while orifices P-A and A-T are overlapped:</p> <ul style="list-style-type: none"> <li>• Orifice P-A initial opening &lt; 0</li> <li>• Orifice P-B initial opening &gt; 0</li> </ul>



## Basic 4-Way Directional Valve Configurations (Continued)

No	Configuration	Initial Openings
		<ul style="list-style-type: none"> <li>• Orifice A-T initial opening <math>&lt; 0</math></li> <li>• Orifice B-T initial opening <math>&gt; 0</math></li> </ul>

All four orifices are assumed to be 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

## 4-Way Directional Valve

---

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, that is, the function values must be specified at vertices of a rectangular array. The argument arrays (openings and pressure differentials) must be strictly increasing. The vertices can be nonuniformly spaced. You have a choice of three interpolation methods and two extrapolation methods.

If you need to simulate a nonsymmetrical 4-way valve (that is, with different orifices), use any of the variable orifice blocks from the Orifices 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**

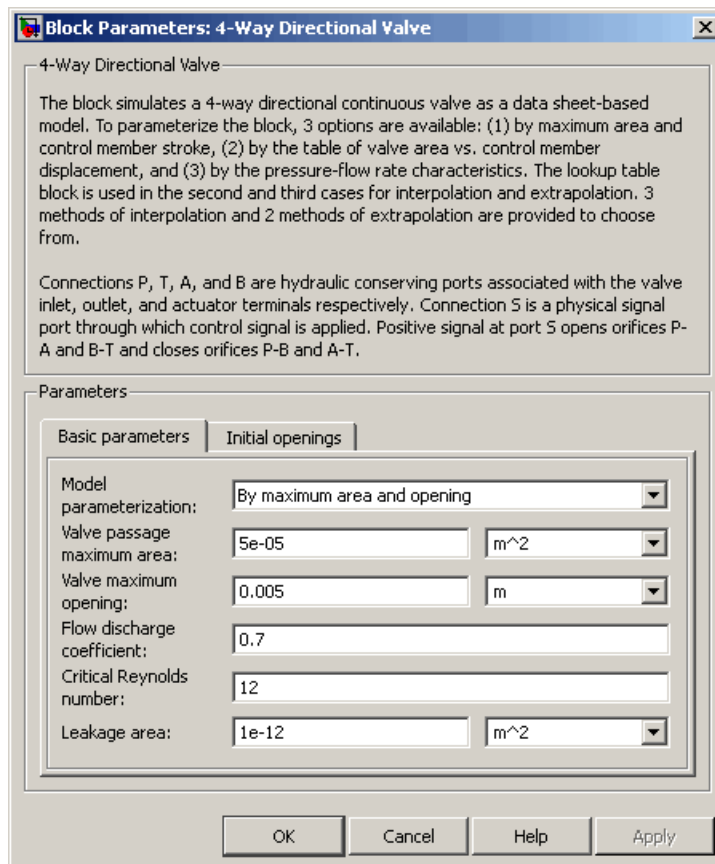
- Fluid inertia is not taken into account.
- Spool loading, such as inertia, spring, hydraulic forces, and so on, is not taken into account.
- Only symmetrical configuration of the valve is considered. In other words, all four orifices are assumed to have the same shape and size.

## Dialog Box and Parameters

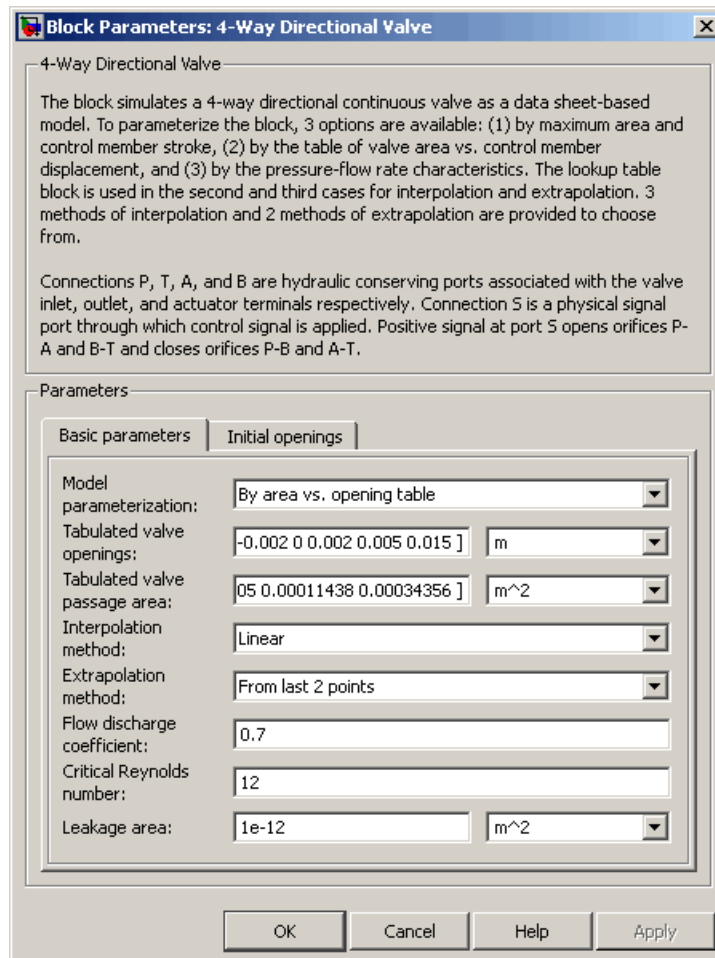
The block dialog box contains two tabs:

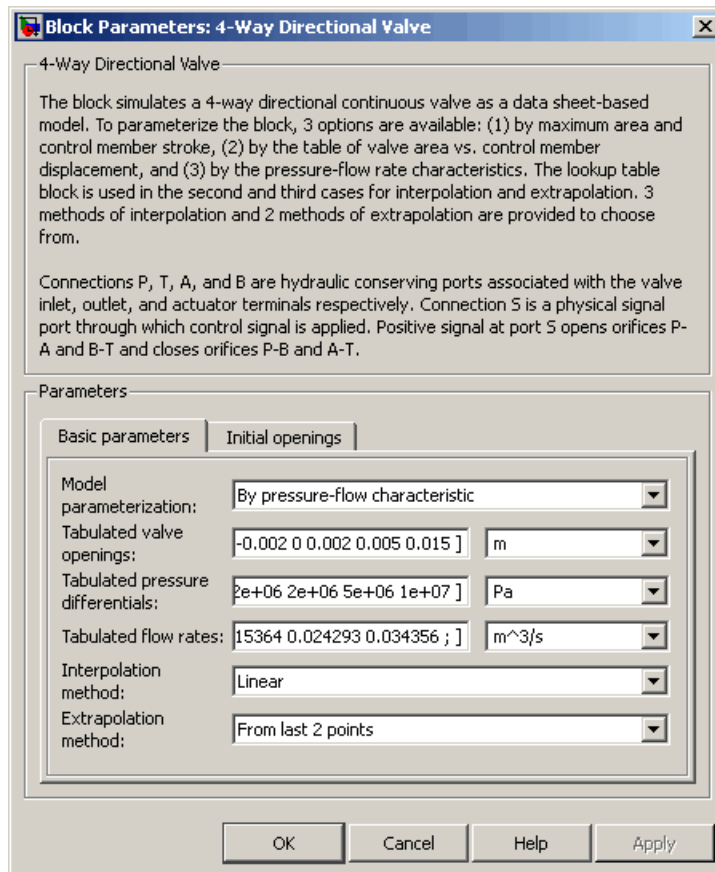
- “Basic Parameters” on page 1-45
- “Initial Openings” on page 1-52

### Basic Parameters



# 4-Way Directional Valve





## Model parameterization

Select one of the following methods for specifying the valve:

- **By maximum area and opening** — Provide values for the maximum valve passage area and the maximum valve opening. The passage area is linearly dependent on the control member displacement, that is, the valve is closed at the initial position of the control member (zero displacement), and the maximum

## 4-Way Directional Valve

---

opening takes place at the maximum displacement. This is the default method.

- **By area vs. opening table** — Provide tabulated data of valve openings and corresponding valve passage areas. The passage area is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.
- **By pressure-flow characteristic** — Provide tabulated data of valve openings, pressure differentials, and corresponding flow rates. The flow rate is determined by two-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.

### **Valve passage maximum area**

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

### **Valve maximum opening**

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

### **Tabulated valve openings**

Specify the vector of input values for valve openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in meters, are  $[-0.002 \ 0 \ 0.002 \ 0.005 \ 0.015]$ . If **Model parameterization** is set to **By area vs. opening table**, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to **By pressure-flow**

characteristic, the **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

## Tabulated valve passage area

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

## Tabulated pressure differentials

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

## Tabulated flow rates

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

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

# 4-Way Directional Valve

---

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

## Interpolation method

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

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

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

## Extrapolation method

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

- **From last 2 points** — Extrapolates using the linear method (regardless of the interpolation method specified), based on the last two output values at the appropriate end of the range. That is, the block uses the first and second specified output



values if the input value is below the specified range, and the two last specified output values if the input value is above the specified range.

- **From last point** — Uses the last specified output value at the appropriate end of the range. That is, the block uses the last specified output value for all input values greater than the last specified input argument, and the first specified output value for all input values less than the first specified input argument.

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**

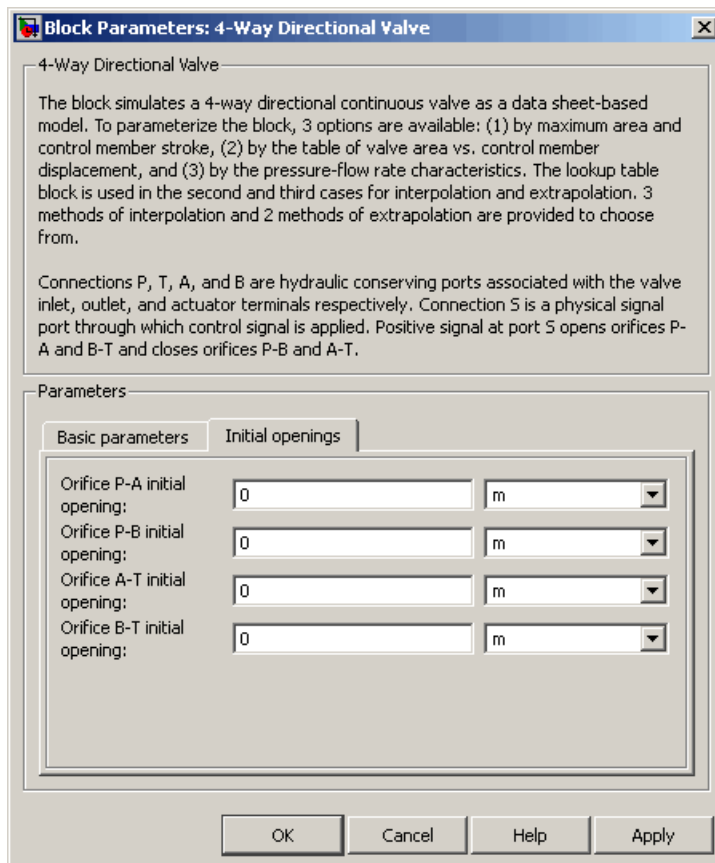
The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic 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 simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

# 4-Way Directional Valve

## Initial Openings



### Orifice P-A initial opening

Initial opening for the Variable Orifice P-A block. 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 Variable Orifice P-B block. 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 Variable Orifice A-T block. 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 Variable Orifice B-T block. 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**

Parameters determined by the type of working fluid:

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

# 4-Way Directional Valve

---

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 example is an open-center, symmetrical valve controlling a double-acting cylinder.

## See Also

2-Way Directional Valve

3-Way Directional Valve

4-Way Directional Valve A

4-Way Directional Valve B

4-Way Directional Valve C

4-Way Directional Valve D

4-Way Directional Valve E

4-Way Directional Valve F

4-Way Directional Valve G

4-Way Directional Valve H

4-Way Directional Valve K

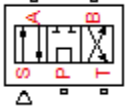
6-Way Directional Valve A

# 4-Way Directional Valve A

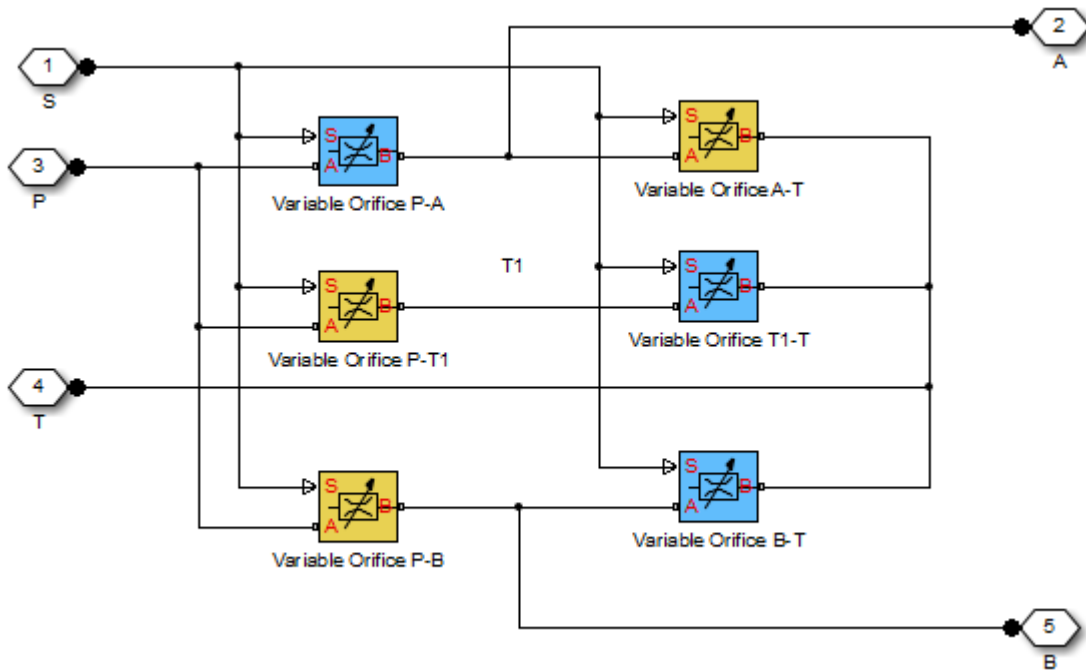
**Purpose** Configuration A of hydraulic continuous 4-way directional valve

**Library** Directional Valves

## Description



The 4-Way Directional Valve A block simulates a configuration of hydraulic continuous 4-way directional valve where the valve unloads the pump in neutral position. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of six Variable Orifice blocks, connected as shown in the following diagram.



# 4-Way Directional Valve A

All blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices P-A, B-T, and T1-T ) and closes the orifices colored yellow (orifices A-T, P-B, and P-T1). As a result, the openings of the orifices are computed as follows:

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

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

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

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

$$h_{PT1} = h_{PT10} - x$$

$$h_{T1T} = h_{T1T0} + x$$

where

$h_{PA}$  Orifice opening for the Variable Orifice P-A block

$h_{PB}$  Orifice opening for the Variable Orifice P-B block

$h_{AT}$  Orifice opening for the Variable Orifice A-T block

$h_{BT}$  Orifice opening for the Variable Orifice B-T block

$h_{PT1}$  Orifice opening for the Variable Orifice P-T1 block

$h_{T1T}$  Orifice opening for the Variable Orifice T1-T block

$h_{PA0}$  Initial opening for the Variable Orifice P-A block

$h_{PB0}$  Initial opening for the Variable Orifice P-B block

$h_{AT0}$  Initial opening for the Variable Orifice A-T block

# 4-Way Directional Valve A

---

$h_{BTO}$	Initial opening for the Variable Orifice B-T block
$h_{PT10}$	Initial opening for the Variable Orifice P-T1 block
$h_{T1T0}$	Initial opening for the Variable Orifice T1-T block
$x$	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

## Dialog Box and Parameters

The block dialog box contains two tabs:

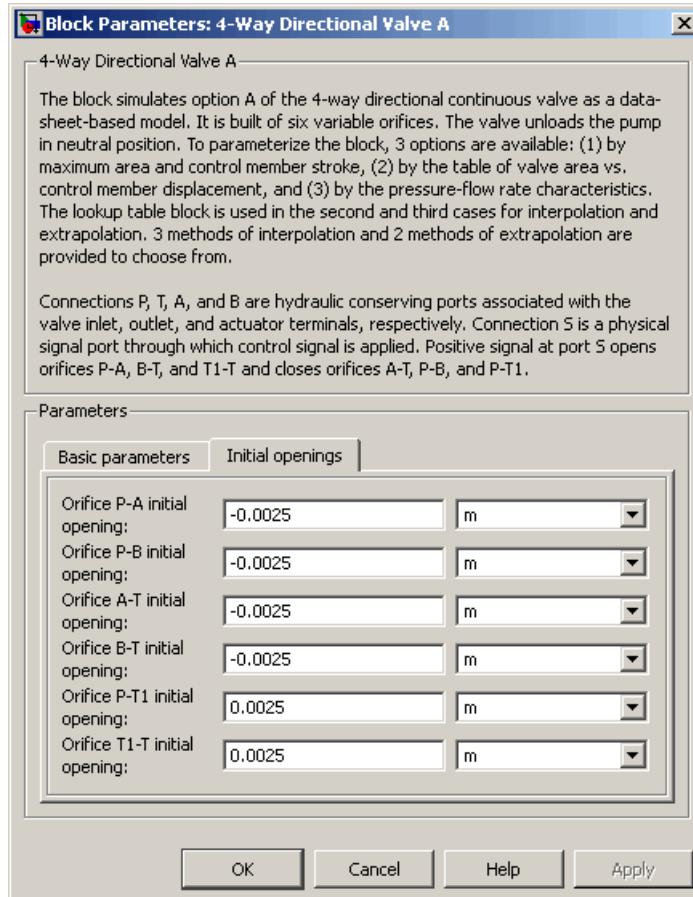
- “Basic Parameters” on page 1-58
- “Initial Openings” on page 1-59

### Basic Parameters

See the documentation for the Basic Parameters tab of the 4-Way Directional Valve block for parameter descriptions and default values.



## Initial Openings



### Orifice P-A initial opening

Initial opening for the Variable Orifice P-A block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is -0.0025 m.

# 4-Way Directional Valve A

---

## **Orifice P-B initial opening**

Initial opening for the Variable Orifice P-B block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is -0.0025 m.

## **Orifice A-T initial opening**

Initial opening for the Variable Orifice A-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is -0.0025 m.

## **Orifice B-T initial opening**

Initial opening for the Variable Orifice B-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is -0.0025 m.

## **Orifice P-T1 initial opening**

Initial opening for the Variable Orifice P-T1 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is 0.0025 m.

## **Orifice T1-T initial opening**

Initial opening for the Variable Orifice T1-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is 0.0025 m.

## **Ports**

The block has the following ports:

P

Hydraulic conserving port associated with the pressure supply line inlet.

T

Hydraulic conserving port associated with the return line connection.

- A Hydraulic conserving port associated with the actuator connection port.
- B Hydraulic conserving port associated with the actuator connection port.
- S Physical signal port to control spool displacement.

## **See Also**

- 4-Way Directional Valve
- 4-Way Directional Valve B
- 4-Way Directional Valve C
- 4-Way Directional Valve D
- 4-Way Directional Valve E
- 4-Way Directional Valve F
- 4-Way Directional Valve G
- 4-Way Directional Valve H
- 4-Way Directional Valve K
- 6-Way Directional Valve A

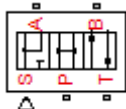
# 4-Way Directional Valve B

---

**Purpose** Configuration B of hydraulic continuous 4-way directional valve

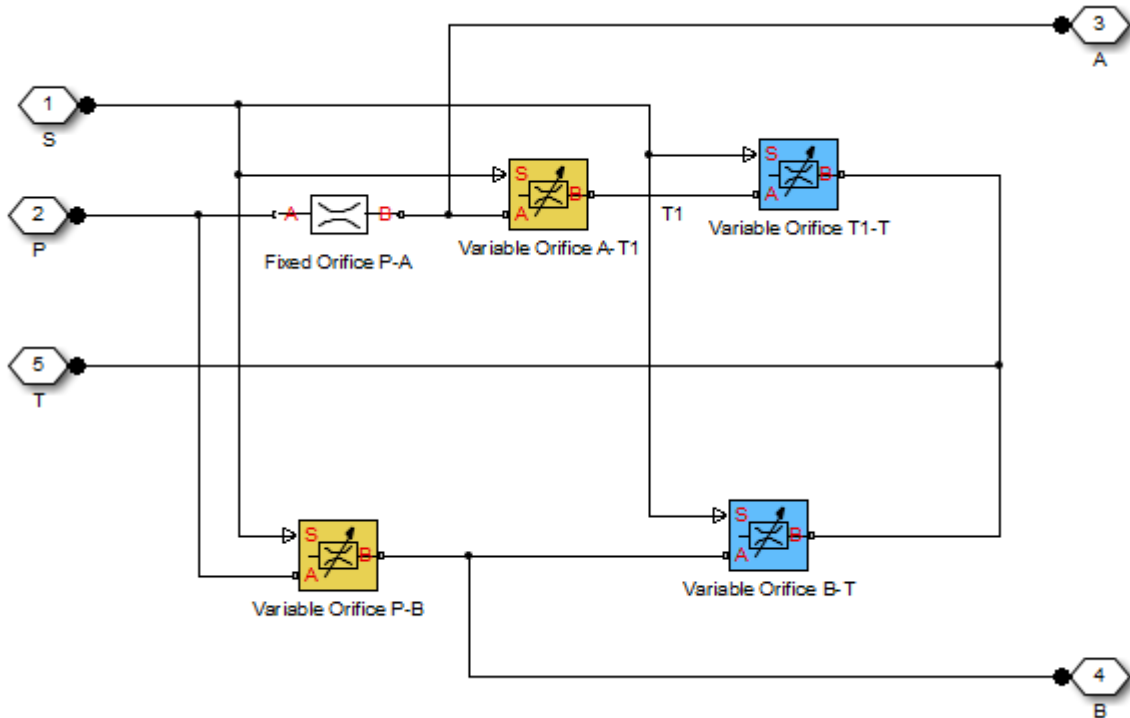
**Library** Directional Valves

**Description**



The 4-Way Directional Valve B block simulates a configuration of hydraulic continuous 4-way directional valve where port A is permanently connected to port P, and all four ports are interconnected in neutral position. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of a Fixed Orifice block in the P-A path and four Variable Orifice blocks, connected as shown in the following diagram.

# 4-Way Directional Valve B



All Variable Orifice blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices A-T1 and P-B) and closes the orifices colored yellow (orifices T1-T and B-T). As a result, the openings of the orifices are computed as follows:

$$h_{AT1} = h_{AT10} - x$$

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

# 4-Way Directional Valve B

---

$$h_{T1T} = h_{T1T0} + x$$

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

where

$h_{AT1}$	Orifice opening for the Variable Orifice A-T1 block
$h_{PB}$	Orifice opening for the Variable Orifice P-B block
$h_{T1T}$	Orifice opening for the Variable Orifice T1-T block
$h_{BT}$	Orifice opening for the Variable Orifice B-T block
$h_{AT10}$	Initial opening for the Variable Orifice A-T1 block
$h_{PBO}$	Initial opening for the Variable Orifice P-B block
$h_{T1T0}$	Initial opening for the Variable Orifice T1-T block
$h_{BT0}$	Initial opening for the Variable Orifice B-T block
$x$	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

## Dialog Box and Parameters

The block dialog box contains two tabs:

- “Basic Parameters” on page 1-64
- “Initial Openings” on page 1-66

### Basic Parameters

See the documentation for the Basic Parameters tab of the 4-Way Directional Valve block for parameter descriptions and default values.

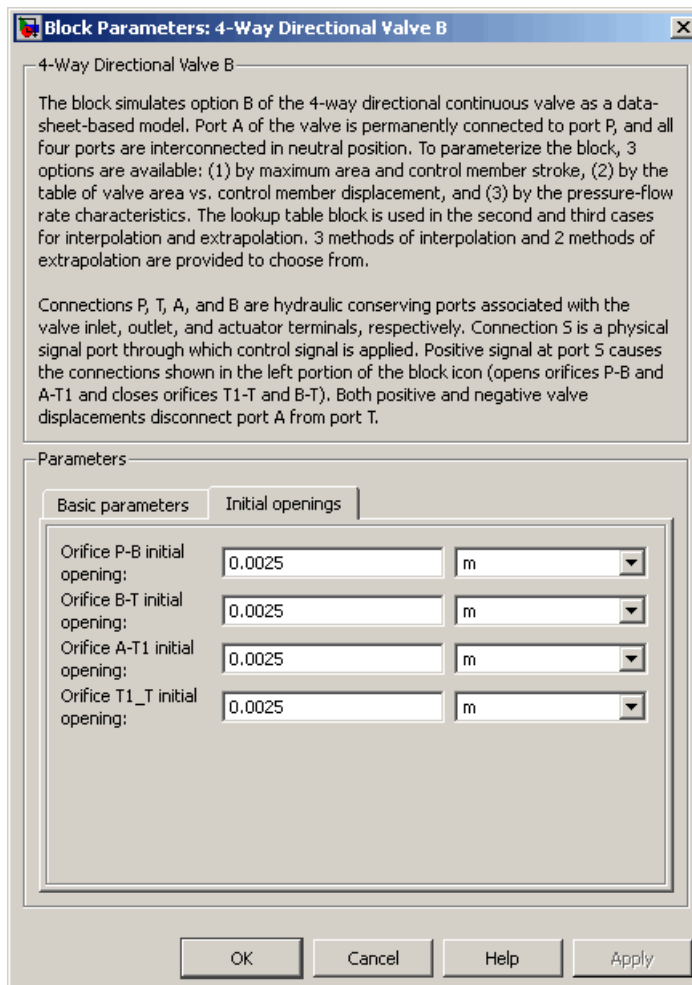
There is one additional parameter:

## **Path P-A passage area**

Specify the area of the P-A passage. The parameter value must be greater than zero. The default value is  $5e-5 \text{ m}^2$ . This parameter is used if **Model parameterization** is set to By maximum area and opening.

# 4-Way Directional Valve B

## Initial Openings



### Orifice P-B initial opening

Initial opening for the Variable Orifice P-B block. The parameter can be positive (underlapped orifice), negative



(overlapped orifice), or equal to zero for zero lap configuration.  
The default value is 0.0025 m.

### **Orifice B-T initial opening**

Initial opening for the Variable Orifice B-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is 0.0025 m.

### **Orifice A-T1 initial opening**

Initial opening for the Variable Orifice A-T1 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is 0.0025 m.

### **Orifice T1-T initial opening**

Initial opening for the Variable Orifice T1-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is 0.0025 m.

## **Ports**

The block has the following ports:

P

Hydraulic conserving port associated with the pressure supply line inlet.

T

Hydraulic conserving port associated with the return line connection.

A

Hydraulic conserving port associated with the actuator connection port.

B

Hydraulic conserving port associated with the actuator connection port.

# 4-Way Directional Valve B

---

S

Physical signal port to control spool displacement.

## **See Also**

4-Way Directional Valve  
4-Way Directional Valve A  
4-Way Directional Valve C  
4-Way Directional Valve D  
4-Way Directional Valve E  
4-Way Directional Valve F  
4-Way Directional Valve G  
4-Way Directional Valve H  
4-Way Directional Valve K  
6-Way Directional Valve A

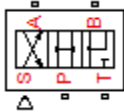
## Purpose

Configuration C of hydraulic continuous 4-way directional valve

## Library

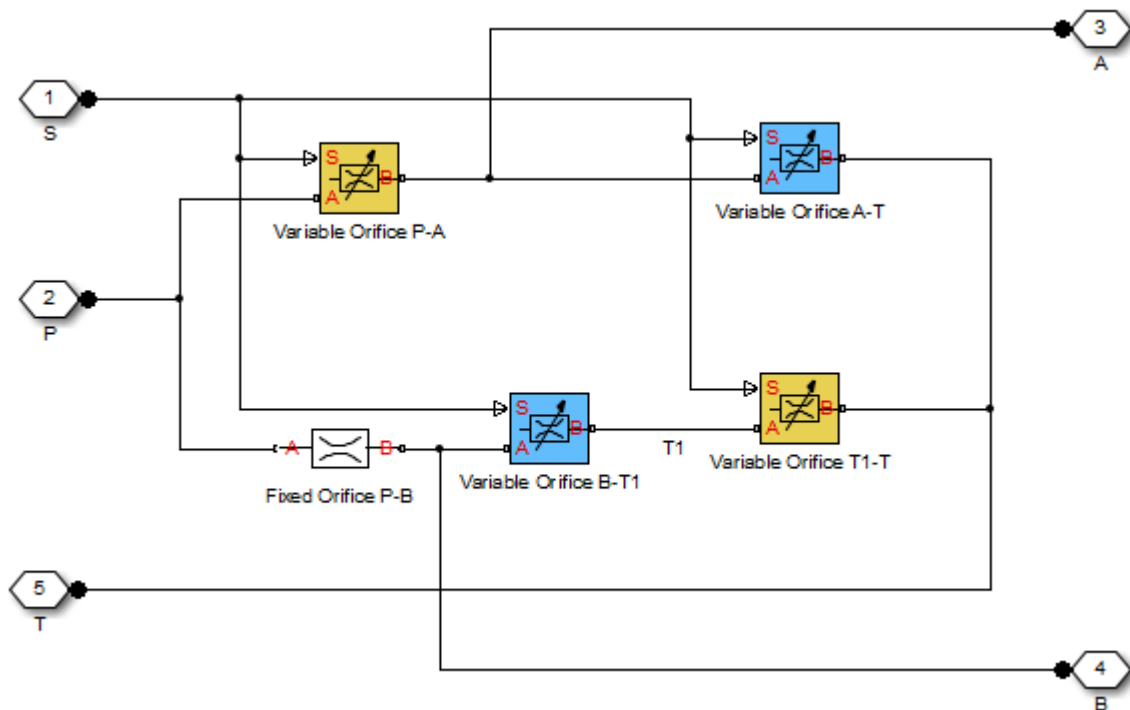
Directional Valves

## Description



The 4-Way Directional Valve C block simulates a configuration of hydraulic continuous 4-way directional valve where port P is connected to port A and port B is connected to port T in the left position; in the right position, both port A and port B are connected to P; and all ports are interconnected in neutral position. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of a Fixed Orifice block in the P-B path and four Variable Orifice blocks, connected as shown in the following diagram.

# 4-Way Directional Valve C



All Variable Orifice blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices B-T1 and A-T ) and closes the orifices colored yellow (orifices P-A and T1-T). As a result, the openings of the orifices are computed as follows:

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

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

$$h_{BT1} = h_{BT10} + x$$

$$h_{T1T} = h_{T1T0} - x$$

where

$h_{PA}$	Orifice opening for the Variable Orifice P-A block
$h_{AT}$	Orifice opening for the Variable Orifice A-T block
$h_{BT1}$	Orifice opening for the Variable Orifice B-T1 block
$h_{T1T}$	Orifice opening for the Variable Orifice T1-T block
$h_{PA0}$	Initial opening for the Variable Orifice P-A block
$h_{PB0}$	Initial opening for the Variable Orifice P-B block
$h_{BT10}$	Initial opening for the Variable Orifice B-T1 block
$h_{T1T0}$	Initial opening for the Variable Orifice T1-T block
$x$	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

## Dialog Box and Parameters

The block dialog box contains two tabs:

- “Basic Parameters” on page 1-71
- “Initial Openings” on page 1-73

### Basic Parameters

See the documentation for the Basic Parameters tab of the 4-Way Directional Valve block for parameter descriptions and default values.

There is one additional parameter:

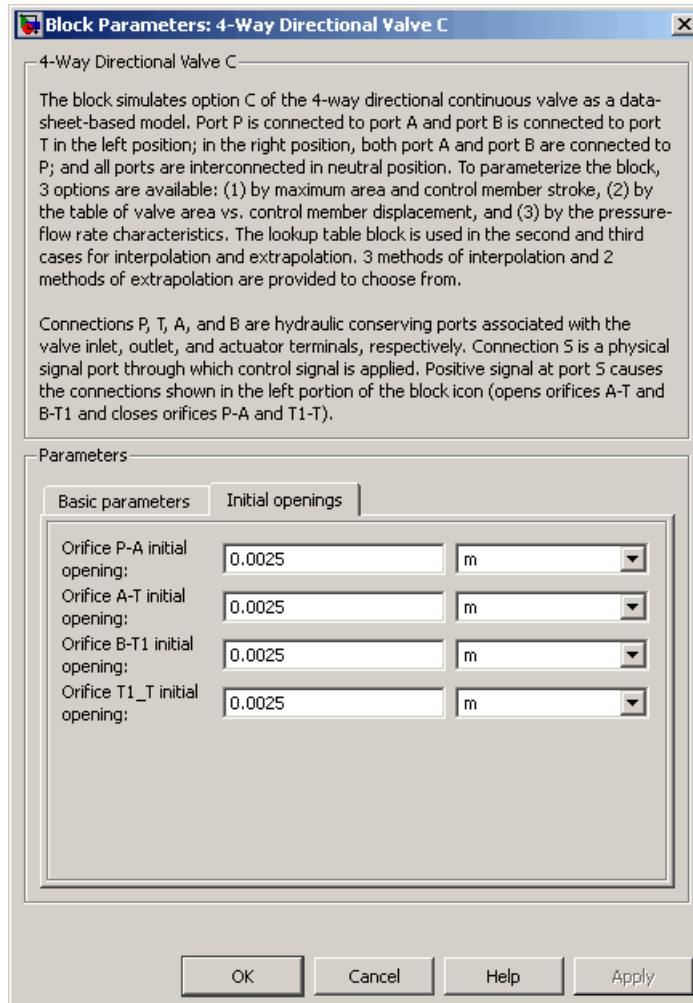
# 4-Way Directional Valve C

---

## **Path P-B passage area**

Specify the area of the P-B passage. The parameter value must be greater than zero. The default value is  $5e-5 \text{ m}^2$ . This parameter is used if **Model parameterization** is set to By maximum area and opening.

## Initial Openings



# 4-Way Directional Valve C

---

## **Orifice P-A initial opening**

Initial opening for the Variable Orifice P-A block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is 0.0025 m.

## **Orifice A-T initial opening**

Initial opening for the Variable Orifice A-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is 0.0025 m.

## **Orifice B-T1 initial opening**

Initial opening for the Variable Orifice B-T1 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is 0.0025 m.

## **Orifice T1-T initial opening**

Initial opening for the Variable Orifice T1-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is 0.0025 m.

## **Ports**

The block has the following ports:

P

Hydraulic conserving port associated with the pressure supply line inlet.

T

Hydraulic conserving port associated with the return line connection.

A

Hydraulic conserving port associated with the actuator connection port.



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

## **See Also**

- 4-Way Directional Valve
- 4-Way Directional Valve A
- 4-Way Directional Valve B
- 4-Way Directional Valve D
- 4-Way Directional Valve E
- 4-Way Directional Valve F
- 4-Way Directional Valve G
- 4-Way Directional Valve H
- 4-Way Directional Valve K
- 6-Way Directional Valve A

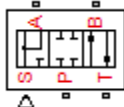
# 4-Way Directional Valve D

---

**Purpose** Configuration D of hydraulic continuous 4-way directional valve

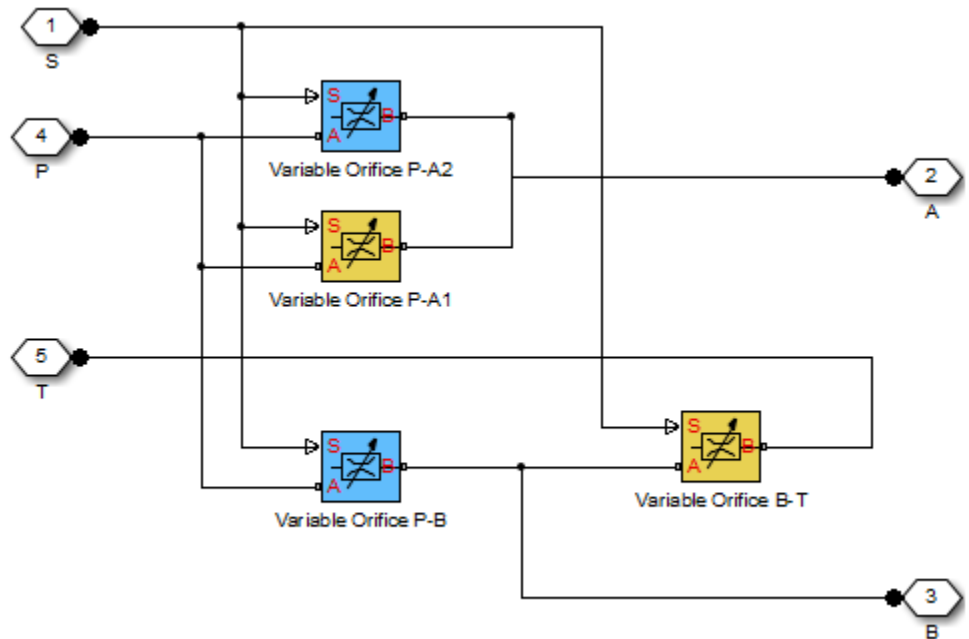
**Library** Directional Valves

## Description



The 4-Way Directional Valve D block simulates a configuration of hydraulic continuous 4-way directional valve. Ports A and B are connected to port P in the left valve position. In the right position, port P is connected to port A, while port B is connected to port T. All connections are blocked in neutral position. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of four Variable Orifice blocks, connected as shown in the following diagram.

# 4-Way Directional Valve D



All Variable Orifice blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices P-A2 and P-B) and closes the orifices colored yellow (orifices P-A1 and B-T). As a result, the openings of the orifices are computed as follows:

$$h_{PA1} = h_{PA10} - x$$

$$h_{PA2} = h_{PA20} + x$$

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

# 4-Way Directional Valve D

---

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

where

$h_{PA1}$	Orifice opening for the Variable Orifice P-A1 block
$h_{PA2}$	Orifice opening for the Variable Orifice P-A2 block
$h_{PB}$	Orifice opening for the Variable Orifice P-B block
$h_{BT}$	Orifice opening for the Variable Orifice B-T block
$h_{PA10}$	Initial opening for the Variable Orifice P-A1 block
$h_{PA20}$	Initial opening for the Variable Orifice P-A2 block
$h_{PB0}$	Initial opening for the Variable Orifice P-B block
$h_{BT0}$	Initial opening for the Variable Orifice B-T block
$x$	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

## Dialog Box and Parameters

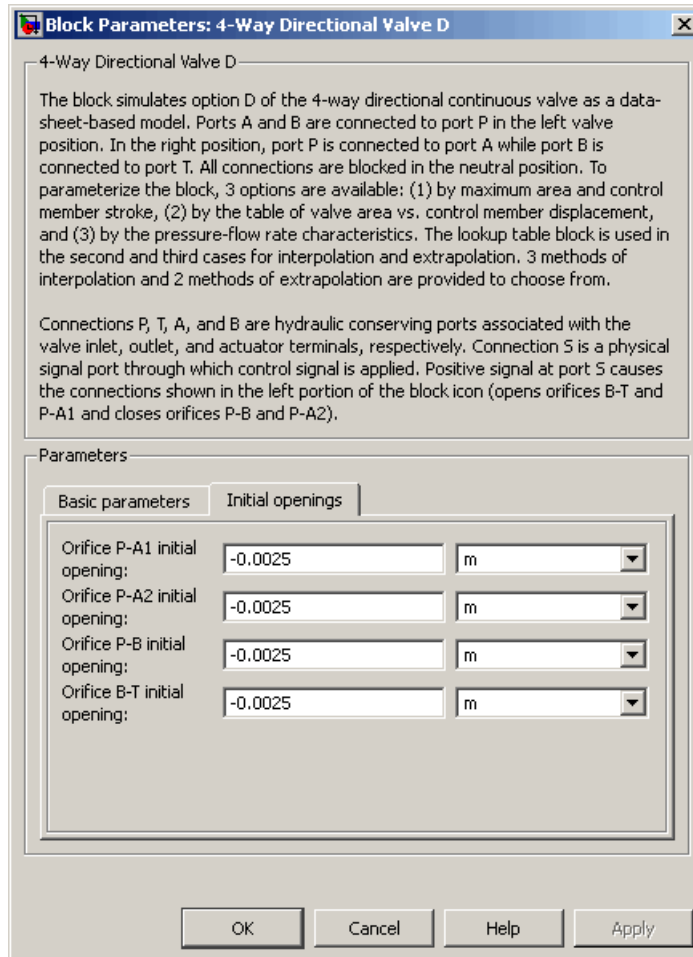
The block dialog box contains two tabs:

- “Basic Parameters” on page 1-78
- “Initial Openings” on page 1-79

### Basic Parameters

See the documentation for the Basic Parameters tab of the 4-Way Directional Valve block for parameter descriptions and default values.

## Initial Openings



### Orifice P-A1 initial opening

Initial opening for the Variable Orifice P-A1 block. The parameter can be positive (underlapped orifice), negative

# 4-Way Directional Valve D

---

(overlapped orifice), or equal to zero for zero lap configuration. The default value is -0.0025 m.

## **Orifice P-A2 initial opening**

Initial opening for the Variable Orifice P-A2 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is -0.0025 m.

## **Orifice P-B initial opening**

Initial opening for the Variable Orifice P-B block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is -0.0025 m.

## **Orifice B-T initial opening**

Initial opening for the Variable Orifice B-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is -0.0025 m.

## **Ports**

The block has the following ports:

- P  
Hydraulic conserving port associated with the pressure supply line inlet.
- T  
Hydraulic conserving port associated with the return line connection.
- A  
Hydraulic conserving port associated with the actuator connection port.
- B  
Hydraulic conserving port associated with the actuator connection port.

S

Physical signal port to control spool displacement.

## **See Also**

4-Way Directional Valve

4-Way Directional Valve A

4-Way Directional Valve B

4-Way Directional Valve C

4-Way Directional Valve E

4-Way Directional Valve F

4-Way Directional Valve G

4-Way Directional Valve H

4-Way Directional Valve K

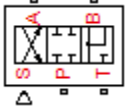
6-Way Directional Valve A

# 4-Way Directional Valve E

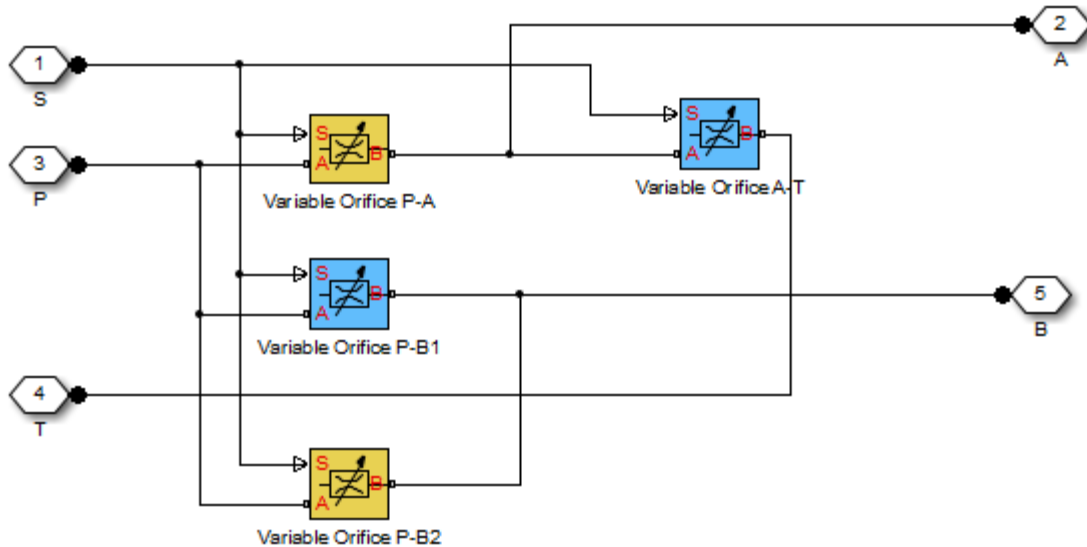
**Purpose** Configuration E of hydraulic continuous 4-way directional valve

**Library** Directional Valves

## Description



The 4-Way Directional Valve E block simulates a configuration of hydraulic continuous 4-way directional valve. Ports A and B are connected to port P in the right valve position. In the left position, port P is connected to port B, while port A is connected to port T. All connections are blocked in neutral position. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of four Variable Orifice blocks, connected as shown in the following diagram.





All Variable Orifice blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices P-B1 and A-T ) and closes the orifices colored yellow (orifices P-A and P-B2). As a result, the openings of the orifices are computed as follows:

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

$$h_{PB1} = h_{PB10} + x$$

$$h_{PB2} = h_{PB20} - x$$

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

where

$h_{PA}$	Orifice opening for the Variable Orifice P-A block
$h_{PB1}$	Orifice opening for the Variable Orifice P-B1 block
$h_{PB2}$	Orifice opening for the Variable Orifice P-B2 block
$h_{AT}$	Orifice opening for the Variable Orifice A-T block
$h_{PA0}$	Initial opening for the Variable Orifice P-A block
$h_{PB10}$	Initial opening for the Variable Orifice P-B1 block
$h_{PB20}$	Initial opening for the Variable Orifice P-B2 block
$h_{AT0}$	Initial opening for the Variable Orifice A-T block
$x$	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

# 4-Way Directional Valve E

---

## **Dialog Box and Parameters**

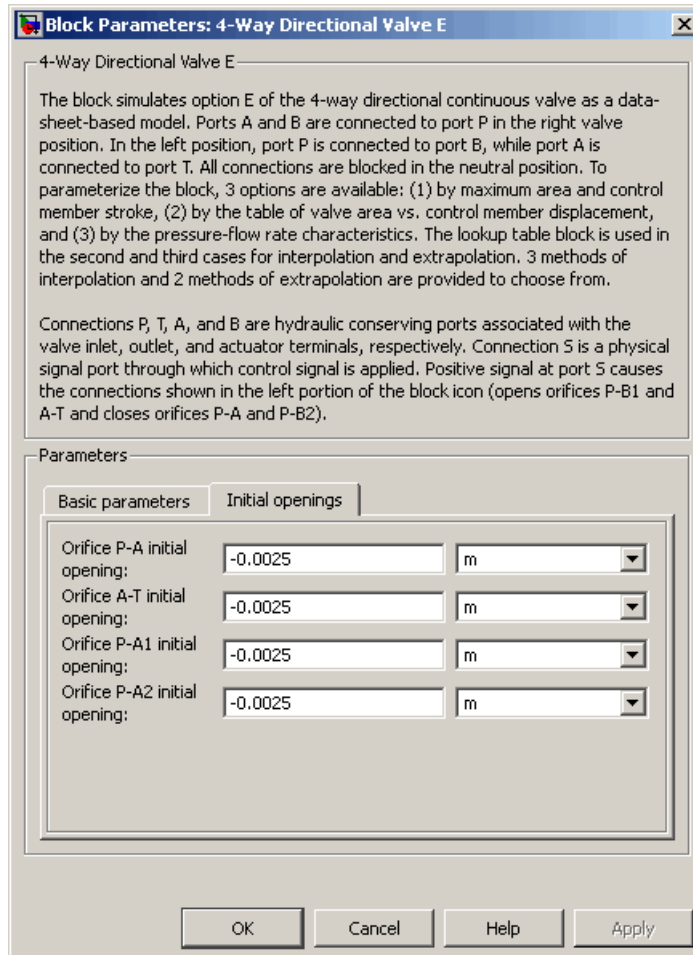
The block dialog box contains two tabs:

- “Basic Parameters” on page 1-84
- “Initial Openings” on page 1-85

### **Basic Parameters**

See the documentation for the Basic Parameters tab of the 4-Way Directional Valve block for parameter descriptions and default values.

## Initial Openings



### Orifice P-A initial opening

Initial opening for the Variable Orifice P-A block. The parameter can be positive (underlapped orifice), negative

## 4-Way Directional Valve E

---

(overlapped orifice), or equal to zero for zero lap configuration. The default value is -0.0025 m.

### **Orifice P-B1 initial opening**

Initial opening for the Variable Orifice P-B1 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is -0.0025 m.

### **Orifice P-B2 initial opening**

Initial opening for the Variable Orifice P-B2 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is -0.0025 m.

### **Orifice A-T initial opening**

Initial opening for the Variable Orifice A-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is -0.0025 m.

## **Ports**

The block has the following ports:

- P  
Hydraulic conserving port associated with the pressure supply line inlet.
- T  
Hydraulic conserving port associated with the return line connection.
- A  
Hydraulic conserving port associated with the actuator connection port.
- B  
Hydraulic conserving port associated with the actuator connection port.

S

Physical signal port to control spool displacement.

**See Also**

4-Way Directional Valve

4-Way Directional Valve A

4-Way Directional Valve B

4-Way Directional Valve C

4-Way Directional Valve D

4-Way Directional Valve F

4-Way Directional Valve G

4-Way Directional Valve H

4-Way Directional Valve K

6-Way Directional Valve A

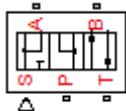
# 4-Way Directional Valve F

---

**Purpose** Configuration F of hydraulic continuous 4-way directional valve

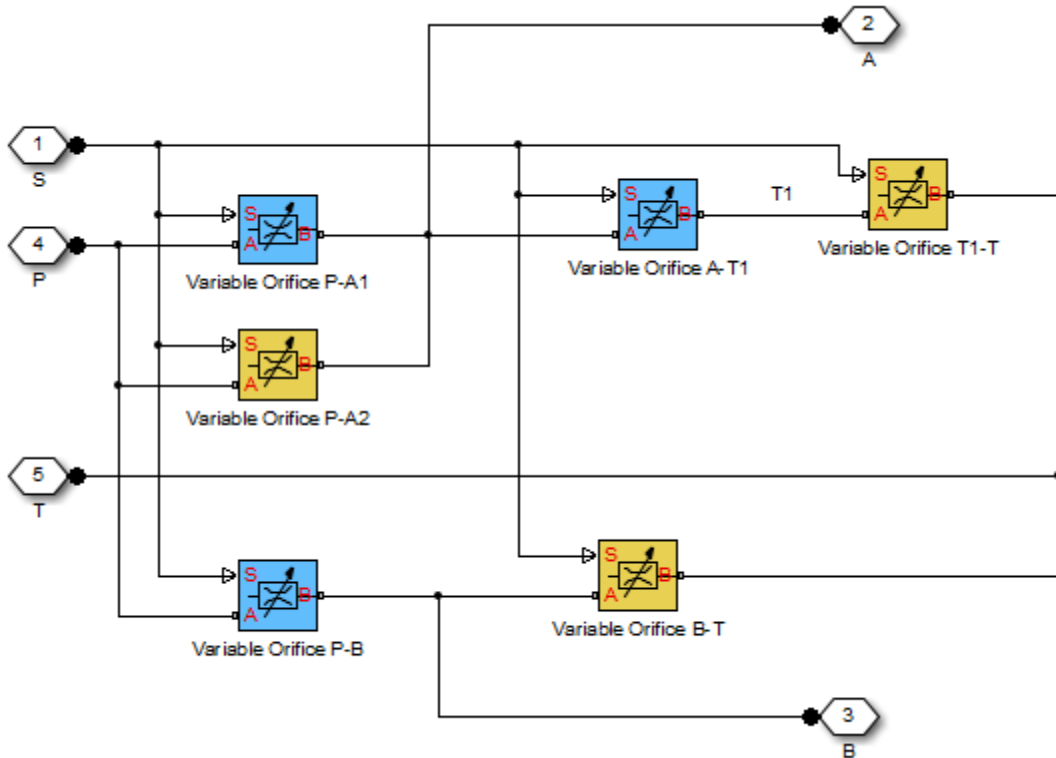
**Library** Directional Valves

**Description**



The 4-Way Directional Valve F block simulates a configuration of hydraulic continuous 4-way directional valve. Ports A and B are connected to port P in the left valve position. In the right position, port P is connected to port A, while port B is connected to port T. In neutral position, ports A and B are connected to port T. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of six Variable Orifice blocks, connected as shown in the following diagram.

# 4-Way Directional Valve F



All Variable Orifice blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices P-A1, A-T1, and P-B) and closes the orifices colored yellow (orifices P-A2, B-T, and T1-T). As a result, the openings of the orifices are computed as follows:

$$h_{PA1} = h_{PA10} + x$$

$$h_{PA2} = h_{PA20} - x$$

## 4-Way Directional Valve F

---

$$h_{AT1} = h_{AT10} + x$$

$$h_{T1T} = h_{T1T0} - x$$

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

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

where

$h_{PA1}$	Orifice opening for the Variable Orifice P-A1 block
$h_{PA2}$	Orifice opening for the Variable Orifice P-A2 block
$h_{AT1}$	Orifice opening for the Variable Orifice A-T1 block
$h_{T1T}$	Orifice opening for the Variable Orifice T1-T block
$h_{PB}$	Orifice opening for the Variable Orifice P-B block
$h_{BT}$	Orifice opening for the Variable Orifice B-T block
$h_{PA10}$	Initial opening for the Variable Orifice P-A1 block
$h_{PA20}$	Initial opening for the Variable Orifice P-A2 block
$h_{AT10}$	Initial opening for the Variable Orifice A-T1 block
$h_{T1T0}$	Initial opening for the Variable Orifice T1-T block
$h_{PB0}$	Initial opening for the Variable Orifice P-B block
$h_{BT0}$	Initial opening for the Variable Orifice B-T block
$x$	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.



## **Dialog Box and Parameters**

The block dialog box contains two tabs:

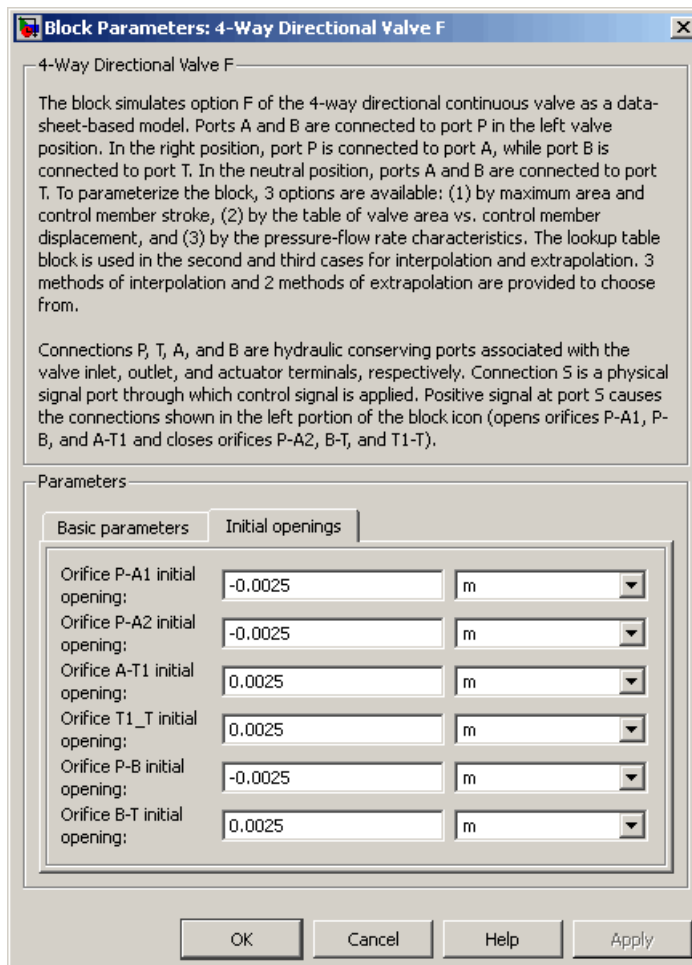
- “Basic Parameters” on page 1-91
- “Initial Openings” on page 1-92

### **Basic Parameters**

See the documentation for the Basic Parameters tab of the 4-Way Directional Valve block for parameter descriptions and default values.

# 4-Way Directional Valve F

## Initial Openings



### Orifice P-A1 initial opening

Initial opening for the Variable Orifice P-A1 block. The parameter can be positive (underlapped orifice), negative

(overlapped orifice), or equal to zero for zero lap configuration.  
The default value is -0.0025 m.

### **Orifice P-A2 initial opening**

Initial opening for the Variable Orifice P-A2 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is -0.0025 m.

### **Orifice A-T1 initial opening**

Initial opening for the Variable Orifice A-T1 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is 0.0025 m.

### **Orifice T1-T initial opening**

Initial opening for the Variable Orifice T1-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is 0.0025 m.

### **Orifice P-B initial opening**

Initial opening for the Variable Orifice P-B block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is -0.0025 m.

### **Orifice B-T initial opening**

Initial opening for the Variable Orifice B-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is 0.0025 m.

## **Ports**

The block has the following ports:

P

Hydraulic conserving port associated with the pressure supply line inlet.

# 4-Way Directional Valve F

---

- T  
Hydraulic conserving port associated with the return line connection.
- A  
Hydraulic conserving port associated with the actuator connection port.
- B  
Hydraulic conserving port associated with the actuator connection port.
- S  
Physical signal port to control spool displacement.

## **See Also**

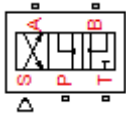
- 4-Way Directional Valve
- 4-Way Directional Valve A
- 4-Way Directional Valve B
- 4-Way Directional Valve C
- 4-Way Directional Valve D
- 4-Way Directional Valve E
- 4-Way Directional Valve G
- 4-Way Directional Valve H
- 4-Way Directional Valve K
- 6-Way Directional Valve A

# 4-Way Directional Valve G

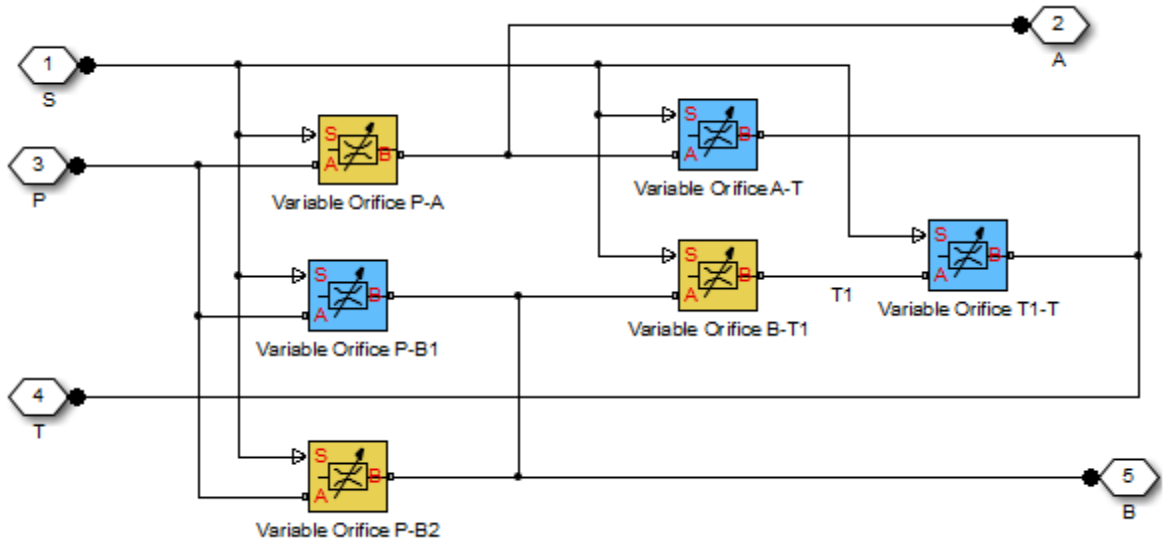
**Purpose** Configuration G of hydraulic continuous 4-way directional valve

**Library** Directional Valves

## Description



The 4-Way Directional Valve G block simulates a configuration of hydraulic continuous 4-way directional valve. Ports A and B are connected to port P in the right valve position. In the left position, port P is connected to port B, while port A is connected to port T. In neutral position, ports A and B are connected to port T. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of six Variable Orifice blocks, connected as shown in the following diagram.



## 4-Way Directional Valve G

---

All Variable Orifice blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices P-B1, A-T, and T1-T) and closes the orifices colored yellow (orifices P-A, B-T1, and P-B2). As a result, the openings of the orifices are computed as follows:

$$h_{PB1} = h_{PB10} + x$$

$$h_{PB2} = h_{PB20} - x$$

$$h_{BT1} = h_{BT10} - x$$

$$h_{T1T} = h_{T1T0} + x$$

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

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

where

$h_{PB1}$	Orifice opening for the Variable Orifice P-B1 block
$h_{PB2}$	Orifice opening for the Variable Orifice P-B2 block
$h_{BT1}$	Orifice opening for the Variable Orifice B-T1 block
$h_{T1T}$	Orifice opening for the Variable Orifice T1-T block
$h_{PA}$	Orifice opening for the Variable Orifice P-A block
$h_{AT}$	Orifice opening for the Variable Orifice A-T block
$h_{PB10}$	Initial opening for the Variable Orifice P-B1 block
$h_{PB20}$	Initial opening for the Variable Orifice P-B2 block
$h_{BT10}$	Initial opening for the Variable Orifice B-T1 block

$h_{T1T0}$	Initial opening for the Variable Orifice T1-T block
$h_{PA0}$	Initial opening for the Variable Orifice P-A block
$h_{AT0}$	Initial opening for the Variable Orifice A-T block
$x$	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

## **Dialog Box and Parameters**

The block dialog box contains two tabs:

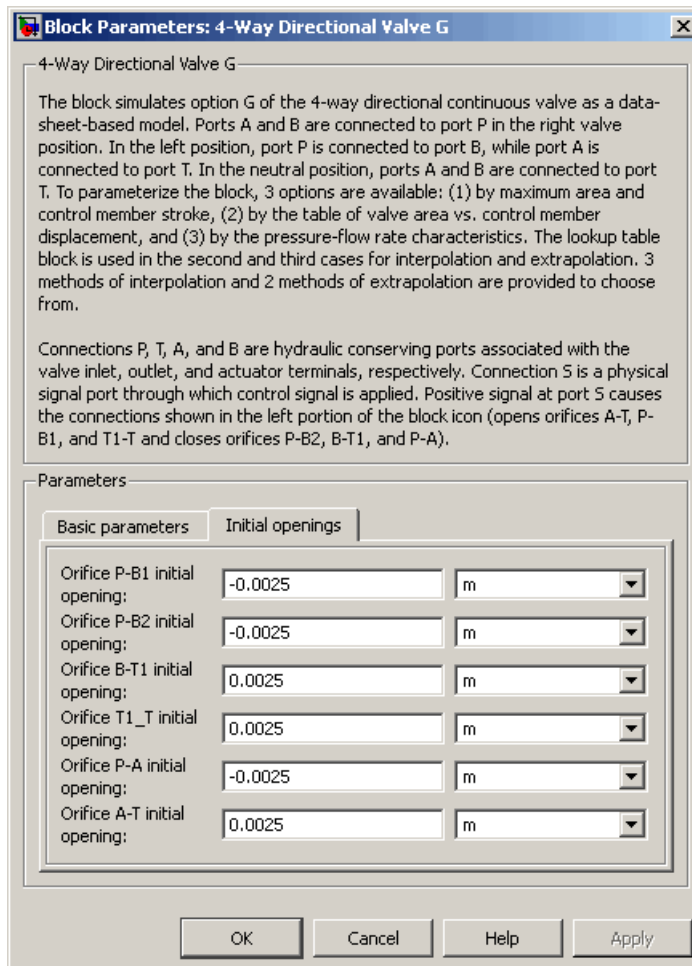
- “Basic Parameters” on page 1-97
- “Initial Openings” on page 1-98

### **Basic Parameters**

See the documentation for the Basic Parameters tab of the 4-Way Directional Valve block for parameter descriptions and default values.

# 4-Way Directional Valve G

## Initial Openings



### Orifice P-B1 initial opening

Initial opening for the Variable Orifice P-B1 block. The parameter can be positive (underlapped orifice), negative



(overlapped orifice), or equal to zero for zero lap configuration.  
The default value is -0.0025 m.

### **Orifice P-B2 initial opening**

Initial opening for the Variable Orifice P-B2 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is -0.0025 m.

### **Orifice B-T1 initial opening**

Initial opening for the Variable Orifice B-T1 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is 0.0025 m.

### **Orifice T1-T initial opening**

Initial opening for the Variable Orifice T1-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is 0.0025 m.

### **Orifice P-A initial opening**

Initial opening for the Variable Orifice P-A block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is -0.0025 m.

### **Orifice A-T initial opening**

Initial opening for the Variable Orifice A-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is 0.0025 m.

## **Ports**

The block has the following ports:

P

Hydraulic conserving port associated with the pressure supply line inlet.

# 4-Way Directional Valve G

---

- T Hydraulic conserving port associated with the return line connection.
- A Hydraulic conserving port associated with the actuator connection port.
- B Hydraulic conserving port associated with the actuator connection port.
- S Physical signal port to control spool displacement.

## **See Also**

- 4-Way Directional Valve
- 4-Way Directional Valve A
- 4-Way Directional Valve B
- 4-Way Directional Valve C
- 4-Way Directional Valve D
- 4-Way Directional Valve E
- 4-Way Directional Valve F
- 4-Way Directional Valve H
- 4-Way Directional Valve K
- 6-Way Directional Valve A

# 4-Way Directional Valve H

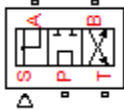
## Purpose

Configuration H of hydraulic continuous 4-way directional valve

## Library

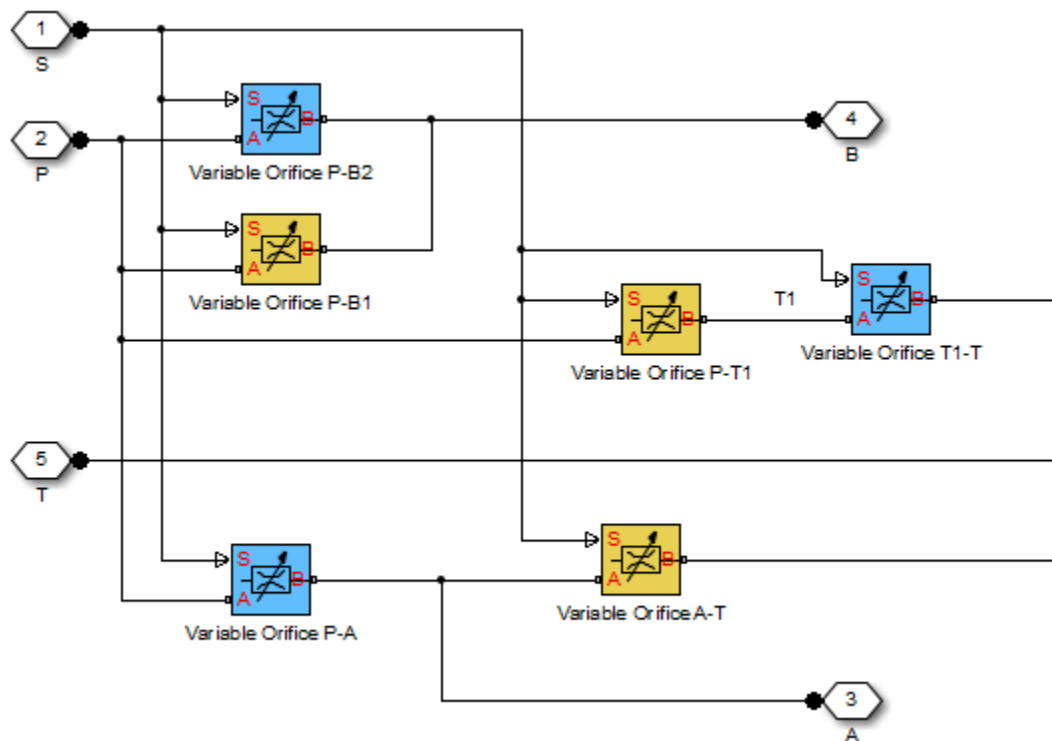
Directional Valves

## Description



The 4-Way Directional Valve H block simulates a configuration of hydraulic continuous 4-way directional valve. Ports A and B are connected to port P in the left valve position. In the right position, port P is connected to port B, while port A is connected to port T. In neutral position, port P is connected to port T to unload the pump. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of six Variable Orifice blocks, connected as shown in the following diagram.

# 4-Way Directional Valve H



All Variable Orifice blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices P-B2, P-A, and T1-T) and closes the orifices colored yellow (orifices P-B1, P-T1, and A-T). As a result, the openings of the orifices are computed as follows:

$$h_{PB1} = h_{PB10} - x$$

$$h_{PB2} = h_{PB20} + x$$

$$h_{PT1} = h_{PT10} - x$$

$$h_{T1T} = h_{T1T0} + x$$

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

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

where

$h_{PB1}$	Orifice opening for the Variable Orifice P-B1 block
$h_{PB2}$	Orifice opening for the Variable Orifice P-B2 block
$h_{PT1}$	Orifice opening for the Variable Orifice P-T1 block
$h_{T1T}$	Orifice opening for the Variable Orifice T1-T block
$h_{PA}$	Orifice opening for the Variable Orifice P-A block
$h_{AT}$	Orifice opening for the Variable Orifice A-T block
$h_{PB10}$	Initial opening for the Variable Orifice P-B1 block
$h_{PB20}$	Initial opening for the Variable Orifice P-B2 block
$h_{PT10}$	Initial opening for the Variable Orifice P-T1 block
$h_{T1T0}$	Initial opening for the Variable Orifice T1-T block
$h_{PA0}$	Initial opening for the Variable Orifice P-A block
$h_{AT0}$	Initial opening for the Variable Orifice A-T block
$x$	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

# 4-Way Directional Valve H

---

## **Dialog Box and Parameters**

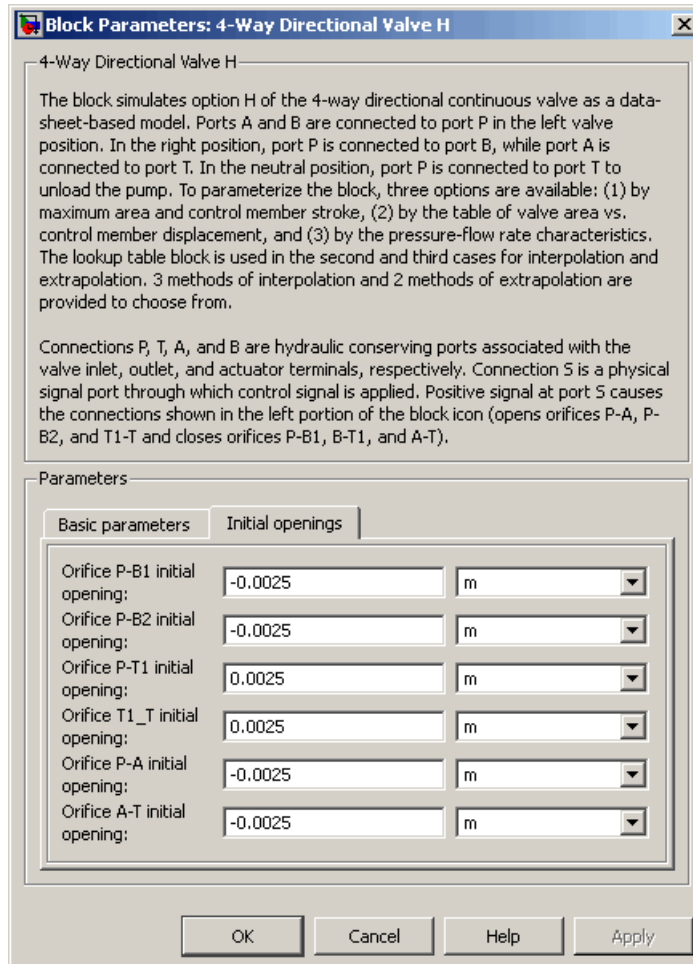
The block dialog box contains two tabs:

- “Basic Parameters” on page 1-104
- “Initial Openings” on page 1-105

### **Basic Parameters**

See the documentation for the Basic Parameters tab of the 4-Way Directional Valve block for parameter descriptions and default values.

## Initial Openings



### Orifice P-B1 initial opening

Initial opening for the Variable Orifice P-B1 block. The parameter can be positive (underlapped orifice), negative

# 4-Way Directional Valve H

---

(overlapped orifice), or equal to zero for zero lap configuration.  
The default value is -0.0025 m.

## **Orifice P-B2 initial opening**

Initial opening for the Variable Orifice P-B2 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is -0.0025 m.

## **Orifice P-T1 initial opening**

Initial opening for the Variable Orifice P-T1 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is 0.0025 m.

## **Orifice T1-T initial opening**

Initial opening for the Variable Orifice T1-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is 0.0025 m.

## **Orifice P-A initial opening**

Initial opening for the Variable Orifice P-A block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is -0.0025 m.

## **Orifice A-T initial opening**

Initial opening for the Variable Orifice A-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is -0.0025 m.

## **Ports**

The block has the following ports:

P

Hydraulic conserving port associated with the pressure supply line inlet.



- T Hydraulic conserving port associated with the return line connection.
- A Hydraulic conserving port associated with the actuator connection port.
- B Hydraulic conserving port associated with the actuator connection port.
- S Physical signal port to control spool displacement.

## **See Also**

- 4-Way Directional Valve
- 4-Way Directional Valve A
- 4-Way Directional Valve B
- 4-Way Directional Valve C
- 4-Way Directional Valve D
- 4-Way Directional Valve E
- 4-Way Directional Valve F
- 4-Way Directional Valve G
- 4-Way Directional Valve K
- 6-Way Directional Valve A

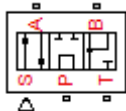
# 4-Way Directional Valve K

---

**Purpose** Configuration K of hydraulic continuous 4-way directional valve

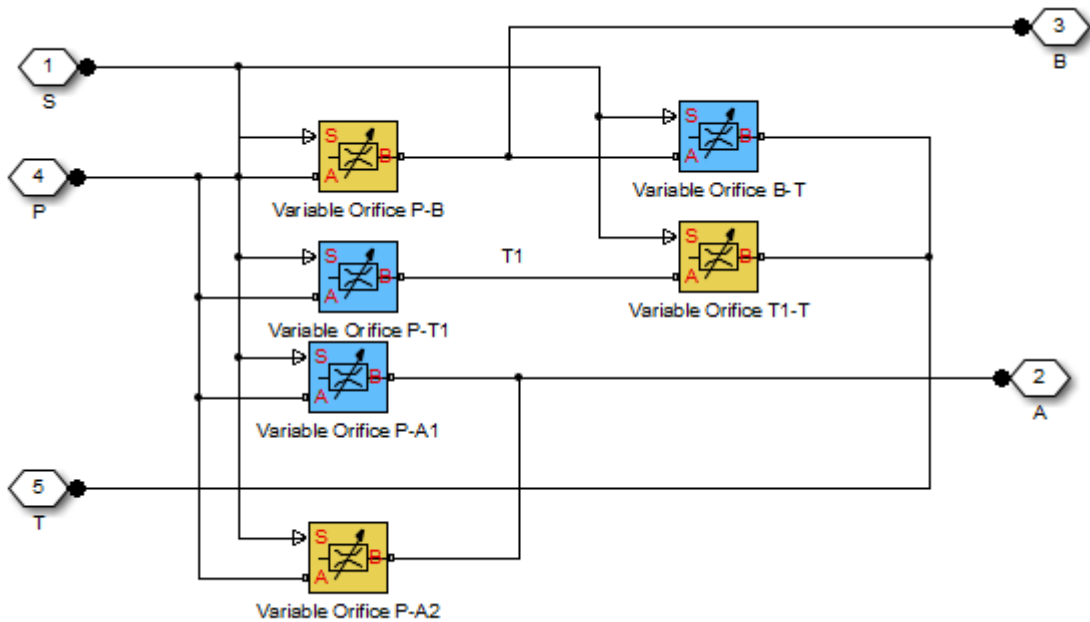
**Library** Directional Valves

## Description



The 4-Way Directional Valve K block simulates a configuration of hydraulic continuous 4-way directional valve. Ports A and B are connected to port P in the right valve position. In the left position, port P is connected to port B, while port A is connected to port T. In neutral position, port P is connected to port T to unload the pump. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of six Variable Orifice blocks, connected as shown in the following diagram.

# 4-Way Directional Valve K



All Variable Orifice blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices P-A1, P-T1, and B-T) and closes the orifices colored yellow (orifices P-A2, P-B, and T1-T). As a result, the openings of the orifices are computed as follows:

$$h_{PA1} = h_{PA10} + x$$

$$h_{PA2} = h_{PA20} - x$$

$$h_{PT1} = h_{PT10} + x$$

$$h_{T1T} = h_{T1T0} - x$$

# 4-Way Directional Valve K

---

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

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

where

$h_{PA1}$	Orifice opening for the Variable Orifice P-A1 block
$h_{PA2}$	Orifice opening for the Variable Orifice P-A2 block
$h_{PT1}$	Orifice opening for the Variable Orifice P-T1 block
$h_{T1T}$	Orifice opening for the Variable Orifice T1-T block
$h_{PB}$	Orifice opening for the Variable Orifice P-B block
$h_{BT}$	Orifice opening for the Variable Orifice B-T block
$h_{PA10}$	Initial opening for the Variable Orifice P-A1 block
$h_{PA20}$	Initial opening for the Variable Orifice P-A2 block
$h_{PT10}$	Initial opening for the Variable Orifice P-T1 block
$h_{T1T0}$	Initial opening for the Variable Orifice T1-T block
$h_{PB0}$	Initial opening for the Variable Orifice P-B block
$h_{BT0}$	Initial opening for the Variable Orifice B-T block
$x$	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

## Dialog Box and Parameters

The block dialog box contains two tabs:

- “Basic Parameters” on page 1-111
- “Initial Openings” on page 1-111

# 4-Way Directional Valve K

## Basic Parameters

See the documentation for the Basic Parameters tab of the 4-Way Directional Valve block for parameter descriptions and default values.

## Initial Openings

**Block Parameters: 4-Way Directional Valve K**

4-Way Directional Valve K

The block simulates option K of the 4-way directional continuous valve as a data-sheet-based model. Ports A and B are connected to port P in the right valve position. In the left position, port P is connected to port B, while port A is connected to port T. In the neutral position, port P is connected to port T to unload the pump. To parameterize the block, three 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 causes the connections shown in the left portion of the block icon (opens orifices P-T1, P-A1, and B-T and closes orifices P-B, P-A2, and T1-T).

Parameters

Basic parameters | Initial openings

Orifice P-A1 initial opening:	-0.0025	m
Orifice P-A2 initial opening:	-0.0025	m
Orifice P-T1 initial opening:	0.0025	m
Orifice T1_T initial opening:	0.0025	m
Orifice P-B initial opening:	-0.0025	m
Orifice B-T initial opening:	-0.0025	m

OK Cancel Help Apply

# 4-Way Directional Valve K

---

## **Orifice P-A1 initial opening**

Initial opening for the Variable Orifice P-A1 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is -0.0025 m.

## **Orifice P-A2 initial opening**

Initial opening for the Variable Orifice P-A2 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is -0.0025 m.

## **Orifice P-T1 initial opening**

Initial opening for the Variable Orifice P-T1 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is 0.0025 m.

## **Orifice T1-T initial opening**

Initial opening for the Variable Orifice T1-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is 0.0025 m.

## **Orifice P-B initial opening**

Initial opening for the Variable Orifice P-B block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is -0.0025 m.

## **Orifice B-T initial opening**

Initial opening for the Variable Orifice B-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is -0.0025 m.

## **Ports**

The block has the following ports:

- P  
Hydraulic conserving port associated with the pressure supply line inlet.
- T  
Hydraulic conserving port associated with the return line connection.
- A  
Hydraulic conserving port associated with the actuator connection port.
- B  
Hydraulic conserving port associated with the actuator connection port.
- S  
Physical signal port to control spool displacement.

## See Also

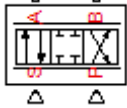
- 4-Way Directional Valve
- 4-Way Directional Valve A
- 4-Way Directional Valve B
- 4-Way Directional Valve C
- 4-Way Directional Valve D
- 4-Way Directional Valve E
- 4-Way Directional Valve F
- 4-Way Directional Valve G
- 4-Way Directional Valve H
- 6-Way Directional Valve A

# 4-Way Ideal Valve

**Purpose** Hydraulic 4-way critically-centered valve

**Library** Directional Valves

## Description



The 4-Way Ideal Valve block represents a 4-way critically-centered valve, where initial openings of all four variable orifices are equal to zero. This significantly simplifies the model of a 4-way directional valve and makes it especially suitable for real-time and HIL (hardware-in-the-loop) simulation, where such assumption is applicable.

The flow rate is computed with the equation:

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

where

- $q$  Flow rate
- $x$  Valve displacement,  $-x_{max} \leq x \leq x_{max}$
- $b$  Orifice width,  $b = A_{max} / x_{max}$
- $A_{max}$  Maximum orifice area
- $x_{max}$  Valve maximum opening
- $C_D$  Flow discharge coefficient
- $\rho$  Fluid density
- $p_S$  Pressure supply
- $p_A, p_B$  Pressures at the load ports A and B, respectively

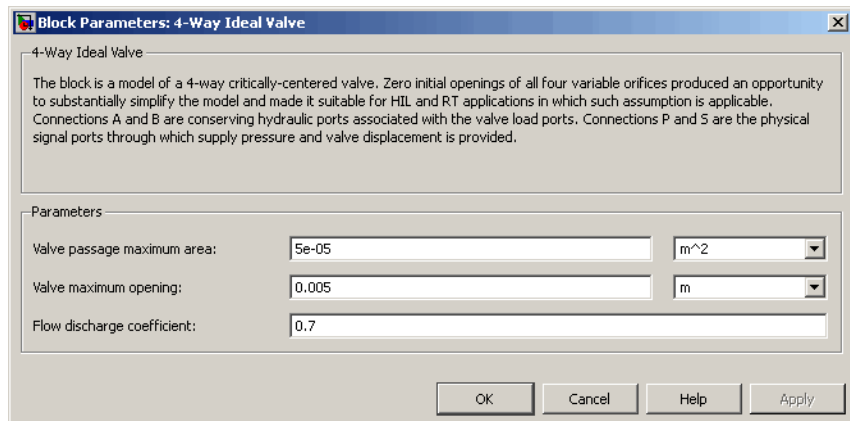
Connections A and B are conserving hydraulic ports associated with the valve load ports. Connections P and S are the physical signal input ports that provide supply pressure and valve displacement values, respectively.



## Basic Assumptions and Limitations

- The valve is of a critically-centered type, that is, all initial openings are equal to zero.
- The return pressure is assumed to be very low and can be treated as a zero pressure.
- All the orifices are assumed to have the same shape and size, that is, the valve is symmetrical.

## Dialog Box and Parameters



### Valve passage maximum area

Specify the area of a fully opened valve. The parameter value must be greater than zero. The default value is  $5e-5 \text{ m}^2$ .

### Valve maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is 0.005 m.

### Flow discharge coefficient

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

# 4-Way Ideal Valve

---

## Global Parameters

Parameter determined by the type of working fluid:

- **Fluid density**

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the actuator connection port.

B

Hydraulic conserving port associated with the actuator connection port.

P

Physical signal port that controls the supply pressure.

S

Physical signal port that controls spool displacement.

## Examples

The Hydraulic Closed-Loop Actuator with Fixed-Step Integration example is an example of using this valve, along with other blocks optimized for real-time and HIL simulation.

## See Also

4-Way Directional Valve

# 6-Way Directional Valve A

## Purpose

Configuration A of hydraulic continuous 6-way directional valve

## Library

Directional Valves

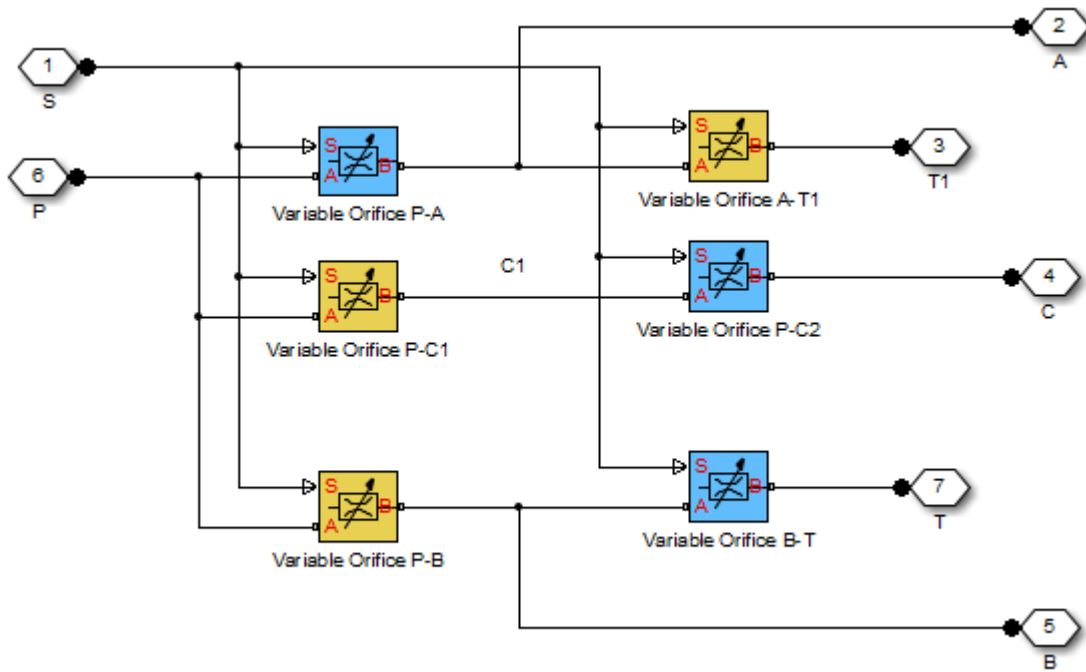
## Description



The 6-Way Directional Valve A block simulates a configuration of hydraulic continuous 6-way directional valve with pump port P, two return ports T and T1, and three actuator ports A, B, and C. Use the valve for applications with two actuators, each being controlled by a valve of this type. When both valves are in neutral position, the pump is unloaded. If any of the valves is shifted from neutral, the diverting line is cut off and the respective actuator is fed at fuel pump pressure. For more details, see the Front-Loader Actuation System example, which uses a similar system of two valves.

The 6-Way Directional Valve A block has six hydraulic connections, corresponding to the valve hydraulic ports (A, B, C, P, T, and T1), and one physical signal port connection (S), which controls the spool position. The block is built of six Variable Orifice blocks, connected as shown in the following diagram.

# 6-Way Directional Valve A



All blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices P-A, B-T, and P-C2) and closes the orifices colored yellow (orifices A-T1, P-B, and P-C1). As a result, the openings of the orifices are computed as follows:

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

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

$$h_{AT1} = h_{AT10} - x$$

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

$$h_{PC1} = h_{PC10} - x$$

$$h_{PC2} = h_{PC20} + x$$

where

$h_{PA}$	Orifice opening for the Variable Orifice P-A block
$h_{PB}$	Orifice opening for the Variable Orifice P-B block
$h_{AT1}$	Orifice opening for the Variable Orifice A-T1 block
$h_{BT}$	Orifice opening for the Variable Orifice B-T block
$h_{PC1}$	Orifice opening for the Variable Orifice P-C1 block
$h_{PC2}$	Orifice opening for the Variable Orifice P-C2 block
$h_{PA0}$	Initial opening for the Variable Orifice P-A block
$h_{PB0}$	Initial opening for the Variable Orifice P-B block
$h_{AT10}$	Initial opening for the Variable Orifice A-T1 block
$h_{BT0}$	Initial opening for the Variable Orifice B-T block
$h_{PC10}$	Initial opening for the Variable Orifice P-C1 block
$h_{PC20}$	Initial opening for the Variable Orifice P-C2 block
$x$	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

# 6-Way Directional Valve A

---

## **Dialog Box and Parameters**

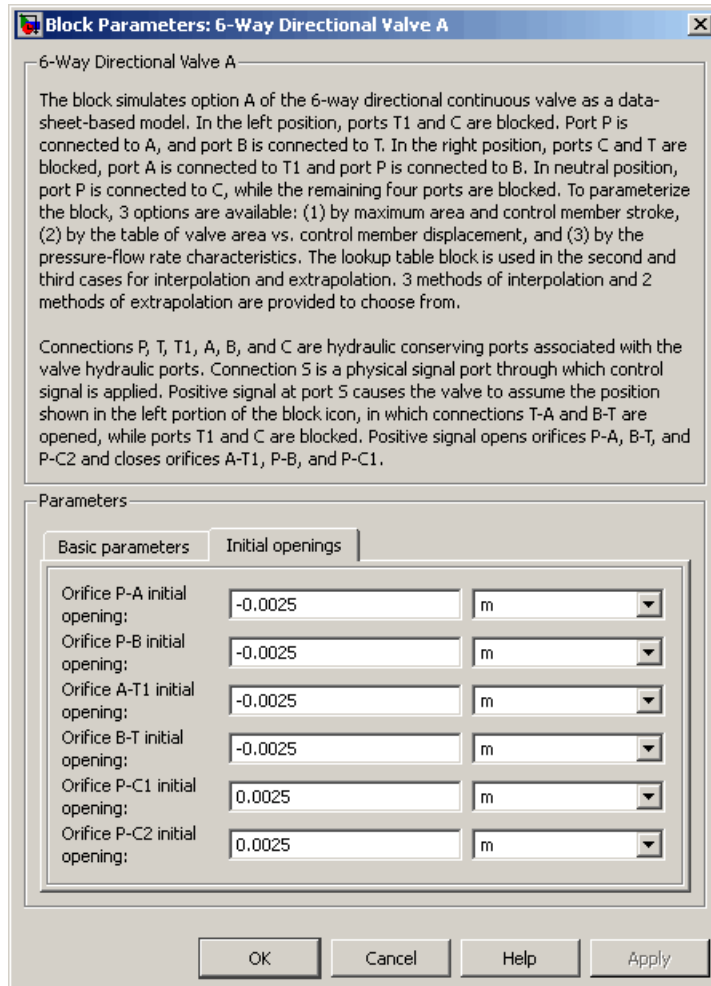
The block dialog box contains two tabs:

- “Basic Parameters” on page 1-120
- “Initial Openings” on page 1-121

### **Basic Parameters**

See the documentation for the Basic Parameters tab of the 4-Way Directional Valve block for parameter descriptions and default values.

## Initial Openings



### Orifice P-A initial opening

Initial opening for the Variable Orifice P-A block. The parameter can be positive (underlapped orifice), negative

# 6-Way Directional Valve A

---

(overlapped orifice), or equal to zero for zero lap configuration.  
The default value is -0.0025 m.

## **Orifice P-B initial opening**

Initial opening for the Variable Orifice P-B block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is -0.0025 m.

## **Orifice A-T1 initial opening**

Initial opening for the Variable Orifice A-T1 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is -0.0025 m.

## **Orifice B-T initial opening**

Initial opening for the Variable Orifice B-T block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is -0.0025 m.

## **Orifice P-C1 initial opening**

Initial opening for the Variable Orifice P-C1 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is 0.0025 m.

## **Orifice P-C2 initial opening**

Initial opening for the Variable Orifice P-C2 block. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration.  
The default value is 0.0025 m.

## **Ports**

The block has the following ports:

P

Hydraulic conserving port associated with the pressure supply line inlet.



- T Hydraulic conserving port associated with the first return line connection.
- T1 Hydraulic conserving port associated with the second return line connection.
- A Hydraulic conserving port associated with the actuator connection port.
- B Hydraulic conserving port associated with the actuator connection port.
- C Hydraulic conserving port associated with the actuator connection port.
- S Physical signal port to control spool displacement.

## See Also

- 4-Way Directional Valve
- 4-Way Directional Valve A
- 4-Way Directional Valve B
- 4-Way Directional Valve C
- 4-Way Directional Valve D
- 4-Way Directional Valve E
- 4-Way Directional Valve F
- 4-Way Directional Valve G
- 4-Way Directional Valve H
- 4-Way Directional Valve K

# Angle Sensor

## Purpose

Ideal angle sensor with measuring range from 0 to 360 degrees

## Library

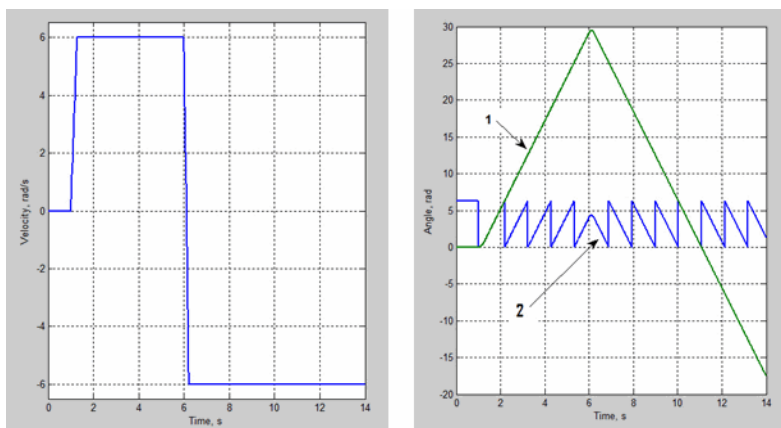
Pumps and Motors

## Description



The Angle Sensor block represents an ideal angle sensor with measuring range from 0 to 360 degrees. To measure an angular displacement, the sensor must be connected to a mechanical rotational port. No load is imposed on the measured object. The sensor output is kept within the range from 0 to  $2\pi$  radians (360 degrees) regardless of the number of revolutions performed by the object and the direction of rotation. The sensor is ideal because it does not account for inertia, friction, and so on.

The difference between the Angle Sensor block and the Ideal Rotational Motion Sensor block, available in the Simscape™ Foundation library, is demonstrated in the following figure.



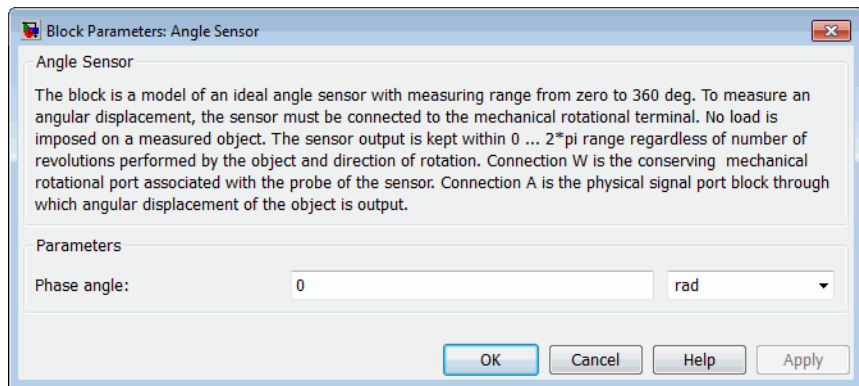
In this example, the object is moving at 6 rad/s in the positive direction for the first 5 seconds, and then switches to the negative direction at the same speed. The Ideal Rotational Motion Sensor block output (line 1) shows that the object turned forward by 30 rad and then turned back in the negative direction, continuing until  $-20$  rad. The reading of the Angle Sensor block (line 2) stays in the range from 0 to  $2\pi$  rad.

Availability of the Angle Sensor block considerably simplifies development of models with complex relationship between model parameters and rotation angle, such as pumps and motors.

## Basic Assumptions and Limitations

- The sensor is assumed to be ideal. No load is imposed on the object.
- No inertial effects are considered.

## Dialog Box and Parameters



### Phase angle

This parameter lets you account for angular shift between the object and the sensor input. The default value is 0.

## Ports

The block has the following ports:

W

Mechanical rotational conserving port associated with the probe of the sensor.

A

Physical signal port that outputs the angular displacement of the object.

# Angle Sensor

---

## Examples

The Hydraulic Axial-Piston Pump with Load-Sensing and Pressure-Limiting Control example models a test rig designed to investigate interaction between an axial-piston pump and a typical control unit, simultaneously performing the load-sensing and pressure-limiting functions. To assure required accuracy, the model of the pump must account for such features as interaction between pistons, swash plate, and porting plate, which makes it necessary to build a detailed pump model.

The Angle Sensor block is used in each of the piston model subsystems (Piston\_1, ..., Piston\_5), where it provides the angular position of the respective piston as input to the Porting Plate Variable Orifice blocks.

## See Also

Ideal Rotational Motion Sensor

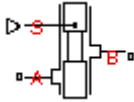
## Purpose

Hydraulic variable orifice created by circular tube and round insert

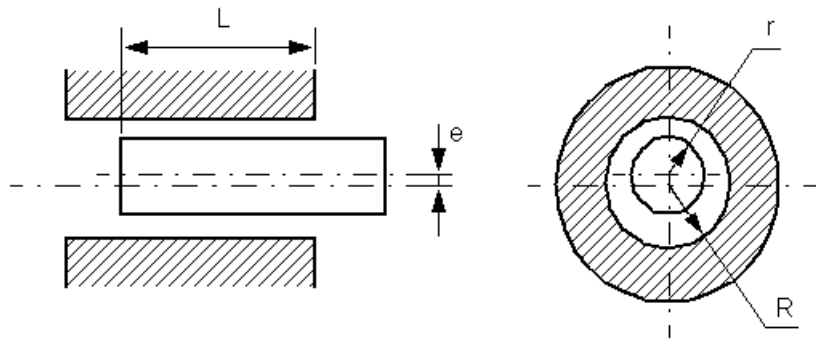
## Library

Orifices

## Description



The Annular Orifice block models a variable orifice created by a circular tube and a round insert, which may be eccentrically located with respect to the tube. The radial gap between the tube and the insert and its axial length are assumed to be essentially smaller than the insert diameter, causing the flow regime to be laminar all the time. A schematic representation of the annular orifice is shown in the following illustration.



The flow rate is computed using the Hagen-Poiseuille equation (see [1]):

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

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

where

# Annular Orifice

---

$q$	Flow rate
$p$	Pressure differential
$R$	Orifice radius
$r$	Insert radius
$L$	Overlap length
$\varepsilon$	Eccentricity ratio
$e$	Eccentricity
$\rho$	Fluid density
$\nu$	Fluid kinematic viscosity

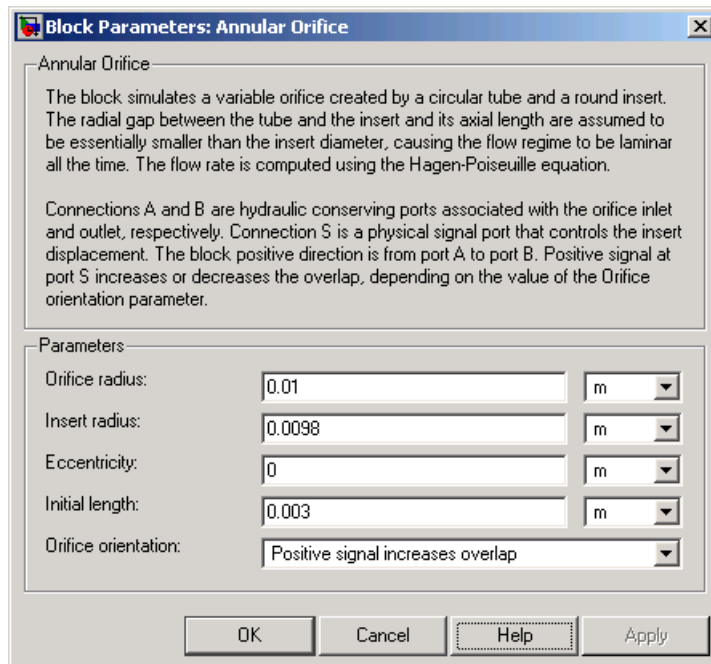
Use this block to simulate leakage path in plungers, valves, and cylinders.

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

## Basic Assumptions and Limitations

Fluid inertia is not taken into account.

## Dialog Box and Parameters



### Orifice radius

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

### Insert radius

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

### Eccentricity

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

### Initial length

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

# Annular Orifice

---

## Orifice orientation

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

## Restricted Parameters

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

- **Orifice orientation**

All other block parameters are available for modification.

## Global Parameters

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

- A  
Hydraulic conserving port associated with the orifice inlet.
- B  
Hydraulic conserving port associated with the orifice outlet.
- S  
Physical signal port that controls the insert displacement.

## References

[1] Noah D. Manring, *Hydraulic Control Systems*, John Wiley & Sons, 2005



## **See Also**

Constant Area Hydraulic Orifice

Fixed Orifice

Orifice with Variable Area Round Holes

Orifice with Variable Area Slot

Variable Area Hydraulic Orifice

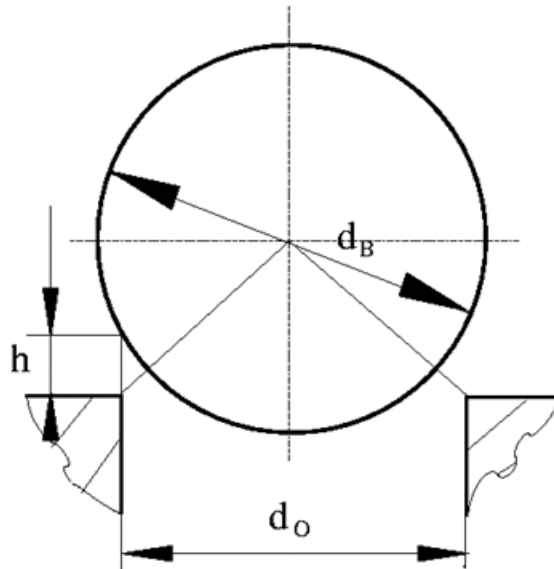
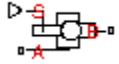
Variable Orifice

# Ball Valve

**Purpose** Hydraulic ball valve

**Library** Flow Control Valves

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



The flow rate through the valve is proportional to the valve opening and to the pressure differential across the valve. The model accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number ( $Re$ ) and comparing its value with the critical Reynolds number ( $Re_{cr}$ ). The flow rate is determined according to the following equations:

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

$$h = x_0 + x$$

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

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

$$p = p_A - p_B$$

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

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

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

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

# Ball Valve

---

$$h_{\max} = r_O \cdot \left( \sqrt{\frac{\left(1 + \sqrt{1 + 4 \frac{d_B^2}{d_O^2}}\right)^2}{4}} - 1 - \sqrt{\frac{d_B^2}{d_O^2} - 1} \right)$$

where

$q$	Flow rate
$\rho$	Pressure differential
$p_A, p_B$	Gauge pressures at the block terminals
$C_D$	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
$x_0$	Initial opening
$x$	Ball displacement from initial position
$h$	Valve opening
$d_o$	Orifice diameter
$r_o$	Orifice radius
$d_B$	Ball diameter
$r_B$	Ball radius
$\rho$	Fluid density
$D_H$	Valve instantaneous hydraulic diameter
$\nu$	Fluid kinematic viscosity
$A_{leak}$	Closed valve leakage area
$A_{max}$	Maximum valve open area
$h_{max}$	Maximum valve opening

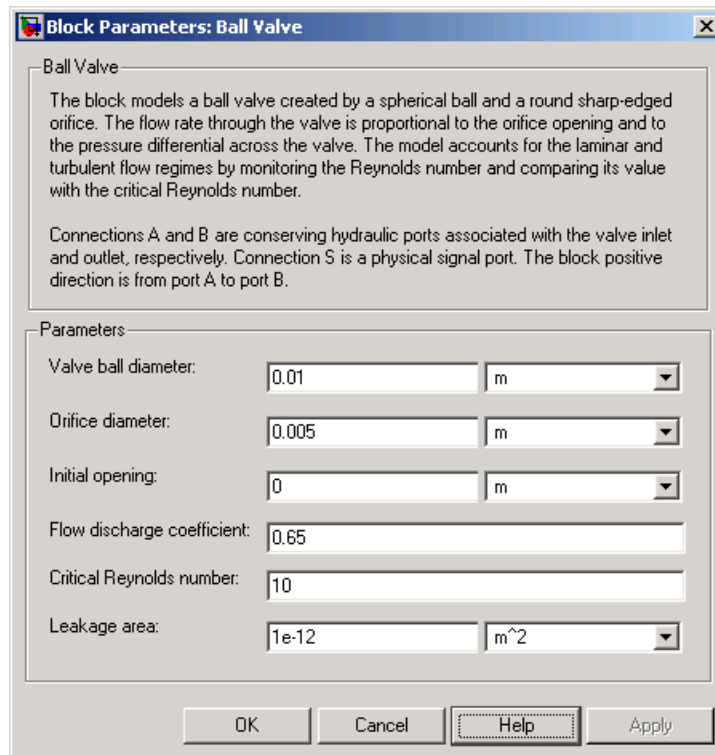
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as  $p = p_A - p_B$ . Positive signal at the physical signal port S opens the valve.

## Basic Assumptions and Limitations

- Fluid inertia is not taken into account.
- The 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.

# Ball Valve

## Dialog Box and Parameters



### 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 simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## Global Parameters

Parameters determined by the type of working fluid:

- Fluid density
- Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

# Ball Valve

---

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

## See Also

Ball Valve with Conical Seat  
Gate Valve  
Needle Valve  
Poppet Valve  
Pressure-Compensated Flow Control Valve



## Purpose

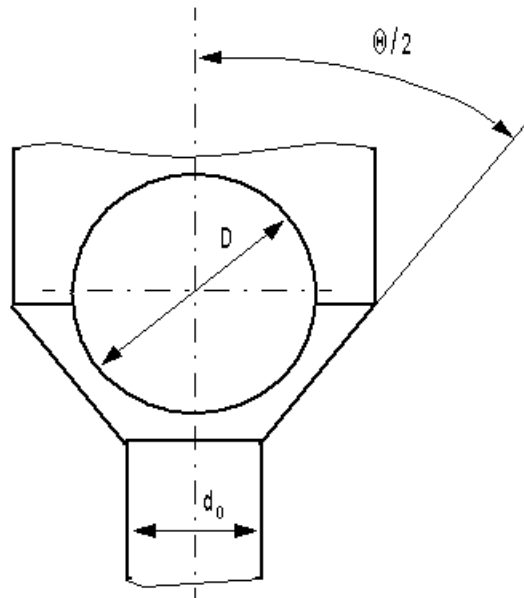
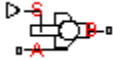
Hydraulic ball valve with conical seat

## Library

Flow Control Valves

## Description

The Ball Valve with Conical Seat block models a valve created by a spherical ball and a conical seat.



The valve is characterized by the ball diameter, cone angle, and orifice diameter. The flow rate through the valve is proportional to the ball displacement and pressure differential. If passage area in the ball-cone contact exceeds the area of the orifice, the latter is assumed as the valve passage area. 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:

# Ball Valve with Conical Seat

---

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

$$h = x_0 + x$$

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

$$p = p_A - p_B$$

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

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

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

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

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
$d_o$	Orifice diameter
$D$	Ball diameter
$\Theta$	Cone angle of the valve seat
$\rho$	Fluid density
$D_H$	Valve instantaneous hydraulic diameter
$\nu$	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

- 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 and the conical seat.

# Ball Valve with Conical Seat

## Dialog Box and Parameters

**Block Parameters: Ball Valve with Conical Seat**

Ball Valve with Conical Seat

The block models a ball valve created by a spherical ball and a conical seat. The valve is characterized by the ball diameter, cone angle, and orifice diameter. The flow rate through the valve is proportional to the ball displacement and pressure differential. If passage area in the ball-cone contact exceeds area of the orifice, the latter is assumed as the valve passage area. 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 through which ball displacement is imported. The block positive direction is from port A to port B.

Parameters

Valve ball diameter:	<input type="text" value="0.01"/>	<input type="text" value="m"/>
Orifice diameter:	<input type="text" value="0.005"/>	<input type="text" value="m"/>
Cone angle:	<input type="text" value="120"/>	<input type="text" value="deg"/>
Initial opening:	<input type="text" value="0"/>	<input type="text" value="m"/>
Flow discharge coefficient:	<input type="text" value="0.65"/>	
Critical Reynolds number:	<input type="text" value="10"/>	
Leakage area:	<input type="text" value="1e-12"/>	<input type="text" value="m^2"/>

OK Cancel Help Apply

### Valve ball diameter

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

### Orifice diameter

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

### Cone angle

The cone angle of the valve seat. The default value is 120 degrees.

## Initial opening

The initial opening of the valve. Its value must be 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 simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## Global Parameters

Parameters determined by the type of working fluid:

- Fluid density
- Fluid kinematic viscosity

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

## Ports

The block has the following ports:

# Ball Valve with Conical Seat

---

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

## **See Also**

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

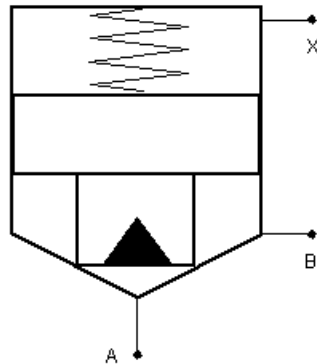
**Purpose** Hydraulic cartridge valve insert

**Library** Directional Valves

## Description

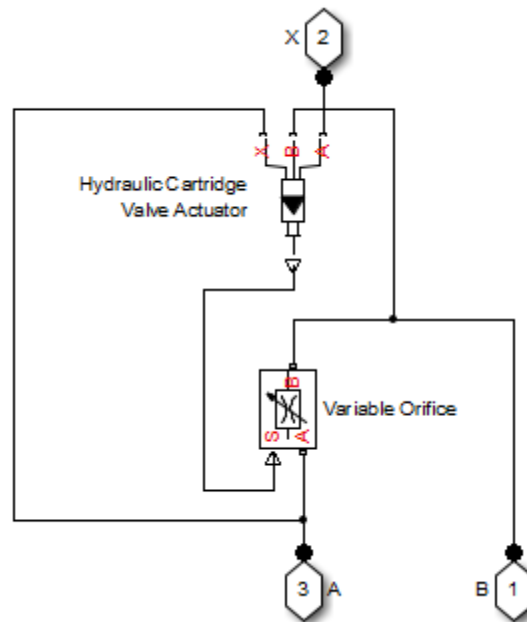


The Cartridge Valve Insert block represents an insert of a hydraulic cartridge valve consisting of a poppet interacting with the seat. The poppet position is determined by pressures at ports A, B, and X and force of the spring. A schematic diagram of the cartridge valve insert is shown in the following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

# Cartridge Valve Insert



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

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



## Basic Assumptions and Limitations

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

# Cartridge Valve Insert

## Dialog Box and Parameters

**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 if the poppet and/or the sleeve have complex geometrical form (e.g. notched, slotted, or chamfered poppet). For inserts with plain cylindrical poppets and conical seats, the Cartridge Valve Insert with Conical Seat is recommended. 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 except pressure and spring forces. Inertial properties of the poppet are accounted by inserting a first order lag between the steady-state and actual poppet displacements.

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 orifice 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:	<input type="text" value="2e-04"/>	<input type="text" value="m^2"/>
Port A to port X area ratio:	<input type="text" value="0.66"/>	
Preload force:	<input type="text" value="26"/>	<input type="text" value="N"/>
Spring rate:	<input type="text" value="1.4e+04"/>	<input type="text" value="N/m"/>
Poppet stroke:	<input type="text" value="0.005"/>	<input type="text" value="m"/>
Initial opening:	<input type="text" value="0"/>	<input type="text" value="m"/>
Orifice specification:	<input type="text" value="By maximum area and opening"/>	
Orifice maximum area:	<input type="text" value="5e-05"/>	<input type="text" value="m^2"/>
Flow discharge coefficient:	<input type="text" value="0.7"/>	
Critical Reynolds number:	<input type="text" value="12"/>	
Leakage area:	<input type="text" value="1e-12"/>	<input type="text" value="m^2"/>
Time constant:	<input type="text" value="0.01"/>	<input type="text" value="s"/>

OK Cancel Help Apply

# Cartridge Valve Insert

**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 if the poppet and/or the sleeve have complex geometrical form (e.g. notched, slotted, or chamfered poppet). For inserts with plain cylindrical poppets and conical seats, the Cartridge Valve Insert with Conical Seat is recommended. 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 except pressure and spring forces. Inertial properties of the poppet are accounted by inserting a first order lag between the steady-state and actual poppet displacements.

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 orifice 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 <sup>2</sup>
Port A to port X area ratio:	0.66	
Preload force:	26	N
Spring rate:	1.4e+04	N/m
Initial opening:	0	m
Orifice specification:	By area vs. opening table	
Tabulated orifice openings:	[ -0.002 0 0.002 0.005 0.015 ]	m
Tabulated orifice area:	1e-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356 ]	m <sup>2</sup>
Interpolation method:	Linear	
Extrapolation method:	From last 2 points	
Flow discharge coefficient:	0.7	
Critical Reynolds number:	12	
Time constant:	0.01	s

OK Cancel Help Apply

# Cartridge Valve Insert

**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 if the poppet and/or the sleeve have complex geometrical form (e.g. notched, slotted, or chamfered poppet). For inserts with plain cylindrical poppets and conical seats, the Cartridge Valve Insert with Conical Seat is recommended. 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 except pressure and spring forces. Inertial properties of the poppet are accounted by inserting a first order lag between the steady-state and actual poppet displacements.

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 orifice 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 <sup>2</sup>
Port A to port X area ratio:	0.66	
Preload force:	26	N
Spring rate:	1.4e+04	N/m
Initial opening:	0	m
Orifice specification:	By pressure-flow characteristic	
Tabulated orifice openings:	[ -0.002 0 0.002 0.005 0.015 ]	m
Tabulated pressure differentials:	[ -1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07 ]	Pa
Tabulated flow rates:	[ 56 -0.024293 -0.015364 0.015364 0.024293 0.034356 ; ]	m <sup>3</sup> /s
Interpolation method:	Linear	
Extrapolation method:	From last 2 points	
Time constant:	0.01	s

OK Cancel Help Apply

## Port A poppet area

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

## Port A to port X area ratio

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

**Preload force**

Spring preload force. The default value is 26 N.

**Spring rate**

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

**Poppet stroke**

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

**Initial opening**

The initial opening of the valve. Its value must be greater than or equal to zero. The default value is 0.

**Orifice specification**

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

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

# Cartridge Valve Insert

---

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

## **Orifice maximum area**

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

## **Tabulated orifice openings**

Specify the vector of input values for orifice openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in meters, are  $[-0.002 \ 0 \ 0.002 \ 0.005 \ 0.015]$ . If **Orifice specification** is set to By area vs. opening table, the **Tabulated orifice openings** values will be used together with **Tabulated orifice area** values for one-dimensional table lookup. If **Orifice specification** is set to By pressure-flow characteristic, the **Tabulated orifice openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

## **Tabulated orifice area**

Specify the vector of orifice areas as a one-dimensional array. The vector must be of the same size as the orifice openings vector. All the values must be positive. The default values, in  $\text{m}^2$ , are  $[1e-09 \ 2.0352e-07 \ 4.0736e-05 \ 0.00011438 \ 0.00034356]$ . This parameter is used if **Orifice specification** is set to By area vs. opening table.

## **Tabulated pressure differentials**

Specify the pressure differential vector as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline

interpolation. The default values, in Pa, are [-1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]. This parameter is used if **Orifice specification** is set to By pressure-flow characteristic.

## Tabulated flow rates

Specify the flow rates as an m-by-n matrix, where m is the number of orifice openings and n is the number of pressure differentials. Each value in the matrix specifies flow rate taking place at a specific combination of orifice opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in m<sup>3</sup>/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 Polynomial (PCHIP). For two-dimensional table lookup (By pressure-flow characteristic), uses the bicubic interpolation algorithm.

# Cartridge Valve Insert

---

- **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 simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## Time constant

Sets the time constant of the first-order lag, which is introduced between the required and the actual poppet positions to account for actuator dynamics. The parameter value must be greater than zero. The default value is 0.01 s.

## Restricted Parameters

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

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

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

# Cartridge Valve Insert

---

## Global Parameters

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

X

Hydraulic conserving port associated with the valve control terminal.

## See Also

Cartridge Valve Insert with Conical Seat

Check Valve

Hydraulic Cartridge Valve Actuator

Pilot-Operated Check Valve

# Cartridge Valve Insert with Conical Seat

## Purpose

Hydraulic cartridge valve insert with conical seat

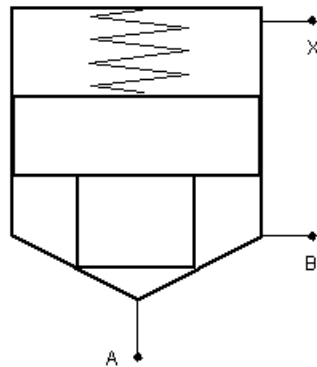
## Library

Directional Valves

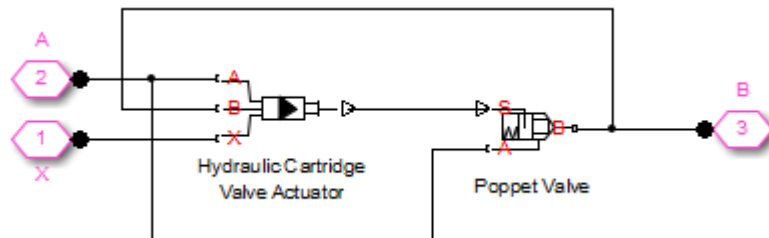
## Description



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



The Cartridge Valve Insert with Conical Seat block is a structural model consisting of a Hydraulic Cartridge Valve Actuator block and a Poppet Valve block, as shown in the next illustration.



# Cartridge Valve Insert with Conical Seat

---

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

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

## **Basic Assumptions and Limitations**

- Valve opening is linearly proportional to the pressure differential.
- No loading on the poppet, such as inertia or friction, is considered.
- The model does not account for flow consumption caused by poppet displacement.

# Cartridge Valve Insert with Conical Seat

## Dialog Box and Parameters

**Block Parameters: Cartridge Valve Insert with Conical Seat**

Cartridge Valve Insert with Conical Seat

This block represents an insert of hydraulic cartridge valve arranged as a cylindrical poppet interacting with the conical 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 except pressure and spring forces. Inertial properties of the poppet are accounted by inserting a first order lag between the steady-state and actual poppet displacements.

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 orifice between the poppet and the seat are simulated with the Poppet Valve block.

Connections A, B, and X are hydraulic conserving ports associated with the valve inlet, valve outlet, and valve control 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:	<input type="text" value="2e-04"/>	<input type="text" value="m^2"/>
Port A to port X area ratio:	<input type="text" value="0.66"/>	
Preload force:	<input type="text" value="26"/>	<input type="text" value="N"/>
Spring rate:	<input type="text" value="1.4e+04"/>	<input type="text" value="N/m"/>
Poppet stroke:	<input type="text" value="0.005"/>	<input type="text" value="m"/>
Poppet diameter:	<input type="text" value="0.01"/>	<input type="text" value="m"/>
Seat cone angle:	<input type="text" value="120"/>	<input type="text" value="deg"/>
Initial opening:	<input type="text" value="0"/>	<input type="text" value="m"/>
Flow discharge coefficient:	<input type="text" value="0.65"/>	
Critical Reynolds number:	<input type="text" value="10"/>	
Leakage area:	<input type="text" value="1e-12"/>	<input type="text" value="m^2"/>
Time constant:	<input type="text" value="0.01"/>	<input type="text" value="s"/>

OK Cancel Help Apply

### Port A poppet area

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

### Port A to port X area ratio

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

# Cartridge Valve Insert with Conical Seat

---

## **Preload force**

Spring preload force. The default value is 26 N.

## **Spring rate**

Spring rate. The default value is  $1.4 \times 10^4$  N/m.

## **Poppet stroke**

Maximum poppet stroke. The parameter value must be greater than zero. The default value is 0.005 m.

## **Poppet diameter**

Maximum poppet diameter. The parameter value must be greater than or equal to zero. The default value is 0.01 m.

## **Seat cone angle**

The cone angle of the valve seat. The default value is 120 degrees.

## **Initial opening**

The initial opening of the valve. Its value must be greater than or equal to zero. The default value is 0.

## **Flow discharge coefficient**

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

## **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

# Cartridge Valve Insert with Conical Seat

---

and even cause simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## Time constant

Sets the time constant of the first-order lag, which is introduced between the required and the actual poppet positions to account for actuator dynamics. The parameter value must be greater than zero. The default value is 0.01 s.

## Global Parameters

Parameters determined by the type of working fluid:

- Fluid density
- Fluid kinematic viscosity

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

## Ports

The block has the following ports:

- A  
Hydraulic conserving port associated with the valve inlet.
- B  
Hydraulic conserving port associated with the valve outlet.
- X  
Hydraulic conserving port associated with the valve control terminal.

## Examples

For an example of using this block, see the Hydraulic Actuation System with Cartridge Valves example.

## See Also

Cartridge Valve Insert  
Hydraulic Cartridge Valve Actuator  
Poppet Valve

# Centrifugal Force in Rotating Cylinder

**Purpose** Centrifugal force in rotating hydraulic cylinders

**Library** Hydraulic Cylinders

## Description



The Centrifugal Force in Rotating Cylinder block is a building block to be used in hydraulic cylinder models, to account for centrifugal forces exerted by fluid on a piston, if the cylinder rotates about its symmetry axis. Such cylinders are used in control mechanisms of various friction clutches, brakes, square-jaw positive clutches, dog clutches, and so on. No inertial effects are considered in the model. In other words, the angular velocity is assumed to be constant or changing at very low speed.

The centrifugal force is computed with the following equation:

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

where

$F$  Centrifugal force

$r_o$  Piston outer radius

$r_i$  Piston inner radius

$r_p$

Fluid entry radius. For design purposes,  $r_p < \sqrt{(r_o^2 + r_i^2)}/2$

$\rho$  Fluid density

$\omega$  Shaft angular velocity

Connections R and C are mechanical translational conserving ports corresponding to the cylinder rod and case, respectively. Connection W is a physical signal port through which shaft angular velocity is imported. The block directionality is adjustable and can be controlled with the **Cylinder orientation** parameter.

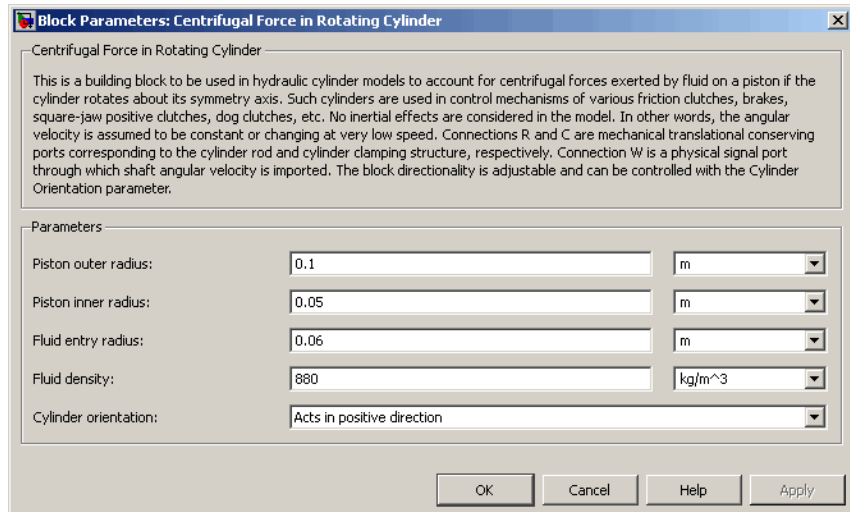


# Centrifugal Force in Rotating Cylinder

## Basic Assumptions and Limitations

- No inertial effects are considered in the model; that is, the angular velocity of the cylinder is assumed to be constant or changing at very low speed.
- Fluid inertia is not taken into account.
- Fluid compressibility is not taken into account.

## Dialog Box and Parameters



### Piston outer radius

The default value is 0.1 m.

### Piston inner radius

The default value is 0.05 m.

### Fluid entry radius

The default value is 0.06 m.

### Fluid density

The default value is 880 kg/m<sup>3</sup>.

# Centrifugal Force in Rotating Cylinder

---

## Cylinder orientation

Specifies block orientation with respect to the globally assigned positive direction. The block can be installed in two different ways, depending upon whether it exerts force in the positive or in the negative direction. If positive velocity applied at port W exerts force in negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

## Restricted Parameters

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

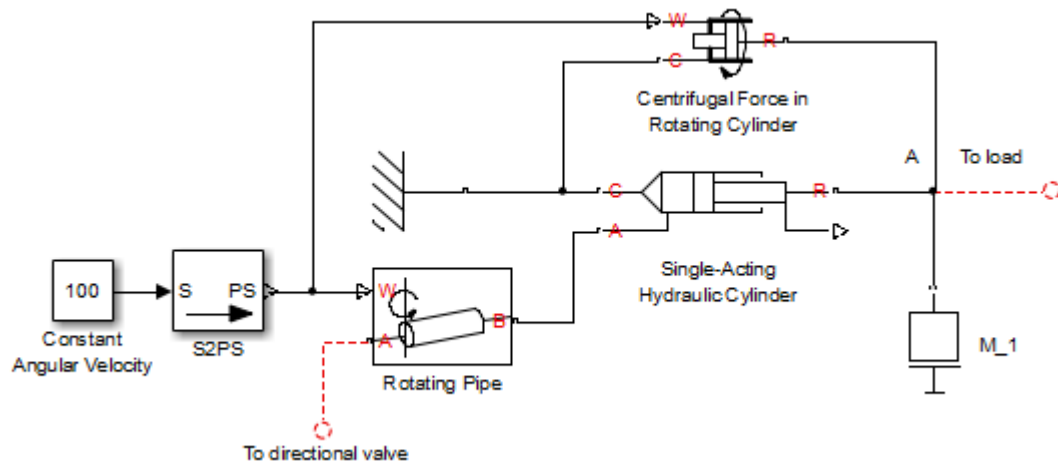
- **Cylinder orientation**

All other block parameters are available for modification.

## Examples

The following schematic diagram shows a model of a custom single-acting hydraulic cylinder built of the Single-Acting Hydraulic Cylinder, Rotating Pipe, and Centrifugal Force in Rotating Cylinder blocks.

# Centrifugal Force in Rotating Cylinder



The centrifugal force generated by the Centrifugal Force in Rotating Cylinder block sums up with that of the cylinder at node A. The Rotating Pipe block simulates the pipeline between the cylinder chamber and the channel in the center of a rotating shaft where the cylinder is installed.

## Ports

The block has the following ports:

- R  
Mechanical translational conserving port associated with the cylinder rod.
- C  
Mechanical translational conserving port associated with the cylinder clamping structure.
- W  
Physical signal port providing the angular velocity value.

## See Also

Rotating Pipe

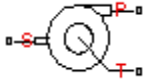
# Centrifugal Pump

---

**Purpose** Centrifugal pump with choice of parameterization options

**Library** Pumps and Motors

## Description



The Centrifugal Pump block represents a centrifugal pump of any type as a data-sheet-based model. Depending on data listed in the manufacturer's catalog or data sheet for your particular pump, you can choose one of the following model parameterization options:

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

These parameterization options are further described in greater detail:

- “Parameterizing the Pump by Approximating Polynomial” on page 1-167
- “Parameterizing the Pump by Pressure Differential and Brake Power Versus Pump Delivery” on page 1-172
- “Parameterizing the Pump by Pressure Differential and Brake Power Versus Pump Delivery at Different Angular Velocities” on page 1-173

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

## Parameterizing the Pump by Approximating Polynomial

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

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

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

where

$p_{ref}$  Pressure differential across the pump for the reference regime, characterized by the reference angular velocity and density

$k$  Correction factor. The factor is introduced to account for dimensional fluctuations, blade incongruity, blade volumes, fluid internal friction, and so on. The factor should be set to 1 if the approximating coefficients are determined experimentally.

$p_E$  Euler pressure

$p_{HL}$  Pressure loss due to hydraulic losses in the pump passages

$p_D$  Pressure loss caused by deviations of the pump delivery from its nominal (rated) value

# Centrifugal Pump

---

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

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

where

- $\rho_{ref}$  Fluid density
- $c_0, c_1$  Approximating coefficients. They can be determined either analytically from the Euler equation [1, 2] or experimentally.
- $q_{ref}$  Pump volumetric delivery at reference regime

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

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

where

- $\rho_{ref}$  Fluid density
- $c_2$  Approximating coefficient
- $q_{ref}$  Pump volumetric delivery at reference regime

The blade profile is determined for a specific fluid velocity, and deviation from this velocity results in pressure loss due to inconsistency between the fluid velocity and blade profile velocity. This pressure loss,  $p_D$ , is estimated with the equation

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

where

$\rho_{ref}$	Fluid density
$c_3$	Approximating coefficient
$q_{ref}$	Pump volumetric delivery at reference regime
$q_D$	Pump design delivery (nominal delivery)

The resulting approximating polynomial takes the form:

$$p_{ref} = \rho_{ref} \left( k(c_0 - c_1 q_{ref}) - c_2 q_{ref}^2 - c_3 (q_D - q_{ref})^2 \right) \quad (1-2)$$

The pump characteristics, approximated with four coefficients  $c_0$ ,  $c_1$ ,  $c_2$ , and  $c_3$ , are determined for a specific fluid and a specific angular velocity of the pump's driving shaft. These two parameters correspond, respectively, to the **Reference density** and **Reference angular velocity** parameters in the block dialog box. To apply the characteristics for another velocity  $\omega$  or density  $\rho$ , the affinity laws are used. With these laws, the delivery at reference regime, which corresponds to given pump delivery and angular velocity, is computed with the expression

$$q_{ref} = q \frac{\omega_{ref}}{\omega} \quad (1-3)$$

where  $q$  and  $\omega$  are the instantaneous values of the pump delivery and angular velocity. Then the pressure differential  $p_{ref}$  at reference regime computed with Equation 1-2 and converted into pressure differential  $p$  at current angular velocity and density

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

Equation 1-2 describes pump characteristic for  $\omega > 0$  and  $q \geq 0$ . Outside this range, the characteristic is approximated with the following relationships:

# Centrifugal Pump

---

$$p = \begin{cases} -k_{leak} \cdot q & \text{for } \omega \leq 0 \\ p_{max} - k_{leak} \cdot q & \text{for } \omega > 0, q < 0 \\ -k_{leak} \cdot (q - q_{max}) & \text{for } \omega > 0, q > q_{max} \end{cases} \quad (1-4)$$

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

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

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

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

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

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

where

$k_{leak}$  Leakage resistance coefficient

$q_{max}$  Maximum pump delivery at given angular velocity. The delivery is determined from Equation 1-2 at  $p = 0$ .

$p_{max}$  Maximum pump pressure at given angular velocity. The pressure is determined from Equation 1-2 at  $q = 0$ .

$k$  Correction factor, as described in Equation 1-1.

The hydraulic power at the pump outlet at reference conditions is

$$N_{hyd} = P_{ref} \cdot q_{ref}$$



The output hydraulic power at arbitrary angular velocity and density is determined with the affinity laws

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

The power at the pump driving shaft consists of the theoretical hydraulic power (power before losses associated with hydraulic loss and deviation from the design delivery) and friction loss at the driving shaft. The theoretical hydraulic power is approximated using the Euler pressure

$$N_{hyd0} = p_{Eref} \cdot q_{ref} \cdot \left( \frac{\omega}{\omega_{ref}} \right)^3$$

where

- $N_{hyd0}$  Pump theoretical hydraulic power
- $p_{Eref}$  Euler pressure. The theoretical pressure developed by the pump before losses associated with hydraulic loss and deviation from the design delivery.

The friction losses are approximated with the relationship:

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

where

- $N_{fr}$  Friction loss power
- $T_0$  Constant torque at driving shaft associated with shaft bearings, seal friction, and so on
- $k_p$  Torque-pressure relationship, which characterizes the influence of pressure on the driving shaft torque

# Centrifugal Pump

---

The power and torque at the pump driving shaft (brake power  $N_{mech}$  and brake torque  $T$ ) are

$$N_{mech} = N_{hyd0} + N_{fr}$$

$$T = \frac{N_{mech}}{\omega}$$

The pump total efficiency  $\eta$  is computed as

$$\eta = \frac{N_{hyd}}{N_{mech}}$$

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

If you set the **Model parameterization** parameter to **By two 1D characteristics: P-Q and N-Q**, the pump characteristics are computed by using two one-dimensional table lookups: for the pressure differential based on the pump delivery and for the pump brake power based on the pump delivery. Both characteristics are specified at the same angular velocity  $\omega_{ref}$  (**Reference angular velocity**) and the same fluid density  $\rho_{ref}$  (**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  $\omega$  and density  $\rho$  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  $\rho$  is different from the reference density  $\rho_{ref}$  at which characteristics have been obtained

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

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

# Centrifugal Pump

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

## Basic Assumptions and Limitations

- Fluid compressibility is neglected.
- The pump rotates in positive direction, with speed that is greater or equal to zero.
- The reverse flow through the pump is allowed only at still shaft.

## Dialog Box and Parameters

**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 approximating polynomial	
First approximating coefficient:	326.8	Pa/(kg/m <sup>3</sup> )
Second approximating coefficient:	3.104e+04	Pa*s/kg
Third approximating coefficient:	1.097e+07	Pa*s <sup>2</sup> /(kg*m <sup>3</sup> )
Fourth approximating coefficient:	2.136e+05	Pa*s <sup>2</sup> /(kg*m <sup>3</sup> )
Correction factor:	0.8	
Pump design delivery:	130	lpm
Reference angular velocity:	1.77e+03	rpm
Reference density:	920	kg/m <sup>3</sup>
Leak resistance:	1e+08	Pa/(m <sup>3</sup> /s)
Drive shaft torque:	0.1	N*m
Torque-pressure coefficient:	1e-06	N*m/Pa

OK Cancel Help Apply

**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 1D characteristics: P-Q and N-Q

Reference angular velocity: 1.77e+03 rpm

Reference density: 920 kg/m<sup>3</sup>

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

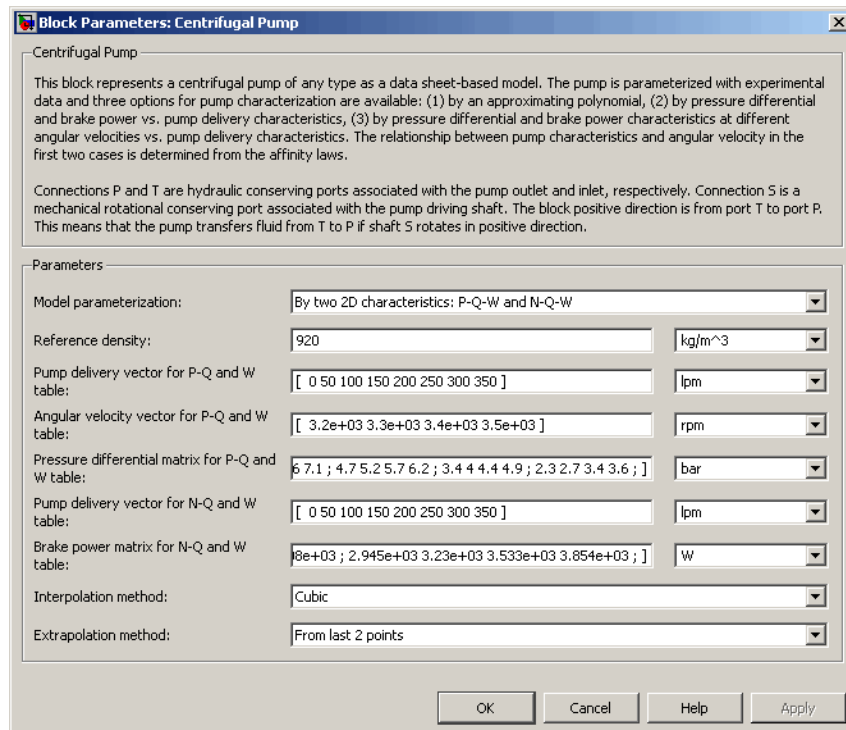
Brake power vector for N-Q table: [ 220 280 310 360 390 420 480 500 550 ] W

Interpolation method: Cubic

Extrapolation method: From last 2 points

OK Cancel Help Apply

# Centrifugal Pump



## 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<sup>3</sup>). 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<sup>2</sup>/(kg\*m<sup>3</sup>). 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<sup>2</sup>/(kg\*m<sup>3</sup>). This parameter is used if **Model parameterization** is set to By approximating polynomial.

# Centrifugal Pump

---

## **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<sup>3</sup>.

## **Leak resistance**

Leakage resistance coefficient (see Equation 1-4). The default value is 1e+8 Pa/(m<sup>3</sup>/s). This parameter is used if **Model parameterization** is set to By approximating polynomial.

## **Drive shaft torque**

The friction torque on the shaft at zero velocity. The default value is 0.1 N\*m. This parameter is used if **Model parameterization** is set to By approximating polynomial.

## **Torque-pressure coefficient**

The coefficient that provides relationship between torque and pump pressure. The default value is 1e-6 N\*m/Pa. This



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

## **Pump delivery vector for P-Q table**

Specify the vector of pump deliveries, as a one-dimensional array, to be used together with the vector of pressure differentials to specify the P-Q pump characteristic. The vector values must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in lpm, are [0 28 90 130 154 182]. This parameter is used if **Model parameterization** is set to By two 1D characteristics: P-Q and N-Q.

## **Pressure differential across pump vector**

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

## **Pump delivery vector for N-Q table**

Specify the vector of pump deliveries, as a one-dimensional array, to be used together with the vector of the pump brake power to specify the N-Q pump characteristic. The vector values must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in lpm, are [0 20 40 60 80 100 120 140 160]. This parameter is used if **Model parameterization** is set to By two 1D characteristics: P-Q and N-Q.

## **Brake power vector for N-Q table**

Specify the vector of pump brake power as a one-dimensional array. The vector will be used together with the pump delivery

# Centrifugal Pump

---

vector to specify the N-Q pump characteristic. The vector must be of the same size as the pump delivery vector for the N-Q table. The default values, in W, are [220 280 310 360 390 420 480 500 550]. This parameter is used if **Model parameterization** is set to By two 1D characteristics: P-Q and N-Q.

## **Pump delivery vector for P-Q and W table**

Specify the vector of pump deliveries, as a one-dimensional array, to be used together with the vector of angular velocities and the pressure differential matrix to specify the pump P-Q-W characteristic. The vector values must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in lpm, are [0 50 100 150 200 250 300 350]. This parameter is used if **Model parameterization** is set to By two 2D characteristics: P-Q-W and N-Q-W.

## **Angular velocity vector for P-Q and W table**

Specify the vector of angular velocities, as a one-dimensional array, to be used for calculating both the pump P-Q-W and N-Q-W characteristics. The vector values must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in rpm, are [3.2e+03 3.3e+03 3.4e+03 3.5e+03]. This parameter is used if **Model parameterization** is set to By two 2D characteristics: P-Q-W and N-Q-W.

## **Pressure differential matrix for P-Q and W table**

Specify the pressure differentials across pump as an m-by-n matrix, where m is the number of the P-Q-W pump delivery values and n is the number of angular velocities. This matrix will define the pump P-Q-W characteristic together with the pump delivery and angular velocity vectors. Each value in the matrix specifies pressure differential for a specific combination of pump delivery

and angular velocity. The matrix size must match the dimensions defined by the pump delivery and angular velocity vectors. The default values, in bar, are:

```
[ 8.3 8.8 9.3 9.9 ;  
  7.8 8.3 8.8 9.4 ;  
  7.2 7.6 8.2 8.7 ;  
  6.5 7 7.5 8 ;  
  5.6 6.1 6.6 7.1 ;  
  4.7 5.2 5.7 6.2 ;  
  3.4 4 4.4 4.9 ;  
  2.3 2.7 3.4 3.6 ; ]
```

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

### **Pump delivery vector for N-Q and W table**

Specify the vector of pump deliveries, as a one-dimensional array, to be used together with the vector of angular velocities and the brake power matrix to specify the pump N-Q-W characteristic. The vector values must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in lpm, are [0 50 100 150 200 250 300 350]. This parameter is used if **Model parameterization** is set to By two 2D characteristics: P-Q-W and N-Q-W.

### **Brake power matrix for N-Q and W table**

Specify the pump brake power as an m-by-n matrix, where m is the number of the N-Q-W pump delivery values and n is the number of angular velocities. This matrix will define the pump N-Q-W characteristic together with the pump delivery and angular velocity vectors. Each value in the matrix specifies brake power for a specific combination of pump delivery and angular velocity. The matrix size must match the dimensions defined by the pump

# Centrifugal 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 Polynomial (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 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 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**

# Centrifugal Pump

---

- **Extrapolation method**

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

## Global Parameters

Parameter determined by the type of working fluid:

- **Fluid density**

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

## Ports

The block has the following ports:

T

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

P

Hydraulic conserving port associated with the pump outlet.

S

Mechanical rotational conserving port associated with the pump driving shaft.

## References

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

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

## See Also

Fixed-Displacement Pump

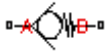
Variable-Displacement Pressure-Compensated Pump

Variable-Displacement Pump

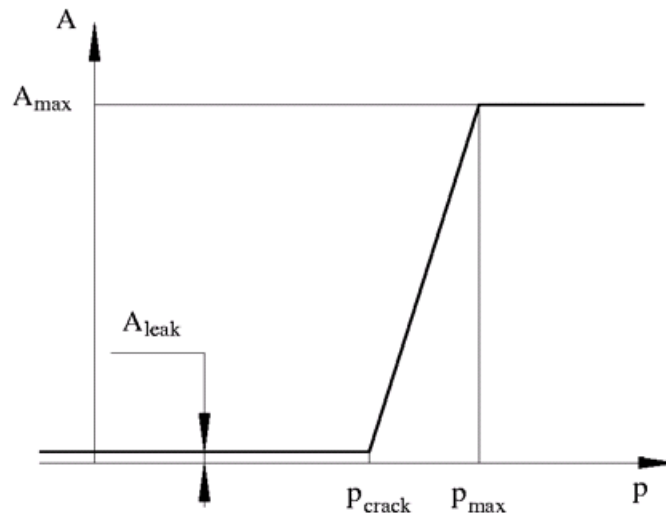
**Purpose** Hydraulic valve that allows flow in one direction only

**Library** Directional Valves

**Description**



The Check Valve block represents a hydraulic check valve as a data-sheet-based model. The purpose of the check valve is to permit flow in one direction and block it in the opposite direction. The following figure shows the typical dependency between the valve passage area  $A$  and the pressure differential across the valve  $p = p_A - p_B$ .



The valve remains closed while pressure differential across the valve is lower than the valve cracking pressure. When cracking pressure is reached, the valve control member (spool, ball, poppet, etc.) is forced off its seat, thus creating a passage between the inlet and outlet. If the flow rate is high enough and pressure continues to rise, the area is further increased until the control member reaches its maximum. At this moment, the valve passage area is at its maximum. The valve

# Check Valve

---

maximum area and the cracking and maximum pressures are generally provided in the catalogs and are the three key parameters of the block.

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

The model accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number ( $Re$ ) and comparing its value with the critical Reynolds number ( $Re_{cr}$ ). The flow rate is determined according to the following equations:

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

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

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

$$p = p_A - p_B$$

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



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

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

where

$q$	Flow rate through the valve
$p$	Pressure differential across the valve
$p_A, p_B$	Gauge pressures at the block terminals
$C_D$	Flow discharge coefficient
$A(p)$	Instantaneous orifice passage area
$A_{max}$	Fully open valve passage area
$A_{leak}$	Closed valve leakage area
$p_{crack}$	Valve cracking pressure
$p_{max}$	Pressure needed to fully open the valve
$D_H$	Instantaneous orifice hydraulic diameter
$\rho$	Fluid density
$\nu$	Fluid kinematic viscosity

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

# Check Valve

## Basic Assumptions and Limitations

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

## Dialog Box and Parameters

**Block Parameters: Check Valve**

Check Valve

This block represents hydraulic check valve as a data sheet-based model. The purpose of the check valve is to permit flow in one direction and block it in the opposite direction. The valve remains closed while pressure differential across the valve is lower than the valve cracking pressure. When cracking pressure is reached, the valve control member is forced off its seat, thus creating a passage between the inlet and outlet. If the flow rate is high enough and pressure continues to rise, the area is further increased until the control member reaches its maximum.

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

Parameters

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

OK Cancel Help Apply

### Maximum passage area

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

**Cracking pressure**

Pressure level at which the orifice of the valve starts to open. The default value is  $3e4$  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 simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12$  m<sup>2</sup>.

**Global Parameters**

Parameters determined by the type of working fluid:

- Fluid density
- Fluid kinematic viscosity

# Check Valve

---

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 example illustrates the use of check valves to build a rectifier that keeps the flow passing through a flow control valve always in the same direction, and to select an appropriate orifice depending on the flow direction.

## See Also

Pilot-Operated Check Valve

## Purpose

Hydraulic reservoir where pressurization and fluid level remain constant regardless of volume change

## Library

Low-Pressure Blocks

## Description



The Constant Head Tank block represents a pressurized hydraulic reservoir, in which fluid is stored under a specified pressure. The size of the tank is assumed to be large enough to neglect the pressurization and fluid level change due to fluid volume. The block accounts for the fluid level elevation with respect to the tank bottom, as well as for pressure loss in the connecting pipe that can be caused by a filter, fittings, or some other local resistance. The loss is specified with the pressure loss coefficient. The block computes the volume of fluid in the tank and exports it outside through the physical signal port V.

The fluid volume value does not affect the results of simulation. It is introduced merely for information purposes. It is possible for the fluid volume to become negative during simulation, which signals that the fluid volume is not enough for the proper operation of the system. By viewing the results of the simulation, you can determine the extent of the fluid shortage.

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

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

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

$$p_{loss} = K \frac{\rho}{2A_p^2} q |q|$$

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

where

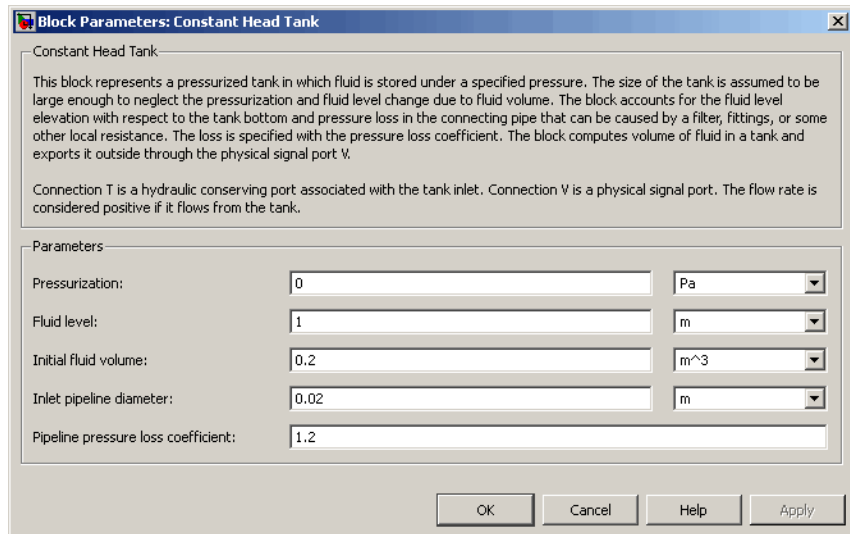
# Constant Head Tank

---

$p$	Pressure at the tank inlet
$p_{elev}$	Pressure due to fluid level
$p_{loss}$	Pressure loss in the connecting pipe
$p_{pr}$	Pressurization
$\rho$	Fluid density
$g$	Acceleration of gravity
$H$	Fluid level with respect to the bottom of the tank
$K$	Pressure loss coefficient
$A_p$	Connecting pipe area
$d$	Connecting pipe diameter
$q$	Flow rate

Connection T is a hydraulic conserving port associated with the tank inlet. Connection V is a physical signal port. The flow rate is considered positive if it flows out of the tank.

## Dialog Box and Parameters



### Pressurization

Gage pressure acting on the surface of the fluid in the tank. It can be created by a gas cushion, membrane, bladder, or piston, as in bootstrap reservoirs. This parameter must be greater than or equal to zero. The default value is 0, which corresponds to a tank connected to atmosphere.

### Fluid level

The fluid level with respect to the tank bottom. This parameter must be greater than zero. The default value is 1 m.

### Initial fluid volume

The initial volume of fluid in the tank. This parameter must be greater than zero. The default value is 0.2 m<sup>3</sup>.

### Inlet pipeline diameter

The diameter of the connecting pipe. This parameter must be greater than zero. The default value is 0.02 m.

# Constant Head Tank

---

## Pipeline pressure loss coefficient

The value of the pressure loss coefficient, to account for pressure loss in the connecting pipe. This parameter must be greater than zero. The default value is 1.2.

For reasons of computational robustness, the loss is computed with the equation similar to that used in the Fixed Orifice block:

$$q = \frac{1}{\sqrt{K}} A_p \sqrt{\frac{2}{\rho} p_{loss}}$$

The Critical Reynolds number is set to 15.

## Ports

The block has the following ports:

T

Hydraulic conserving port associated with the tank inlet.

V

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

## See Also

Reservoir

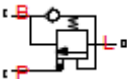
Variable Head Tank



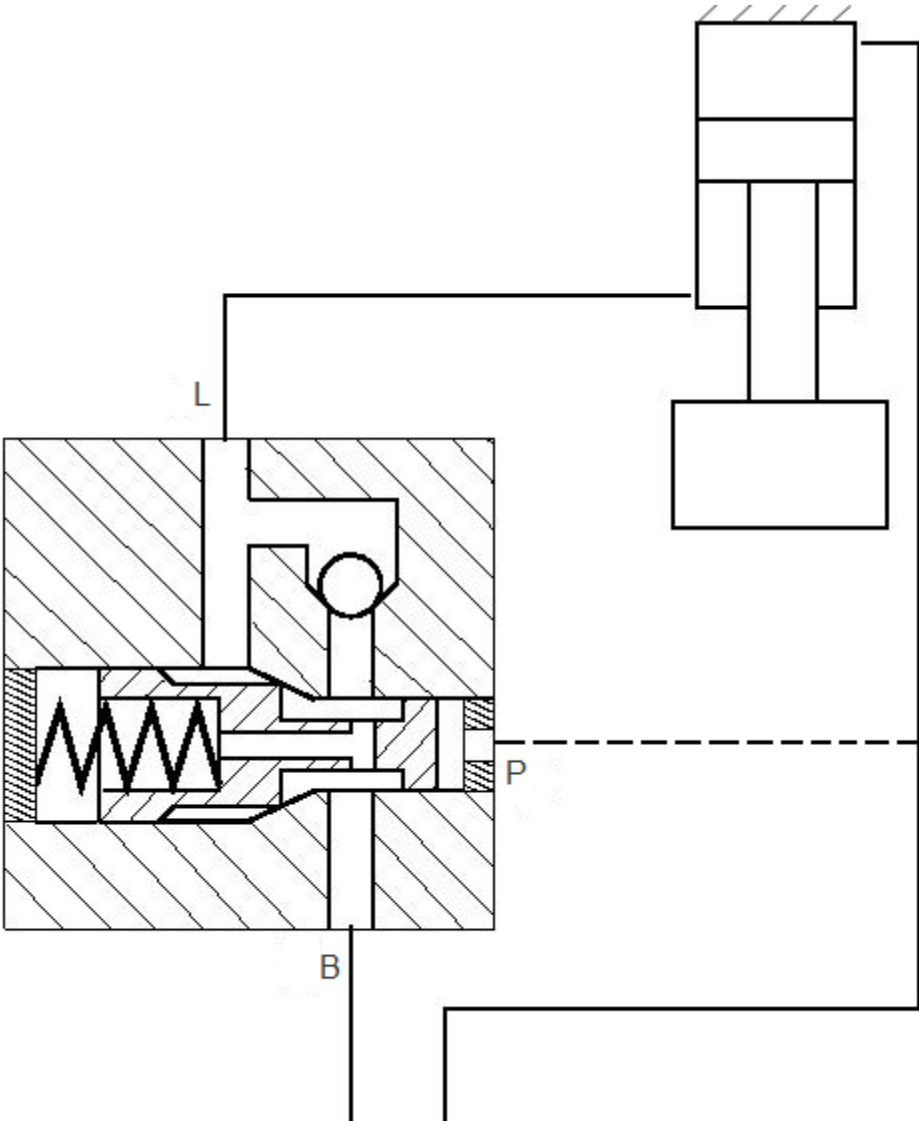
**Purpose** Hydraulic counterbalance valve

**Library** Flow Control Valves

**Description** Counterbalance valves are used in hydraulic systems working with overriding (running-away) or suspended load. They are designed to create backpressure at the return line of the actuator to prevent losing control over the load. The following illustration shows a counterbalance valve schematic.



# Counterbalance Valve



If a directional valve (not shown in the picture) is shifted into position that lowers the load, then the fluid from the rod chamber of the cylinder can exit only if pressure at port P (pilot pressure) and port L (load pressure) create enough force to overcome the spring force and open the valve. In statics, the valve is described with the equation

$$F_0 + c \cdot x = p_{pilot} \cdot A_{pilot} + p_{load} \cdot A_{load} - p_{back} \cdot A_{back} \quad (1-5)$$

where

$F_0$	Spring setting
$c$	Spring rate
$x$	Valve opening
$p_{pilot}$	Pilot pressure (pressure at port P)
$p_{load}$	Load pressure (pressure at port L)
$p_{back}$	Backpressure (pressure at return port B)
$A_{pilot}$	Valve effective area at pilot port P
$A_{load}$	Valve effective area at load port L
$A_{back}$	Valve effective area at return port B

Counterbalance valve, classified by type, is an internally-externally piloted valve because both the pilot pressure and the load pressure tend to open the valve. After minor rearrangements, Equation 1-5 takes the form

$$p_{set} + c_p \cdot x = p_{pilot} \cdot k_{pilot} + p_{load} - p_{back} \cdot k_{back} \quad (1-6)$$

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

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

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

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

# Counterbalance Valve

---

where

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

The valve displacement is determined from Equation 1-6

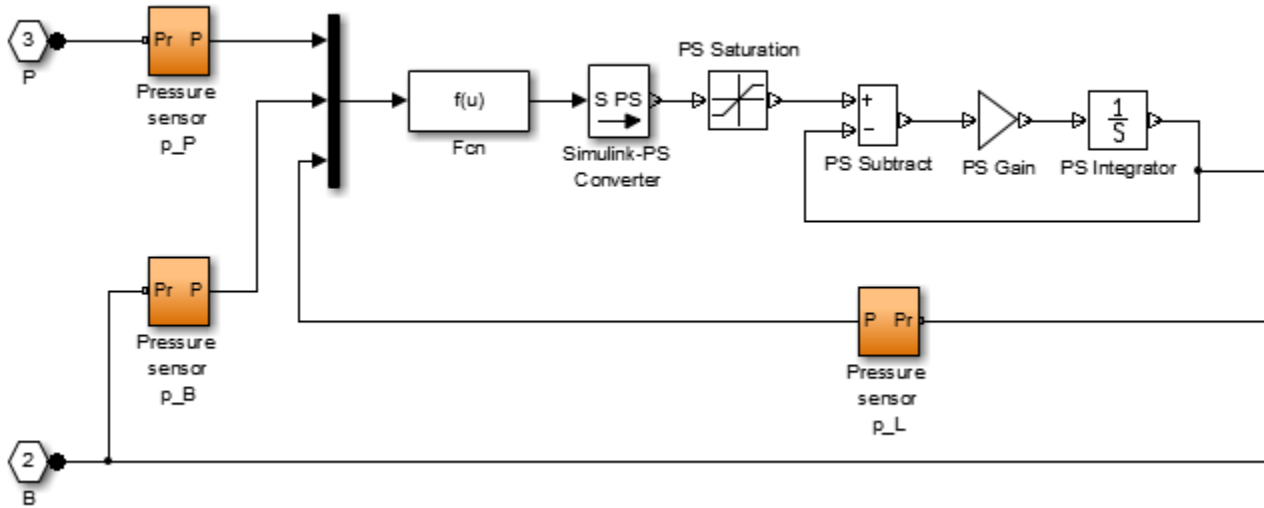
$$x = \left( p_{set} - (p_{pilot} \cdot k_{pilot} + p_{load} - p_{back} \cdot k_{back}) \right) / c_p \quad (1-7)$$

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

where  $x_{max}$  is the maximum valve displacement.

The Counterbalance Valve block can be represented as the following structural model.

# Counterbalance Valve



The pressure sensors measure pressure at respective ports and convey their values to the Fcn block, which, together with the PS Saturation block, performs calculations in accordance with Equation 1-7. The valve displacement is passed through the first order lag block, built of the PS Subtract, PS Gain, and PS Integrator blocks, to account for valve dynamics. The gain of the PS Gain block is set to  $1/T$ , where  $T$  is the time constant. The Variable Orifice and Check Valve blocks simulate the counterbalance valve orifice and check valve. In the actual Counterbalance Valve block model, the operations performed by the sensors and the Fcn block are executed in the block equation section.

The Counterbalance Valve block is essentially a data-sheet-based model. Depending on data listed in the manufacturer's catalogs or data sheets for your particular valve, you can choose one of the following model parameterization options:

# Counterbalance Valve

---

- **By maximum area and opening** — Use this option if the data sheet provides only the orifice maximum area and the control member maximum stroke.
- **By area vs. opening table** — Use this option if the catalog or data sheet provides a table of the orifice passage area based on the control member displacement.

In the latter case, the PS Saturation block in the structural model is replaced with the PS Lookup Table (1D) block, and you can choose from three interpolation and two extrapolation methods.

Connections L and B are hydraulic conserving ports associated with the load and backpressure ports of the valve. The hydraulic conserving port P is associated with the pilot port. The block positive direction is from port L to port B. Positive pressure at port P opens the valve.

## **Assumptions and Limitations**

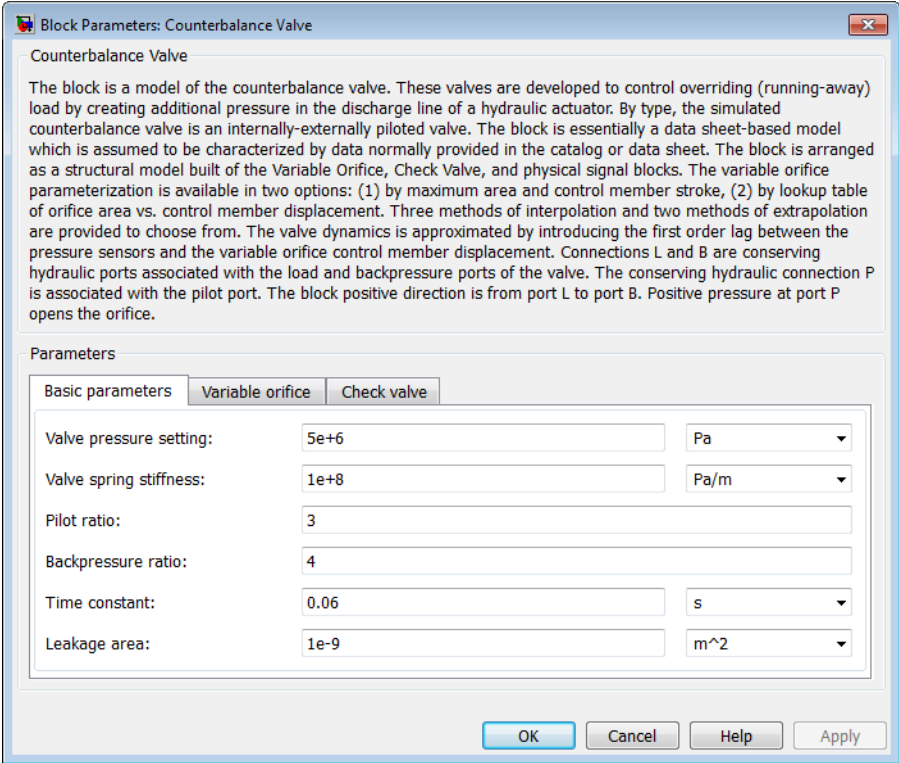
- Valve dynamics are approximated by introducing the first order lag between the pressure sensors and the variable orifice control member displacement.
- Inertia, friction, or hydraulic forces acting on the valve control member are not taken into account.

## **Dialog Box and Parameters**

The block dialog box contains three tabs:

- “Basic Parameters” on page 1-201
- “Variable Orifice” on page 1-203
- “Check Valve” on page 1-208

## Basic Parameters



### Valve pressure setting

The parameter specifies the pressure at port L necessary to start opening the valve. The setting is controlled with the valve spring. The default value is 50e5 Pa.

### Valve spring stiffness

The valve spring pressure stiffness  $c_p$ . The default value is 1e8 Pa/m.

# Counterbalance Valve

---

## **Pilot ratio**

The ratio between the effective areas of the control member face in the pilot chamber and in the load chamber. This is one of the fundamental valve characteristics in a catalog or data sheet. The default value is 3.

## **Backpressure ratio**

The ratio between the effective area of the control member, onto which the backpressure acts, and the load chamber valve area. There is a wide variety of counterbalance valves with compensated or partially-compensated backpressure. The parameter can take zero value if the valve is completely compensated. The default value is 4.

## **Time constant**

The valve dynamics are approximated with the first order lag. This parameter is the time constant of the lag. The default value is 0.06 s.

## **Leakage area**

The total area of possible leaks in the completely closed orifice and check valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the orifice or check valve is completely closed. 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  $1e-9 \text{ m}^2$ .



## Variable Orifice

**Block Parameters: Counterbalance Valve**

Counterbalance Valve

The block is a model of the counterbalance valve. These valves are developed to control overriding (running-away) load by creating additional pressure in the discharge line of a hydraulic actuator. By type, the simulated counterbalance valve is an internally-externally piloted valve. The block is essentially a data sheet-based model which is assumed to be characterized by data normally provided in the catalog or data sheet. The block is arranged as a structural model built of the Variable Orifice, Check Valve, and physical signal blocks. The variable orifice parameterization is available in two options: (1) by maximum area and control member stroke, (2) by lookup table of orifice area vs. control member displacement. Three methods of interpolation and two methods of extrapolation are provided to choose from. The valve dynamics is approximated by introducing the first order lag between the pressure sensors and the variable orifice control member displacement. Connections L and B are conserving hydraulic ports associated with the load and backpressure ports of the valve. The conserving hydraulic connection P is associated with the pilot port. The block positive direction is from port L to port B. Positive pressure at port P opens the orifice.

Parameters

Basic parameters Variable orifice Check valve

Variable orifice parameterization: By maximum area and opening

Orifice maximum area: 8e-5 m<sup>2</sup>

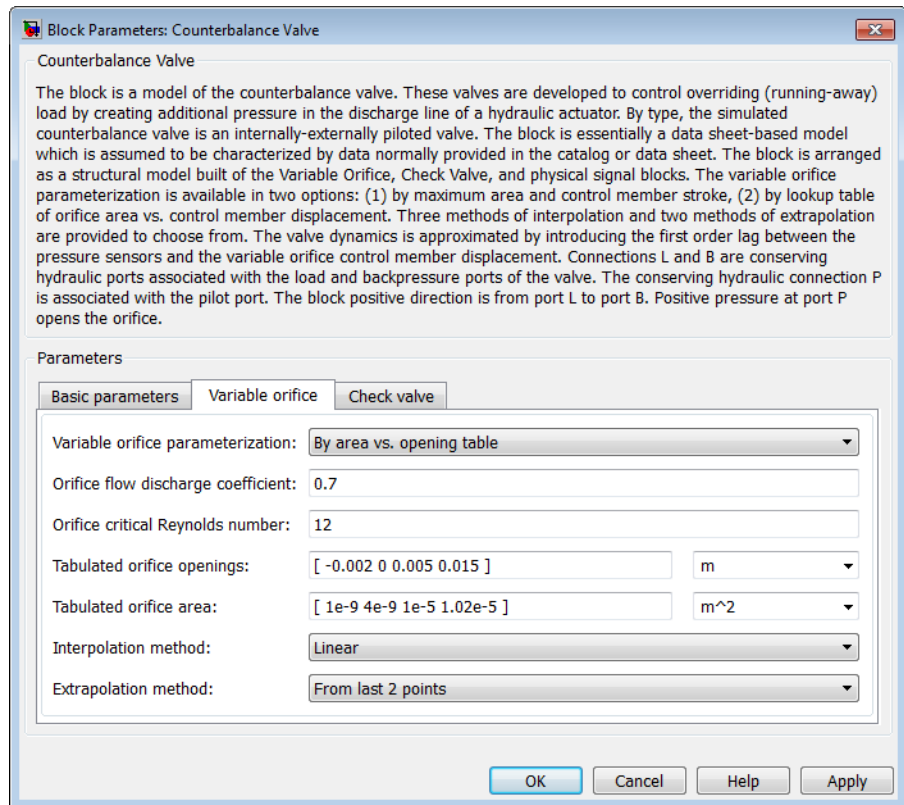
Orifice maximum opening: 0.005 m

Orifice flow discharge coefficient: 0.7

Orifice critical Reynolds number: 12

OK Cancel Help Apply

# Counterbalance Valve



## Variable orifice parameterization

Select one of the following methods for specifying the orifice:

- By maximum area and opening — Provide values for the maximum orifice area and the maximum orifice opening. The passage area is linearly dependent on the control member displacement, that is, the orifice is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.

- **By area vs. opening table** — Provide tabulated data of orifice openings and corresponding orifice areas. The passage area is determined by one-dimensional table lookup. You have a choice of 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  $8e-5 \text{ m}^2$ . This parameter is used if **Variable orifice parameterization** is set to **By maximum area and opening**.

### **Orifice maximum opening**

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is  $5e-4 \text{ m}$ . This parameter is used if **Variable orifice parameterization** is set to **By maximum area and opening**.

### **Orifice flow discharge coefficient**

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

### **Orifice 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.

### **Tabulated orifice openings**

Specify the vector of input values for orifice openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default

# Counterbalance Valve

---

values, in meters, are  $[-2e-3, 0, 5e-3, 15e-3]$ . This parameter is used if **Variable orifice parameterization** is set to **By area vs. opening table**. **Tabulated orifice openings** values will be used together with **Tabulated orifice area** values for one-dimensional table lookup.

## Tabulated orifice area

Specify the vector of orifice areas as a one-dimensional array. The vector must be of the same size as the orifice openings vector. All the values must be positive. The default values, in  $m^2$ , are  $[1e-12, 4e-12, 1.e-5, 1.02e-5]$ . This parameter is used if **Variable orifice parameterization** is set to **By area vs. opening table**.

## Interpolation method

This parameter is used if **Variable orifice parameterization** is set to **By area vs. opening table**. Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- **Linear** — Uses a linear interpolation function.
- **Cubic** — Uses the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP).
- **Spline** — Uses the cubic spline interpolation algorithm.

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

## Extrapolation method

This parameter is used if **Variable orifice parameterization** is set to **By area vs. opening table**. Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- **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

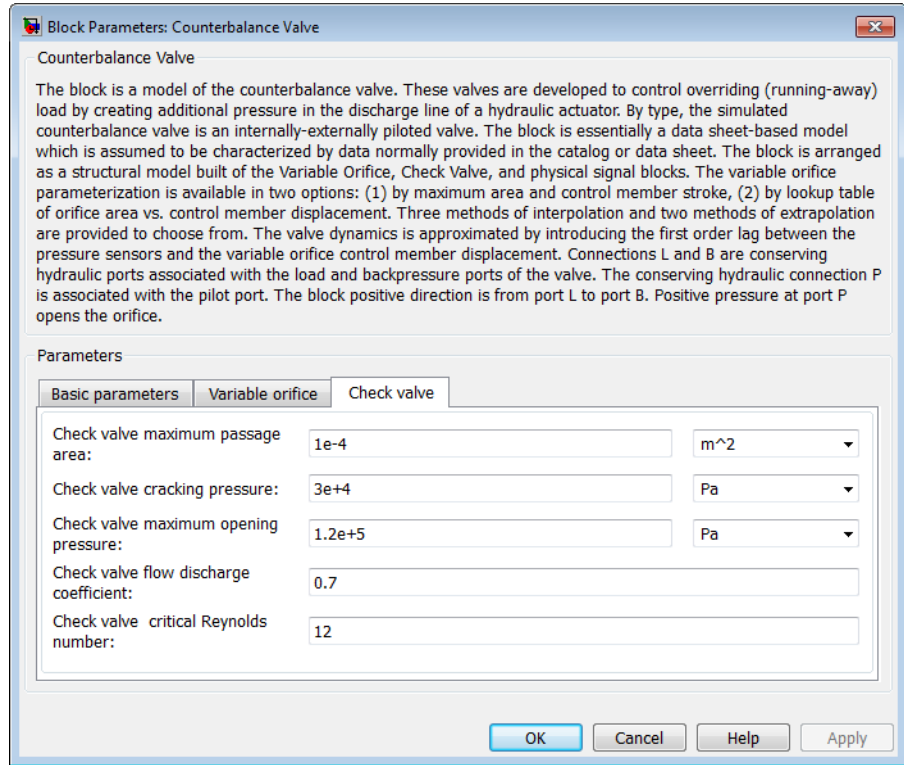
values if the input value is below the specified range, and the two last specified output values if the input value is above the specified range.

- **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.

# Counterbalance Valve

## Check Valve



### Check valve maximum passage area

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

### Check valve cracking pressure

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

## **Check valve maximum opening pressure**

Pressure differential across the valve needed to fully open the valve. Its value must be higher than the cracking pressure. The default value is  $1.2 \times 10^5$  Pa.

## **Check valve flow discharge coefficient**

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

## **Check valve 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.

## **Restricted Parameters**

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

- **Variable orifice parameterization**
- **Interpolation method**
- **Extrapolation method**

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

## **Global Parameters**

Parameters determined by the type of working fluid:

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

# Counterbalance Valve

---

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

## Ports

The block has the following ports:

P

Hydraulic conserving port associated with the valve pilot port.

L

Hydraulic conserving port associated with the valve load port.

B

Hydraulic conserving port associated with the valve backpressure port.

## Examples

The Hydraulic System with Counterbalance Valve example illustrates the use of counterbalance valve to balance the load of a double-acting cylinder controlled by an open-center 4-way directional valve.

## See Also

Check Valve

Variable Orifice



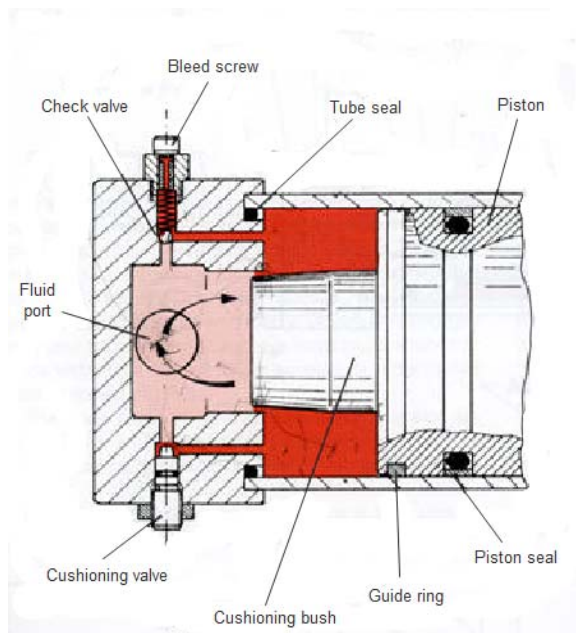
**Purpose** Cushion in hydraulic cylinders

**Library** Hydraulic Cylinders

**Description**



The Cylinder Cushion block models a hydraulic cylinder cushion, the device that decelerates the cylinder rod near the end of the stroke by restricting the flow rate leaving the cylinder chamber. The figure shows a typical design of a cylinder cushion [1].

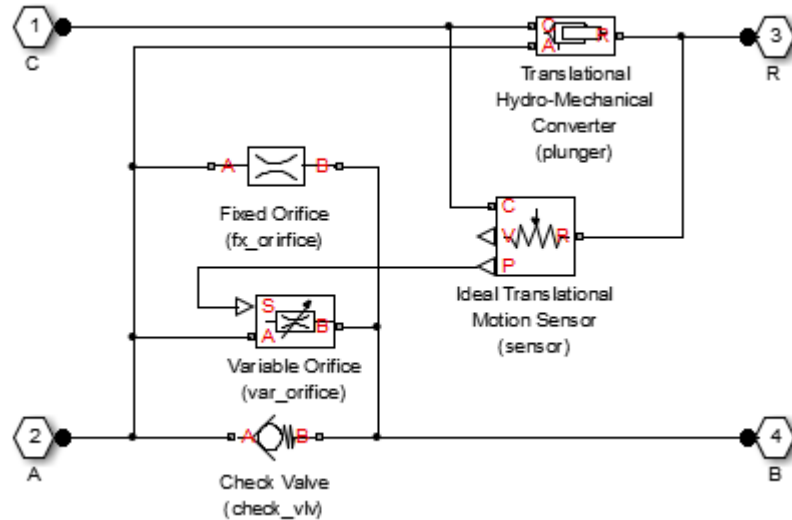


As the piston moves toward the cap (to the left in the figure), the cushioning bush enters the chamber in the cap and creates an additional resistance to the fluid leaving the chamber. The bush profile determines the desired deceleration. Near the end of the stroke, the fluid flows through the gap between the bush and the cap and through the cushioning valve with constant cross-sectional area. The check valve

# Cylinder Cushion

located between the chambers allows free flow to the piston chamber to ease the piston breakaway from the end position.

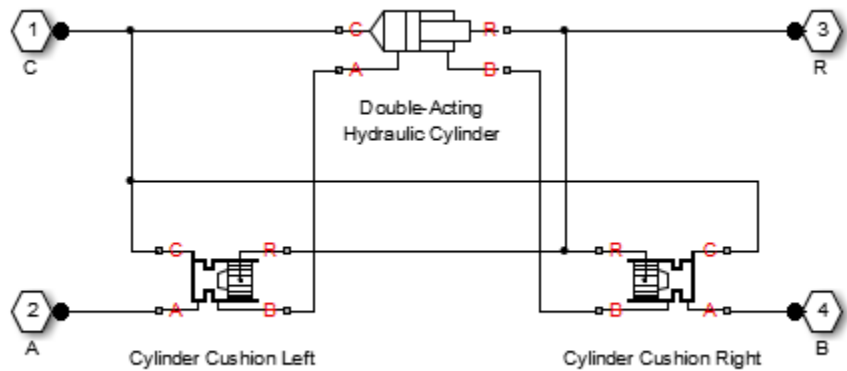
The block is implemented as a structural model that replicates a cushioning device, as shown in this diagram.



The Variable Orifice block represents a variable gap between the bush and the cavity machined in the end cap. The lookup table of the Variable Orifice block implements the relationship between the orifice area and the piston displacement. The Fixed Orifice and the Check Valve blocks simulate the cushioning valve and the check valve installed between chambers. The Translational Hydro-Mechanical Converter represents a plunger created by the bush and the cavity. The Ideal Translational Motion Sensor block monitors the piston displacement and conveys it (with the initial piston position accounted) to the Variable Orifice block. The names assigned to the nested blocks in the model are shown in parentheses.

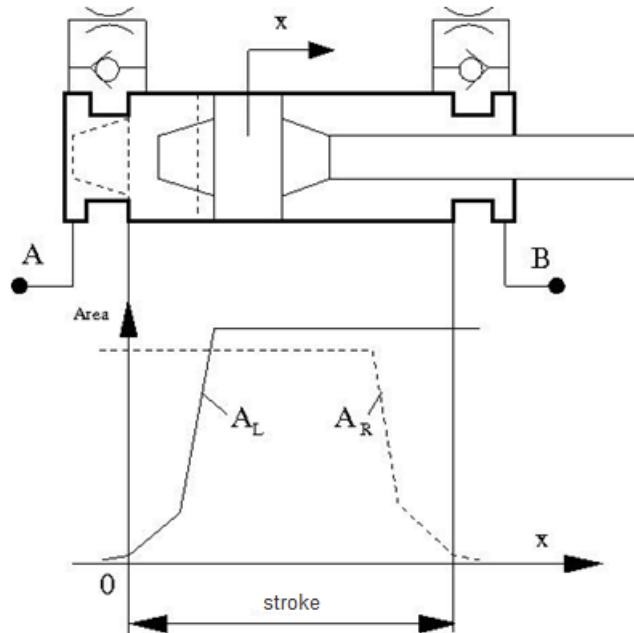
The block develops a cushioning effect for the flow rate from port B to port A. The check valve in the block is oriented from port A to port B.

You can use this block with any of the cylinder blocks in the library to model a single-acting or double-acting cylinder with cushion. The following diagram shows the model of a double-sided hydraulic cylinder with cushion built from a Double-Acting Hydraulic Cylinder block and two Cylinder Cushion blocks.



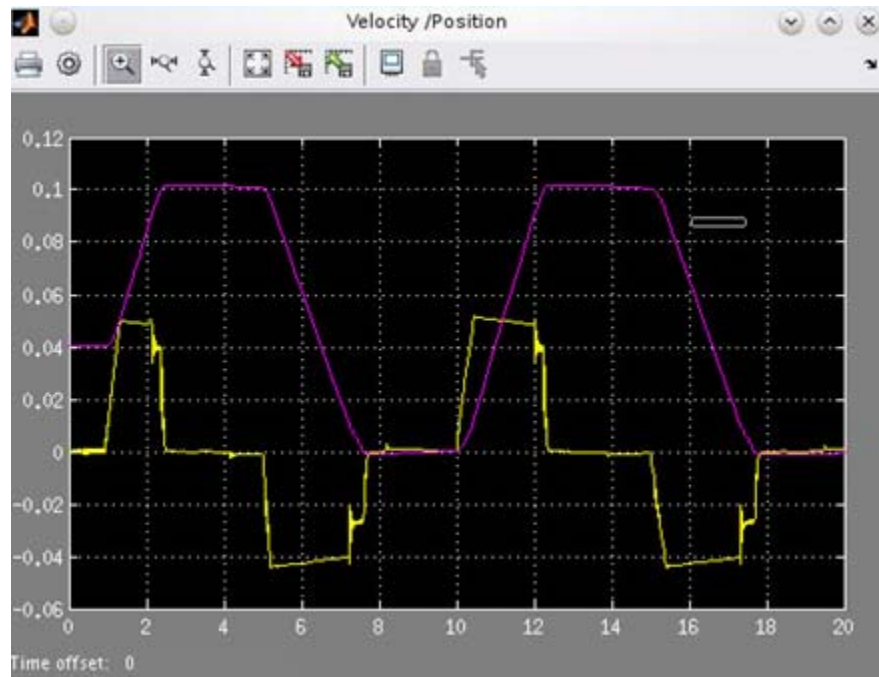
You can adjust the cushioning effect by changing the area of the fixed orifice and the profile of the cushioning bush (variable orifice). Specify the profile using the one-dimensional lookup table of the orifice area versus piston displacement. The next figure shows a typical configuration of a double-acting cylinder with the double-sided cushioning, similar to the model shown in the block diagram above.

# Cylinder Cushion



To ensure cushioning on both sides of the stroke, set the variable orifice area of the left cushion ( $A_L$ ) and the right cushion ( $A_R$ ) similar to the profile shown in the figure. The origin of the plot is located at the position where the piston touches the cap. If the cylinder acts in the negative direction, the piston displacements are negative, and you must make the profile specification in the fourth quadrant.

The following figure shows a typical motion diagram of a cylinder with the double-sided cushioning.

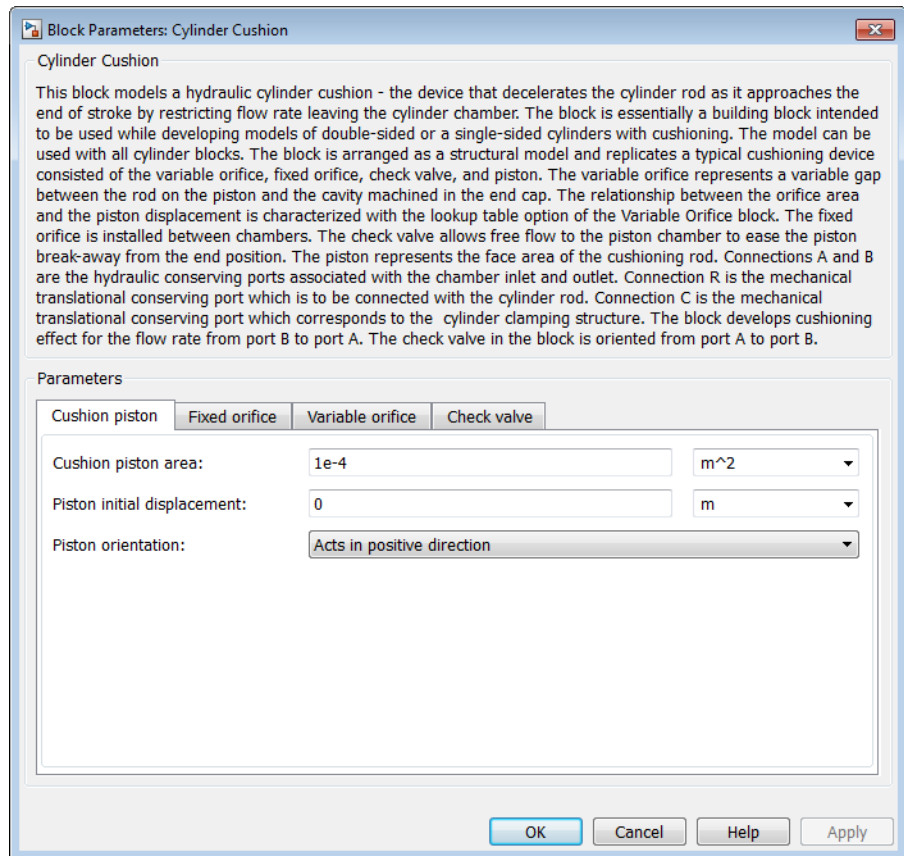


The cushions are set to provide deceleration at  $\sim 10$  mm before the end of the stroke. The stroke of the cylinder is 10 cm, and the initial position of the piston is 0.04 m. The plot shows the velocity (yellow line) and motion (magenta line) profiles.

Connections A and B are hydraulic conserving ports associated with the device hydraulic inlet and outlet. Connection R is a mechanical translational conserving port that connects to the cylinder rod. Connection C is a mechanical translational conserving port that connects to the cylinder clamping structure.

# Cylinder Cushion

## Dialog Box and Parameters



- “Cushion Piston Tab” on page 1-217
- “Fixed Orifice Tab” on page 1-217
- “Variable Orifice Tab” on page 1-218
- “Check Valve Tab” on page 1-220

## Cushion Piston Tab

### Cushion piston area

Addition of the cushion to a cylinder converts the respective cylinder into two cylinders, rigidly connected and acting in parallel, with the total effective area equal to the piston area before addition. This parameter sets the area of the cushion piston, which is the face area of the cushion bush. The default value is  $1e-4 \text{ m}^2$ .

### Piston initial displacement

The distance between the cylinder piston and cap A at the start of simulation. The default value is 0, which corresponds to the fully retracted position of the piston.

### Piston orientation

Piston orientation with respect to the globally assigned positive direction. Since the cushion piston is part of the cylinder piston, its orientation must be the same as the orientation of the cylinder piston at the side the cushion is attached to. Similar to a cylinder model, if pressure applied at port A exerts force in the negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

## Fixed Orifice Tab

### Fixed orifice area

The area of the fixed orifice installed between cushion chambers. The default value is  $1e-6 \text{ m}^2$ .

### Fixed orifice flow discharge coefficient

Semi-empirical coefficient that is used in the computation of flow rate through the fixed orifice. The default value is 0.7.

### Fixed orifice critical Reynolds number

The maximum Reynolds number for laminar flow through the fixed orifice. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on the orifice

geometrical profile. You can find the recommendations on the parameter value in hydraulic textbooks. The default value is 10.

## **Variable Orifice Tab**

### **Tabulated piston displacements**

Vector of input values for piston displacements, specified as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The **Tabulated piston displacements** values are used together with **Tabulated orifice area** values for one-dimensional table lookup. Due to the nature of the cylinder hard stops, the piston can move below zero and above the stroke value. It is good practice to account for piston deformation and provide piston displacements beyond the ideal stroke range to avoid extrapolation. The default values, in mm, are [-2, 10, 11, 89, 90, 102].

### **Tabulated orifice area**

Vector of orifice areas, specified as a one-dimensional array. The vector must be the same size as the piston displacements vector. All the values must be positive. The default values, in  $\text{cm}^2$ , are [0.004, 0.006, 9, 9, 9, 9].

### **Variable orifice interpolation method**

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

- **Linear** — For one-dimensional table lookup (By area vs. opening table), uses a linear interpolation function. For two-dimensional table lookup (By pressure-flow characteristic), uses a bilinear interpolation algorithm, which is an extension of linear interpolation for functions in two variables.



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

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

### **Variable orifice extrapolation method**

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

- **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.

# Cylinder Cushion

---

## **Variable orifice flow discharge coefficient**

Semi-empirical coefficient that is used in the computation of flow rate through the variable orifice. The default value is 0.7.

## **Variable orifice critical Reynolds number**

The maximum Reynolds number for laminar flow through the variable orifice. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on the orifice geometrical profile. You can find the recommendations on the parameter value in hydraulic textbooks. The default value is 10.

## **Variable orifice leakage area**

The total area of possible leaks in the completely closed orifice. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## **Check Valve Tab**

### **Check valve maximum area**

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

### **Check valve cracking pressure**

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

### **Check valve full opening pressure**

Pressure differential across the valve needed to fully open the valve. Its value must be higher than the cracking pressure. The default value is  $1.5e5 \text{ Pa}$ .

### **Check valve flow discharge coefficient**

Semi-empirical coefficient that is used in the computation of flow rate through the check valve. The default value is 0.7.

## **Check valve critical Reynolds number**

The maximum Reynolds number for laminar flow through the check valve. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The default value is 10.

## **Check valve leakage area**

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## **Restricted Parameters**

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

- **Piston orientation**
- **Variable orifice interpolation method**
- **Variable orifice extrapolation method**

All other block parameters are available for modification.

## **Global Parameters**

Parameters determined by the type of working fluid:

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

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

## **Ports**

The block has the following ports:

# Cylinder Cushion

---

- A Hydraulic conserving port connected to the cylinder inlet.
- B Hydraulic conserving port connected to the cylinder outlet.
- R Mechanical translational conserving port connected to the cylinder rod.
- C Mechanical translational conserving port connected to the cylinder clamping structure.

## References

[1] Rohner, P. *Industrial Hydraulic Control*. Fourth edition. Brisbane : John Wiley & Sons, 1995.

## See Also

Check Valve  
Fixed Orifice  
Variable Orifice

**Purpose** 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:

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

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

where

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

# Cylinder Friction

$v$  Relative velocity in the contact

$f_{vfr}$  Viscous friction coefficient

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

$$F = K \square v$$

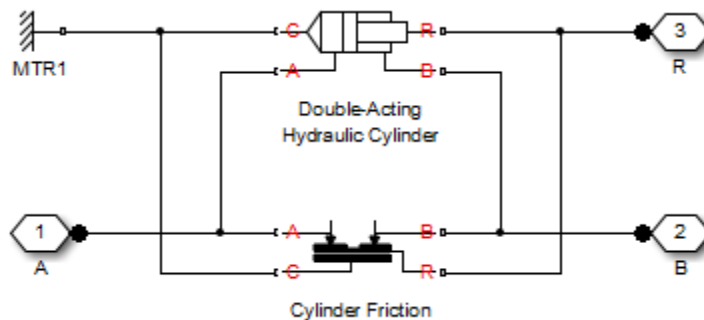
$$K = \frac{F_C (1 + (K_{brk} - 1) \square \exp(-c_v v_{th})) + f_{vfr} \square v_{th}}{v_{th}}$$

where

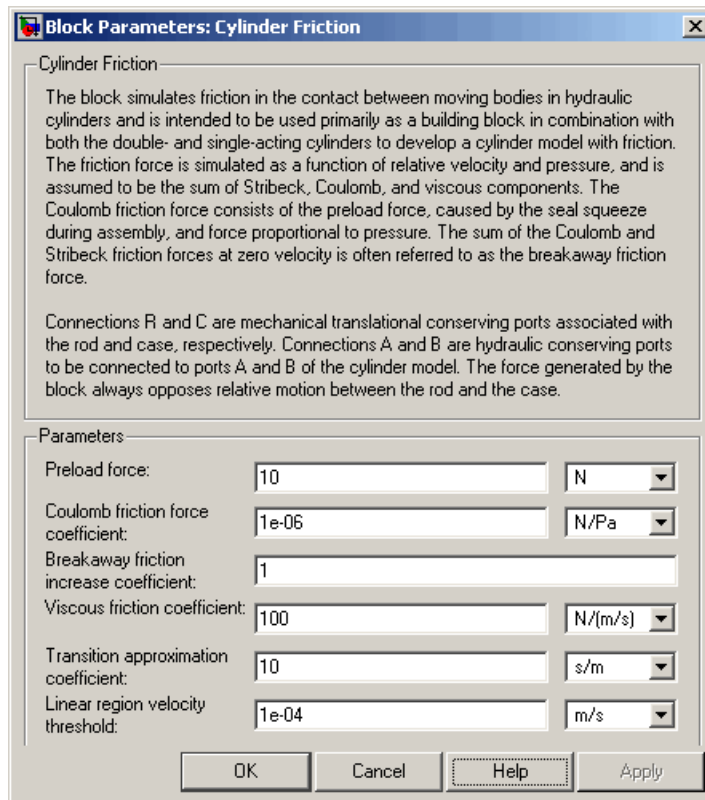
$K$  Proportionality coefficient

$v_{th}$  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.



## Dialog Box and Parameters



### Preload force

The preload force, caused by the seal squeeze during assembly. The default value is 10 N.

### Coulomb friction force coefficient

Coulomb friction coefficient, which defines the proportionality between the Coulomb friction force and the pressure in cylinder chambers. The default value is 1e-6 N/Pa.

# Cylinder Friction

---

## **Breakaway friction increase coefficient**

The friction force increase over the Coulomb friction. The Coulomb friction force, multiplied by this coefficient, is referred to as breakaway friction force. The default value is 1.

## **Viscous friction coefficient**

Proportionality coefficient between the viscous friction force and the relative velocity. The parameter value must be greater than or equal to zero. The default value is 100 N/(m/s).

## **Transition approximation coefficient**

The parameter sets the value of coefficient  $c_v$ , which is used for the approximation of the transition between the breakaway and the Coulomb frictions. Its value is assigned based on the following considerations: the Stribeck friction component reaches approximately 5% of its steady-state value at velocity  $3/c_v$ , and 2% at velocity  $4/c_v$ , which makes it possible to develop an approximate relationship  $c_v \approx 4/v_{min}$ , where  $v_{min}$  is the relative velocity at which friction force has its minimum value. By default,  $c_v$  is set to 10 s/m, which corresponds to a minimum friction at velocity of about 0.4 m/s.

## **Linear region velocity threshold**

The parameter sets the small vicinity near zero velocity, within which friction force is considered to be linearly proportional to the relative velocity. MathWorks recommends that you use values in the range between  $1e-6$  and  $1e-4$  m/s. The default value is  $1e-4$  m/s.

## **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** Hydraulic actuator exerting force in both directions

**Library** Hydraulic Cylinders

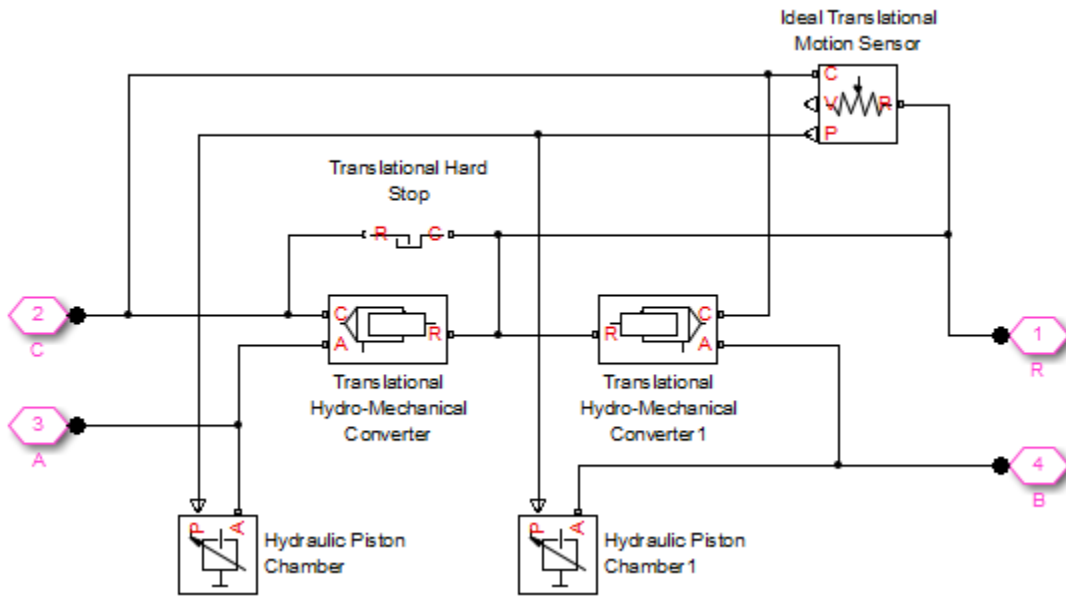
## Description



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, with the exception of the two Translational Hydro-Mechanical Converter blocks. These custom hydro-mechanical converters take into account pressure-induced density variations, unlike the Foundation library blocks, and therefore give increased accuracy for mass conservation in fluid compressibility computations. The schematic diagram of the model is shown below.

# Double-Acting Hydraulic Cylinder



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 Hydraulic 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 displacement from initial position. The displacement is necessary to control the Hydraulic Piston Chamber blocks.

# Double-Acting Hydraulic Cylinder

---

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

## **Basic Assumptions and Limitations**

- No leakage, internal or external, is taken into account.
- No loading on piston rod, such as inertia, friction, spring, and so on, is taken into account. If necessary, you can easily add them by connecting an appropriate building block to cylinder port R.

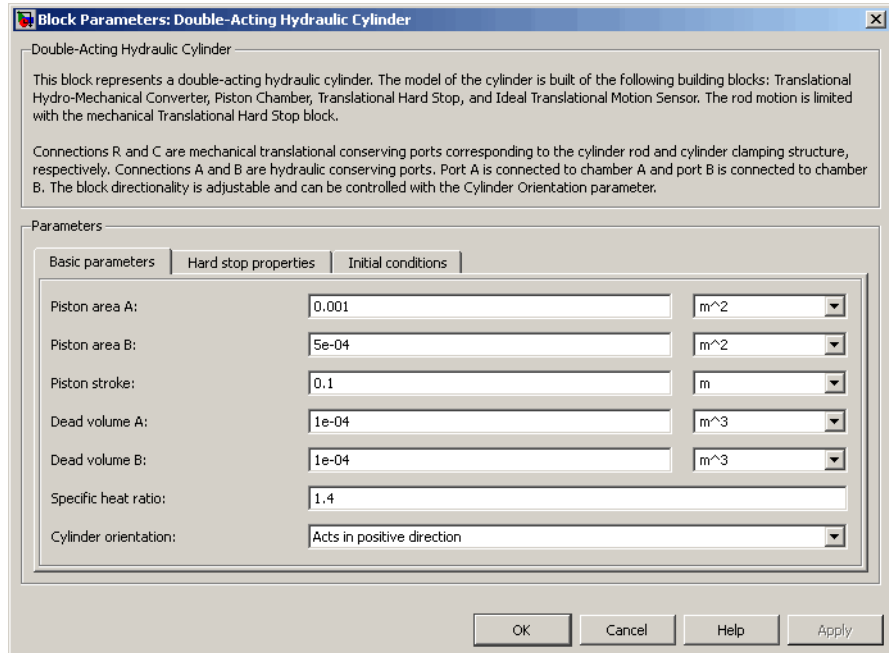
## **Dialog Box and Parameters**

The block dialog box contains three tabs:

- “Basic Parameters” on page 1-231
- “Hard Stop Properties” on page 1-233
- “Initial Conditions” on page 1-234

# Double-Acting Hydraulic Cylinder

## Basic Parameters



### Piston area A

Chamber A effective piston area. The default value is 0.001 m<sup>2</sup>.

### Piston area B

Chamber B effective piston area. The default value is 5e-5 m<sup>2</sup>.

### Piston stroke

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

### Dead volume A

Fluid volume in chamber A that remains in the chamber after the rod is fully retracted. The default value is 1e-4 m<sup>3</sup>.

# Double-Acting Hydraulic Cylinder

---

## **Dead volume B**

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

## **Specific heat ratio**

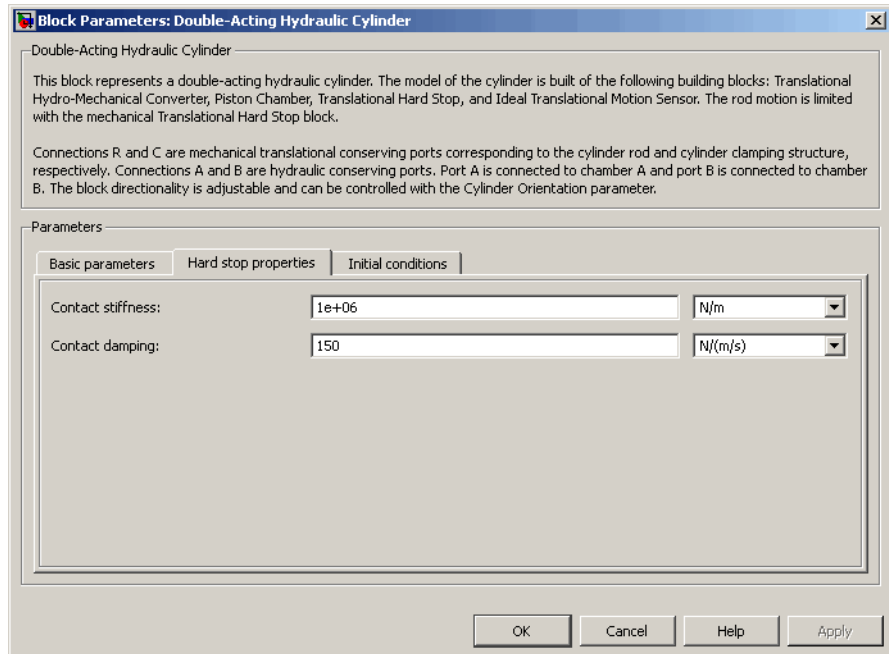
Gas-specific heat ratio for the Hydraulic Piston Chamber blocks. The default value is 1.4.

## **Cylinder orientation**

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

# Double-Acting Hydraulic Cylinder

## Hard Stop Properties



### 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.

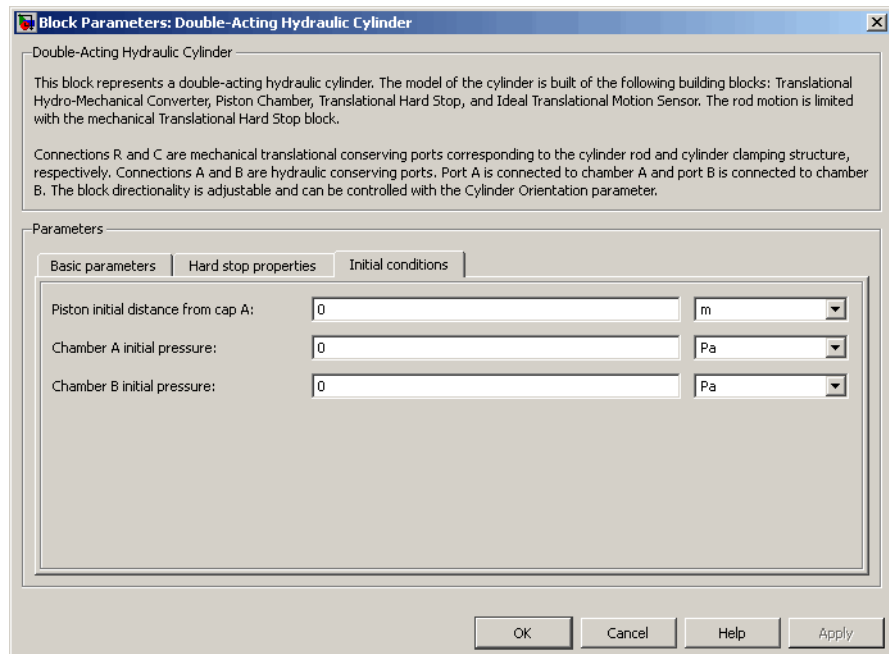
### Contact damping

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

# Double-Acting Hydraulic Cylinder

slider is in contact with the stop, including the period when slider is pulled back from the contact. For computational efficiency and convergence reasons, MathWorks recommends that you assign a nonzero value to this parameter. The default value is 150 N\*s/m.

## Initial Conditions



### Piston initial distance from cap A

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

### Chamber A initial pressure

Pressure in the cylinder chamber A at the beginning of simulation. The default value is 0.



# Double-Acting Hydraulic Cylinder

---

## Chamber B initial pressure

Pressure in the cylinder chamber B at the beginning of simulation.  
The default value is 0.

## Restricted Parameters

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

- **Cylinder orientation**

All other block parameters are available for modification.

## Global Parameters

Parameter determined by the type of working fluid:

- **Fluid bulk modulus**

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the cylinder chamber A.

B

Hydraulic conserving port associated with the cylinder chamber B.

R

Mechanical translational conserving port associated with the cylinder rod.

C

Mechanical translational conserving port associated with the cylinder clamping structure.

## Examples

The Double-Acting Hydraulic Cylinder with Flexible Clamping example illustrates simulation of a cylinder whose clamping is too flexible to

# Double-Acting Hydraulic Cylinder

---

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 example demonstrates the use of a 4-way valve in combination with a double-acting cylinder in a simple closed-loop actuator. The example shows how to connect the blocks and set the initial orifice openings for the 4-way valve to model the forward and return strokes of the cylinder under load.

## See Also

Single-Acting Hydraulic Cylinder

Ideal Translational Motion Sensor

Translational Hard Stop

Translational Hydro-Mechanical Converter

Hydraulic Piston Chamber

# Double-Acting Hydraulic Cylinder (Simple)

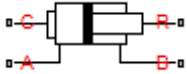
## Purpose

Basic functionality of double-acting hydraulic cylinder

## Library

Hydraulic Cylinders

## Description



The Double-Acting Hydraulic Cylinder (Simple) block represents a simplified version of a double-acting hydraulic cylinder, developed for applications where only the basic cylinder functionality must be reproduced, in exchange for better numerical efficiency. For these reasons, such factors as fluid compressibility, friction, and leakages are assumed to be negligible. The hard stops are assumed to be fully inelastic, to eliminate any possible oscillations at the end of the stroke. The model is especially suitable for real-time and HIL (hardware-in-the-loop) simulation, if such simplifications are acceptable.

The model is described with the following equations:

$$F = A_A \cdot p_A - A_B \cdot p_B - F_c$$

$$q_A = A_A \cdot v$$

$$q_B = A_B \cdot v$$

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

$$v = v_R - v_C$$

$$F_c = \begin{cases} (x - x_E) \cdot K_p \cdot v & \text{if } x > x_E, v > 0 \\ (x - x_R) \cdot K_p \cdot v & \text{if } x < x_R, v < 0 \\ 0 & \text{otherwise} \end{cases}$$

$$x_E = S - x_0$$

# Double-Acting Hydraulic Cylinder (Simple)

---

$$x_R = -x_0$$

where

$F$	Force developed by the cylinder
$v$	Cylinder rod velocity
$v_R, v_C$	Absolute velocities of cylinder rod and cylinder case, respectively
$A_A$	Piston area at port A side
$A_B$	Piston area at port B side
$p_A$	Pressure at the cylinder port A
$p_B$	Pressure at the cylinder port B
$q_A$	Flow rate through port A into the cylinder
$q_B$	Flow rate through port B from the cylinder
$x$	Piston position
$x_0$	Initial distance between piston and cap A
$F_c$	Hard stop force
$x_E$	Distance the piston can travel to fully extend from initial position
$x_R$	Distance the piston can travel to fully retract from initial position
$K_p$	Penetration coefficient
$S$	Piston stroke

The main difference between this block and the Double-Acting Hydraulic Cylinder block is the representation of the hard stop:

- The Double-Acting Hydraulic Cylinder block utilizes the elastic model of the stop, which is essentially a combination of a spring and a damper, connected in parallel. The stiffness of the spring,

# Double-Acting Hydraulic Cylinder (Simple)

---

which represents contact stiffness of colliding bodies, is very high, resulting in high-frequency, low-magnitude oscillations at the moment the bodies collide. These oscillations can hardly be noticed experimentally, but they are difficult to process by a numerical simulation and generally decrease numerical efficiency.

- The hard stop in the Double-Acting Hydraulic Cylinder (Simple) block is represented with the inelastic model, which is essentially a viscous damper with the penetration-dependent damping coefficient. This coefficient is referred to as the penetration coefficient. With inelastic model, no oscillation is generated during an impact, resulting in improved numerical robustness and efficiency. But inelastic stop model has a feature that you must consider while selecting the model: colliding bodies continue slowly moving into each other as long as the contact is loaded with the compressing force. In real life, this phenomenon is similar to collision of two bodies separated by a sizable layer of viscous liquid. It takes some time to squeeze the liquid before bodies themselves come into contact.

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

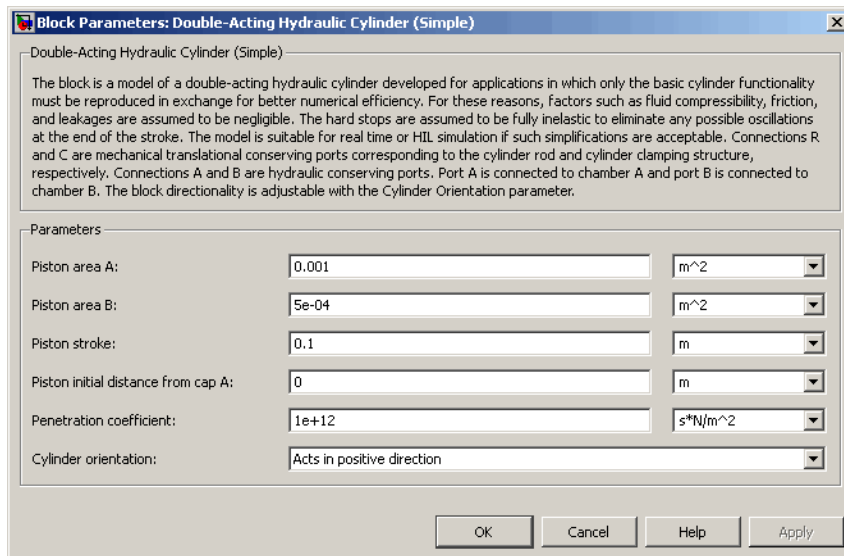
You can adjust the block directionality with the **Cylinder orientation** parameter.

## Basic Assumptions and Limitations

- Friction between moving parts is not taken into account.
- Inertia effects are not taken into account.
- Fluid compressibility is not taken into account.
- Leakage flows are assumed to be negligible.
- The hard stops are assumed to be fully inelastic, as explained above.

# Double-Acting Hydraulic Cylinder (Simple)

## Dialog Box and Parameters



### Piston area A

Chamber A effective piston area. The default value is 0.001 m<sup>2</sup>.

### Piston area B

Chamber B effective piston area. The default value is 5e-4 m<sup>2</sup>.

### Piston stroke

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

### Piston initial distance from cap A

The distance between the piston and cap A at the beginning of simulation. This value cannot exceed the piston stroke. The default value is 0.

### Penetration coefficient

Specifies the penetration property of colliding bodies. The higher the value of the coefficient, the less the bodies penetrate into each other, but the sharper the increase of the impact force. To find out the value of the coefficient for a particular application, it is recommended to run the same model with different hard stop

# Double-Acting Hydraulic Cylinder (Simple)

---

models (or with different cylinder models) until they show close results. The default value of the coefficient is  $1e12$  N/m/(m/s).

## Cylinder orientation

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

## Restricted Parameters

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

- **Cylinder orientation**

All other block parameters are available for modification.

## Ports

The block has the following ports:

- A Hydraulic conserving port associated with the cylinder chamber A.
- B Hydraulic conserving port associated with the cylinder chamber B.
- R Mechanical translational conserving port associated with the cylinder rod.
- C Mechanical translational conserving port associated with the cylinder clamping structure.

# Double-Acting Hydraulic Cylinder (Simple)

---

## **Examples**

The Hydraulic Closed-Loop Actuator with Fixed-Step Integration example shows how to use this cylinder along with other blocks optimized for real-time and HIL simulation.

## **See Also**

[Double-Acting Hydraulic Cylinder](#)

[Single-Acting Hydraulic Cylinder](#)

[Single-Acting Hydraulic Cylinder \(Simple\)](#)

[Translational Hard Stop](#)

[Translational Hydro-Mechanical Converter](#)

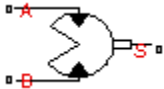


# Double-Acting Rotary Actuator

**Purpose** Double-acting hydraulic rotary actuator

**Library** Hydraulic Cylinders

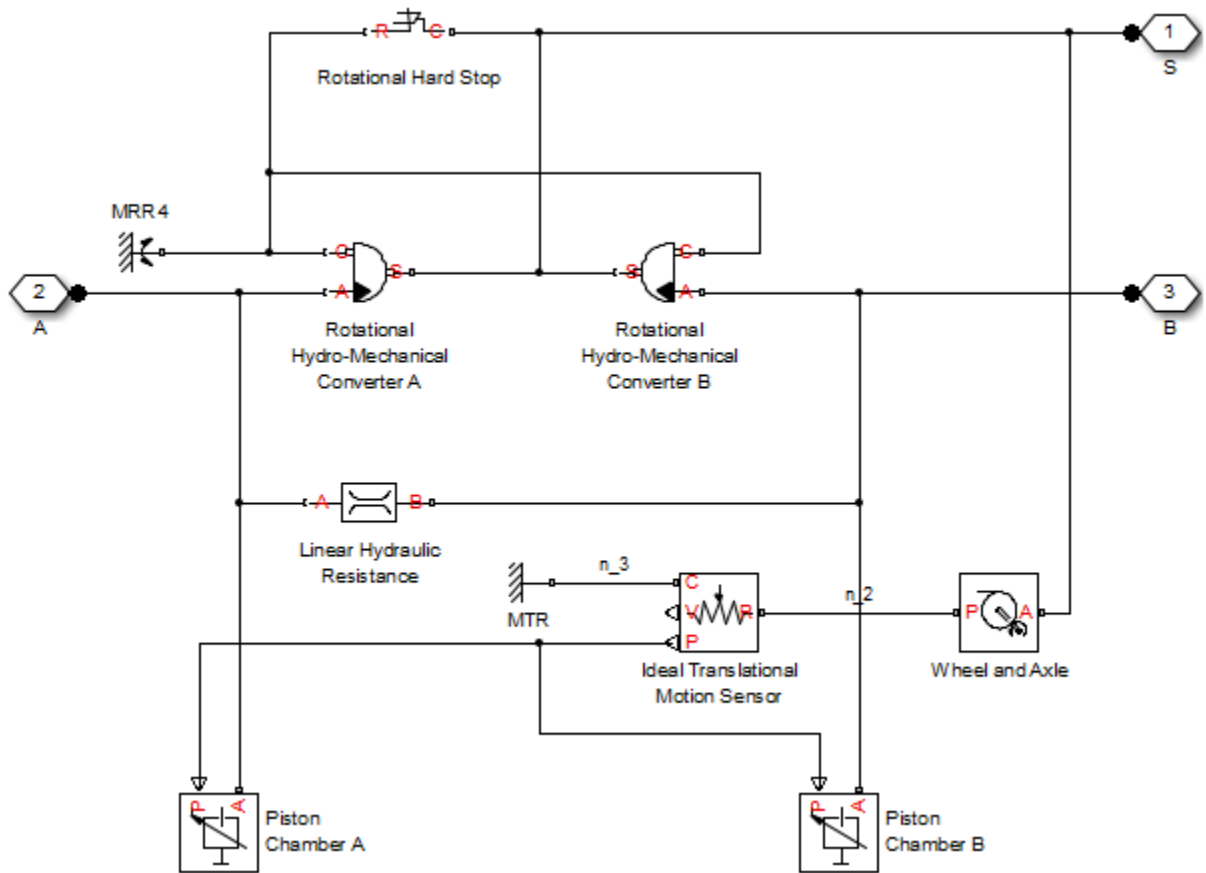
## Description



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, with the exception of the two Rotational Hydro-Mechanical Converter blocks. These custom hydro-mechanical converters take into account pressure-induced density variations, unlike the Foundation library blocks, and therefore give increased accuracy for mass conservation in fluid compressibility computations. The schematic diagram of the model is shown below.

# Double-Acting Rotary Actuator



The blocks in the diagram perform the following functions:

# Double-Acting Rotary Actuator

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 Hydraulic 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.

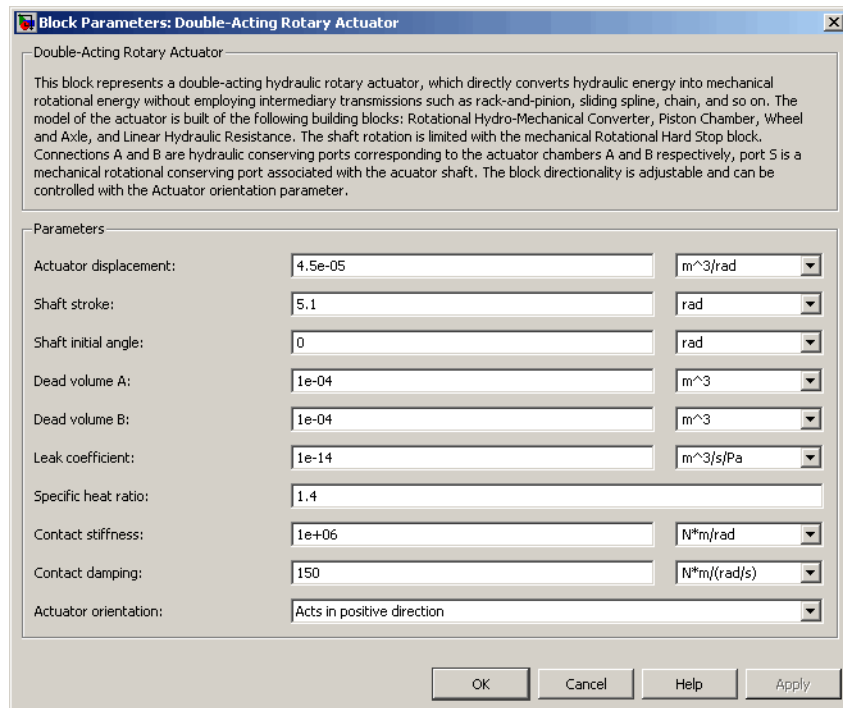
The block directionality is adjustable and can be controlled with the **Actuator orientation** parameter.

## Basic Assumptions and Limitations

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.

# Double-Acting Rotary Actuator

## Dialog Box and Parameters



### Actuator displacement

Effective displacement of the actuator. The default value is  $4.5e-5$  m<sup>3</sup>/rad.

### Shaft stroke

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

### Shaft initial angle

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

**Dead volume A**

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

**Dead volume B**

Fluid volume in chamber B that remains in the chamber when the shaft is positioned at the end of the stroke. The default value is  $1e-4 \text{ m}^3$ .

**Leak coefficient**

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

**Specific heat ratio**

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

**Contact stiffness**

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

**Contact damping**

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

**Actuator orientation**

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

# Double-Acting Rotary Actuator

---

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

## Restricted Parameters

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

- **Actuator orientation**

All other block parameters are available for modification.

## Global Parameters

Parameter determined by the type of working fluid:

- **Fluid bulk modulus**

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

## Ports

The block has the following ports:

- A  
Hydraulic conserving port associated with the actuator chamber A.
- B  
Hydraulic conserving port associated with the actuator chamber B.
- S  
Mechanical rotational conserving port associated with the actuator shaft.

## **See Also**

Ideal Translational Motion Sensor

Linear Hydraulic Resistance

Rotational Hard Stop

Rotational Hydro-Mechanical Converter

Hydraulic Piston Chamber

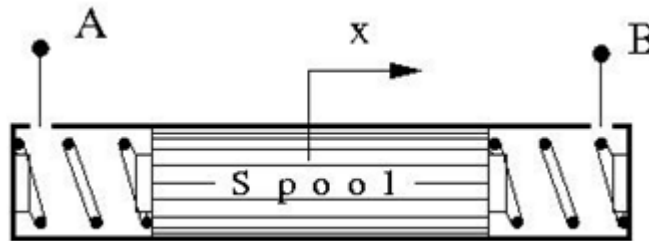
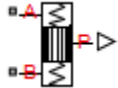
Wheel and Axle

# Double-Acting Servo Cylinder

**Purpose** Hydraulic double-acting servo cylinder with spring-centered plunger

**Library** Hydraulic Cylinders

**Description** The Double-Acting Servo Cylinder block models a hydraulic double-acting servo cylinder implemented as a spring-centered plunger.

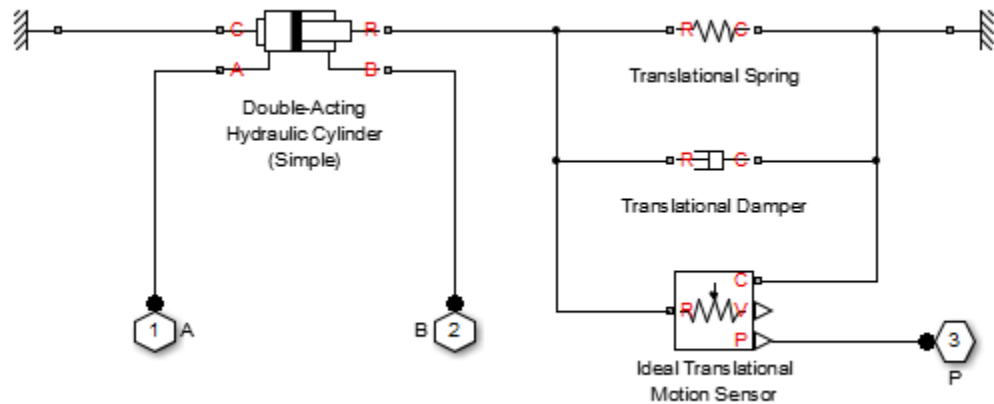


Changing pressure at ports A and B shifts the spool from the neutral position in proportion to the pressure differential. Use the servo cylinder for control purposes.

The model of the cylinder uses the Double-Acting Hydraulic Cylinder (Simple), Translational Spring, Translational Damper, and Ideal Translational Motion Sensor blocks. The following figure shows the schematic diagram of the model.



# Double-Acting Servo Cylinder



The plunger is located initially in the middle of the stroke. The plunger displacement is computed with the following equations:

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

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

where

- $p_A, p_B$  Pressures at port A and B, respectively
- $A$  Spool face area
- $k$  Spring rate
- $B$  Damping coefficient
- $F_{HS}$  Hard stop force
- $K_p$  Stop penetration coefficient

# Double-Acting Servo Cylinder

---

$S$	Piston stroke
$v$	Piston velocity
$x$	Piston displacement

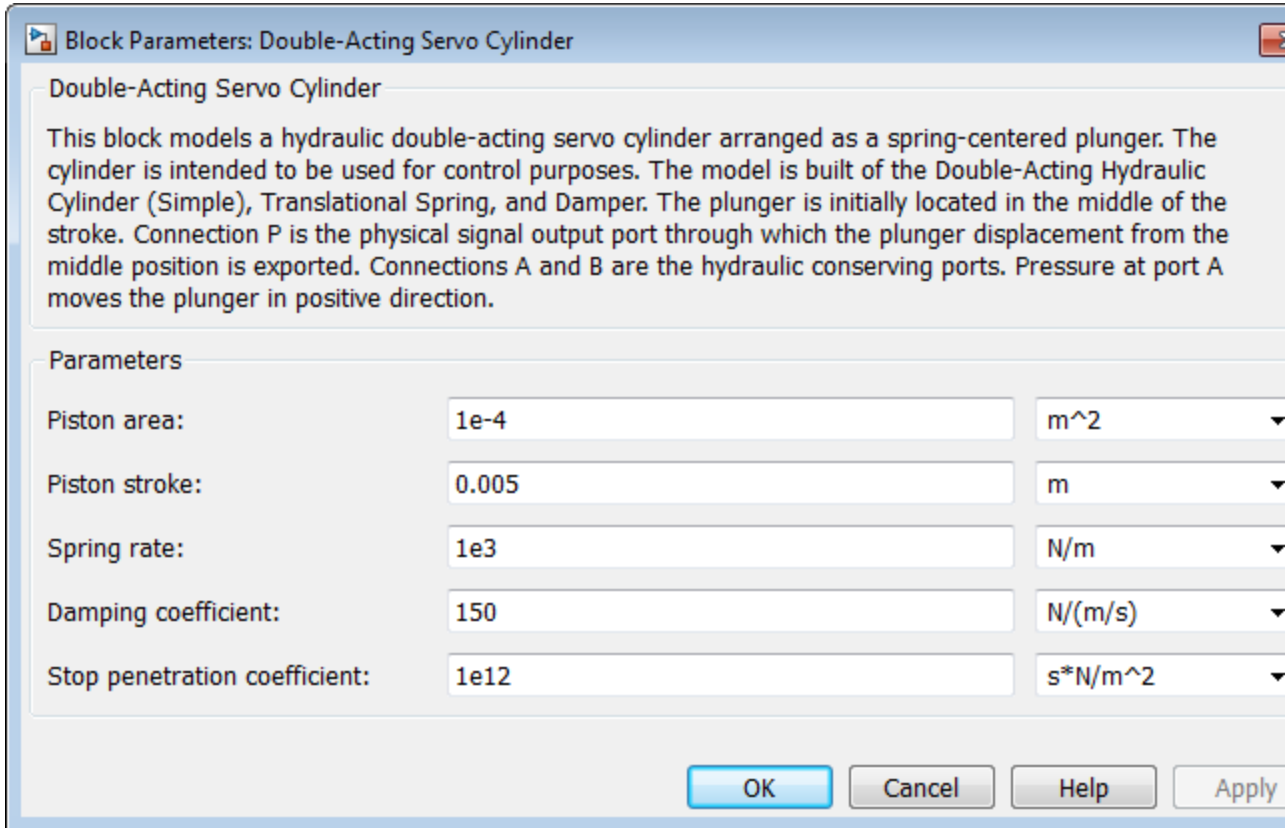
Connection P is a physical signal output port that outputs the plunger displacement from the central position. Connections A and B are hydraulic conserving ports. Pressure at port A moves the plunger in the positive direction.

## **Basic Assumptions and Limitations**

- Friction between moving parts is not taken into account.
- Inertia effects are not taken into account.
- Fluid compressibility is not taken into account.
- Leakage flows are assumed to be negligible.
- The hard stops are assumed to be fully inelastic. For details, see the Double-Acting Hydraulic Cylinder (Simple) block reference page.

# Double-Acting Servo Cylinder

## Dialog Box and Parameters



### Piston area

The face area of the piston. The default value is  $1e-4$  m<sup>2</sup>.

### Piston stroke

The full piston stroke, from one hard stop to another. The piston is located initially in the middle of the stroke and can travel half a

# Double-Acting Servo Cylinder

---

stroke in the positive and negative direction. The default value is 0.005 m.

## **Spring rate**

The spring rate of the centering springs. The default value is 1000 N/m.

## **Damping coefficient**

The damping coefficient in the contact between the piston and the case. The default value is 150 N/(m/s).

## **Stop penetration coefficient**

The penetration property of the piston hard stop. The hard stop is represented as absolutely inelastic, and the property is characterized by the penetration coefficient. The default value of the coefficient is  $1e12$  N/m/(m/s).

## **Ports**

The block has the following ports:

A

Hydraulic conserving port associated with the cylinder chamber A.

B

Hydraulic conserving port associated with the cylinder chamber B.

P

Physical signal output port that outputs the piston displacement.

## **See Also**

Double-Acting Hydraulic Cylinder (Simple)

Ideal Translational Motion Sensor

Translational Damper

Translational Spring

**Purpose** 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
$\rho$	Fluid density

The flow regime is checked in the underlying Local Resistance block by comparing the Reynolds number to the specified critical Reynolds number value. For laminar flow regime, the formula for pressure loss computation is modified, as described in the reference documentation for the Local Resistance block.

# Elbow

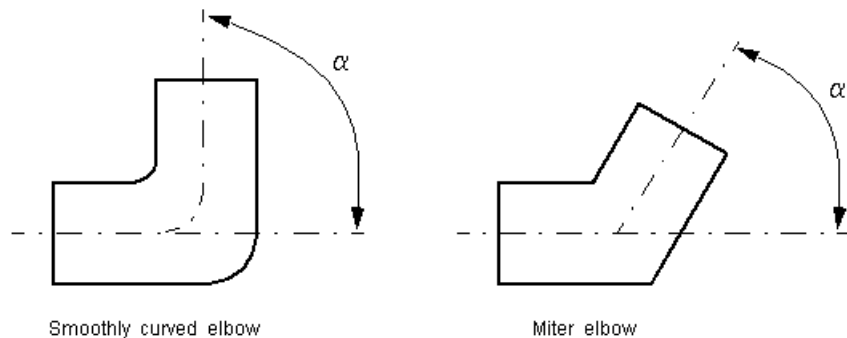
The core data for the pressure loss coefficient computation is the table-specified relationship between the friction factor  $f_T$  and the internal diameter for clean commercial steel pipes, with flow in the zone of complete turbulence (see [1], p. A-26). For smoothly curved, standard 90° elbows, the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

$$K = 30f_T$$

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

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

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



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

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

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

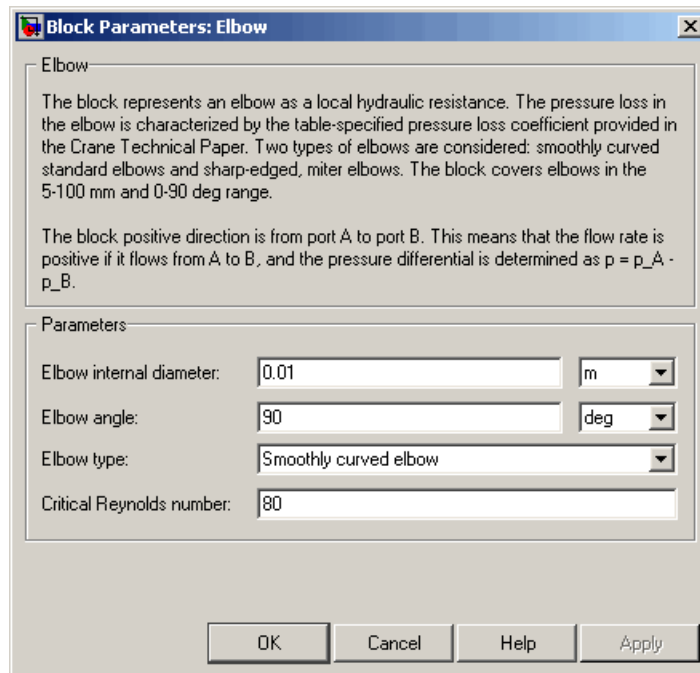
**The formulas used in the Elbow block are very approximate, especially in the laminar and transient flow regions. For more accurate results, use the Local Resistance block with a table-specified  $K=f(Re)$  relationship.**

### Basic Assumptions and Limitations

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

# Elbow

## Dialog Box and Parameters



### 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

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

- A  
Hydraulic conserving port associated with the elbow inlet.
- B  
Hydraulic conserving port associated with the elbow outlet.

## References

- [1] *Flow of Fluids Through Valves, Fittings, and Pipe*, Crane Valves North America, Technical Paper No. 410M
- [2] George R. Keller, *Hydraulic System Analysis*, Published by the Editors of Hydraulics & Pneumatics Magazine, 1970

## See Also

Gradual Area Change

# Elbow

---

Local Resistance

Pipe Bend

Sudden Area Change

T-junction

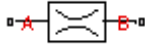
## Purpose

Hydraulic orifice with constant cross-sectional area

## Library

Orifices

## Description



The Fixed Orifice block models a sharp-edged constant-area orifice, flow rate through which is proportional to the pressure differential across the orifice. The model accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number ( $Re$ ) and comparing its value with the critical Reynolds number ( $Re_{cr}$ ). The flow rate is determined according to the following equations:

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

$$p = p_A - p_B$$

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

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

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

where

$q$  Flow rate

$p$  Pressure differential

$p_A, p_B$  Gauge pressures at the block terminals

# Fixed Orifice

$C_D$	Flow discharge coefficient
$A$	Orifice passage area
$D_H$	Orifice hydraulic diameter
$\rho$	Fluid density
$\nu$	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

- 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: Fixed Orifice**

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:

Flow discharge coefficient:

Critical Reynolds number:

OK Cancel Help Apply

**Orifice area**

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

**Flow discharge coefficient**

Semi-empirical parameter for orifice capacity characterization.

Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets.

The default value is 0.7.

**Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12, which corresponds to a round orifice in thin material with sharp edges.

**Global Parameters**

Parameters determined by the type of working fluid:

- Fluid density
- Fluid kinematic viscosity

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

**Ports**

The block has the following ports:

A

Hydraulic conserving port associated with the orifice inlet.

B

Hydraulic conserving port associated with the orifice outlet.

**See Also**

Annular Orifice

Constant Area Hydraulic Orifice

# Fixed Orifice

---

Fixed Orifice Empirical

Fixed Orifice with Fluid Inertia

Orifice with Variable Area Round Holes

Orifice with Variable Area Slot

Variable Area Hydraulic Orifice

Variable Orifice

## Purpose

Fixed hydraulic orifice based on empirical data

## Library

Orifices

## Description



The Fixed Orifice Empirical block models a fixed orifice in hydraulic systems with data obtained from experimental testing. The data required for characterization is the set of pressure differentials across the orifice, together with associated flow rates determined for a particular fluid and temperature.

The flow rate through the orifice is determined with the table lookup block from the set of experimental measurements

$$q = f(p)$$

where

$q$  Volumetric flow rate

$p$  Pressure differential

You can select from three interpolation and two extrapolation methods.

---

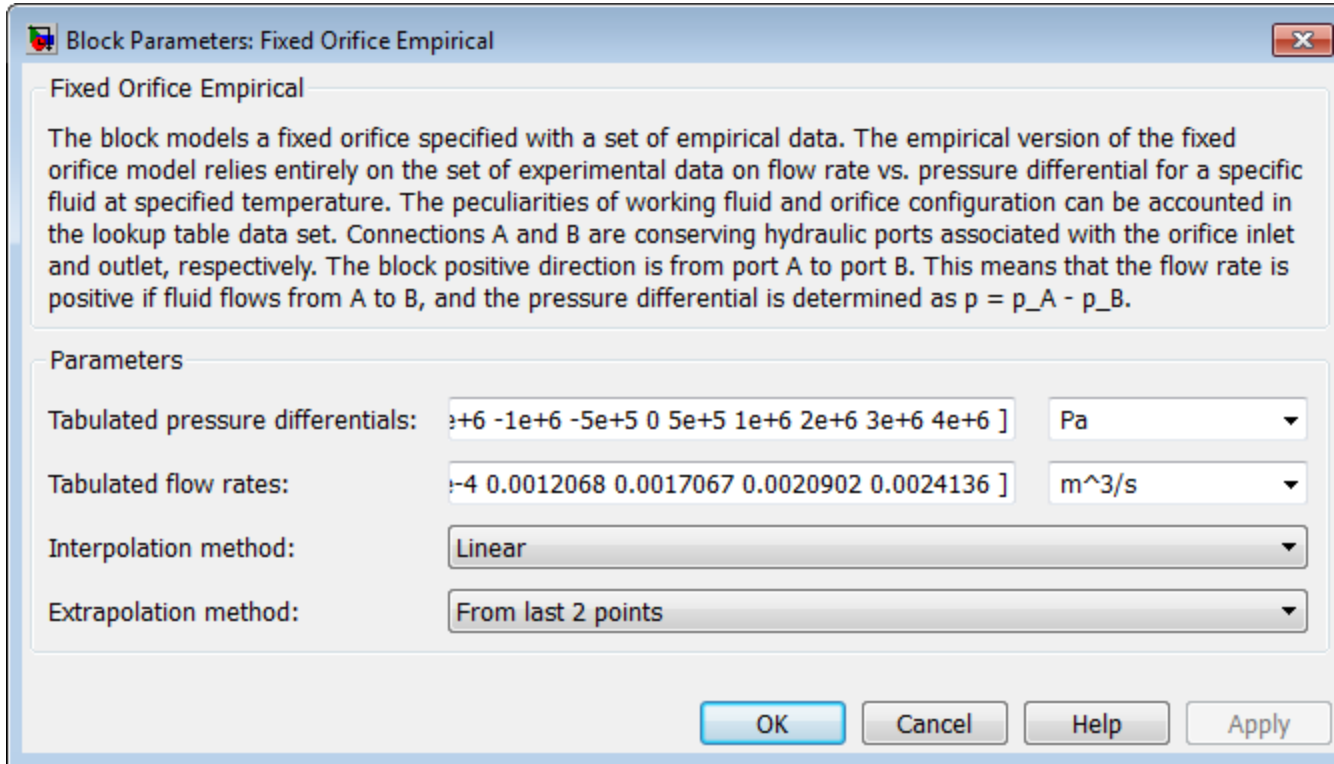
**Note** Extrapolation could result in large errors if the operating point deviates significantly from the area of experiments.

---

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

# Fixed Orifice Empirical

## Dialog Box and Parameters



### Tabulated pressure differentials

Specify the pressure differential vector as a one-dimensional array. The vector contains values of pressure differentials obtained during experimental characterization of the orifice. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values



for linear interpolation, at least three values for cubic or spline interpolation. The default values, in Pa, are [ -4e+6 -3e+6 -2e+6 -1e+6 -5e+5 0 5e+5 1e+6 2e+6 3e+6 4e+6 ].

## Tabulated flow rates

Specify the vector of flow rates through the orifice, corresponding to these pressure differentials, as a one-dimensional array. The vector must be of the same size as the **Tabulated pressure differentials** vector. The default values, in m<sup>3</sup>/s, are [ -0.0024436 -0.0021248 -0.0016852 -0.0012248 -8.4476e-4 0 8.5333e-4 0.0012068 0.0017067 0.0020902 0.0024136 ].

## Interpolation method

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

- **Linear** — Uses a linear interpolation function.
- **Cubic** — Uses the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP).
- **Spline** — Uses the cubic spline interpolation algorithm.

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

## Extrapolation method

Use extrapolation with extreme caution, because moving beyond the measurement limits could result in large errors. Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- **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

# Fixed Orifice Empirical

---

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.

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the orifice inlet.

B

Hydraulic conserving port associated with the orifice outlet.

## See Also

Fixed Orifice

Fixed Orifice with Fluid Inertia

**Purpose** Fixed hydraulic orifice accounting for flow inertia

**Library** Orifices

## Description



The Fixed Orifice with Fluid Inertia block models a hydraulic fixed orifice and accounts for the fluid inertia, in addition to the static pressure loss. 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 ( $Re_{cr}$ ).

Fluid inertia plays a noticeable role in orifices with a large ratio of orifice length to the orifice hydraulic diameter ( $L / D_H$ ) and in sharp-edged short orifices when the rate of change of flow rate (fluid acceleration) is relatively large.

The orifice is based on the following equations:

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

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

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

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

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

# Fixed Orifice with Fluid Inertia

---

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

where

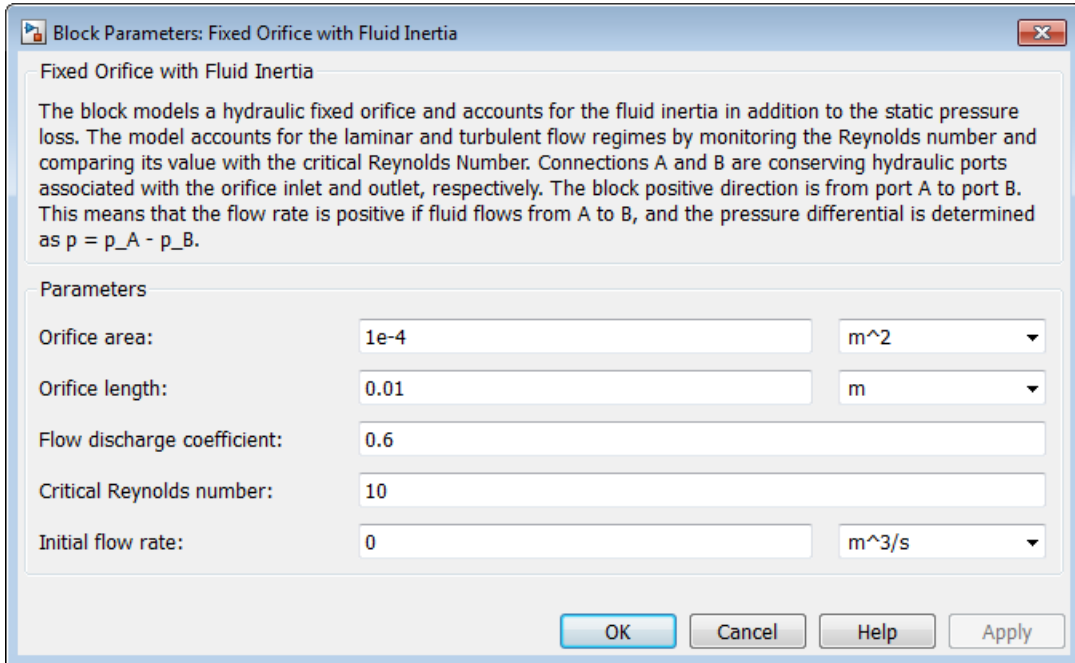
$q$	Volumetric flow rate
$p$	Total pressure differential
$p_{in}$	Inertial pressure drop
$p_r$	Resistive pressure drop
$C_D$	Flow discharge coefficient
$A$	Orifice passage area
$L$	Orifice length
$D_H$	Orifice hydraulic diameter
$\rho$	Fluid density
$\nu$	Fluid kinematic viscosity

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

## Basic Assumptions and Limitations

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

## Dialog Box and Parameters



### Orifice area

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

### Orifice length

Total length of the orifice. Generally, increase the geometrical length of the orifice up to  $2 \cdot 0.8 \cdot D_H$  (where  $D_H$  is the orifice hydraulic diameter) to take into account the added volumes of fluid on both sides of the orifice. The default value is  $0.01 \text{ m}$ .

# Fixed Orifice with Fluid Inertia

---

## Flow discharge coefficient

Semi-empirical parameter for orifice capacity characterization. The coefficient affects the resistive pressure drop in the orifice. The default value is 0.6.

## Critical Reynolds number

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches this value. The default value is 10.

## Initial flow rate

Flow rate through the orifice at the start of simulation. This parameter specifies the initial condition for use in computing the block's state at the beginning of a simulation run. For more information, see "Initial Conditions Computation". The default value is 0.

## Global Parameters

Parameters determined by the type of working fluid:

- Fluid density
- Fluid kinematic viscosity

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

## Ports

The block has the following ports:

- A  
Hydraulic conserving port associated with the orifice inlet.
- B  
Hydraulic conserving port associated with the orifice outlet.

## See Also

Fixed Orifice  
Fixed Orifice Empirical

# Fixed-Displacement Motor (External Efficiencies)

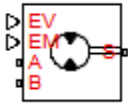
## Purpose

Fixed-displacement hydraulic motor with externally specified mechanical and volumetric efficiencies

## Library

Pumps and Motors

## Description



The Fixed-Displacement Motor (External Efficiencies) block represents a fixed-displacement hydraulic motor with both mechanical and volumetric efficiencies provided through the external physical signal ports. Hydraulic losses are implemented as a leakage between hydraulic ports, while mechanical losses are introduced through the friction imposed on the driving shaft. Both the leakage and the friction are determined by the efficiency values. The ideal hydraulic motor is described with the following equations:

$$q_{id} = D \cdot \omega$$

$$T_{id} = D \cdot p$$

where

$q_{id}$  Ideal volumetric flow rate

$p$  Pressure differential across the motor

$D$  Displacement

$\omega$  Angular velocity

$T_{id}$  Ideal torque

When accounting for mechanical and hydraulic losses, the flow rate through the motor is increased by the leakage value

$$q_{leak} = D \cdot \omega \cdot (1 - \eta_V)$$

and output torque becomes lower due to mechanical losses simulated with the friction torque

# Fixed-Displacement Motor (External Efficiencies)

---

$$T_{FR} = T_{id} \cdot (1 - \eta_M)$$

where

$q_{leak}$  Leakage flow through the motor

$\eta_V$  Volumetric efficiency

$T_{FR}$  Friction torque

$\eta_M$  Mechanical efficiency

For numerical stability purposes, the friction torque is implemented as

$$T_{FR} = T_{id} \cdot (1 - \eta_M) \cdot \tanh\left(4 \frac{\omega}{\omega_{th}}\right)$$

where  $\omega_{th}$  is the threshold angular velocity, a parameter that specifies how steeply the tanh function changes in the vicinity of zero 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. Physical signal ports EV and EM import volumetric and mechanical efficiencies, respectively. Use a separate block to compute these efficiencies. 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.

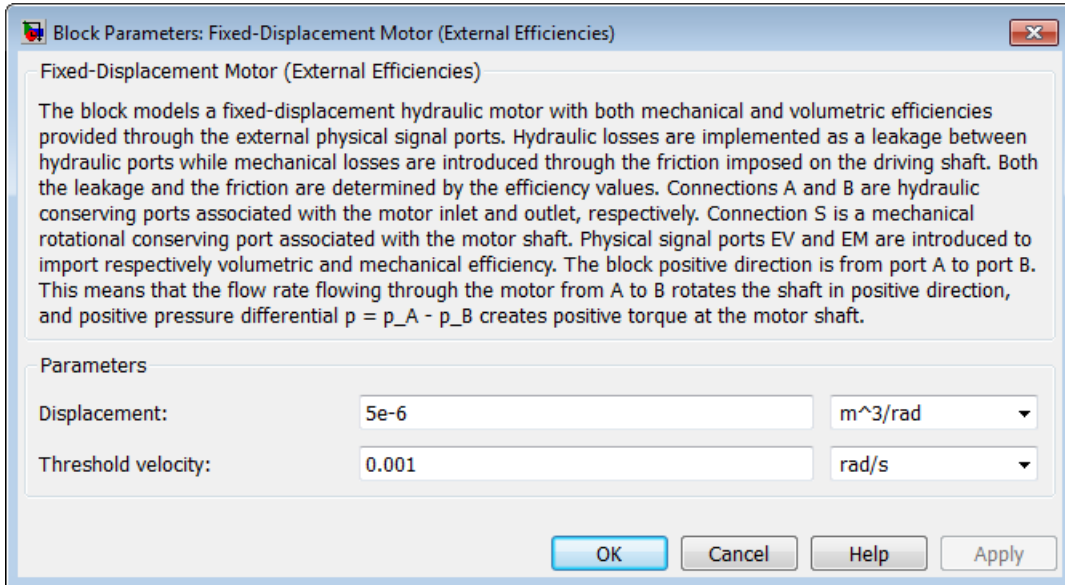
## Basic Assumptions and Limitations

- Fluid compressibility is neglected.
- Hydraulic loss is simulated as leakage through the motor.
- Mechanical loss is represented with the friction torque applied to the motor shaft.



# Fixed-Displacement Motor (External Efficiencies)

## Dialog Box and Parameters



### Displacement

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

### Threshold velocity

This parameter specifies how steeply the tanh function changes in the vicinity of zero velocity. It is introduced to avoid sharp change in the friction torque. The default value is  $0.001 \text{ rad/s}$ .

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the motor inlet.

# Fixed-Displacement Motor (External Efficiencies)

---

- B Hydraulic conserving port associated with the motor outlet.
- S Mechanical rotational conserving port associated with the motor shaft.
- EV Physical signal port that imports the motor volumetric efficiency.
- EM Physical signal port that imports the motor mechanical efficiency.

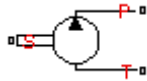
## See Also

- Hydraulic Motor
- Variable-Displacement Motor

**Purpose** 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:

$$q = D\omega - k_{leak}\rho$$

$$T = D\rho / \eta_{mech}$$

$$k_{leak} = k_{HP} / \sqrt{\rho}$$

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

$$\rho = p_P - p_T$$

where

- $q$  Pump delivery
- $\rho$  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
- $\omega$  Pump angular velocity
- $k_{leak}$  Leakage coefficient

# Fixed-Displacement Pump

---

$k_{HP}$	Hagen-Poiseuille coefficient
$\eta_V$	Pump volumetric efficiency
$\eta_{mech}$	Pump mechanical efficiency
$\nu$	Fluid kinematic viscosity
$\rho$	Fluid density
$p_{nom}$	Pump nominal pressure
$\omega_{nom}$	Pump nominal angular velocity
$\nu_{nom}$	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
$\mu$	Fluid dynamic viscosity, $\mu = \nu\rho$

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

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

which provides the formula to determine the Hagen-Poiseuille coefficient

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

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

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

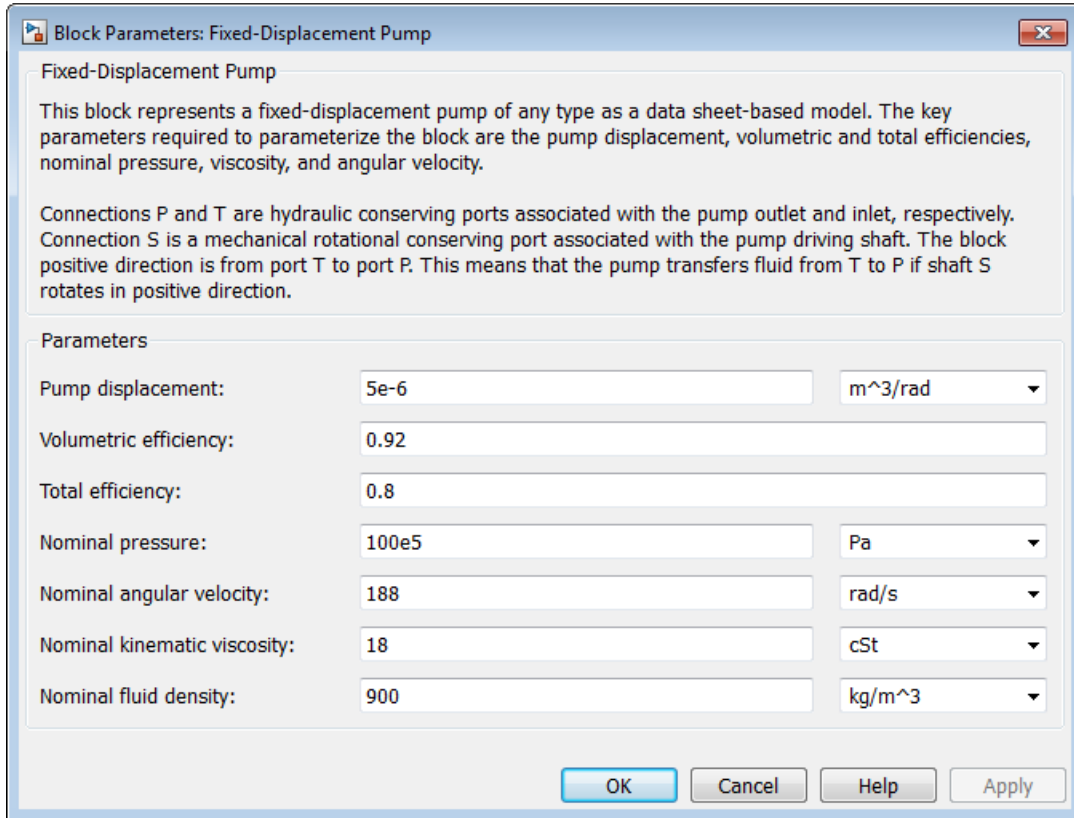
The block positive direction is from port T to port P. This means that the pump transfers fluid from T to P provided that the shaft S rotates in the positive direction. The pressure differential across the pump is determined as  $p = p_P - p_T$ .

## **Basic Assumptions and Limitations**

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

# Fixed-Displacement Pump

## Dialog Box and Parameters



### **Pump displacement**

Pump displacement. The default value is  $5e-6$  m<sup>3</sup>/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.

## **Nominal fluid density**

Working fluid density, at which both the volumetric and total efficiencies are specified. The default value is  $900 \text{ kg/m}^3$ .

## **Global Parameters**

Parameter determined by the type of working fluid:

- **Fluid kinematic viscosity**

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

## **Ports**

The block has the following ports:

T

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

P

Hydraulic conserving port associated with the pump outlet.

# Fixed-Displacement Pump

---

S

Mechanical rotational conserving port associated with the pump driving shaft.

## Examples

The Power Unit with Fixed-Displacement Pump example 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

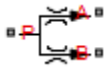
Variable-Displacement Pump



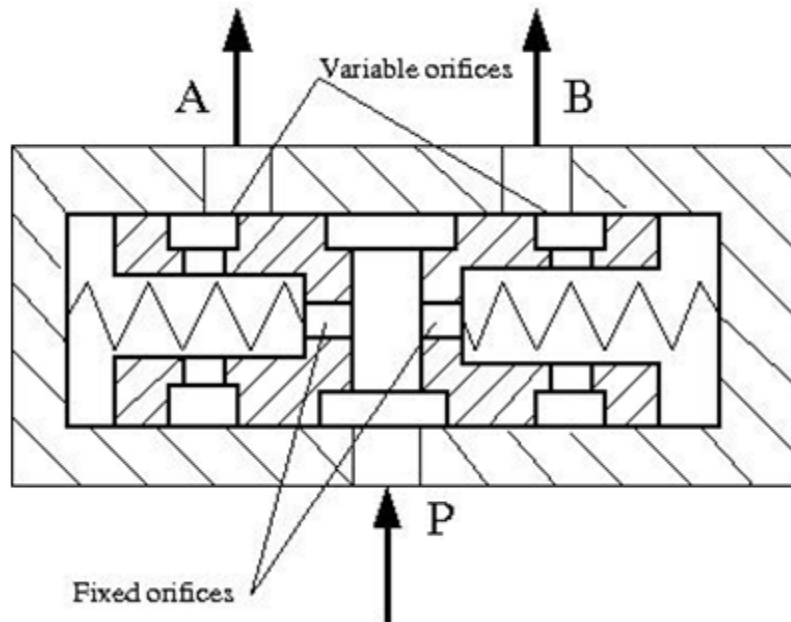
**Purpose** Hydraulic two-path flow divider

**Library** Flow Control Valves

**Description**



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



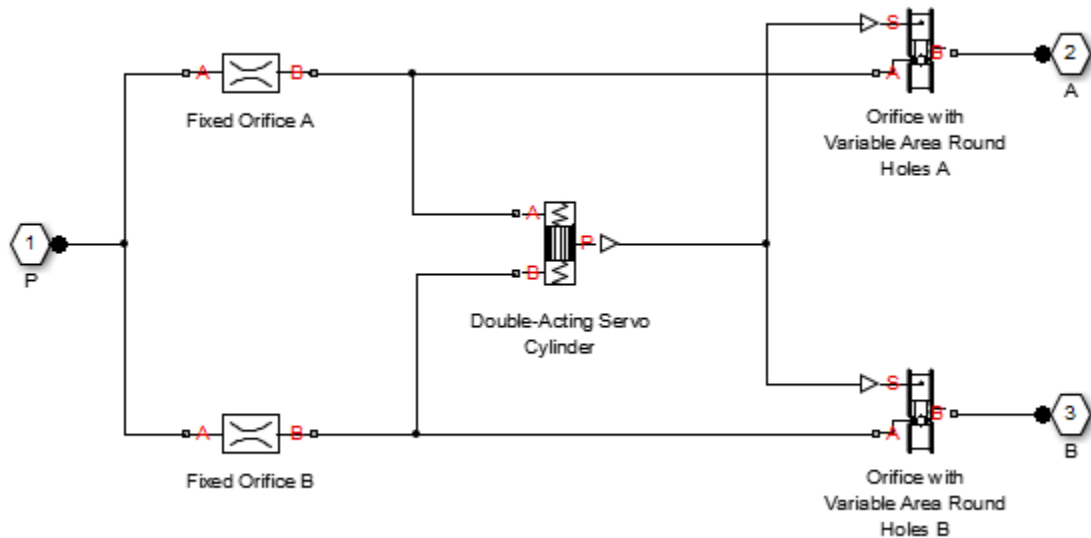
The flow from the source enters the valve through port P and is split into two parts flowing through the P-A and P-B paths. Each path contains a fixed orifice and a variable orifice. The fixed orifices must be precisely matched to divide flow in equal parts, or arranged in a certain proportion if unequal division is required.

The purpose of variable orifices is to maintain a constant pressure drop across the fixed orifices, regardless of pressure fluctuations at valve outlets. The load increase on any outlet causes the pressure drop across

# Flow Divider

the spool (and across fixed orifices) to change, and thus shift the spool. As a result, the passage areas of variable orifices change until the pressure drop values across the fixed orifices even out.

The model of the flow divider uses the Double-Acting Servo Cylinder, Fixed Orifice, and Orifice with Variable Area Round Holes blocks. The following figure shows the schematic diagram of the model. The Double-Acting Servo Cylinder detects the pressure drop and shifts the variable orifice control member accordingly.



---

**Note** You cannot use this block as a flow combiner.

---

## Assumptions and Limitations

- Friction between moving parts is not taken into account.
- Inertia effects are not taken into account.
- Fluid compressibility is not taken into account.
- Leakage flows are assumed to be negligible.
- The hard stops in the Double-Acting Servo Cylinder are assumed to be fully inelastic.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at  $Re=Re_{cr}$ .

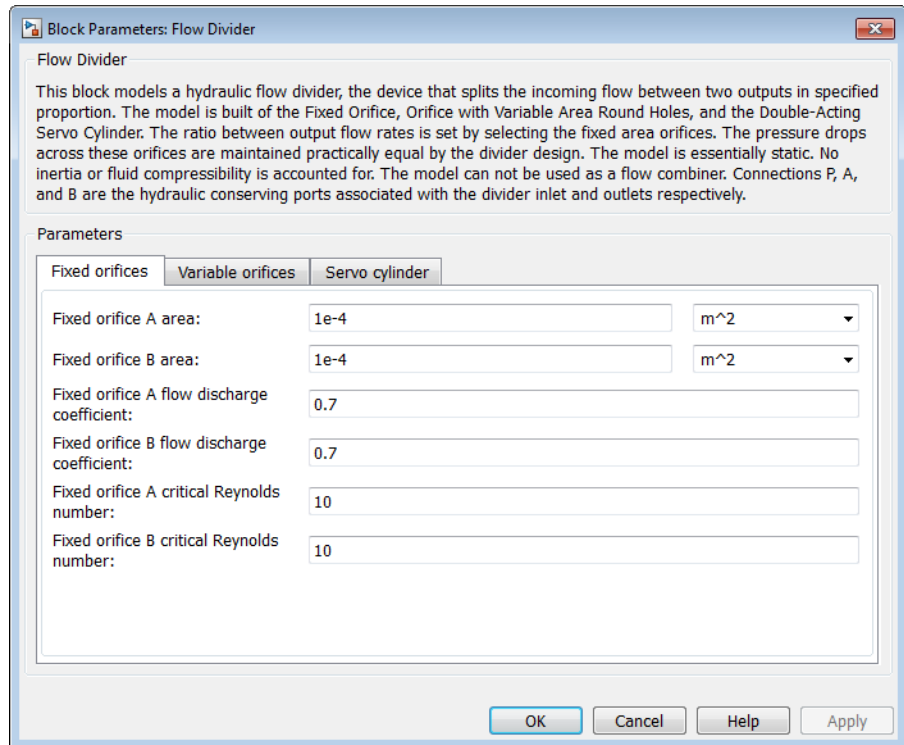
## Dialog Box and Parameters

The block dialog box has three tabs:

- “Fixed Orifices” on page 1-286
- “Variable Orifices” on page 1-288
- “Servo Cylinder” on page 1-290

# Flow Divider

## Fixed Orifices



### Fixed orifice A area

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

### Fixed orifice B area

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

### Fixed orifice A flow discharge coefficient

Semi-empirical parameter defining the capacity of the fixed orifice in the P–A path. The value depends on the geometrical

properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is 0.7.

**Fixed orifice B flow discharge coefficient**

Semi-empirical parameter defining the capacity of the fixed orifice in the P–B path. The value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is 0.7.

**Fixed orifice A critical Reynolds number**

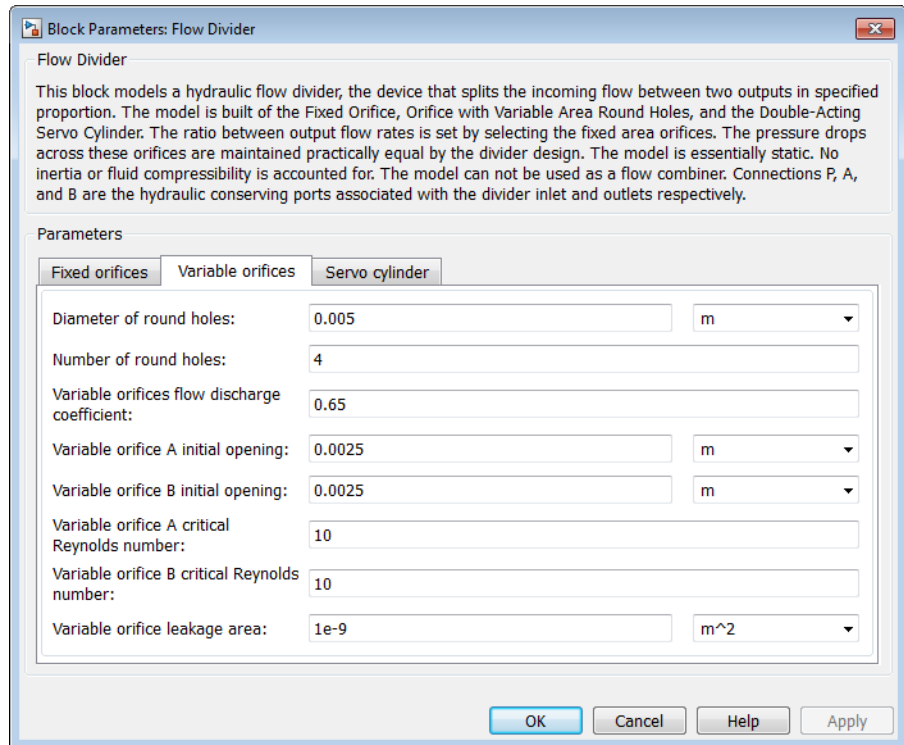
The maximum Reynolds number for laminar flow for the fixed orifice in the P–A path. The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches this value. The default value is 10.

**Fixed orifice B critical Reynolds number**

The maximum Reynolds number for laminar flow for the fixed orifice in the P–B path. The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches this value. The default value is 10.

# Flow Divider

## Variable Orifices



### Diameter of round holes

Diameter of the round holes in the two identical Variable Orifice with Round Holes blocks. The default value is 0.005 m.

### Number of round holes

Number of holes in each of the Variable Orifice with Round Holes blocks. The default value is 4.

### Variable orifices flow discharge coefficient

Semi-empirical parameter defining the orifice capacity of the Variable Orifice with Round Holes blocks. The value depends on

the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is 0.65.

### **Variable orifice A initial opening**

Initial opening of the variable orifice in the P–A path. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or 0 for zero-lap configuration. The value of initial opening does not depend on the orifice orientation. The default value is 0.0025.

### **Variable orifice B initial opening**

Initial opening of the variable orifice in the P–B path. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or 0 for zero-lap configuration. The value of initial opening does not depend on the orifice orientation. The default value is 0.0025.

### **Variable orifice A critical Reynolds number**

The maximum Reynolds number for laminar flow through the variable orifice in the P–A path. The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches this value. The default value is 10.

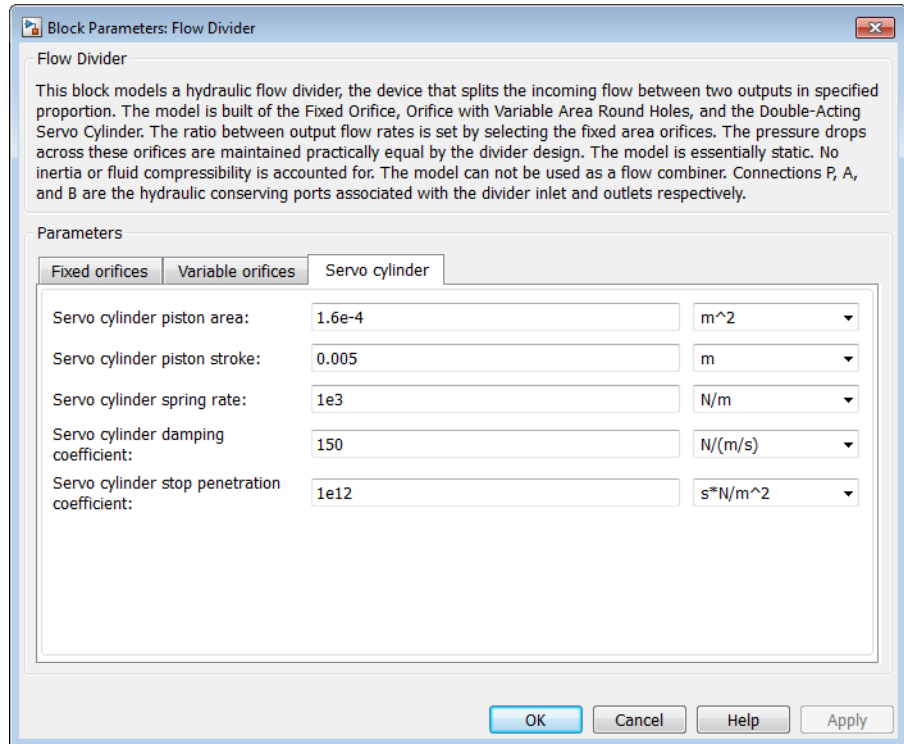
### **Variable orifice B critical Reynolds number**

The maximum Reynolds number for laminar flow through the variable orifice in the P–B path. The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches this value. The default value is 10.

### **Variable orifice leakage area**

The total area of possible leaks in each variable orifice when it is completely closed. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from becoming isolated after the orifice is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-9 \text{ m}^2$ .

## Servo Cylinder



### Servo cylinder piston area

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

### Servo cylinder piston stroke

The full piston stroke in the servo cylinder, from one hard stop to another. The piston is located initially in the middle of the stroke and can travel half a stroke in the positive and negative direction. The default value is  $0.005 \text{ m}$ .



## **Servo cylinder spring rate**

The spring rate of the centering springs in the servo cylinder.  
The default value is 1000 N/m.

## **Servo cylinder damping coefficient**

The damping coefficient in the contact between the piston and the case of the servo cylinder. The default value is 150 N/(m/s).

## **Servo cylinder stop penetration coefficient**

The penetration property of the piston hard stop in the servo cylinder. The hard stop is represented as absolutely inelastic, and its property is characterized by the penetration coefficient. The default value of the coefficient is  $1e12$  N/m/(m/s).

## **Global Parameters**

Parameters determined by the type of working fluid:

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

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

## **Ports**

The block has the following ports:

- P  
Hydraulic conserving port associated with the inlet port P.
- A  
Hydraulic conserving port associated with the outlet port A.
- B  
Hydraulic conserving port associated with the outlet port B.

## **Examples**

The Hydraulic System with Flow Divider example shows the use of a Flow Divider block to split the flow equally between two consumers represented by Orifice with Variable Area Slot blocks.

## **See Also**

Double-Acting Servo Cylinder

# Flow Divider

---

Fixed Orifice

Orifice with Variable Area Round Holes

**Purpose** 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 another kind of elastic diaphragm.

If the fluid pressure at the accumulator inlet becomes higher than the preload 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 preload 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 preload 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 preload value or higher, the chambers start interacting again.

To improve computational convergence and efficiency, the accumulator inlet chamber is assumed to be compliant. As a result, the fluid starts accumulating in the chamber even before the preload pressure is reached, so that at preload pressure the accumulator already stores a certain volume of fluid. The structural compliance can be set to a very small value, but not to zero.

The accumulator is described with the following equations:

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

# Gas-Charged Accumulator

---

$$V_F = \begin{cases} K_s(p + p_a) & \text{for } p \leq p_{pr} \\ V_{pr} \left( \frac{p_{pr} + p_a}{p + p_a} \right)^{\frac{1}{k}} + V_A \left[ 1 - \left( \frac{p_{pr} + p_a}{p + p_a} \right)^{\frac{1}{k}} \right] & \text{for } p > p_{pr} \end{cases}$$

$$V_{pr} = K_s(p_{pr} + p_a)$$

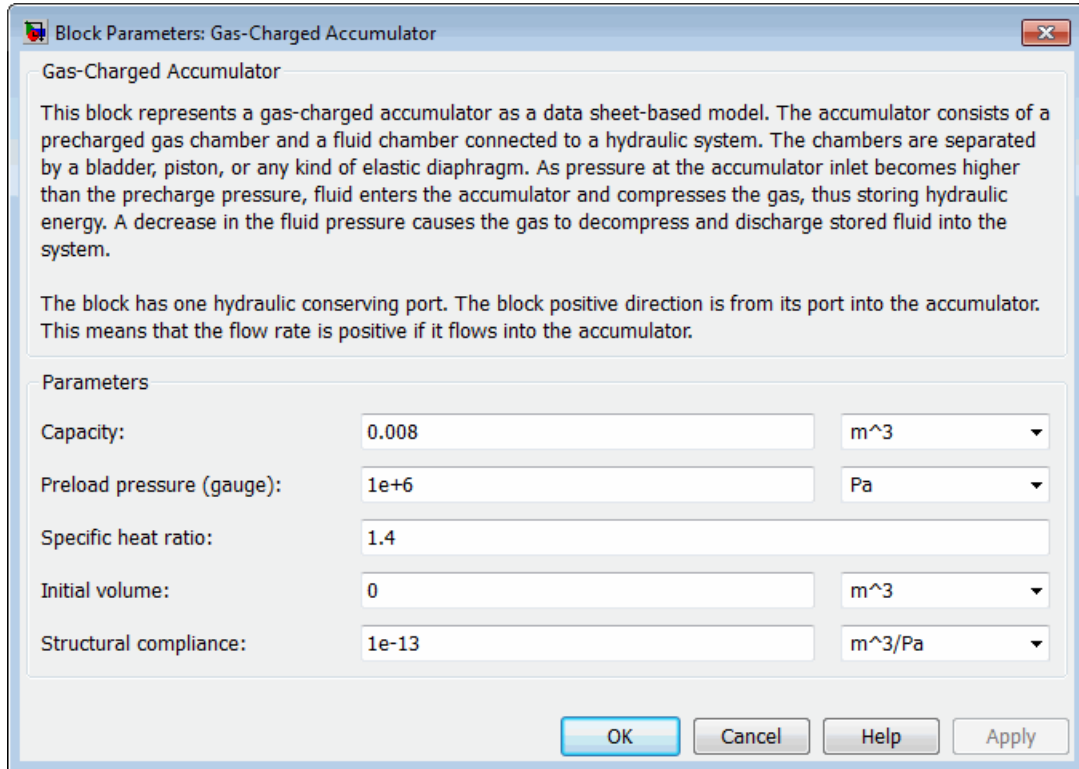
where

$V_F$	Volume of fluid in the accumulator
$V_A$	Accumulator capacity
$V_{pr}$	Volume of fluid at preload pressure
$p$	Inlet gauge pressure
$p_{pr}$	Preload pressure
$p_a$	Atmospheric pressure
$K_s$	Structural compliance of the accumulator inlet port structure
$k$	Specific heat ratio
$q$	Volumetric flow rate
$t$	Time

## Basic Assumptions and Limitations

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

## Dialog Box and Parameters



### Capacity

Accumulator capacity. The default value is 0.008 m<sup>3</sup>.

### Preload pressure (gauge)

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

# Gas-Charged Accumulator

---

## 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 to a value between 1 and 2, depending on the properties of the gas being used in the accumulator. For example, for dry air at 20 degrees C, this value will be 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 "Initial Conditions Computation". The default value is 0.

## Structural compliance

The compliance of the inlet port structure. This parameter is introduced to improve convergence and computational efficiency. It must be greater than zero. Values less than  $1e-16 \text{ m}^3/\text{Pa}$  may result in numerical instability. The default value is  $1e-13 \text{ m}^3/\text{Pa}$ .

## Global Parameters

### Atmospheric pressure

Absolute pressure of the environment. The default value is 101325 Pa.

## Ports

The block has one hydraulic conserving port associated with the accumulator inlet.

The flow rate is positive if fluid flows into the accumulator.

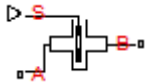
## See Also

Spring-Loaded Accumulator

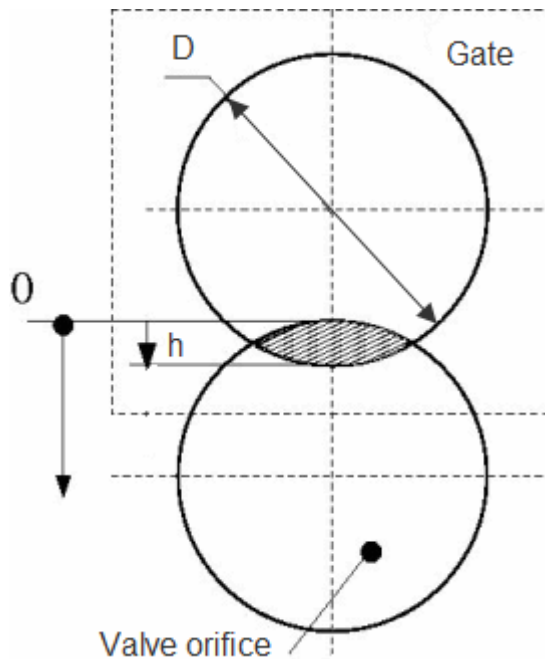
**Purpose** Hydraulic gate valve

**Library** Flow Control Valves

**Description**



The Gate Valve block models a valve consisting of a round orifice in the valve housing and a flat gate, which moves perpendicular to the orifice axis. The orifice in the gate has the same diameter as the orifice in the housing. As the gate moves, it opens or closes the valve passage (shown as a shaded area in the following illustration).



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

# Gate Valve

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

$$h = x_0 + x$$

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

$$p = p_A - p_B$$

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

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

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

where

$q$  Flow rate

$p$  Pressure differential

$p_A, p_B$  Gauge pressures at the block terminals



$C_D$	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
$x_0$	Initial opening
$x$	Gate displacement from initial position
$h$	Valve opening
$D$	Orifice diameter
$\rho$	Fluid density
$D_H$	Valve instantaneous hydraulic diameter
$\nu$	Fluid kinematic viscosity
$A_{leak}$	Closed valve leakage area

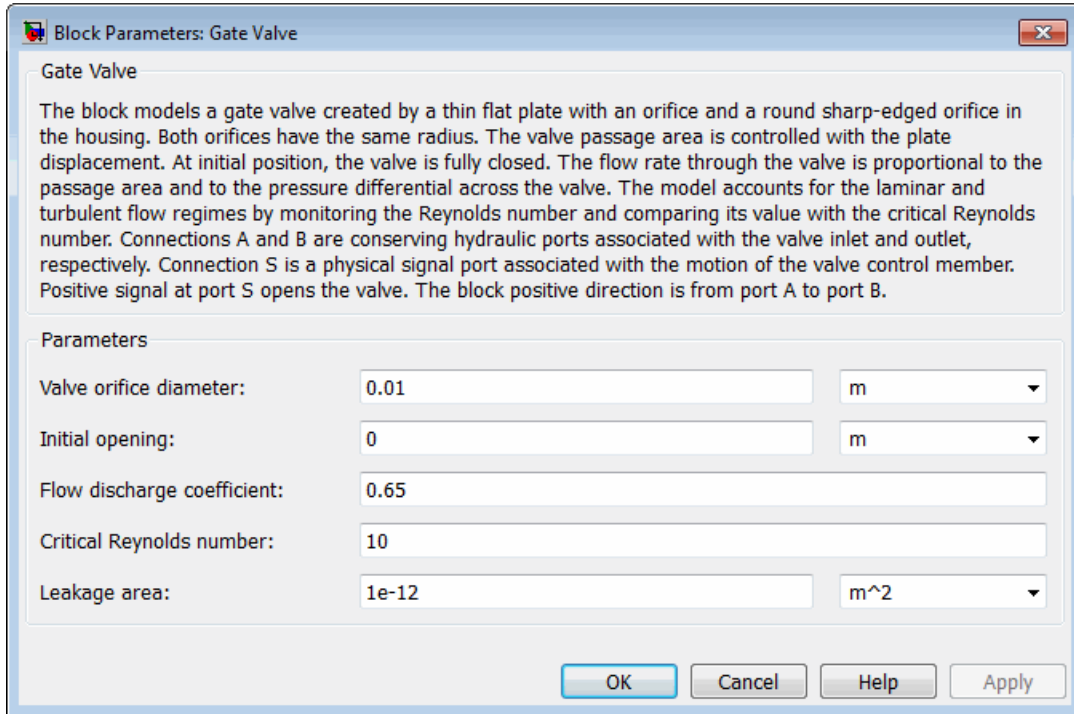
Connections A and B are hydraulic conserving ports. Connection S is a physical signal port that controls the gate displacement. The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as  $p = p_A - p_B$ . Positive signal at the physical signal port S opens the valve.

## Basic Assumptions and Limitations

- No inertial effects are taken into account.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at  $Re=Re_{cr}$ .

# Gate Valve

## Dialog Box and Parameters



### Valve orifice diameter

The diameter of the valve orifice. The orifices in the valve housing and in the gate have the same diameter. The default value is 0.01 m.

### Initial opening

The initial opening of the valve. The parameter can take both positive and negative values. The default value is 0.

## Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is 0.65.

## 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 simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## Global Parameters

Parameters determined by the type of working fluid:

- Fluid density
- Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

# Gate Valve

---

B

Hydraulic conserving port associated with the valve outlet.

S

Physical signal port that controls the gate displacement. The signal applied to this port is treated as translational motion, in meters.

## See Also

Ball Valve

Ball Valve with Conical Seat

Needle Valve

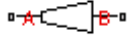
Poppet Valve

Pressure-Compensated Flow Control Valve

**Purpose** Gradual enlargement or contraction

**Library** Local Hydraulic Resistances

**Description**



The Gradual Area Change block represents a local hydraulic resistance, such as a gradual cross-sectional area change. The resistance represents a gradual enlargement (diffuser) if fluid flows from inlet to outlet, or a gradual contraction if fluid flows from outlet to inlet. The block is based on the Local Resistance block. It determines the pressure loss coefficient and passes its value to the underlying Local Resistance block. The block offers two methods of parameterization: by applying semi-empirical formulas (with a constant value of the pressure loss coefficient) or by table lookup for the pressure loss coefficient based on the Reynolds number.

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

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

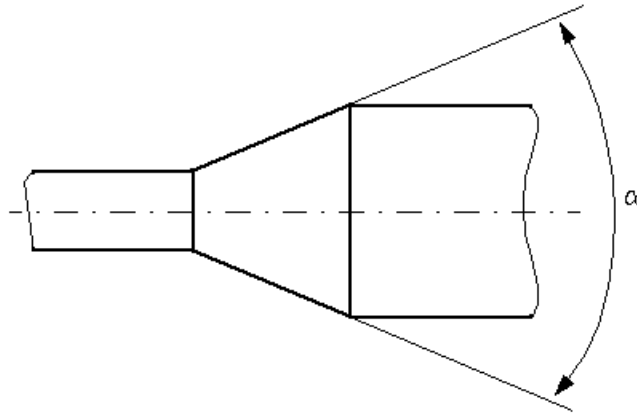
$$K_{GC} = \begin{cases} K_{cor} \square 0.5 \left(1 - \frac{A_s}{A_L}\right)^{0.75} \square 1.6 \sin \frac{\alpha}{2} & \text{for } 0 < \alpha \leq 45^\circ \\ K_{cor} \square 0.5 \left(1 - \frac{A_s}{A_L}\right)^{0.75} \square \sqrt{\sin \frac{\alpha}{2}} & \text{for } 45^\circ < \alpha < 180^\circ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

where

# Gradual Area Change

---

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

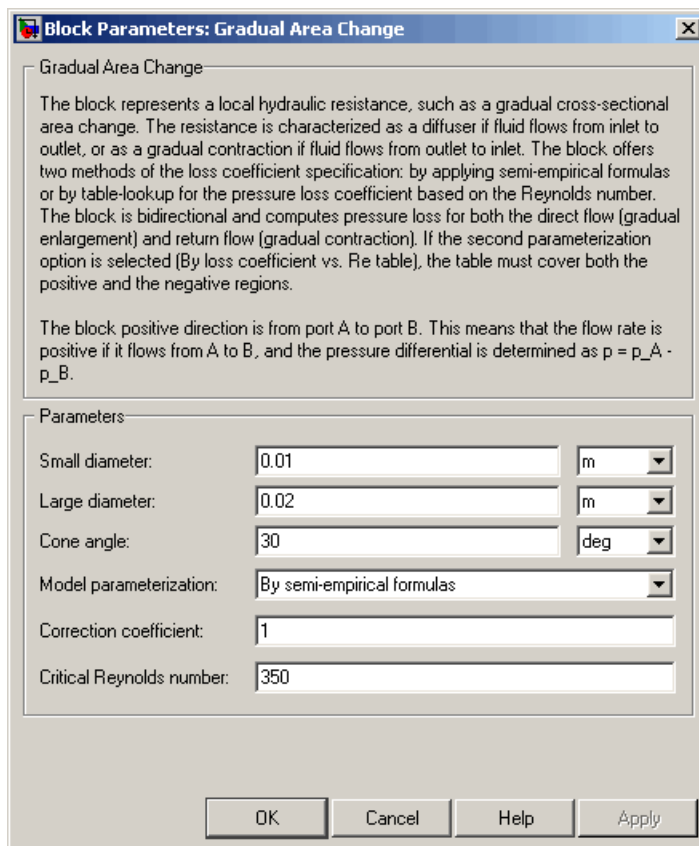
The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure loss is determined as  $p = p_A - p_B$ .

## Basic Assumptions and Limitations

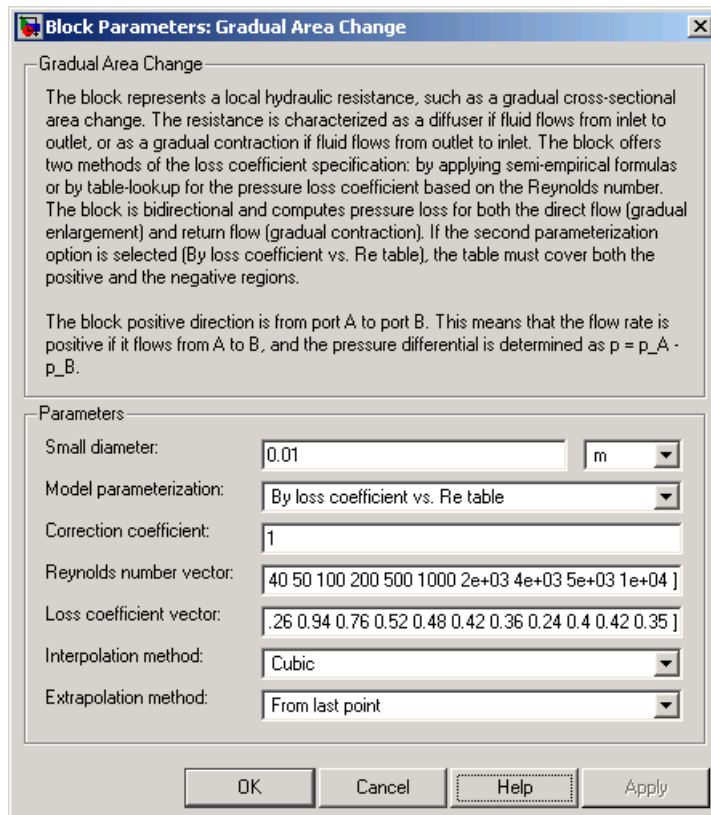
- Fluid inertia is not taken into account.
- If you select parameterization by semi-empirical formulas, the transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at  $Re=Re_{cr}$ .
- If you select parameterization by the table-specified relationship  $K=f(Re)$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

# Gradual Area Change

## Dialog Box and Parameters







## Small diameter

Resistance small diameter. The default value is 0.01 m.

## Large diameter

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

# Gradual Area Change

---

## Cone angle

The enclosed angle. The default value is 30 deg. This parameter is used if **Model parameterization** is set to By semi-empirical formulas.

## Model parameterization

Select one of the following methods for block parameterization:

- **By semi-empirical formulas** — Provide geometrical parameters of the resistance. This is the default method.
- **By loss coefficient vs. Re table** — Provide tabulated relationship between the loss coefficient and the Reynolds number. The loss coefficient is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods. The table must cover both the positive and the negative flow regions.

## Correction coefficient

Correction factor used in the formula for computation of the loss coefficient. The default value is 1. This parameter is used if **Model parameterization** is set to By semi-empirical formulas.

## Critical Reynolds number

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 350. This parameter is used if **Model parameterization** is set to By semi-empirical formulas.

## Reynolds number vector

Specify the vector of input values for Reynolds numbers as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at

least three values for cubic or spline interpolation. The default values are [-4000, -3000, -2000, -1000, -500, -200, -100, -50, -40, -30, -20, -15, -10, 10, 20, 30, 40, 50, 100, 200, 500, 1000, 2000, 4000, 5000, 10000]. This parameter is used if **Model parameterization** is set to By loss coefficient vs. Re table.

### Loss coefficient vector

Specify the vector of the loss coefficient values as a one-dimensional array. The loss coefficient vector must be of the same size as the Reynolds numbers vector. The default values are [0.25, 0.3, 0.65, 0.9, 0.65, 0.75, 0.90, 1.15, 1.35, 1.65, 2.3, 2.8, 3.10, 5, 2.7, 1.8, 1.46, 1.3, 0.9, 0.65, 0.42, 0.3, 0.20, 0.40, 0.42, 0.25]. This parameter is used if **Model parameterization** is set to By loss coefficient vs. Re table.

### Interpolation method

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

- **Linear** — Uses a linear interpolation function.
- **Cubic** — Uses the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP).
- **Spline** — Uses the cubic spline interpolation algorithm.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to By loss coefficient vs. Re table.

### Extrapolation method

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

# Gradual Area Change

---

- **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

Parameters determined by the type of working fluid:

- **Fluid density**

- **Fluid kinematic viscosity**

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

## **Ports**

The block has the following ports:

A

Hydraulic conserving port associated with the resistance inlet.

B

Hydraulic conserving port associated with the resistance outlet.

## **References**

[1] *Flow of Fluids Through Valves, Fittings, and Pipe*, Crane Valves North America, Technical Paper No. 410M

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

## **See Also**

Elbow

Local Resistance

Pipe Bend

Sudden Area Change

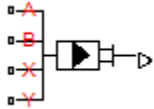
T-junction

# Hydraulic 4-Port Cartridge Valve Actuator

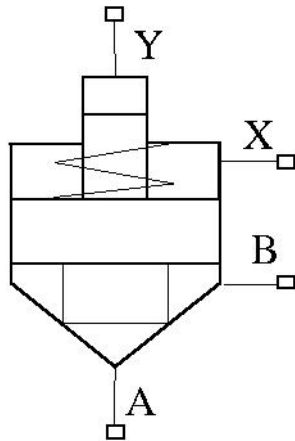
**Purpose** Double-acting hydraulic valve actuator driven by four pressures

**Library** Valve Actuators

## Description



The Hydraulic 4-Port Cartridge Valve Actuator block represents a double-acting hydraulic valve actuator driven by four pressures. Use it as a pilot actuator for cartridge valves, pilot-operated pressure and control valves, and similar devices. A schematic diagram of the 4-port cartridge valve actuator is shown in the following illustration.



The actuator drives a valve (spool, poppet, and so on) whose position depends on pressures at ports A, B, X, and Y and the spring force. Pressures at ports A and B tend to open the valve, while pressures at control ports X and Y, together with the spring force, act to close it. The model does not account for flow consumption and loading forces, except spring and pressure forces.

Inertial properties of the actuator are accounted for by adding a first order lag between the steady-state and actual valve displacements. The lag is simulated with the combination of the PS Gain and PS Integrator blocks, enveloped by the unity feedback.

The valve remains closed as long as the aggregate pressure force is lower than the spring preload force. The poppet is forced off its seat as

# Hydraulic 4-Port Cartridge Valve Actuator

the preload force is reached and moves up proportionally to pressure increase until it passes the full stroke.

The force acting on the poppet and the poppet displacement are computed with the following equations:

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

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

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

where

$F_P$  Pressure force acting on the poppet

$p_A, p_B, p_X, p_Y$  Pressures at ports A, B, X, and Y, respectively

$p_X, p_Y$

$A_A, A_B, A_X, A_Y$  Areas at ports A, B, X, and Y, respectively. Areas at ports A and Y are block parameters. Area at port X is specified by its ratio to that at port A,  $A_A / A_X$ . Area at port B is determined as  $A_B = A_X - A_A$ .

$x$  Actual poppet displacement

$x_s$  Steady-state poppet displacement

$x_{\max}$  Full stroke

$F_{PR}$  Spring preload force

$k_{spr}$  Spring rate

Connections A, B, X, and Y are hydraulic conserving ports associated with the actuator ports. Connection P is a physical signal port whose

# Hydraulic 4-Port Cartridge Valve Actuator

---

output corresponds to poppet displacement. Pressures applied at ports A and B move the poppet in the positive or negative direction, depending on the value of the **Actuator orientation** parameter. Pressures at ports X and Y act in the opposite direction to pressures at ports A and B.

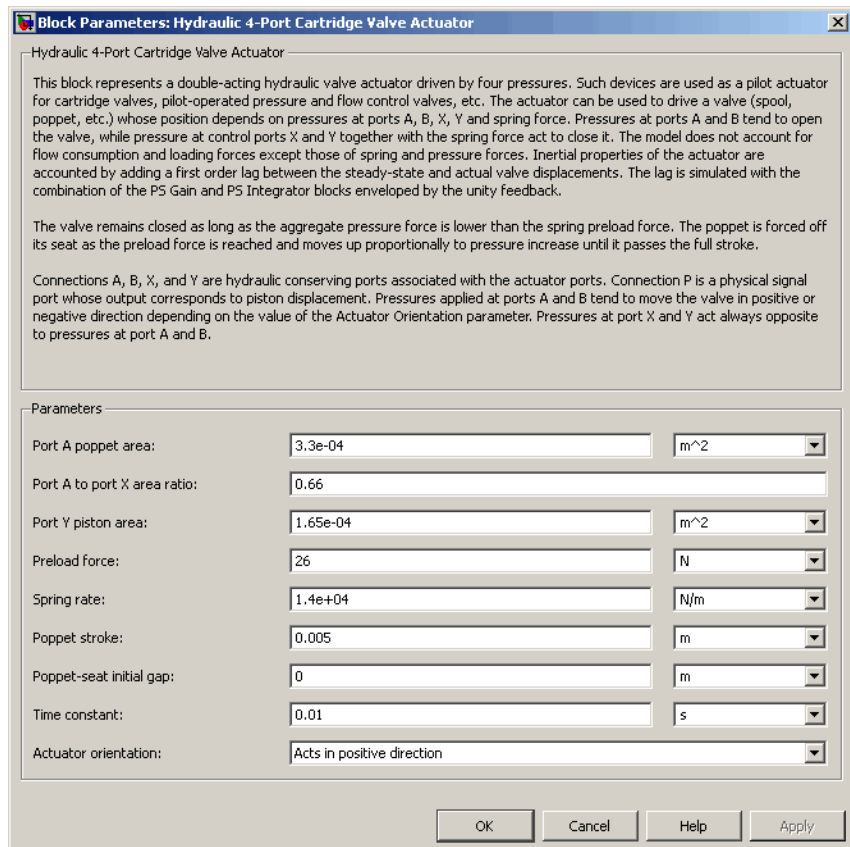
## **Basic Assumptions and Limitations**

- The flow consumption associated with the valve motion is assumed to be negligible.
- The inertia, friction, and hydraulic axial forces are assumed to be small and are not taken into account.



# Hydraulic 4-Port Cartridge Valve Actuator

## Dialog Box and Parameters



### Port A poppet area

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

### Port A to port X area ratio

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

# Hydraulic 4-Port Cartridge Valve Actuator

---

## **Port Y piston area**

Effective piston area at port Y. The parameter value must be greater than zero. The default value is  $1.65e-4 \text{ m}^2$ .

## **Preload force**

Spring preload force. The default value is 26 N.

## **Spring rate**

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

## **Poppet stroke**

Maximum poppet stroke. The parameter value must be greater than zero. The default value is  $5e-3 \text{ m}$ .

## **Poppet-seat initial gap**

Initial gap between the poppet and the seat. The parameter value must be greater than or equal to zero. The default value is 0.

## **Time constant**

Time constant of the first-order lag. The default value is 0.01 s.

## **Actuator orientation**

Specifies actuator orientation with respect to the globally assigned positive direction. The actuator can be installed in two different ways, depending upon whether it moves the poppet in the positive or in the negative direction when pressure is applied at its inlet. If pressures applied at ports A and B move the poppet in the negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

## **Restricted Parameters**

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

- **Actuator orientation**

All other block parameters are available for modification.

## **Ports**

The block has the following ports:

# Hydraulic 4-Port Cartridge Valve Actuator

---

- A Hydraulic conserving port associated with the valve inlet.
- B Hydraulic conserving port associated with the valve outlet.
- X Hydraulic conserving port associated with the valve control terminal.
- Y Hydraulic conserving port associated with the valve control terminal.
- P Physical signal port that outputs poppet displacement.

## Examples

For an example of using this block, see the Hydraulic Actuation System with Cartridge Valves example.

## See Also

Hydraulic Cartridge Valve Actuator

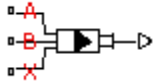
# Hydraulic Cartridge Valve Actuator

---

**Purpose** Double-acting hydraulic actuator for cartridge valves

**Library** Valve Actuators

## Description



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

Inertial properties of the actuator are accounted for by adding a first order lag between the steady-state and actual valve displacements. The lag is simulated with the combination of the PS Gain and PS Integrator blocks, enveloped by the unity feedback.

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

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

## Basic Assumptions and Limitations

- The flow consumption associated with the valve motion is assumed to be negligible.
- The inertia, friction, and hydraulic axial forces are assumed to be small and are not taken into account.
- The clearances between the valve and the washers are not taken into account.

# Hydraulic Cartridge Valve Actuator

## Dialog Box and Parameters

**Block Parameters: Hydraulic Cartridge Valve Actuator**

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 for flow consumption and loading forces except those of spring and pressure forces. Inertial properties of the actuator are accounted by adding a first order lag between the steady-state and actual valve displacements. The lag is simulated with the combination of the PS Gain and PS Integrator blocks enveloped by the unity feedback.

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

Connections A, B, and X are hydraulic conserving ports associated with the actuator ports. Connection P is a physical signal port whose output corresponds to 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 at port A and B.

**Parameters**

Port A poppet area:	<input type="text" value="3.3e-04"/>	<input type="text" value="m^2"/>
Port A to port X area ratio:	<input type="text" value="0.66"/>	
Preload force:	<input type="text" value="26"/>	<input type="text" value="N"/>
Spring rate:	<input type="text" value="1.4e+04"/>	<input type="text" value="N/m"/>
Poppet stroke:	<input type="text" value="0.005"/>	<input type="text" value="m"/>
Poppet-seat initial gap:	<input type="text" value="0"/>	<input type="text" value="m"/>
Time constant:	<input type="text" value="0.01"/>	<input type="text" value="s"/>
Actuator orientation:	<input type="text" value="Acts in positive direction"/>	

OK Cancel Help Apply

### Port A poppet area

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

### Port A to port X area ratio

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

### Preload force

Spring preload force. The default value is  $26 \text{ N}$ .

# Hydraulic Cartridge Valve Actuator

---

## Spring rate

Spring rate. The default value is  $1.4 \times 10^4$  N/m.

## Poppet stroke

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

## Poppet-seat initial gap

Initial gap between the poppet and the seat. The parameter value must be greater than or equal to zero. The default value is 0.

## Time constant

Time constant of the first-order lag. The default value is 0.01 s.

## Actuator orientation

Specifies actuator orientation with respect to the globally assigned positive direction. The actuator can be installed in two different ways, depending upon whether it moves the poppet in the positive or in the negative direction when pressure is applied at its inlet. If pressures applied at ports A and B move the poppet in the negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

## Restricted Parameters

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

- **Actuator orientation**

All other block parameters are available for modification.

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

# Hydraulic Cartridge Valve Actuator

---

- 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

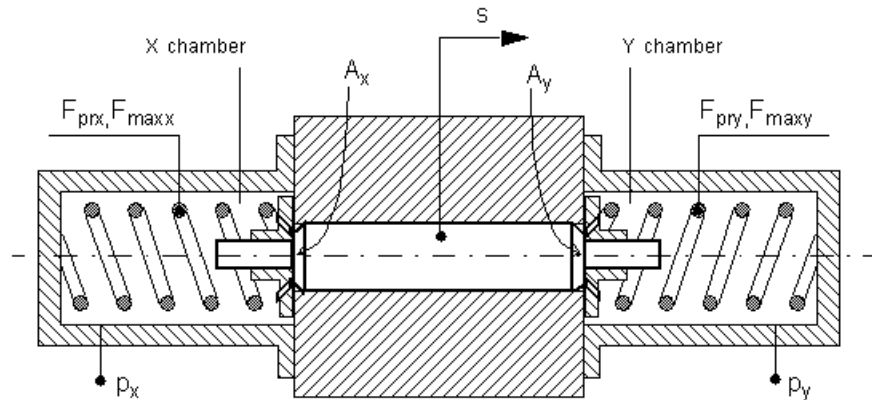
**Purpose** Double-acting hydraulic valve actuator

**Library** Valve Actuators

## Description



Use the Hydraulic Double-Acting Valve Actuator block as a pilot actuator for directional, pressure, or flow control valves in applications where all the forces, except spring force, and flow consumption can be neglected. The actuator consists of two single-acting actuators acting against each other. Each single-acting actuator consists of a piston, centering spring, and centering washer. When control pressure is applied to either hydraulic port, only one centering spring is compressed by its washer while the other butts against the valve body and exerts no force on the spool. When both control pressures are released, the springs force the washers against the valve body, and the spool centers between them. This design allows each actuator to have a different spring, preload force, and piston area.



As pressure applied to the piston develops enough force to overcome the spring preload, the piston moves to the opposite position until it reaches its maximum stroke. Pressure applied at port X shifts the valve in the x-direction, overcoming the spring located in the Y chamber. Pressure



# Hydraulic Double-Acting Valve Actuator

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

The actuator is simulated according to the following equations:

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

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

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

If  $F \geq 0$ ,

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

If  $F < 0$ ,

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

where

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

# Hydraulic Double-Acting Valve Actuator

---

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

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

## Basic Assumptions and Limitations

- The flow consumption associated with the valve motion is assumed to be negligible.
- The inertia, friction, and hydraulic axial forces are assumed to be small and are not taken into account.
- The clearances between the valve and the washers are not taken into account.

# Hydraulic Double-Acting Valve Actuator

## Dialog Box and Parameters

**Block Parameters: Hydraulic Double-Acting Valve Actuator**

Hydraulic Double-Acting Valve Actuator

This block represents a double-acting hydraulic valve actuator. Use it as a pilot actuator for directional, pressure, or flow control valves in applications where all the forces, except spring force, and flow consumption can be neglected.

The actuator consists of two single-acting actuators acting against each other. Each single-acting actuator consists of a piston, centering spring, and centering washer. When control pressure is applied to either hydraulic port, only one centering spring is compressed by its washer while the other butts against the valve body and exerts no force on the spool. When both control pressures are released, the springs force the washers against the valve body, and the spool centers between them. This design allows each actuator to have a different spring, preload force, and piston area. As pressure applied to the piston develops enough force to overcome the spring preload, the piston moves to the opposite position until it reaches its maximum stroke. Pressure applied at port X shifts the valve in the X-direction, overcoming the spring located in the Y chamber. Pressure applied at port Y shifts the valve in the Y-direction, overcoming the spring located in the X chamber.

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

**Parameters**

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

OK Cancel Help Apply

# Hydraulic Double-Acting Valve Actuator

---

## **Piston area at port X**

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

## **Piston area at port Y**

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

## **Preload force at port X**

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

## **Preload force at port Y**

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

## **Spring maximum force at port X**

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

## **Spring maximum force at port Y**

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

## **Piston stroke at port X**

Piston stroke in chamber X. The parameter value must be greater than or equal to zero. The default value is  $5e-3 \text{ m}$ .

## **Piston stroke at port Y**

Piston stroke in chamber Y. The parameter value must be greater than or equal to zero. The default value is  $5e-3 \text{ m}$ .

## **Actuator orientation**

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

# Hydraulic Double-Acting Valve Actuator

---

## Restricted Parameters

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

- **Actuator orientation**

All other block parameters are available for modification.

## Ports

The block has the following ports:

X

Hydraulic conserving port associated with the valve X chamber.

Y

Hydraulic conserving port associated with the valve Y chamber.

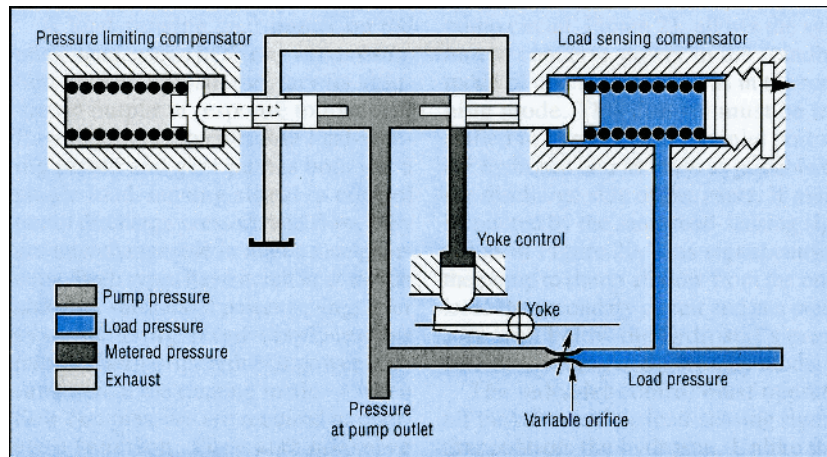
P

Physical signal port that outputs piston displacement.

## Examples

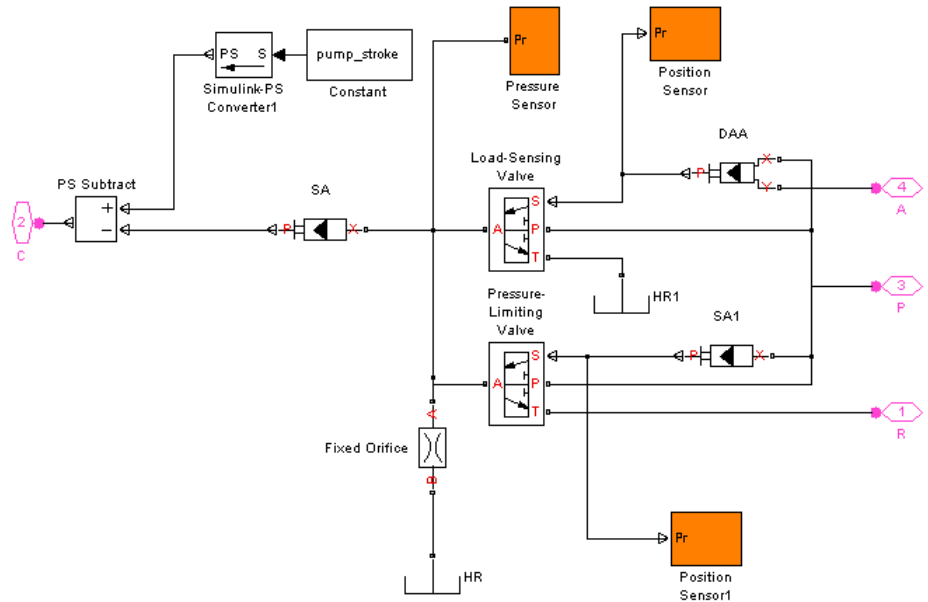
The following illustration shows a typical control unit of a variable-displacement pump that provides load sensing and pressure limiting (see [1]). In the unit, the load-sensing compensator varies the pump displacement to maintain a preset pressure differential across the variable orifice, while the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

# Hydraulic Double-Acting Valve Actuator



The Hydraulic Actuator with Load-Sensing Variable-Displacement Pump example implements this type of control. The next illustration shows the schematic of the Load-Sensing and Pressure-Limiting Control block in the example model.

# 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 example model to see the parameter settings for the blocks.

## References

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

# Hydraulic Double-Acting Valve Actuator

---

## **See Also**

2-Position Valve Actuator

3-Position Valve Actuator

Hydraulic Single-Acting Valve Actuator

Proportional and Servo-Valve Actuator



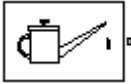
## Purpose

Working fluid properties, set by selecting from list of predefined fluids

## Library

Hydraulic Utilities

## Description



The Hydraulic Fluid block lets you specify the type of hydraulic fluid used in a loop of hydraulic blocks. It provides the hydraulic fluid properties, such as kinematic viscosity, density, and bulk modulus, for all the hydraulic blocks in the loop. These fluid properties are assumed to be constant during simulation time. The density is determined by the type of fluid, while kinematic viscosity additionally requires that the temperature is specified.

The bulk modulus value shown in the block dialog box is the bulk modulus of pure liquid, and is determined by the type of fluid and by the temperature. When the fluid properties are used in hydraulic blocks, such as Constant Volume Hydraulic Chamber or Variable Hydraulic Chamber, the fluid is represented as a mixture of liquid and a small amount of entrained, nondissolved gas, which is specified in the Hydraulic Fluid block as **Relative amount of trapped air**. The mixture bulk modulus in these blocks is determined as:

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

where

$E_l$  Pure liquid bulk modulus

$p_a$  Atmospheric pressure

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

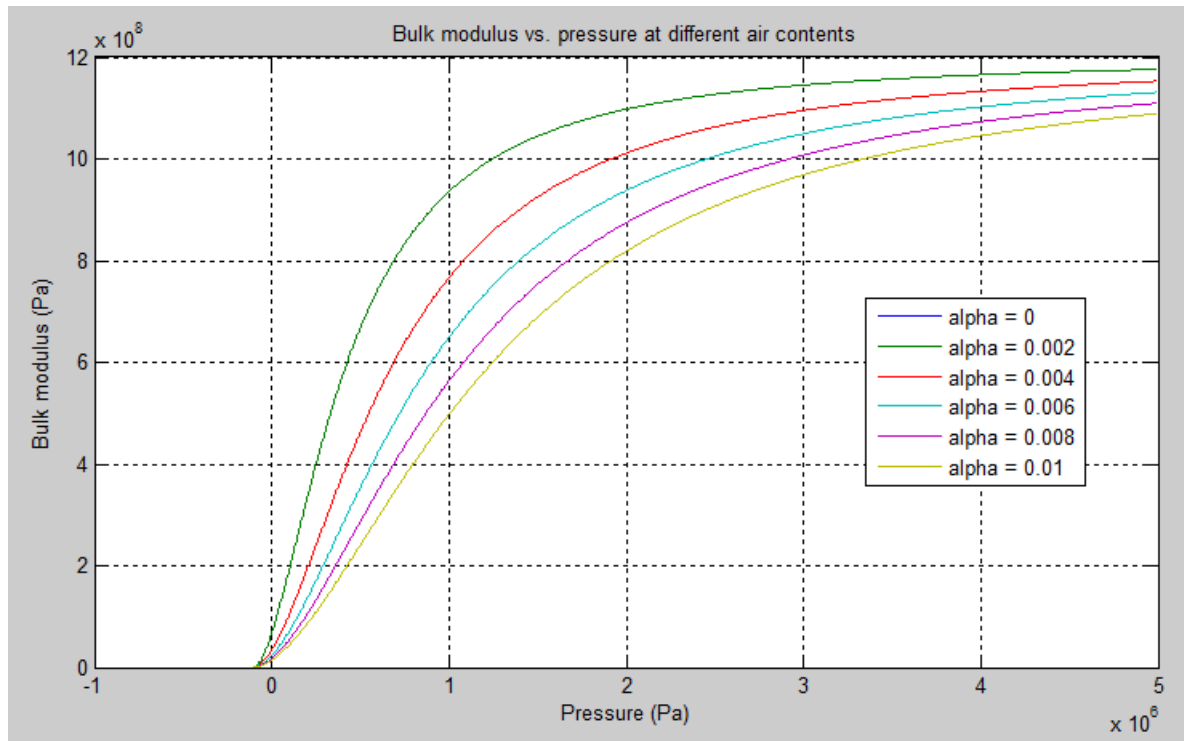
$V_g$  Gas volume at atmospheric pressure

# Hydraulic Fluid

$V_L$  Volume of liquid  
 $n$  Gas-specific heat ratio

The main objective of representing fluid as a mixture of liquid and gas is to introduce an approximate model of cavitation, which takes place in a chamber if pressure drops below fluid vapor saturation level. As it is seen in the graph below, the bulk modulus of a mixture decreases at  $p \rightarrow p_a$ , thus considerably slowing down further pressure change.

At high pressure,  $p \gg p_a$ , a small amount of nondissolved gas has practically no effect on the system behavior.



Cavitation is an inherently thermodynamic process, requiring consideration of multiple-phase fluids, heat transfers, and so on, 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.

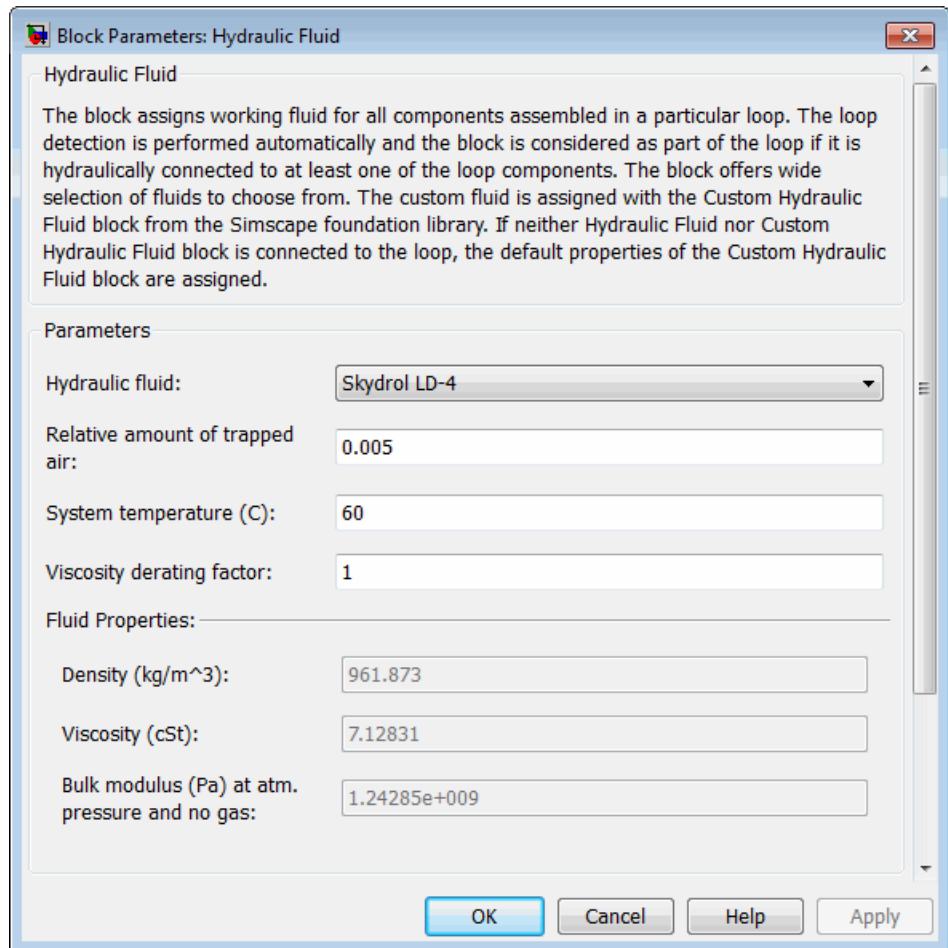
The Hydraulic Fluid block offers a selection of predefined fluids. See “Examples” on page 1-336 for how you can get information on the fluid properties used in the block. Once you select a fluid name, you can also specify the temperature of the fluid and the relative amount of entrained, nondissolved gas.

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

Each topologically distinct hydraulic loop in a diagram requires the properties of its working fluid to be specified. You can specify these properties either by selecting a predefined fluid using a Hydraulic Fluid block, or by specifying parameters of a custom fluid using a Custom Hydraulic Fluid block, which is available in the Simscape Foundation library. If no Hydraulic Fluid block or Custom Hydraulic Fluid block is attached to a loop, the hydraulic blocks in this loop use the default fluid, which is equivalent to fluid defined by a Custom Hydraulic Fluid block with the default parameter values.

# Hydraulic Fluid

## Dialog Box and Parameters



### Hydraulic fluid

Hydraulic fluid type. Select one of the predefined fluids:

- Skydrol LD-4 (default)
- Skydrol 500B-4

- Skydrol-5
- HyJet-4A
- Fluid MIL-F-83282
- Fluid MIL-F-5606
- Fluid MIL-F-87257
- Oil-10W
- Oil-30W
- Oil-50W
- Oil SAE-30
- Oil SAE-50
- Transmission fluid ATF (Dexron III)
- ISO VG 22 (ESSO UNIVIS N 22)
- ISO VG 32 (ESSO UNIVIS N 32)
- ISO VG 46 (ESSO UNIVIS N 46)
- Brake fluid DOT3
- Brake fluid DOT4
- Brake fluid DOT5
- Gasoline
- Diesel fuel
- Jet fuel
- Water-Glycol 60/40
- Water

### **Relative amount of trapped air**

Amount of entrained, nondissolved gas in the fluid. The amount is specified as the ratio of gas volume at normal conditions to the fluid volume in the chamber. In practice, the relative amount

# Hydraulic Fluid

---

of trapped air is always greater than 0. If set to 0, ideal fluid is assumed, but you will get a warning upon simulation. 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]
```

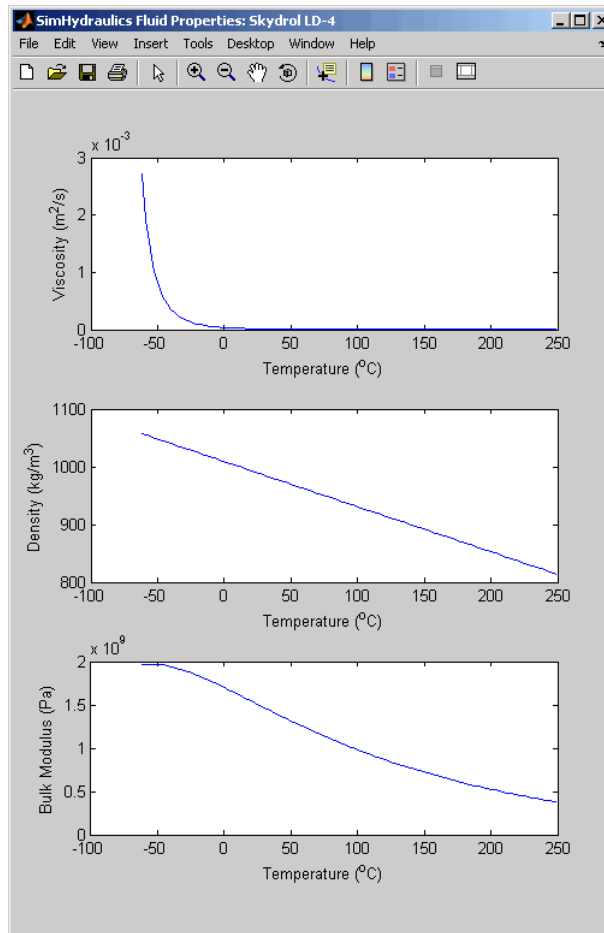
```
skydrol_5: [1x1 struct]
  hy_jet: [1x1 struct]
  f_83282: [1x1 struct]
  f_5606: [1x1 struct]
  f_87257: [1x1 struct]
  oil_10w: [1x1 struct]
  oil_30w: [1x1 struct]
  oil_50w: [1x1 struct]
oil_sae_30: [1x1 struct]
oil_sae_50: [1x1 struct]
atf_dexron: [1x1 struct]
  iso_vg_32: [1x1 struct]
  gasoline: [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:

# Hydraulic Fluid



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



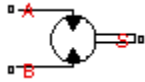
## Purpose

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:

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

$$T = Dp\eta_{mech}$$

$$k_{leak} = k_{HP} / v_p$$

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

$$p = p_A - p_B$$

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
$\omega$	Output shaft angular velocity
$k_{leak}$	Leakage coefficient

# Hydraulic Motor

---

$k_{HP}$	Hagen-Poiseuille coefficient
$\eta_V$	Motor volumetric efficiency
$\eta_{mech}$	Motor mechanical efficiency
$\nu$	Fluid kinematic viscosity
$\rho$	Fluid density
$p_{nom}$	Motor nominal pressure
$\omega_{nom}$	Motor nominal angular velocity
$\nu_{nom}$	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
$\mu$	Fluid dynamic viscosity, $\mu = \nu\rho$

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

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

which provides the formula to determine the Hagen-Poiseuille coefficient

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

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

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

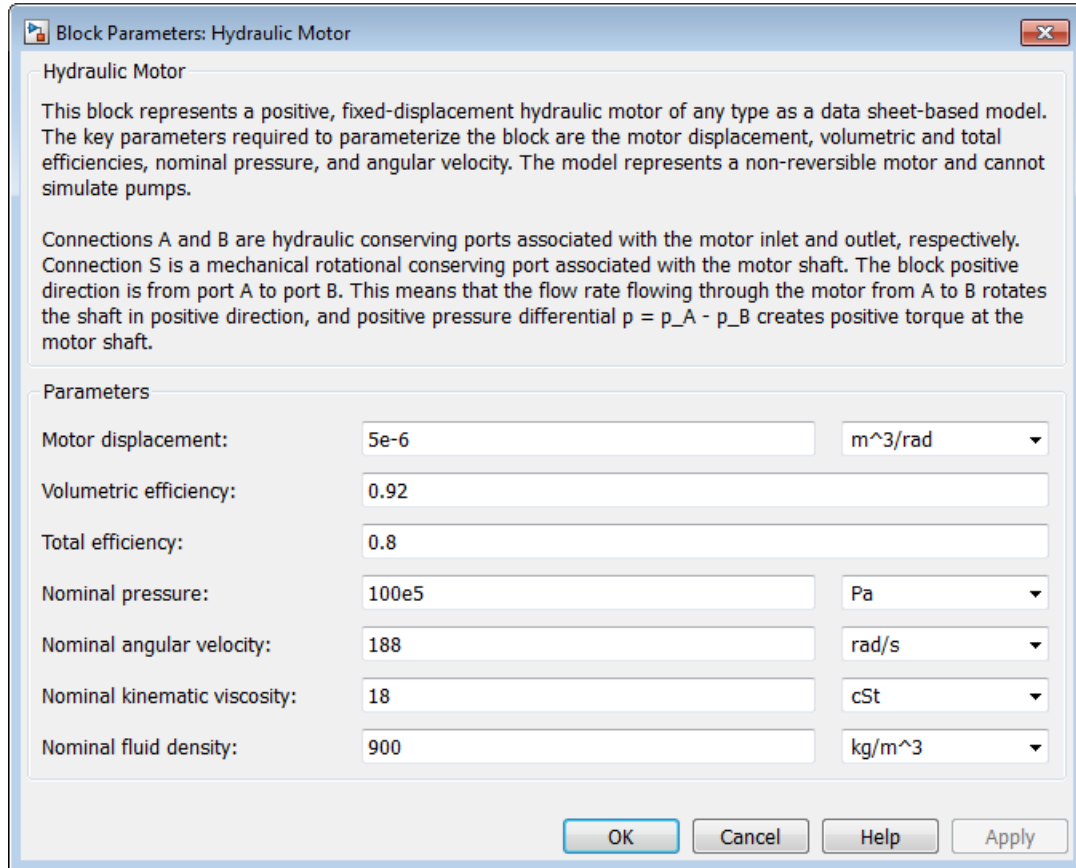
The block hydraulic positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and rotates the output shaft in the globally assigned positive direction. The pressure differential across the motor is determined as  $p = p_A - p_B$ , and positive pressure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

## **Basic Assumptions and Limitations**

- 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.

# Hydraulic Motor

## Dialog Box and Parameters



### Motor displacement

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

**Volumetric efficiency**

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

**Total efficiency**

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

**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.

**Nominal fluid density**

Working fluid density, at which both the volumetric and total efficiencies are specified. The default value is 900 kg/m<sup>3</sup>.

**Global Parameters**

Parameter determined by the type of working fluid:

- Fluid kinematic viscosity

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

**Ports**

The block has the following ports:

A

Hydraulic conserving port associated with the motor inlet.

# Hydraulic Motor

---

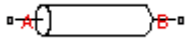
- B Hydraulic conserving port associated with the motor outlet.
- S Mechanical rotational conserving port associated with the motor output shaft.

**See Also** Variable-Displacement Motor

**Purpose** Hydraulic pipeline with resistive and fluid compressibility properties

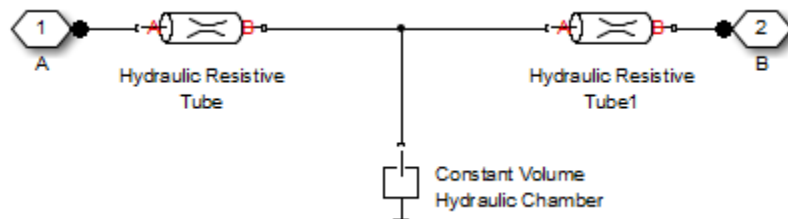
**Library** Pipelines

## Description



The Hydraulic Pipeline block models hydraulic pipelines with circular and noncircular cross sections. The block accounts for friction loss along the pipe length and for fluid compressibility. The block does not account for fluid inertia and cannot be used for predicting effects like water hammer or changes in pressure caused by fluid acceleration.

The model is built of Simscape Foundation library building blocks and its schematic diagram is shown below.



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

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

# Hydraulic Pipeline

## 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:	<input type="text" value="Circular"/>	<input type="button" value="v"/>
Pipe internal diameter:	<input type="text" value="0.01"/>	<input type="text" value="m"/> <input type="button" value="v"/>
Geometrical shape factor:	<input type="text" value="64"/>	
Pipe length:	<input type="text" value="5"/>	<input type="text" value="m"/> <input type="button" value="v"/>
Aggregate equivalent length of local resistances:	<input type="text" value="1"/>	<input type="text" value="m"/> <input type="button" value="v"/>
Internal surface roughness height:	<input type="text" value="1.5e-5"/>	<input type="text" value="m"/> <input type="button" value="v"/>
Laminar flow upper margin:	<input type="text" value="2e+3"/>	
Turbulent flow lower margin:	<input type="text" value="4e+3"/>	
Pipe wall type:	<input type="text" value="Rigid"/>	<input type="button" value="v"/>
Specific heat ratio:	<input type="text" value="1.4"/>	
Initial pressure:	<input type="text" value="0"/>	<input type="text" value="Pa"/> <input type="button" value="v"/>



**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.

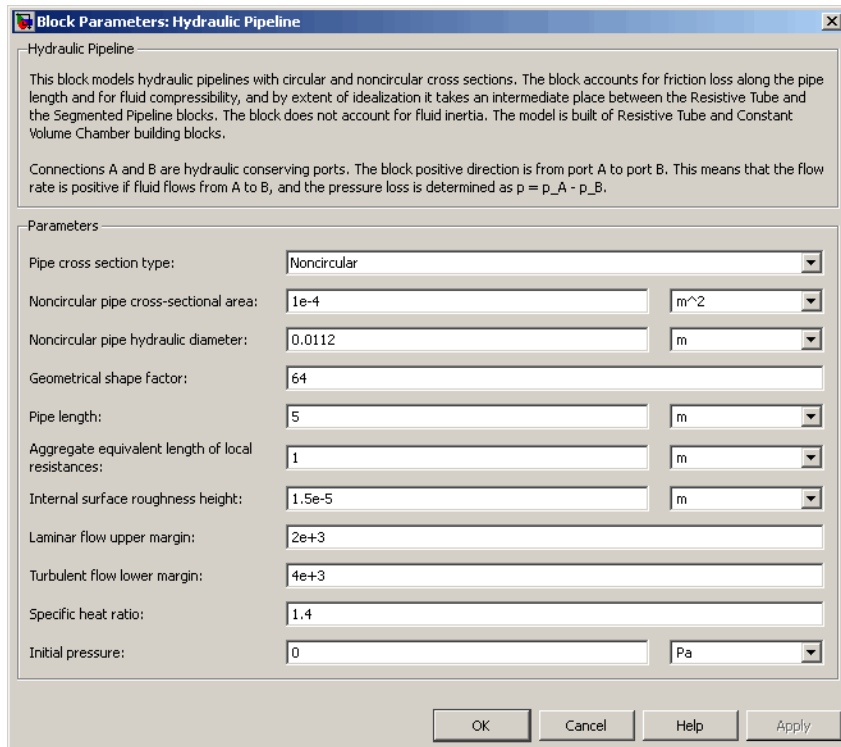
Connections A and B are hydraulic conserving ports. The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure loss is determined as  $p = p_A - p_B$ .

Parameters

Pipe cross section type:	<input type="text" value="Circular"/>	
Pipe internal diameter:	<input type="text" value="0.01"/>	<input type="text" value="m"/>
Geometrical shape factor:	<input type="text" value="64"/>	
Pipe length:	<input type="text" value="5"/>	<input type="text" value="m"/>
Aggregate equivalent length of local resistances:	<input type="text" value="1"/>	<input type="text" value="m"/>
Internal surface roughness height:	<input type="text" value="1.5e-5"/>	<input type="text" value="m"/>
Laminar flow upper margin:	<input type="text" value="2e+3"/>	
Turbulent flow lower margin:	<input type="text" value="4e+3"/>	
Pipe wall type:	<input type="text" value="Flexible"/>	
Static pressure-diameter coefficient:	<input type="text" value="2e-12"/>	<input type="text" value="m/Pa"/>
Viscoelastic process time constant:	<input type="text" value="0.01"/>	<input type="text" value="s"/>
Specific heat ratio:	<input type="text" value="1.4"/>	
Initial pressure:	<input type="text" value="0"/>	<input type="text" value="Pa"/>

OK Cancel Help Apply

# Hydraulic Pipeline



## Pipe cross section type

The type of pipe cross section: **Circular** or **Noncircular**. For a circular pipe, you specify its internal diameter. For a noncircular pipe, you specify its hydraulic diameter and pipe cross-sectional area. The default value of the parameter is **Circular**.

## Pipe internal diameter

Pipe internal diameter. The parameter is used if **Pipe cross section type** is set to **Circular**. The default value is 0.01 m.

## **Noncircular pipe cross-sectional area**

Pipe cross-sectional area. The parameter is used if **Pipe cross section type** is set to Noncircular. The default value is  $1e-4 \text{ m}^2$ .

## **Noncircular pipe hydraulic diameter**

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

## **Geometrical shape factor**

Used for computing friction factor at laminar flow. The shape of the pipe cross section determines the value. For a pipe with a noncircular cross section, set the factor to an appropriate value, for example, 56 for a square, 96 for concentric annulus, 62 for rectangle (2:1), and so on [1]. 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  $1.5e-5 \text{ m}$ , 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,

# Hydraulic Pipeline

---

this is the maximum Reynolds number at fully developed laminar flow. The default value is 2000.

## **Turbulent flow lower margin**

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

## **Pipe wall type**

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

## **Static pressure-diameter coefficient**

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

## **Viscoelastic process time constant**

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

## **Specific heat ratio**

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

## Initial pressure

Initial pressure in the pipe. This parameter specifies the initial condition for use in computing the block's initial state at the beginning of a simulation run. For more information, see “Initial Conditions Computation” in the *Simscape User's Guide*. The default value is 0.

## Restricted Parameters

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

- **Pipe cross section type**
- **Pipe wall type**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the values of the **Pipe cross section type** and **Pipe wall type** parameters at the time the model entered Restricted mode.

## Global Parameters

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

- A  
Hydraulic conserving port associated with the pipe inlet.
- B  
Hydraulic conserving port associated with the pipe outlet.

# Hydraulic Pipeline

---

## References

[1] White, F.M., *Viscous Fluid Flow*, McGraw-Hill, 1991

## See Also

Linear Hydraulic Resistance

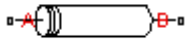
Hydraulic Resistive Tube

Segmented Pipeline

**Purpose** Hydraulic pipeline with resistive, fluid compressibility, and elevation properties

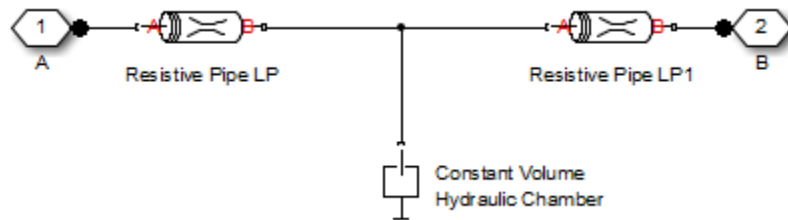
**Library** Low-Pressure Blocks

## Description



The Hydraulic Pipe LP block models hydraulic pipelines with circular and noncircular cross sections. The block accounts for friction loss along the pipe length and for fluid compressibility. The block does not account for fluid inertia and cannot be used for predicting effects like water hammer or changes in pressure caused by fluid acceleration.

The model is a structural model and its schematic diagram is shown below.



The Resistive Pipe LP blocks account for friction losses, while the Constant Volume Hydraulic Chamber block accounts for fluid compressibility. To reduce model complexity, you can use this block to simulate not only a pipe itself, but also a combination of pipes and local resistances such as bends, fittings, inlet and outlet losses, associated with the pipe. You must convert the resistances into their equivalent lengths, and then sum up all the resistances to obtain their aggregate length. Then add this length to the pipe geometrical length. By using the block parameters, you can set the model to simulate pipeline with rigid or compliant walls, including simulation of hydraulic hoses with elastic and viscoelastic properties.

The difference in elevation between ports A and B is distributed evenly between pipe segments.

# Hydraulic Pipe LP

---

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

## **Basic Assumptions and Limitations**

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

## **Dialog Box and Parameters**

The block dialog box contains three tabs:

- “Basic Parameters” on page 1-355
- “Wall Compliance” on page 1-358
- “Vertical Position” on page 1-361



## Basic Parameters

**Block Parameters: Hydraulic Pipe LP**

Hydraulic Pipe LP

This block models a hydraulic pipe with circular and noncircular cross sections. The block is intended to be used for low-pressure system simulation and, for this reason, requires elevation of both ports to be specified. 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 Pipe LP and the Segmented Pipe LP blocks. The block does not account for fluid inertia. The model is built of the Resistive Pipe LP and Constant Volume Chamber building blocks. To reduce model complexity, the block can be used to simulate not only a pipe itself, but also a combination of pipes and local resistances such as bends, fittings, inlet and outlet losses, etc., associated with the pipe. The resistances must be converted into their equivalent lengths, and then the total length of all the resistances is added to the pipe geometrical length. 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

Basic parameters | Wall compliance | Vertical position

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

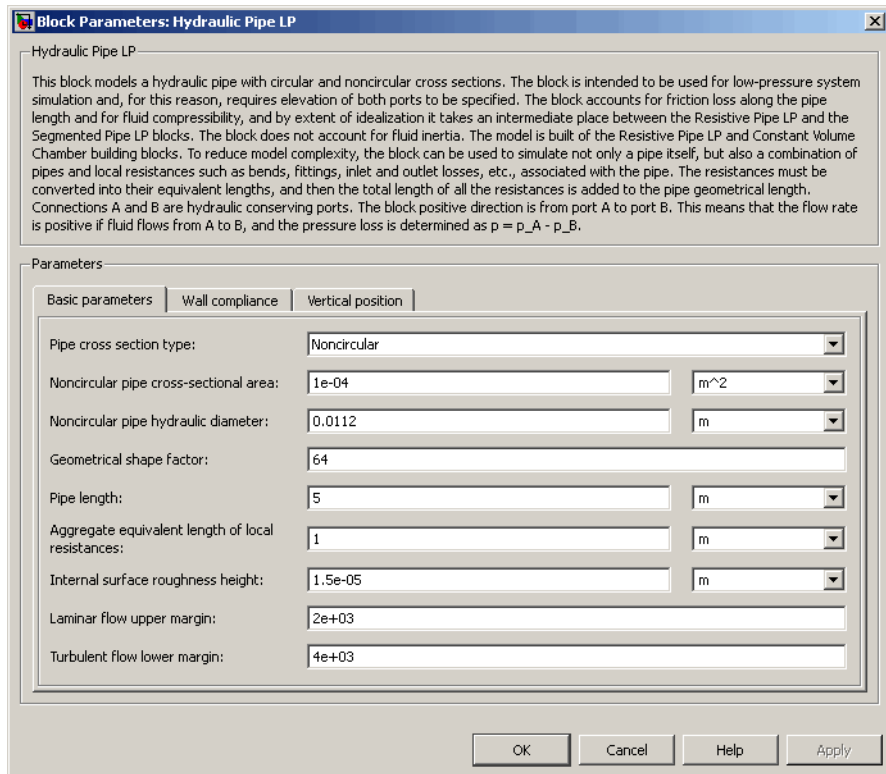
Internal surface roughness height: 1.5e-05 m

Laminar flow upper margin: 2e+03

Turbulent flow lower margin: 4e+03

OK Cancel Help Apply

# Hydraulic Pipe LP



## Pipe cross section type

The type of pipe cross section: **Circular** or **Noncircular**. For a circular pipe, you specify its internal diameter. For a noncircular pipe, you specify its hydraulic diameter and pipe cross-sectional area. The default value of the parameter is **Circular**.

## Pipe internal diameter

Pipe internal diameter. The parameter is used if **Pipe cross section type** is set to **Circular**. The default value is 0.01 m.

## **Noncircular pipe cross-sectional area**

Pipe cross-sectional area. The parameter is used if **Pipe cross section type** is set to Noncircular. The default value is  $1e-4 \text{ m}^2$ .

## **Noncircular pipe hydraulic diameter**

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

## **Geometrical shape factor**

Used for computing friction factor at laminar flow. The shape of the pipe cross section determines the value. For a pipe with a noncircular cross section, set the factor to an appropriate value, for example, 56 for a square, 96 for concentric annulus, 62 for rectangle (2:1), and so on [1]. 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  $1.5e-5 \text{ m}$ , 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,

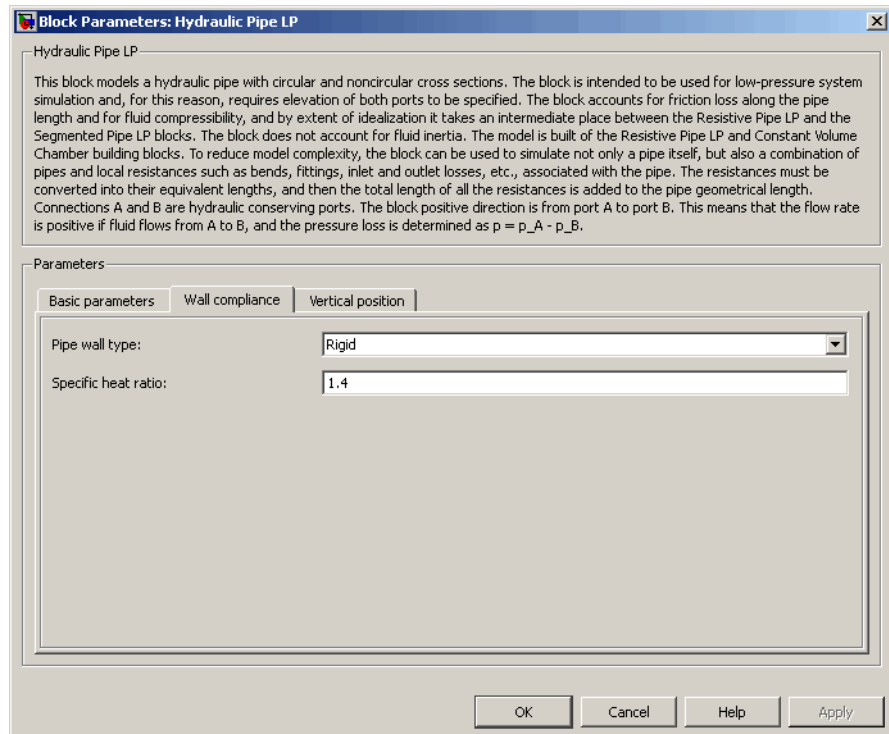
# Hydraulic Pipe LP

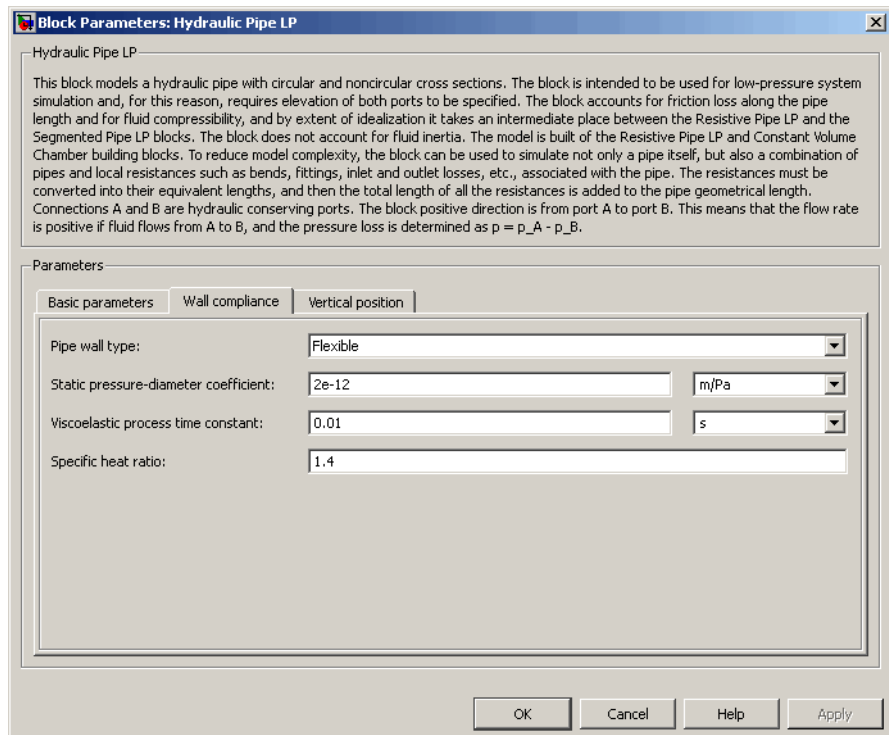
this is the maximum Reynolds number at fully developed laminar flow. The default value is 2000.

## 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.

## Wall Compliance





## 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**

# Hydraulic Pipe LP

---

**type** parameter is set to **Flexible**. The default value is  $2e-12$  m/Pa.

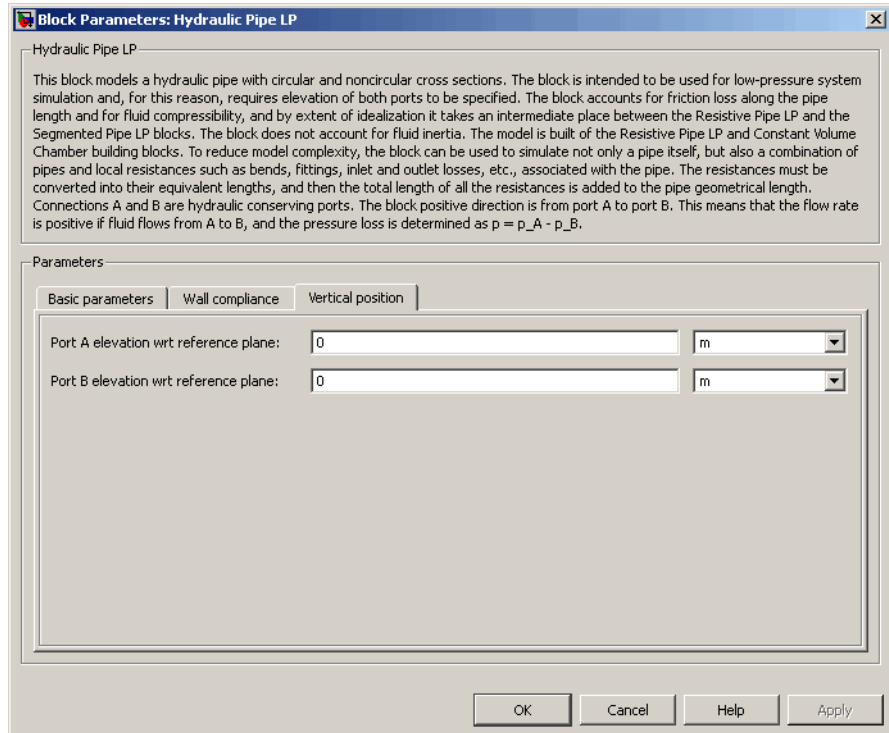
## **Viscoelastic process time constant**

Time constant in the transfer function that relates pipe internal diameter to pressure variations. By using this parameter, the simulated elastic or viscoelastic process is approximated with the first-order lag. The value is determined experimentally or provided by the manufacturer. The parameter is used if the **Pipe wall type** parameter is set to **Flexible**. The default value is 0.01 s.

## **Specific heat ratio**

Gas-specific heat ratio for the Constant Volume Hydraulic Chamber block. The default value is 1.4. If **Pipe cross section type** is set to **Noncircular**, then this is the only parameter on the Wall Compliance tab.

## Vertical Position



### Port A elevation wrt reference plane

The parameter specifies vertical position of the pipe port A with respect to the reference plane. The default value is 0.

### Port B elevation wrt reference plane

The parameter specifies vertical position of the pipe port B with respect to the reference plane. The default value is 0.

### Restricted Parameters

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

# Hydraulic Pipe LP

---

- **Pipe cross section type**
- **Pipe wall type**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the values of the **Pipe cross section type** and **Pipe wall type** parameters at the time the model entered Restricted mode.

## Global Parameters

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the pipe inlet.

B

Hydraulic conserving port associated with the pipe outlet.

## References

[1] White, F.M., *Viscous Fluid Flow*, McGraw-Hill, 1991

## See Also

Hydraulic Pipeline

Hydraulic Pipe LP with Variable Elevation

Linear Hydraulic Resistance

Resistive Pipe LP

Resistive Pipe LP with Variable Elevation

Hydraulic Resistive Tube

Segmented Pipeline



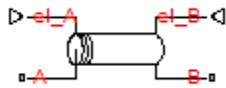
Segmented Pipe LP

# Hydraulic Pipe LP with Variable Elevation

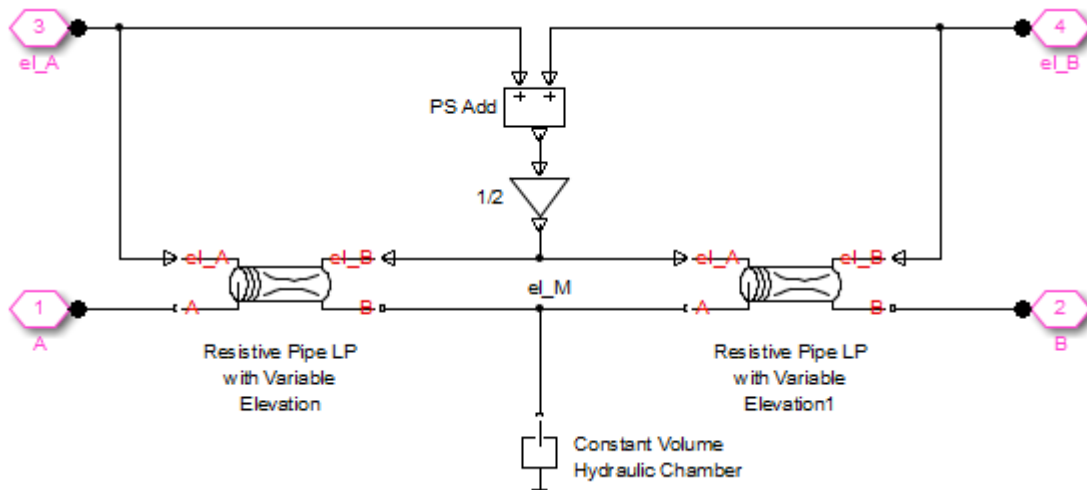
**Purpose** Hydraulic pipeline with resistive, fluid compressibility, and variable elevation properties

**Library** Low-Pressure Blocks

**Description** The Hydraulic Pipe LP with Variable Elevation block models hydraulic pipelines with circular and noncircular cross sections. The block accounts for friction loss along the pipe length and for fluid compressibility. The block does not account for fluid inertia and cannot be used for predicting effects like water hammer or changes in pressure caused by fluid acceleration. Use this block for low-pressure system simulation in which the pipe ends change their positions with respect to the reference plane. The elevations are provided through respective physical signal inputs.



The model is a structural model and its schematic diagram is shown below.



The Resistive Pipe LP with Variable Elevation blocks account for friction losses, while the Constant Volume Hydraulic Chamber block

# Hydraulic Pipe LP with Variable Elevation

accounts for fluid compressibility. To reduce model complexity, you can use this block to simulate not only a pipe itself, but also a combination of pipes and local resistances such as bends, fittings, inlet and outlet losses, associated with the pipe. You must convert the resistances into their equivalent lengths, and then sum up all the resistances to obtain their aggregate length. Then add this length to the pipe geometrical length. By using the block parameters, you can set the model to simulate pipeline with rigid or compliant walls, including simulation of hydraulic hoses with elastic and viscoelastic properties.

The difference in elevation between ports A and B is assumed to be distributed evenly between pipe segments. Thus, the elevation of the pipe middle point is computed as

$$el\_M = \frac{el\_A + el\_B}{2}$$

where

$el\_M$  Elevation of the pipe middle point

$el\_A$ , Elevations of the pipe ends A and B, respectively

$el\_B$

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

## Basic Assumptions and Limitations

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

# Hydraulic Pipe LP with Variable Elevation

## Dialog Box and Parameters

The block dialog box contains two tabs:

- “Basic Parameters” on page 1-366
- “Wall Compliance” on page 1-369

### Basic Parameters

**Block Parameters: Hydraulic Pipe LP with Variable Elevation**

Hydraulic Pipe LP with Variable Elevation

This block models a hydraulic pipe with circular and noncircular cross sections. The block is intended to be used for low-pressure system simulation in which pipe ends change their positions. The ends elevations are provided through their respective physical signal ports. The block accounts for friction loss along the pipe length and for fluid compressibility. The block does not account for fluid inertia. The model is built of the Resistive Pipe LP with Variable Elevation and Constant Volume Chamber building blocks. To reduce model complexity, the block can be used to simulate not only a pipe itself, but also a combination of pipes and local resistances such as bends, fittings, inlet and outlet losses, etc., associated with the pipe. The resistances must be converted into their equivalent lengths, and then the total length of all the resistances is added to the pipe geometrical length. Connections A and B are hydraulic conserving ports. Connections el\_A and el\_B are the physical signal port through which pipe ends elevations are imported. 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

Basic parameters | Wall compliance

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

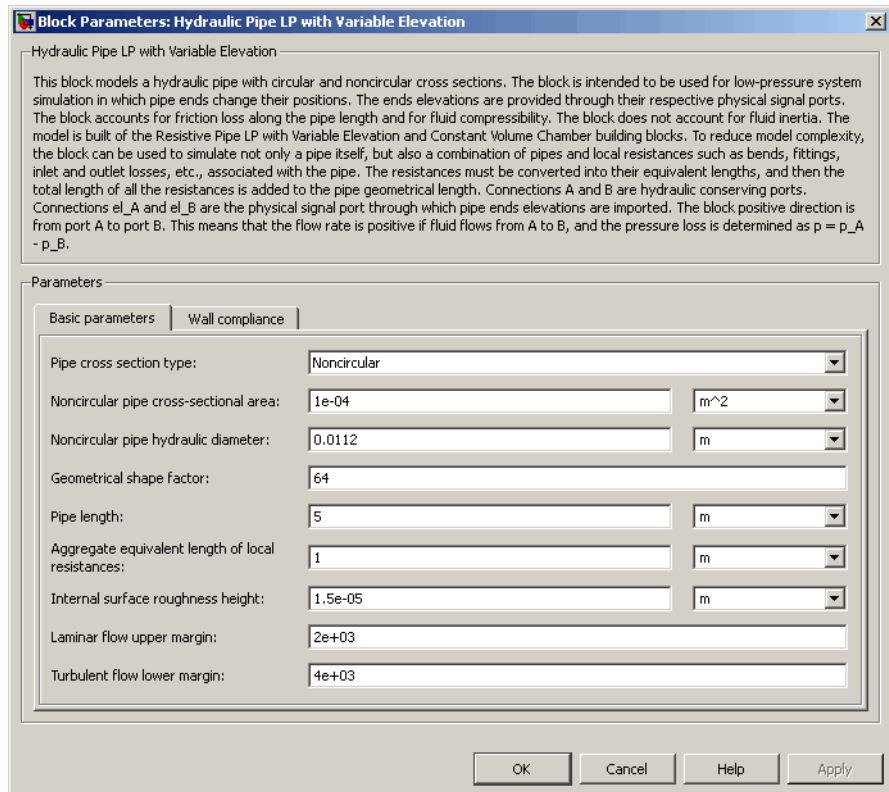
Internal surface roughness height: 1.5e-05 m

Laminar flow upper margin: 2e+03

Turbulent flow lower margin: 4e+03

OK Cancel Help Apply

# Hydraulic Pipe LP with Variable Elevation



## Pipe cross section type

The type of pipe cross section: **Circular** or **Noncircular**. For a circular pipe, you specify its internal diameter. For a noncircular pipe, you specify its hydraulic diameter and pipe cross-sectional area. The default value of the parameter is **Circular**.

## Pipe internal diameter

Pipe internal diameter. The parameter is used if **Pipe cross section type** is set to **Circular**. The default value is 0.01 m.

# Hydraulic Pipe LP with Variable Elevation

---

## **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<sup>2</sup>.

## **Noncircular pipe hydraulic diameter**

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

## **Geometrical shape factor**

Used for computing friction factor at laminar flow. The shape of the pipe cross section determines the value. For a pipe with a noncircular cross section, set the factor to an appropriate value, for example, 56 for a square, 96 for concentric annulus, 62 for rectangle (2:1), and so on [1]. 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,

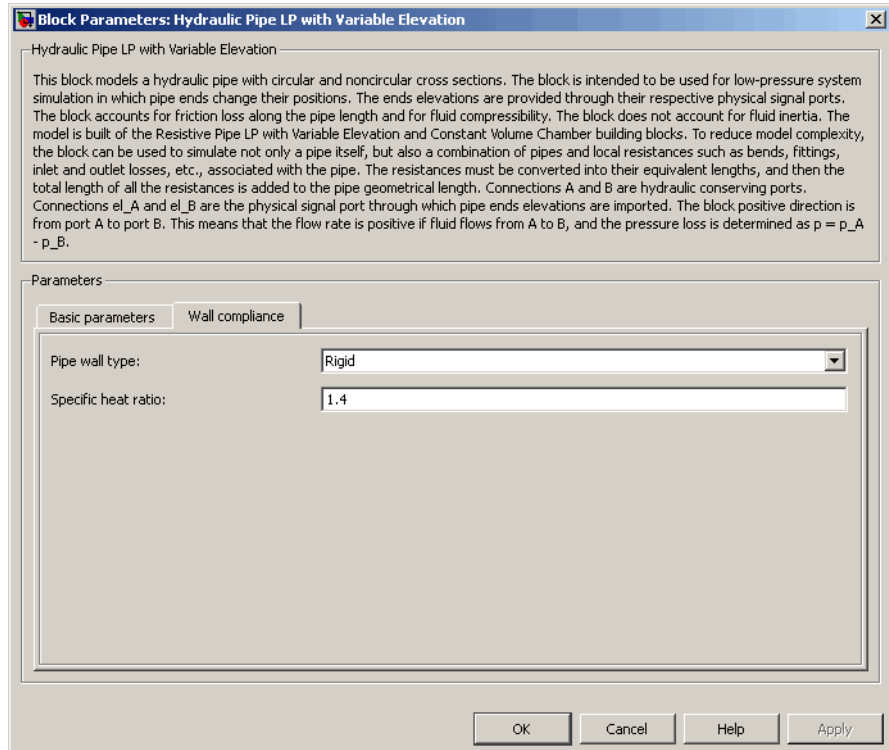
# Hydraulic Pipe LP with Variable Elevation

this is the maximum Reynolds number at fully developed laminar flow. The default value is 2000.

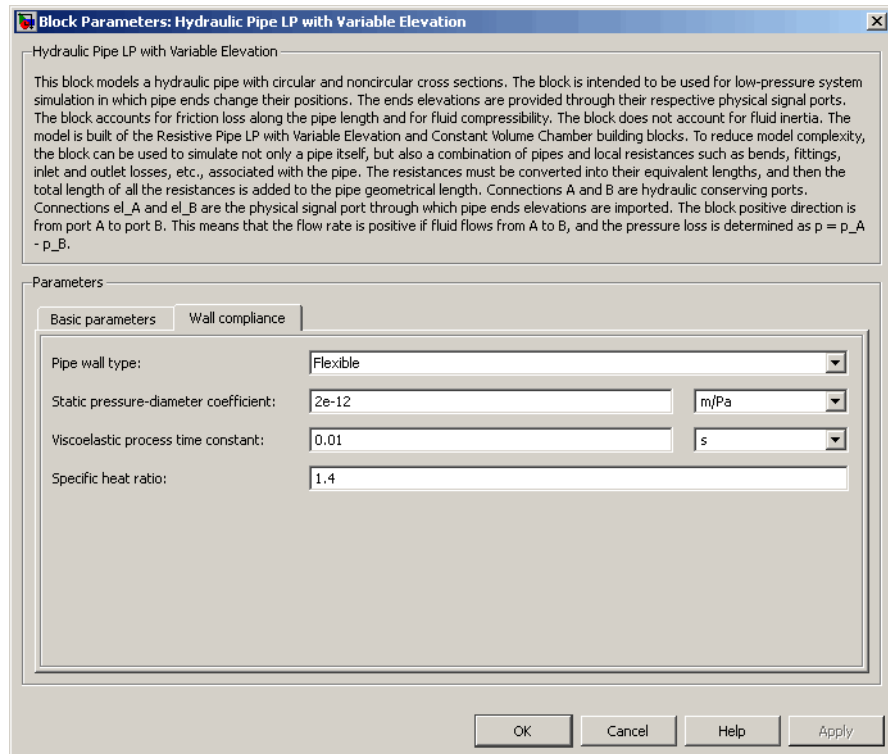
## Turbulent flow lower margin

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.

## Wall Compliance



# Hydraulic Pipe LP with Variable Elevation



## 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**



# Hydraulic Pipe LP with Variable Elevation

---

**type** parameter is set to **Flexible**. The default value is  $2e-12$  m/Pa.

## Viscoelastic process time constant

Time constant in the transfer function that relates pipe internal diameter to pressure variations. By using this parameter, the simulated elastic or viscoelastic process is approximated with the first-order lag. The value is determined experimentally or provided by the manufacturer. The parameter is used if the **Pipe wall type** parameter is set to **Flexible**. The default value is 0.01 s.

## Specific heat ratio

Gas-specific heat ratio for the Constant Volume Hydraulic Chamber block. The default value is 1.4. If **Pipe cross section type** is set to **Noncircular**, then this is the only parameter on the Wall Compliance tab.

## Global Parameters

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

- A  
Hydraulic conserving port associated with the pipe inlet.
- B  
Hydraulic conserving port associated with the pipe outlet.
- e1\_A  
Physical signal input port that controls pipe elevation at port A.
- e1\_B  
Physical signal input port that controls pipe elevation at port B.

# Hydraulic Pipe LP with Variable Elevation

---

## **Examples**

For an example of using this block, see the Fuel Supply System with Variable Elevation example.

## **References**

[1] White, F.M., *Viscous Fluid Flow*, McGraw-Hill, 1991

## **See Also**

Hydraulic Pipeline

Hydraulic Pipe LP

Hydraulic Resistive Tube

Linear Hydraulic Resistance

Resistive Pipe LP

Resistive Pipe LP with Variable Elevation

Segmented Pipeline

Segmented Pipe LP

# Hydraulic Single-Acting Valve Actuator

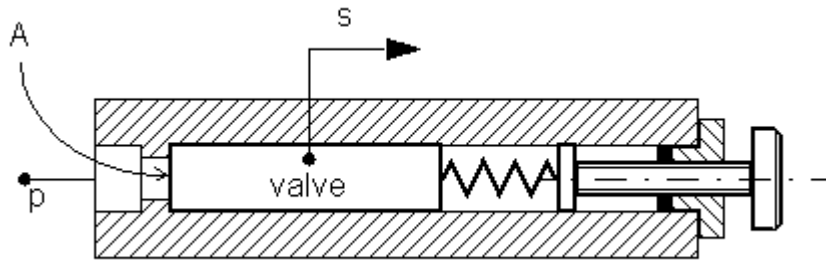
**Purpose** Single-acting hydraulic valve actuator

**Library** Valve Actuators

## Description



Use the Hydraulic Single-Acting Valve Actuator block as a pilot actuator for directional, pressure, or flow control valves in applications where all the forces, except spring force, and flow consumption can be neglected.



The actuator consists of a piston and a spring. The spring, which can be preloaded, tends to keep the piston at the initial position. As pressure applied to the piston develops enough force to overcome the spring preload, the piston moves to the opposite position until it reaches its maximum stroke.

The actuator is simulated according to the following equations:

$$F = p \cdot A$$

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

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

# Hydraulic Single-Acting Valve Actuator

---

where

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

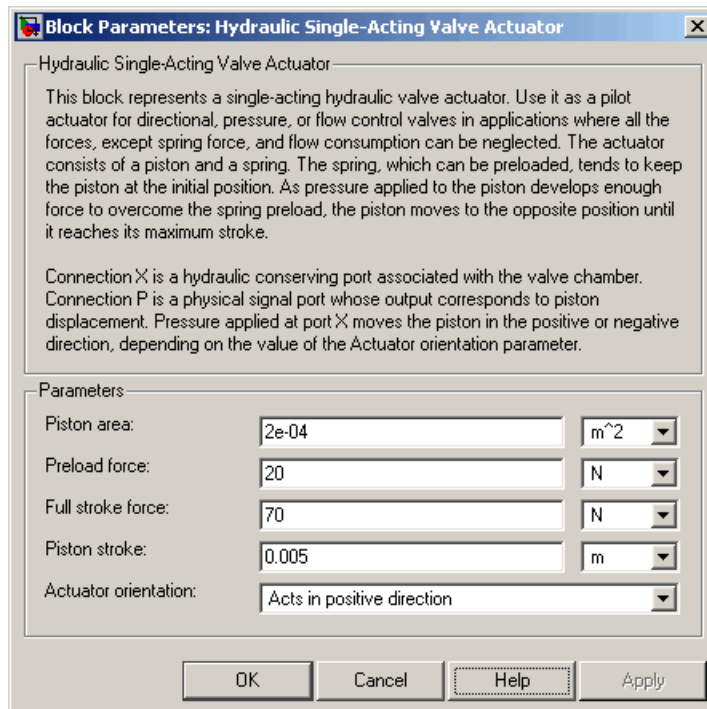
Connection X is a hydraulic conserving port associated with the valve chamber. Connection P is a physical signal port whose output corresponds to piston displacement. Pressure applied at port X moves the piston in the positive or negative direction, depending on the value of the **Actuator orientation**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

## Basic Assumptions and Limitations

- No loading, such as inertia, friction, hydraulic force, and so on, is taken into account. The only force considered is a spring force.
- No flow consumption associated with the piston motion, leakage, or fluid compressibility is taken into account.

# Hydraulic Single-Acting Valve Actuator

## Dialog Box and Parameters



### Piston area

Effective piston area. The default value is  $2e-4 \text{ m}^2$ .

### Preload force

Spring preload force. The default value is 20 N.

### Full stroke force

Force necessary to move the piston to maximum stroke. The default value is 70 N.

### Piston stroke

Piston stroke. The default value is  $5e-3 \text{ m}$ .

### Actuator orientation

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

# Hydraulic Single-Acting Valve Actuator

---

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

## Restricted Parameters

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

- **Actuator orientation**

All other block parameters are available for modification.

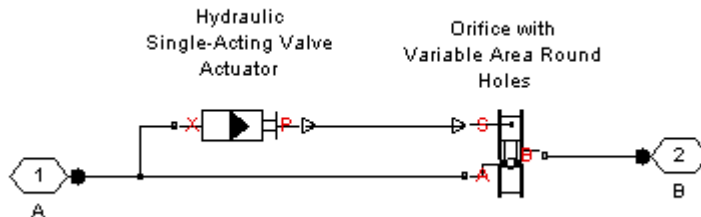
## Ports

The block has the following ports:

- X  
Hydraulic conserving port associated with the valve chamber.
- P  
Physical signal port that outputs piston displacement.

## Examples

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



# Hydraulic Single-Acting Valve Actuator

---

## **See Also**

2-Position Valve Actuator

3-Position Valve Actuator

Hydraulic Double-Acting Valve Actuator

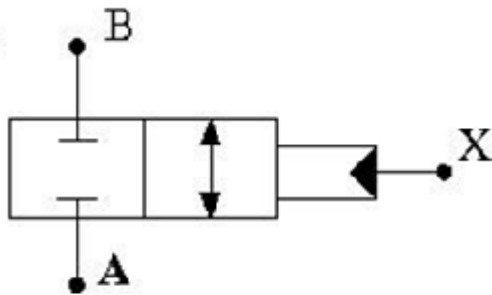
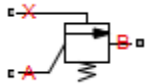
Proportional and Servo-Valve Actuator

# Hydraulically Operated Remote Control Valve

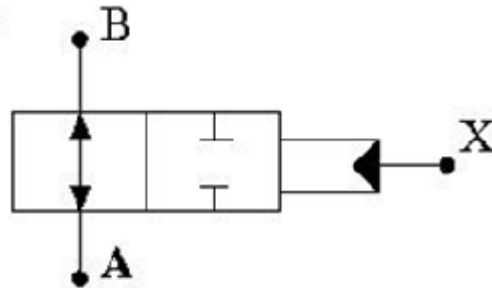
**Purpose** Normally closed or normally open hydraulically operated remote control valve

**Library** Directional Valves

**Description** The Hydraulically Operated Remote Control Valve block represents a hydraulically operated remote control valve as a data-sheet-based model, meaning that most of the model parameters are generally available in catalogs or manufacturer data sheets. Hydraulically operated remote control valves are widely used in hydraulic systems as hydraulic switches, unloading and sequencing valves. You can also use them as pressure-relief and pressure-reducing valves. The block covers both the normally closed and normally open configurations, shown in the following figure.



Normally closed  
valve configuration



Normally open  
valve configuration

The valve opens (closes) by the pilot pressure. The valve control member remains in its initial position as long as the pilot pressure is lower than the cracking pressure. When cracking pressure is reached, the valve control member (spool, ball, poppet, and so on) is forced off its seat and starts opening the normally closed valve, or closing the normally open valve. The control member displacement is directly proportional to pilot pressure. The member reaches its maximum displacement after



# Hydraulically Operated Remote Control Valve

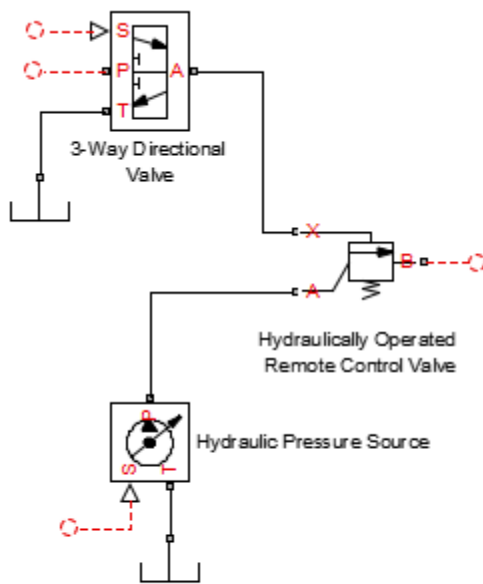
---

the pilot pressure becomes equal or greater than the preset maximum value. The valve maximum area, cracking pressure, and maximum pressure are the key parameters of the block. These three parameters are usually provided in catalogs or data sheets.

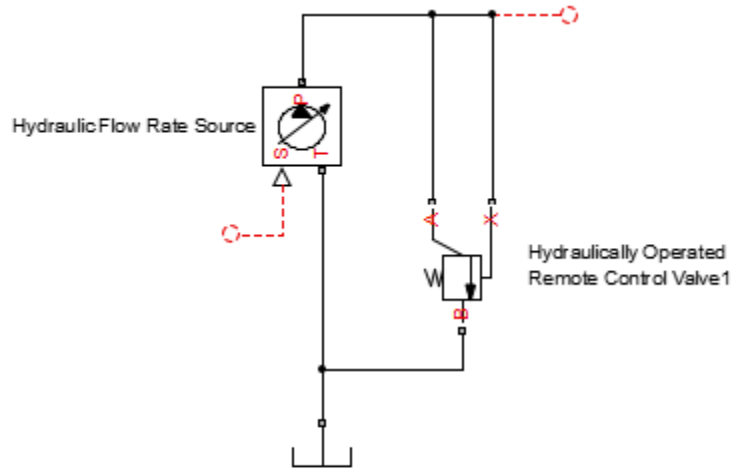
In addition to the maximum area, the leakage area is also required to characterize the valve. The main purpose of the parameter is not to account for possible leakage, even though this is also important, but to maintain numerical integrity of the circuit by preventing a portion of the system from getting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

Schematic fragments in the next illustration show some typical valve applications: remote control valve (a), pressure-relief valve (b), and pressure-reducing valve (c).

# Hydraulically Operated Remote Control Valve



a) Remote control valve



b) Pressure-relief valve

The flow rate through the orifice is proportional to the orifice opening and 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:

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

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

# Hydraulically Operated Remote Control Valve

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

For the normally open valve, the equations are similar:

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

The rest of the equations apply to both valve configurations:

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

$$p = p_A - p_B$$

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

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

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

where

- $q$  Flow rate through the valve
- $p$  Pressure differential across the valve
- $p_A, p_B$  Gauge pressures at the block terminals

# Hydraulically Operated Remote Control Valve

---

$p_p$	Gage pressure at the pilot terminal
$C_D$	Flow discharge coefficient
$A(p)$	Instantaneous orifice passage area
$A_{\max}$	Fully open valve passage area
$A_{\text{leak}}$	Closed valve leakage area
$p_{\text{crack}}$	Valve cracking pressure
$p_{\max}$	Pilot pressure to shift the control member to its maximum
$D_H$	Instantaneous orifice hydraulic diameter
$\rho$	Fluid density
$\nu$	Fluid kinematic viscosity

Connections A and B are hydraulic conserving ports associated with the inlet and the outlet of the valve. Connection X is the pilot port, which is a hydraulic conserving port that provides the pilot pressure. The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as  $p = p_A - p_B$ .

## Basic Assumptions and Limitations

- Control member displacement is linearly proportional to pilot pressure.
- No flow consumption is associated with the pilot chamber.
- No loading on the valve, such as inertia, friction, spring, and so on, is considered.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at  $Re=Re_{cr}$ .

# Hydraulically Operated Remote Control Valve

## Dialog Box and Parameters

Block Parameters: Hydraulically Operated Remote Control Valve

Hydraulically Operated Remote Control Valve

This block represents a hydraulically operated remote control valve. The valve control member (spool, poppet) can be shifted from its initial position after pressure at the pilot port exceeds the cracking pressure limit. The control member moves to its extreme position and remains at this position as the pilot pressure becomes equal or greater than the maximum pressure setting. The valve passage area is assumed to be directly proportional to pilot pressure. The model covers both normally-closed and normally-open valve configurations. The pilot pressure opens the normally-closed valve while the normally-open valve requires pilot pressure increase to be closed. Connections A, B, and X are hydraulic conserving ports associated with the valve inlet, valve outlet, and valve pilot terminals, respectively. The block positive direction is from port A to port B. Pressure at port X acts to open the normally-closed valve or close the normally-opened valve.

Parameters

Valve type:	Normally closed valve	
Maximum passage area:	1e-4	m <sup>2</sup>
Cracking pressure:	3e+4	Pa
Maximum control member displacement pressure:	1.2e+5	Pa
Flow discharge coefficient:	0.7	
Critical Reynolds number:	12	
Leakage area:	1e-9	m <sup>2</sup>

OK Cancel Help Apply

# Hydraulically Operated Remote Control Valve

---

## **Valve type**

Select the valve configuration: Normally closed valve or Normally open valve. The default is Normally closed valve.

## **Maximum passage area**

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

## **Cracking pressure**

Pressure level at which the valve control member is forced off its seat and starts to either open or close the valve, depending on the valve type. The default value is  $3e4 \text{ Pa}$ .

## **Maximum control member displacement pressure**

Pilot pressure at which the valve control member shifts to its maximum displacement and remains there until the pressure falls below this level. Its value must be higher than the cracking pressure. The default value is  $1.2e5 \text{ Pa}$ .

## **Flow discharge coefficient**

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

## **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 simulation to fail. Therefore, MathWorks

# Hydraulically Operated Remote Control Valve

---

recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## Global Parameters

Parameters determined by the type of working fluid:

- Fluid density
- Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

X

Hydraulic conserving port that acts as the control port and provides the pilot pressure.

## See Also

Pressure Reducing Valve

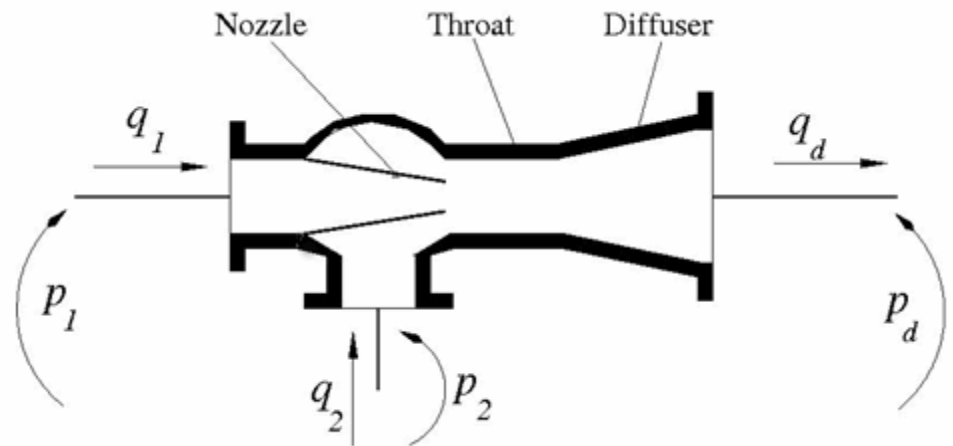
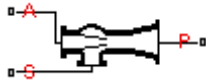
Pressure Relief Valve

# Jet Pump

**Purpose** Jet liquid-liquid pump

**Library** Pumps and Motors

**Description** The Jet Pump block represents a jet liquid-liquid pump consisting of a nozzle, throat, and diffuser, as shown in the following illustration.



The model is based on the following equations, described in [1]:



$$q_1 = \frac{A_n}{\sqrt{1 + K_n}} \sqrt{\frac{2}{\rho} (p_1 - p_0)} \quad (1-8)$$

$$q_2 = \frac{A_n \cdot c}{\sqrt{1 + K_{en}}} \sqrt{\frac{2}{\rho} (p_2 - p_0)} \quad (1-9)$$

$$p_d - p_0 = Zb^2 \left( \frac{2}{b} + \frac{2}{1-b} M^2 - (1+M)^2 \cdot (1 + K_{th} + K_{di} + a^2) \right) \quad (1-10)$$

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

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

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

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

where

- $q_1$  Primary flow rate pumped through the nozzle
- $q_2$  Secondary flow rate
- $q_d$  Output flow rate
- $p_1$  Pressure at the nozzle inlet
- $p_2$  Pressure at the secondary flow rate inlet
- $p_0$  Pressure at the throat inlet
- $p_d$  Pressure at the pump outlet

# Jet Pump

---

$A_n$	Nozzle area
$A_{th}$	Throat area
$a$	Diffuser area ratio, $A_{th} / A_d$
$A_d$	Diffuser outlet area
$K_n$	Nozzle hydraulic loss coefficient
$K_{en}$	Throat entry hydraulic loss coefficient
$K_{th}$	Throat hydraulic loss coefficient
$K_{di}$	Diffuser hydraulic loss coefficient
$\rho$	Fluid density

Equation 1-8 describes the nozzle, Equation 1-9 – throat entry, and Equation 1-10 – the combination of the throat and the diffuser. The equations correspond to a standard configuration of the pump, where all the longitudinal dimensions conform to established, empirically determined values. For more details, see [1].

The pump parameters are closely related to each other, and the methodology described in [1] is recommended to determine their initial values.

## Basic Assumptions and Limitations

- The model is based on the one-dimensional theory.
- The primary and secondary flows enter the mixing throat with uniform velocity distribution, and the mixed flow leaves the diffuser with uniform velocity distribution.
- The fluid in the primary and secondary flows is the same.
- The fluid is assumed to be incompressible and containing no gas.

## Dialog Box and Parameters

**Block Parameters: Jet Pump**

Jet Pump

This block represents a jet liquid-liquid pump operating with identical primary and secondary fluids containing no gas. The longitudinal pump dimensions conform to standard pump sizes summarized in the "Pump Handbook" by I. J. Karassik et al. Connections A and S are the conserving hydraulic ports associated with the inlet of the primary and secondary fluids respectively. Connection P is the conserving hydraulic port associated with the pump outlet. Internal node N is associated with the throat entry point of the pump.

Parameters

Nozzle area:	1e-4	m <sup>2</sup>
Throat area:	4e-4	m <sup>2</sup>
Diffuser inlet/outlet area ratio:	0.224	
Nozzle loss coefficient:	0.05	
Throat entry loss coefficient:	0.005	
Throat loss coefficient:	0.1	
Diffuser loss coefficient:	0.1	

OK Cancel Help Apply

### Nozzle area

Cross-sectional area of the nozzle. The parameter must be greater than zero. The default value is 1 cm<sup>2</sup>.

### Throat area

Cross-sectional area of the throat. The throat area is usually two to four times larger than the nozzle area. The default value is 4 cm<sup>2</sup>.

### Diffuser inlet/outlet area ratio

The ratio between the inlet and outlet diffuser areas. For a standard pump with a 5° – 7° included-angle diffuser, the ratio is close to 0.2. The parameter must be greater or equal to zero. The default value is 0.224.

### Nozzle loss coefficient

The hydraulic friction loss coefficient in the nozzle. The parameter must be greater than zero. The default value is 0.05.

## Throat entry loss coefficient

The hydraulic friction loss coefficient in the throat entry. The parameter must be greater than zero. The default value is 0.005.

## Throat loss coefficient

The hydraulic friction loss coefficient in the throat. The parameter must be greater than zero. The default value is 0.1.

## Diffuser loss coefficient

The hydraulic friction loss coefficient in the diffuser. The parameter must be greater than zero. The default value is 0.1.

## Global Parameters

Parameters determined by the type of working fluid:

- Fluid density

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the nozzle entry (primary flow entry).

S

Hydraulic conserving port associated with the pump suction (secondary flow entry).

P

Hydraulic conserving port associated with the pump outlet.

N

Internal nonvisible hydraulic conserving port associated with the throat entry section of the pump. You can view the variables associated with the port by logging simulation data. For more information, see “How to Log Simulation Data”.

**Examples**

The Well Jet Pump example represents a well jet pump installation, consisting of a surface-mounted centrifugal pump and a jet pump installed in the well below water level.

**References**

[1] I.J. Karassic, J.P. Messina, P. Cooper, C.C. Heald, *Pump Handbook*, Fourth edition, McGraw-Hill, NY, 2008

**See Also**

Centrifugal Pump

Fixed-Displacement Pump

Variable-Displacement Pressure-Compensated Pump

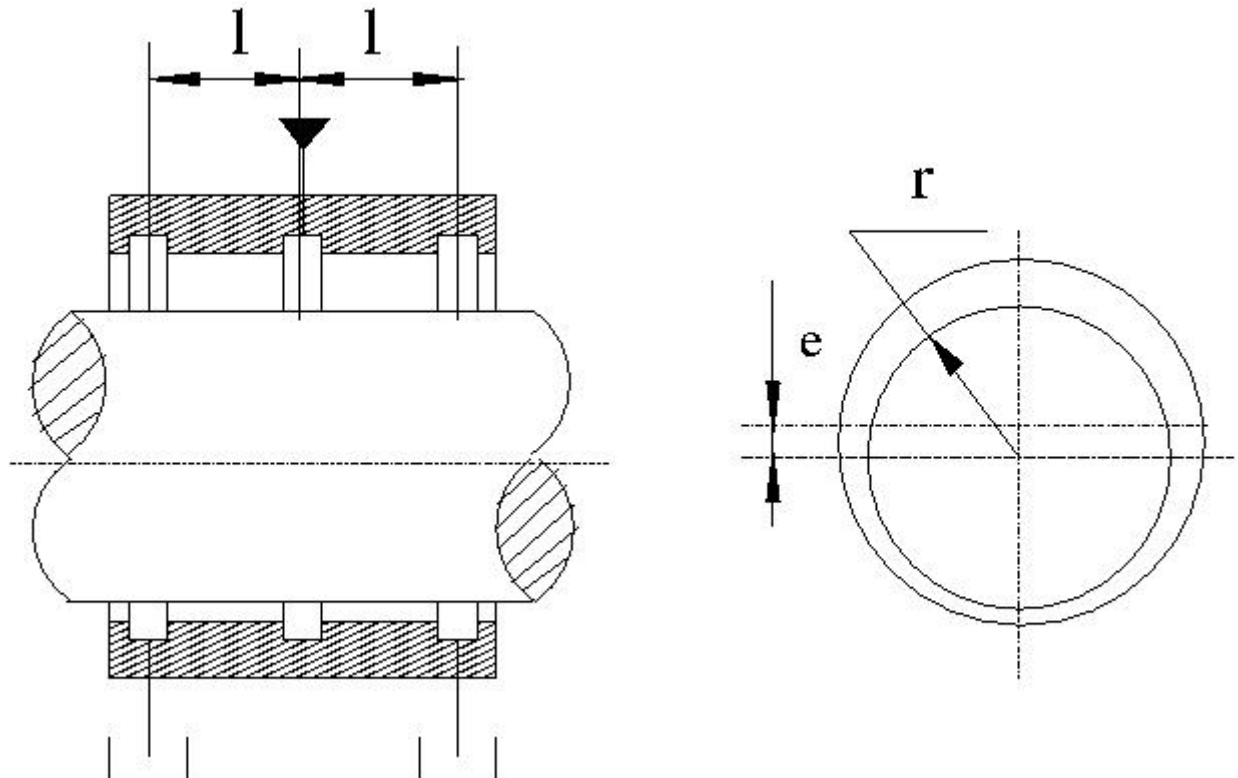
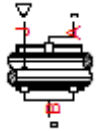
Variable-Displacement Pump

# Journal Bearing Pressure-Fed

**Purpose** Hydraulic portion of pressure-fed journal bearing

**Library** Orifices

**Description** The Journal Bearing Pressure-Fed block simulates the hydraulic portion of a pressure-fed journal bearing, shown in the following illustration.



The lubricant under pressure  $p$  is pumped into the circumferential groove at the center of the bearing. The groove divides the bearing into two half-bearings. The lubricant exits through the end grooves located at a distance  $l$  from the central groove. The model is intended to be used in lubrication system simulation to assess the flow consumption through the pressure-fed journal bearing. The flow regime is assumed to be laminar due to very small clearances between the journal and the bushing.

The flow rate is computed using the Hagen-Poiseuille equation (see [1]):

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

where

$q$	Volumetric flow rate
$p$	Pressure differential across the bearing
$r$	Journal radius
$c$	Radial clearance at neutral position
$\mu$	Dynamic viscosity
$l$	Length of the half-bearing
$\varepsilon$	Relative eccentricity, $\varepsilon = e / r$
$e$	Eccentricity or journal deflection from the central position

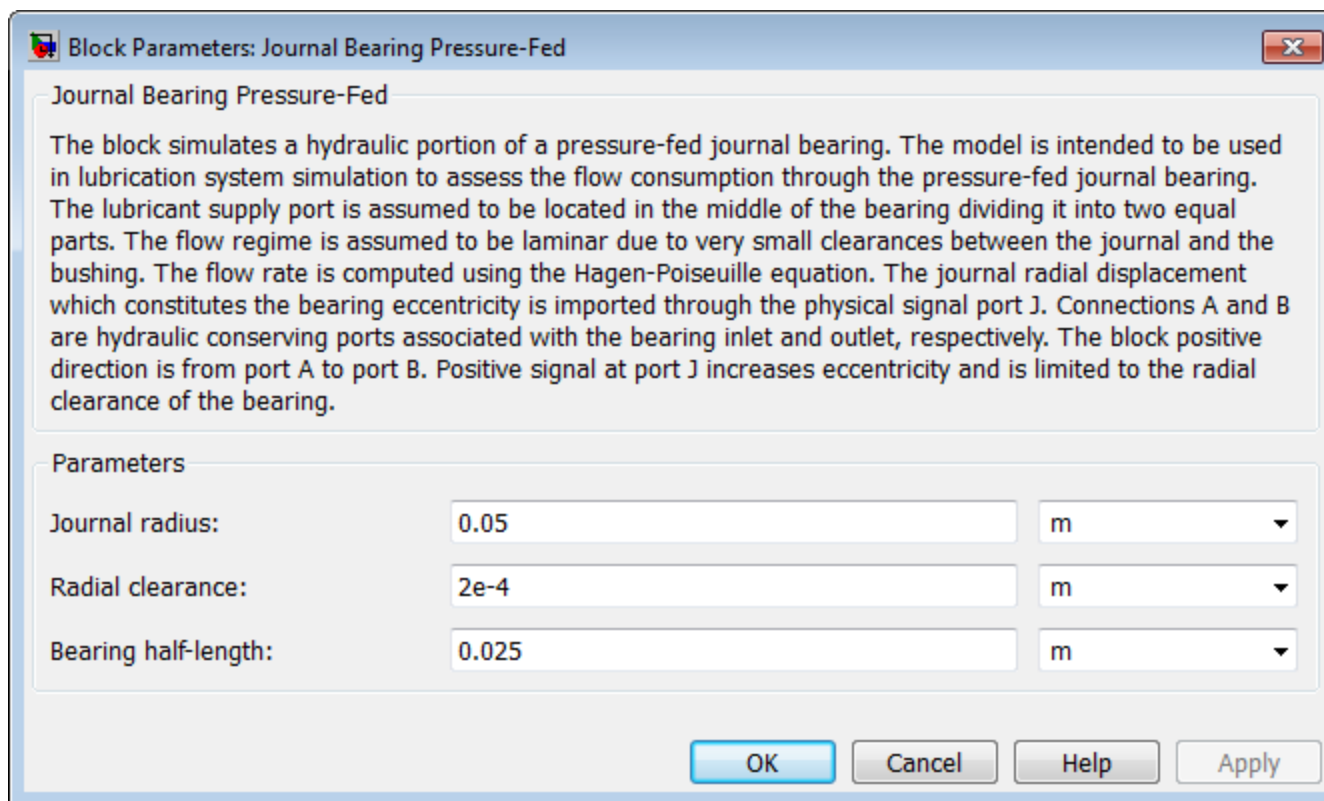
The journal radial displacement, which controls the bearing eccentricity, is imported through the physical signal port J. Connections A and B are hydraulic conserving ports associated with the bearing inlet and outlet, respectively. The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as  $p = p_A - p_B$ . Positive signal at the physical signal port J increases the eccentricity and is limited to the radial clearance of the bearing.

# Journal Bearing Pressure-Fed

## Basic Assumptions and Limitations

The flow regime is assumed to be laminar due to small clearances.

## Dialog Box and Parameters





## Journal radius

The radius of the journal. The parameter must be positive. The default value is 0.05 m.

## Radial clearance

The radial clearance between the journal and the bushing at neutral position. The parameter must be positive. The default value is  $2e-4$  m.

## Bearing half-length

The length of the half-bearing, that is, the distance between each of the end grooves and the central groove. The parameter must be positive. The default value is 0.025 m.

## Global Parameters

Parameters determined by the type of working fluid:

- Fluid density
- Fluid kinematic viscosity

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

## Ports

The block has the following ports:

- A Hydraulic conserving port associated with the bearing inlet.
- B Hydraulic conserving port associated with the bearing outlet.
- J Physical signal port that controls the journal deflection.

## References

[1] Shigley, J., C. Mischke, and R. Budynas. *Mechanical Engineering Design*. New York: McGraw-Hill, 2004.

# Local Resistance

---

**Purpose** Hydraulic resistance specified by loss coefficient

**Library** Local Hydraulic Resistances

**Description**



The Local Resistance block represents a generic local hydraulic resistance, such as a bend, elbow, fitting, filter, local change in the flow cross section, and so on. The pressure loss caused by resistance is computed based on the pressure loss coefficient, which is usually provided in catalogs, data sheets, or hydraulic textbooks. The pressure loss coefficient can be specified either as a constant, or by a table, in which it is tabulated versus Reynolds number.

The pressure loss is determined according to the following equations:

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

$$p = p_A - p_B$$

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

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

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

where

$q$  Flow rate

$p$  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
$\rho$	Fluid density
$\nu$	Fluid kinematic viscosity

Two block parameterization options are available:

- By semi-empirical formulas — The pressure loss coefficient is assumed to be constant for a specific flow direction. The flow regime can be either laminar or turbulent, depending on the Reynolds number.
- By table-specified  $K=f(Re)$  relationship — The pressure loss coefficient is specified as function of the Reynolds number. The flow regime is assumed to be turbulent all the time. It is your responsibility to provide the appropriate values in the  $K=f(Re)$  table to ensure turbulent flow.

The resistance can be symmetrical or asymmetrical. In symmetrical resistances, the pressure loss practically does not depend on flow direction and one value of the coefficient is used for both the direct and reverse flow. For asymmetrical resistances, the separate coefficients are provided for each flow direction. If the loss coefficient is specified by a table, the table must cover both the positive and the negative flow regions.

# Local Resistance

---

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

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

## Basic Assumptions and Limitations

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

## Dialog Box and Parameters

**Block Parameters: Local Resistance**

Local Resistance

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

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

Parameters

Resistance area:

Model parameterization:

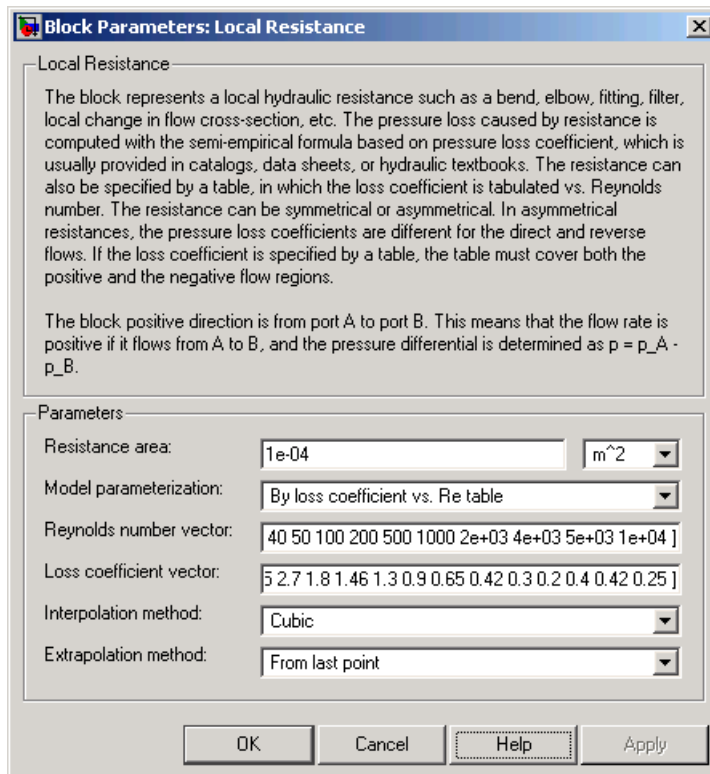
Pressure loss coefficient for direct flow:

Pressure loss coefficient for reverse flow:

Critical Reynolds number:

OK Cancel Help Apply

# Local Resistance



## Resistance area

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

## Model parameterization

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

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

- **By loss coefficient vs. Re table** — Provide tabulated data of loss coefficients and corresponding Reynolds numbers. The loss coefficient is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods. For asymmetrical resistances, the table must cover both the positive and the negative flow regions.

### **Pressure loss coefficient for direct flow**

Loss coefficient for the direct flow (flowing from A to B). For simple ideal configurations, the value of the coefficient can be determined analytically, but in most cases its value is determined empirically and provided in textbooks and data sheets (for example, see [1]). The default value is 2. This parameter is used if **Model parameterization** is set to **By semi-empirical formulas**.

### **Pressure loss coefficient for reverse flow**

Loss coefficient for the reverse flow (flowing from B to A). The parameter is similar to the loss coefficient for the direct flow and must be set to the same value if the resistance is symmetrical. The default value is 2. This parameter is used if **Model parameterization** is set to **By semi-empirical formulas**.

### **Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 150. This parameter is used if **Model parameterization** is set to **By semi-empirical formulas**.

### **Reynolds number vector**

Specify the vector of input values for Reynolds numbers as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default

# Local Resistance

---

values are [-4000, -3000, -2000, -1000, -500, -200, -100, -50, -40, -30, -20, -15, -10, 10, 20, 30, 40, 50, 100, 200, 500, 1000, 2000, 4000, 5000, 10000]. This parameter is used if **Model parameterization** is set to By loss coefficient vs. Re table.

## Loss coefficient vector

Specify the vector of the loss coefficient values as a one-dimensional array. The loss coefficient vector must be of the same size as the Reynolds numbers vector. The default values are [0.25, 0.3, 0.65, 0.9, 0.65, 0.75, 0.90, 1.15, 1.35, 1.65, 2.3, 2.8, 3.10, 5, 2.7, 1.8, 1.46, 1.3, 0.9, 0.65, 0.42, 0.3, 0.20, 0.40, 0.42, 0.25]. This parameter is used if **Model parameterization** is set to By loss coefficient vs. Re table.

## Interpolation method

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

- **Linear** — Uses a linear interpolation function.
- **Cubic** — Uses the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP).
- **Spline** — Uses the cubic spline interpolation algorithm.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to By loss coefficient vs. Re table.

## Extrapolation method

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

- **From last 2 points** — Extrapolates using the linear method (regardless of the interpolation method specified), based on



the last two output values at the appropriate end of the range. That is, the block uses the first and second specified output values if the input value is below the specified range, and the two last specified output values if the input value is above the specified range.

- **From last point** — Uses the last specified output value at the appropriate end of the range. That is, the block uses the last specified output value for all input values greater than the last specified input argument, and the first specified output value for all input values less than the first specified input argument.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to **By loss coefficient vs. Re table**.

## Restricted Parameters

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

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

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

## Global Parameters

Parameters determined by the type of working fluid:

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

# Local Resistance

---

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

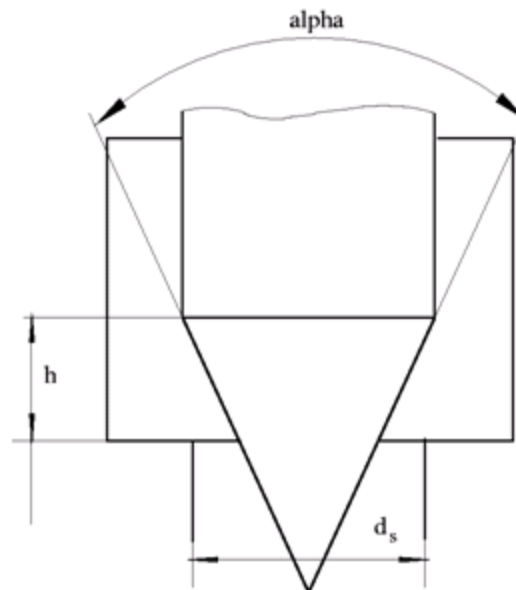
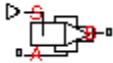
Sudden Area Change

T-junction

**Purpose** Hydraulic needle valve

**Library** Flow Control Valves

**Description** The Needle Valve block models a variable orifice created by a conical needle and a round sharp-edged orifice in thin material.



The flow rate through the valve is proportional to the valve opening and to the pressure differential across the valve. The model accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number ( $Re$ ) and comparing its value with the critical Reynolds number ( $Re_{cr}$ ). The flow rate is determined according to the following equations:

# Needle Valve

---

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

$$h = x_0 + x$$

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

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

$$p = p_A - p_B$$

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

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

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

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

where

$q$	Flow rate
$p$	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
$\alpha$	Needle angle
$\rho$	Fluid density
$D_H$	Valve instantaneous hydraulic diameter
$\nu$	Fluid kinematic viscosity
$A_{leak}$	Closed valve leakage area
$A_{max}$	Maximum valve open area

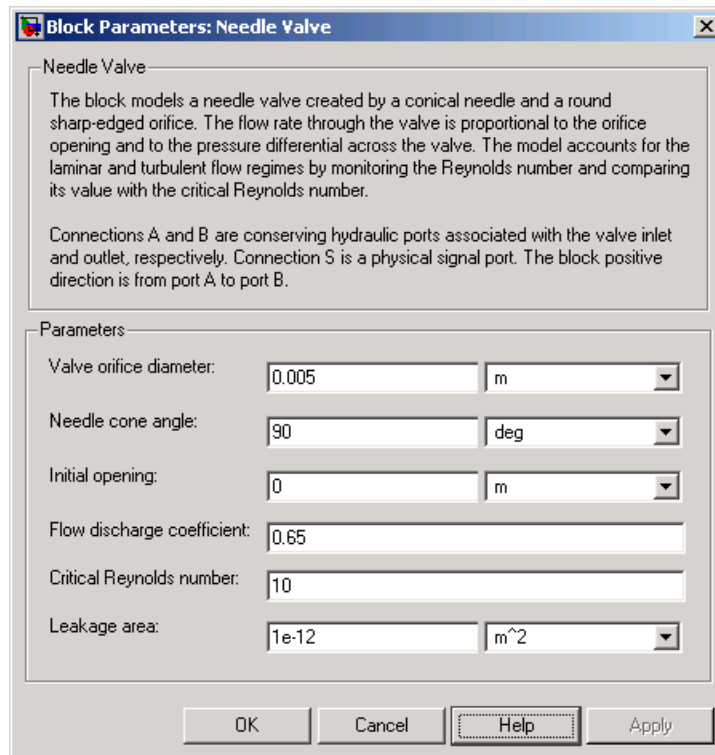
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as  $p = p_A - p_B$ . Positive signal at the physical signal port S opens the valve.

## Basic Assumptions and Limitations

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

# Needle Valve

## Dialog Box and Parameters



### Valve orifice diameter

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

### Needle cone angle

The angle of the valve conical needle. The parameter value must be in the range between 0 and 180 degrees. The default value is 90 degrees.

### Initial opening

The initial opening of the valve. You can specify both positive and negative values. The default value is 0.

## Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is 0.65.

## Critical Reynolds number

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 10.

## Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## Global Parameters

Parameters determined by the type of working fluid:

- Fluid density
- Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

# Needle Valve

---

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

## **See Also**

- Ball Valve
- Ball Valve with Conical Seat
- Gate Valve
- Poppet Valve
- Pressure-Compensated Flow Control Valve



# Orifice with Variable Area Round Holes

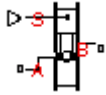
## Purpose

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

## Library

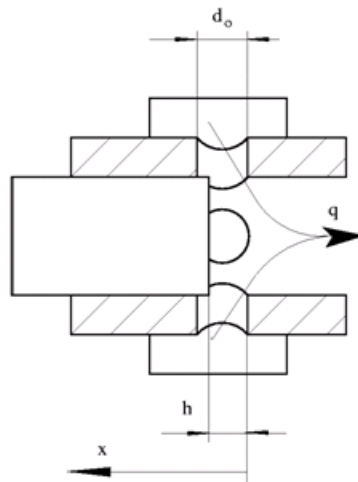
Orifices

## 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
- $d_o$  Orifice hole diameter



# Orifice with Variable Area Round Holes

The model accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number ( $Re$ ) and comparing its value with the critical Reynolds number ( $Re_{cr}$ ). The flow rate is determined according to the following equations:

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

$$h = x_0 + x_{or}$$

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

$$p = p_A - p_B$$

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

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

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

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

# Orifice with Variable Area Round Holes

---

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
$d_o$	Hole diameter
$z$	Number of holes
$x_o$	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.
$\rho$	Fluid density
$D_H$	Instantaneous orifice hydraulic diameter
$\nu$	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**.

# Orifice with Variable Area Round Holes

## Basic Assumptions and Limitations

- 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 Round Holes**

Orifice with Variable Area Round Holes

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

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

Parameters

Diameter of round holes:  m

Number of round holes:

Flow discharge coefficient:

Initial opening:  m

Orifice orientation:

Critical Reynolds number:

Leakage area:  m<sup>2</sup>

OK Cancel Help Apply

### 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.

# Orifice with Variable Area Round Holes

---

## **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 simulation to fail. Therefore, MathWorks

# Orifice with Variable Area Round Holes

---

recommends that you do not set this parameter to 0. The default value is  $1e-15 \text{ m}^2$ .

## Restricted Parameters

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

- **Orifice orientation**

All other block parameters are available for modification.

## Global Parameters

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

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

The flow rate is positive if fluid flows from port A to port B. Positive signal at the physical signal port S opens or closes the orifice depending on the value of the parameter **Orifice orientation**.

## See Also

Annular Orifice  
Constant Area Hydraulic Orifice

# Orifice with Variable Area Round Holes

---

Fixed Orifice

Orifice with Variable Area Slot

Variable Area Hydraulic Orifice

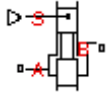
Variable Orifice

# Orifice with Variable Area Slot

**Purpose** Hydraulic variable orifice shaped as rectangular slot

**Library** Orifices

## Description



The block models a variable orifice created by a cylindrical sharp-edged spool and a rectangular slot in a sleeve. The flow rate through the orifice is proportional to the orifice opening and to the pressure differential across the orifice. The model accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number ( $Re$ ) and comparing its value with the critical Reynolds number ( $Re_{cr}$ ). The flow rate is determined according to the following equations:

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

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

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

$$p = p_A - p_B$$

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

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

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



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.
$\rho$	Fluid density
$D_H$	Instantaneous orifice hydraulic diameter
$\nu$	Fluid kinematic viscosity
$A_{leak}$	Closed orifice leakage area

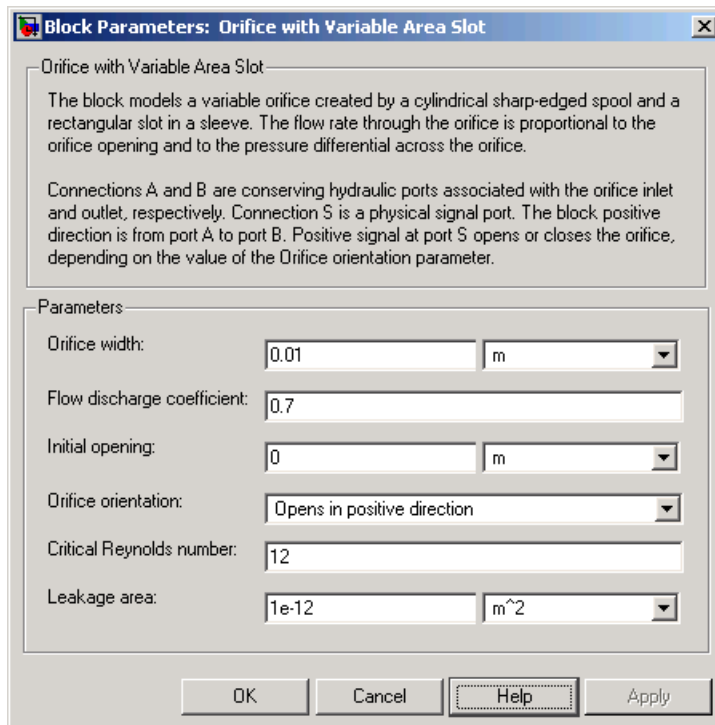
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as  $p = p_A - p_B$ . Positive signal at the physical signal port S opens or closes the orifice depending on the value of the parameter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

## Basic Assumptions and Limitations

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

# Orifice with Variable Area Slot

## Dialog Box and Parameters



### Orifice width

The width of the rectangular slot. The default value is  $1 \text{e-}2 \text{ m}$ .

### 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 simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## Restricted Parameters

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

- **Orifice orientation**

All other block parameters are available for modification.

# Orifice with Variable Area Slot

---

## Global Parameters

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the orifice inlet.

B

Hydraulic conserving port associated with the orifice outlet.

S

Physical signal port to control spool displacement.

The flow rate is positive if fluid flows from port A to port B. Positive signal at the physical signal port S opens or closes the orifice depending on the value of the parameter **Orifice orientation**.

## See Also

Annular Orifice

Constant Area Hydraulic Orifice

Fixed Orifice

Orifice with Variable Area Round Holes

Variable Area Hydraulic Orifice

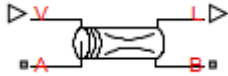
Variable Orifice

# Partially Filled Vertical Pipe LP

**Purpose** Partially filled vertical pipe connecting two tanks

**Library** Low-Pressure Blocks

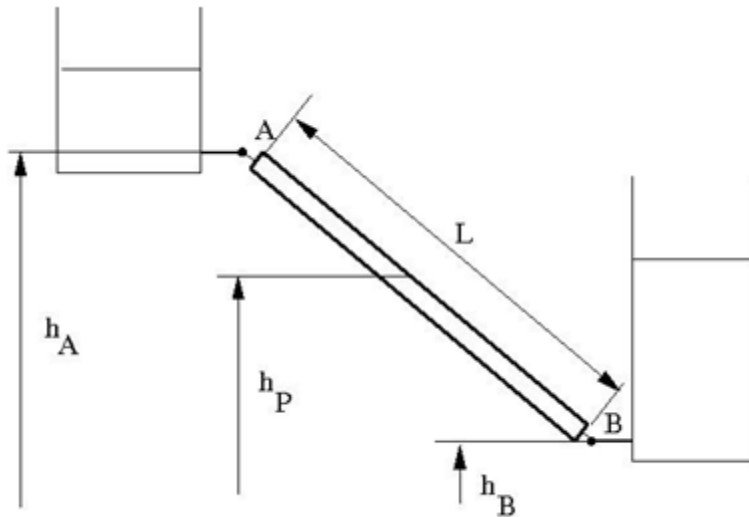
## Description



The Partially Filled Vertical Pipe LP block models a vertical pipe connecting two tanks. The block takes into account the possibility that the top tank gets emptied, and therefore the top port of the pipe can be exposed. The fluid level in the pipe then settles at some intermediate position, depending on pressures at both ports.

The following schematic diagram shows two tanks connected by a partially filled vertical pipe, where:

- $h_A$  and  $h_B$  are the elevations of ports A and B, respectively
- $h_p$  is the fluid level in the pipe
- $L$  is the length of the pipe



# Partially Filled Vertical Pipe LP

---

The tank is assumed to be empty when fluid volume becomes less than a specified value. If the upstream tank is emptied, the flow rate through port A is assumed to be 0, and the tank pressurization pressure propagates through port A to the pipe. When pressure increases at the bottom port B, the fluid flows upstream. After the pipe is completely filled, the fluid can start filling the top tank.

You must connect port A to a component that provides data on fluid volume (such as the Reservoir block or any of the tank blocks). Port A elevation must be greater than port B elevation.

You can connect port B to any hydraulic component, with one restriction: the component must keep the port submerged at all times.

The pipe hydraulic resistance is proportional to the length of fluid in the pipe (wet length). The shorter the wet length, the lower the resistance. To avoid zero resistance as wet length approaches 0, the volume of fluid in the pipe is limited to the same minimum value as the tank. After the minimum volume is reached, the flow rate through port B is assumed to be 0.

It is a good practice to connect port B either to another tank or to a pressure source. When you connect it to a pressure source, it is good practice to insert some hydraulic resistance (fixed orifice, valve, and so on) between the pipe and the source, to avoid zero resistance.

The block models pipes with circular and noncircular cross sections. The block accounts for friction losses, variable fluid level in the pipe, and resulting variable heads at the ports. The block is based on the assumption of steady-state fluid momentum conditions. For additional information, see “Basic Assumptions and Limitations” on page 1-428. To account for local resistances (such as bends, fittings, inlet and outlet losses, and so on), convert the resistances into their equivalent lengths, sum up all the resistances to obtain their aggregate length, and then add this length to the pipe geometrical length. The ratio between the added and the current wet length of the pipe is maintained constant as the pipe is emptied or filled with fluid.

Flow rates through the pipe ports are computed individually because they are different when the top port is exposed. The flow rates

are proportional to the flow-regime-dependent friction factor and the pressure differential. The friction factor in turbulent regime is determined with the Haaland approximation [1]:

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

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

where

$f$	Friction factor
$k$	Height of the roughness on the pipe internal surface
$Re$	Reynolds number
$q$	Flow rate
$D_H$	Pipe hydraulic diameter
$A$	Pipe cross-sectional area
$\nu$	Fluid kinematic viscosity

At laminar regime, the friction factor is determined as

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

where  $s$  is the geometrical shape factor, or Poiseuille number [1], defined as a function of the Fanning factor for various noncircular shapes: rectangle, concentric annulus, ellipse, and circular sector.

The friction factor during the transition from laminar to turbulent regimes is determined with the linear interpolation between extreme

# Partially Filled Vertical Pipe LP

points of the regimes. As a result of these assumptions, the pipe is simulated according to the following equations:

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

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

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

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



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

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

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

where

$q_A, q_B$  Flow rates through port A and B, respectively

$p_A, p_B$  Pressures at port A and B, respectively

$h_A, h_B$  Port A and port B elevations, respectively

$L$  Geometrical pipe length

$L_{ad}$  Aggregate equivalent length of local resistances

$L_F$  Pipe wet length

$L_{ef}$  Effective pipe length

$\rho$  Fluid density

$p$  Pressure differential across the pipe

$V$  Fluid volume in the upstream tank

$V_p$  Volume of fluid in the pipe

$V_{\min}$  Minimum volume of fluid in the tank, or in the pipe. If  $V$  becomes less than  $V_{\min}$ , then the upstream tank is considered empty, the flow rate through port A is assumed to be 0, and the tank pressurization pressure propagates through port A to the pipe. If  $V_p$  becomes less than  $V_{\min}$ , then the pipe is considered empty and the flow rate through port B is assumed to be 0.

$h_p$  Fluid elevation in the pipe

# Partially Filled Vertical Pipe LP

---

$Re_{cr}$  Critical Reynolds number

$g$  Gravity acceleration

Flow rate A is considered positive when fluid flows into the pipe. Flow rate B is considered positive when fluid flows out of the pipe.

## Basic Assumptions and Limitations

- The block is based on the assumption of steady-state fluid momentum conditions.
- Fluid inertia and fluid compressibility are not taken into account.
- Flow is assumed to be fully developed along the pipe length, therefore the end effects are not considered.
- To account for local resistances in the pipe, you must convert the resistances into their equivalent lengths, sum them up to obtain their aggregate length, and then add this aggregate length to the pipe geometrical length.
- The elevation of port A must be greater than that of port B.
- Port A must be connected to the hydraulic port of the top tank. The fluid volume in the top tank must be input into the Partially Filled Vertical Pipe LP block through the physical signal port V. If you use one of the standard SimHydraulics reservoir or tank blocks, connect its physical output port directly to the input port V. If you use a custom tank block, it must have a port that exports fluid volume in the tank as a physical signal, and you must connect this output port to the input port V.
- Port B must not be exposed.
- The friction factor at turbulent regime is computed with the Haaland approximation.
- The friction factor at laminar regime is determined as the Poiseuille number divided by the Reynolds number.
- The friction factor during transition from laminar to turbulent regime is computed with the linear interpolation.

## Dialog Box and Parameters

The block dialog box has two tabs:

- “Basic Parameters” on page 1-429
- “Vertical Position” on page 1-433

### Basic Parameters

**Block Parameters: Partially Filled Vertical Pipe LP**

Partially Filled Vertical Pipe LP

This block models a hydraulic pipe which is connected to the installed at high elevation tank and accounts for the fluid volume in the tank and in the pipe. The fluid volume in the tank, connected to port A, is imported via physical signal port V. If tank gets emptied, the pressure at port A becomes constant and equal to the tank pressurization. Port B must not be exposed and must be submerged all the time. The block accounts for the friction loss and loss at local resistances such as bends, fittings, filters, etc. The resistances must be converted into their equivalent lengths, and then the total length of all the resistances has to be added to the pipe geometrical length. The ratio between the geometrical and added length is maintained constant while pipe is filled or emptied. The fluid inertia and fluid compressibility are assumed to be negligible. Connections A and B are hydraulic conserving ports. Connection V is the physical signal port through which the volume in the upstream tank is imported. Connection L is the physical signal port which conveys the fluid level in the pipe. 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

Basic parameters    Vertical position

Pipe type: Circular

Pipe internal diameter: 0.1 m

Geometrical shape factor: 64

Pipe length: 100 m

Aggregate equivalent length of local resistances: 50 m

Tank minimum volume: 1e-4 m<sup>3</sup>

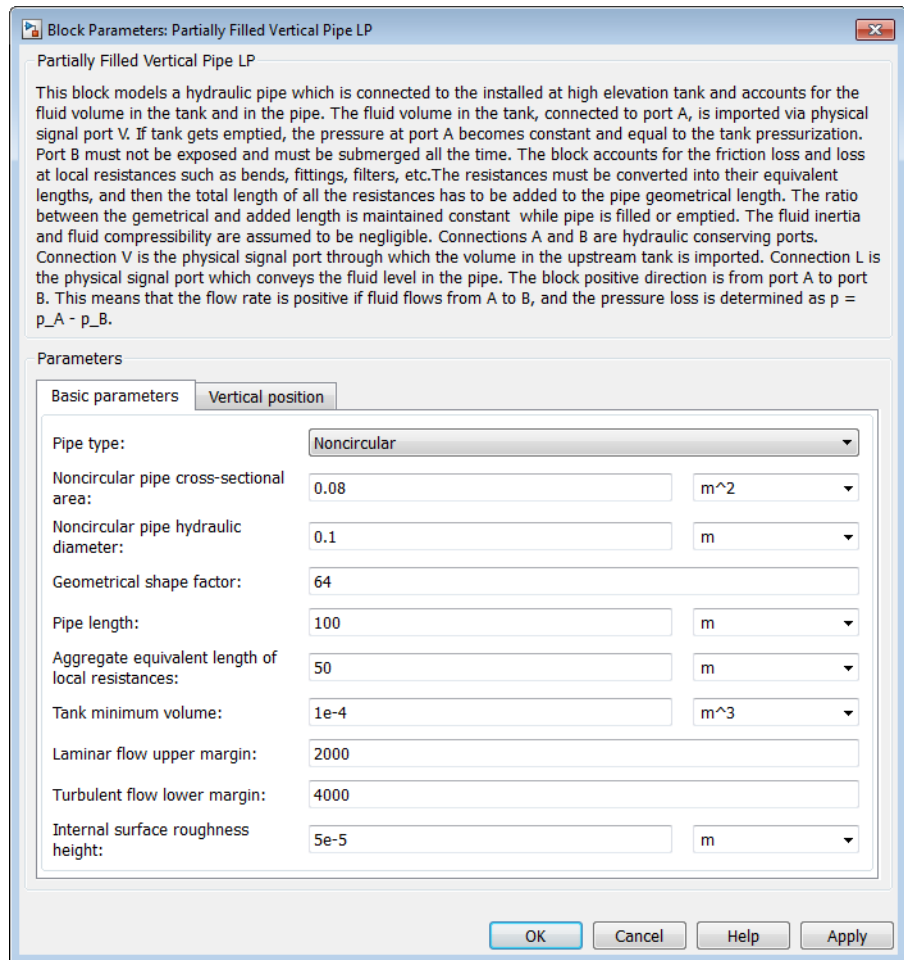
Laminar flow upper margin: 2000

Turbulent flow lower margin: 4000

Internal surface roughness height: 5e-5 m

OK    Cancel    Help    Apply

# Partially Filled Vertical Pipe LP



## Pipe type

The type of pipe cross section: **Circular** or **Noncircular**. For a circular pipe, you specify its internal diameter. For a noncircular pipe, you specify its hydraulic diameter and pipe cross-sectional area. The default value of the parameter is **Circular**.

**Pipe internal diameter**

Pipe internal diameter. The parameter is available if **Pipe type** is set to **Circular**. The default value is 0.01 m.

**Noncircular pipe cross-sectional area**

Pipe cross-sectional area. The parameter is available if **Pipe type** is set to **Noncircular**. The default value is 0.08 m<sup>2</sup>.

**Noncircular pipe hydraulic diameter**

Hydraulic diameter of the pipe cross section. The parameter is available if **Pipe type** is set to **Noncircular**. The default value is 0.1 m.

**Geometrical shape factor**

Used for computing friction factor at laminar flow. The shape of the pipe cross section determines the value. For a pipe with a noncircular cross section, set the factor to an appropriate value, for example, 56 for a square, 96 for concentric annulus, 62 for rectangle (2:1), and so on [1]. The default value is 64, which corresponds to a pipe with a circular cross section.

**Pipe length**

Pipe geometrical length. The default value is 100 m.

**Aggregate equivalent length of local resistances**

Represents total equivalent length of all local resistances associated with the pipe. You can account for the pressure loss caused by local resistances, such as bends, fittings, armature, inlet/outlet losses, and so on, by adding to the pipe geometrical length an aggregate equivalent length of all the local resistances. The default value is 50 m.

**Tank minimum volume**

Minimum volume of fluid in the upstream tank, or in the pipe, corresponding to  $V_{\min}$  in the block equations. The tank or pipe is considered empty if its fluid volume becomes less than  $V_{\min}$ . The default value is  $1e-4$  m<sup>3</sup>.

# Partially Filled Vertical Pipe LP

---

## **Laminar flow upper margin**

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

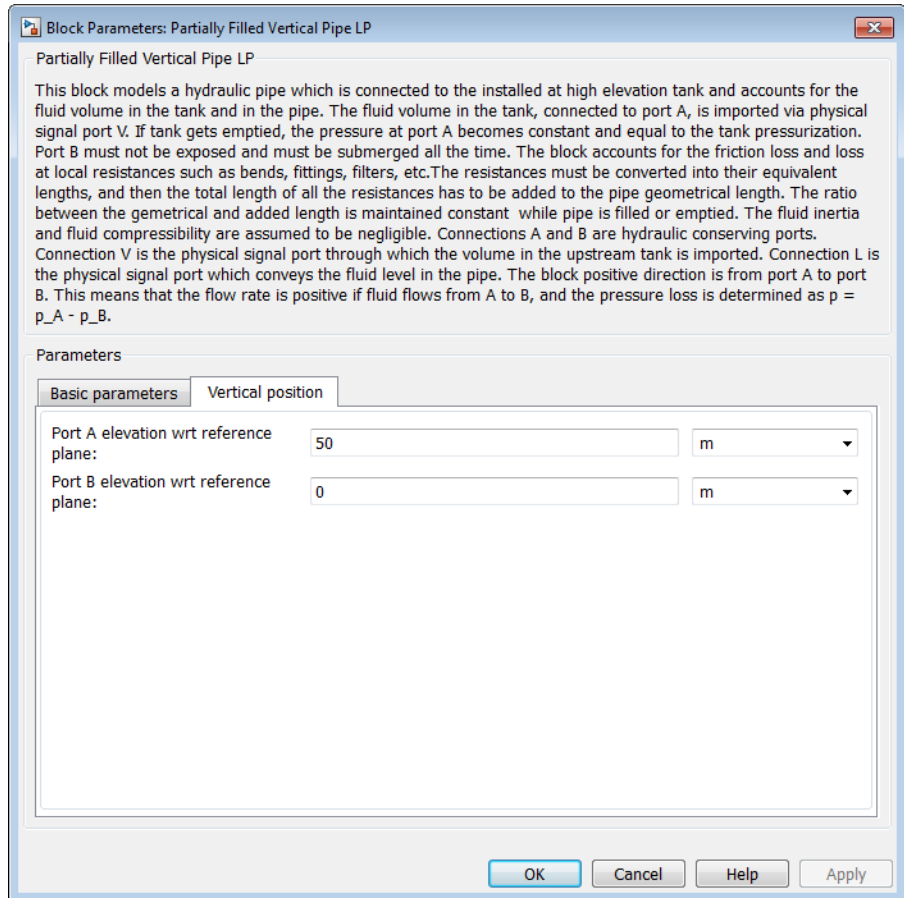
## **Turbulent flow lower margin**

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

## **Internal surface roughness height**

Roughness height on the pipe internal surface. The parameter is typically provided in data sheets or manufacturer catalogs. The default value is  $5e-5$  m.

## Vertical Position



### Port A elevation wrt reference plane

The vertical position of the pipe port A with respect to the reference plane. The default value is 50 m.

# Partially Filled Vertical Pipe LP

---

## Port B elevation wrt reference plane

The vertical position of the pipe port B with respect to the reference plane. The default value is 0.

## Restricted Parameters

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

- **Pipe type**

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

## Global Parameters

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

- A  
Hydraulic conserving port associated with the pipe inlet.
- B  
Hydraulic conserving port associated with the pipe outlet.
- V  
Physical signal input port that provides data on fluid volume in the upstream tank.
- L  
Physical signal output port that exports the pipe fluid level.



## References

[1] White, F.M., *Viscous Fluid Flow*, McGraw-Hill, 1991

## See Also

Hydraulic Pipe LP with Variable Elevation

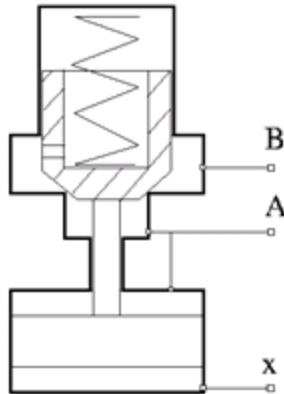
Resistive Pipe LP with Variable Elevation

# Pilot-Operated Check Valve

**Purpose** 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

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

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

$$p_e = p_A + p_X \kappa_p - p_B$$

where  $\kappa_p = A_X/A_A$  is usually referred to as pilot ratio and  $p_e$  is the equivalent pressure differential across the poppet. The valve remains closed while this pressure differential across the valve is lower than the valve cracking pressure. When cracking pressure is reached, the valve control member (spool, ball, poppet, etc.) is forced off its seat, thus creating a passage between the inlet and outlet. If the flow rate is high enough and pressure continues to rise, the area is further increased until the control member reaches its maximum. At this moment, the valve passage area is at its maximum. The valve maximum area and the cracking and maximum pressures are generally provided in the catalogs and are the three key parameters of the block.

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

The model accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number ( $Re$ ) and comparing its value with the critical Reynolds number ( $Re_{cr}$ ). The flow rate is determined according to the following equations:

# Pilot-Operated Check Valve

---

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

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

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

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

$$p = p_A - p_B$$

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

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

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

where

- $q$  Flow rate through the valve
- $p$  Pressure differential across the valve
- $p_e$  Equivalent pressure differential across the control member

$p_A, p_B$	Gauge pressures at the valve terminals
$p_X$	Gauge pressure at the pilot terminal
$k_p$	Pilot ratio, $k_p = p_X/p_A$
$k$	Valve gain coefficient
$C_D$	Flow discharge coefficient
$A(\rho)$	Instantaneous orifice passage area
$A_{max}$	Fully open valve passage area
$A_{leak}$	Closed valve leakage area
$p_{crack}$	Valve cracking pressure
$p_{max}$	Pressure needed to fully open the valve
$D_H$	Instantaneous orifice hydraulic diameter
$\rho$	Fluid density
$\nu$	Fluid kinematic viscosity

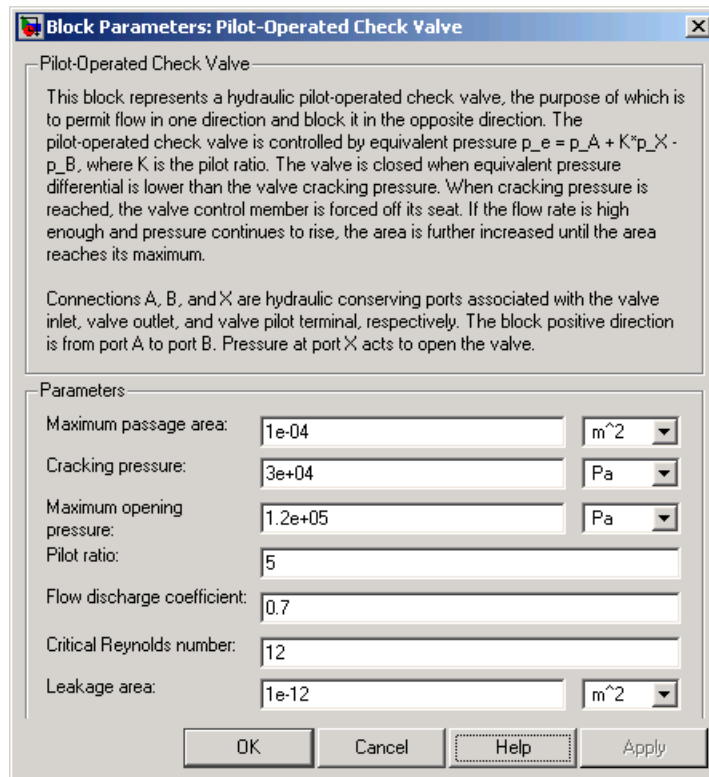
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as  $p = p_A - p_B$ .

## Basic Assumptions and Limitations

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

# Pilot-Operated Check Valve

## Dialog Box and Parameters



### Maximum passage area

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

### Cracking pressure

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

### Maximum opening pressure

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

## **Pilot ratio**

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

## **Flow discharge coefficient**

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

## **Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

## **Leakage area**

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## **Global Parameters**

Parameters determined by the type of working fluid:

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

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

## **Ports**

The block has the following ports:

# Pilot-Operated Check Valve

---

- A Hydraulic conserving port associated with the valve inlet.
- B Hydraulic conserving port associated with the valve outlet.
- X Hydraulic conserving port associated with the valve pilot terminal.

**See Also** Check Valve



**Purpose** Hydraulic resistance in pipe bend

**Library** Local Hydraulic Resistances

**Description**



The Pipe Bend block represents a pipe bend as a local hydraulic resistance. The pressure loss in the bend is assumed to consist of

- Loss in the straight pipe
- Loss due to curvature

The loss in a straight pipe is simulated with the Hydraulic Resistive Tube block. The loss due to curvature is simulated with the Local Resistance block, and the pressure loss coefficient is determined in accordance with the Crane Co. recommendations (see [1], 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
- $\rho$  Fluid density

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

# Pipe Bend

The pressure loss coefficient is determined according to recommendation provided in [1]:

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

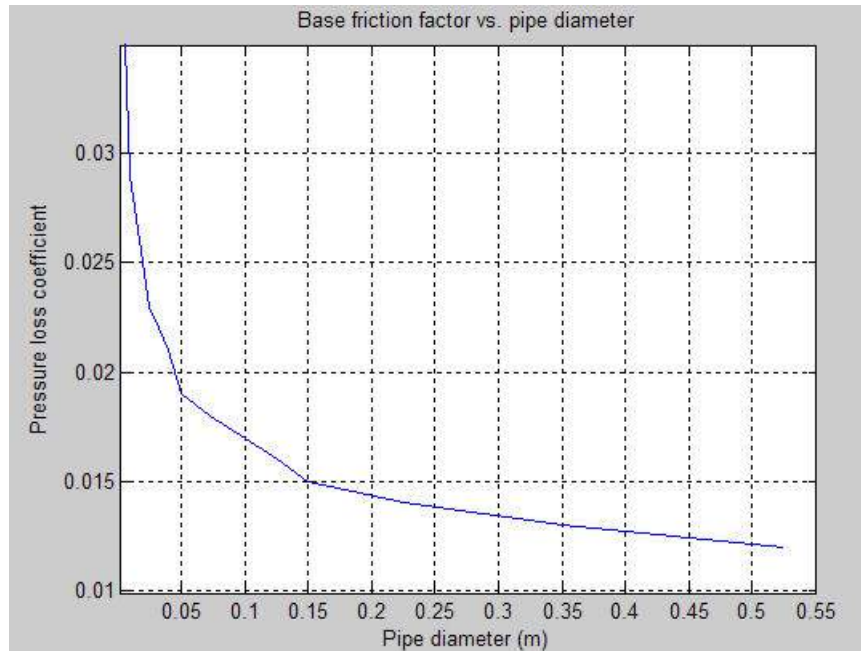
where

$K_d$  Base friction factor coefficient

$K_r$  Correction coefficient accounting for the bend curvature

$K_\alpha$  Correction coefficient accounting for the bend angle

The base friction factor coefficient is determined according to the following table.

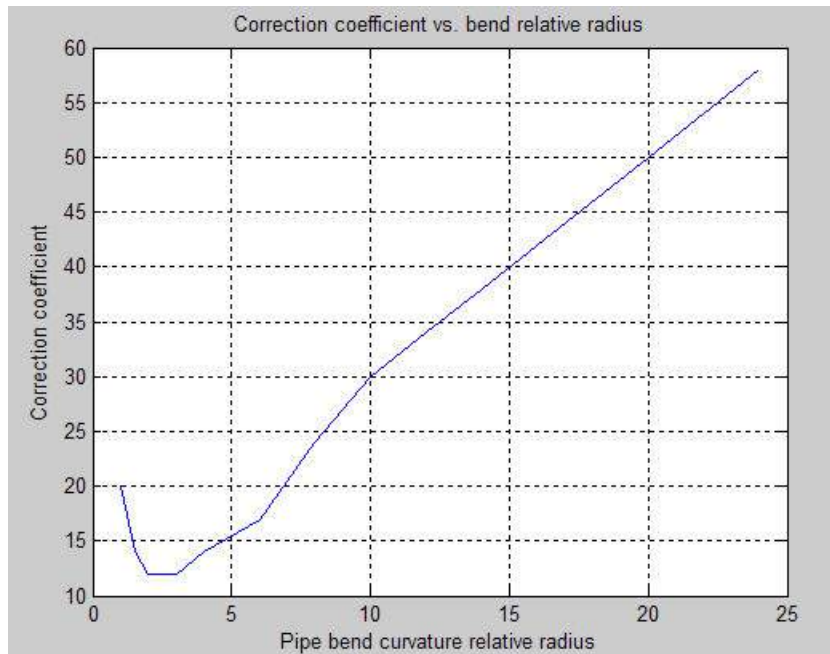


---

**Note** Friction factors for pipes with diameters greater than 525 mm are determined by extrapolation.

---

The correction coefficient accounting for the bend curvature is determined according to the next table.



The bend curvature relative radius is calculated as

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

# Pipe Bend

---

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

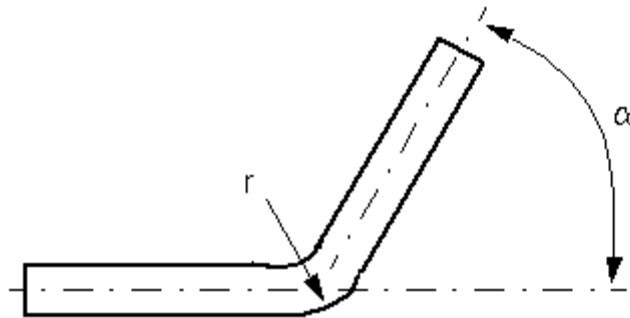
---

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

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

where

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



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

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

## Warning

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

## Basic Assumptions and Limitations

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

# Pipe Bend

## Dialog Box and Parameters

**Block Parameters: Pipe Bend**

Pipe Bend

The block represents a local hydraulic resistance, such as a pipe bend. The pressure loss in the bend is assumed to consist of (1) the loss in the straight pipe and (2) the loss due to curvature. The loss in a straight pipe is simulated with the Resistive Tube block. The loss due to curvature is simulated with the Local Resistance block, and the pressure loss coefficient is determined in accordance with the Crane Co. recommendations.

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

Parameters

Pipe diameter: 0.01 m

Bend radius: 0.04 m

Bend angle: 90 deg

Internal surface roughness height: 1.5e-05 m

Critical Reynolds number: 350

OK Cancel Help Apply

### Pipe diameter

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

### Bend radius

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

### Bend angle

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

## Internal surface roughness height

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

## Critical Reynolds number

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 350.

## Global Parameters

Parameters determined by the type of working fluid:

- Fluid density
- Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the bend inlet.

B

Hydraulic conserving port associated with the bend outlet.

## References

[1] *Flow of Fluids Through Valves, Fittings, and Pipe*, Crane Valves North America, Technical Paper No. 410M

[2] George R. Keller, *Hydraulic System Analysis*, Published by the Editors of Hydraulics & Pneumatics Magazine, 1970

## See Also

Elbow

Gradual Area Change

# Pipe Bend

---

Local Resistance

Hydraulic Resistive Tube

Sudden Area Change

T-junction

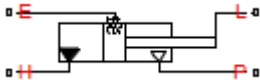


# Pneumo-Hydraulic Actuator

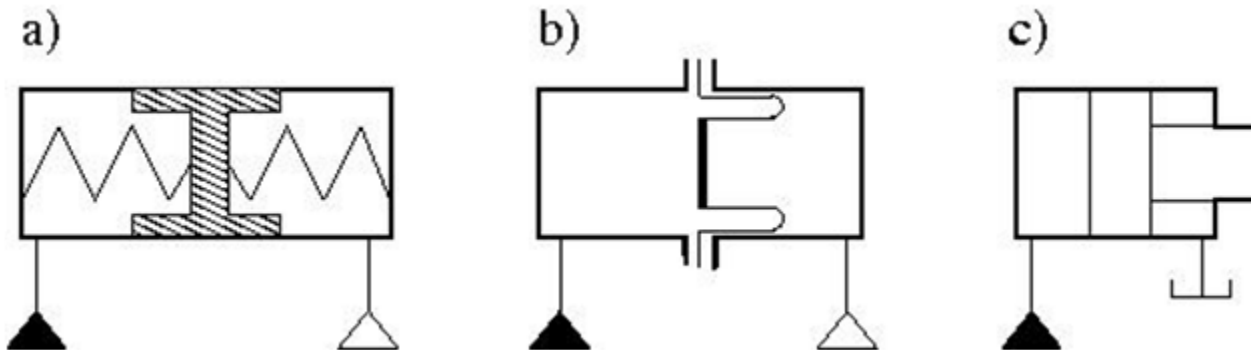
**Purpose** Double-acting cylinder operated by hydraulic and pneumatic power

**Library** Hydraulic Cylinders

## Description



The Pneumo-Hydraulic Actuator block simulates a pneumo-hydraulic actuator, implemented as a double-acting cylinder with one side connected to a hydraulic power supply and another side operated by pneumatic power. Such devices are widely used as pneumo-hydraulic pumps, intensifiers, and converters of various types. The following illustration shows a few examples of the pneumo-hydraulic actuator: a) with rigid separator; b) with flexible separator; c) pneumo-hydraulic intensifier.



The hydraulic part of the model accounts only for fluid consumption associated with the piston velocity. The pneumatic part of the model is built with the ideal gas relationships. To simulate the limit on the piston motion, the hard stop is included in the model. The piston effective area is assumed to be constant. As a result, the model is described with the following equations:

# Pneumo-Hydraulic Actuator

---

$$q_H = A_H \cdot v$$

$$F_H = A_H \cdot p$$

$$F_P = A_P \cdot p$$

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

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

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

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

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

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

where

$q_H$	Hydraulic volumetric flow rate
$p$	Pressure in the actuator chambers
$A_H$	Hydraulic side effective area
$A_P$	Pneumatic side effective area
$v$	Piston velocity
$F_H$	Force developed by piston on hydraulic side

$F_P$	Force developed by piston on pneumatic side
$F_{HS}$	Hard stop force
$F_L$	Force developed by external load connected to port L
$K$	Hard stop stiffness
$D$	Hard stop damping
<i>stroke</i>	Piston stroke
$x$	Piston displacement
$V_P$	Chamber volume on pneumatic side
$V_0$	Chamber dead volume on pneumatic side
$G$	Gas mass flow rate
$R$	Gas constant
$T$	Gas absolute temperature
$Q_P$	Heat flow through the pneumatic chamber
$c_v$	Gas specific heat at constant volume
$c_p$	Gas specific heat at constant pressure
$Q_{HE}$	Heat flow through the thermal port E

The model is suitable for building pneumo-hydraulic or hydro-pneumatic pumps, intensifiers, and similar devices. You can simulate piston loading (such as inertia, springs, friction) by modeling the load externally and connecting it to port L. Similarly, simulate the heat exchange with the environment through the external thermal port E, which corresponds to the gas in the chamber. Use blocks from the Simscape Foundation library, such as the Convective Heat Transfer, Conductive Heat Transfer, Thermal Mass, and so on, depending on the actual system configuration.

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

# Pneumo-Hydraulic Actuator

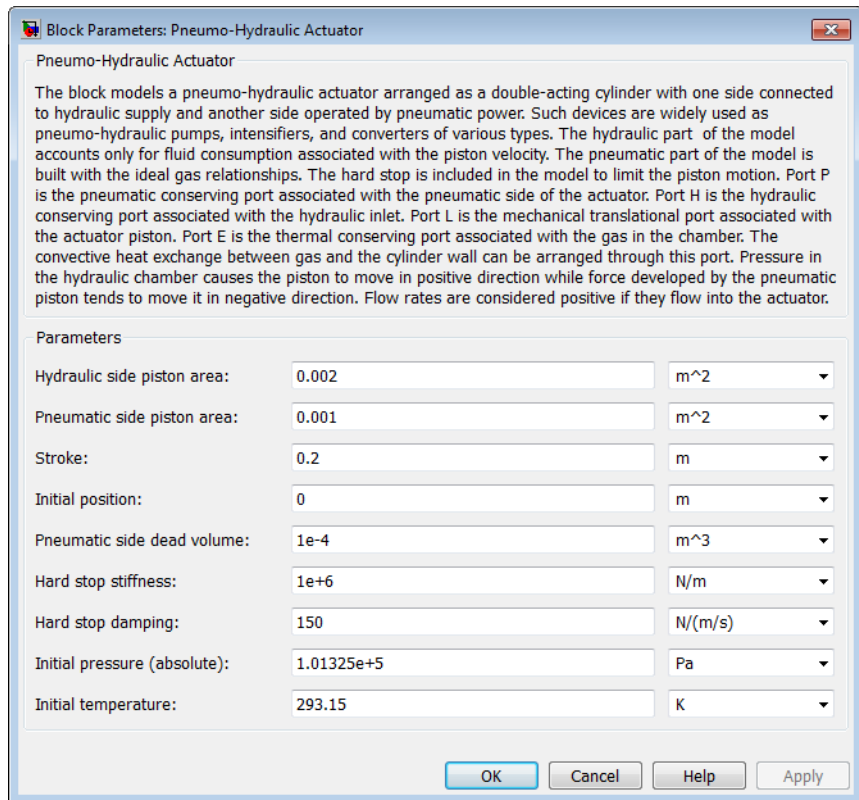
---

The block directionality assumes that pressure in the hydraulic chamber causes the piston to move in the positive direction, while pressure in the pneumatic chamber tends to move the piston in the negative direction. Flow rates are considered positive if they flow into the actuator.

## **Basic Assumptions and Limitations**

- The effective piston area in each chamber is assumed to be constant.
- The leakage flow between chambers is assumed to be negligible because pressures in the chambers are equal.
- On the hydraulic side, fluid compressibility is not taken into account.
- On the pneumatic side, the mass flow rate and heat flow computations assume that the gas is ideal.

## Dialog Box and Parameters



### Hydraulic side piston area

Effective piston area on the hydraulic side. The default value is  $20e-4 \text{ m}^2$ .

### Pneumatic side piston area

Effective piston area on the pneumatic side. The default value is  $10e-4 \text{ m}^2$ .

### Stroke

Piston maximum travel between caps. The default value is 0.2 m.

# Pneumo-Hydraulic Actuator

---

## **Initial position**

The distance between the piston and the cap on the hydraulic side at the beginning of simulation. This value cannot exceed the piston stroke. The default value is 0.

## **Pneumatic side dead volume**

Chamber dead volume on the pneumatic side. The default value is  $0.1 \times 10^{-3} \text{ m}^3$ .

## **Hard stop stiffness**

The hard stop model implemented in the block assumes that the stop resists penetration of the piston with force proportional to the penetration. This parameter sets the stiffness of the contact between the stop and the piston. The default value is  $1 \times 10^6 \text{ N/m}$ .

## **Hard stop damping**

The damping in the hard stop model accounts for dissipation in the piston-stop contact. The default value is  $150 \text{ N*s/m}$ .

## **Initial pressure (absolute)**

The initial absolute pressure in the pneumatic chamber. The default value is  $101325 \text{ Pa}$ .

## **Initial temperature**

The initial gas temperature in the pneumatic chamber. The default value is  $293.15 \text{ K}$ .

## **Ports**

The block has the following ports:

H

Hydraulic conserving port associated with the actuator hydraulic chamber.

P

Pneumatic conserving port associated with the actuator pneumatic chamber.

L

Mechanical translational conserving port associated with the actuator piston. You can model the load on the piston, such as

external force, inertia, friction, or spring, and connect it through this port.

E

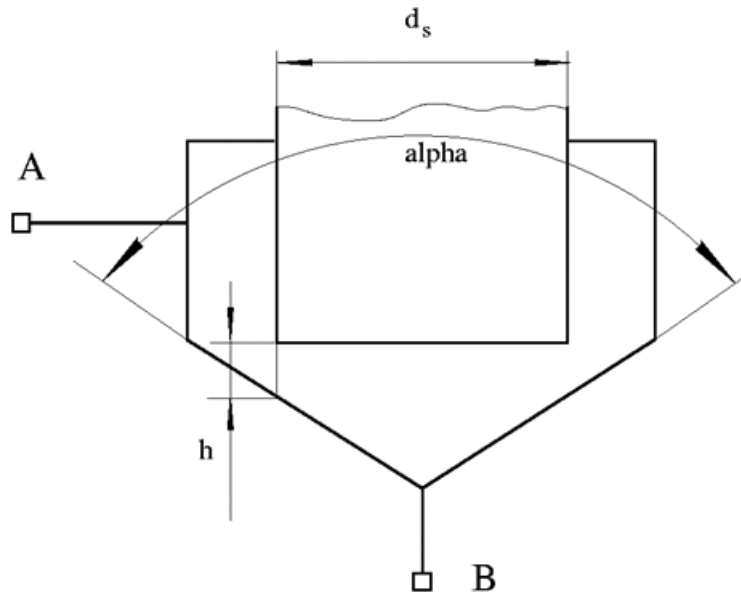
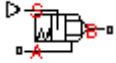
Thermal conserving port associated with the gas in the pneumatic chamber. You can simulate the heat exchange with the environment through this port.

# Poppet Valve

**Purpose** 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:



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

$$h = x_0 + x$$

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

$$p = p_A - p_B$$

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

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

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

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

where

$q$  Flow rate

$\rho$  Pressure differential

# Poppet Valve

---

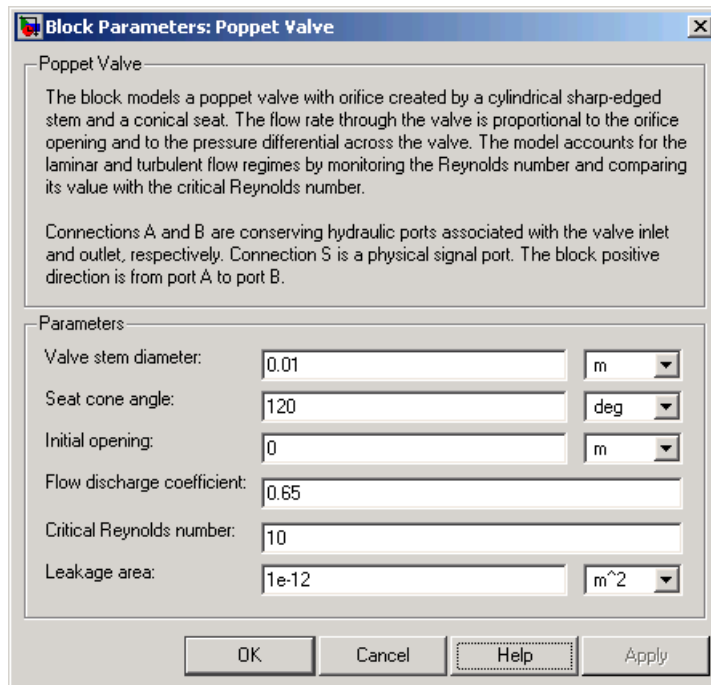
$p_A, p_B$	Gauge pressures at the block terminals
$C_D$	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
$x_0$	Initial opening
$x$	Stem displacement from initial position
$h$	Valve opening
$h_{max}$	Maximum valve opening. The passage area remains constant and equal to $A_{max}$ after this.
$d_s$	Stem diameter
$\alpha$	Cone angle
$\rho$	Fluid density
$D_H$	Valve instantaneous hydraulic diameter
$\nu$	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

- 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



### 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

# Poppet Valve

---

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 simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## **Global Parameters**

Parameters determined by the type of working fluid:

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

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

## **Ports**

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

S

Physical signal port to control spool displacement.

## **See Also**

Ball Valve

Ball Valve with Conical Seat

Gate Valve

Needle Valve

Pressure-Compensated Flow Control Valve

# Porting Plate Variable Orifice

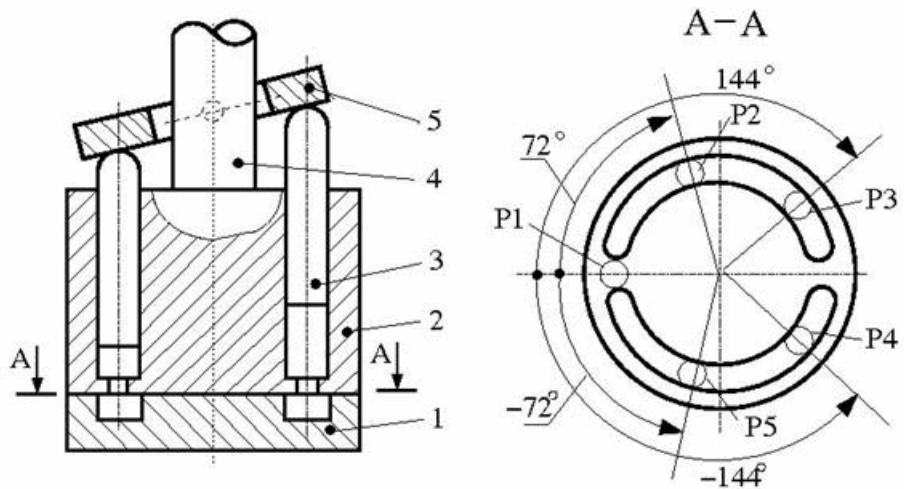
**Purpose** Variable orifice between piston and porting plate

**Library** Pumps and Motors

## Description



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



The preceding diagram shows an axial-piston machine with five pistons, where:

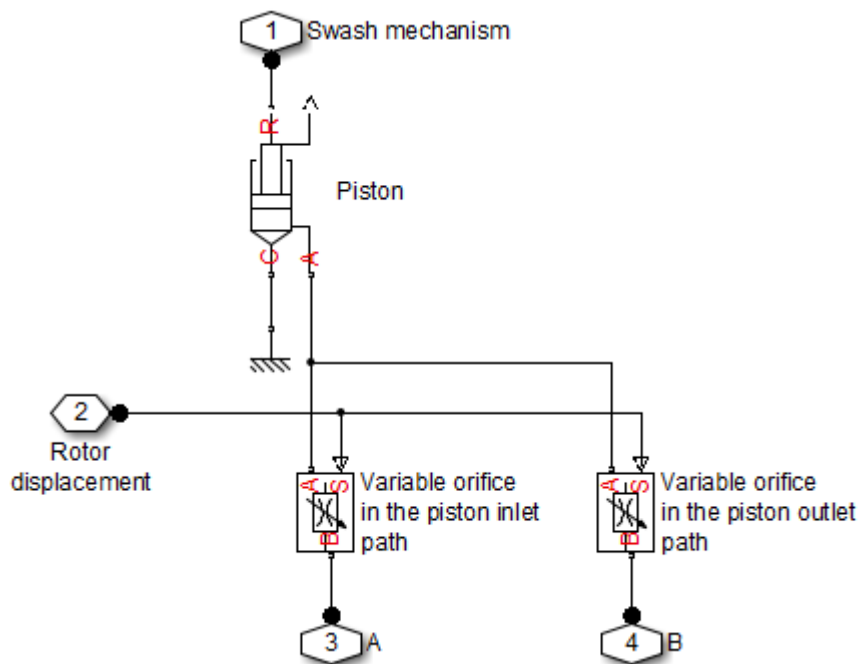
**1** Porting plate

**2** Rotor

# Porting Plate Variable Orifice

- 3 Piston
- 4 Driving shaft
- 5 Swash plate

During rotor rotation, every piston gets connected to one of the porting slots. These piston connections, modeled as variable orifices, are reflected in the schematic model of a piston, shown in the following illustration.

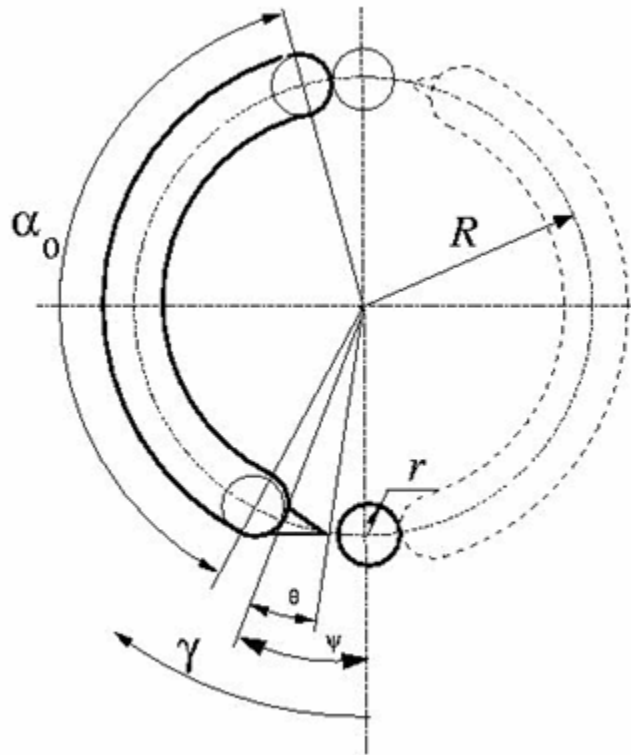


The Porting Plate Variable Orifice block is a model of a variable orifice created between the piston chamber and a slot on a porting plate. Two Porting Plate Variable Orifice blocks are necessary in the model of a

# Porting Plate Variable Orifice

piston, to simulate connection to the intake and the discharge port, respectively.

The calculation diagram of the porting plate variable orifice is shown in the next illustration.



The model assumes that the slot is shifted by a pressure carryover angle  $\psi$  from the reference point in the direction of orifice rotation. Generally,  $r/R \leq \psi \leq 3r/R$ , where  $r$  is the orifice radius and  $R$  is the piston pitch radius. A small triangular-shaped transition slot is placed at the beginning of the major slot, to avoid sudden pressure change. To



avoid direct connection between the intake and discharge slots, the transition slot angle  $\theta$  must be less than  $\psi - r / R$ . The transition slot area is assumed to be linearly dependent on the rotation angle and characterized by its maximum area. There are six distinctive angles defining the relationship between the rotation angle  $\gamma$  and the orifice opening, as listed in the following table.

No	Nomenclature	Value	Description
1	$\gamma_1$	$\psi - \theta - r / R$	Opening of transition slot starts.
2	$\gamma_2$	$\psi - r / R$	Opening of the major slot starts. The orifice contacts with the transition slot.
3	$\gamma_3$	$\psi + r / R$	Major slot full opening starts.
4	$\gamma_4$	$\psi + r / R + 0.01$	Transition slot ends.
5	$\gamma_5$	$\pi - 2 \cdot r / R$	Major slot full opening ends.
6	$\gamma_6$	$\pi$	Major slot opening ends.

The variable orifice is fully opened in the  $\alpha_0$  region

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

where  $\gamma$  is the rotation angle.

The orifice area is computed with the following equations:

$$\begin{aligned}
 A &= A_{leak} && \text{for } \gamma_6 < \gamma \leq \gamma_1 \\
 A &= G \cdot (\gamma - \gamma_1) + A_{leak} && \text{for } \gamma_1 < \gamma \leq \gamma_2 \\
 A &= A_{tr} + r^2 (\beta - \sin \beta) + A_{leak} && \text{for } \gamma_2 < \gamma \leq \gamma_3 \\
 A &= A_{tr} \cdot (1 - 100 \cdot (\gamma - \gamma_3)) + \pi \cdot r^2 + A_{leak} && \text{for } \gamma_3 < \gamma \leq \gamma_4 \\
 A &= \pi \cdot r^2 + A_{leak} && \text{for } \gamma_4 < \gamma \leq \gamma_5 \\
 A &= r^2 (\beta - \sin \beta) + A_{leak} && \text{for } \gamma_5 < \gamma \leq \gamma_6
 \end{aligned}$$

# Porting Plate Variable Orifice

---

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

where

$A$  Orifice area

$A_{tr}$  Transition slot maximum area

$A_{leak}$  Closed orifice leakage area

After the area has been determined, the flow rate through the orifice is computed with the following equations:

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

$$p = p_A - p_B$$

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

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

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

where

$q$	Flow rate
$p$	Pressure differential
$p_A, p_B$	Gauge pressures at the block terminals
$C_D$	Flow discharge coefficient
$D_H$	Orifice instantaneous hydraulic diameter
$\rho$	Fluid density
$\nu$	Fluid kinematic viscosity
$Re$	Reynolds number
$Re_{cr}$	Critical Reynolds number

The Porting Plate Variable Orifice block model is essentially a building block intended for use in various piston machines. The model accounts for the flow regime by computing the Reynolds number and comparing it with its critical value. No inertial effects are considered in the model.

Connections A and B are hydraulic conserving ports associated with the inlet and outlet of the orifice. Connection G is a physical signal port associated with the input signal for the cylinder block angle. The input signal at port G is treated as an angle, in radians, therefore it is essential that the input signal representing angular displacement is kept within the range from 0 to  $2\pi$ . Connect port G to the output port of an Angle Sensor block to meet this requirement.

The flow rate is considered positive if it flows from A to B. At initial position, the orifice is assumed to be shifted by the pressure carryover angle from the slot. When the block rotates in the positive direction, the orifice starts opening. To adjust the initial position of the orifice with respect to the slot, use the **Phase angle** parameter.

# Porting Plate Variable Orifice

## Basic Assumptions and Limitations

- The model accounts for the viscous friction in the piston-plate contact.
- No inertial effects are considered.
- The plate angular displacements are considered to be small.
- The joint between the piston and the plate permanently maintains contact between the piston and the plate.

## Dialog Box and Parameters

Block Parameters: Porting Plate Variable Orifice

Porting Plate Variable Orifice

The block is a model of a variable orifice located between the piston chamber and a crescent-shaped slot on a porting plate in hydraulic axial-piston machines. The pistons, fitted in the machine rotor, are carried along the porting plate slots, thus being periodically connected to either the intake or discharge port of the machine. The block is a model of a fluid path between the piston and one of the slots. The model is essentially a building block intended to build models of various hydraulic machines. The model accounts for the flow regime by computing the Reynolds number and comparing it with its critical value. No inertial effects are considered in the model. Connections A and B are the conserving hydraulic ports associated with the inlet and outlet of the orifice. Connection G is a physical signal port through which the cylinder block angle is imported. Whatever signal is connected to port G, it is treated as an angle in rad. The flow rate is considered positive if it flows from A to B, and pressure differential is determined as  $p = P_A - P_B$ . At initial position, the orifice is assumed to be shifted by the pressure carryover angle from the slot. The front edge of the slot is equipped with the transition slot. The orifice starts getting opened provided the block rotates in positive direction. By using the Phase angle parameter, the initial position can be adjusted to reflect the actual position of the orifice with respect to the slot. The angular displacement of the block provided to the model via port G must be kept within  $0 - 2\pi$  range.

Parameters

Piston pitch radius:	0.05	m
Orifice diameter:	0.005	m
Pressure carryover angle:	0.06	rad
Phase angle:	0	rad
Transition slot angle:	0.01	rad
Transition slot maximum area:	1e-6	m <sup>2</sup>
Flow discharge coefficient:	0.65	
Critical Reynolds number:	12	
Leakage area:	1e-9	m <sup>2</sup>

OK Cancel Help Apply

**Piston pitch radius**

The radius of the pitch circle where the pistons are located. The parameter must be greater than zero. The default value is 0.05 m.

**Orifice diameter**

The diameter of the orifice at the bottom of the piston chamber. The parameter must be greater than zero. The default value is 0.005 m.

**Pressure carryover angle**

The angle introduced between to slot and the orifice to avoid abrupt pressure change and decrease leakage. This angle is marked  $\psi$  in the preceding calculation diagram. The angle is expected to be in the range  $r / R \leq \psi \leq 3 r / R$ , where  $r$  is the orifice radius and  $R$  is the piston pitch radius. The default value is 0.06 rad.

**Phase angle**

This parameter sets the orifice initial angular position with respect to the slot. The default value is 0, which means that the orifice is shifted by the pressure carryover angle from the slot.

**Transition slot angle**

The angle covered by a small, triangular-shaped transition slot placed before the major slot to avoid sudden pressure change, as shown in the preceding calculation diagram. To avoid direct connection between the intake and discharge slots, the transition slot angle  $\theta$  must be less than  $\psi - r / R$ . The default value is 0.01 rad.

**Transition slot maximum area**

The maximum cross-sectional area of the transition slot. The slot area is assumed to be linearly-dependent on the rotation angle and reaches its maximum when the orifice approaches the major slot. The default value is 1e-6 m.

**Flow discharge coefficient**

Semi-empirical parameter for orifice capacity characterization. Its value depends on the geometrical properties of the orifice, and

# Porting Plate Variable Orifice

---

usually is provided in textbooks or manufacturer data sheets. The default value is 0.6.

## Critical Reynolds number

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is 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.

## Leakage area

The total area of possible leaks in the completely closed orifice. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-9 \text{ m}^2$ .

## Global Parameters

Parameters determined by the type of working fluid:

- Fluid density
- Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the orifice inlet.

B

Hydraulic conserving port associated with the orifice outlet.

G

Physical signal input port that conveys the angular position of the orifice to the block. The signal applied to the port is treated as an angle, in radians, and must be in the range between 0 and  $2\pi$ .

## Examples

The Hydraulic Axial-Piston Pump with Load-Sensing and Pressure-Limiting Control example models a test rig designed to investigate interaction between an axial-piston pump and a typical control unit, simultaneously performing the load-sensing and pressure-limiting functions. To assure required accuracy, the model of the pump must account for such features as interaction between pistons, swash plate, and porting plate, which makes it necessary to build a detailed pump model.

## See Also

Angle Sensor

Swash Plate

# Pressure-Compensated 3-Way Flow Control Valve

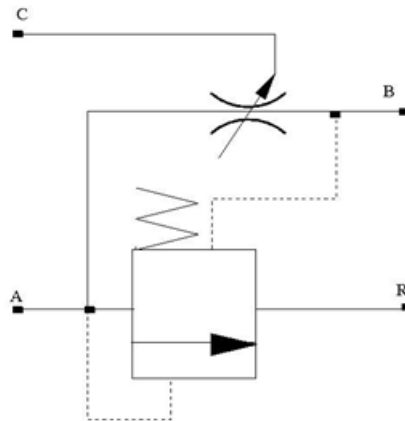
**Purpose** Hydraulic pressure compensating 3-way valve

**Library** Flow Control Valves

## Description



The Pressure-Compensated 3-Way Flow Control Valve block represents a 3-way pressure-compensated flow control valve as a data-sheet-based model. The valve model includes a variable orifice and a normally closed pressure-control valve, connected in parallel with the orifice. The purpose of the pressure-control valve is to maintain a preset pressure differential across the orifice by diverting some flow from port A to the tank (port R) if the pressure differential exceeds the preset value. Port C controls the orifice opening, as shown in the following figure.



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

- By maximum area and opening — Use this option if the data sheet provides only the orifice maximum area and the control member maximum stroke.



# Pressure-Compensated 3-Way Flow Control Valve

- By area vs. opening table — Use this option if the catalog or data sheet provides a table of the orifice passage area based on the control member displacement  $A=A(h)$ .

In the first case, the passage area is assumed to be linearly dependent on the control member displacement, that is, the orifice is assumed to be closed if the initial orifice opening is set to zero and the control member position also equals zero. The maximum orifice opening takes place at the maximum displacement. In the second case, the passage area is determined by one-dimensional interpolation from the table  $A=A(h)$ .

The pressure-control valve representation does not account for inertia, friction, or hydraulic forces. The valve has the following area-pressure differential relationship:

$$A_{pc} = \begin{cases} 0 & \text{for } p < p_{set} \\ (p - p_{set}) \frac{A_{max\_pc}}{p_{reg}} & \text{for } p_{set} \leq p \leq (p_{set} + p_{reg}) \\ A_{max\_pc} & \text{for } p > (p_{set} + p_{reg}) \end{cases}$$

where

$A_{pc}$	Pressure-control valve passage area
$p$	Pressure differential across the orifice
$p_{set}$	Preset pressure differential
$p_{reg}$	Regulation range
$A_{max\_pc}$	Pressure-control valve maximum area

Both for the variable orifice and the pressure compensator, a small leakage area is assumed to exist even after the orifice is completely closed. Physically, it represents a possible clearance in the closed valve, but the main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or

# Pressure-Compensated 3-Way Flow Control Valve

---

“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 ( $Re_{cr}$ ). After the area has been determined, the block computes the flow rate for both the orifice and the pressure compensator according to the following equations:

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

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

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

$$h = x_0 + x$$

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

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

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

# Pressure-Compensated 3-Way Flow Control Valve

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

where

$q$	Flow rate
$p$	Pressure differential
$p_A, p_B, p_R$	Gauge pressures at the block terminals
$C_D$	Flow discharge coefficient
$A$	Instantaneous orifice passage area
$A_{\max}$	Orifice maximum area
$h_{\max}$	Control member maximum displacement
$x_0$	Initial opening
$x$	Control member displacement from initial position
$h$	Orifice opening
$\rho$	Fluid density
$D_H$	Instantaneous orifice hydraulic diameter
$\nu$	Fluid kinematic viscosity
$A_{\text{leak}}$	Closed orifice leakage area

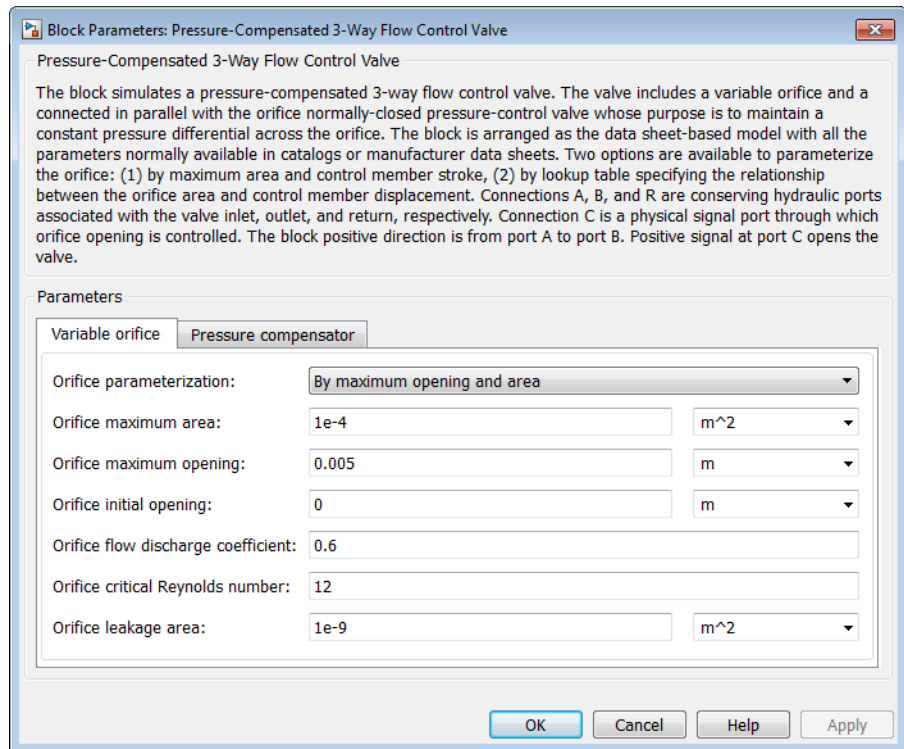
Connections A, B, and R are the conserving hydraulic ports associated with the valve inlet, outlet, and return, respectively. Connection C is a physical signal port through which the orifice opening is controlled. The block positive direction is from port A to port B. Positive signal at port C opens the valve.

## Assumptions and Limitations

Fluid inertia and pressure compensator loading are not taken into account.

# Pressure-Compensated 3-Way Flow Control Valve

## Dialog Box and Parameters



- “Variable Orifice Tab” on page 1-478
- “Pressure Compensator Tab” on page 1-481

## Variable Orifice Tab

### Orifice parameterization

Select one of the following methods for specifying the orifice:

- By maximum opening and area — Provide values for the maximum orifice area and the maximum orifice opening. The passage area is linearly dependent on the control member displacement, that is, the orifice is closed at the initial position of the control member (zero displacement), and the maximum

# Pressure-Compensated 3-Way Flow Control Valve

---

opening takes place at the maximum displacement. This is the default method.

- **By orifice area vs. orifice opening table** — Provide tabulated data of orifice openings and corresponding orifice areas. The passage area is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.

## **Orifice maximum area**

The area of a fully opened orifice. The parameter value must be greater than zero. The default value is  $1e-4 \text{ m}^2$ . This parameter is used if **Orifice parameterization** is set to **By maximum opening and area**.

## **Orifice maximum opening**

The maximum displacement of the control member. The parameter value must be greater than zero. The default value is 0.005 m. This parameter is used if **Orifice parameterization** is set to **By maximum opening and area**.

## **Tabulated orifice openings**

Vector of input values for orifice openings, specified as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in mm, are [-6, 0, 1, 3, 6, 9, 15]. This parameter is used if **Orifice parameterization** is set to **By orifice area vs. orifice opening table**. **Tabulated orifice openings** values will be used together with **Tabulated orifice area** values for one-dimensional table lookup.

## **Tabulated orifice area**

Vector of orifice areas, specified as a one-dimensional array. The vector must be the same size as the orifice openings vector. All the values must be positive. The default values, in  $\text{cm}^2$ , are [ $1e-7$ ,  $1e-6$ , 0.08, 0.145, 0.96, 1, 1.001]. This parameter is used

# Pressure-Compensated 3-Way Flow Control Valve

---

if **Orifice parameterization** is set to By orifice area vs. orifice opening table.

## Interpolation method

This parameter is used if **Orifice parameterization** is set to By orifice area vs. orifice opening table. Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- **Linear** — Uses a linear interpolation function.
- **Cubic** — Uses the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP).
- **Spline** — Uses the cubic spline interpolation algorithm.

For more information on interpolation algorithms, see the [tablelookup](#) reference page.

## Extrapolation method

This parameter is used if **Orifice parameterization** is set to By orifice area vs. orifice opening table. Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- **Linear** — 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.
- **Nearest** — Uses the last specified output value at the appropriate end of the range. That is, the block uses the last specified output value for all input values greater than the last specified input argument, and the first specified output value for all input values less than the first specified input argument.

# Pressure-Compensated 3-Way Flow Control Valve

---

For more information on extrapolation algorithms, see the `tablelookup` reference page.

## **Orifice initial opening**

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

## **Orifice flow discharge coefficient**

Semi-empirical coefficient that is used in the computation of flow rate through the orifice. Its value depends on the geometrical properties of the orifice and usually is provided in textbooks or manufacturer data sheets. The default value is 0.6.

## **Orifice 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 the orifice geometrical profile. You can find the recommendations on the parameter value in hydraulic textbooks. The default value is 12.

## **Orifice leakage area**

The total area of possible leaks in the completely closed orifice. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-9 \text{ m}^2$ .

## **Pressure Compensator Tab**

### **Pressure differential across orifice setting**

Pressure difference that must be maintained across the variable orifice by the pressure compensator. The default value is  $6e5 \text{ Pa}$ .

# Pressure-Compensated 3-Way Flow Control Valve

---

## **Pressure differential regulation range**

Pressure increase over the preset level needed to fully open the pressure compensator orifice. The lower the value of the range, the higher the valve sensitivity. The default value is  $0.5e5$  Pa.

## **Pressure compensator maximum area**

The area of a fully opened pressure compensator. The parameter value must be greater than zero. The default value is  $1e-4$  m<sup>2</sup>.

## **Pressure compensator flow discharge coefficient**

Semi-empirical coefficient that is used in the computation of flow rate through the pressure compensator. The default value is 0.6.

## **Pressure compensator 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 the orifice geometrical profile. You can find the recommendations on the parameter value in hydraulic textbooks. The default value is 12.

## **Pressure compensator leakage area**

The total area of possible leaks in the completely closed orifice. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-9$  m<sup>2</sup>.

## **Restricted Parameters**

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

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



# Pressure-Compensated 3-Way Flow Control Valve

---

- **Extrapolation method**

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

## **Global Parameters**

Parameters determined by the type of working fluid:

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

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

## **Ports**

The block has the following ports:

- A  
Hydraulic conserving port associated with the valve inlet.
- B  
Hydraulic conserving port associated with the valve outlet.
- R  
Hydraulic conserving port that connects with the tank.
- C  
Physical signal control port.

## **See Also**

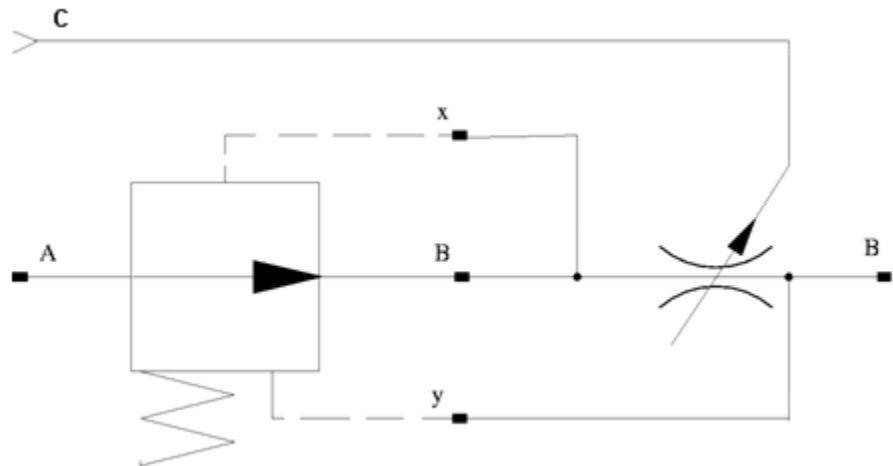
Pressure-Compensated Flow Control Valve

# Pressure-Compensated Flow Control Valve

**Purpose** Hydraulic pressure compensating valve

**Library** Flow Control Valves

**Description** The Pressure-Compensated Flow Control Valve block represents a pressure-compensated flow control valve as a data-sheet-based model. The valve is based on a Pressure Compensator block installed upstream from a Variable Orifice block, as shown in the following illustration.



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

- By maximum area and opening — Use this option if the data sheet provides only the orifice maximum area and the control member maximum stroke.
- By area vs. opening table — Use this option if the catalog or data sheet provides a table of the orifice passage area based on the control member displacement  $A=A(h)$ .

# Pressure-Compensated Flow Control Valve

---

In the first case, the passage area is assumed to be linearly dependent on the control member displacement, that is, the orifice is assumed to be closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. In the second case, the passage area is determined by one-dimensional interpolation from the table  $A=A(h)$ . In both cases, a small leakage area is assumed to exist even after the orifice is completely closed. Physically, it represents a possible clearance in the closed valve, but the main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation.

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

## **Assumptions and Limitations**

Fluid inertia is not taken into account.

# Pressure-Compensated Flow Control Valve

## Dialog Box and Parameters

**Block Parameters: Pressure-Compensated Flow Control Valve**

Pressure-Compensated Flow Control Valve

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

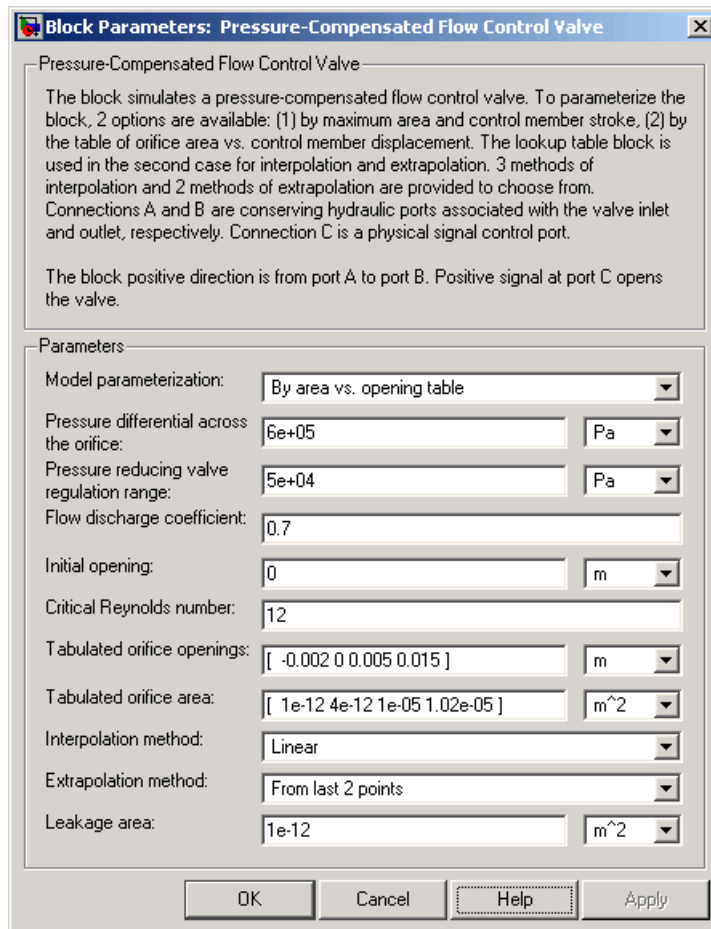
The block positive direction is from port A to port B. Positive signal at port C opens the valve.

Parameters:

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

OK Cancel Help Apply

# Pressure-Compensated Flow Control Valve



## Model parameterization

Select one of the following methods for specifying the orifice:

- **By maximum area and opening** — Provide values for the maximum orifice area and the maximum orifice opening. The passage area is linearly dependent on the control member displacement, that is, the orifice is closed at the initial position

# Pressure-Compensated Flow Control Valve

---

of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.

- **By area vs. opening table** — Provide tabulated data of orifice openings and corresponding orifice areas. The passage area is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.

## **Orifice maximum area**

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

## **Orifice maximum opening**

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

## **Tabulated orifice openings**

Specify the vector of input values for orifice openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in meters, are  $[-2e-3, 0, 5e-3, 15e-3]$ . This parameter is used if **Model parameterization** is set to **By area vs. opening table**. **Tabulated orifice openings** values will be used together with **Tabulated orifice area** values for one-dimensional table lookup.

## **Tabulated orifice area**

Specify the vector of orifice areas as a one-dimensional array. The vector must be of the same size as the orifice openings vector. All the values must be positive. The default values, in  $\text{m}^2$ , are

# Pressure-Compensated Flow Control Valve

---

[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 Polynomial (PCHIP).
- **Spline** — Uses the cubic spline interpolation algorithm.

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

## Extrapolation method

This parameter is used if **Model parameterization** is set to By area vs. opening table. Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

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

# Pressure-Compensated Flow Control Valve

---

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

## **Pressure differential across the orifice**

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

## **Pressure reducing valve regulation range**

Pressure increase over the preset level needed to fully close the valve. 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 simulation to fail. Therefore, MathWorks



# Pressure-Compensated Flow Control Valve

---

recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## Restricted Parameters

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

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

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

## Global Parameters

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

- A  
Hydraulic conserving port associated with the valve inlet.
- B  
Hydraulic conserving port associated with the valve outlet.
- C  
Physical signal control port.

# Pressure-Compensated Flow Control Valve

---

## **See Also**

Ball Valve

Ball Valve with Conical Seat

Gate Valve

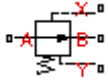
Needle Valve

Poppet Valve

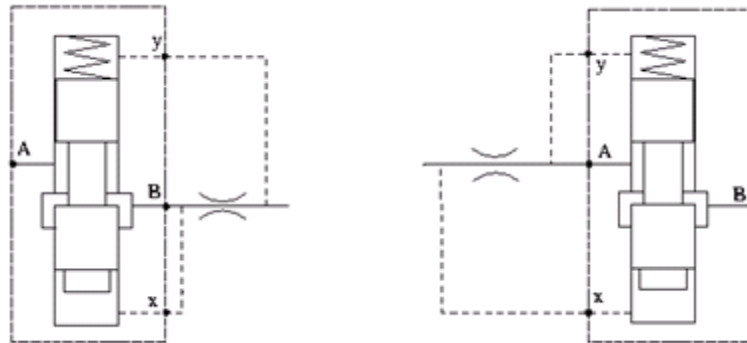
**Purpose** Hydraulic pressure compensating valve

**Library** Pressure Control Valves

## Description



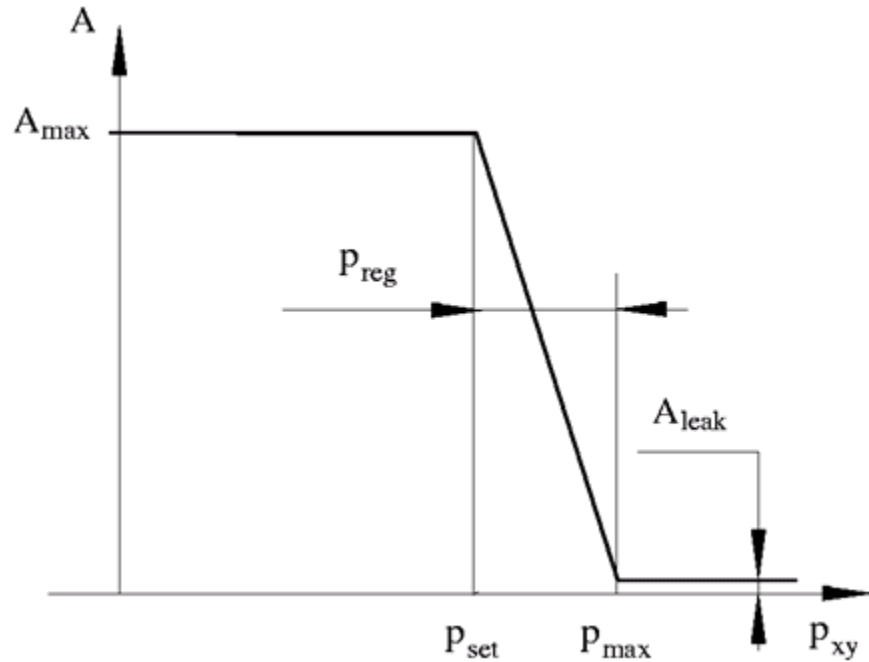
The Pressure Compensator block represents a hydraulic pressure compensating valve, or pressure compensator. Pressure compensators are used to maintain preset pressure differential across a hydraulic component to minimize the influence of pressure variation on a flow rate passing through the component.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



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

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

# Pressure Compensator



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

The main parameters of the block are the valve maximum area and regulation range. In addition, you need to specify the leakage area of the valve. Physically, it represents a possible clearance in the closed valve, but the main purpose of the parameter is to maintain numerical

integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation.

The model accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number ( $Re$ ) and comparing its value with the critical Reynolds number ( $Re_{cr}$ ). The flow rate is computed according to the following equations:

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

$$h = x_0 + x \cdot \rho r$$

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

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

$$p = p_A - p_B$$

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

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

# Pressure Compensator

---

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

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

where

$q$	Flow rate
$p$	Pressure differential across the valve
$p_{xy}$	Pressure differential across valve control terminals
$p_A, p_B$	Gauge pressures at the valve main terminals
$p_x, p_y$	Gauge pressures at the valve control terminals
$p_{set}$	Valve preset pressure
$p_{max}$	Pressure needed to fully close the orifice
$p_{reg}$	Regulation range
$A(h)$	Instantaneous orifice passage area
$A_{max}$	Orifice maximum area
$C_D$	Flow discharge coefficient
$\rho$	Fluid density
$D_H$	Instantaneous orifice hydraulic diameter
$\nu$	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

- Valve opening is linearly proportional to the pressure differential.
- No loading on the valve, such as inertia, friction, spring, and so on, is considered.
- Flow consumption associated with the spool motion is neglected.

## Dialog Box and Parameters

**Block Parameters: Pressure Compensator**

Pressure Compensator

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

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

Parameters

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

OK Cancel Help Apply

### Maximum passage area

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

# Pressure Compensator

---

## **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 simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12$  m<sup>2</sup>.

## **Global Parameters**

Parameters determined by the type of working fluid:

- Fluid density
- Fluid kinematic viscosity



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

## Ports

The block has the following ports:

- A Hydraulic conserving port associated with the valve inlet.
- B Hydraulic conserving port associated with the valve outlet.
- X Hydraulic conserving port associated with the pressure control terminal that opens the orifice.
- Y Hydraulic conserving port associated with the pressure control terminal that closes the orifice.

## See Also

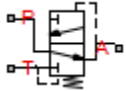
Pressure Reducing Valve  
Pressure Relief Valve

# Pressure Reducing 3-Way Valve

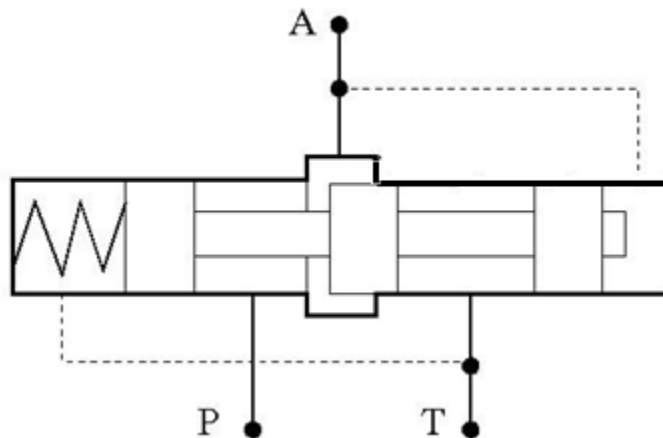
**Purpose** Pressure reducing/relieving valve

**Library** Pressure Control Valves

## Description



The Pressure Reducing 3-Way Valve block represents a hydraulic 3-way valve that is also known as a pressure reducing/relieving valve. The valve reduces inlet pressure to a preset value, similar to a conventional pressure reducing valve, but, additionally, starts working as a pressure-relief valve if the pressure continues to rise.



Initially, orifice P-A is fully open. It remains fully open while the outlet pressure at port A is lower than the valve preset pressure. When the preset pressure is reached, the valve control member is forced off its stop and starts closing the orifice, thus trying to maintain outlet pressure at the preset level. Any further increase in the outlet pressure causes the control member to close the orifice even more, until the orifice is fully closed. The pressure increase needed to close the valve is referred to as regulation range, and is generally provided in the catalogs, along with the valve maximum area. In a conventional pressure reducing valve, outlet pressure is no longer under control after

# Pressure Reducing 3-Way Valve

orifice P-A is closed. The pressure reducing 3-way valve provides an additional relieving function by diverting some flow from the outlet to a tank through an additional orifice A-T.

The valve has the following area-pressure differential relationship:

$$A_{\text{red}} = A_{\text{med}} - (A_{\text{max}} - A_{\text{med}}) \cdot \tanh(k \cdot (p - p_{\text{red\_med}}) / (p_{\text{red\_max}} - p_{\text{red\_med}}))$$

$$A_{\text{rel}} = A_{\text{med}} + (A_{\text{max}} - A_{\text{med}}) \cdot \tanh(k \cdot (p - p_{\text{rel\_med}}) / (p_{\text{rel\_max}} - p_{\text{rel\_med}}))$$

$$A_{\text{med}} = (A_{\text{max}} + A_{\text{leak}}) / 2$$

$$p_{\text{red\_max}} = p_{\text{set}} + p_{\text{reg}}$$

$$p_{\text{red\_med}} = (p_{\text{set}} + p_{\text{reg}}) / 2$$

$$p_{\text{rel\_set}} = p_{\text{set}} + p_{\text{reg}} + p_{\text{tr}}$$

$$p_{\text{rel\_max}} = p_{\text{rel\_set}} + p_{\text{reg}}$$

$$p_{\text{rel\_med}} = (p_{\text{rel\_set}} + p_{\text{reg}}) / 2$$

where

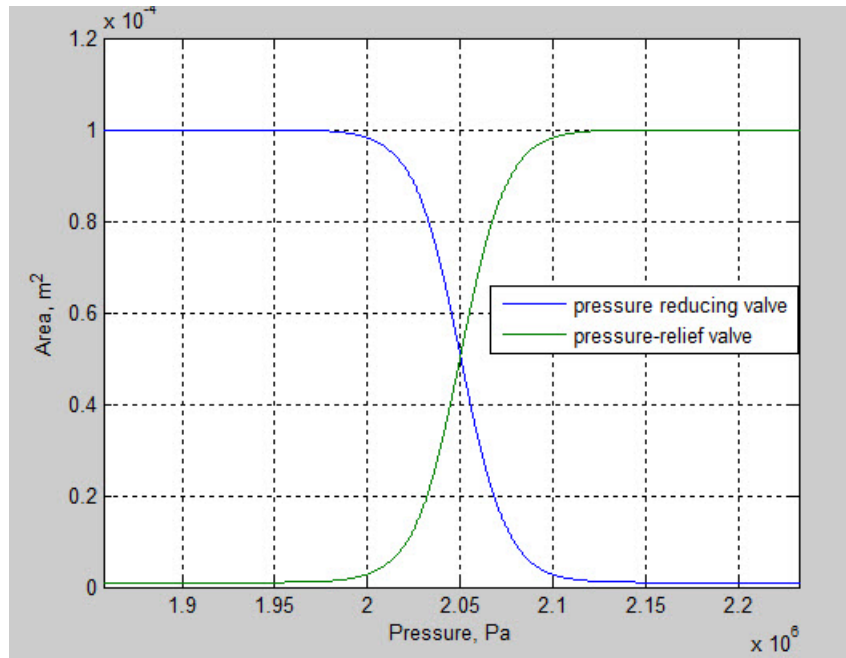
$A_{\text{max}}$	Maximum opening area of both the reducing and relieving valves
$A_{\text{red}}$	Pressure reducing (orifice P-A) valve opening area
$A_{\text{rel}}$	Pressure-relief (orifice A-T) valve opening area
$A_{\text{leak}}$	Leakage area (the area that remains open even after the orifice is completely closed)
$p$	Pressure drop across the valve, $p_A - p_T$
$p_{\text{set}}$	Preset pressure differential
$p_{\text{reg}}$	Regulation range

# Pressure Reducing 3-Way Valve

$p_{tr}$  Transition pressure (the pressure increment above the pressure of the fully closed reducing valve and setting pressure of the pressure-relief valve)

$k$  Valve opening adjustment coefficient

The distinctive feature of a pressure reducing 3-way valve is a sharp change of orifice openings. To avoid computational problems, the openings are approximated using the hyperbolic tangent function. The following figure shows an example of the relationship between the opening area and pressure.



In the example, the setting pressure is set to 20e5 Pa, and the regulation range is set to 1e5 Pa. The valve opening adjustment coefficient is set to 2. The higher the value of the coefficient, the closer the transition is to

# Pressure Reducing 3-Way Valve

a linear relationship. The transition is close to the experimental data at  $k$  in the range of 3 to 4.

For both orifices, a small leakage area is assumed to exist even after the orifice is completely closed. Physically, it represents a possible clearance in the closed valve, but the main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or "hanging" part of the system could affect computational efficiency and even cause failure of computation.

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 areas have been determined, the block computes the flow rate for both orifices according to the following equations:

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

$$A = \begin{cases} A_{red} & \text{for orifice P-A} \\ A_{rel} & \text{for orifice A-T} \end{cases}$$

$$p = \begin{cases} p_P - p_A & \text{for orifice P-A} \\ p_A - p_T & \text{for orifice A-T} \end{cases}$$

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

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

# Pressure Reducing 3-Way Valve

---

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

where

$q$	Flow rate
$p$	Pressure differential
$p_A, p_B, p_R$	Gauge pressures at the block terminals
$C_D$	Flow discharge coefficient
$A$	Instantaneous orifice passage area
$\rho$	Fluid density
$D_H$	Instantaneous orifice hydraulic diameter
$\nu$	Fluid kinematic viscosity

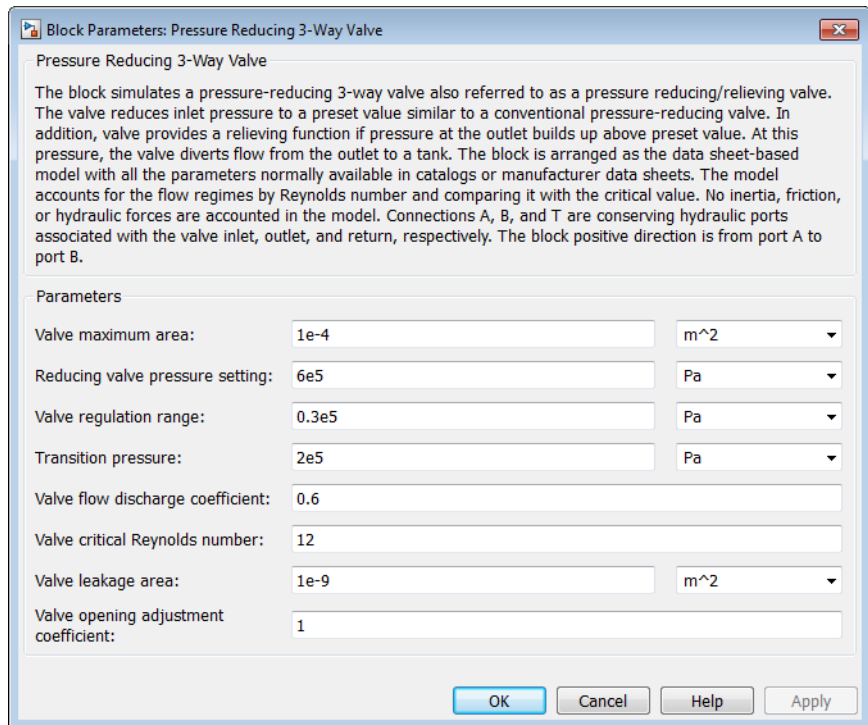
Connections P, A, and T are the conserving hydraulic ports associated with the valve inlet, outlet, and return, respectively. The block positive direction is from port P to port A and from port A to port T.

## **Assumptions and Limitations**

Fluid inertia, valve inertia, friction, and hydraulic forces are not taken into account.

# Pressure Reducing 3-Way Valve

## Dialog Box and Parameters



**Block Parameters: Pressure Reducing 3-Way Valve**

Pressure Reducing 3-Way Valve

The block simulates a pressure-reducing 3-way valve also referred to as a pressure reducing/relieving valve. The valve reduces inlet pressure to a preset value similar to a conventional pressure-reducing valve. In addition, valve provides a relieving function if pressure at the outlet builds up above preset value. At this pressure, the valve diverts flow from the outlet to a tank. The block is arranged as the data sheet-based model with all the parameters normally available in catalogs or manufacturer data sheets. The model accounts for the flow regimes by Reynolds number and comparing it with the critical value. No inertia, friction, or hydraulic forces are accounted in the model. Connections A, B, and T are conserving hydraulic ports associated with the valve inlet, outlet, and return, respectively. The block positive direction is from port A to port B.

Parameters

Valve maximum area:	<input type="text" value="1e-4"/>	<input type="text" value="m^2"/>
Reducing valve pressure setting:	<input type="text" value="6e5"/>	<input type="text" value="Pa"/>
Valve regulation range:	<input type="text" value="0.3e5"/>	<input type="text" value="Pa"/>
Transition pressure:	<input type="text" value="2e5"/>	<input type="text" value="Pa"/>
Valve flow discharge coefficient:	<input type="text" value="0.6"/>	
Valve critical Reynolds number:	<input type="text" value="12"/>	
Valve leakage area:	<input type="text" value="1e-9"/>	<input type="text" value="m^2"/>
Valve opening adjustment coefficient:	<input type="text" value="1"/>	

OK Cancel Help Apply

### Valve maximum area

The area of a fully opened orifice. Both orifices are assumed to have the same maximum area. The parameter value must be greater than zero. The default value is  $1e-4 \text{ m}^2$ .

### Reducing valve pressure setting

Preset pressure level, at which the orifice P-A of the valve starts to close. The default value is  $6e5 \text{ Pa}$ .

### Valve regulation range

Pressure increase over the preset level needed to fully close the pressure reducing valve. The lower the value of the range, the higher the valve sensitivity. The default value is  $0.3e5 \text{ Pa}$ .

# Pressure Reducing 3-Way Valve

---

## **Transition pressure**

Pressure increment above the pressure of the fully closed reducing valve needed to reach the pressure at which the pressure-relief valve starts opening. The transition pressure must be greater than or equal to zero. The default value is 2e5 Pa.

## **Valve flow discharge coefficient**

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

## **Valve 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 the orifice geometrical profile. You can find recommendations on the parameter value in hydraulic textbooks. The default value is 12.

## **Valve leakage area**

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12\text{m}^2$ .

## **Valve opening adjustment coefficient**

The coefficient controls how close the hyperbolic tangent function approximates the linear relationship between the orifice area and control pressure. See the block description for more information. The default value is 4.

## **Global Parameters**

Parameters determined by the type of working fluid:

- Fluid density



# Pressure Reducing 3-Way Valve

---

- **Fluid kinematic viscosity**

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

## **Ports**

The block has the following ports:

P

Hydraulic conserving port associated with the valve inlet.

A

Hydraulic conserving port associated with the valve outlet.

T

Hydraulic conserving port that connects with the tank.

## **See Also**

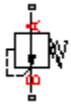
Pressure Reducing Valve

# Pressure Reducing Valve

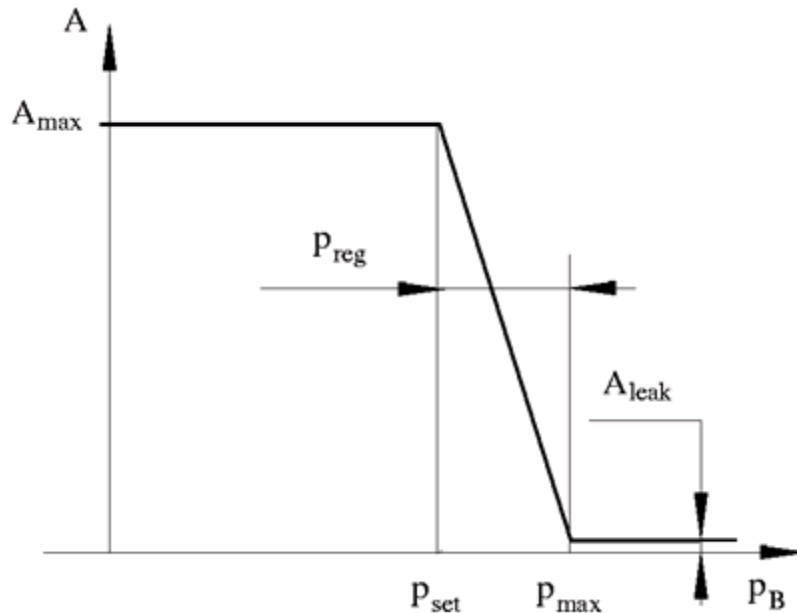
**Purpose** Pressure control valve maintaining reduced pressure in portion of system

**Library** Pressure Control Valves

## Description



The Pressure Reducing Valve block represents a hydraulic pressure-reducing valve as a data-sheet-based model. Pressure-reducing valves are used to maintain reduced pressure in a portion of a system. The following figure shows the typical dependency between the valve passage area  $A$  and the pressure  $p_B$  downstream from the valve.



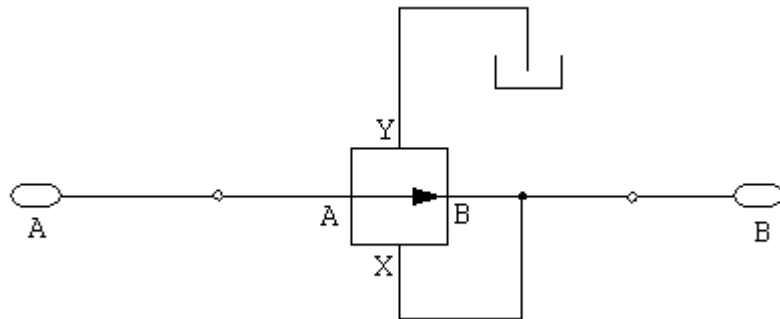
The pressure-reducing valve is a normally open valve and it remains fully open while outlet pressure is lower than the valve preset pressure. When the preset pressure is reached, the valve control member (spool, ball, poppet, etc.) is forced off its stop and starts closing the orifice, thus

# Pressure Reducing Valve

trying to maintain outlet pressure at preset level. Any further increase in the outlet pressure causes the control member to close the orifice even more until the point when the orifice is fully closed. The pressure increase that is necessary to close the valve is referred to as regulation range, and is generally provided in the catalogs, along with the valve maximum area. The valve maximum area and regulation range are the key parameters of the block.

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

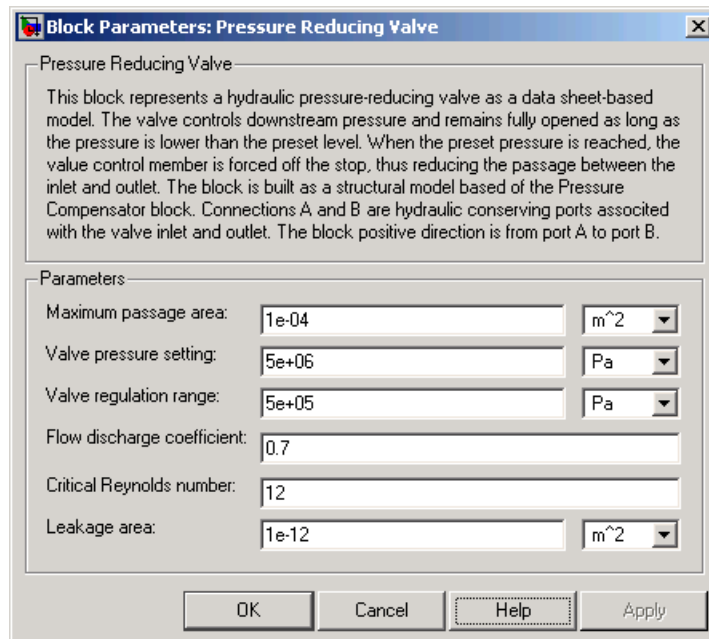
The block is built as a structural model based on the Pressure Compensator block, as shown in the following schematic.



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

# Pressure Reducing Valve

## Dialog Box and Parameters



### Maximum passage area

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

### Valve pressure setting

Preset pressure level, at which the orifice of the valve starts to close. The default value is  $5e6 \text{ Pa}$ .

### Valve regulation range

Pressure increase over the preset level needed to fully close the valve. Must be less than 0.2 of the **Valve pressure setting** parameter value. The default value is  $5e5 \text{ Pa}$ .

### Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the orifice, and

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 simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12m^2$ .

## **Global Parameters**

Parameters determined by the type of working fluid:

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

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

## **Ports**

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

# Pressure Reducing Valve

---

## **Examples**

The Power Unit with Pressure Reducing Valve example illustrates the use of the Pressure Reducing Valve block in hydraulic systems. The pressure reducing valve is set to  $20 \times 10^5$  Pa and maintains this pressure downstream, as long as the upstream pressure is higher than this setting.

## **See Also**

Pressure Compensator

Pressure Relief Valve

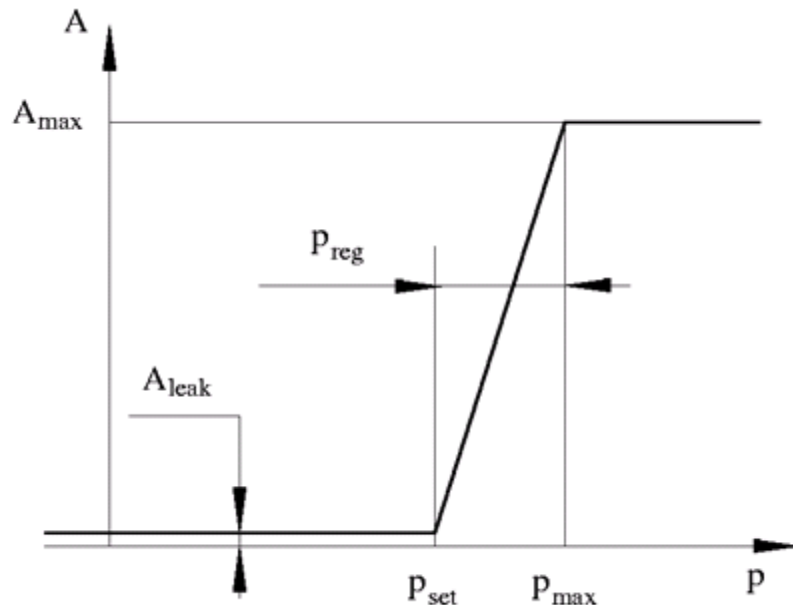
**Purpose** Pressure control valve maintaining preset pressure in system

**Library** Pressure Control Valves

## Description



The Pressure Relief Valve block represents a hydraulic pressure relief valve as a data-sheet-based model. The following figure shows the typical dependency between the valve passage area  $A$  and the pressure differential  $p$  across the valve.



The valve remains closed while pressure at the valve inlet is lower than the valve preset pressure. When the preset pressure is reached, the valve control member (spool, ball, poppet, etc.) is forced off its seat, thus creating a passage between the inlet and outlet. Some fluid is diverted to a tank through this orifice, thus reducing the pressure at the inlet. If

# Pressure Relief Valve

---

this flow rate is not enough and pressure continues to rise, the area is further increased until the control member reaches its maximum. At this moment, the maximum flow rate is passing through the valve. The value of a maximum flow rate and the pressure increase over the preset level to pass this flow rate are generally provided in the catalogs. The pressure increase over the preset level is frequently referred to as valve steady state error, or regulation range. The valve maximum area and regulation range are the key parameters of the block.

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

The model accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number ( $Re$ ) and comparing its value with the critical Reynolds number ( $Re_{cr}$ ). The flow rate is determined according to the following equations:

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

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

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



$$p = p_A - p_B$$

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

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

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

where

$q$	Flow rate through the valve
$p$	Pressure differential across the valve
$p_A, p_B$	Gauge pressures at the block terminals
$C_D$	Flow discharge coefficient
$A(p)$	Instantaneous orifice passage area
$A_{max}$	Fully open valve passage area
$A_{leak}$	Closed valve leakage area
$p_{reg}$	Regulation range
$p_{set}$	Valve preset pressure
$p_{max}$	Valve pressure at maximum opening
$D_H$	Instantaneous orifice hydraulic diameter
$\rho$	Fluid density
$\nu$	Fluid kinematic viscosity

# Pressure Relief Valve

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

## Basic Assumptions and Limitations

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

## Dialog Box and Parameters

**Block Parameters: Pressure Relief Valve**

Pressure Relief Valve

This block represents a hydraulic pressure relief valve as a data sheet-based model. The valve remains closed while pressure at the valve inlet is lower than the valve preset pressure. When the preset pressure is reached, the valve control member is forced off its seat, thus creating a passage between the inlet and outlet. Some fluid is diverted to a tank through this orifice, thus reducing the pressure at the inlet. If this flow rate is not enough and pressure continues to rise, the area is further increased until the control member reaches its maximum.

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

Parameters

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

OK Cancel Help Apply

**Maximum passage area**

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

**Valve pressure setting**

Preset pressure level, at which the orifice of the valve starts to open. The default value is  $50e5 \text{ Pa}$ .

**Valve regulation range**

Pressure increase over the preset level needed to fully open the valve. MathWorks recommends using values less than 0.2 of the **Valve pressure setting** parameter value. The default value is  $5e5 \text{ Pa}$ .

**Flow discharge coefficient**

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

**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 simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

# Pressure Relief Valve

---

## Global Parameters

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

## Examples

The Power Unit with Fixed-Displacement Pump example illustrates the use of the Pressure Relief Valve block in hydraulic systems. The valve is set to  $75e5$  Pa and starts diverting fluid to tank as soon as the pressure at its inlet reaches this value.

## See Also

Pressure Compensator

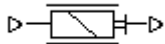
Pressure Reducing Valve

# Proportional and Servo-Valve Actuator

**Purpose** Continuous valve driver with output proportional to input signal

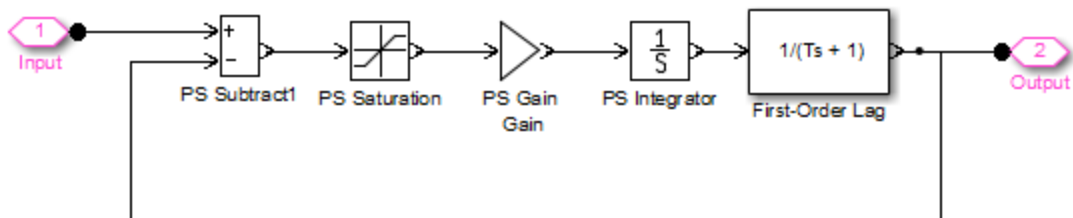
**Library** Valve Actuators

## Description

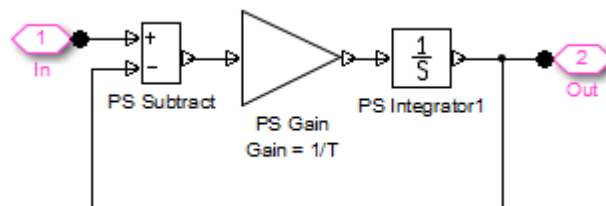


The Proportional and Servo-Valve Actuator block represents an electromagnetic actuator that is used in proportional and servo-valves to drive a spool or other working member. The block is intended to work with one of the directional valve models to form a desirable configuration of a proportional or servo-valve. The block is implemented as a data-sheet-based model and reproduces only the input/output relationship, or the actuator's transient response, as presented in the catalog or data sheet.

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



The model of the actuator is built of the Foundation Library Physical Signals blocks: PS Subtract, PS Saturation, PS Gain, PS Integrator. The first-order lag is arranged as a subsystem, as shown below,



# Proportional and Servo-Valve Actuator

---

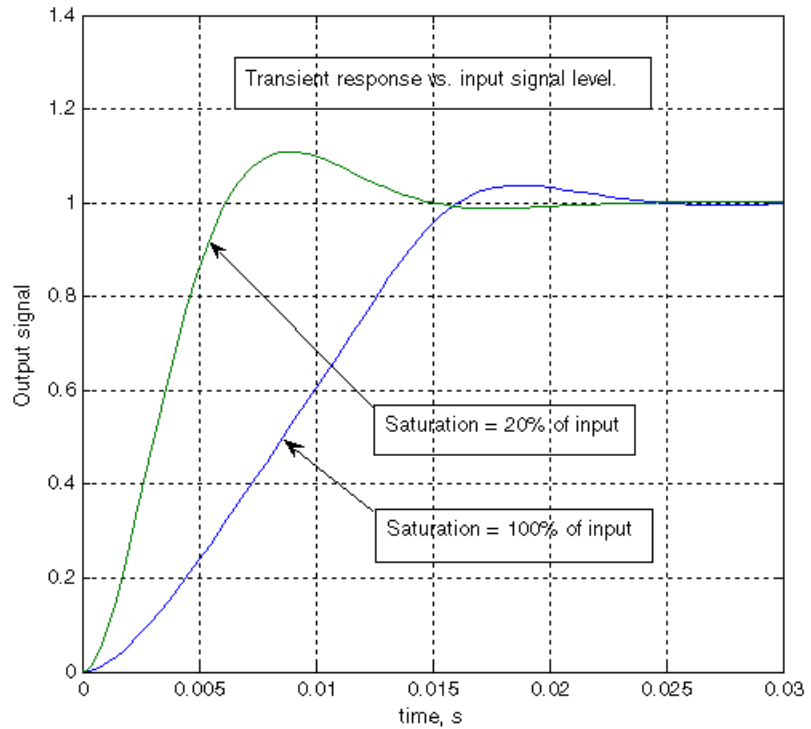
and is also built of the Physical Signals blocks: PS Subtract, PS Gain, and PS Integrator. The transfer function of the lag is

$$H(s) = \frac{1}{Ts + 1}$$

where  $T$  is the **Time constant** of the lag. The value of this parameter is set through the parameter **Gain** of the PS Gain block in the First-Order Lag subsystem by setting it equal to  $1/T$ .

The typical transient responses of a servo-valve or a high-quality proportional valve are shown in the following figure. The only difference between the two responses in the figure is the value of the saturation.

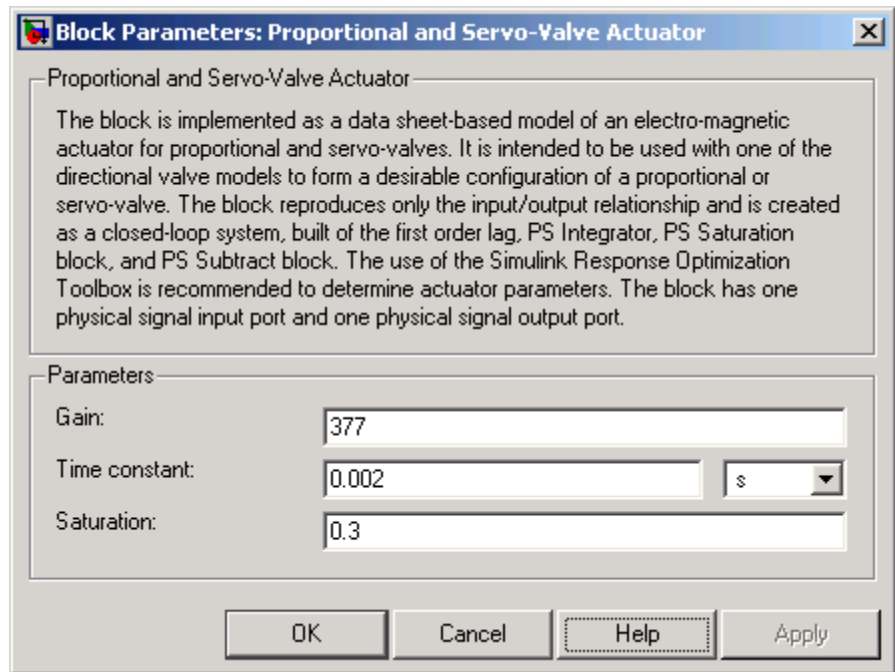
# Proportional and Servo-Valve Actuator



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

# Proportional and Servo-Valve Actuator

## Dialog Box and Parameters



### Gain

Gain of the PS Gain block in the top level of the block diagram of the actuator model. The default value is 377.

### Time constant

Time constant of the first-order lag. The **Gain** parameter of the PS Gain block in the First-Order Lag subsystem is equal to  $1/\text{Time constant}$ . The default value is 0.002 s.

### Saturation

Saturation level of the PS Saturation block in the actuator model. The default value is 0.3.

## Ports

The block has one physical signal input port and one physical signal output port.



# Proportional and Servo-Valve Actuator

---

## **Examples**

The Closed-Loop Electrohydraulic Actuator with Proportional Valve example illustrates the use of the Proportional and Servo-Valve Actuator block in hydraulic systems.

## **See Also**

2-Position Valve Actuator

3-Position Valve Actuator

Hydraulic Double-Acting Valve Actuator

Hydraulic Single-Acting Valve Actuator

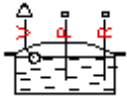
# Reservoir

---

**Purpose** Pressurized hydraulic reservoir

**Library** Hydraulic Utilities

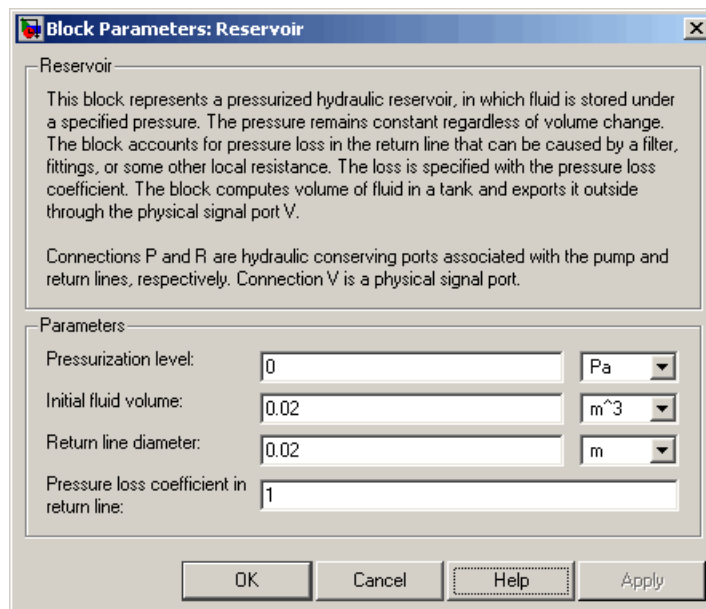
**Description**



The Reservoir block represents a pressurized hydraulic reservoir, in which fluid is stored under a specified pressure. The pressure remains constant regardless of volume change. The block accounts for pressure loss in the return line that can be caused by a filter, fittings, or some other local resistance. The loss is specified with the pressure loss coefficient. The block computes the volume of fluid in the tank and exports it outside through the physical signal port V.

The fluid volume value does not affect the results of simulation. It is introduced merely for information purposes. It is possible for the fluid volume to become negative during simulation, which signals that the fluid volume is not enough for the proper operation of the system. By viewing the results of the simulation, you can determine the extent of the fluid shortage.

## Dialog Box and Parameters



### Pressurization level

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

### Initial fluid volume

The initial volume of fluid in the tank. The default value is 0.02 m<sup>3</sup>.

### Return line diameter

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

### Pressure loss coefficient in return line

The value of the pressure loss coefficient, to account for pressure loss in the return line. This parameter must be greater than zero. The default value is 1.

For reasons of computational robustness, the loss is computed with the equation similar to that used in the Fixed Orifice block:

# Reservoir

---

$$q = \frac{1}{\sqrt{K}} A_p \sqrt{\frac{2}{\rho} P_{loss}}$$

The Critical Reynolds number is set to 15.

## Ports

The block has the following ports:

P

Hydraulic conserving port associated with the pump line.

R

Hydraulic conserving port associated with the return line.

V

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

## See Also

Constant Head Tank

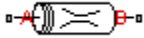
Hydraulic Reference

Variable Head Tank

**Purpose** Hydraulic pipeline which accounts for friction losses and port elevations

**Library** Low-Pressure Blocks

## Description



The Resistive Pipe LP block models hydraulic pipelines with circular and noncircular cross sections and accounts for resistive property only. In other words, the block is developed with the basic assumption of the steady state fluid momentum conditions. Neither fluid compressibility nor fluid inertia is considered in the model, meaning that features such as water hammer cannot be investigated. If necessary, you can add fluid compressibility, fluid inertia, and other effects to your model using other blocks, thus producing a more comprehensive model.

The end effects are also not considered, assuming that the flow is fully developed along the entire pipe length. To account for local resistances, such as bends, fittings, inlet and outlet losses, and so on, convert the resistances into their equivalent lengths, and then sum up all the resistances to obtain their aggregate length. Then add this length to the pipe geometrical length.

Pressure loss due to friction is computed with the Darcy equation, in which losses are proportional to the flow regime-dependable friction factor and the square of the flow rate. The friction factor in turbulent regime is determined with the Haaland approximation (see [1]). The friction factor during transition from laminar to turbulent regimes is determined with the linear interpolation between extreme points of the regimes. As a result of these assumptions, the tube is simulated according to the following equations:

$$p = f \frac{(L + L_{eq})}{D_H} \frac{\rho}{2A^2} |q| q + \rho g (z_B - z_A)$$

# Resistive Pipe LP

$$f = \begin{cases} K_s / Re & \text{for } Re \leq Re_L \\ f_L + \frac{f_T - f_L}{Re_T - Re_L} (Re - Re_L) & \text{for } Re_L < Re < Re_T \\ \frac{1}{\left( -1.8 \log_{10} \left( \frac{6.9}{Re} + \left( \frac{r/D_H}{3.7} \right)^{1.11} \right) \right)^2} & \text{for } Re \geq Re_T \end{cases}$$

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

where

- $p$  Pressure loss along the pipe due to friction
- $q$  Flow rate through the pipe
- $Re$  Reynolds number
- $Re_L$  Maximum Reynolds number at laminar flow
- $Re_T$  Minimum Reynolds number at turbulent flow
- $K_s$  Shape factor that characterizes the pipe cross section
- $f_L$  Friction factor at laminar border
- $f_T$  Friction factor at turbulent border
- $A$  Pipe cross-sectional area
- $D_H$  Pipe hydraulic diameter
- $L$  Pipe geometrical length
- $L_{eq}$  Aggregate equivalent length of local resistances
- $r$  Height of the roughness on the pipe internal surface

- $\nu$  Fluid kinematic viscosity
- $z_A, z_B$  Elevations of the pipe port A and port B, respectively
- $g$  Gravity acceleration

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

## Basic Assumptions and Limitations

- Flow is assumed to be fully developed along the pipe length.
- Fluid inertia, fluid compressibility, and wall compliance are not taken into account.

## Dialog Box and Parameters

The block dialog box contains two tabs:

- “Basic Parameters” on page 1-530
- “Vertical Position” on page 1-533

# Resistive Pipe LP

## Basic Parameters

**Block Parameters: Resistive Pipe LP**

Resistive Pipe LP

This block models hydraulic pipe with circular and noncircular cross sections and accounts for resistive property only. The block is intended to be used for low-pressure system simulation and, for this reason, requires elevation of both ports to be specified. To reduce model complexity, the block can be used to simulate not only a pipe itself, but also a combination of pipes and local resistances such as bends, fittings, inlet and outlet losses, etc., associated with the pipe. The resistances must be converted into their equivalent lengths, and then the total length of all the resistances is added to the pipe geometrical length. 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

Basic parameters | Vertical position

Pipe cross section type: Circular

Internal diameter: 0.01 m

Geometrical shape factor: 64

Pipe length: 5 m

Aggregate equivalent length of local resistances: 1 m

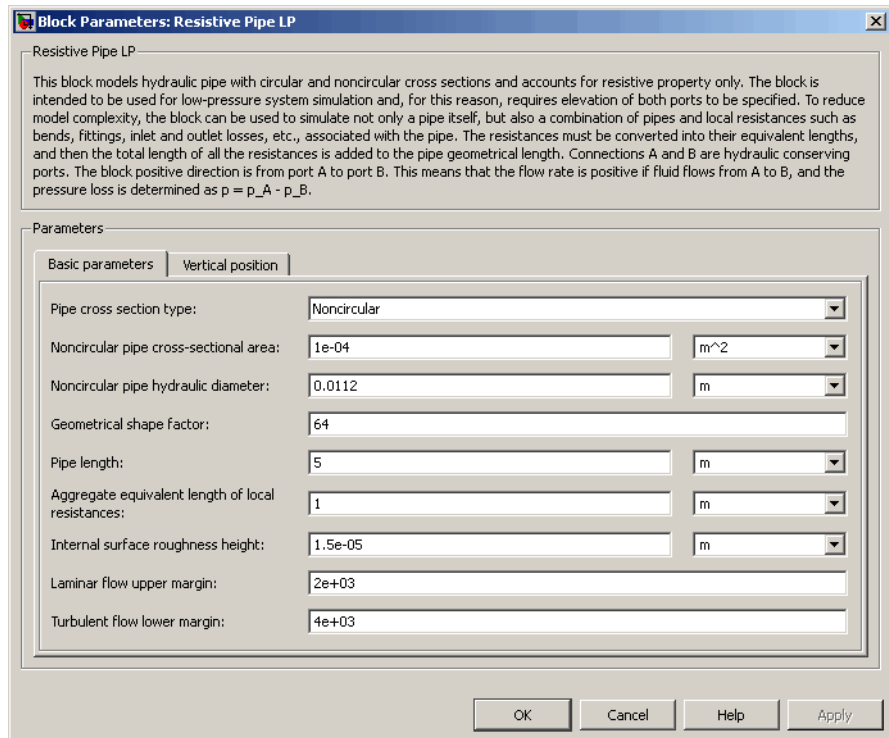
Internal surface roughness height: 1.5e-05 m

Laminar flow upper margin: 2e+03

Turbulent flow lower margin: 4e+03

OK Cancel Help Apply





## Pipe cross section type

The type of pipe cross section: **Circular** or **Noncircular**. For a circular pipe, you specify its internal diameter. For a noncircular pipe, you specify its hydraulic diameter and pipe cross-sectional area. The default value of the parameter is **Circular**.

## Internal diameter

Pipe internal diameter. The parameter is used if **Pipe cross section type** is set to **Circular**. The default value is 0.01 m.

## Noncircular pipe cross-sectional area

Pipe cross-sectional area. The parameter is used if **Pipe cross section type** is set to **Noncircular**. The default value is  $1e-4 \text{ m}^2$ .

# Resistive Pipe LP

---

## **Noncircular pipe hydraulic diameter**

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

## **Geometrical shape factor**

Used for computing friction factor at laminar flow. The shape of the pipe cross section determines the value. For a pipe with a noncircular cross section, set the factor to an appropriate value, for example, 56 for a square, 96 for concentric annulus, 62 for rectangle (2:1), and so on [1]. The default value is 64, which corresponds to a pipe with a circular cross section.

## **Pipe length**

Pipe geometrical length. The default value is 5 m.

## **Aggregate equivalent length of local resistances**

This parameter represents total equivalent length of all local resistances associated with the pipe. You can account for the pressure loss caused by local resistances, such as bends, fittings, armature, inlet/outlet losses, and so on, by adding to the pipe geometrical length an aggregate equivalent length of all the local resistances. The default value is 1 m.

## **Internal surface roughness height**

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

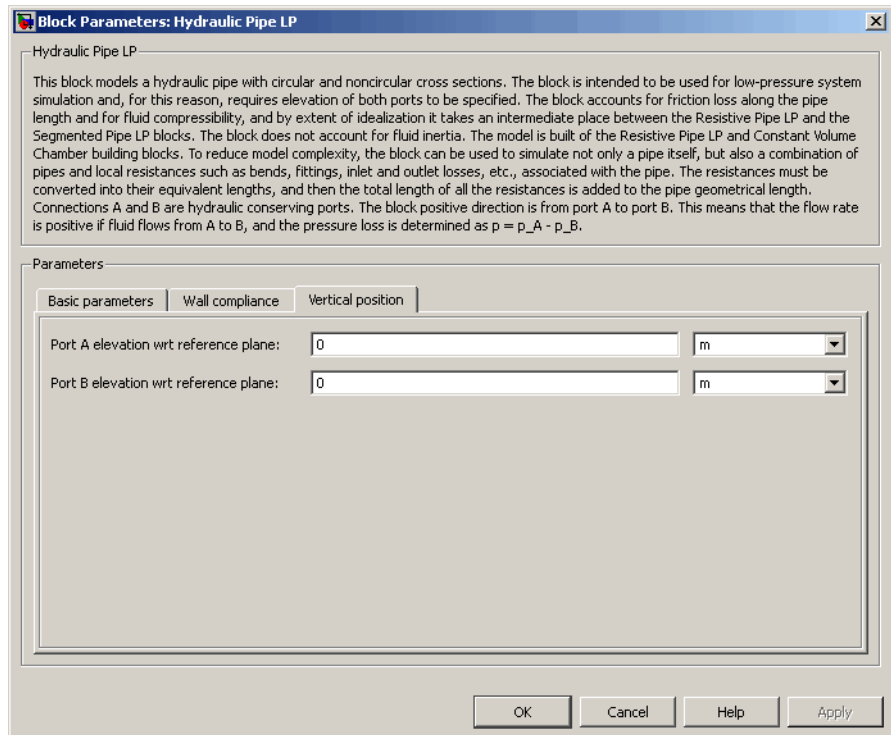
## **Laminar flow upper margin**

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

## **Turbulent flow lower margin**

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

## Vertical Position



### Port A elevation wrt reference plane

The parameter specifies vertical position of the pipe port A with respect to the reference plane. The default value is 0.

### Port B elevation wrt reference plane

The parameter specifies vertical position of the pipe port B with respect to the reference plane. The default value is 0.

### Restricted Parameters

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

# Resistive Pipe LP

---

- **Pipe cross section type**

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

## Global Parameters

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the pipe inlet.

B

Hydraulic conserving port associated with the pipe outlet.

## References

[1] White, F.M., *Viscous Fluid Flow*, McGraw-Hill, 1991

## See Also

Hydraulic Pipeline

Hydraulic Pipe LP

Hydraulic Pipe LP with Variable Elevation

Hydraulic Resistive Tube

Linear Hydraulic Resistance

Resistive Pipe LP with Variable Elevation

Segmented Pipeline

Segmented Pipe LP

# Resistive Pipe LP with Variable Elevation

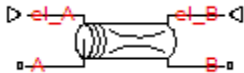
## Purpose

Hydraulic pipeline which accounts for friction losses and variable port elevations

## Library

Low-Pressure Blocks

## Description



The Resistive Pipe LP with Variable Elevation block models hydraulic pipelines with circular and noncircular cross sections and accounts for resistive property only. Use this block for low-pressure system simulation in which the pipe ends change their positions with respect to the reference plane. The elevations are provided through respective physical signal inputs.

To reduce model complexity, you can use this block to simulate not only a pipe itself, but also a combination of pipes and local resistances such as bends, fittings, inlet and outlet losses, associated with the pipe. You must convert the resistances into their equivalent lengths, and then sum up all the resistances to obtain their aggregate length. Then add this length to the pipe geometrical length.

Pressure loss due to friction is computed with the Darcy equation, in which losses are proportional to the flow regime-dependable friction factor and the square of the flow rate. The friction factor in turbulent regime is determined with the Haaland approximation (see [1]). The friction factor during transition from laminar to turbulent regimes is determined with the linear interpolation between extreme points of the regimes. As a result of these assumptions, the tube is simulated according to the following equations:

$$p = f \frac{(L + L_{eq})}{D_H} \frac{\rho}{2A^2} |q| q + \rho g (z_B - z_A)$$

# Resistive Pipe LP with Variable Elevation

---

$$f = \begin{cases} K_s / Re & \text{for } Re \leq Re_L \\ f_L + \frac{f_T - f_L}{Re_T - Re_L} (Re - Re_L) & \text{for } Re_L < Re < Re_T \\ \frac{1}{\left( -1.8 \log_{10} \left( \frac{6.9}{Re} + \left( \frac{r/D_H}{3.7} \right)^{1.11} \right) \right)^2} & \text{for } Re \geq Re_T \end{cases}$$

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

where

- $p$  Pressure loss along the pipe due to friction
- $q$  Flow rate through the pipe
- $Re$  Reynolds number
- $Re_L$  Maximum Reynolds number at laminar flow
- $Re_T$  Minimum Reynolds number at turbulent flow
- $K_s$  Shape factor that characterizes the pipe cross section
- $f_L$  Friction factor at laminar border
- $f_T$  Friction factor at turbulent border
- $A$  Pipe cross-sectional area
- $D_H$  Pipe hydraulic diameter
- $L$  Pipe geometrical length
- $L_{eq}$  Aggregate equivalent length of local resistances
- $r$  Height of the roughness on the pipe internal surface

# Resistive Pipe LP with Variable Elevation

---

- $\nu$  Fluid kinematic viscosity
- $z_A, z_B$  Elevations of the pipe port A and port B, respectively
- $g$  Gravity acceleration

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

## Basic Assumptions and Limitations

- Flow is assumed to be fully developed along the pipe length.
- Fluid inertia, fluid compressibility, and wall compliance are not taken into account.

# Resistive Pipe LP with Variable Elevation

## Dialog Box and Parameters

**Block Parameters: Resistive Pipe LP with Variable Elevation**

Resistive Pipe LP with Variable Elevation

This block models hydraulic pipe with circular and noncircular cross sections and accounts for resistive property only. The block is intended to be used for low-pressure system simulation in which pipe ends change their positions with respect to the reference plane. The elevations are provided through respective physical signal inputs. To reduce model complexity, the block can be used to simulate not only a pipe itself, but also a combination of pipes and local resistances such as bends, fittings, inlet and outlet losses, etc., associated with the pipe. The resistances must be converted into their equivalent lengths, and then the total length of all the resistances is added to the pipe geometrical length. Connections A and B are hydraulic conserving ports. Connections el\_A and el\_B are the physical signal 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

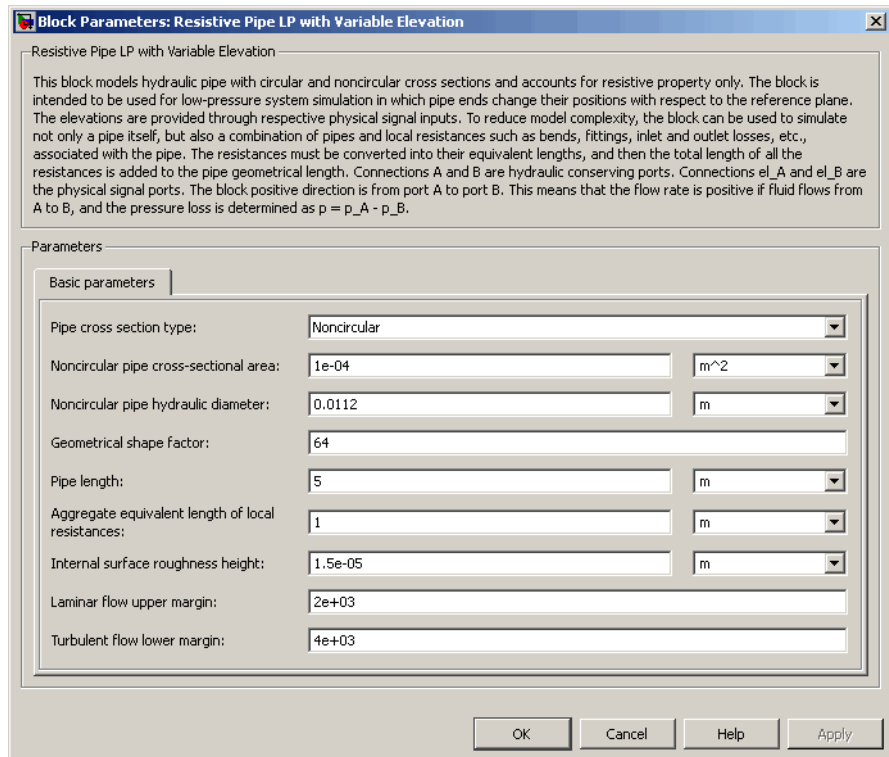
Basic parameters

Pipe cross section type:	Circular	
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	

OK Cancel Help Apply



# Resistive Pipe LP with Variable Elevation



## Pipe cross section type

The type of pipe cross section: Circular or Noncircular. For a circular pipe, you specify its internal diameter. For a noncircular pipe, you specify its hydraulic diameter and pipe cross-sectional area. The default value of the parameter is Circular.

## Internal diameter

Pipe internal diameter. The parameter is used if **Pipe cross section type** is set to Circular. The default value is 0.01 m.

# Resistive Pipe LP with Variable Elevation

---

## **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<sup>2</sup>.

## **Noncircular pipe hydraulic diameter**

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

## **Geometrical shape factor**

Used for computing friction factor at laminar flow. The shape of the pipe cross section determines the value. For a pipe with a noncircular cross section, set the factor to an appropriate value, for example, 56 for a square, 96 for concentric annulus, 62 for rectangle (2:1), and so on [1]. The default value is 64, which corresponds to a pipe with a circular cross section.

## **Pipe length**

Pipe geometrical length. The default value is 5 m.

## **Aggregate equivalent length of local resistances**

This parameter represents total equivalent length of all local resistances associated with the pipe. You can account for the pressure loss caused by local resistances, such as bends, fittings, armature, inlet/outlet losses, and so on, by adding to the pipe geometrical length an aggregate equivalent length of all the local resistances. The default value is 1 m.

## **Internal surface roughness height**

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

## **Laminar flow upper margin**

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

# Resistive Pipe LP with Variable Elevation

---

## 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.

## Global Parameters

Parameters determined by the type of working fluid:

- Fluid density
- Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the pipe inlet.

B

Hydraulic conserving port associated with the pipe outlet.

e1\_A

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

e1\_B

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

## References

[1] White, F.M., *Viscous Fluid Flow*, McGraw-Hill, 1991

## See Also

Hydraulic Pipeline

Hydraulic Pipe LP

Hydraulic Pipe LP with Variable Elevation

Hydraulic Resistive Tube

Linear Hydraulic Resistance

# Resistive Pipe LP with Variable Elevation

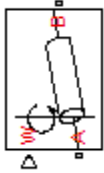
---

Resistive Pipe LP  
Segmented Pipeline  
Segmented Pipe LP

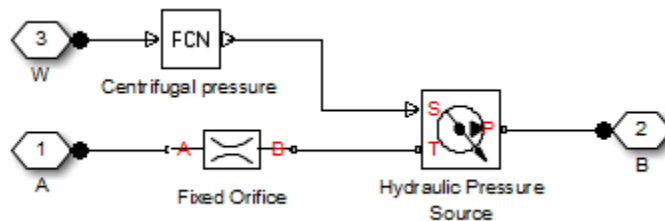
**Purpose** Hydraulic pipeline created by bore in rotating housing

**Library** Pipelines

## Description



The Rotating Pipe block models a hydraulic pipeline represented as a short bore drilled through a shaft at some angle to its axis, or as a short pipe that connects actuator or bearings, mounted on the shaft, with the channel inside the shaft. Such connections are typical for lubrication systems and for hydraulically-operated clutches and mechanisms. The shaft angular velocity is assumed to be constant or changing at very low speed. The block is implemented as a structural model, shown in the diagram below:



The Fixed Orifice block simulates pressure losses in the bore. The pipe is assumed to be short enough to be characterized by a flow-pressure relationship similar to that of the fixed orifice. The model accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number ( $Re$ ) and comparing its value with the critical Reynolds number ( $Re_{cr}$ ). The Hydraulic Pressure Source block represents pumping action developed by a rotating pipe, which is calculated with the equation:

$$p = \rho \cdot \frac{\omega^2 \cdot r_B^2}{2}$$

where

# Rotating Pipe

$p$	Pressure at the pipe outlet
$\rho$	Fluid density
$\omega$	Shaft angular velocity
$r_B$	Radius of the pipe outlet

Connections A and B are conserving hydraulic ports associated with the pipe inlet and outlet, respectively. Connection W is the physical signal input port that provides the shaft angular velocity value.

## Basic Assumptions and Limitations

- The pipe is assumed to be short, to be simulated with the fixed orifice.
- Flow is assumed to be fully developed along the pipe length.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at  $Re=Re_{cr}$ .
- Fluid inertia is not taken into account.
- Fluid compressibility is not taken into account.

## Dialog Box and Parameters

**Block Parameters: Rotating Pipe**

Rotating Pipe

The block represents a short hole drilled through a shaft or a short pipe that connects mounted on the shaft actuator or bearings with the channel inside the shaft. Such connections are typical for lubrication systems and for hydraulically-operated clutches and mechanisms. The shaft angular velocity is assumed to be constant. The pipe is assumed to be short enough to be characterized by a flow-pressure relationship similar to that of the fixed orifice with pressure differential being determined as a sum of static pressure drop and centrifugal pressure. 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 pipe 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

Pipe cross-section area:	<input type="text" value="1e-04"/>	<input type="text" value="m^2"/>
Port B radius of rotation:	<input type="text" value="0.03"/>	<input type="text" value="m"/>
Flow discharge coefficient:	<input type="text" value="0.6"/>	
Critical Reynolds number:	<input type="text" value="12"/>	

OK Cancel Help Apply

**Pipe cross-section area**

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

**Port B radius of rotation**

Radius of rotation of the outer end of the pipe. The default value is 0.03 m.

**Flow discharge coefficient**

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

**Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on pipe 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**

Parameters determined by the type of working fluid:

- Fluid density
- Fluid kinematic viscosity

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

**Examples**

For an example of using the Rotating Pipe block to model a custom single-acting hydraulic cylinder, see the Centrifugal Force in Rotating Cylinder block reference page.

**Ports**

The block has the following ports:

# Rotating Pipe

---

- A Hydraulic conserving port associated with the pipe inlet.
- B Hydraulic conserving port associated with the pipe outlet.
- W Physical signal port providing the angular velocity value.

**See Also** Centrifugal Force in Rotating Cylinder

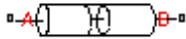


# Segmented Pipeline

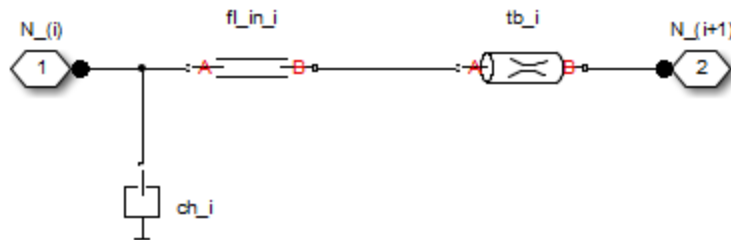
**Purpose** Hydraulic pipeline with resistive, fluid inertia, and fluid compressibility properties

**Library** Pipelines

**Description** The Segmented Pipeline block models hydraulic pipelines with circular cross sections. Hydraulic pipelines, which are inherently distributed parameter elements, are represented with sets of identical, connected in series, lumped parameter segments. It is assumed that the larger the number of segments, the closer the lumped parameter model becomes to its distributed parameter counterpart. The equivalent circuit of a pipeline adopted in the block is shown below, along with the segment configuration.



## Pipeline Equivalent Circuit



## Segment Configuration

The model contains as many Constant Volume Hydraulic Chamber blocks as there are segments. The chamber lumps fluid volume equal to

# Segmented Pipeline

---

$$V = \frac{\pi d^2 L}{4 N}$$

where

- $V$  Fluid volume
- $d$  Pipe diameter
- $L$  Pipe length
- $N$  Number of segments

The Constant Volume Hydraulic Chamber block is placed between two branches, each consisting of a Hydraulic Resistive Tube block and a Fluid Inertia block. Every Hydraulic Resistive Tube block lumps  $(L+L_{ad}) / (N+1)$ -th portion of the pipe length, while Fluid Inertia block has  $L / (N+1)$  length ( $L_{ad}$  denotes additional pipe length equal to aggregate equivalent length of pipe local resistances, such as fitting, elbows, bends, and so on).

The nodes to which Constant Volume Hydraulic Chamber blocks are connected are assigned names  $N_1, N_2, \dots, N_n$  ( $n$  is the number of segments). Pressures at these nodes are assumed to be equal to average pressure of the segment. Intermediate nodes between Hydraulic Resistive Tube and Fluid Inertia blocks are assigned names  $nn_0, nn_1, nn_2, \dots, nn_n$ . The Constant Volume Hydraulic Chamber blocks are named  $ch_1, ch_2, \dots, ch_n$ , Hydraulic Resistive Tube blocks are named  $tb_0, tb_1, tb_2, \dots, tb_n$ , and Fluid Inertia blocks are named  $fl\_in_0, fl\_in_1, fl\_in_2, \dots, fl\_in_n$ .

The number of segments 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:

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

where

- $N$  Number of segments
- $L$  Pipe length
- $c$  Speed of sound in the fluid
- $\omega$  Maximum frequency to be observed in the pipe response

The table below contains an example of simulation of a pipeline where the first four true eigenfrequencies are 89.1 Hz, 267 Hz, 446 Hz, and 624 Hz.

Number of Segments	1st Mode	2nd Mode	3rd Mode	4th Mode
1	112.3	–	–	–
2	107.2	271.8	–	–
4	97.7	284.4	432.9	689
8	93.2	271.9	435.5	628

As you can see, the error is less than 5% if an eight-segmented version is used.

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

# Segmented Pipeline

---

## **Basic Assumptions and Limitations**

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

## Dialog Box and Parameters

**Block Parameters: Segmented Pipeline**

**Segmented Pipeline**

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 distributed 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  $p = p_A - p_B$ .

**Parameters**

Pipe internal diameter:	<input type="text" value="0.01"/>	<input type="text" value="m"/>
Pipe length:	<input type="text" value="5"/>	<input type="text" value="m"/>
Number of segments:	<input type="text" value="1"/>	
Aggregate equivalent length of local resistances:	<input type="text" value="1"/>	<input type="text" value="m"/>
Internal surface roughness height:	<input type="text" value="1.5e-5"/>	<input type="text" value="m"/>
Laminar flow upper margin:	<input type="text" value="2e+3"/>	
Turbulent flow lower margin:	<input type="text" value="4e+3"/>	
Pipe wall type:	<input type="text" value="Rigid"/>	
Specific heat ratio:	<input type="text" value="1.4"/>	
Initial pressures at model nodes:	<input type="text" value="The same initial pressure for all nodes"/>	
Initial pressure:	<input type="text" value="0"/>	<input type="text" value="Pa"/>
Initial flow rate:	<input type="text" value="0"/>	<input type="text" value="m^3/s"/>

# Segmented Pipeline

---

## **Pipe internal diameter**

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

## **Pipe length**

Pipe geometrical length. The default value is 5 m.

## **Number of segments**

Number of lumped parameter segments in the pipeline model. The default value is 1.

## **Aggregate equivalent length of local resistances**

This parameter represents total equivalent length of all local resistances associated with the pipe. You can account for the pressure loss caused by local resistances, such as bends, fittings, armature, inlet/outlet losses, and so on, by adding to the pipe geometrical length an aggregate equivalent length of all the local resistances. This length is added to the geometrical pipe length only for hydraulic resistance computation. Both the fluid volume and fluid inertia are determined based on pipe geometrical length only. The default value is 1 m.

## **Internal surface roughness height**

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

## **Laminar flow upper margin**

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

## **Turbulent flow lower margin**

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

## Pipe wall type

The parameter can have one of two values: **Rigid** or **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 Hydraulic Chamber block. The default value is  $1.4$ .

## Initial pressures at model nodes

Lets you specify the initial condition for pressure inside the pipe segments. The parameter can have one of two values:

- **The same initial pressure for all nodes** — The initial pressure in all pipe segments is the same, and is specified by the **Initial pressure** parameter value. This is the default.
- **Custom** — Lets you specify initial pressure individually for each pipe segment, by using the **Initial pressure vector** parameter. The vector size must be equal to the number of pipe segments, defined by the **Number of segments** parameter value.

# Segmented Pipeline

---

## **Initial pressure**

Specifies the initial pressure in all pipe segments. The parameter is used if the **Initial pressures at model nodes** parameter is set to The same initial pressure for all nodes, and the default value is 0.

## **Initial pressure vector**

Lets you specify initial pressure individually for each pipe segment. The parameter is used if the **Initial pressures at model nodes** parameter is set to Custom. The vector size must be equal to the number of pipe segments, defined by the **Number of segments** parameter value.

## **Initial flow rate**

Specifies the initial flow rate through the pipe. The default value is 0.

## **Restricted Parameters**

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

- **Pipe wall type**
- **Initial pressures at model nodes**

All other block parameters are available for modification.

## **Global Parameters**

Parameters determined by the type of working fluid:

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

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

## **Ports**

The block has the following ports:



A Hydraulic conserving port associated with the pipe inlet.

B Hydraulic conserving port associated with the pipe outlet.

## **See Also**

Hydraulic Pipeline

Linear Hydraulic Resistance

Hydraulic Resistive Tube

# Segmented Pipe LP

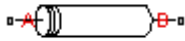
## Purpose

Hydraulic pipeline with resistive, fluid inertia, fluid compressibility, and elevation properties

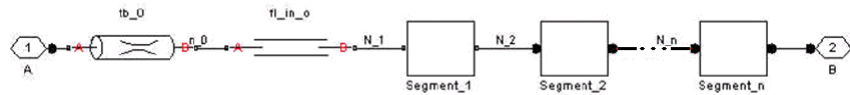
## Library

Low-Pressure Blocks

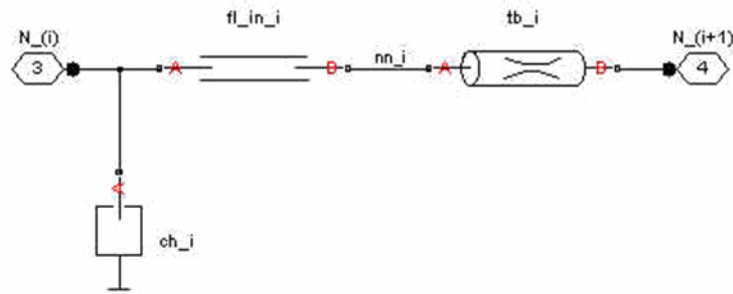
## Description



The Segmented Pipe LP block models hydraulic pipelines with circular cross sections. Hydraulic pipelines, which are inherently distributed parameter elements, are represented with sets of identical, connected in series, lumped parameter segments. It is assumed that the larger the number of segments, the closer the lumped parameter model becomes to its distributed parameter counterpart. The equivalent circuit of a pipeline adopted in the block is shown below, along with the segment configuration.



## Pipeline Equivalent Circuit



## Segment Configuration

The model contains as many Constant Volume Hydraulic Chamber blocks as there are segments. The chamber lumps fluid volume equal to

$$V = \frac{\pi d^2 L}{4 N}$$

where

- $V$  Fluid volume
- $d$  Pipe diameter
- $L$  Pipe length
- $N$  Number of segments

The Constant Volume Hydraulic Chamber block is placed between two branches, each consisting of a Hydraulic Resistive Tube block and a Fluid Inertia block. Every Hydraulic Resistive Tube block lumps  $(L+L_{ad}) / (N+1)$ -th portion of the pipe length, while Fluid Inertia block has  $L / (N+1)$  length ( $L_{ad}$  denotes additional pipe length equal to aggregate equivalent length of pipe local resistances, such as fitting, elbows, bends, and so on).

The nodes to which Constant Volume Hydraulic Chamber blocks are connected are assigned names  $N_1, N_2, \dots, N_n$  ( $n$  is the number of segments). Pressures at these nodes are assumed to be equal to average pressure of the segment. Intermediate nodes between Hydraulic Resistive Tube and Fluid Inertia blocks are assigned names  $nn_0, nn_1, nn_2, \dots, nn_n$ . The Constant Volume Hydraulic Chamber blocks are named  $ch_1, ch_2, \dots, ch_n$ , Hydraulic Resistive Tube blocks are named  $tb_0, tb_1, tb_2, \dots, tb_n$ , and Fluid Inertia blocks are named  $fl\_in_0, fl\_in_1, fl\_in_2, \dots, fl\_in_n$ .

The number of segments 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

# Segmented Pipe LP

---

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

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

where

- $N$  Number of segments
- $L$  Pipe length
- $c$  Speed of sound in the fluid
- $\omega$  Maximum frequency to be observed in the pipe response

The table below contains an example of simulation of a pipeline where the first four true eigenfrequencies are 89.1 Hz, 267 Hz, 446 Hz, and 624 Hz.

Number of Segments	1st Mode	2nd Mode	3rd Mode	4th Mode
1	112.3	–	–	–
2	107.2	271.8	–	–
4	97.7	284.4	432.9	689
8	93.2	271.9	435.5	628

As you can see, the error is less than 5% if an eight-segmented version is used.

The difference in elevation between ports A and B is distributed evenly between pipe segments.

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

## Basic Assumptions and Limitations

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

## Dialog Box and Parameters

The block dialog box contains three tabs:

- “Basic Parameters” on page 1-559
- “Wall Compliance” on page 1-561
- “Vertical Position” on page 1-564

## Basic Parameters

**Block Parameters: Segmented Pipe LP**

Segmented Pipe LP

This block represents hydraulic pipes 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 long hydraulic pipes, within lumped parameter simulator. The larger the number of segments, the closer the lumped parameter model becomes to its distributed parameter counterpart. Each segment consists of Resistive Pipe LP, Fluid Inertia, and Constant Volume Chamber building blocks. As a result, the block accounts for friction loss, fluid inertia, and fluid compressibility. The block is intended to be used for low-pressure system simulation and, for this reason, requires elevation of both ports to be specified. To reduce system model complexity, the block can be used to simulate not only a pipe itself, but also a combination of pipes and local resistances such as bends, fittings, inlet and outlet losses, etc., associated with the pipe. The resistances must be converted into their equivalent lengths, and then the total length of all the resistances is added to the pipe geometrical length. 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

Basic parameters | Wall compliance | Vertical position

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

OK Cancel Help Apply

# Segmented Pipe LP

---

## **Pipe internal diameter**

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

## **Pipe length**

Pipe geometrical length. The default value is 5 m.

## **Number of segments**

Number of lumped parameter segments in the pipeline model. The default value is 1.

## **Aggregate equivalent length of local resistances**

This parameter represents total equivalent length of all local resistances associated with the pipe. You can account for the pressure loss caused by local resistances, such as bends, fittings, armature, inlet/outlet losses, and so on, by adding to the pipe geometrical length an aggregate equivalent length of all the local resistances. This length is added to the geometrical pipe length only for hydraulic resistance computation. Both the fluid volume and fluid inertia are determined based on pipe geometrical length only. The default value is 1 m.

## **Internal surface roughness height**

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

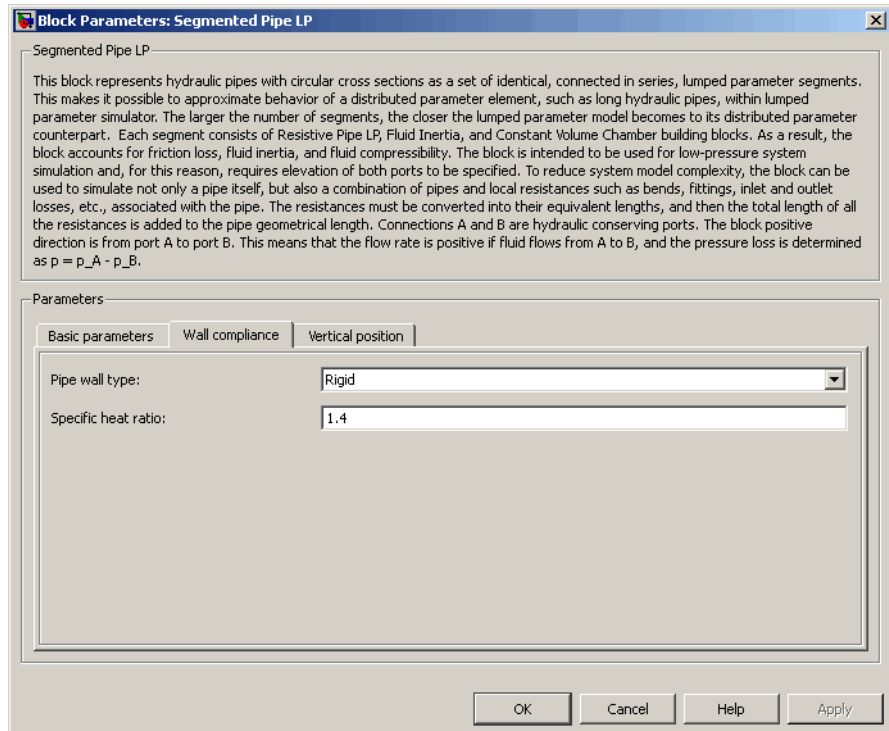
## **Laminar flow upper margin**

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

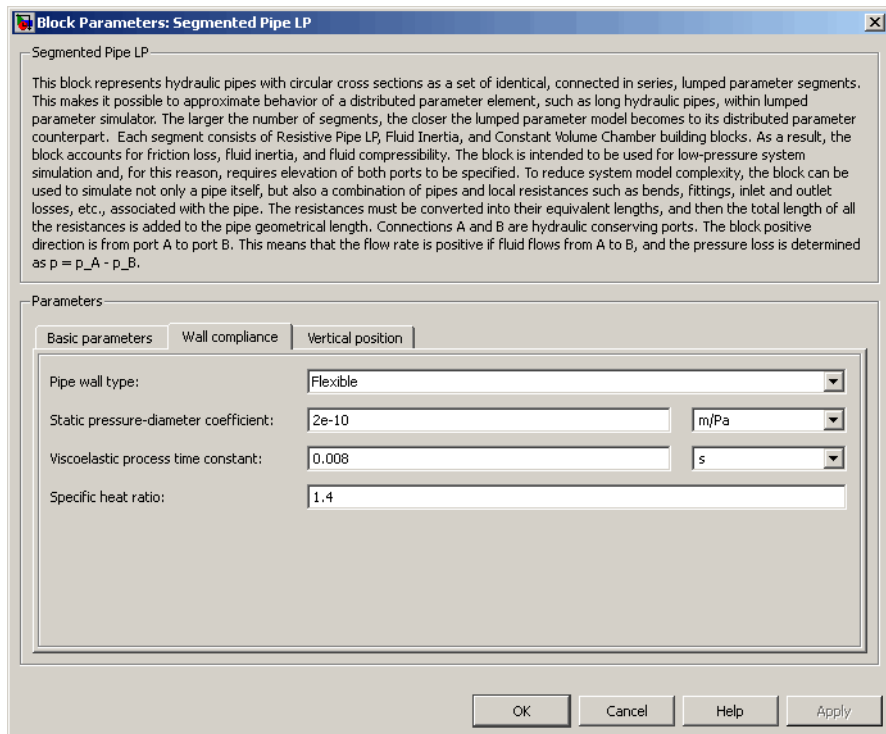
## **Turbulent flow lower margin**

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

## Wall Compliance



# Segmented Pipe LP



## Pipe wall type

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

## Static pressure-diameter coefficient

Coefficient that establishes relationship between the pressure and the internal diameter at steady-state conditions. This coefficient can be determined analytically for cylindrical metal pipes or experimentally for hoses. The parameter is used if the **Pipe wall**



**type** parameter is set to **Flexible**. The default value is  $2e-12$  m/Pa.

### **Viscoelastic process time constant**

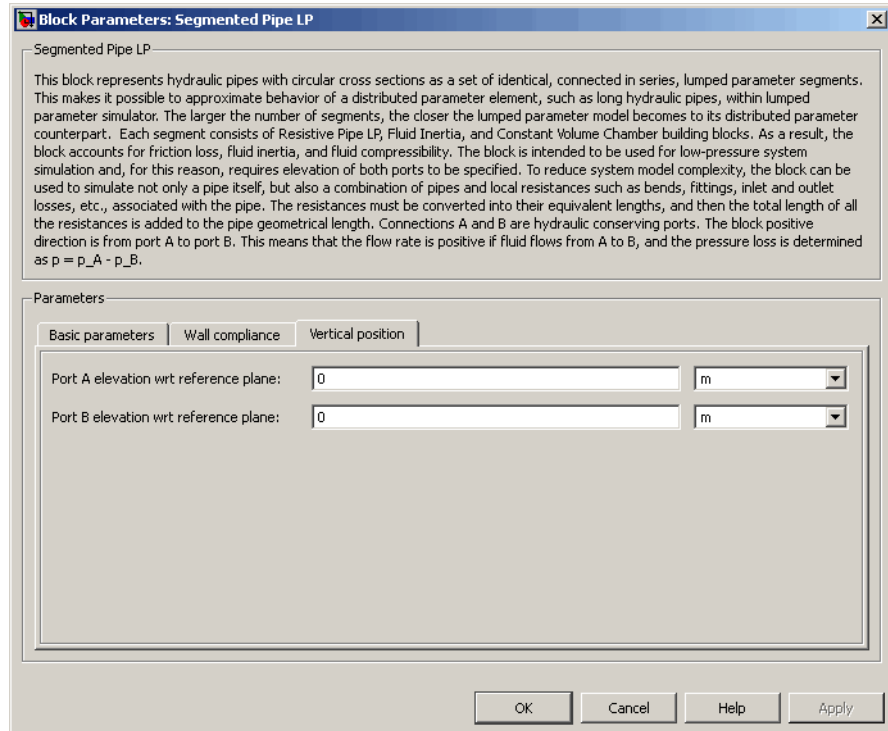
Time constant in the transfer function that relates pipe internal diameter to pressure variations. By using this parameter, the simulated elastic or viscoelastic process is approximated with the first-order lag. The value is determined experimentally or provided by the manufacturer. The parameter is used if the **Pipe wall type** parameter is set to **Flexible**. The default value is 0.01 s.

### **Specific heat ratio**

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

# Segmented Pipe LP

## Vertical Position



### Port A elevation wrt reference plane

The parameter specifies vertical position of the pipe port A with respect to the reference plane. The default value is 0.

### Port B elevation wrt reference plane

The parameter specifies vertical position of the pipe port B with respect to the reference plane. The default value is 0.

### Restricted Parameters

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

- **Pipe wall type**

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

## Global Parameters

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the pipe inlet.

B

Hydraulic conserving port associated with the pipe outlet.

## References

[1] White, F.M., *Viscous Fluid Flow*, McGraw-Hill, 1991

## See Also

Hydraulic Pipeline

Hydraulic Pipe LP

Linear Hydraulic Resistance

Resistive Pipe LP

Hydraulic Resistive Tube

Segmented Pipeline

# Shuttle Valve

## Purpose

Hydraulic valve that allows flow in one direction only

## Library

Directional Valves

## Description

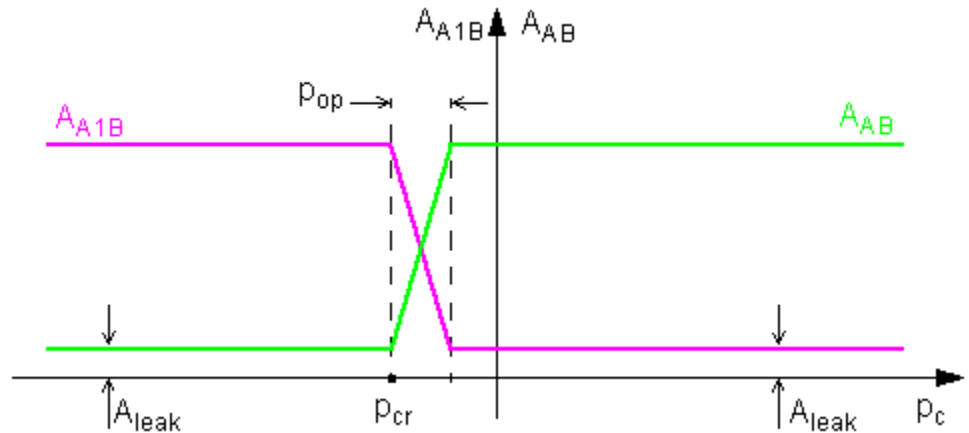


The Shuttle Valve block represents a hydraulic shuttle valve as a data-sheet-based model. The valve has two inlet ports (A and A1) and one outlet port (B). The valve is controlled by pressure differential

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

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

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



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

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

# Shuttle Valve

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

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

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

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

$$p_c = p_A - p_{A1}$$

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

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

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

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

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

where

$q_{AB}, q_{A1B}$	Flow rates through the AB and A1B orifices
$p_{AB}, p_{A1B}$	Pressure differentials across the AB and A1B orifices
$p_A, p_{A1}, p_B$	Gauge pressures at the block terminals
$C_D$	Flow discharge coefficient
$A_{AB}, A_{A1B}$	Instantaneous orifice AB and A1B passage areas
$A_{max}$	Fully open orifice passage area
$A_{leak}$	Closed valve leakage area
$p_c$	Valve control pressure differential
$p_{cr}$	Valve cracking pressure differential
$p_{op}$	Pressure differential needed to fully shift the valve
$D_{HAB}, D_{HA1B}$	Instantaneous orifice hydraulic diameters
$\rho$	Fluid density
$\nu$	Fluid kinematic viscosity

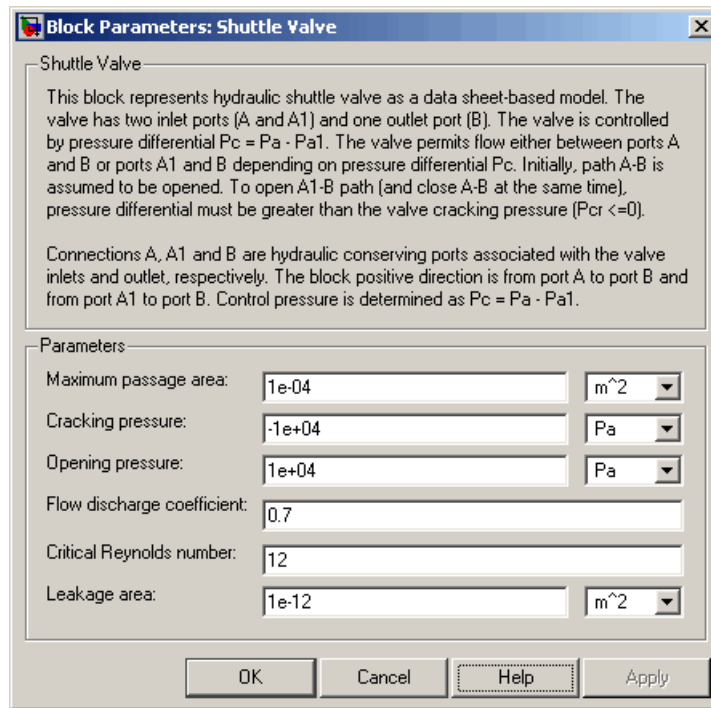
The block positive direction is from port A to port B and from port A1 to port B. Control pressure is determined as  $p_c = p_A - p_{A1}$ .

## Basic Assumptions and Limitations

- 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  $Re = Re_{cr}$ .

# Shuttle Valve

## Dialog Box and Parameters



### Maximum passage area

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

### Cracking pressure

Pressure differential level at which the orifice of the valve starts to open. The default value is  $-1e4 \text{ Pa}$ .

### Opening pressure

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

### Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the orifice, and



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

### **Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

### **Leakage area**

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## **Global Parameters**

Parameters determined by the type of working fluid:

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

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

## **Ports**

The block has the following ports:

- A  
Hydraulic conserving port associated with the valve inlet.
- A1  
Hydraulic conserving port associated with the valve inlet.

# Shuttle Valve

---

B  
Hydraulic conserving port associated with the valve outlet.

## **See Also**

Check Valve

Pilot-Operated Check Valve

# Single-Acting Hydraulic Cylinder

## Purpose

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, with the exception of the Translational Hydro-Mechanical Converter block. This custom hydro-mechanical converter takes into account pressure-induced density variations, unlike the Foundation library block, and therefore gives increased accuracy for mass conservation in fluid compressibility computations. The schematic diagram of the model is shown below.



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 piston position, which is necessary for the Hydraulic Piston Chamber block. The piston position is also conveyed to the physical signal port P, but on its way it is corrected for the piston initial position.

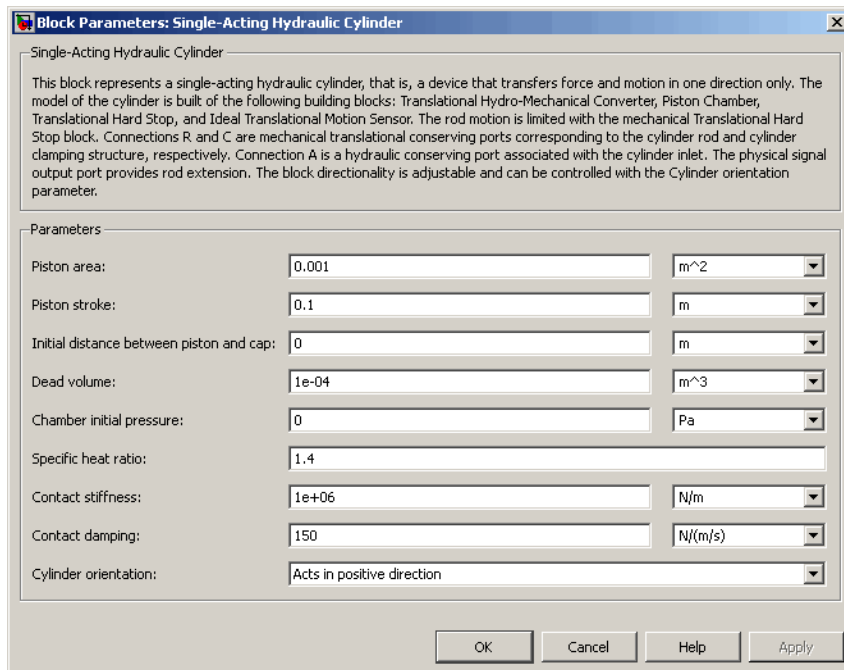
The block directionality is adjustable and can be controlled with the **Cylinder orientation** parameter.

## **Basic Assumptions and Limitations**

- No leakage, internal or external, is taken into account.
- No loading on piston rod, such as inertia, friction, spring, and so on, is taken into account. If necessary, you can easily add them by connecting an appropriate building block to cylinder port R.

# Single-Acting Hydraulic Cylinder

## Dialog Box and Parameters



**Block Parameters: Single-Acting Hydraulic Cylinder**

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

Parameters

Piston area:	0.001	m <sup>2</sup>
Piston stroke:	0.1	m
Initial distance between piston and cap:	0	m
Dead volume:	1e-04	m <sup>3</sup>
Chamber initial pressure:	0	Pa
Specific heat ratio:	1.4	
Contact stiffness:	1e+06	N/m
Contact damping:	150	N/(m/s)
Cylinder orientation:	Acts in positive direction	

OK Cancel Help Apply

### Piston area

Effective piston area. The default value is 0.001 m<sup>2</sup>.

### Piston stroke

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

### Initial distance between piston and cap

The distance between the piston and cap at the beginning of simulation. This value cannot exceed the piston stroke. The default value is 0, which corresponds to the fully retracted position.

### Dead volume

Fluid volume that remains in the chamber after the rod is fully retracted. The default value is 1e-4 m<sup>3</sup>.

# Single-Acting Hydraulic Cylinder

---

## **Chamber initial pressure**

Pressure in the cylinder chamber at the beginning of simulation. The default value is 0.

## **Specific heat ratio**

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

## **Contact stiffness**

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

## **Contact damping**

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

## **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`.

# Single-Acting Hydraulic Cylinder

---

## Restricted Parameters

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

- **Cylinder orientation**

All other block parameters are available for modification.

## Global Parameters

Parameter determined by the type of working fluid:

- **Fluid bulk modulus**

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the cylinder inlet.

R

Mechanical translational conserving port associated with the cylinder rod.

C

Mechanical translational conserving port associated with the cylinder clamping structure.

P

Physical signal output port that outputs rod extension.

## See Also

Single-Acting Hydraulic Cylinder (Simple)

Double-Acting Hydraulic Cylinder

Ideal Translational Motion Sensor

Translational Hard Stop



# Single-Acting Hydraulic Cylinder

---

Translational Hydro-Mechanical Converter

Hydraulic Piston Chamber

# Single-Acting Hydraulic Cylinder (Simple)

## Purpose

Basic functionality of single-acting hydraulic cylinder

## Library

Hydraulic Cylinders

## Description



The Single-Acting Hydraulic Cylinder (Simple) block represents a simplified version of a hydraulic cylinder, developed for applications where only the basic cylinder functionality must be reproduced, in exchange for better numerical efficiency. For these reasons, such factors as fluid compressibility, friction, and leakages are assumed to be negligible. The hard stops are assumed to be fully inelastic, to eliminate any possible oscillations at the end of the stroke. The model is especially suitable for real-time and HIL (hardware-in-the-loop) simulation, if such simplifications are acceptable.

The model is described with the following equations:

$$F = A \cdot p - F_c$$

$$q = A \cdot v$$

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

$$v = v_R - v_C$$

$$F_c = \begin{cases} (x - x_E) \cdot K_p \cdot v & \text{if } x > x_E, v > 0 \\ (x - x_R) \cdot K_p \cdot v & \text{if } x < x_R, v < 0 \\ 0 & \text{otherwise} \end{cases}$$

$$x_E = S - x_0$$

$$x_R = -x_0$$

where

# Single-Acting Hydraulic Cylinder (Simple)

---

$F$	Force developed by the cylinder
$v$	Cylinder rod velocity
$v_R, v_C$	Absolute velocities of cylinder rod and cylinder case, respectively
$A$	Piston area
$p$	Pressure at the cylinder inlet
$q$	Flow rate at the cylinder inlet
$x$	Piston position
$x_0$	Initial distance between piston and the cap
$F_c$	Hard stop force
$x_E$	Distance the piston can travel to fully extend from initial position
$x_R$	Distance the piston can travel to fully retract from initial position
$K_p$	Penetration coefficient
$S$	Piston stroke

The main difference between this block and the Single-Acting Hydraulic Cylinder block is the representation of the hard stop:

- The Single-Acting Hydraulic Cylinder block utilizes the elastic model of the stop, which is essentially a combination of a spring and a damper, connected in parallel. The stiffness of the spring, which represents contact stiffness of colliding bodies, is very high, resulting in high-frequency, low-magnitude oscillations at the moment the bodies collide. These oscillations can hardly be noticed experimentally, but they are difficult to process by a numerical simulation and generally decrease numerical efficiency.
- The hard stop in the Single-Acting Hydraulic Cylinder (Simple) block is represented with the inelastic model, which is essentially a viscous damper with the penetration-dependent damping coefficient. This

# Single-Acting Hydraulic Cylinder (Simple)

---

coefficient is referred to as the penetration coefficient. With inelastic model, no oscillation is generated during an impact, resulting in improved numerical robustness and efficiency. But inelastic stop model has a feature that you must consider while selecting the model: colliding bodies continue slowly moving into each other as long as the contact is loaded with the compressing force. In real life, this phenomenon is similar to collision of two bodies separated by a sizable layer of viscous liquid. It takes some time to squeeze the liquid before bodies themselves come into contact.

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

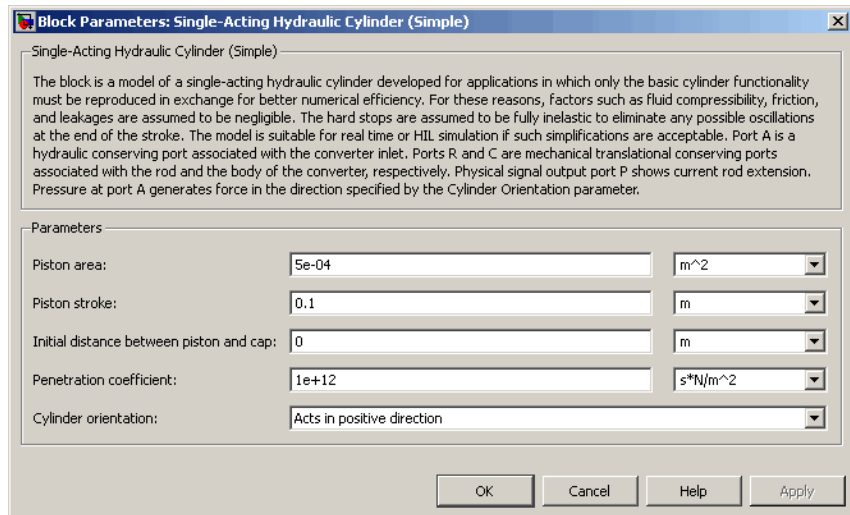
Pressure at port A generates force in the direction specified by the **Cylinder orientation** parameter.

## **Basic Assumptions and Limitations**

- Friction between moving parts is not taken into account.
- Inertia effects are not taken into account.
- Fluid compressibility is not taken into account.
- Leakage flows are assumed to be negligible.
- The hard stops are assumed to be fully inelastic, as explained above.

# Single-Acting Hydraulic Cylinder (Simple)

## Dialog Box and Parameters



### Piston area

Effective piston area. The default value is  $5e-4 \text{ m}^2$ .

### Piston stroke

Piston maximum travel between caps. The default value is  $0.1 \text{ m}$ .

### Initial distance between piston and cap

The distance between the piston and cap at the beginning of simulation. This value cannot exceed the piston stroke. The default value is  $0$ , which corresponds to the fully retracted position.

### Penetration coefficient

Specifies the penetration property of colliding bodies. The higher the value of the coefficient, the less the bodies penetrate into each other, but the sharper the increase of the impact force. To find out the value of the coefficient for a particular application, it is recommended to run the same model with different hard stop models (or with different cylinder models) until they show close results. The default value of the coefficient is  $1e12 \text{ N/m/(m/s)}$ .

# Single-Acting Hydraulic Cylinder (Simple)

---

## Cylinder orientation

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

## Restricted Parameters

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

- **Cylinder orientation**

All other block parameters are available for modification.

## Ports

The block has the following ports:

- A  
Hydraulic conserving port associated with the cylinder inlet.
- R  
Mechanical translational conserving port associated with the cylinder rod.
- C  
Mechanical translational conserving port associated with the cylinder clamping structure.
- P  
Physical signal output port that outputs rod extension.

## See Also

Double-Acting Hydraulic Cylinder  
Double-Acting Hydraulic Cylinder (Simple)  
Single-Acting Hydraulic Cylinder

# Single-Acting Hydraulic Cylinder (Simple)

---

Translational Hard Stop

Translational Hydro-Mechanical Converter

# Single-Acting Rotary Actuator

---

**Purpose** Single-acting hydraulic rotary actuator

**Library** Hydraulic Cylinders

## Description

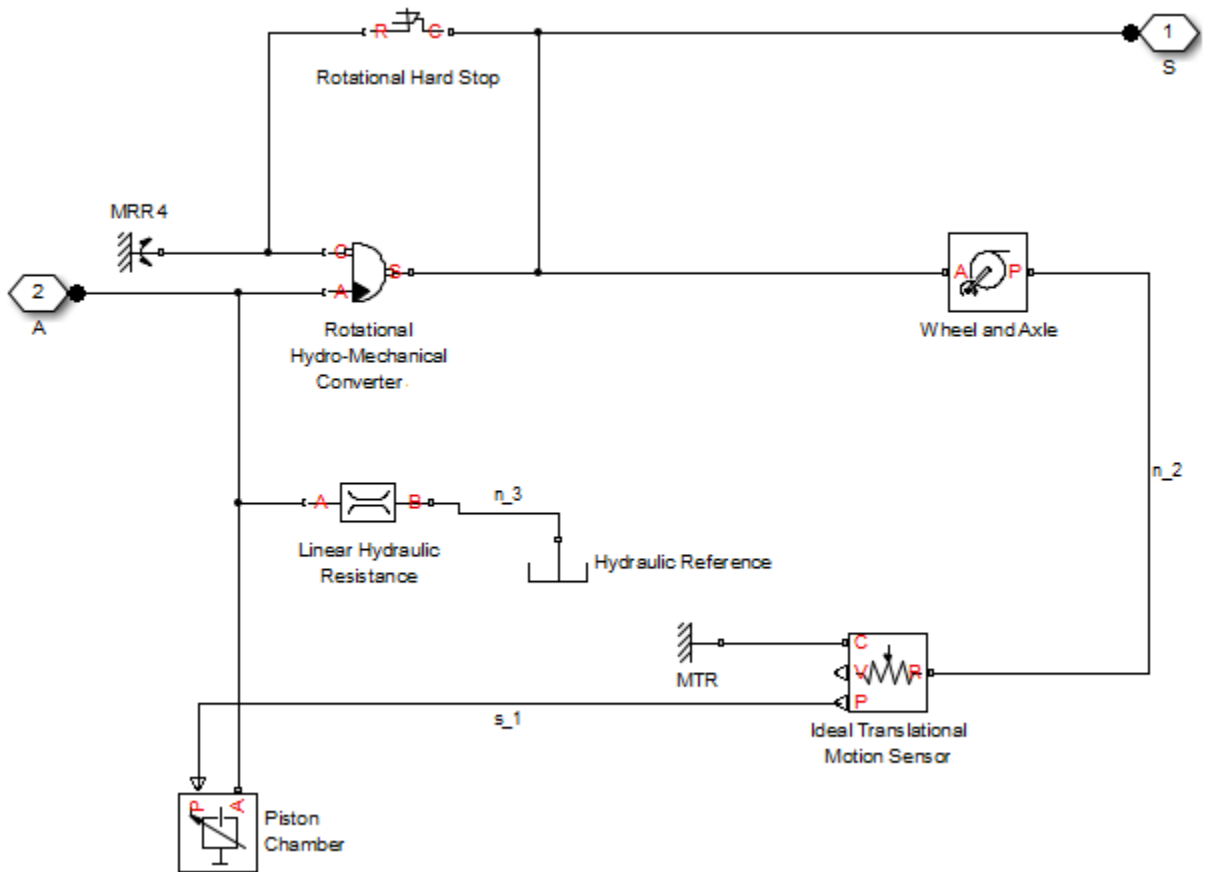


The Single-Acting Rotary Actuator block models a single-acting hydraulic rotary actuator, which directly converts hydraulic energy into mechanical rotational energy without employing intermediary transmissions such as rack-and-pinion, sliding spline, chain, and so on. Single-acting actuators generate 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, with the exception of the Rotational Hydro-Mechanical Converter block. This custom hydro-mechanical converter takes into account pressure-induced density variations, unlike the Foundation library block, and therefore gives increased accuracy for mass conservation in fluid compressibility computations. The schematic diagram of the model is shown below.



# Single-Acting Rotary Actuator



The blocks in the diagram perform the following functions:

Rotational  
Hydro-Mechanical  
Converter

Converts hydraulics energy into  
mechanical rotational energy and vice  
versa.

Rotational Hard Stop

Imposes limits on shaft rotation.

# Single-Acting Rotary Actuator

---

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 Hydraulic Piston Chamber block.
Wheel and Axle	Converts shaft rotation into translational motion to provide input to the Ideal Translational Motion Sensor block

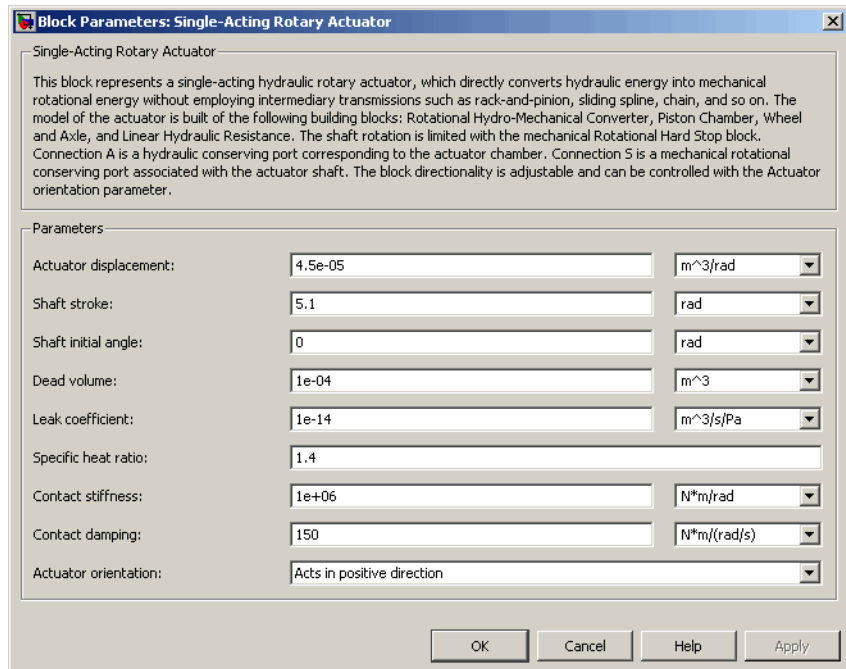
Connection A is a hydraulic conserving port corresponding to the actuator chamber. Connection S is a mechanical rotational conserving port associated with the actuator shaft.

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

## **Basic Assumptions and Limitations**

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



### Actuator displacement

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

### Shaft stroke

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

### Shaft initial angle

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

# Single-Acting Rotary Actuator

---

## **Dead volume**

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

## **Leak coefficient**

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

## **Specific heat ratio**

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

## **Contact stiffness**

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

## **Contact damping**

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

## **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

Parameter determined by the type of working fluid:

- **Fluid bulk modulus**

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

## Ports

The block has the following ports:

- A  
Hydraulic conserving port associated with the actuator inlet.
- S  
Mechanical rotational conserving port associated with the actuator shaft.

## See Also

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

# Spool Orifice Hydraulic Force

**Purpose** Axial hydraulic force exerted on spool

**Library** Valve Forces

## Description



The Spool Orifice Hydraulic Force block simulates the steady-state axial hydraulic force exerted on the spool by fluid flowing through the orifice. The spool is assumed to be sharp-edged. You have two parameterization options:

- A rectangular slot, to be used with the Orifice with Variable Area Slot block
- A round hole, or a set of round holes, to be used with the Orifice with Variable Area Round Holes block

The width of the slot, or the diameter of the holes, must be considerably larger than the radial clearance between the spool and the sleeve. Connect the Spool Orifice Hydraulic Force block in series with the respective orifice block to monitor the flow rate.

The force is simulated according to the following equations:

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

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

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

For rectangular slot,

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

For round holes,

$$A = \begin{cases} A_{\min} + Z \frac{d^2}{8} (\alpha - \sin \alpha) & \text{for } d \geq x \geq 0 \\ A_{\min} & \text{for } x < 0 \\ A_{\min} + A_{\max} & \text{for } x > d \end{cases}$$

$$\alpha = 2a \cos \left( 1 - 2 \frac{x}{d} \right)$$

$$A_{\min} = Z \cdot d \cdot \delta$$

$$A_{\max} = \pi d^2 / 4$$

where

$F$	Axial hydraulic force
$q$	Flow rate through the orifice
$\rho$	Fluid density
$A$	Orifice area
$\Theta$	Jet angle (rad)
$x_0$	Orifice initial opening
$s$	Spool displacement
$b$	Orifice width, for rectangular slot
$D$	Orifice diameter, for round holes
$Z$	Number of round holes

# Spool Orifice Hydraulic Force

---

- $\delta$  Radial clearance
- or* Orientation parameter with respect to the globally assigned positive direction. If the orifice is opened while the spool is shifted in positive direction, *or* equals 1. If the orifice is opened while the spool is shifted in negative direction, *or* equals  $-1$ .

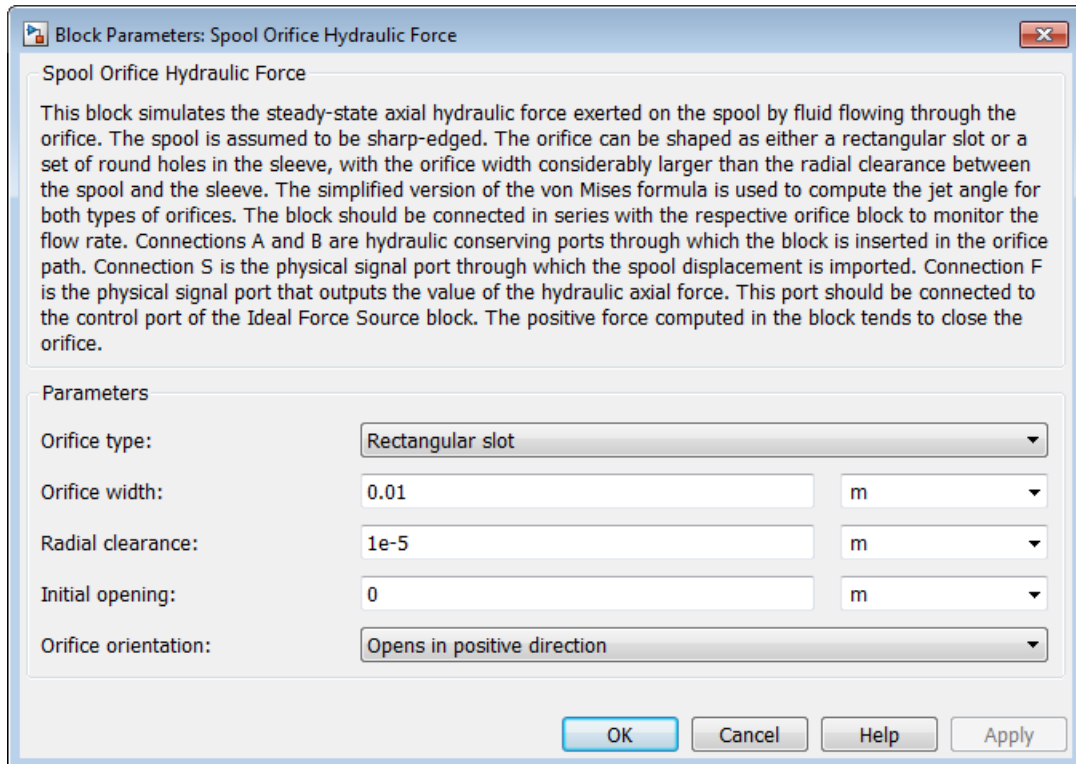
Connections A and B are hydraulic conserving ports that should be connected in series with the orifice block to monitor the flow rate. Connection S is a physical signal port that provides the spool displacement. Connection F is a physical signal port that outputs the hydraulic axial force value. This port should be connected to the control port of an Ideal Force Source block. The force computed in the block always acts to close the orifice.

## Basic Assumptions and Limitations

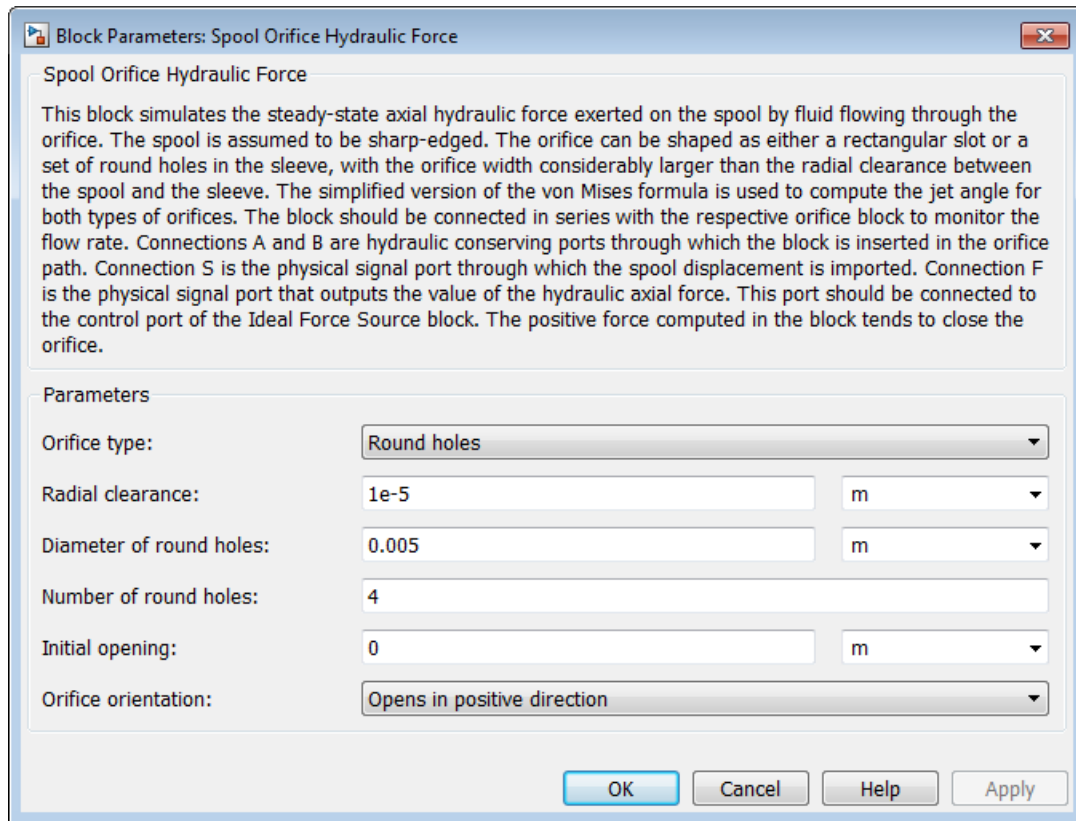
- The transient effects are assumed to be negligible.
- The jet angle approximation is based on the Richard von Mises equation.
- The jet angle variation with the orifice opening is identical for the rectangular slot and the round holes orifices.
- The block can be used with orifices whose width or diameter is considerably larger than the axial opening.



## Dialog Box and Parameters



# Spool Orifice Hydraulic Force



## Orifice type

Select the type of orifice:

- **Rectangular slot** — Orifice created by the sharp-edged spool and a rectangular slot in the sleeve. This is the default.
- **Round holes** — Orifice created by the sharp-edged spool and a set of round holes evenly distributed along the sleeve perimeter.

## **Orifice width**

Orifice width. The parameter must be greater than zero. The default value is 0.01 m. The parameter is used if **Orifice type** is set to Rectangular slot.

## **Diameter of round holes**

Orifice diameter. If multiple orifices are used, all of them must be of the same diameter, identically aligned with respect to the spool edge, and spaced evenly among the sleeve perimeter. The parameter must be greater than zero. The default value is 0.005 m. The parameter is used if **Orifice type** is set to Round holes.

## **Number of round holes**

Number of round holes in the sleeve. The parameter must be greater than zero. The default value is 4. The parameter is used if **Orifice type** is set to Round holes.

## **Radial clearance**

The radial clearance between the spool and the sleeve. The default value is  $1e-5$  m.

## **Initial opening**

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

## **Orifice orientation**

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

## **Restricted Parameters**

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

# Spool Orifice Hydraulic Force

---

- **Orifice type**
- **Orifice orientation**

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

## Ports

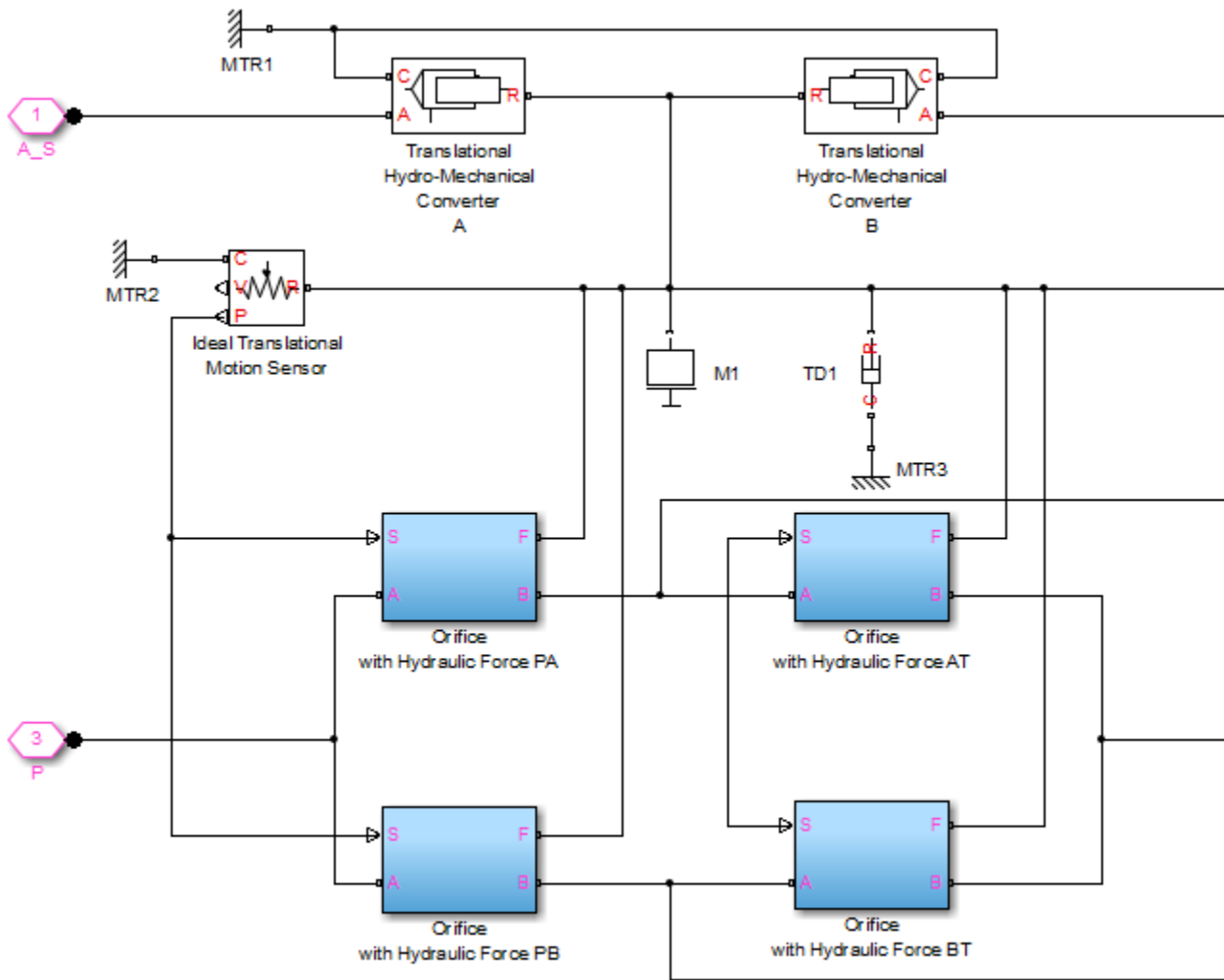
The block has the following ports:

- A Hydraulic conserving port associated with an orifice inlet.
- B Hydraulic conserving port associated with an orifice outlet.
- S Physical signal port that provides the spool displacement.
- F Physical signal port that outputs hydraulic axial force.

## Examples

The following example shows a model of a 4-way, 3-position, hydraulically-operated directional valve where the hydraulic axial forces acting on the spool are being taken into consideration.

# Spool Orifice Hydraulic Force



The spool (mass M1, viscous friction TD1) is shifted by the servo-actuator simulated by two Translational Hydro-Mechanical Converter blocks.

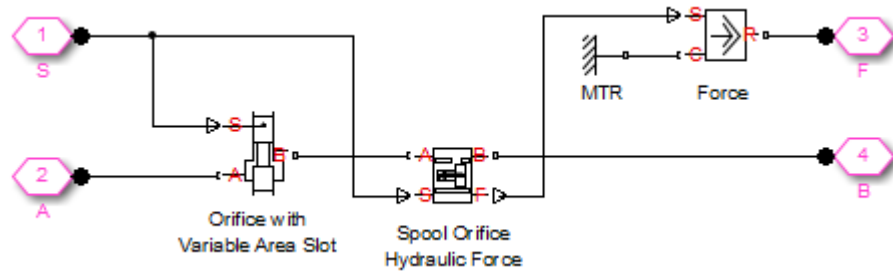
# Spool Orifice Hydraulic Force

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 example.

## See Also

Valve Hydraulic Force

## Purpose

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

To improve computational convergence and efficiency, the accumulator shell is assumed to be compliant. As a result, the fluid starts accumulating in the chamber even before the preload pressure is reached, so that at preload pressure the accumulator already stores a certain volume of fluid. The structural compliance can be set to a very small value, but not to zero.

The accumulator is described with the following equations:

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

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

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

where

# Spring-Loaded Accumulator

---

$p$	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
$V_{pr}$	Volume of fluid at preload pressure
$p_{pr}$	Preload pressure
$p_{\max}$	Pressure needed to fully fill the accumulator
$K_s$	Structural compliance of the accumulator shell
$k$	Gain coefficient

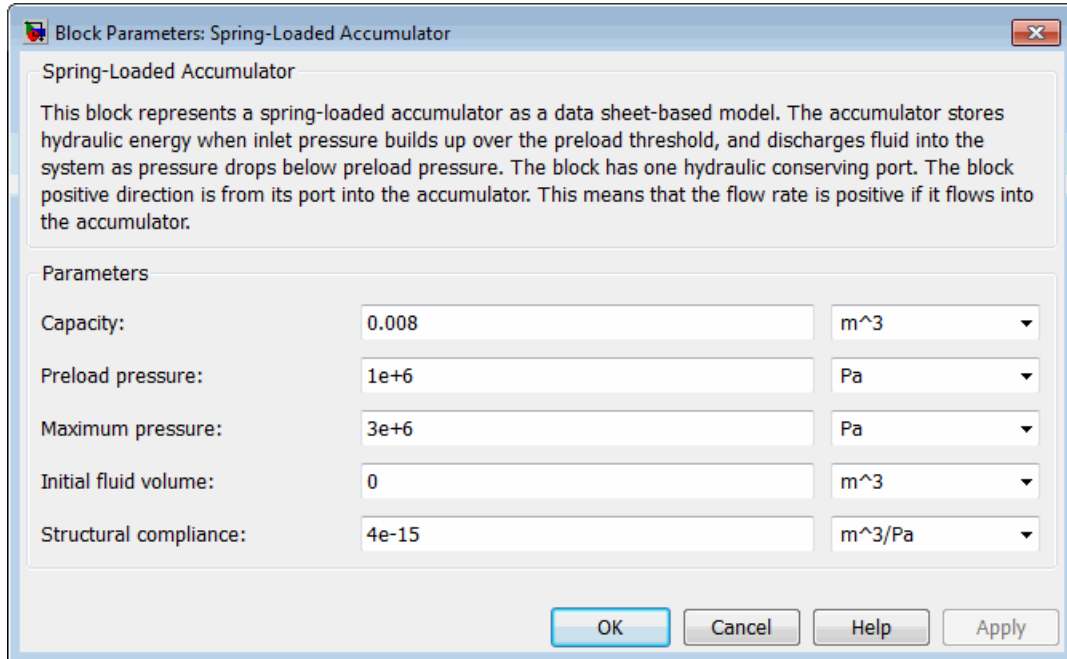
The block positive direction is from port A into the accumulator. This means that the flow rate is positive if it flows into the accumulator.

## Basic Assumptions and Limitations

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



## Dialog Box and Parameters



### Capacity

Accumulator volumetric capacity. The default value is 0.008 m<sup>3</sup>.

### Preload pressure

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

### Maximum pressure

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

# Spring-Loaded Accumulator

---

## Initial fluid volume

Initial volume of fluid in the accumulator. This parameter specifies the initial condition for use in computing the block's initial state at the beginning of a simulation run. For more information, see "Initial Conditions Computation". The default value is 0.

## Structural compliance

The compliance of the accumulator shell. This parameter is introduced to improve convergence and computational efficiency. It must be greater than zero. Values less than  $1e-16 \text{ m}^3/\text{Pa}$  may result in numerical instability. The default value is  $4e-15 \text{ m}^3/\text{Pa}$ .

## 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

Gas-Charged Accumulator

**Purpose** Sudden enlargement or contraction

**Library** Local Hydraulic Resistances

## Description



The Sudden Area Change block represents a local hydraulic resistance, such as a sudden cross-sectional area change. The resistance represents a sudden enlargement if fluid flows from inlet to outlet, or a sudden contraction if fluid flows from outlet to inlet. The block is based on the Local Resistance block. It determines the pressure loss coefficient and passes its value to the underlying Local Resistance block. The block offers two methods of parameterization: by applying semi-empirical formulas (with a constant value of the pressure loss coefficient) or by table lookup for the pressure loss coefficient based on the Reynolds number.

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

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

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

where

$K_{SE}$  Pressure loss coefficient for the sudden enlargement, which takes place if fluid flows from inlet to outlet

$K_{SC}$  Pressure loss coefficient for the sudden contraction, which takes place if fluid flows from outlet to inlet

$K_{cor}$  Correction factor

# Sudden Area Change

---

$A_S$  Small area

$A_L$  Large area

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

The pressure loss coefficient, determined by either of the two methods, is then passed to the underlying Local Resistance block, which computes the pressure loss according to the formulas explained in the reference documentation for that block. The flow regime is checked in the underlying Local Resistance block by comparing the Reynolds number to the specified critical Reynolds number value, and depending on the result, the appropriate formula for pressure loss computation is used.

The Sudden Area Change block is bidirectional and computes pressure loss for both the direct flow (sudden enlargement) and return flow (sudden contraction). If the loss coefficient is specified by a table, the table must cover both the positive and the negative flow regions.

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

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

## Basic Assumptions and Limitations

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

## Dialog Box and Parameters

**Block Parameters: Sudden Area Change**

Sudden Area Change

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

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

Parameters

Small diameter:  m

Large diameter:  m

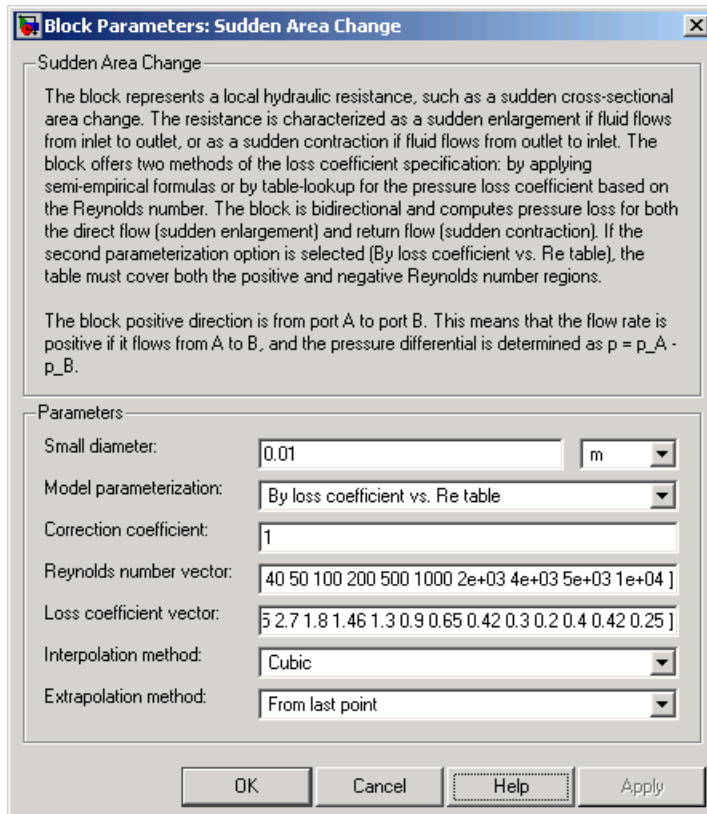
Model parameterization:

Correction coefficient:

Critical Reynolds number:

OK Cancel Help Apply

# Sudden Area Change



## Small diameter

Resistance small diameter. The default value is 0.01 m.

## Large diameter

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

## Model parameterization

Select one of the following methods for block parameterization:

- **By semi-empirical formulas** — Provide geometrical parameters of the resistance. This is the default method.
- **By loss coefficient vs. Re table** — Provide tabulated relationship between the loss coefficient and the Reynolds number. The loss coefficient is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods. The table must cover both the positive and the negative flow regions.

### **Correction coefficient**

Correction factor used in the formula for computation of the loss coefficient. The default value is 1. This parameter is used if **Model parameterization** is set to **By semi-empirical formulas**.

### **Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 110. This parameter is used if **Model parameterization** is set to **By semi-empirical formulas**.

### **Reynolds number vector**

Specify the vector of input values for Reynolds numbers as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values are [-4000, -3000, -2000, -1000, -500, -200, -100, -50, -40, -30, -20, -15, -10, 10, 20, 30, 40, 50, 100, 200, 500, 1000, 2000, 4000, 5000, 10000]. This parameter is used if **Model parameterization** is set to **By loss coefficient vs. Re table**.

# Sudden Area Change

---

## Loss coefficient vector

Specify the vector of the loss coefficient values as a one-dimensional array. The loss coefficient vector must be of the same size as the Reynolds numbers vector. The default values are [0.25, 0.3, 0.65, 0.9, 0.65, 0.75, 0.90, 1.15, 1.35, 1.65, 2.3, 2.8, 3.10, 5, 2.7, 1.8, 1.46, 1.3, 0.9, 0.65, 0.42, 0.3, 0.20, 0.40, 0.42, 0.25]. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

## Interpolation method

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

- **Linear** — Uses a linear interpolation function.
- **Cubic** — Uses the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP).
- **Spline** — Uses the cubic spline interpolation algorithm.

For more information on interpolation algorithms, see the `PS Lookup Table (1D)` block reference page. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

## Extrapolation method

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

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



- **From last point** — Uses the last specified output value at the appropriate end of the range. That is, the block uses the last specified output value for all input values greater than the last specified input argument, and the first specified output value for all input values less than the first specified input argument.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to **By loss coefficient vs. Re table**.

## Restricted Parameters

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

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

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

## Global Parameters

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

# Sudden Area Change

---

- A Hydraulic conserving port associated with the resistance inlet.
- B Hydraulic conserving port associated with the resistance outlet.

## References

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

## See Also

- Elbow
- Gradual Area Change
- Local Resistance
- Pipe Bend
- T-junction

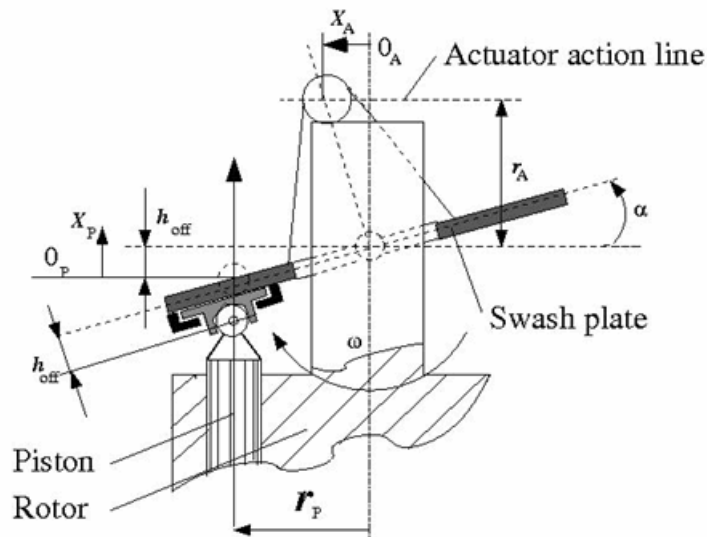
**Purpose** Swash plate mechanism

**Library** Pumps and Motors

**Description**



The Swash Plate block is a model of a swash plate mechanism used in axial-piston hydraulic pumps and motors to convert translational motion of a piston into rotational motion of a drive shaft. The mechanism is a combination of a cylinder block or rotor, with the piston fitted inside, and an angled swash plate.



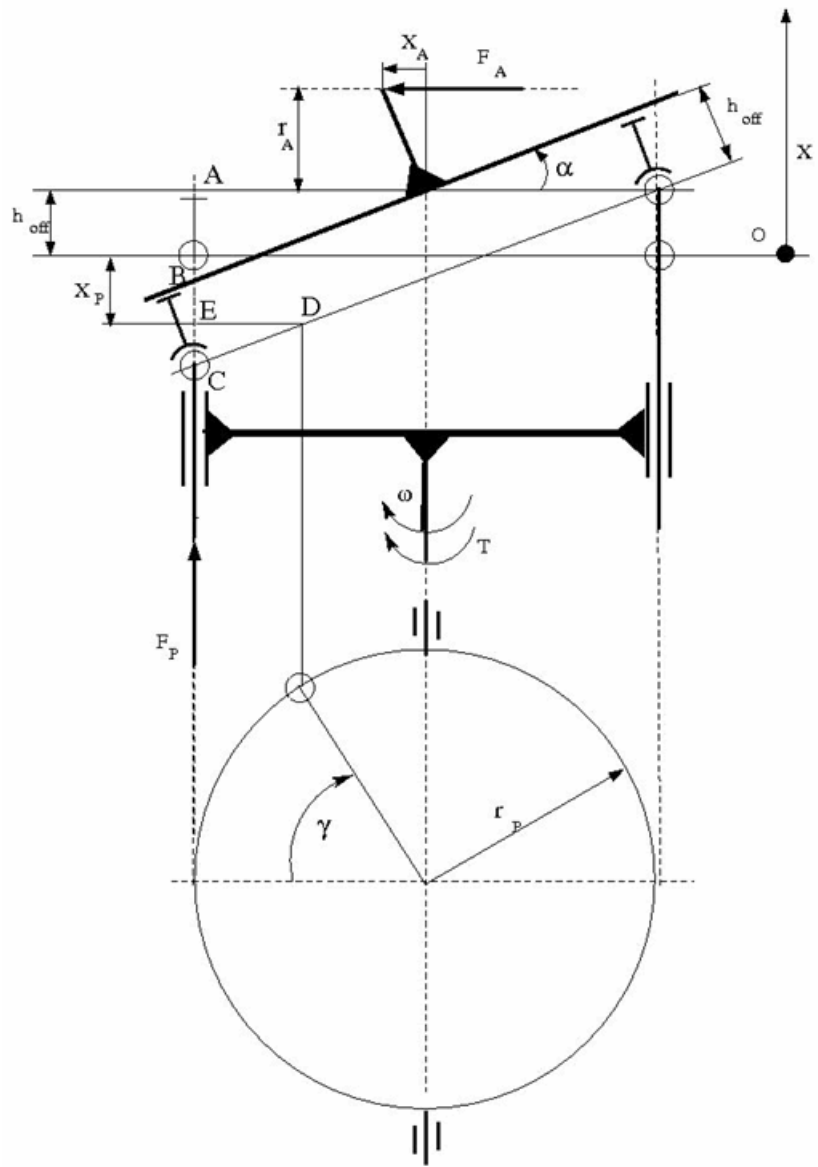
The angular position of the plate is controlled by an actuator, usually a servo cylinder, acting along the actuator action line (marked in the preceding diagram). The distance between the actuator line and the swash plate at zero angle is equal to the actuator arm length  $r_A$ . The pistons are offset by  $h_{off}$  from the swash plate. The swash plate angle can be changed to increase or decrease the piston stroke. As the rotor turns, the piston follows the plate, which causes the piston to reciprocate.

# Swash Plate

---

In a hydraulic machine, the piston moves along the slots in the porting plate (not shown in the preceding diagram) thus being periodically connected to either the intake or the discharge port of the machine. The model simulates a single piston interacting with the swash plate. Mechanically, the mechanism performs summation of two motions, caused by rotation of the swash plate with respect to its axis and rotation of the rotor, and converts them into displacement of the piston.

The following illustration shows the calculation diagram of the mechanism.



# Swash Plate

---

The positive direction of the actuator is assumed to be directed to the left from the vertical axis, while the positive direction of the piston motion is directed upward, as it is shown in the diagram. The reference point for both motions corresponds to the plate being perpendicular to the rotor axis. As far as the angular motion of the rotor is concerned, the clockwise rotation (looking from the top) is assumed to be positive.

The piston displacement at arbitrary rotor rotation angle  $\gamma$  and actuator displacement  $x_A$  is determined from the relationship:

$$x_P = -(AB - h_{off} + BC + DE \cdot tg\alpha)$$

$$tg\alpha = \frac{x_A}{r_A}$$

which, after minor rearrangement, yields:

$$x_P = -\frac{r_P}{r_A} x_A \cos(\gamma + \beta) - h_{off} \left( \sqrt{1 + \frac{x_A^2}{r_A^2}} - 1 \right) \quad (1-11)$$

$$\gamma = \omega \cdot t$$

where

$x_P$	Piston displacement
$x_A$	Actuator displacement
$r_P$	Cylinder block pitch radius
$r_A$	Actuator arm
$\gamma$	Rotor angle of rotation
$h_{off}$	Piston offset
$\beta$	Piston phase angle

$\omega$  Rotor angular velocity

$t$  Time

The kinematic relationship between piston and actuator displacement must be converted into a relationship between the basic Across variables for the physical network representation, that is, velocities:

$$V_P = \frac{dx_P}{dt} = -\frac{r_P}{r_A} V_A \cos(\gamma + \beta) + \frac{r_P}{r_A} x_A \omega \sin(\gamma + \beta) - \frac{h_{off}}{r_A^2} \frac{x_A V_A}{\sqrt{1 + \frac{x_A^2}{r_A^2}}}$$

where

$V_P$  Piston velocity

$V_A$  Actuator velocity

The inertia properties of the mechanism are assumed to be negligible. Therefore, any state of the mechanism can be considered as equilibrium. According to the principle of virtual work, if the system is in equilibrium the aggregate work of all the applied forces and torques on the system virtual displacements is equal to zero. For the mechanism under consideration, the principle of virtual work yields:

$$F_P \delta x_P + F_A \delta x_A + (T - T_{res}) \delta \gamma = 0 \quad (1-12)$$

where

$F_P$  Piston force

$F_A$  Actuator force

$T$  Rotor torque

$T_{res}$  Resistance torque acting on the rotor due to viscous friction

# Swash Plate

Virtual displacements  $\delta x_p$ ,  $\delta x_A$ , and  $\delta y$  are determined from Equation 1-11 and substituted into Equation 1-12. Since virtual displacements  $\delta x_A$  and  $\delta y$  are independent, any of them can be equal to zero. As a result, we obtain two more equations of the mechanism:

$$F_A = \frac{r_P}{r_A} F_P \cos(\gamma + \beta) + \frac{h_{off}}{r_A^2} \frac{x_A}{\sqrt{1 + \frac{x_A^2}{r_A^2}}} F_P$$

$$T = -\frac{r_P}{r_A} x_A \cdot F_P \cdot \sin(\gamma + \beta) + T_{res}$$

The resistance torque is:

$$T_{res} = \mu \cdot r_P \cdot F_P$$

where  $\mu$  is the viscous friction coefficient in the contact between the piston and the plate.

The final set of equations describing the swash plate mechanism takes the form:

$$V_P = \frac{dx_P}{dt} = -\frac{r_P}{r_A} V_A \cos(\gamma + \beta) + \frac{r_P}{r_A} x_A \omega \sin(\gamma + \beta) - \frac{h_{off}}{r_A^2} \frac{x_A V_A}{\sqrt{1 + \frac{x_A^2}{r_A^2}}}$$

$$F_A = \frac{r_P}{r_A} F_P \cos(\gamma + \beta) + \frac{h_{off}}{r_A^2} \frac{x_A}{\sqrt{1 + \frac{x_A^2}{r_A^2}}} F_P$$

$$T = -\frac{r_P}{r_A} x_A \cdot F_P \cdot \sin(\gamma + \beta) + \mu \cdot r_P \cdot F_P$$

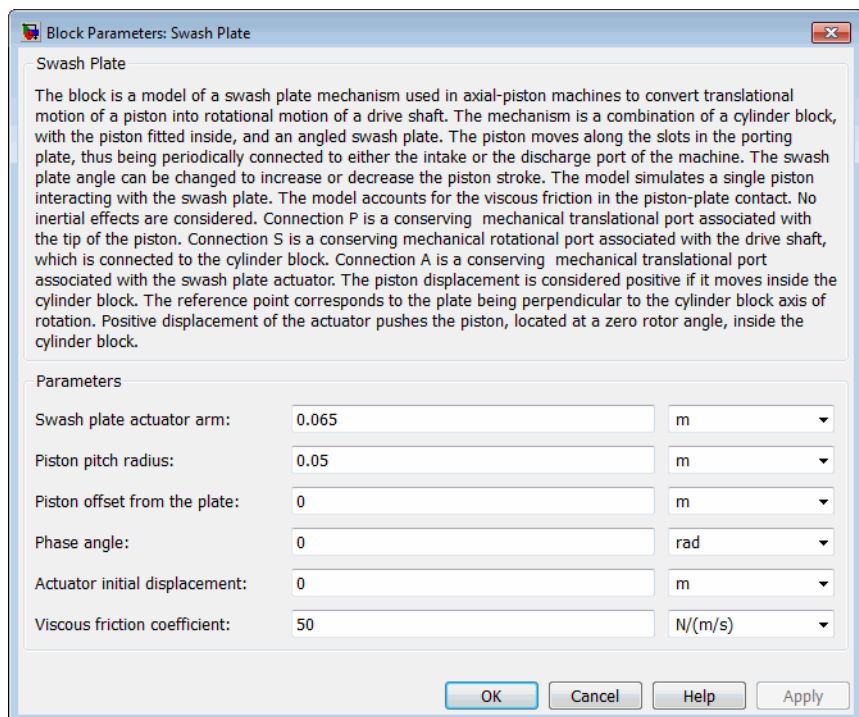


Connection P is a mechanical translational conserving port associated with the tip of the piston. Connection S is a mechanical rotational conserving port associated with the drive shaft, which is connected to the cylinder block. Connection A is a mechanical translational conserving port associated with the swash plate actuator.

## Basic Assumptions and Limitations

- The model accounts for the viscous friction in the piston-plate contact.
- No inertial effects are considered.
- The plate angular displacements are considered to be small.
- The joint between the piston and the plate permanently maintains contact between the piston and the plate.

## Dialog Box and Parameters



# Swash Plate

---

## **Swash plate actuator arm**

The distance between the actuator line of action and the center of the rotor rotation. The parameter must be greater than zero. The default value is 0.065 m.

## **Piston pitch radius**

The distance between the actuator line of action and the center of the rotor rotation. The parameter must be greater than zero. The default value is 0.05 m.

## **Piston offset from the plate**

The offset between the piston tip and the swash plate. The parameter must be greater than or equal to zero. The default value is 0.

## **Phase angle**

Parameter sets the piston initial angular position with respect to the reference point which corresponds to the zero angle. The default value is 0.

## **Actuator initial displacement**

Parameter sets the initial position of the actuator with respect to the reference point which corresponds to the plate being perpendicular to the axis of rotor rotation. The default value is 0.

## **Viscous friction coefficient**

The parameter specifies the coefficient of viscous friction in the contact between the piston and the plate. The parameter must be greater than zero. The default value is 50 N/(m/s).

## **Ports**

The block has the following ports:

A

Mechanical translational conserving port associated with the actuator joint.

P

Mechanical translational conserving port associated with the tip of the piston.

S

Mechanical rotational conserving port associated with the drive shaft.

## Examples

The Hydraulic Axial-Piston Pump with Load-Sensing and Pressure-Limiting Control example models a test rig designed to investigate interaction between an axial-piston pump and a typical control unit, simultaneously performing the load-sensing and pressure-limiting functions. To assure required accuracy, the model of the pump must account for such features as interaction between pistons, swash plate, and porting plate, which makes it necessary to build a detailed pump model.

## See Also

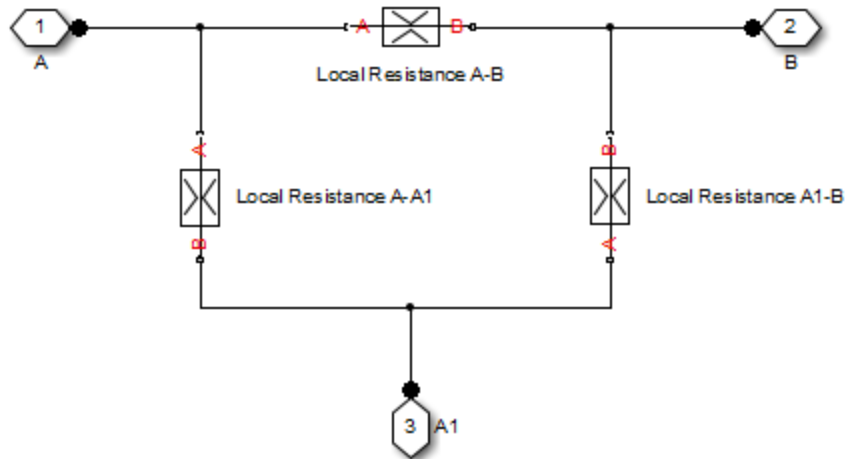
Porting Plate Variable Orifice

# T-junction

**Purpose** Hydraulic resistance of T-junction in pipe

**Library** Local Hydraulic Resistances

**Description** The T-junction block represents a T-junction (wye connection) consisting, in general, of a main run and a branch merging to the main run. The junction as a hydraulic resistance is built of three Local Resistance blocks, as shown in the following diagram.



To specify pressure loss for all possible flow directions, you have to provide six pressure loss coefficients. The flow regime is checked in the underlying Local Resistance blocks by comparing the Reynolds number to the specified critical Reynolds number value, and depending on the result, the appropriate formula for pressure loss computation is used. For more information, see the reference documentation for the Local Resistance block.

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

## Basic Assumptions and Limitations

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

## Dialog Box and Parameters

**Block Parameters: T-junction**

T-junction

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

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

Parameters

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

OK Cancel Help Apply

# T-junction

---

## **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

Parameters determined by the type of working fluid:

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

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

## Ports

The block has the following ports:

A  
Hydraulic conserving port associated with the main run inlet.

B  
Hydraulic conserving port associated with the main run outlet.

A1  
Hydraulic conserving port associated with the branch inlet.

## See Also

Elbow

Gradual Area Change

Local Resistance

Pipe Bend

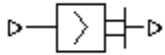
Sudden Area Change

# Valve Actuator

**Purpose** Simplified model of valve driver

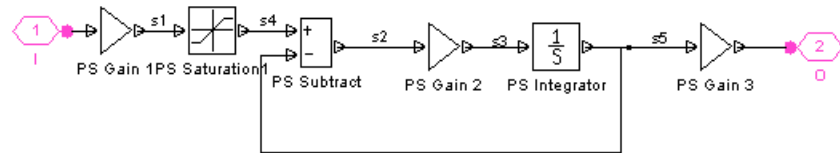
**Library** Valve Actuators

## Description



The Valve Actuator block represents a simplified model of a valve actuator, which is especially suitable for real-time and HIL (hardware-in-the-loop) simulation. Its functionality is similar to the 2-Position Valve Actuator and 3-Position Valve Actuator blocks, but its characterization requires fewer parameters because it is built of components that are practically linear.

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



The gain of the PS Gain 1 block establishes the steady-state relationship between the input and output signals. The PS Saturation block keeps the output signal within the  $\pm$  valve stroke range. The gain of the PS Gain 2 block is  $1/\text{Time constant}$ , which makes it possible to reproduce the first-order lag by combining this block with the PS Subtract and PS Integrator blocks.

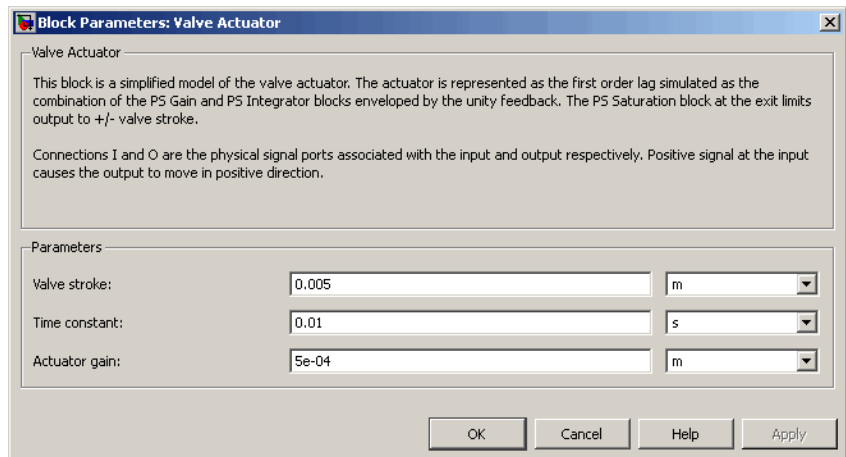
The transfer function of the lag is

$$H(s) = \frac{1}{Ts + 1}$$

where  $T$  is the time constant. The gain of the PS Gain 3 block is set to 1.



## Dialog Box and Parameters



### Valve stroke

The desired valve stroke, which defines the saturation level of the PS Saturation block in the actuator model. The output of the actuator is kept within the +/- valve stroke range. The default value is 0.005 m.

### Time constant

Time constant of the first-order lag. The **Gain** parameter of the PS Gain 2 block is equal to  $1/\text{Time constant}$ . The default value is 0.01 s.

### Actuator gain

Gain of the PS Gain 1 block in the block diagram of the actuator model. The default value is  $5e-4$  m.

## Ports

The block has one physical signal input port and one physical signal output port.

## See Also

4-Way Ideal Valve

Proportional and Servo-Valve Actuator

# Valve Hydraulic Force

---

**Purpose** Axial hydraulic static force exerted on valve

**Library** Valve Forces

## Description



The Valve Hydraulic Force block simulates axial hydraulic static force exerted on a valve by fluid flowing through the orifice. The relationship between the valve opening, the pressure drop, and the force is provided as a two-dimensional table, which is processed by the PS Lookup Table (2D) block. The table can be obtained experimentally or analytically and can represent both the hydraulic static axial force and pressure forces. The force matrix must be rectangular and contain as many rows as there are pressure differential measurements and as many columns as there are valve openings. The pressure differential and opening vectors must be arranged in strictly ascending order and cover the whole range of valve operation. Connect the block in parallel with the orifice whose flow induces the force.

Connections A and B are hydraulic conserving ports that should be connected to the valve block ports in such a way as to monitor the pressure differential across the valve. Connection S is a physical signal port that provides the valve control member displacement. Connection F is a physical signal port that outputs the hydraulic axial force value. This port should be connected to the control port of an Ideal Force Source block. The pressure differential inside the block is determined as  $p = p_A - p_B$ . The force orientation is specified by the table values and can be positive or negative with respect to the globally assigned positive direction, depending on the value of the **Orifice orientation** parameter.

## Basic Assumptions and Limitations

No transient effects can be simulated.

## Dialog Box and Parameters

**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:  m

Orifice orientation: Opens in positive direction

Tabulated valve openings:  m

Tabulated pressure differentials:  Pa

Hydraulic axial force table:  N

Interpolation method: Linear

Extrapolation method: Linear

# Valve Hydraulic Force

---

## **Initial opening**

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

## **Orifice orientation**

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

## **Tabulated valve openings**

Specify the vector of input values for valve openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in meters, are [0, 1e-3, 2e-3, 3e-3, 4e-3]. The **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** for two-dimensional table lookup in the **Hydraulic axial force table**.

## **Tabulated pressure differentials**

Specify the vector of input values for pressure differentials as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in Pa, are [-100e5, -75e5, -50e5, -25e5, 0, 25e5, 50e5, 75e5, 100e5].

## **Hydraulic axial force table**

Specify the hydraulic axial force as an m-by-n matrix, where m is the number of valve openings and n is the number of pressure differentials. Each value in the matrix specifies an axial force

corresponding to a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in N, are:

```
[0, -127.3576, -27.8944, 227.2513, 575.3104; ...
 0, -95.5182, -20.9208, 170.4385, 431.4828; ...
 0, -63.6788, -13.9472, 113.6256, 287.6552; ...
 0, -31.8394, -6.9736, 56.8128, 143.8276; ...
 0, 0, 0, 0, 0; ...
196.3495, 120.7506, 97.5709, 111.9898, 150.9306; ...
392.6991, 241.5013, 195.1418, 223.9797, 301.8613; ...
589.0486, 362.2519, 292.7126, 335.9695, 452.7919; ...
785.3982, 483.0025, 390.2835, 447.9594, 603.7225]
```

### Interpolation method

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

- **Linear** — 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.

### Extrapolation method

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

- **Linear** — 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.

# Valve Hydraulic Force

---

- **Nearest** — 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.

## Restricted Parameters

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

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

All other block parameters are available for modification.

## Ports

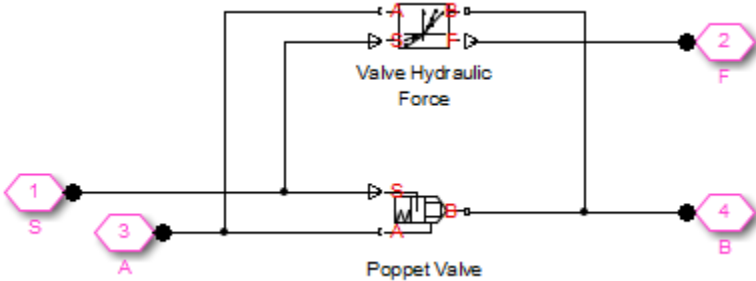
The block has the following ports:

- A  
Hydraulic conserving port associated with a valve port.
- B  
Hydraulic conserving port associated with another valve port to monitor the pressure differential.
- S  
Physical signal port that provides the valve control member displacement.
- F  
Physical signal port that outputs hydraulic axial force.

## Examples

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

# Valve Hydraulic Force



**See Also**

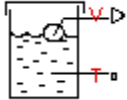
Spool Orifice Hydraulic Force

# Variable Head Tank

**Purpose** Tank with constant pressurization and volume-dependent fluid level

**Library** Low-Pressure Blocks

## Description



The Variable Head Tank block represents a pressurized hydraulic reservoir, in which fluid is stored under a specified pressure. The pressurization remains constant regardless of volume change. The block accounts for the fluid level change caused by the volume variation, as well as for pressure loss in the connecting pipe that can be caused by a filter, fittings, or some other local resistance. The loss is specified with the pressure loss coefficient. The block computes the volume of fluid in the tank and exports it outside through the physical signal port  $V$ .

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

$$p = p_{elev} - p_{loss} + p_{pr}$$

$$p_{elev} = \rho g H$$

$$p_{loss} = K \frac{\rho}{2A_p^2} q |q|$$

$$A_p = \frac{\pi d^2}{4}$$

$$H = \begin{cases} \frac{V}{A} & \text{for constant-area tank} \\ f(V) & \text{for table-specified tank} \end{cases}$$

$$V = V_0 + q t$$

where



$p$	Pressure at the tank inlet
$p_{elev}$	Pressure due to fluid level
$p_{loss}$	Pressure loss in the connecting pipe
$p_{pr}$	Pressurization
$\rho$	Fluid density
$g$	Acceleration of gravity
$H$	Fluid level with respect to the bottom of the tank
$K$	Pressure loss coefficient
$A_p$	Connecting pipe area
$d$	Connecting pipe diameter
$q$	Flow rate
$V$	Instantaneous fluid volume
$V_0$	Initial fluid volume
$A$	Tank cross-sectional area
$t$	Simulation time

For a tank with a variable cross-sectional area, the relationship between fluid level and volume is specified with the table lookup

$$H = f(V)$$

You have a choice of three interpolation methods and two extrapolation methods.

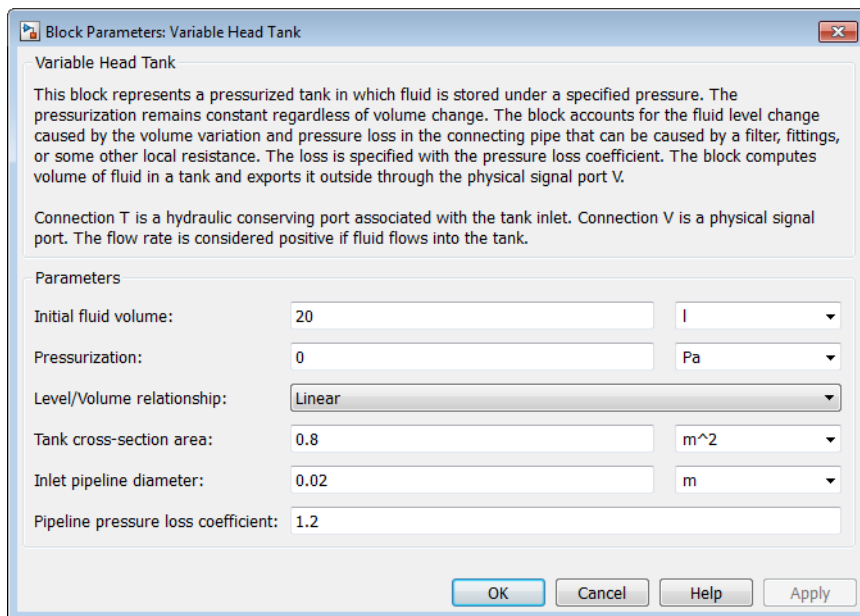
Connection T is a hydraulic conserving port associated with the tank inlet. Connection V is a physical signal port. The flow rate is considered positive if fluid flows into the tank.

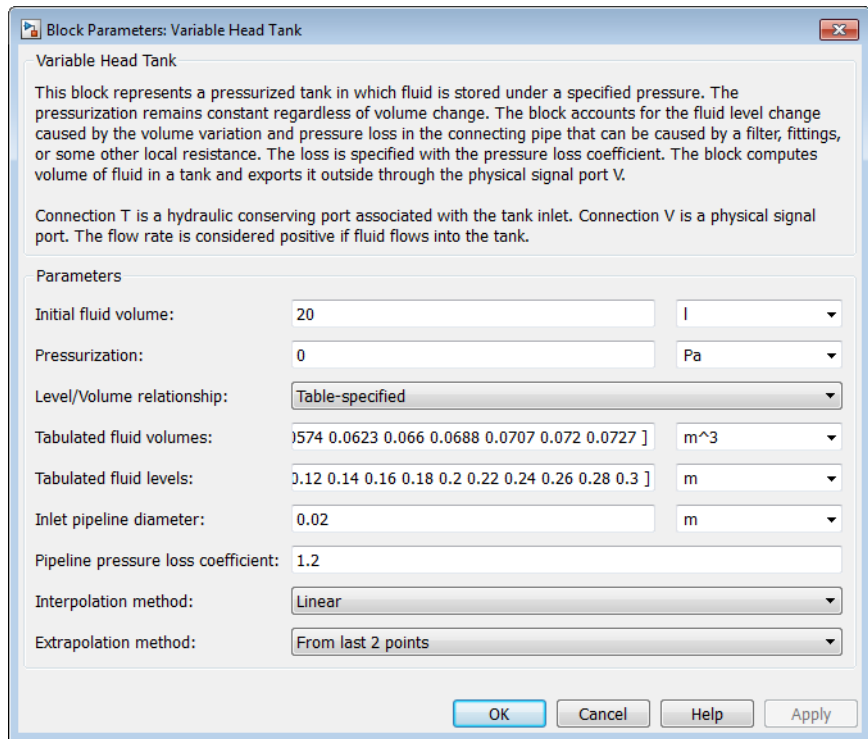
# Variable Head Tank

## Warning

If fluid level becomes so low that the tank inlet gets exposed, no warnings will be issued. The simulation will continue and pressure at the inlet will be set to the pressurization pressure. If this is not acceptable, MathWorks recommends that you employ the necessary control measures to guard against this situation in your models.

## Dialog Box and Parameters





## Initial fluid volume

The initial volume of fluid in the tank. This parameter must be greater than zero. The default value is 20 l.

## Pressurization

Gage pressure acting on the surface of the fluid in the tank. It can be created by a gas cushion, membrane, bladder, or piston, as in bootstrap reservoirs. This parameter must be greater than or equal to zero. The default value is 0, which corresponds to a tank connected to atmosphere.

## Level/Volume relationship

Select one of the following block parameterization options:

# Variable Head Tank

---

- **Linear** — Provide a value for the tank cross-sectional area. The level is assumed to be linearly dependent on the fluid volume. This is the default method.
- **Table-specified** — Provide tabulated data of fluid volumes and fluid levels. The level is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.

## **Tank cross-section area**

The cross-sectional area of the tank. This parameter must be greater than zero. The default value is  $0.8 \text{ m}^2$ . This parameter is used if **Level/Volume relationship** is set to **Linear**.

## **Tabulated fluid volumes**

Specify the vector of input values for fluid volume as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in  $\text{m}^3$ , are [0 0.0028 0.0065 0.0114 0.0176 0.0252 0.0344 0.0436 0.0512 0.0574 0.0623 0.066 0.0688 0.0707 0.072 0.0727]. This parameter is used if **Level/Volume relationship** is set to **Table-specified**.

## **Tabulated fluid levels**

Specify the vector of fluid levels as a one-dimensional array. The fluid levels vector must be of the same size as the fluid volumes vector. The default values, in meters, are [0 0.02 0.04 0.06 0.08 0.1 0.12 0.14 0.16 0.18 0.2 0.22 0.24 0.26 0.28 0.3]. This parameter is used if **Level/Volume relationship** is set to **Table-specified**.

## **Inlet pipeline diameter**

The diameter of the connecting pipe. This parameter must be greater than zero. The default value is  $0.02 \text{ m}$ .

## Pipeline pressure loss coefficient

The value of the pressure loss coefficient, to account for pressure loss in the connecting pipe. This parameter must be greater than zero. The default value is 1.2.

For reasons of computational robustness, the loss is computed with the equation similar to that used in the Fixed Orifice block:

$$q = \frac{1}{\sqrt{K}} A_p \sqrt{\frac{2}{\rho} P_{loss}}$$

The Critical Reynolds number is set to 15.

## Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- **Linear** — Uses a linear interpolation function.
- **Cubic** — Uses the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP).
- **Spline** — Uses the cubic spline interpolation algorithm.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Level/Volume relationship** is set to `Table-specified`.

## 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

# Variable Head Tank

---

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 **Level/Volume relationship** is set to **Table-specified**.

## Ports

The block has the following ports:

T

Hydraulic conserving port associated with the tank inlet.

V

Physical signal port that outputs the volume of fluid in the tank.

## See Also

Constant Head Tank

Reservoir

Variable Head Three-Arm Tank

Variable Head Two-Arm Tank

# Variable Head Three-Arm Tank

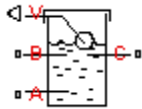
## Purpose

Tank with three hydraulic ports, constant pressurization, and volume-dependent fluid level

## Library

Low-Pressure Blocks

## Description



The Variable Head Three-Arm Tank block represents a three-arm pressurized tank, in which fluid is stored under a specified pressure. The pressurization remains constant regardless of volume change. The block accounts for the fluid level change caused by the volume variation, as well as for pressure loss in the connecting pipes that can be caused by a filter, fittings, or some other local resistance. The loss is specified with the pressure loss coefficients. The block computes the volume of fluid in the tank and exports it outside through the physical signal port V.

The pressure at the tank outlets is computed with the following equations:

$$p_A = p_{elevA} - p_{lossA} + p_{pr}$$

$$p_B = p_{elevB} - p_{lossB} + p_{pr}$$

$$p_C = p_{elevC} - p_{lossC} + p_{pr}$$

$$p_{elevA} = \rho g H$$

$$p_{elevB} = \rho g (H - h_{BA})$$

$$p_{elevC} = \rho g (H - h_{CA})$$

$$p_{lossA} = K_A \frac{\rho}{2A_A^2} q_A |q_A|$$

$$p_{lossB} = K_B \frac{\rho}{2A_B^2} q_B |q_B|$$

# Variable Head Three-Arm Tank

---

$$p_{lossC} = K_C \frac{\rho}{2A_C^2} q_C |q_C|$$

$$A_A = \frac{\pi d_A^2}{4}$$

$$A_B = \frac{\pi d_B^2}{4}$$

$$A_C = \frac{\pi d_C^2}{4}$$

$$H = \begin{cases} \frac{V}{A} & \text{for constant-area tank} \\ f(V) & \text{for table-specified tank} \end{cases}$$

$$V = V_0 + q \bar{t}$$

where

$p_A$	Pressure at the tank outlet A
$p_B$	Pressure at the tank outlet B
$p_C$	Pressure at the tank outlet C
$p_{elevA}$	Pressure due to fluid level at outlet A
$p_{elevB}$	Pressure due to fluid level at outlet B
$p_{elevC}$	Pressure due to fluid level at outlet C
$p_{lossA}$	Pressure loss in the connecting pipe A
$p_{lossB}$	Pressure loss in the connecting pipe B
$p_{lossC}$	Pressure loss in the connecting pipe C
$p_{pr}$	Pressurization



# Variable Head Three-Arm Tank

---

$\rho$	Fluid density
$g$	Acceleration of gravity
$H$	Fluid level with respect to outlet A
$h_{AB}$	Elevation of outlet B with respect to outlet A
$h_{AC}$	Elevation of outlet C with respect to outlet A
$K_A$	Pressure loss coefficient at outlet A
$K_B$	Pressure loss coefficient at outlet B
$K_C$	Pressure loss coefficient at outlet C
$A_A$	Connecting pipe area at outlet A
$A_B$	Connecting pipe area at outlet B
$A_C$	Connecting pipe area at outlet C
$d_A$	Connecting pipe diameter at outlet A
$d_B$	Connecting pipe diameter at outlet B
$d_C$	Connecting pipe diameter at outlet C
$q_A$	Flow rate through outlet A
$q_B$	Flow rate through outlet B
$q_C$	Flow rate through outlet C
$V$	Instantaneous fluid volume
$V_0$	Initial fluid volume
$A$	Tank cross-sectional area
$t$	Simulation time

For a tank with a variable cross-sectional area, the relationship between fluid level and volume is specified with the table lookup

$$H = f(V)$$

# Variable Head Three-Arm Tank

---

You have a choice of three interpolation methods and two extrapolation methods.

Connections A, B, and C are hydraulic conserving ports associated with the tank outlets. Connection V is a physical signal port. The flow rates are considered positive if fluid flows into the tank.

## **Warning**

**If fluid level becomes so low that some of the tank outlets get exposed, no warnings will be issued. The simulation will continue and pressure at exposed outlet(s) will be set to the pressurization pressure level. If this is not acceptable, MathWorks recommends that you employ the necessary control measures to guard against this situation in your models.**

# Variable Head Three-Arm Tank

## Dialog Box and Parameters

Block Parameters: Variable Head Three-Arm Tank

Variable Head Three-Arm Tank

This block represents a three-arm pressurized tank which stores fluid under a specified pressure and the pressurization remains constant regardless of volume change. The block accounts for the fluid level change caused by the volume variation and pressure loss in the connecting pipes that can be caused by a filter, fittings, or some other local resistances. 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 A, B, and C are the hydraulic conserving ports associated with the tank ports. Connection V is a physical signal port. The flow rates are considered positive if fluid flows into the tank.

Parameters

Initial fluid volume:	<input type="text" value="20"/>	<input type="text" value="l"/>
Pressurization:	<input type="text" value="0"/>	<input type="text" value="Pa"/>
Level/Volume relationship:	<input type="text" value="Linear"/>	
Tank cross-section area:	<input type="text" value="0.8"/>	<input type="text" value="m^2"/>
Port A pipeline diameter:	<input type="text" value="0.025"/>	<input type="text" value="m"/>
Port A pressure loss coefficient:	<input type="text" value="1.2"/>	
Port B pipeline diameter:	<input type="text" value="0.02"/>	<input type="text" value="m"/>
Port B height above port A:	<input type="text" value="0.8"/>	<input type="text" value="m"/>
Port B pressure loss coefficient:	<input type="text" value="1.2"/>	
Port C pipeline diameter:	<input type="text" value="0.02"/>	<input type="text" value="m"/>
Port C height above port A:	<input type="text" value="0.8"/>	<input type="text" value="m"/>
Port C pressure loss coefficient:	<input type="text" value="1.2"/>	

OK Cancel Help Apply

# Variable Head Three-Arm Tank

**Block Parameters: Variable Head Three-Arm Tank**

Variable Head Three-Arm Tank

This block represents a three-arm pressurized tank which stores fluid under a specified pressure and the pressurization remains constant regardless of volume change. The block accounts for the fluid level change caused by the volume variation and pressure loss in the connecting pipes that can be caused by a filter, fittings, or some other local resistances. 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 A, B, and C are the hydraulic conserving ports associated with the tank ports. Connection V is a physical signal port. The flow rates are considered positive if fluid flows into the tank.

**Parameters**

Initial fluid volume:	<input type="text" value="20"/>	<input type="text" value="l"/>
Pressurization:	<input type="text" value="0"/>	<input type="text" value="Pa"/>
Level/Volume relationship:	<input type="text" value="Table-specified"/>	
Tabulated fluid volumes:	<input type="text" value="1574 0.0623 0.066 0.0688 0.0707 0.072 0.0727 ]"/>	<input type="text" value="m^3"/>
Tabulated fluid levels:	<input type="text" value="0.12 0.14 0.16 0.18 0.2 0.22 0.24 0.26 0.28 0.3 ]"/>	<input type="text" value="m"/>
Port A pipeline diameter:	<input type="text" value="0.025"/>	<input type="text" value="m"/>
Port A pressure loss coefficient:	<input type="text" value="1.2"/>	
Port B pipeline diameter:	<input type="text" value="0.02"/>	<input type="text" value="m"/>
Port B height above port A:	<input type="text" value="0.8"/>	<input type="text" value="m"/>
Port B pressure loss coefficient:	<input type="text" value="1.2"/>	
Port C pipeline diameter:	<input type="text" value="0.02"/>	<input type="text" value="m"/>
Port C height above port A:	<input type="text" value="0.8"/>	<input type="text" value="m"/>
Port C pressure loss coefficient:	<input type="text" value="1.2"/>	
Interpolation method:	<input type="text" value="Linear"/>	
Extrapolation method:	<input type="text" value="From last 2 points"/>	

## Initial fluid volume

The initial volume of fluid in the tank. This parameter must be greater than zero. The default value is 20 l.

## Pressurization

Gage pressure acting on the surface of the fluid in the tank. It can be created by a gas cushion, membrane, bladder, or piston, as in bootstrap reservoirs. This parameter must be greater than or equal to zero. The default value is 0, which corresponds to a tank connected to atmosphere.

## Level/Volume relationship

Select one of the following block parameterization options:

- **Linear** — Provide a value for the tank cross-sectional area. The level is assumed to be linearly dependent on the fluid volume. This is the default method.
- **Table-specified** — Provide tabulated data of fluid volumes and fluid levels. The level is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.

## Tank cross-section area

The cross-sectional area of the tank. This parameter must be greater than zero. The default value is  $0.8 \text{ m}^2$ . This parameter is used if **Level/Volume relationship** is set to **Linear**.

## Tabulated fluid volumes

Specify the vector of input values for fluid volume as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in  $\text{m}^3$ , are [0 0.0028 0.0065 0.0114 0.0176 0.0252 0.0344 0.0436 0.0512 0.0574 0.0623 0.066 0.0688 0.0707 0.072 0.0727]. This parameter is used if **Level/Volume relationship** is set to **Table-specified**.

## Tabulated fluid levels

Specify the vector of fluid levels as a one-dimensional array. The fluid levels vector must be of the same size as the fluid volumes vector. The default values, in meters, are [0 0.02 0.04 0.06

# Variable Head Three-Arm Tank

---

0.08 0.1 0.12 0.14 0.16 0.18 0.2 0.22 0.24 0.26 0.28 0.3]. This parameter is used if **Level/Volume relationship** is set to **Table-specified**.

## Port A pipeline diameter

The diameter of the connecting pipe at port A. This parameter must be greater than zero. The default value is 0.025 m.

## Port A pressure loss coefficient

The value of the pressure loss coefficient, to account for pressure loss in the connecting pipe at port A. This parameter must be greater than zero. The default value is 1.2.

For reasons of computational robustness, the loss is computed with the equation similar to that used in the Fixed Orifice block:

$$q_A = \frac{1}{\sqrt{K}} A_A \sqrt{\frac{2}{\rho} p_{lossA}}$$

The Critical Reynolds number is set to 15.

## Port B pipeline diameter

The diameter of the connecting pipe at port B. This parameter must be greater than zero. The default value is 0.02 m.

## Port B height above port A

The elevation of port B above port A. If port A is higher than port B, enter a negative value. The default value is 0.8 m.

## Port B pressure loss coefficient

The value of the pressure loss coefficient, to account for pressure loss in the connecting pipe at port B. The loss is computed with the equation similar to the one given for port A. This parameter must be greater than zero. The default value is 1.2.

## Port C pipeline diameter

The diameter of the connecting pipe at port C. This parameter must be greater than zero. The default value is 0.02 m.

## **Port C height above port A**

The elevation of port C above port A. If port A is higher than port C, enter a negative value. The default value is 0.8 m.

## **Port C pressure loss coefficient**

The value of the pressure loss coefficient, to account for pressure loss in the connecting pipe at port C. The loss is computed with the equation similar to the one given for port A. This parameter must be greater than zero. The default value is 1.2.

## **Interpolation method**

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- **Linear** — Uses a linear interpolation function.
- **Cubic** — Uses the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP).
- **Spline** — Uses the cubic spline interpolation algorithm.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Level/Volume relationship** is set to **Table-specified**.

## **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

# Variable Head Three-Arm Tank

---

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 **Level/Volume relationship** is set to **Table-specified**.

## Ports

The block has the following ports:

- A  
Hydraulic conserving port associated with the tank outlet A.
- B  
Hydraulic conserving port associated with the tank outlet B.
- C  
Hydraulic conserving port associated with the tank outlet C.
- V  
Physical signal port that outputs the volume of fluid in the tank.

## See Also

Constant Head Tank  
Reservoir  
Variable Head Tank  
Variable Head Two-Arm Tank



# Variable Head Two-Arm Tank

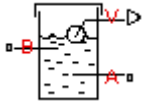
## Purpose

Tank with two hydraulic ports, constant pressurization, and volume-dependent fluid level

## Library

Low-Pressure Blocks

## Description



The Variable Head Two-Arm Tank block represents a two-arm pressurized tank, in which fluid is stored under a specified pressure. The pressurization remains constant regardless of volume change. The block accounts for the fluid level change caused by the volume variation, as well as for pressure loss in the connecting pipes that can be caused by a filter, fittings, or some other local resistance. The loss is specified with the pressure loss coefficients. The block computes the volume of fluid in the tank and exports it outside through the physical signal port V.

The pressure at the tank outlets is computed with the following equations:

$$p_A = p_{elevA} - p_{lossA} + p_{pr}$$

$$p_B = p_{elevB} - p_{lossB} + p_{pr}$$

$$p_{elevA} = \rho g H$$

$$p_{elevB} = \rho g (H - h_{BA})$$

$$p_{lossA} = K_A \frac{\rho}{2A_A^2} q_A |q_A|$$

$$p_{lossB} = K_B \frac{\rho}{2A_B^2} q_B |q_B|$$

$$A_A = \frac{\pi d_A^2}{4}$$

# Variable Head Two-Arm Tank

---

$$A_B = \frac{\pi d_B^2}{4}$$

$$H = \begin{cases} \frac{V}{A} & \text{for constant-area tank} \\ f(V) & \text{for table-specified tank} \end{cases}$$

$$V = V_0 + q \bar{t}$$

where

$p_A$	Pressure at the tank outlet A
$p_B$	Pressure at the tank outlet B
$p_{elevA}$	Pressure due to fluid level at outlet A
$p_{elevB}$	Pressure due to fluid level at outlet B
$p_{lossA}$	Pressure loss in the connecting pipe A
$p_{lossB}$	Pressure loss in the connecting pipe B
$p_{pr}$	Pressurization
$\rho$	Fluid density
$g$	Acceleration of gravity
$H$	Fluid level with respect to outlet A
$h_{AB}$	Elevation of outlet B with respect to outlet A
$K_A$	Pressure loss coefficient at outlet A
$K_B$	Pressure loss coefficient at outlet B
$A_A$	Connecting pipe area at outlet A
$A_B$	Connecting pipe area at outlet B
$d_A$	Connecting pipe diameter at outlet A
$d_B$	Connecting pipe diameter at outlet B

# Variable Head Two-Arm Tank

---

$q_A$	Flow rate through outlet A
$q_B$	Flow rate through outlet B
$V$	Instantaneous fluid volume
$V_0$	Initial fluid volume
$A$	Tank cross-sectional area
$t$	Simulation time

For a tank with a variable cross-sectional area, the relationship between fluid level and volume is specified with the table lookup

$$H = f(V)$$

You have a choice of three interpolation methods and two extrapolation methods.

Connections A and B are hydraulic conserving ports associated with the tank outlets. Connection V is a physical signal port. The flow rates are considered positive if fluid flows into the tank.

## **Warning**

**If fluid level becomes so low that one or both of the tank outlets get exposed, no warnings will be issued. The simulation will continue and pressure at exposed outlet(s) will be set to the pressurization pressure level. If this is not acceptable, MathWorks recommends that you employ the necessary control measures to guard against this situation in your models.**

# Variable Head Two-Arm Tank

## Dialog Box and Parameters

**Block Parameters: Variable Head Two-Arm Tank**

Variable Head Two-Arm Tank

This block represents a two-arm pressurized tank in which fluid is stored under a specified pressure and the pressurization remains constant regardless of volume change. The block accounts for the fluid level change caused by the volume variation and pressure loss in the connecting pipes that can be caused by a filter, fittings, or some other local resistances. 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 A and B are the hydraulic conserving ports associated with the tank ports. Connection V is a physical signal port. The flow rates are considered positive if fluid flows into the tank.

**Parameters**

Initial fluid volume:	<input type="text" value="20"/>	<input type="text" value="l"/>
Pressurization:	<input type="text" value="0"/>	<input type="text" value="Pa"/>
Level/Volume relationship:	<input type="text" value="Linear"/>	
Tank cross-section area:	<input type="text" value="0.8"/>	<input type="text" value="m^2"/>
Port A pipeline diameter:	<input type="text" value="0.025"/>	<input type="text" value="m"/>
Port A pressure loss coefficient:	<input type="text" value="1.2"/>	
Port B pipeline diameter:	<input type="text" value="0.02"/>	<input type="text" value="m"/>
Port B height above port A:	<input type="text" value="0.8"/>	<input type="text" value="m"/>
Port B pressure loss coefficient:	<input type="text" value="1.2"/>	

OK Cancel Help Apply

# Variable Head Two-Arm Tank

**Block Parameters: Variable Head Two-Arm Tank**

Variable Head Two-Arm Tank

This block represents a two-arm pressurized tank in which fluid is stored under a specified pressure and the pressurization remains constant regardless of volume change. The block accounts for the fluid level change caused by the volume variation and pressure loss in the connecting pipes that can be caused by a filter, fittings, or some other local resistances. 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 A and B are the hydraulic conserving ports associated with the tank ports. Connection V is a physical signal port. The flow rates are considered positive if fluid flows into the tank.

**Parameters**

Initial fluid volume: 20 l

Pressurization: 0 Pa

Level/Volume relationship: Table-specified

Tabulated fluid volumes: [574 0.0623 0.066 0.0688 0.0707 0.072 0.0727 ] m<sup>3</sup>

Tabulated fluid levels: [0.12 0.14 0.16 0.18 0.2 0.22 0.24 0.26 0.28 0.3 ] m

Port A pipeline diameter: 0.025 m

Port A pressure loss coefficient: 1.2

Port B pipeline diameter: 0.02 m

Port B height above port A: 0.8 m

Port B pressure loss coefficient: 1.2

Interpolation method: Linear

Extrapolation method: From last 2 points

OK Cancel Help Apply

## Initial fluid volume

The initial volume of fluid in the tank. This parameter must be greater than zero. The default value is 20 l.

## Pressurization

Gage pressure acting on the surface of the fluid in the tank. It can be created by a gas cushion, membrane, bladder, or piston, as in bootstrap reservoirs. This parameter must be greater than

# Variable Head Two-Arm Tank

---

or equal to zero. The default value is 0, which corresponds to a tank connected to atmosphere.

## Level/Volume relationship

Select one of the following block parameterization options:

- **Linear** — Provide a value for the tank cross-sectional area. The level is assumed to be linearly dependent on the fluid volume. This is the default method.
- **Table-specified** — Provide tabulated data of fluid volumes and fluid levels. The level is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.

## Tank cross-section area

The cross-sectional area of the tank. This parameter must be greater than zero. The default value is  $0.8 \text{ m}^2$ . This parameter is used if **Level/Volume relationship** is set to **Linear**.

## Tabulated fluid volumes

Specify the vector of input values for fluid volume as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in  $\text{m}^3$ , are [0 0.0028 0.0065 0.0114 0.0176 0.0252 0.0344 0.0436 0.0512 0.0574 0.0623 0.066 0.0688 0.0707 0.072 0.0727]. This parameter is used if **Level/Volume relationship** is set to **Table-specified**.

## Tabulated fluid levels

Specify the vector of fluid levels as a one-dimensional array. The fluid levels vector must be of the same size as the fluid volumes vector. The default values, in meters, are [0 0.02 0.04 0.06 0.08 0.1 0.12 0.14 0.16 0.18 0.2 0.22 0.24 0.26 0.28 0.3]. This parameter is used if **Level/Volume relationship** is set to **Table-specified**.

## Port A pipeline diameter

The diameter of the connecting pipe at port A. This parameter must be greater than zero. The default value is 0.025 m.

## Port A pressure loss coefficient

The value of the pressure loss coefficient, to account for pressure loss in the connecting pipe at port A. This parameter must be greater than zero. The default value is 1.2.

For reasons of computational robustness, the loss is computed with the equation similar to that used in the Fixed Orifice block:

$$q_A = \frac{1}{\sqrt{K}} A_A \sqrt{\frac{2}{\rho} p_{lossA}}$$

The Critical Reynolds number is set to 15.

## Port B pipeline diameter

The diameter of the connecting pipe at port B. This parameter must be greater than zero. The default value is 0.02 m.

## Port B height above port A

The elevation of port B above port A. If port A is higher than port B, enter a negative value. The default value is 0.8 m.

## Port B pressure loss coefficient

The value of the pressure loss coefficient, to account for pressure loss in the connecting pipe at port B. The loss is computed with the equation similar to the one given for port A. This parameter must be greater than zero. The default value is 1.2.

## Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- Linear — Uses a linear interpolation function.
- Cubic — Uses the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP).

# Variable Head Two-Arm Tank

---

- **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 **Level/Volume relationship** is set to **Table-specified**.

## 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 **Level/Volume relationship** is set to **Table-specified**.

## Ports

The block has the following ports:

- A  
Hydraulic conserving port associated with the tank outlet A.
- B  
Hydraulic conserving port associated with the tank outlet B.
- V  
Physical signal port that outputs the volume of fluid in the tank.



**See Also**

Constant Head Tank

Reservoir

Variable Head Tank

Variable Head Three-Arm Tank

# Variable Orifice

---

**Purpose** Generic hydraulic variable orifice

**Library** Orifices

**Description** The block represents a variable orifice of any type as a data-sheet-based model. Depending on data listed in the manufacturer's catalogs or data sheets for your particular orifice, you can choose one of the following model parameterization options:



- By maximum area and opening — Use this option if the data sheet provides only the orifice maximum area and the control member maximum stroke.
- By area vs. opening table — Use this option if the catalog or data sheet provides a table of the orifice passage area based on the control member displacement  $A=A(h)$ .
- By pressure-flow characteristic — Use this option if the catalog or data sheet provides a two-dimensional table of the pressure-flow characteristics  $q=q(p, h)$ .

In the first case, the passage area is assumed to be linearly dependent on the control member displacement, that is, the orifice is assumed to be closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. In the second case, the passage area is determined by one-dimensional interpolation from the table  $A=A(h)$ . In both cases, a small leakage area is assumed to exist even after the orifice is completely closed. Physically, it represents a possible clearance in the closed valve, but the main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation.

In the first and second cases, the model accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number ( $Re$ ) and comparing its value with the critical Reynolds number ( $Re_{cr}$ ). After the

area has been determined, the flow rate is computed according to the following equations:

$$q = \begin{cases} C_D \cdot A \sqrt{\frac{2}{\rho} |p|} \cdot \text{sign}(p) & \text{for } Re \geq Re_{cr} \\ 2C_{DL} \cdot A \frac{D_H}{v \cdot \rho} p & \text{for } Re < Re_{cr} \end{cases}$$

$$h = x_0 + x_{or}$$

$$A(h) = \begin{cases} h A_{max} / h_{max} + A_{leak} & \text{for } h > 0 \\ A_{leak} & \text{for } h \leq 0 \end{cases}$$

$$p = p_A - p_B$$

$$Re = \frac{q D_H}{A(h) v}$$

$$C_{DL} = \left( \frac{C_D}{\sqrt{Re_{cr}}} \right)^2$$

$$D_H = \sqrt{\frac{4A(h)}{\pi}}$$

where

- $q$  Flow rate
- $p$  Pressure differential
- $p_A, p_B$  Gauge pressures at the block terminals
- $C_D$  Flow discharge coefficient
- $A(h)$  Instantaneous orifice passage area

# Variable Orifice

---

$A_{max}$	Orifice maximum area
$h_{max}$	Control member maximum displacement
$x_0$	Initial opening
$x$	Control member displacement from initial position
$h$	Orifice opening
$or$	Orifice orientation indicator. The variable assumes +1 value if the control member displacement in the globally assigned positive direction opens the orifice, and -1 if positive motion decreases the opening.
$\rho$	Fluid density
$D_H$	Instantaneous orifice hydraulic diameter
$\nu$	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 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

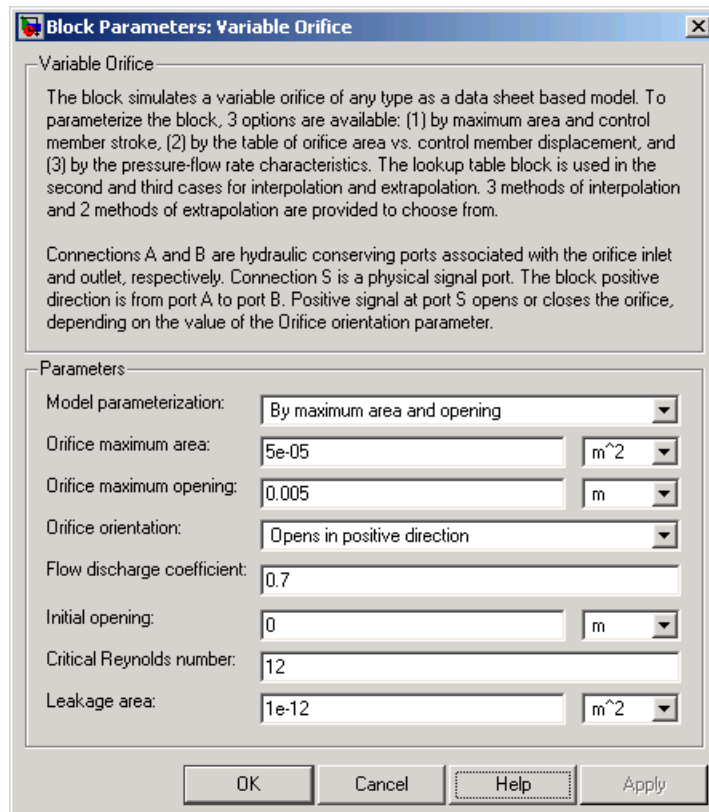
port S opens or closes the orifice depending on the value of the orifice orientation indicator.

## Basic Assumptions and Limitations

- Fluid inertia is not taken into account.
- For orifices specified by the passage area (the first two parameterization options), the transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at  $Re=Re_{cr}$ .
- For orifices specified by pressure-flow characteristics (the third parameterization option), the model does not explicitly account for the flow regime or leakage flow rate, because the tabulated data is assumed to account for these characteristics.

# Variable Orifice

## Dialog Box and Parameters



**Block Parameters: Variable Orifice**

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 <sup>2</sup>
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 <sup>2</sup>

OK Cancel Help Apply

**Block Parameters: Variable Orifice**

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:

Tabulated orifice openings:

Tabulated orifice area:

Interpolation method:

Extrapolation method:

Orifice orientation:

Flow discharge coefficient:

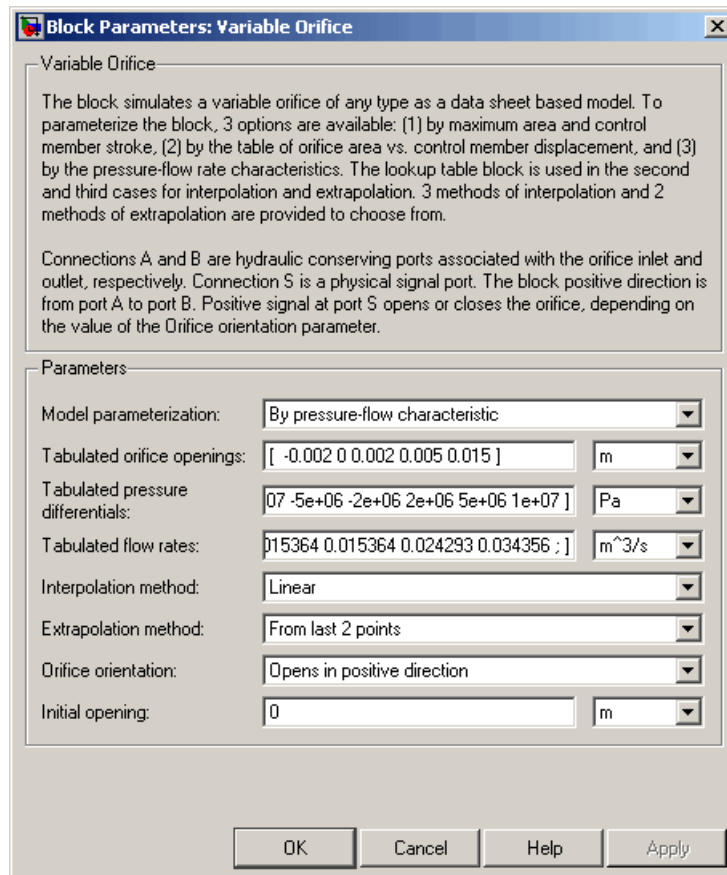
Initial opening:

Critical Reynolds number:

Leakage area:

OK Cancel Help Apply

# Variable Orifice



## Model parameterization

Select one of the following methods for specifying the orifice:

- By maximum area and opening — Provide values for the maximum orifice area and the maximum orifice opening. The passage area is linearly dependent on the control member displacement, that is, the orifice is closed at the initial position of the control member (zero displacement), and the maximum



opening takes place at the maximum displacement. This is the default method.

- **By area vs. opening table** — Provide tabulated data of orifice openings and corresponding orifice areas. The passage area is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.
- **By pressure-flow characteristic** — Provide tabulated data of orifice openings, pressure differentials, and corresponding flow rates. The flow rate is determined by two-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.

### **Orifice maximum area**

Specify the area of a fully opened orifice. The parameter value must be greater than zero. The default value is  $5e-5 \text{ m}^2$ . This parameter is used if **Model parameterization** is set to **By maximum area and opening**.

### **Orifice maximum opening**

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is  $5e-4 \text{ m}$ . This parameter is used if **Model parameterization** is set to **By maximum area and opening**.

### **Tabulated orifice openings**

Specify the vector of input values for orifice openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in meters, are  $[-0.002 \ 0 \ 0.002 \ 0.005 \ 0.015]$ . 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**

# Variable Orifice

---

characteristic, the **Tabulated orifice openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

## Tabulated orifice area

Specify the vector of orifice areas as a one-dimensional array. The vector must be of the same size as the orifice openings vector. All the values must be positive. The default values, in  $\text{m}^2$ , 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 pressure differential vector as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in Pa, are [-1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]. This parameter is used if **Model parameterization** is set to By pressure-flow characteristic.

## Tabulated flow rates

Specify the flow rates as an  $m$ -by- $n$  matrix, where  $m$  is the number of orifice openings and  $n$  is the number of pressure differentials. Each value in the matrix specifies flow rate taking place at a specific combination of orifice opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in  $\text{m}^3/\text{s}$ , 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 Polynomial (PCHIP). For two-dimensional table lookup (By `pressure-flow characteristic`), uses the bicubic interpolation algorithm.
- **Spline** — For one-dimensional table lookup (By `area vs. opening table`), uses the cubic spline interpolation algorithm. For two-dimensional table lookup (By `pressure-flow characteristic`), uses the bicubic spline interpolation algorithm.

For more information on interpolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

## Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- **From last 2 points** — Extrapolates using the linear method (regardless of the interpolation method specified), based on the last two output values at the appropriate end of the range. That is, the block uses the first and second specified output values if the input value is below the specified range, and the

# Variable Orifice

---

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 orifice. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause simulation to fail. Therefore, MathWorks recommends that you do not set this parameter to 0. The default value is  $1e-12 \text{ m}^2$ .

## Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Model parameterization**
- **Orifice orientation**
- **Interpolation method**
- **Extrapolation method**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Model parameterization** parameter at the time the model entered Restricted mode.

## Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

# Variable Orifice

---

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the orifice inlet.

B

Hydraulic conserving port associated with the orifice outlet.

S

Physical signal port to control spool displacement.

The flow rate is positive if fluid flows from port A to port B. Positive signal at the physical signal port S opens or closes the orifice depending on the value of the parameter **Orifice orientation**.

## Examples

The Hydraulic Flapper-Nozzle Amplifier example illustrates the use of the Variable Orifice block in hydraulic systems.

## See Also

Annular Orifice

Constant Area Hydraulic Orifice

Fixed Orifice

Orifice with Variable Area Round Holes

Orifice with Variable Area Slot

PS Lookup Table (1D)

PS Lookup Table (2D)

Variable Area Hydraulic Orifice

# Variable-Displacement Hydraulic Machine

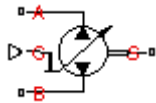
## Purpose

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:

$$q = D\omega - k_m \square q_L$$

$$T = D\square p + k_m \square T_{fr}$$

# Variable-Displacement Hydraulic Machine

---

$$D = \begin{cases} D_{\max} & x \\ x_{\max} & \\ D(x) & \end{cases}$$

$$p = p_A - p_B$$

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
$\omega$	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]

$$q_L = D \omega k_{L1} \left( \frac{p}{p_{nom}} \right)^{k_{LP}} \left( \frac{D}{D_{\max}} \right)^{k_{LD}} \left( \frac{\omega}{\omega_{nom}} \right)^{k_{L\omega}}$$



# Variable-Displacement Hydraulic Machine

$$T_{fr} = D_p k_{F1} \left( \frac{p}{p_{nom}} \right)^{k_{FP}} \left( \frac{D}{D_{max}} \right)^{k_{FD}} \left( \frac{\omega}{\omega_{nom}} \right)^{k_{F\omega}}$$

where

- $p_{nom}$  Nominal pressure
- $\omega_{nom}$  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

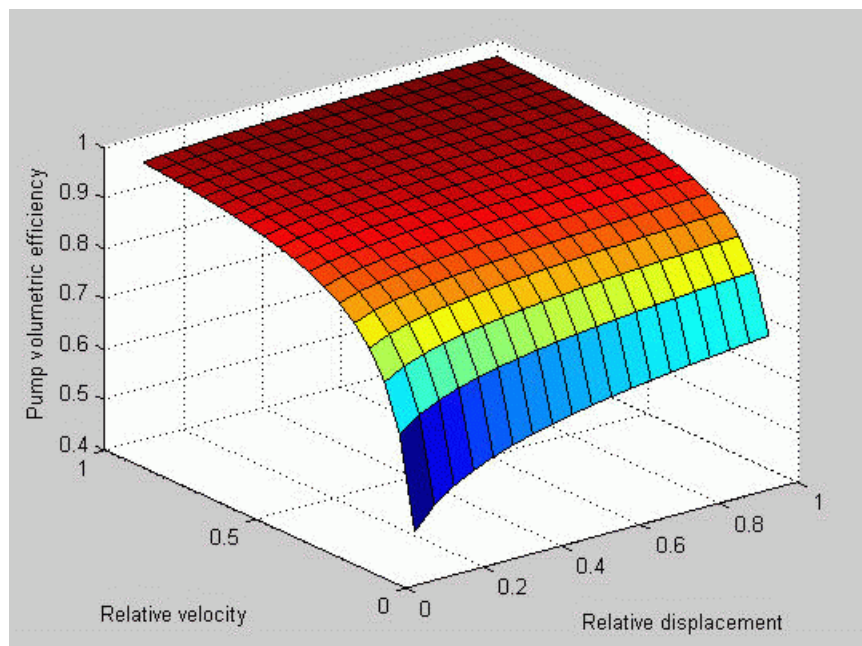
# Variable-Displacement Hydraulic Machine

$$\eta_{mp} = \frac{D_p}{D_p + T_{fr}} = \frac{1}{1 + k_{F1} \left( \frac{p}{p_{nom}} \right)^{k_{FP}} \left( \frac{D}{D_{max}} \right)^{k_{FD}} \left( \frac{\omega}{\omega_{nom}} \right)^{k_{F\omega}}}$$

$$\eta_{mm} = \frac{D_p - T_{fr}}{D_p} = 1 - k_{F1} \left( \frac{p}{p_{nom}} \right)^{k_{FP}} \left( \frac{D}{D_{max}} \right)^{k_{FD}} \left( \frac{\omega}{\omega_{nom}} \right)^{k_{F\omega}}$$

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.



# Variable-Displacement Hydraulic Machine

The procedure can be performed with the Optimization Toolbox software. For instance, the pump volumetric efficiency approximating coefficients can be found by solving the following problem:

$$\min_x F(x)$$

$$x = [k_{L1}, k_{LP}, k_{LD}, k_{L\omega}]$$

$$F(x) = \sum_i \sum_j \sum_k \left( \eta_{\text{exp}}(p_i, D_j, \omega_k) - \left( 1 - k_{L1} \left( \frac{p_i}{p_{\text{nom}}} \right)^{k_{LP}} \left( \frac{D_j}{D_{\text{max}}} \right)^{k_{LD}} \left( \frac{\omega_k}{\omega_{\text{nom}}} \right)^{k_{L\omega}} \right) \right)^2$$

where

- $i$  Number of experimental pressure points, from 1 to  $n$
- $j$  Number of experimental displacement points, from 1 to  $m$
- $k$  Number of experimental angular velocity points, from 1 to  $l$

Connections A and B are hydraulic conserving ports associated with the machine inlet and outlet, respectively. Connection S is a mechanical rotational conserving port associated with the machine shaft. Connection C is a physical signal port that controls machine displacement. The flow rate from port A to port B causes the shaft to rotate in positive direction, provided positive signal is applied to port C.

## Basic Assumptions and Limitations

- Fluid compressibility is neglected.
- No inertia on the machine shaft is considered.
- The model is applicable only for fluid and fluid temperature at which the approximating coefficients have been determined.
- Exercise extreme caution to not exceed the limits within which the approximating coefficients have been determined. The extrapolation could result in large errors.

# Variable-Displacement Hydraulic Machine

---

## **Dialog Box and Parameters**

The block dialog box contains three tabs:

- “Displacement” on page 1-679
- “Nominal Parameters” on page 1-683
- “Efficiencies” on page 1-685

# Variable-Displacement Hydraulic Machine

## Displacement

**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    Nominal parameters    Efficiencies

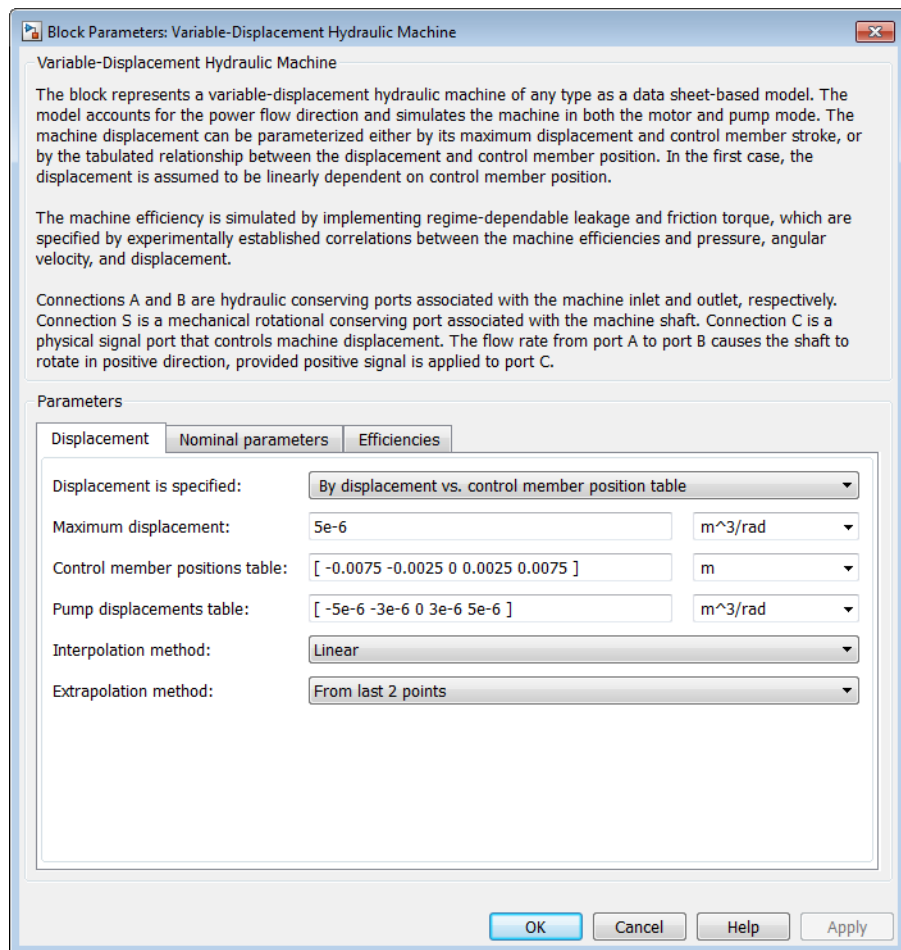
Displacement is specified:

Maximum displacement:

Maximum stroke:

OK    Cancel    Help    Apply

# Variable-Displacement Hydraulic Machine



## 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

# Variable-Displacement Hydraulic Machine

---

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<sup>3</sup>/rad.

## **Maximum stroke**

Maximum control member stroke. The default value is 0.005 m. This parameter is used if displacement is specified as **By maximum displacement and control member stroke**.

## **Control member positions table**

Specify the vector of input values for control member position as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in meters, are [-0.0075 -0.0025 0 0.0025 0.0075]. This parameter is used if displacement is specified as **By displacement vs. control member position table**.

## **Pump displacements table**

Specify the vector of machine displacements as a one-dimensional array. The machine displacements vector must be of the same size as the control member positions vector. The default values, in m<sup>3</sup>/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**.

# Variable-Displacement Hydraulic Machine

---

## Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- **Linear** — Uses a linear interpolation function.
- **Cubic** — Uses the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP).
- **Spline** — Uses the cubic spline interpolation algorithm.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if displacement is specified as `By displacement vs. control member position table`.

## Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

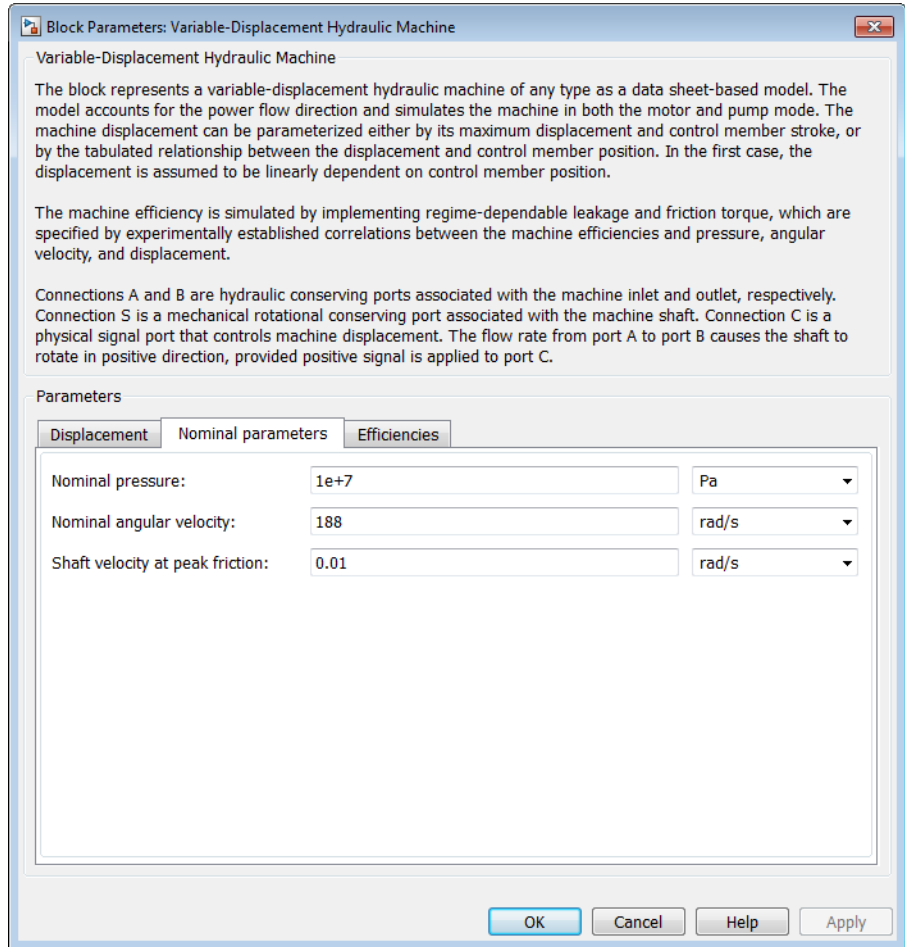
- **From last 2 points** — Extrapolates using the linear method (regardless of the interpolation method specified), based on the last two output values at the appropriate end of the range. That is, the block uses the first and second specified output values if the input value is below the specified range, and the two last specified output values if the input value is above the specified range.
- **From last point** — Uses the last specified output value at the appropriate end of the range. That is, the block uses the last specified output value for all input values greater than the last specified input argument, and the first specified output value for all input values less than the first specified input argument.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if displacement is specified as `By displacement vs. control member position table`.



# Variable-Displacement Hydraulic Machine

## Nominal Parameters



### Nominal pressure

Nominal pressure differential across the machine. The default value is  $1e7$  Pa.

# Variable-Displacement Hydraulic Machine

---

## **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  $T_{fr} \text{sign}(\omega)$ . To avoid discontinuity at  $\omega \rightarrow 0$ , the friction is defined as  $T_{fr} \tanh(4\omega / \omega_{\max})$ , where  $\omega_{\max}$  is a small velocity, representing the shaft velocity at peak friction, at which  $\tanh(4\omega / \omega_{\max})$  is equal to 0.999. The default value of  $\omega_{\max}$  is 0.01 rad/s.

# Variable-Displacement Hydraulic Machine

## Efficiencies

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	Nominal parameters	Efficiencies
		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

OK Cancel Help Apply

### Volumetric efficiency proportionality coefficient

Approximating coefficient  $k_{L1}$  in the block description preceding.  
The default value is 0.05.

# Variable-Displacement Hydraulic Machine

---

## **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  $k_L$  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  $k_F$  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

# Variable-Displacement Hydraulic Machine

---

**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

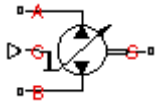
Variable-Displacement Motor  
Variable-Displacement Pump

# Variable-Displacement Motor

**Purpose** Variable-displacement bidirectional hydraulic motor

**Library** Pumps and Motors

## Description



The Variable-Displacement Motor block represents a variable-displacement bidirectional 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:

$$q = D\omega - k_{leak}p$$

$$T = Dp\eta_{mech}$$

$$D = \begin{cases} D_{max}x \\ x_{max} \\ D(x) \end{cases}$$

$$k_{leak} = k_{HP} / v_p$$

# Variable-Displacement Motor

$$k_{HP} = \frac{D_{max} \omega_{nom} (1 - \eta_V) v_{nom} \rho}{p_{nom}}$$

$$p = p_A - p_B$$

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
$\omega$	Output shaft angular velocity
$k_{leak}$	Leakage coefficient
$k_{HP}$	Hagen-Poiseuille coefficient
$\eta_V$	Motor volumetric efficiency
$\eta_{mech}$	Motor mechanical efficiency
$v$	Fluid kinematic viscosity
$\rho$	Fluid density
$p_{nom}$	Motor nominal pressure
$\omega_{nom}$	Motor nominal angular velocity
$v_{nom}$	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

# Variable-Displacement Motor

---

$$p = \frac{128\mu l}{\pi d^4} q_{leak} = \frac{\mu}{k_{HP}} q_{leak}$$

where

$q_{leak}$  Leakage flow

$d, l$  Geometric parameters of the leakage path

$\mu$  Fluid dynamic viscosity,  $\mu = \nu \rho$

The leakage flow at  $p = p_{nom}$  and  $\nu = \nu_{nom}$  can be determined from the catalog data

$$q_{leak} = D\omega_{nom} (1 - \eta_V)$$

which provides the formula to determine the Hagen-Poiseuille coefficient

$$k_{HP} = \frac{D\omega_{nom} (1 - \eta_V) \nu_{nom} \rho}{p_{nom}}$$

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**

- Fluid compressibility is neglected.
- No loading on the motor shaft, such as inertia, friction, spring, and so on, is considered.
- Leakage inside the motor is assumed to be linearly proportional to its pressure differential.

# Variable-Displacement Motor

## Dialog Box and Parameters

**Block Parameters: Variable-Displacement Motor**

Variable-Displacement Motor

This block represents a variable-displacement bidirectional 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 the motor displacement and control member position. In the first case, the displacement is assumed to be linearly dependent on control member position. The motor efficiency is determined based on 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 physical signal port that controls motor displacement. 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.

**Parameters**

Displacement specification:	By maximum displacement and control member stroke	
Maximum displacement:	5e-6	m <sup>3</sup> /rad
Maximum stroke:	0.005	m
Volumetric efficiency:	0.85	
Total efficiency:	0.75	
Nominal pressure:	100e5	Pa
Nominal angular velocity:	188	rad/s
Nominal kinematic viscosity:	18	cSt
Nominal fluid density:	900	kg/m <sup>3</sup>

OK Cancel Help Apply

# Variable-Displacement Motor

**Block Parameters: Variable-Displacement Motor**

**Variable-Displacement Motor**

This block represents a variable-displacement bidirectional 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 the motor displacement and control member position. In the first case, the displacement is assumed to be linearly dependent on control member position. The motor efficiency is determined based on 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 physical signal port that controls motor displacement. 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.

**Parameters**

Displacement specification:	By displacement vs. control member position table	
Maximum displacement:	5e-6	m <sup>3</sup> /rad
Control member positions table:	[- 7.5e-3, - 2.5e-3, 0, 2.5e-3, 7.5e-3]	m
Pump displacements table:	[- 5e-6, - 3e-6, 0, 3e-6, 5e-6]	m <sup>3</sup> /rad
Interpolation method:	Linear	
Extrapolation method:	From last 2 points	
Volumetric efficiency:	0.85	
Total efficiency:	0.75	
Nominal pressure:	100e5	Pa
Nominal angular velocity:	188	rad/s
Nominal kinematic viscosity:	18	cSt
Nominal fluid density:	900	kg/m <sup>3</sup>

OK Cancel Help Apply

# Variable-Displacement Motor

---

## Model parameterization

Select one of the following block parameterization options:

- By maximum displacement and control member stroke  
— Provide values for maximum motor displacement and maximum stroke. The displacement is assumed to be linearly dependent on the control member position. This is the default method.
- By displacement vs. control member position table  
— Provide tabulated data of motor displacements and control member positions. The displacement is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.

## Maximum displacement

Motor maximum displacement. The default value is  $5e-6$  m<sup>3</sup>/rad.

## Maximum stroke

Maximum control member stroke. The default value is 0.005 m. This parameter is visible 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 one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in meters, are [-0.0075 -0.0025 0 0.0025 0.0075]. This parameter is visible if **Model parameterization** is set to By displacement vs. control member position table.

## Pump displacements table

Specify the vector of motor displacements as a one-dimensional array. The motor displacements vector must be of the same size as the control member positions vector. The default values, in m<sup>3</sup>/rad, are [-5e-06 -3e-06 0 3e-06 5e-06]. This parameter

is visible 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 Polynomial (PCHIP).
- **Spline** — Uses the cubic spline interpolation algorithm.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is visible 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.

# Variable-Displacement Motor

---

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is visible 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.

## **Nominal fluid density**

Working fluid density, at which both the volumetric and total efficiencies are specified. The default value is 900 kg/m<sup>3</sup>.

## **Restricted Parameters**

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Model parameterization**
- **Interpolation method**

- **Extrapolation method**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Model parameterization** parameter at the time the model entered Restricted mode.

## **Global Parameters**

Parameter determined by the type of working fluid:

- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

## **Ports**

The block has the following ports:

A

Hydraulic conserving port associated with the 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

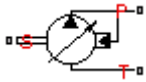
## Purpose

Hydraulic pump maintaining preset pressure at outlet by regulating its flow delivery

## Library

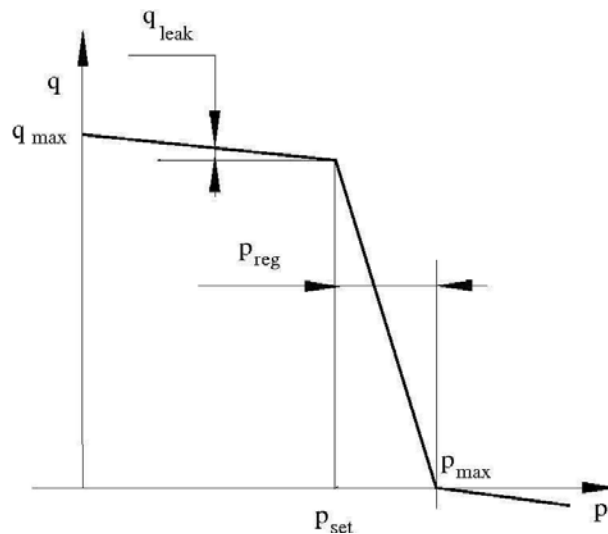
Pumps and Motors

## Description



The Variable-Displacement Pressure-Compensated Pump block represents a positive, variable-displacement, pressure-compensated pump of any type as a data-sheet-based model. The key parameters required to parameterize the block are the pump maximum displacement, regulation range, volumetric and total efficiencies, nominal pressure, and angular velocity. All these parameters are generally provided in the data sheets or catalogs.

The following figure shows the delivery-pressure characteristic of the pump.



The pump tries to maintain preset pressure at its outlet by adjusting its delivery flow in accordance with the system requirements. If pressure differential across the pump is less than the setting pressure, the pump outputs its maximum delivery corrected for internal leakage. After



# Variable-Displacement Pressure-Compensated Pump

the pressure setting has been reached, the output flow is regulated to maintain preset pressure by changing the pump's displacement. The displacement can be changed from its maximum value down to zero, depending upon system flow requirements. The pressure range between the preset pressure and the maximum pressure, at which the displacement is zero, is referred to as regulation range. The smaller the range, the higher the accuracy at which preset pressure is maintained. The range size also affects the pump stability, and decreasing the range generally causes stability to decrease.

The variable-displacement, pressure-compensated pump is represented with the following equations:

$$q = D\omega - k_{leak}p$$

$$T = Dp / \eta_{mech}$$

$$D = \begin{cases} D_{max} & \text{for } p \leq p_{set} \\ D_{max} - K(p - p_{set}) & \text{for } p_{set} < p < p_{max} \\ 0 & \text{for } p \geq p_{max} \end{cases}$$

$$p_{max} = p_{set} + p_{reg}$$

$$K = D_{max} / (p_{max} - p_{set})$$

$$k_{leak} = k_{HP} / v_p$$

$$k_{HP} = \frac{D_{max} \omega_{nom} (1 - \eta_V) v_{nom}}{p_{nom}}$$

$$p = p_p - p_T$$

where

# Variable-Displacement Pressure-Compensated Pump

---

$q$	Pump delivery
$p$	Pressure differential across the pump
$p_p, p_T$	Gauge pressures at the block terminals
$D$	Pump instantaneous displacement
$D_{max}$	Pump maximum displacement
$p_{set}$	Pump setting pressure
$p_{max}$	Maximum pressure, at which the pump displacement is zero
$T$	Torque at the pump driving shaft
$\omega$	Pump angular velocity
$k_{leak}$	Leakage coefficient
$k_{HP}$	Hagen-Poiseuille coefficient
$\eta_V$	Pump volumetric efficiency
$\eta_{mech}$	Pump mechanical efficiency
$\nu$	Fluid kinematic viscosity
$\rho$	Fluid density
$p_{nom}$	Pump nominal pressure
$\omega_{nom}$	Pump nominal angular velocity
$\nu_{nom}$	Nominal fluid kinematic viscosity

The leakage flow is determined based on the assumption that it is linearly proportional to the pressure differential across the pump and can be computed by using the Hagen-Poiseuille formula

$$p = \frac{128\mu l}{\pi d^4} q_{leak} = \frac{\mu}{k_{HP}} q_{leak}$$

where

# Variable-Displacement Pressure-Compensated Pump

- $q_{leak}$  Leakage flow  
 $d, l$  Geometric parameters of the leakage path  
 $\mu$  Fluid dynamic viscosity,  $\mu = \nu \rho$

The leakage flow at  $p = p_{nom}$  and  $v = v_{nom}$  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

$$k_{HP} = \frac{D\omega_{nom}(1 - \eta_V)\nu_{nom}\rho}{p_{nom}}$$

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

$$\eta_{mech} = \eta_{total} / \eta_V$$

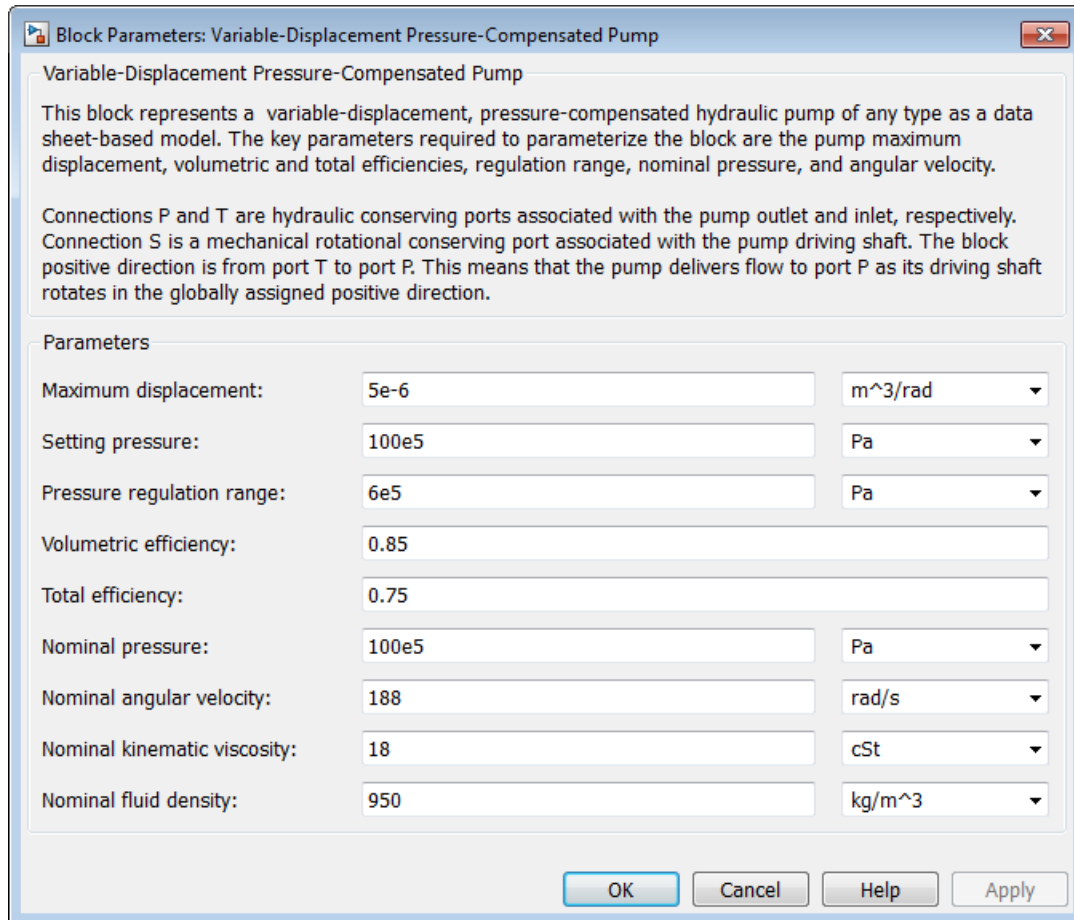
The block positive direction is from port T to port P. This means that the pump transfers fluid from T to P provided that the shaft S rotates in the positive direction. The pressure differential across the pump is determined as  $p = p_P - p_T$ .

## Basic Assumptions and Limitations

- Fluid compressibility is neglected.
- No loading on the pump shaft, such as inertia, friction, spring, and so on, is considered.
- Leakage inside the pump is assumed to be linearly proportional to its pressure differential.

# Variable-Displacement Pressure-Compensated Pump

## Dialog Box and Parameters



**Block Parameters: Variable-Displacement Pressure-Compensated Pump**

Variable-Displacement Pressure-Compensated Pump

This block represents a variable-displacement, pressure-compensated hydraulic pump of any type as a data sheet-based model. The key parameters required to parameterize the block are the pump maximum displacement, volumetric and total efficiencies, regulation range, nominal pressure, and angular velocity.

Connections P and T are hydraulic conserving ports associated with the pump outlet and inlet, respectively. Connection S is a mechanical rotational conserving port associated with the pump driving shaft. The block positive direction is from port T to port P. This means that the pump delivers flow to port P as its driving shaft rotates in the globally assigned positive direction.

**Parameters**

Maximum displacement:	5e-6	m <sup>3</sup> /rad
Setting pressure:	100e5	Pa
Pressure regulation range:	6e5	Pa
Volumetric efficiency:	0.85	
Total efficiency:	0.75	
Nominal pressure:	100e5	Pa
Nominal angular velocity:	188	rad/s
Nominal kinematic viscosity:	18	cSt
Nominal fluid density:	950	kg/m <sup>3</sup>

OK Cancel Help Apply

### Maximum displacement

Pump displacement. The default value is 5e-6 m<sup>3</sup>/rad.

# Variable-Displacement Pressure-Compensated Pump

---

## Setting pressure

Pump pressure setting. The default value is  $1e7$  Pa.

## Pressure regulation range

Pressure range required to change the pump displacement from its maximum to zero. The default value is  $6e5$  Pa.

## Volumetric efficiency

Pump volumetric efficiency specified at nominal pressure, angular velocity, and fluid viscosity. The default value is  $0.85$ .

## Total efficiency

Pump total efficiency, which is determined as a ratio between the hydraulic power at the pump outlet and mechanical power at the driving shaft at nominal pressure, angular velocity, and fluid viscosity. The default value is  $0.75$ .

## Nominal pressure

Pressure differential across the pump, at which both the volumetric and total efficiencies are specified. The default value is  $1e7$  Pa.

## Nominal angular velocity

Angular velocity of the driving shaft, at which both the volumetric and total efficiencies are specified. The default value is  $188$  rad/s.

## Nominal kinematic viscosity

Working fluid kinematic viscosity, at which both the volumetric and total efficiencies are specified. The default value is  $18$  cSt.

## Nominal fluid density

Working fluid density, at which both the volumetric and total efficiencies are specified. The default value is  $900$  kg/m<sup>3</sup>.

## Global Parameters

Parameter determined by the type of working fluid:

- Fluid kinematic viscosity

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

# Variable-Displacement Pressure-Compensated Pump

---

## Ports

The block has the following ports:

T

Hydraulic conserving port associated with the pump suction, or inlet.

P

Hydraulic conserving port associated with the pump outlet.

S

Mechanical rotational conserving port associated with the pump driving shaft.

## Examples

The Closed-Loop Electrohydraulic Actuator with Proportional Valve example illustrates the use of the Variable-Displacement Pressure-Compensated Pump block in hydraulic systems.

## See Also

Centrifugal Pump

Fixed-Displacement Pump

Variable-Displacement Pump

## Purpose

Variable-displacement bidirectional hydraulic pump

## Library

Pumps and Motors

## Description



The Variable-Displacement Pump block represents a variable-displacement bidirectional 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:

$$q = D\omega - k_{leak}p$$

$$T = Dp / \eta_{mech}$$

$$D = \begin{cases} D_{max}x \\ x_{max} \\ D(x) \end{cases}$$

$$k_{leak} = k_{HP} / v_p$$

# Variable-Displacement Pump

---

$$k_{HP} = \frac{D_{max} \omega_{nom} (1 - \eta_V) \nu_{nom} \rho}{p_{nom}}$$

$$p = p_P - p_T$$

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
$\omega$	Pump angular velocity
$k_{leak}$	Leakage coefficient
$k_{HP}$	Hagen-Poiseuille coefficient
$\eta_V$	Pump volumetric efficiency
$\eta_{mech}$	Pump mechanical efficiency
$\nu$	Fluid kinematic viscosity
$\rho$	Fluid density
$p_{nom}$	Pump nominal pressure
$\omega_{nom}$	Pump nominal angular velocity
$\nu_{nom}$	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
- $\mu$  Fluid dynamic viscosity,  $\mu = \nu \rho$

The leakage flow at  $p = p_{nom}$  and  $v = v_{nom}$  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

$$k_{HP} = \frac{D\omega_{nom} (1 - \eta_V) \nu_{nom} \rho}{p_{nom}}$$

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.

# Variable-Displacement Pump

---

## **Basic Assumptions and Limitations**

- Fluid compressibility is neglected.
- No loading on the pump shaft, such as inertia, friction, spring, and so on, is considered.
- Leakage inside the pump is assumed to be linearly proportional to its pressure differential.

# Variable-Displacement Pump

## Dialog Box and Parameters

**Block Parameters: Variable-Displacement Pump**

**Variable-Displacement Pump**

This block represents a variable-displacement bidirectional 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.

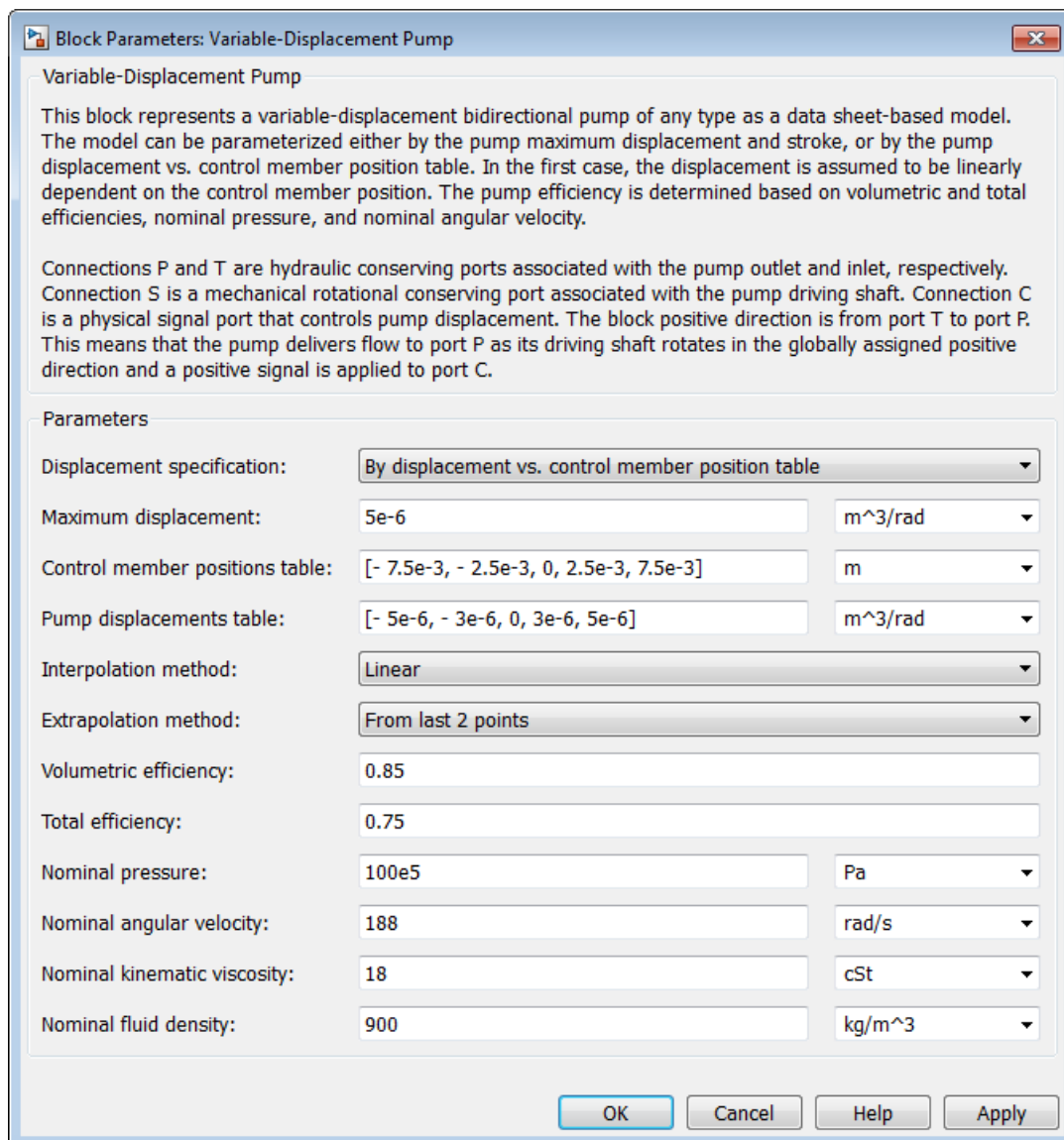
Connections P and T are hydraulic conserving ports associated with the pump outlet and inlet, respectively. Connection S is a mechanical rotational conserving port associated with the pump driving shaft. Connection C is a physical signal port that controls pump displacement. The block positive direction 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**

Displacement specification:	By maximum displacement and control member stroke	
Maximum displacement:	5e-6	m <sup>3</sup> /rad
Maximum stroke:	0.005	m
Volumetric efficiency:	0.85	
Total efficiency:	0.75	
Nominal pressure:	100e5	Pa
Nominal angular velocity:	188	rad/s
Nominal kinematic viscosity:	18	cSt
Nominal fluid density:	900	kg/m <sup>3</sup>

OK Cancel Help Apply

# Variable-Displacement Pump



## Model parameterization

Select one of the following block parameterization options:

- **By maximum displacement and control member stroke** — Provide values for maximum pump displacement and maximum control member stroke. The displacement is assumed to be linearly dependent on the control member position. This is the default method.
- **By displacement vs. control member position table** — Provide tabulated data of pump displacements and control member positions. The displacement is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.

## Maximum displacement

Pump maximum displacement. The default value is  $5e-6$  m<sup>3</sup>/rad.

## Maximum stroke

Maximum control member stroke. The default value is 0.005 m. This parameter is visible 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 one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for cubic or spline interpolation. The default values, in meters, are [-0.0075 -0.0025 0 0.0025 0.0075]. This parameter is visible if **Model parameterization** is set to **By displacement vs. control member position table**.

## Pump displacements table

Specify the vector of pump displacements as a one-dimensional array. The pump displacements vector must be of the same size as the control member positions vector. The default values, in m<sup>3</sup>/rad, are [-5e-06 -3e-06 0 3e-06 5e-06]. This parameter

# Variable-Displacement Pump

---

is visible 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 Polynomial (PCHIP).
- **Spline** — Uses the cubic spline interpolation algorithm.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is visible 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 visible 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.

## **Nominal fluid density**

Working fluid density, at which both the volumetric and total efficiencies are specified. The default value is 900 kg/m<sup>3</sup>.

## **Restricted Parameters**

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Model parameterization**
- **Interpolation method**

# Variable-Displacement Pump

---

- **Extrapolation method**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Model parameterization** parameter at the time the model entered Restricted mode.

## **Global Parameters**

Parameter determined by the type of working fluid:

- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

## **Ports**

The block has the following ports:

T

Hydraulic conserving port associated with the pump suction, or inlet.

P

Hydraulic conserving port associated with the pump outlet.

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



# Bibliography

---

- [1] Cornell, C.R. “Dynamic Simulation of a Hydrostatically Propelled Vehicle.” SAE paper 811253, 1981, p. 22.
- [2] “Flow of Fluids Through Valves, Fittings, and Pipe.” Crane Valves North America. Technical Paper No. 410.
- [3] Hicks, T. G. and T. W. Edwards. *Pump Application Engineering*, New York: McGraw-Hill, 1971.
- [4] Idelchik, I. E. *Handbook of Hydraulic Resistance*. New York: CRC Begell House, 1994.
- [5] Karassic, I. J., J. P. Messina, P. Cooper, and C. C. Heald. *Pump Handbook*. Fourth edition. New York: McGraw-Hill, 2008.
- [6] Keller, George R. *Hydraulic System Analysis*. Seattle, WA: Hydraulics & Pneumatics Magazine, 1970.
- [7] Manring, Noah D. *Hydraulic Control Systems*. Hoboken, NJ: John Wiley & Sons, 2005.
- [8] White, F. M. *Viscous Fluid Flow*. New York: McGraw-Hill, 1991.
- [9] Yeapple, F. *Fluid Power Design Handbook*, New York: Marcel Dekker, 1995.

**across variables**

Variables that are measured with a gauge connected in parallel to an element.

**add-on products**

Products in the Physical Modeling family that use Simscape platform and, as a result, share common functionality such as physical units management, editing modes, and so on. SimHydraulics software is one of the Simscape add-on products.

**behavioral block implementation model**

A block that is implemented based on its physical behavior, described by a system of mathematical equations. An example of a behavioral block implementation is the Variable Orifice block.

**conserving ports**

Bidirectional hydraulic or mechanical ports that represent physical connections and relate physical variables based on the Physical Network approach.

**constructional block implementation model**

A block that is constructed out of other blocks, connected in a certain way. An example of a constructional block implementation is the 4-Way Directional Valve block, which is constructed based on four Variable Orifice blocks.

**data-sheet-based model**

A block with a set of parameters determined by data that is usually listed in the manufacturer's catalogs or data sheets.

**globally assigned positive direction**

Direction considered positive for a model diagram.

**nonrestricted parameters**

Parameters that are available for modification when you open a model in Restricted mode. Usually, these are the block parameters with plain numerical values, such as **Pipe internal diameter** or **Resistance**

**area.** Information on restricted and nonrestricted parameters is listed in block reference pages.

**physical signal ports**

Unidirectional ports (inports and outports) transferring signals that use an internal physical modeling engine for computations.

**restricted parameters**

Parameters that are not available for modification when you open a model in Restricted mode. You have to be in Full mode to modify them. Usually, these are the block parameterization options, such as **Pipe cross section type** or **Interpolation method**. Information on restricted and nonrestricted parameters is listed in block reference pages.

**through variables**

Variables that are measured with a gauge connected in series to an element.

## Symbols and Numerics

2-Position Valve Actuator block 1-2  
2-Way Directional Valve block 1-8  
3-Position Valve Actuator block 1-19  
3-Way Directional Valve block 1-25  
4-Way Directional Valve A block 1-56  
4-Way Directional Valve B block 1-62  
4-Way Directional Valve block 1-37  
4-Way Directional Valve C block 1-69  
4-Way Directional Valve D block 1-76  
4-Way Directional Valve E block 1-82  
4-Way Directional Valve F block 1-88  
4-Way Directional Valve G block 1-95  
4-Way Directional Valve H block 1-101  
4-Way Directional Valve K block 1-108  
4-Way Ideal Valve block 1-114  
6-Way Directional Valve A block 1-117

## A

accumulators  
    gas-charged 1-293  
    spring-loaded 1-601  
Angle Sensor block 1-124  
Annular Orifice block 1-127

## B

Ball Valve block 1-132  
Ball Valve with Conical Seat block 1-139

## C

Cartridge Valve Insert block 1-145  
Cartridge Valve Insert with Conical Seat  
    block 1-157  
Centrifugal Force in Rotating Cylinder  
    block 1-162  
Centrifugal Pump block 1-166  
Check Valve block 1-185

Constant Head Tank block 1-191  
Counterbalance Valve block 1-195  
Cylinder Cushion block 1-211  
Cylinder Friction block 1-223

## D

Double-Acting Hydraulic Cylinder (Simple)  
    block 1-237  
Double-Acting Hydraulic Cylinder block 1-228  
Double-Acting Rotary Actuator block 1-243  
Double-Acting Servo Cylinder block 1-250

## E

Elbow block 1-255

## F

Fixed Orifice block 1-261  
Fixed Orifice Empirical block 1-265  
Fixed Orifice with Fluid Inertia block 1-269  
Fixed-Displacement Motor (External Efficiencies)  
    block 1-273  
Fixed-Displacement Pump block 1-277  
Flow Divider block 1-283

## G

Gas-Charged Accumulator block 1-293  
Gate Valve block 1-297  
Gradual Area Change block 1-303

## H

Hydraulic 4-Port Cartridge Valve Actuator  
    block 1-312  
Hydraulic Cartridge Valve Actuator block 1-318  
Hydraulic Double-Acting Valve Actuator  
    block 1-322  
Hydraulic Fluid block 1-331

Hydraulic Motor block 1-339  
Hydraulic Pipe LP block 1-353  
Hydraulic Pipe LP with Variable Elevation  
block 1-364  
Hydraulic Pipeline block 1-345  
Hydraulic Single-Acting Valve Actuator  
block 1-373  
Hydraulically Operated Remote Control Valve  
block 1-378

**J**

Jet Pump block 1-386  
Journal Bearing Pressure-Fed block 1-392

**L**

Local Resistance block 1-396

**N**

Needle Valve block 1-405

**O**

Orifice with Variable Area Round Holes  
block 1-411  
Orifice with Variable Area Slot block 1-418

**P**

Partially Filled Vertical Pipe LP block 1-423  
Pilot-Operated Check Valve block 1-436  
Pipe Bend block 1-443  
Pneumo-Hydraulic Actuator block 1-451  
Poppet Valve block 1-458  
Porting Plate Variable Orifice block 1-464  
Pressure Compensator block 1-493  
Pressure Reducing 3-Way Valve block 1-500  
Pressure Reducing Valve block 1-508  
Pressure Relief Valve block 1-513

Pressure-Compensated 3-Way Flow Control  
Valve block 1-474  
Pressure-Compensated Flow Control Valve  
block 1-484  
Proportional and Servo-Valve Actuator  
block 1-519

**R**

Reservoir block 1-524  
Resistive Pipe LP block 1-527  
Resistive Pipe LP with Variable Elevation  
block 1-535  
Rotating Pipe block 1-543

**S**

Segmented Pipe LP block 1-556  
Segmented Pipeline block 1-547  
Shuttle Valve block 1-566  
Single-Acting Hydraulic Cylinder (Simple)  
block 1-580  
Single-Acting Hydraulic Cylinder block 1-573  
Single-Acting Rotary Actuator block 1-586  
Spool Orifice Hydraulic Force block 1-592  
Spring-Loaded Accumulator block 1-601  
Sudden Area Change block 1-605  
Swash Plate block 1-613

**T**

T-junction block 1-622  
terminology Glossary-1

**V**

Valve Actuator block 1-626  
Valve Hydraulic Force block 1-628  
Variable Head Tank block 1-634  
Variable Head Three-Arm Tank block 1-641  
Variable Head Two-Arm Tank block 1-651

Variable Orifice block 1-660  
Variable-Displacement Hydraulic Machine  
block 1-673  
Variable-Displacement Motor block 1-688

Variable-Displacement Pressure-Compensated  
Pump block 1-698  
Variable-Displacement Pump block 1-705