

# SimHydraulics

**For Use with Simulink®**

- Modeling
- Simulation
- Implementation

Reference

*Version 1*



## How to Contact The MathWorks



www.mathworks.com  
comp.soft-sys.matlab  
www.mathworks.com/contact\_TS.html

Web  
Newsgroup  
Technical Support



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

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



508-647-7000 (Phone)



508-647-7001 (Fax)



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

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

*SimHydraulics Reference*

© COPYRIGHT 2006 by The MathWorks, Inc.

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

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

### Trademarks

MATLAB, Simulink, Stateflow, Handle Graphics, Real-Time Workshop, and xPC TargetBox are registered trademarks, and SimBiology, SimEvents, and SimHydraulics are trademarks of The MathWorks, Inc.

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

### Patents

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

**Revision History**

March 2006 Online only  
September 2006 Online only

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



## Blocks — By Category

**1**

---

<b>Foundation</b> .....	<b>1-2</b>
Hydraulic .....	<b>1-2</b>
Mechanical .....	<b>1-4</b>
Physical Signals .....	<b>1-6</b>
<b>SimHydraulics</b> .....	<b>1-8</b>
Accumulators .....	<b>1-8</b>
Hydraulic Cylinders .....	<b>1-8</b>
Hydraulic Utilities .....	<b>1-9</b>
Local Hydraulic Resistances .....	<b>1-9</b>
Pipelines .....	<b>1-9</b>
Pumps and Motors .....	<b>1-10</b>
Valves .....	<b>1-10</b>
<b>Utilities</b> .....	<b>1-12</b>

## Blocks — Alphabetical List

**2**

## Commands — Alphabetical List

**3**

---

## Index



# Blocks — By Category

---

---

**Note** SimHydraulics uses the Physical Networks library as its main library. To open the Physical Networks library, use the Simulink Library Browser or type `pn_lib` in the MATLAB Command Window.

---

Foundation (p. 1-2)

Basic hydraulic, mechanical, and physical signal blocks

SimHydraulics (p. 1-8)

Advanced hydraulic diagram blocks, such as valves, cylinders, pipelines, pumps, and accumulators

Utilities (p. 1-12)

Essential environment blocks for creating Physical Networks models

## Foundation

Hydraulic (p. 1-2)

Basic hydraulic diagram blocks, such as orifices, chambers, sensors and sources, and utilities

Mechanical (p. 1-4)

Mechanical elements for rotational and translational motion, as well as mechanical sensors and sources

Physical Signals (p. 1-6)

Blocks for transmitting physical control signals

## Hydraulic

Hydraulic Elements (p. 1-2)

Hydraulic building blocks, such as orifices, chambers, and hydro-mechanical converters

Hydraulic Sensors and Sources (p. 1-3)

Hydraulic sensors and sources

Hydraulic Utilities (p. 1-4)

Basic hydraulic environment blocks, such as custom hydraulic fluid

## Hydraulic Elements

Annular Orifice

Simulate hydraulic variable orifice created by circular tube and round insert

Constant Volume Chamber

Simulate hydraulic capacity of constant volume

Fixed Orifice

Simulate hydraulic orifice with constant cross-sectional area

Fluid Inertia

Simulate pressure differential across tube or channel due to change in fluid velocity



Hydraulic Reference	Simulate connection to atmospheric pressure
Linear Hydraulic Resistance	Simulate hydraulic pipeline with linear resistance losses
Orifice with Variable Area Round Holes	Simulate hydraulic variable orifice shaped as set of round holes drilled in sleeve
Orifice with Variable Area Slot	Simulate hydraulic variable orifice shaped as rectangular slot
Pressure Compensator	Simulate hydraulic pressure compensating valve
Resistive Tube	Simulate hydraulic pipeline which accounts for friction losses only
Translational Hydro-Mechanical Converter	Simulate single chamber of hydraulic cylinder as building block for various cylinder models
Variable Orifice	Simulate generic hydraulic variable orifice
Variable Volume Chamber	Simulate hydraulic capacity of variable volume with compressible fluid

## **Hydraulic Sensors and Sources**

Ideal Hydraulic Flow Rate Sensor	Simulate ideal flow meter
Ideal Hydraulic Flow Rate Source	Simulate ideal source of hydraulic energy, characterized by flow rate
Ideal Hydraulic Pressure Sensor	Simulate ideal pressure sensing device
Ideal Hydraulic Pressure Source	Simulate ideal source of hydraulic energy, characterized by pressure

## Hydraulic Utilities

Custom Hydraulic Fluid	Set working fluid properties by specifying parameter values
------------------------	-------------------------------------------------------------

## Mechanical

Mechanical Sensors and Sources (p. 1-4)	Mechanical sensors and sources
Mechanisms (p. 1-5)	Various simple mechanisms
Rotational Elements (p. 1-5)	Mechanical elements for rotational motion
Translational Elements (p. 1-6)	Mechanical elements for translational motion

## Mechanical Sensors and Sources

Ideal Angular Velocity Source	Simulate ideal angular velocity source in mechanical rotational systems
Ideal Force Sensor	Simulate force sensor in mechanical translational systems
Ideal Force Source	Simulate ideal source of mechanical energy that generates force proportional to the input signal
Ideal Rotational Motion Sensor	Simulate motion sensor in mechanical rotational systems
Ideal Torque Sensor	Simulate torque sensor in mechanical rotational systems
Ideal Torque Source	Simulate ideal source of mechanical energy that generates torque proportional to the input signal

Ideal Translational Motion Sensor	Simulate motion sensor in mechanical translational systems
Ideal Translational Velocity Source	Simulate ideal velocity source in mechanical translational systems

## **Mechanisms**

Gear Box	Simulate gear boxes in mechanical systems
Lever	Simulate lever in mechanical systems
Wheel and Axle	Simulate wheel and axle mechanism in mechanical systems

## **Rotational Elements**

Inertia	Simulate inertia in mechanical rotational systems
Mechanical Rotational Reference	Simulate reference for mechanical rotational ports
Rotational Damper	Simulate viscous damper in mechanical rotational systems
Rotational Friction	Simulate friction in contact between rotating bodies
Rotational Hard Stop	Simulate double-sided rotational hard stop
Rotational Spring	Simulate ideal spring in mechanical rotational systems

## **Translational Elements**

Mass	Simulate mass in mechanical translational systems
Mechanical Translational Reference	Simulate reference for mechanical translational ports
Translational Damper	Simulate viscous damper in mechanical translational systems
Translational Friction	Simulate friction in contact between moving bodies
Translational Hard Stop	Simulate double-sided translational hard stop
Translational Spring	Simulate ideal spring in mechanical translational systems

## **Physical Signals**

Functions (p. 1-6)	Perform math operations on physical signals
Linear Operators (p. 1-7)	Simulate continuous-time functions for physical signals
Lookup Tables (p. 1-7)	Perform one- and two-dimensional table lookup to generate physical signals
Nonlinear Operators (p. 1-7)	Simulate discontinuities, such as saturation or dead zone, for physical signals

## **Functions**

PS Add	Add two physical signal inputs
PS Divide	Compute simple division of two input physical signals

PS Gain	Multiply input physical signal by constant
PS Product	Multiply two physical signal inputs
PS Subtract	Compute simple subtraction of two input physical signals

### **Linear Operators**

PS Integrator	Integrate physical signal
---------------	---------------------------

### **Lookup Tables**

PS Lookup Table (1D)	Approximate one-dimensional function using specified lookup method
PS Lookup Table (2D)	Approximate two-dimensional function using specified lookup method

### **Nonlinear Operators**

PS Dead Zone	Provide region of zero output for physical signals
PS Saturation	Limit range of physical signal

## **SimHydraulics**

Accumulators (p. 1-8)	Hydraulic accumulators
Hydraulic Cylinders (p. 1-8)	Hydraulic cylinders
Hydraulic Utilities (p. 1-9)	Environment blocks, such as hydraulic fluid
Local Hydraulic Resistances (p. 1-9)	Various local hydraulic resistances
Pipelines (p. 1-9)	Hydraulic pipelines
Pumps and Motors (p. 1-10)	Hydraulic pumps and motors
Valves (p. 1-10)	Hydraulic valves

### **Accumulators**

Gas-Charged Accumulator	Simulate hydraulic accumulator with gas as compressible medium
Spring-Loaded Accumulator	Simulate hydraulic accumulator with spring used for energy storage

### **Hydraulic Cylinders**

Cylinder Friction	Simulate friction in hydraulic cylinders
Double-Acting Hydraulic Cylinder	Simulate hydraulic actuator exerting force in both directions
Single-Acting Hydraulic Cylinder	Simulate hydraulic actuator exerting force in one direction

## Hydraulic Utilities

Hydraulic Fluid	Set working fluid properties by selecting from list of predefined fluids
Reservoir	Simulate pressurized hydraulic reservoir

## Local Hydraulic Resistances

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

## Pipelines

Hydraulic Pipeline	Simulate hydraulic pipeline with resistive and fluid compressibility properties
Segmented Pipeline	Simulate hydraulic pipeline with resistive, fluid inertia, and fluid compressibility properties

## **Pumps and Motors**

Fixed-Displacement Pump	Simulate fixed-displacement hydraulic pump
Hydraulic Motor	Simulate fixed-displacement hydraulic motor
Variable-Displacement Motor	Simulate variable-displacement reversible hydraulic motor
Variable-Displacement Pressure-Compensated Pump	Simulate hydraulic pump maintaining preset pressure at outlet by regulating its flow delivery
Variable-Displacement Pump	Simulate variable-displacement reversible hydraulic pump

## **Valves**

Directional Valves (p. 1-10)	Hydraulic directional valves
Flow Control Valves (p. 1-11)	Hydraulic flow control valves
Pressure Control Valves (p. 1-11)	Hydraulic pressure control valves
Valve Actuators (p. 1-11)	Actuators for driving directional valves

## **Directional Valves**

2-Way Directional Valve	Simulate hydraulic continuous 2-way directional valve
3-Way Directional Valve	Simulate hydraulic continuous 3-way directional valve
4-Way Directional Valve	Simulate hydraulic continuous 4-way directional valve
Check Valve	Simulate hydraulic valve that allows flow in one direction only



Pilot-Operated Check Valve	Simulate hydraulic check valve that allows flow in one direction, but can be disabled by pilot pressure
Shuttle Valve	Simulate hydraulic valve that allows flow in one direction only

### **Flow Control Valves**

Ball Valve	Simulate hydraulic ball valve
Needle Valve	Simulate hydraulic needle valve
Poppet Valve	Simulate hydraulic poppet valve
Pressure-Compensated Flow Control Valve	Simulate hydraulic pressure compensating valve

### **Pressure Control Valves**

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

### **Valve Actuators**

2-Position Valve Actuator	Simulate actuator for two-position valves
3-Position Valve Actuator	Simulate actuator for three-position valves
Proportional and Servo-Valve Actuator	Simulate continuous valve driver with output proportional to input signal

## Utilities

Connection Port	Create Physical Modeling connector port for subsystem
PS-Simulink Converter	Convert physical signal into Simulink output signal
Simulink-PS Converter	Convert Simulink input signal into physical signal
Solver Configuration	Represent Physical Networks environment and solver configuration

# Blocks — Alphabetical List

---

# 2-Position Valve Actuator

---

**Purpose** Simulate actuator for two-position valves

**Library** Valve Actuators

**Description**

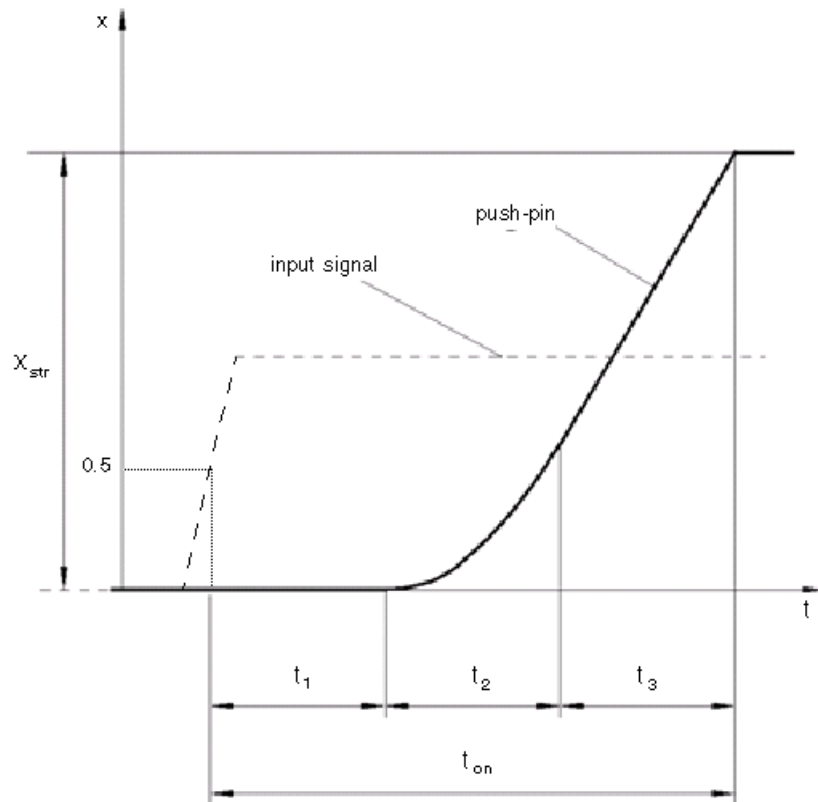


The 2-Position Valve Actuator block represents an actuator that you can use with directional valves to control their position. This actuator can drive a two-position valve. The block is developed as a data-sheet-based model and all its parameters are generally provided in catalogs or data sheets. The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

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

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

## 2-Position Valve Actuator



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

## 2-Position Valve Actuator

---

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

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

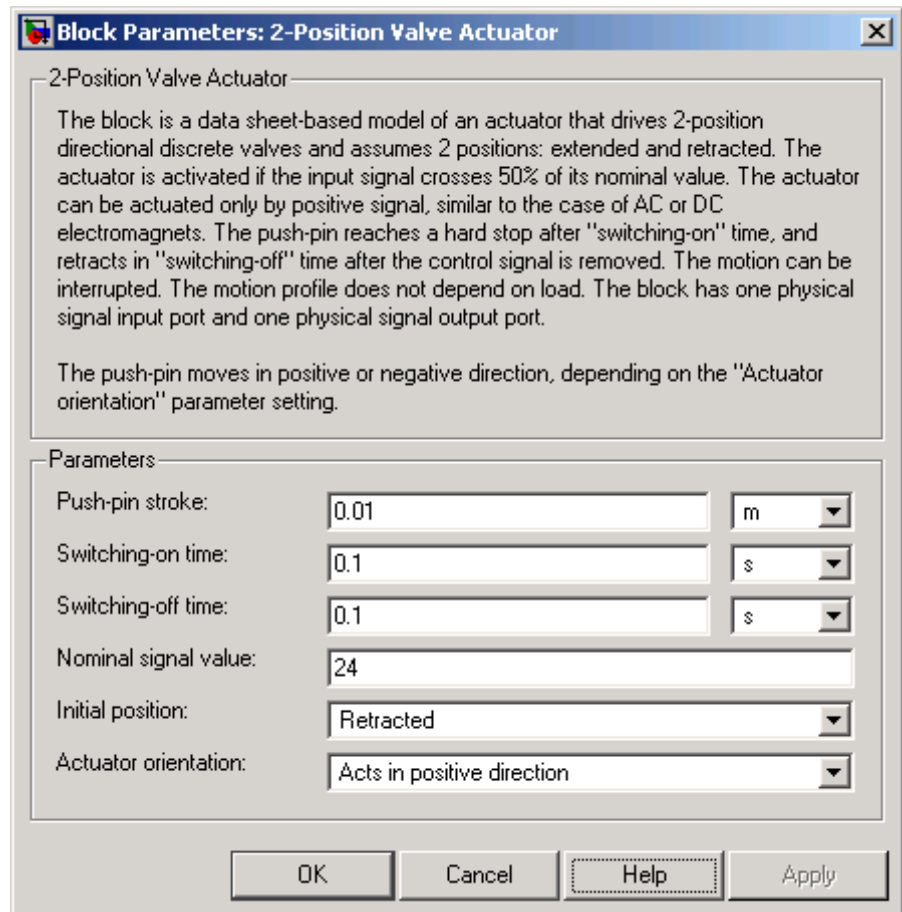
The push-pin is actuated only by positive signal, similar to the AC or DC electromagnets. The direction of push-pin motion is controlled by the **Actuator orientation** parameter, which can have one of two values: Acts in positive direction or Acts in negative direction.

### **Basic Assumptions and Limitations**

The model is based on the following assumption:

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

## Dialog Box and Parameters



### Push-pin stroke

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

### Switching-on time

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

# 2-Position Valve Actuator

---

## **Switching-off time**

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

## **Nominal signal value**

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

## **Initial position**

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

## **Actuator orientation**

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

## **Ports**

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

## **Examples**

In the 2-Position Valve Actuator demo (sh\_2\_pos\_valve\_actuator), the hydraulic circuit contains two actuators. The first one is set to start from the retracted position, while the second one starts from the extended position. Both actuators are driven with a Pulse Generator. The actuators start extending at 1 s, but the second actuator first retracts from 0.01 m to zero, since it was initially extended and there was no signal keeping it there.



In the Hydraulic Circuit with Single-Acting Cylinder demo (sh\_circuit\_sa\_cylinder), the 2-Position Valve Actuator block is used along with a 3-Way Directional Valve block to simulate an electrically operated 3-way directional valve.

### **See Also**

3-Position Valve Actuator

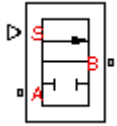
Proportional and Servo-Valve Actuator

# 2-Way Directional Valve

**Purpose** Simulate hydraulic continuous 2-way directional valve

**Library** Directional Valves

## Description



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

$$h = x_0 + x$$

where

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

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

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

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

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

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

# 2-Way Directional Valve

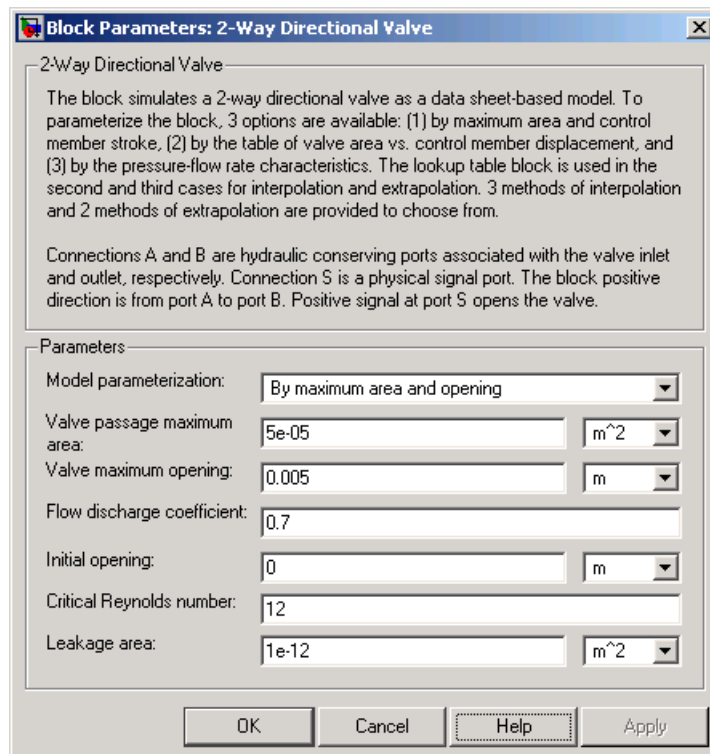
is determined as  $p = p_A - p_B$ . Positive signal at the physical signal port S opens the valve.

## Basic Assumptions and Limitations

The model is based on the following assumptions:

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

## Dialog Box and Parameters



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

2-Way Directional Valve

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

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

Parameters

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

OK Cancel Help Apply

# 2-Way Directional Valve

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

2-Way Directional Valve

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

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

Parameters

Model parameterization: By area vs. opening table

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

Tabulated valve passage area: [ 1e-12 4e-12 1e-05 1.02e-05 ] m<sup>2</sup>

Interpolation method: Linear

Extrapolation method: From last 2 points

Flow discharge coefficient: 0.7

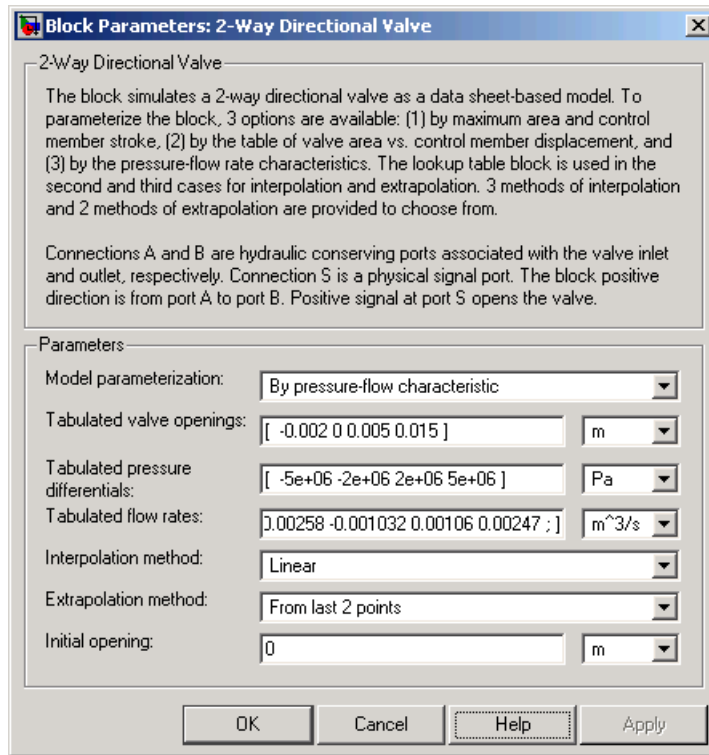
Initial opening: 0 m

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 tabulated 1-by- $m$  array. The input values vector must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. The default values, in meters, are  $[-2e-3, 0, 5e-3, 15e-3]$ . If **Model parameterization** is set to By area vs. opening table, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to By pressure-flow characteristic, the **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

### **Tabulated valve passage area**

Specify the vector of output values for valve passage area as a tabulated 1-by- $m$  array. The valve passage area vector

## 2-Way Directional Valve

---

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

### Tabulated pressure differentials

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

### Tabulated flow rates

Specify the output values for flow rates as a tabulated m-by-n matrix, defining the function values at the input grid vertices. Each value in the matrix specifies flow rate taking place at a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in  $m^3/s$ , are:

```
[ -1e-12,  -0.6e-13,   0.6e13,    0.1e12; ...  
  -0.8e-7, -0.4e-7,   0.4e7,     0.8e7; ...  
  -2.5e-3, -1.03e-3,  1.e-3,    2.4e-3; ...  
  -2.58e-3, -1.032e-3, 1.06e-3,  2.47e-3]
```

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

### Interpolation method

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

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



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

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

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

### **Extrapolation method**

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

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

## 2-Way Directional Valve

---

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

### **Flow discharge coefficient**

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

### **Initial opening**

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

### **Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

### **Leakage area**

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is  $1e-12 \text{ m}^2$ .

## **Global Parameters**

### **Fluid density**

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

## **Fluid kinematic viscosity**

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

## **Ports**

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

S

Physical signal port to control spool displacement.

## **Examples**

In the Hydraulic Closed-Loop Circuit with 2-Way Valve demo (sh\_closed\_loop\_circuit\_2\_way\_valve), the 2-Way Directional Valve block is used to control the position of a double-acting cylinder. At the start of simulation, the valve is open by 0.42 mm to make the circuit initial position as close as possible to its neutral position.

## **See Also**

3-Way Directional Valve

4-Way Directional Valve

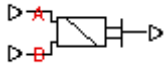
# 3-Position Valve Actuator

---

**Purpose** Simulate actuator for three-position valves

**Library** Valve Actuators

## Description

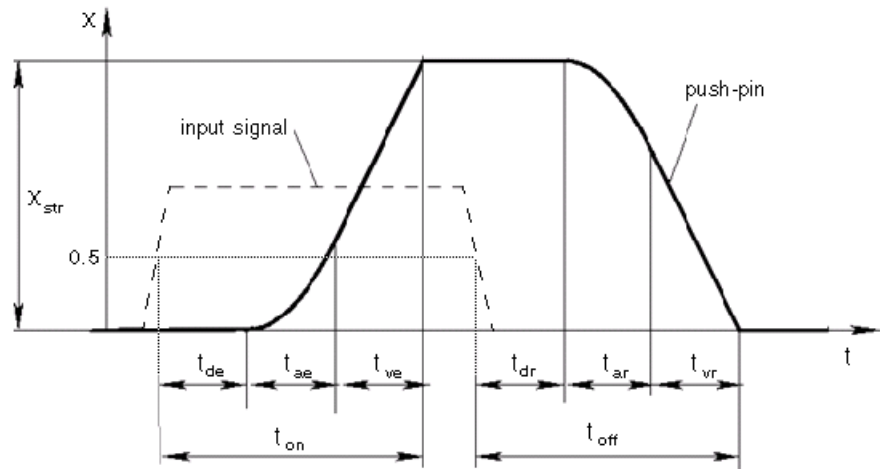


The 3-Position Valve Actuator block represents an actuator that you can use with directional valves to control their position. This actuator can drive a three-position valve. The block is developed as a data-sheet-based model and all its parameters are generally provided in catalogs or data sheets. The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

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

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

## 3-Position Valve Actuator



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

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

## 3-Position Valve Actuator

---

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

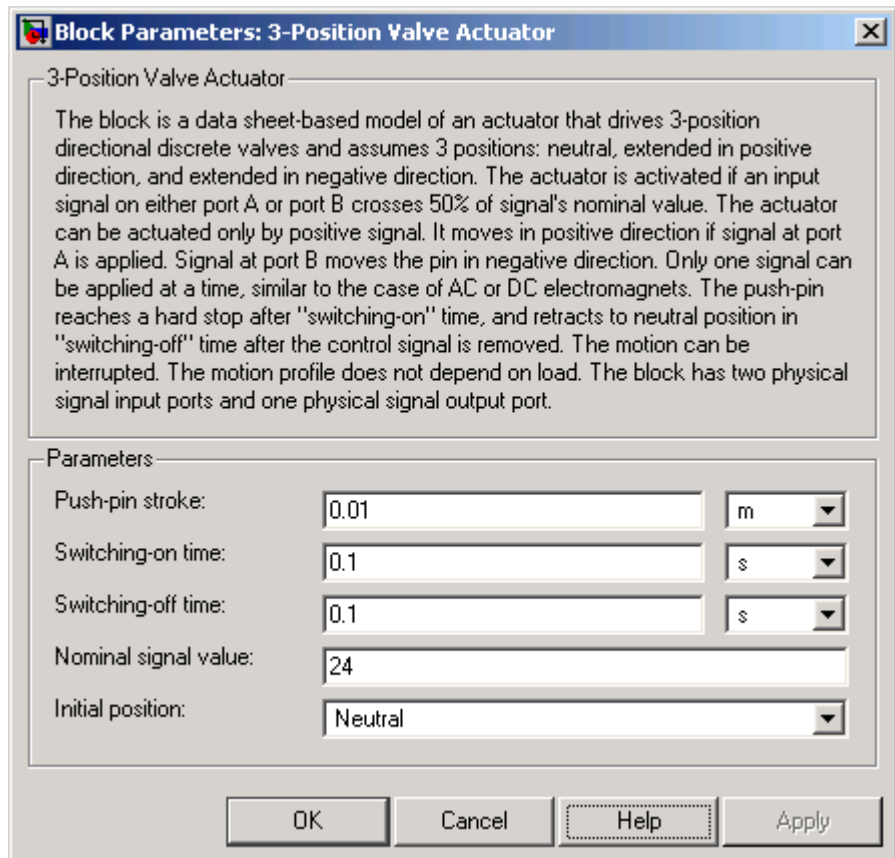
Only positive signals activate the actuator. In other words, negative signals at ports A and B have no effect on the actuator, which is similar to the behavior of electromagnetically controlled 3-position directional valves.

### **Basic Assumptions and Limitations**

The model is based on the following assumption:

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

## Dialog Box and Parameters



### Push-pin stroke

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

### Switching-on time

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

### Switching-off time

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

# 3-Position Valve Actuator

---

## Nominal signal value

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

## Initial position

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

## Ports

The block has the following ports:

A

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

B

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

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

## Examples

In the 3-Position Valve Actuator demo (`sh_3_pos_valve_actuator`), all three actuators are set to different strokes, switch-on and switch-off times, and initial positions. If the initial position is not `Neutral` and the control signal at the beginning of simulation equals zero, the push-pin starts moving towards neutral position, as the actuators A and C show in the demo.

## See Also

2-Position Valve Actuator

Proportional and Servo-Valve Actuator



# 3-Way Directional Valve

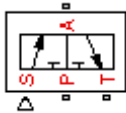
## Purpose

Simulate hydraulic continuous 3-way directional valve

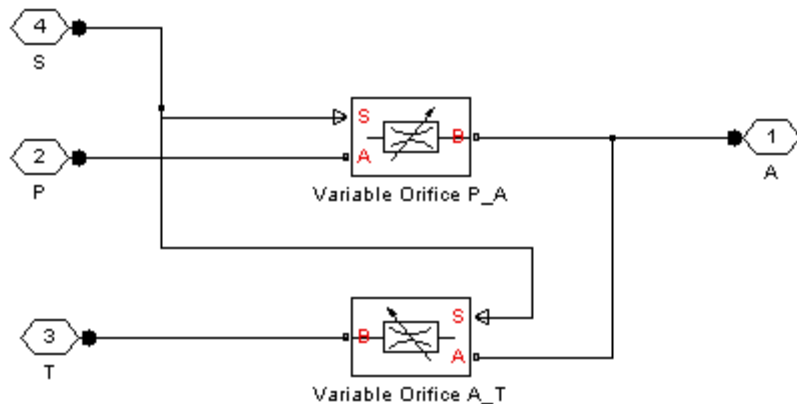
## Library

Directional Valves

## Description



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



One Variable Orifice block, called `orifice_PA`, is installed in the P-A path. The second Variable Orifice block, called `orifice_AT`, is installed in the A-T path. Both blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens `orifice_PA` and closes `orifice_AT`. As a result, the openings of the orifices are computed as follows:

## 3-Way Directional Valve

---

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

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

where

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

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

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

In the first case, the passage area is assumed to be linearly dependent on the control member displacement, that is, the orifice is assumed to be closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. In the second case, the passage area is determined by one-dimensional interpolation from the table  $A=A(h)$ . Flow rate is determined analytically, which additionally requires data such as flow discharge

coefficient, critical Reynolds number, and fluid density and viscosity. The computation accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number and comparing its value with the critical Reynolds number. See the Variable Orifice block reference page for details. In both cases, a small leakage area is assumed to exist even after the orifice is completely closed. Physically, it represents a possible clearance in the closed valve, but the main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation.

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

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

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

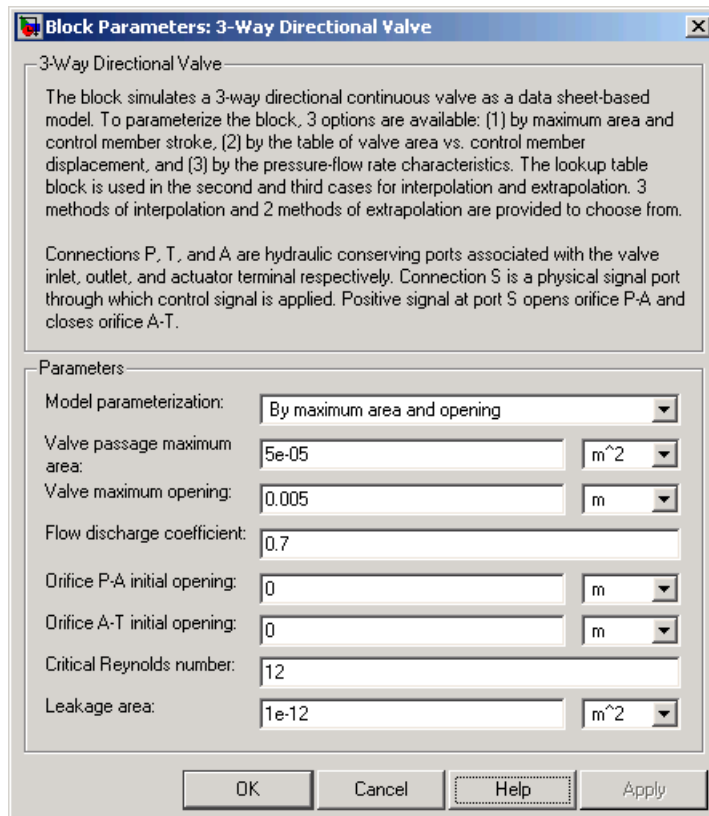
# 3-Way Directional Valve

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- Fluid inertia is not taken into account.
- Spool loading, such as inertia, spring, hydraulic forces, and so on, is not taken into account.
- 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: By area vs. opening table

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

Tabulated valve passage area: [ 1e-12 4e-12 1e-05 1.02e-05 ] m<sup>2</sup>

Interpolation method: Linear

Extrapolation method: From last 2 points

Flow discharge coefficient: 0.7

Orifice P-A initial opening: 0 m

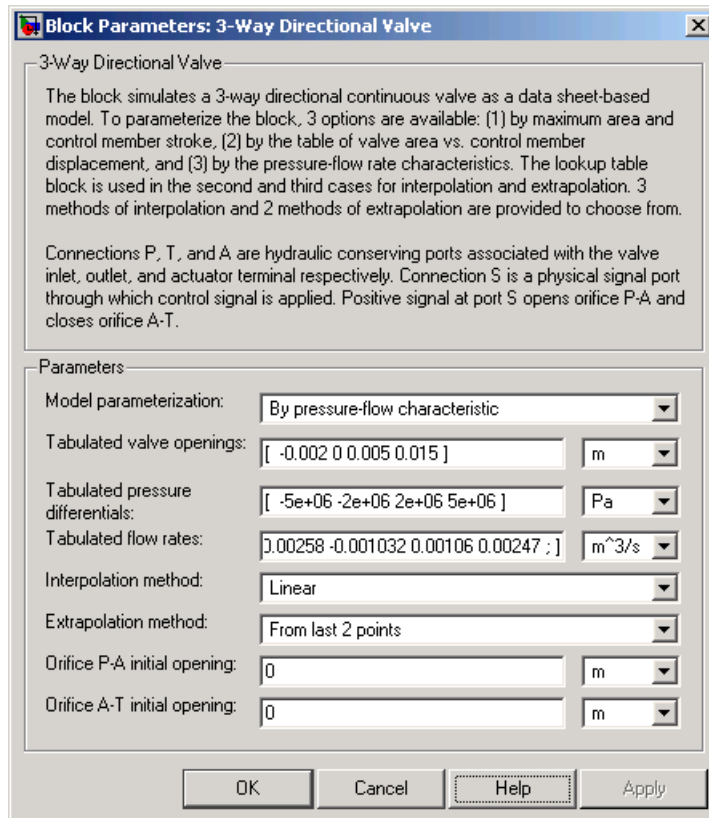
Orifice A-T initial opening: 0 m

Critical Reynolds number: 12

Leakage area: 1e-12 m<sup>2</sup>

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 tabulated 1-by-m array. The input values vector must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. The default values, in meters, are  $[-2e-3, 0, 5e-3, 15e-3]$ . If **Model parameterization** is set to By area vs. opening table, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to By pressure-flow characteristic, the **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

## 3-Way Directional Valve

---

### Tabulated valve passage area

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

### Tabulated pressure differentials

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

### Tabulated flow rates

Specify the output values for flow rates as a tabulated m-by-n matrix, defining the function values at the input grid vertices. Each value in the matrix specifies flow rate taking place at a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in  $m^3/s$ , are:

```
[ -1e-12,  -0.6e-13,   0.6e13,    0.1e12; ...  
  -0.8e-7, -0.4e-7,   0.4e7,     0.8e7; ...  
  -2.5e-3, -1.03e-3,  1.e-3,    2.4e-3; ...  
  -2.58e-3, -1.032e-3, 1.06e-3,  2.47e-3]
```

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

### Interpolation method

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

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



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

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

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

### **Extrapolation method**

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

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

## 3-Way Directional Valve

---

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

### **Flow discharge coefficient**

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

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

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

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

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

### **Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

### **Leakage area**

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is  $1e-12 \text{ m}^2$ .

## Global Parameters

### Fluid density

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

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

P

Hydraulic conserving port associated with the pressure supply line inlet.

T

Hydraulic conserving port associated with the return line connection.

A

Hydraulic conserving port associated with the actuator connection port.

S

Physical signal port to control spool displacement.

## Examples

The 3-Way Directional Valve block is demonstrated in the Hydraulic Circuit with 3-Way Valve and Differential Cylinder demo (sh\_circuit\_3\_way\_valve\_diff\_cylinder), where it is used to switch between a conventional and differential connection of the cylinder.

## See Also

2-Way Directional Valve

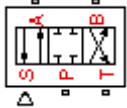
4-Way Directional Valve

# 4-Way Directional Valve

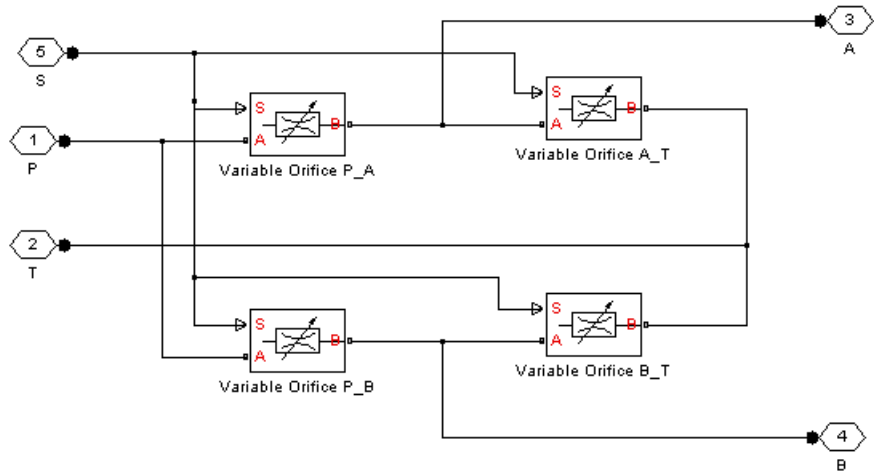
**Purpose** Simulate hydraulic continuous 4-way directional valve

**Library** Directional Valves

## Description



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



The Variable Orifice blocks are installed as follows: orifice\_PA is in the P-A path, orifice\_PB is in the P-B path, orifice\_AT is in the A-T path, and orifice\_BT is in the B-T path. All blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens orifice\_PA and

orifice\_BT and closes orifice\_PB and orifice\_AT. As a result, the openings of the orifices are computed as follows:

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

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

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

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

where

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

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

- By maximum area and opening — Use this option if the data sheet provides only the orifice maximum area and the control member maximum stroke.

## 4-Way Directional Valve

---

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

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

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

(openings and pressure differentials) must be strictly monotonically increasing. The vertices can be nonuniformly spaced. You have a choice of three interpolation methods and two extrapolation methods.

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

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

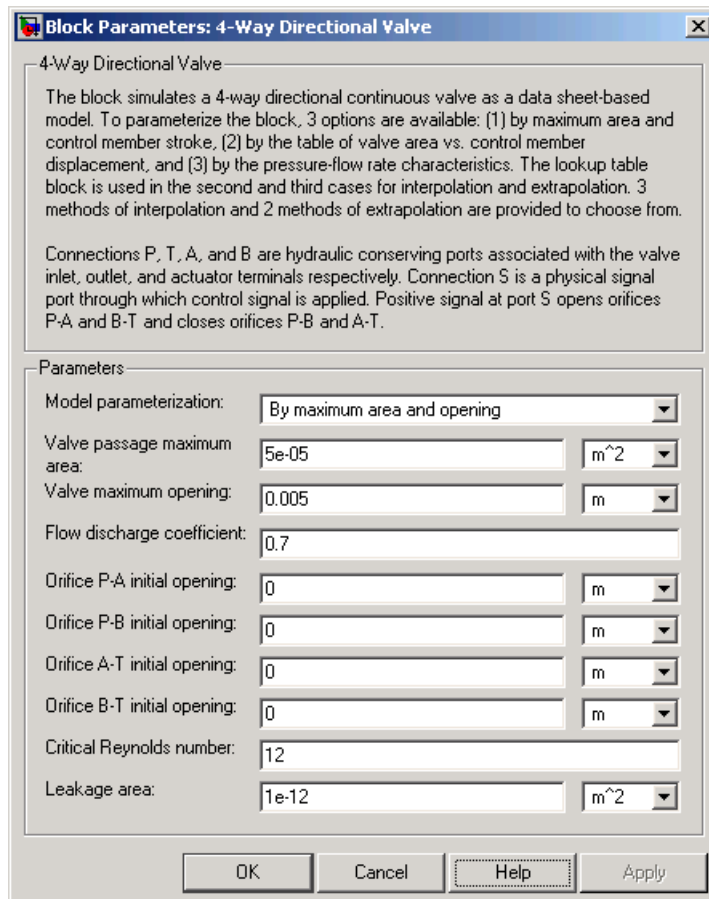
## **Basic Assumptions and Limitations**

The model is based on the following assumptions:

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

# 4-Way Directional Valve

## Dialog Box and Parameters





# 4-Way Directional Valve

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

4-Way Directional Valve

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

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

Parameters

Model parameterization: By area vs. opening table

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

Tabulated valve passage area: [ 1e-12 4e-12 1e-05 1.02e-05 ] m<sup>2</sup>

Interpolation method: Linear

Extrapolation method: From last 2 points

Flow discharge coefficient: 0.7

Orifice P-A initial opening: 0 m

Orifice P-B initial opening: 0 m

Orifice A-T initial opening: 0 m

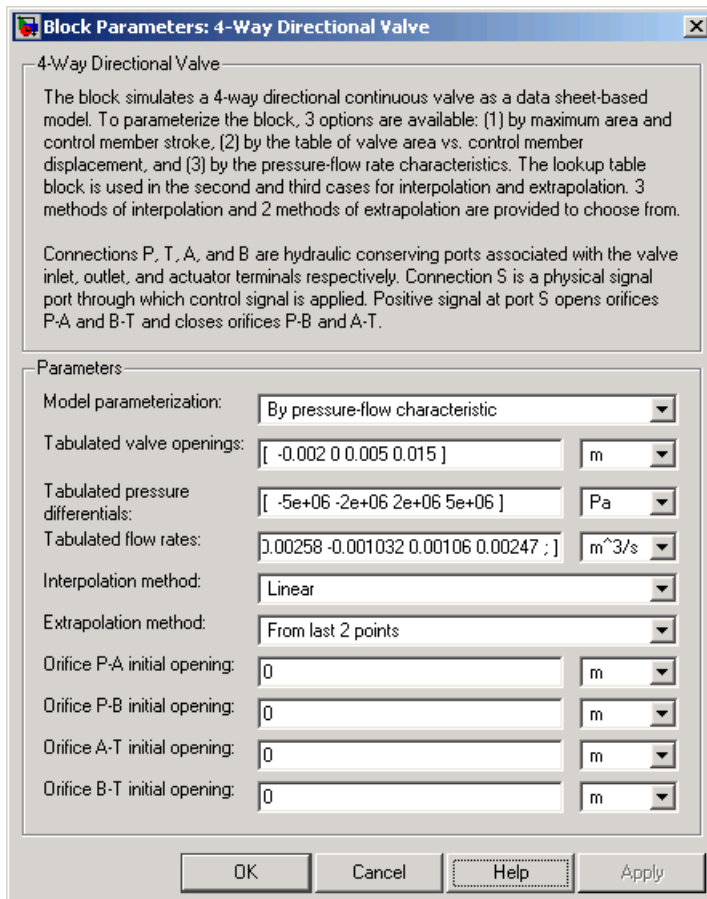
Orifice B-T initial opening: 0 m

Critical Reynolds number: 12

Leakage area: 1e-12 m<sup>2</sup>

OK Cancel Help Apply

# 4-Way Directional Valve



## Model parameterization

Select one of the following methods for specifying the valve:

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

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

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

### **Valve passage maximum area**

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

### **Valve maximum opening**

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

### **Tabulated valve openings**

Specify the vector of input values for valve openings as a tabulated 1-by-m array. The input values vector must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. The default values, in meters, are  $[-2e-3, 0, 5e-3, 15e-3]$ . If **Model parameterization** is set to **By area vs. opening table**, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to **By pressure-flow characteristic**, the **Tabulated valve openings** values will

## 4-Way Directional Valve

---

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

### Tabulated valve passage area

Specify the vector of output values for valve passage area as a tabulated 1-by-m array. The valve passage area vector must be the same size as the valve openings vector. All the values must be positive. The default values, in m<sup>2</sup>, are [1e-12, 4e-12, 1.e-5, 1.02e-5]. This parameter is used if **Model parameterization** is set to By area vs. opening table.

### Tabulated pressure differentials

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

### Tabulated flow rates

Specify the output values for flow rates as a tabulated m-by-n matrix, defining the function values at the input grid vertices. Each value in the matrix specifies flow rate taking place at a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in m<sup>3</sup>/s, are:

```
[ -1e-12,  -0.6e-13,   0.6e13,   0.1e12; ...  
  -0.8e-7, -0.4e-7,   0.4e7,    0.8e7; ...  
  -2.5e-3, -1.03e-3,  1.e-3,    2.4e-3; ...  
  -2.58e-3, -1.032e-3, 1.06e-3,  2.47e-3]
```

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

### Interpolation method

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

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

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

## **Extrapolation method**

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

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

## 4-Way Directional Valve

---

specified input argument, and the first specified output value for all input values less than the first specified input argument.

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

### **Flow discharge coefficient**

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

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

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

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

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

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

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

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

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

### **Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

## Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is  $1e-12 \text{ m}^2$ .

## Global Parameters

### Fluid density

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

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

P

Hydraulic conserving port associated with the pressure supply line inlet.

T

Hydraulic conserving port associated with the return line connection.

A

Hydraulic conserving port associated with the actuator connection port.

B

Hydraulic conserving port associated with the actuator connection port.

# 4-Way Directional Valve

---

S

Physical signal port to control spool displacement.

## Examples

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

## See Also

2-Way Directional Valve

3-Way Directional Valve



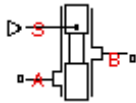
## Purpose

Simulate hydraulic variable orifice created by circular tube and round insert

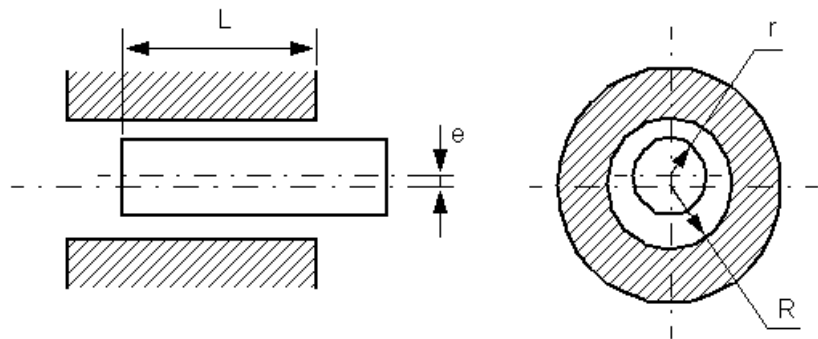
## Library

Hydraulic Elements

## 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
$\epsilon$	Eccentricity ratio
$e$	Eccentricity
$\rho$	Fluid density
$\nu$	Fluid kinematic viscosity

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

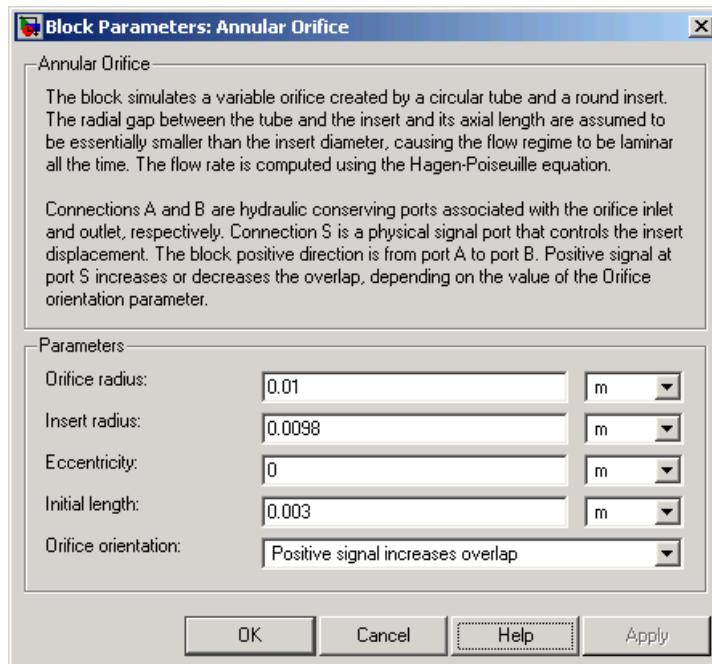
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as  $p = p_A - p_B$ . Positive signal at the physical signal port S increases or decreases the overlap, depending on the value of the parameter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

## Basic Assumptions and Limitations

The model is based on the following assumption:

- Fluid inertia is not taken into account.

## Dialog Box and Parameters



### Orifice radius

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

### Insert radius

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

### Eccentricity

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

### Initial length

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

# Annular Orifice

---

## Orifice orientation

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

## Global Parameters

### Fluid density

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

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the orifice inlet.

B

Hydraulic conserving port associated with the orifice outlet.

S

Physical signal port that controls the insert displacement.

## References

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

## See Also

Orifice with Variable Area Round Holes

Orifice with Variable Area Slot

Variable Orifice

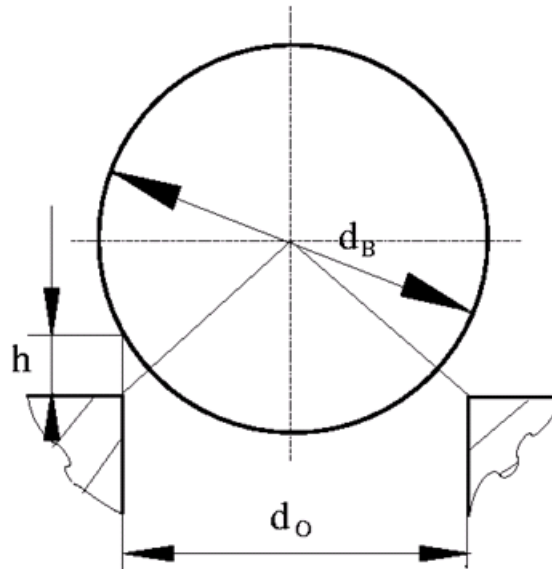
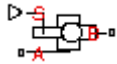
Fixed Orifice

**Purpose** Simulate hydraulic ball valve

**Library** Flow Control Valves

**Description**

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



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

# Ball Valve

---

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

$$h = x_0 + x$$

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

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

$$p = p_A - p_B$$

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

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

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

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

where

$q$	Flow rate
$p$	Pressure differential
$p_A, p_B$	Gauge pressures at the block terminals
$C_D$	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
$x_0$	Initial opening
$x$	Ball displacement from initial position
$h$	Valve opening
$d_o$	Orifice diameter
$r_o$	Orifice radius
$d_B$	Ball diameter
$r_B$	Ball radius
$\rho$	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

The model is based on the following assumptions:

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

# Ball Valve

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

## Dialog Box and Parameters

**Block Parameters: Ball Valve**

Ball Valve

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

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

Parameters

Valve ball diameter: 0.01 m

Orifice diameter: 0.005 m

Initial opening: 0 m

Flow discharge coefficient: 0.65

Critical Reynolds number: 10

Leakage area: 1e-12 m<sup>2</sup>

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.



**Initial opening**

The initial opening of the valve. Its value must be nonnegative. The default value is 0.

**Flow discharge coefficient**

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is 0.65.

**Critical Reynolds number**

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

**Leakage area**

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is  $1e-12 \text{ m}^2$ .

**Global Parameters****Fluid density**

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

**Fluid kinematic viscosity**

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

**Ports**

The block has the following ports:

# Ball Valve

---

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

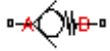
## See Also

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

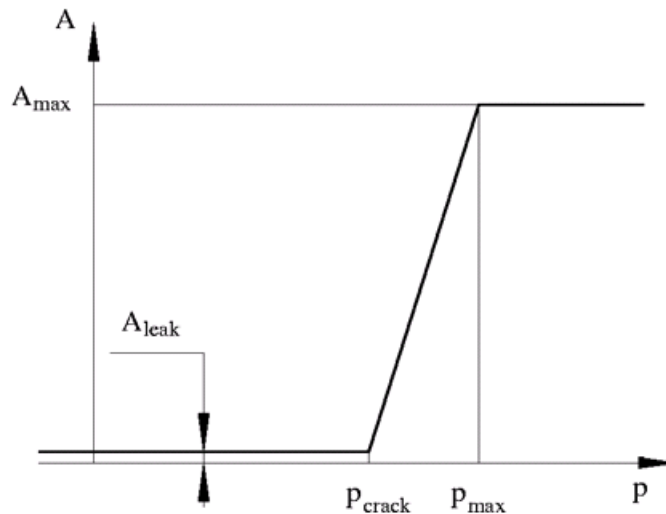
**Purpose** Simulate hydraulic valve that allows flow in one direction only

**Library** Directional Valves

## Description



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



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

# Check Valve

---

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

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

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

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

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

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

$$p = p_A - p_B$$

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

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

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

where

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

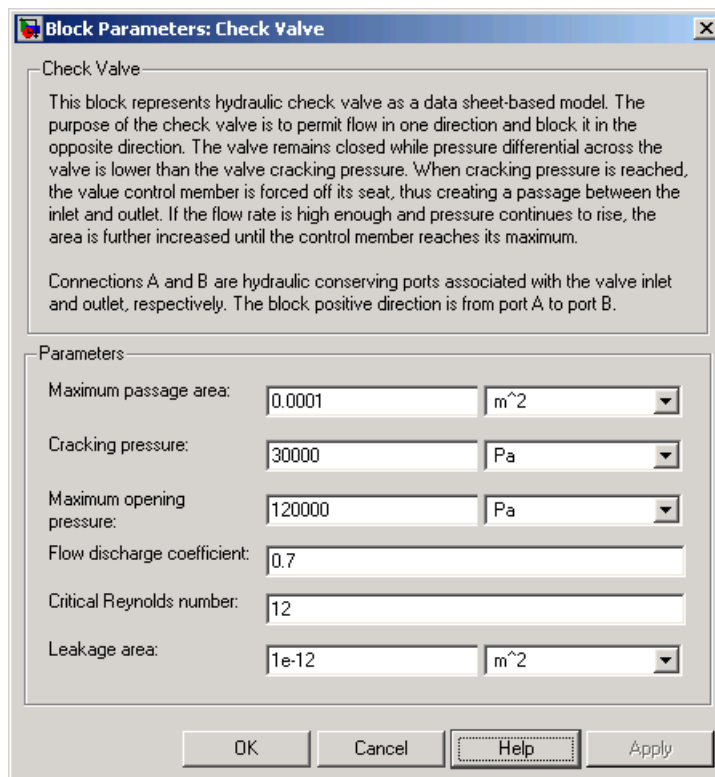
# Check Valve

## Basic Assumptions and Limitations

The model is based on the following assumptions:

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

## Dialog Box and Parameters



**Block Parameters: Check Valve**

Check Valve

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

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

Parameters

Maximum passage area:	0.0001	m <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  $3e4 \text{ Pa}$ .

**Maximum opening pressure**

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

**Flow discharge coefficient**

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

**Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

**Leakage area**

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is  $1e-12 \text{ m}^2$ .

# Check Valve

---

## Global Parameters

### Fluid density

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

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

## Examples

The Graetz Flow Control Circuit demo (sh\_Graetz\_circuit) illustrates the use of check valves to build a rectifier that keeps the flow passing through a flow control valve always in the same direction, and to select an appropriate orifice depending on the flow direction.

## See Also

Pilot-Operated Check Valve





**Purpose** Create Physical Modeling connector port for subsystem

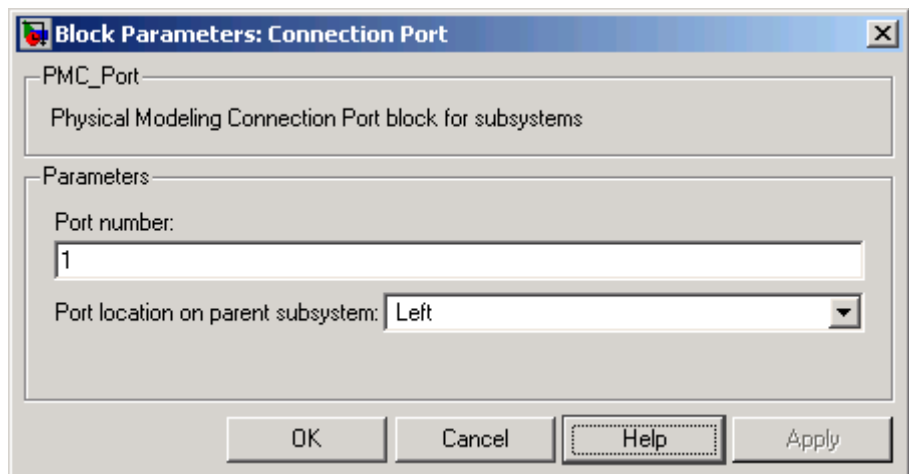
**Library** Utilities

**Description** The Connection Port block is used to export both the conserving and the physical signal connections to the outside boundary of a subsystem block, similar to the Inport and Outport blocks in Simulink.



The ports on the subsystem boundary appear as the square Conserving ports  or as triangular Physical Signal ports , depending on the type of port to which the Connection Port block is connected inside the subsystem. In other words, if a Connection Port block is connected to a Conserving port in a subsystem, it appears as a Conserving port on the outside boundary of the subsystem. If it is connected to a Physical Signal inport or outport inside the subsystem, it appears as a Physical Signal inport or outport, respectively, on the outside boundary of the subsystem.

## Dialog Box and Parameters



# Connection Port

---

## **Port number**

Labels the subsystem connector port created by this block. Each connector port on the boundary of a single subsystem requires a unique number as a label. The default value for the first port is 1.

## **Port location on parent subsystem**

Choose here on which side of the parent subsystem boundary the port is placed. The choices are `Left` or `Right`. The default choice is `Left`.

## **See Also**

In the Using Simulink documentation, see “Creating Masked Subsystems”.

## Purpose

Simulate hydraulic capacity of constant volume

## Library

Hydraulic Elements

## Description



The Constant Volume Chamber block models a fixed-volume chamber with rigid or flexible walls, to be used in hydraulic valves, pumps, manifolds, pipes, hoses, and so on. Use this block in models where you have to account for some form of fluid compressibility. You can select the appropriate representation of fluid compressibility using the block parameters.

Fluid compressibility in its simplest form is simulated according to the following equations:

$$V_f = V_c + \frac{V_c}{E} p$$

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

where

- $q$  Flow rate into the chamber
- $V_f$  Volume of fluid in the chamber
- $V_c$  Geometrical chamber volume
- $E$  Fluid bulk modulus
- $\rho$  Gauge pressure of fluid in the chamber

If pressure in the chamber is likely to fall to negative values and approach cavitation limit, the above equations must be enhanced. In this block, it is done by representing the fluid in the chamber as a mixture of liquid and a small amount of entrained, nondissolved gas. The mixture bulk modulus is determined as:

# Constant Volume Chamber

---

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

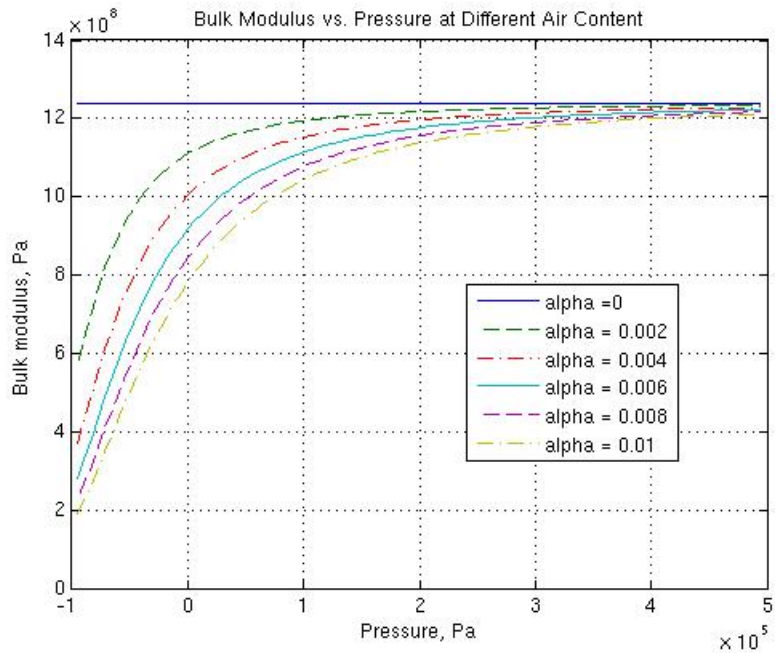
where

- $E_l$  Pure liquid bulk modulus
- $p_a$  Atmospheric pressure
- $\alpha$  Relative gas content at atmospheric pressure,  $\alpha = V_g/V_L$
- $V_g$  Gas volume at atmospheric pressure
- $V_L$  Volume of liquid
- $n$  Gas-specific heat ratio

The main objective of representing fluid as a mixture of liquid and gas is to introduce an approximate model of cavitation, which takes place in a chamber if pressure drops below fluid vapor saturation level. As it is seen in the graph below, the bulk modulus of a mixture decreases at  $p \rightarrow p_a$ , thus considerably slowing down further pressure change.

At high pressure,  $p \gg p_a$ , a small amount of nondissolved gas has practically no effect on the system behavior.

# Constant Volume Chamber



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

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

If chamber walls have noticeable compliance, the above equations must be further enhanced by representing geometrical chamber volume as a function of pressure:

# Constant Volume Chamber

---

$$V_c = \pi d^2 / 4 \cdot L$$

$$d(s) = \frac{K_p}{1 + \tau s} p(s)$$

where

$d$	Internal diameter of the cylindrical chamber
$L$	Length of the cylindrical chamber
$K_p$	Proportionality coefficient (m/Pa)
$\tau$	Time constant
$s$	Laplace operator

Coefficient  $K_p$  establishes relationship between pressure and the internal diameter at steady-state conditions. For metal tubes, the coefficient can be computed as (see [1]):

$$K_p = \frac{d}{E_M} \left( \frac{D^2 + d^2}{D^2 - d^2} + \nu \right)$$

where

$D$	Pipe external diameter
$E_M$	Modulus of elasticity (Young's modulus) for the pipe material
$\nu$	Poisson's ratio for the pipe material

For hoses, the coefficient can be provided by the manufacturer.

The process of expansion and contraction in pipes and especially in hoses is a complex combination of nonlinear elastic and viscoelastic deformations. This process is approximated in the block with the

first-order lag, whose time constant is determined empirically (for example, see [2]).

As a result, by selecting appropriate values, you can implement four different models of fluid compressibility with this block:

- Chamber with rigid walls, no entrained gas in the fluid
- Cylindrical chamber with compliant walls, no entrained gas in the fluid
- Chamber with rigid walls, fluid with entrained gas
- Cylindrical chamber with compliant walls, fluid with entrained gas

The block allows two methods of specifying the chamber size:

- By volume — Use this option for cylindrical or non-cylindrical chambers with rigid walls. You only need to know the volume of the chamber. This chamber type does not account for wall compliance.
- By length and diameter — Use this option for cylindrical chambers with rigid or compliant walls, such as circular pipes or hoses.

The block has one hydraulic conserving port associated with the chamber inlet. The block positive direction is from its port to the reference point. This means that the flow rate is positive if it flows into the chamber.

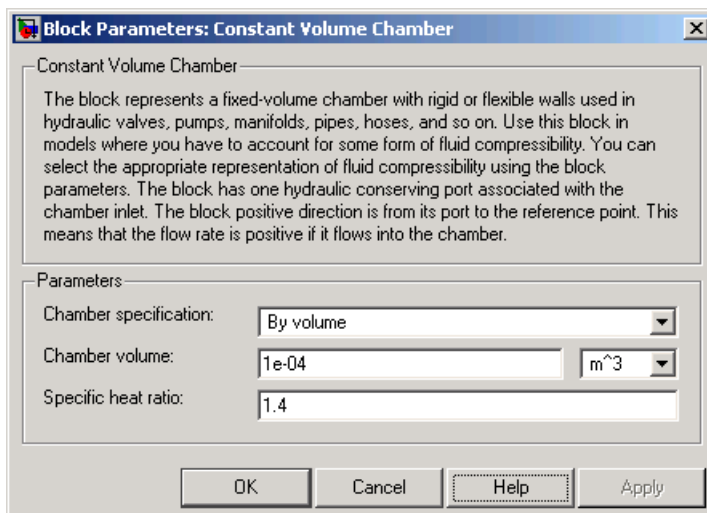
## **Basic Assumptions and Limitations**

The model is based on the following assumptions:

- No inertia associated with pipe walls is taken into account.
- Chamber with compliant walls is assumed to have a cylindrical shape. Chamber with rigid wall can have any shape.

# Constant Volume Chamber

## Dialog Box and Parameters



**Block Parameters: Constant Volume Chamber**

Constant Volume Chamber

The block represents a fixed-volume chamber with rigid or flexible walls used in hydraulic valves, pumps, manifolds, pipes, hoses, and so on. Use this block in models where you have to account for some form of fluid compressibility. You can select the appropriate representation of fluid compressibility using the block parameters. The block has one hydraulic conserving port associated with the chamber inlet. The block positive direction is from its port to the reference point. This means that the flow rate is positive if it flows into the chamber.

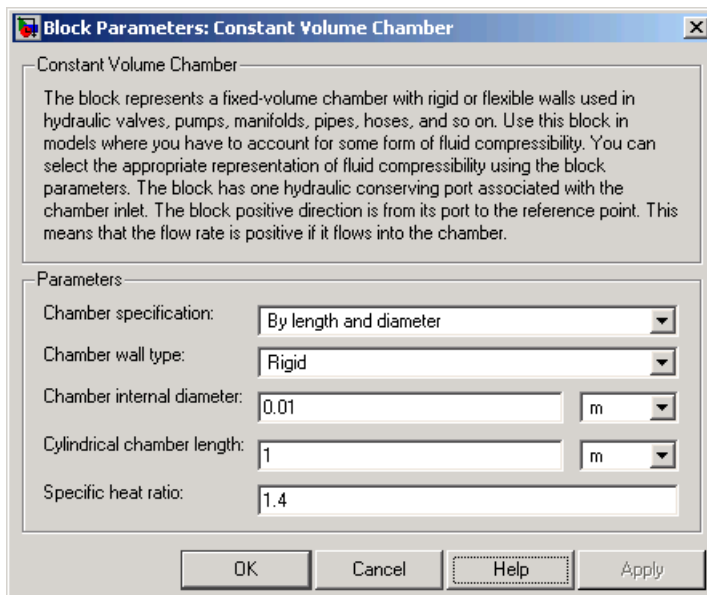
Parameters

Chamber specification: By volume

Chamber volume: 1e-04 m<sup>3</sup>

Specific heat ratio: 1.4

OK Cancel Help Apply



**Block Parameters: Constant Volume Chamber**

Constant Volume Chamber

The block represents a fixed-volume chamber with rigid or flexible walls used in hydraulic valves, pumps, manifolds, pipes, hoses, and so on. Use this block in models where you have to account for some form of fluid compressibility. You can select the appropriate representation of fluid compressibility using the block parameters. The block has one hydraulic conserving port associated with the chamber inlet. The block positive direction is from its port to the reference point. This means that the flow rate is positive if it flows into the chamber.

Parameters

Chamber specification: By length and diameter

Chamber wall type: Rigid

Chamber internal diameter: 0.01 m

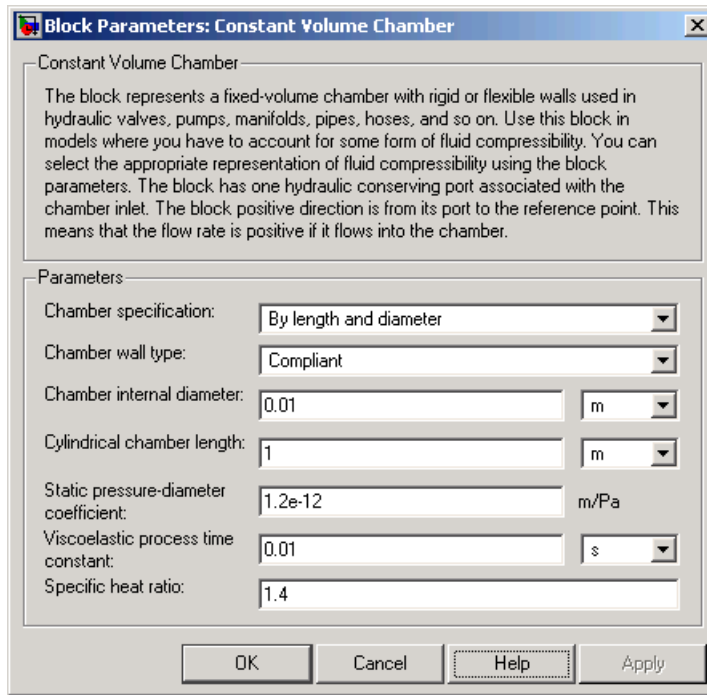
Cylindrical chamber length: 1 m

Specific heat ratio: 1.4

OK Cancel Help Apply



# Constant Volume Chamber



## Chamber specification

The parameter can have one of two values: By volume or By length and diameter. The value By length and diameter is recommended if a chamber is formed by a circular pipe. If the parameter is set to By volume, wall compliance is not taken into account. The default value of the parameter is By volume.

## Chamber 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 compliance can affect the system behavior. The default value of the parameter is Rigid. The parameter is used if the **Chamber specification** parameter is set to By length and diameter.

# Constant Volume Chamber

---

## Chamber volume

Volume of fluid in the chamber. The default value is  $1e-4 \text{ m}^3$ . The parameter is used if the **Chamber specification** parameter is set to **By volume**.

## Chamber internal diameter

Internal diameter of the cylindrical chamber. The default value is 0.01 m. The parameter is used if the **Chamber specification** parameter is set to **By length and diameter**.

## Cylindrical chamber length

Length of the cylindrical chamber. The default value is 1 m. The parameter is used if the **Chamber specification** parameter is set to **By length and diameter**.

## Static pressure-diameter coefficient

Coefficient  $K_p$  that establishes relationship between pressure and the internal diameter at steady-state conditions. The parameter can be determined analytically or experimentally. The default value is  $1.2e-12 \text{ m/Pa}$ . The parameter is used if **Chamber wall type** is set to **Compliant**.

## Viscoelastic process time constant

Time constant in the transfer function relating pipe internal diameter to pressure variations. With this parameter, the simulated elastic or viscoelastic process is approximated with the first-order lag. The parameter is determined experimentally or provided by the manufacturer. The default value is 0.01 s. The parameter is used if **Chamber wall type** is set to **Compliant**.

## Specific heat ratio

Gas-specific heat ratio. The default value is 1.4.

## Global Parameters

## Fluid bulk modulus

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

## **Nondissolved gas ratio**

Nondissolved gas relative content determined as a ratio of gas volume to the liquid volume. The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

## **Ports**

The block has one hydraulic conserving port associated with the chamber inlet.

## **References**

- [1] Meritt, H.E., *Hydraulic Control Systems*, John Wiley & Sons, New York, 1967
- [2] Holcke, Jan, *Frequency Response of Hydraulic Hoses*, RIT, FTH, Stockholm, 2002

## **Examples**

The Constant Volume Chamber Test Rig demo (sh\_constant\_chamber\_test\_rig) is specifically designed to demonstrate the Constant Volume Chamber block behavior at different regimes. The chamber is placed between two linear hydraulic resistances and subjected to an abrupt pressure change from 0 to 5 MPa at the beginning of simulation.

If air content is set to zero and chamber walls are rigid, the pressure change can easily be determined analytically. The Simulink Transfer Fcn block simulates this regime for a metal cylindrical pipe with 0.03 m internal diameter, 0.036 m external diameter, and 16.5 m length. The output of this block serves as a reference during all other regimes. You can investigate chamber characteristics, for example, by changing air content in the fluid, by switching from rigid to compliant walls, by changing the viscoelastic time constant or the pressure-diameter coefficient.

## **See Also**

Variable Volume Chamber

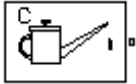
# Custom Hydraulic Fluid

---

**Purpose** Set working fluid properties by specifying parameter values

**Library** Hydraulic Utilities

**Description**



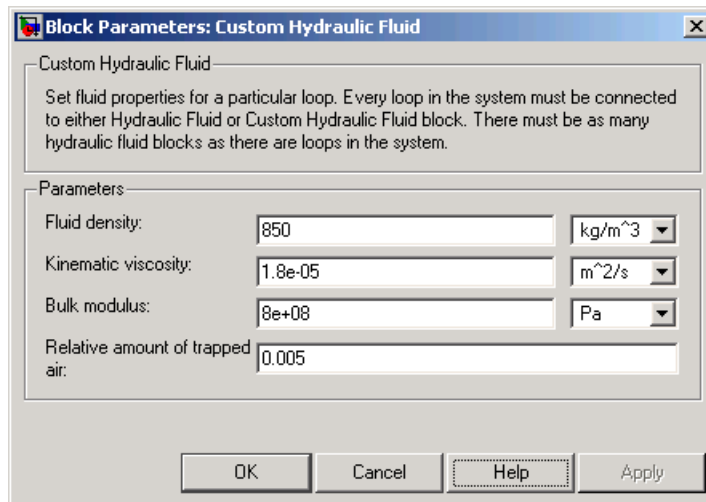
The Custom Hydraulic Fluid block lets you specify the type of hydraulic fluid used in a loop of hydraulic blocks. It provides the hydraulic fluid properties, such as kinematic viscosity, density, and bulk modulus, for all the hydraulic blocks in the loop. These fluid properties are assumed to be constant during simulation time.

The Custom Hydraulic Fluid block lets you specify the fluid properties, such as kinematic viscosity, density, bulk modulus, and relative amount of entrapped air, as block parameters.

The Custom 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 Custom Hydraulic Fluid block to a hydraulic line, SimHydraulics automatically identifies the hydraulic blocks connected to the particular loop and propagates the hydraulic fluid properties to all the hydraulic blocks in the loop.

Each topologically distinct hydraulic loop in a diagram requires exactly one Custom Hydraulic Fluid block or Hydraulic Fluid block to be connected to it. Therefore, there must be as many Custom Hydraulic Fluid blocks (or Hydraulic Fluid blocks) as there are loops in the system.

## Dialog Box and Parameters



### Fluid density

Density of the working fluid. The default value is  $850 \text{ kg/m}^3$ .

### Kinematic viscosity

Kinematic viscosity of the working fluid. The default value is  $1.8\text{e-}5 \text{ m}^2/\text{s}$ .

### Bulk modulus

Bulk modulus of the working fluid. The default value is  $8\text{e}8 \text{ Pa}$ .

### Relative amount of trapped air

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

## Ports

The block has one hydraulic conserving port.

## See Also

Hydraulic Fluid

# Cylinder Friction

**Purpose** Simulate friction in hydraulic cylinders

**Library** Hydraulic Cylinders

## Description



The Cylinder Friction block simulates friction in the contact between moving bodies in hydraulic cylinders and is intended to be used primarily as a building block in combination with both the double- and single-acting cylinders to develop a cylinder model with friction. The friction force is simulated as a function of relative velocity and pressure, and is assumed to be the sum of Stribeck, Coulomb, and viscous components. The Coulomb friction force consists of the preload force, caused by the seal squeeze during assembly, and the force proportional to pressure. The sum of the Coulomb and Stribeck friction forces at zero velocity is often referred to as the breakaway friction force. For more information, see the Translational Friction block reference page.

The friction force is approximated with the following equations:

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

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

where

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

$v$  Relative velocity in the contact

$f_{vfr}$  Viscous friction coefficient

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

$$F = K \cdot v$$

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

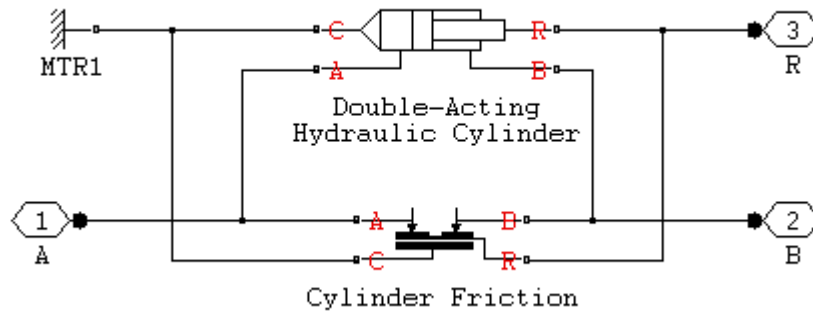
where

$K$  Proportionality coefficient

$v_{th}$  Velocity threshold

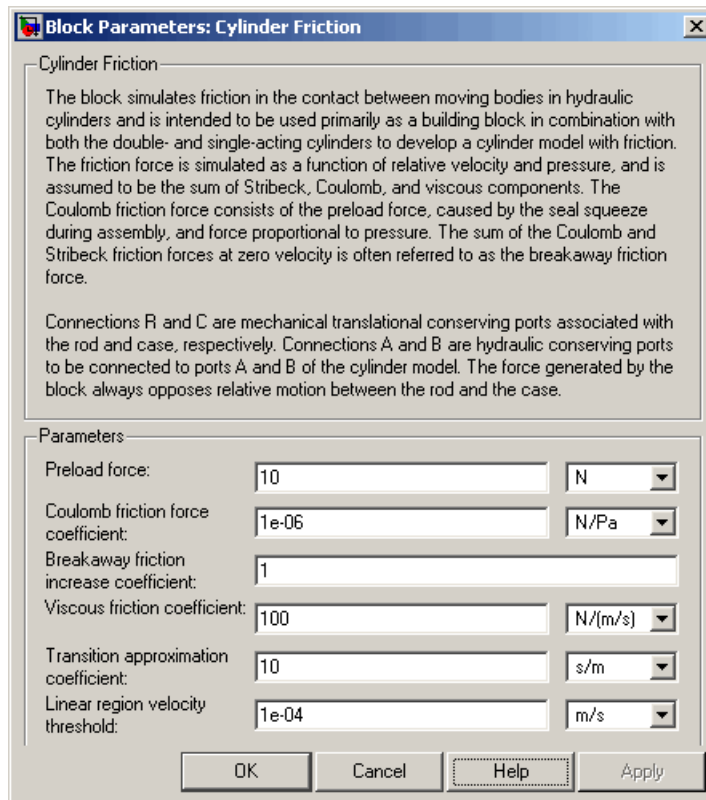
Connections R and C are mechanical translational conserving ports associated with the rod and case, respectively. Connections A and B are hydraulic conserving ports to be connected to ports A and B of the cylinder model, as shown in the following illustration. The force generated by the block always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

# Cylinder Friction





## Dialog Box and Parameters



### Preload force

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

### Coulomb friction force coefficient

Coulomb friction coefficient, which defines the proportionality between the Coulomb friction force and the pressure in cylinder chambers. The default value is 1 e-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  $c_v \sim 4/v_{min}$ , where  $v_{min}$  is the relative velocity at which friction force has its minimum value. By default,  $c_v$  is set to 10 s/m, which corresponds to a minimum friction at velocity of about 0.4 m/s.

## **Linear region velocity threshold**

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

## **Ports**

The block has the following ports:

A

Hydraulic conserving port connected to the cylinder inlet.

B

Hydraulic conserving port connected to the cylinder outlet.

R Mechanical translational conserving port associated with the cylinder rod.

C Mechanical translational conserving port associated with the cylinder clamping structure.

## **See Also**

Double-Acting Hydraulic Cylinder

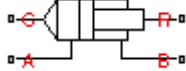
Single-Acting Hydraulic Cylinder

# Double-Acting Hydraulic Cylinder

**Purpose** Simulate hydraulic actuator exerting force in both directions

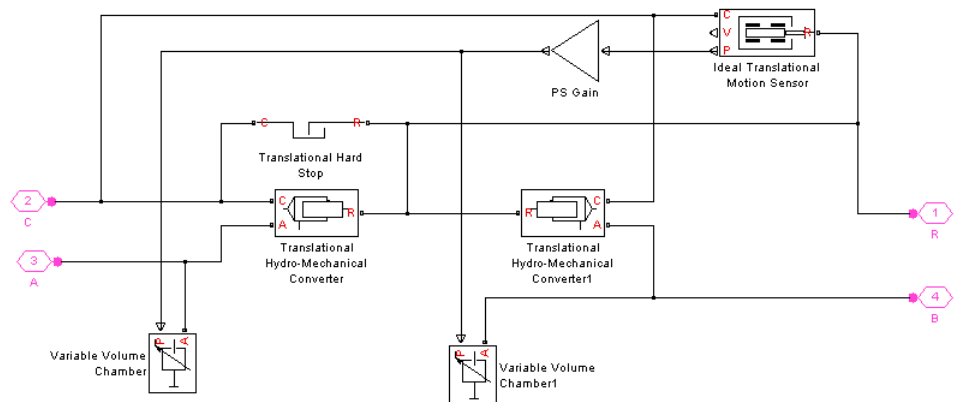
**Library** Hydraulic Cylinders

## Description



The Double-Acting Hydraulic Cylinder block models a device that converts hydraulic energy into mechanical energy in the form of translational motion. Hydraulic fluid pumped under pressure into one of the two cylinder chambers forces the piston to move and exert force on the cylinder rod. Double-acting cylinders transfer force and motion in both directions.

The model of the cylinder is built of other SimHydraulics blocks. The schematic diagram of the model is shown below.



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

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

compressibility in the cylinder chamber. The rod motion is limited with the mechanical Translational Hard Stop block in such a way that the rod can travel only between cylinder caps. The Ideal Translational Motion Sensor block in the schematic is introduced to determine an instantaneous piston position, which is necessary for the Variable Volume Chamber blocks.

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

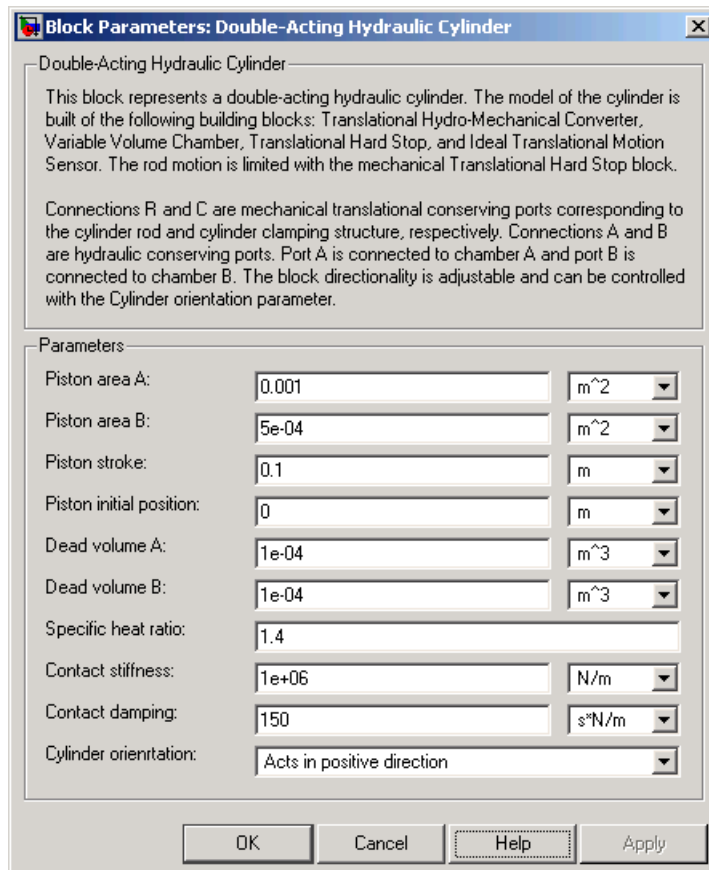
## **Basic Assumptions and Limitations**

The model is based on the following assumptions:

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

# Double-Acting Hydraulic Cylinder

## Dialog Box and Parameters



### Chamber A piston area

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

### Chamber B piston area

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.

# Double-Acting Hydraulic Cylinder

---

## **Piston initial position**

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

## **Dead volume A**

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

## **Dead volume B**

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

## **Specific heat ratio**

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

## **Contact stiffness**

Specifies the elastic property of colliding bodies for the Translational Hard Stop block. The greater the value of the parameter, the less the bodies penetrate into each other, the more rigid the impact becomes. Lesser value of the parameter makes contact softer, but generally improves convergence and computational efficiency. The default value is  $1 \text{e}6 \text{ N/m}$ .

## **Contact damping**

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

# Double-Acting Hydraulic Cylinder

---

## 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`.

## Global Parameters

### Fluid bulk modulus

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the cylinder chamber A.

B

Hydraulic conserving port associated with the cylinder chamber B.

R

Mechanical translational conserving port associated with the cylinder rod.

C

Mechanical translational conserving port associated with the cylinder clamping structure.

## Examples

The `Double-Acting Hydraulic Cylinder with Flexible Clamping demo (sh_cylinder_da_flexible_clamping)` illustrates simulation of a cylinder whose clamping is too flexible to be neglected. The structure compliance is represented with a spring and a damper, installed between the cylinder case and reference point. The cylinder performs forward and return strokes, and is loaded with inertia, viscous friction, and constant opposing load of 400 N.



# Double-Acting Hydraulic Cylinder

---

## **See Also**

Single-Acting Hydraulic Cylinder

Ideal Translational Motion Sensor

Translational Hard Stop

Translational Hydro-Mechanical Converter

Variable Volume Chamber

# Elbow

---

**Purpose** Simulate hydraulic resistance in elbow

**Library** Local Hydraulic Resistances

**Description**



The Elbow block represents an elbow as a local hydraulic resistance. The pressure loss is computed with the semi-empirical formula based on pressure loss coefficient, which is determined in accordance with the Crane Co. recommendations (see [1], p. A-29). Two types of elbow are considered: smoothly curved (standard) and sharp-edged (miter). The block covers elbows in the 5–100 mm and 0–90 degrees range.

The pressure loss 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 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

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

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

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

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

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

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

# Elbow

## Warning

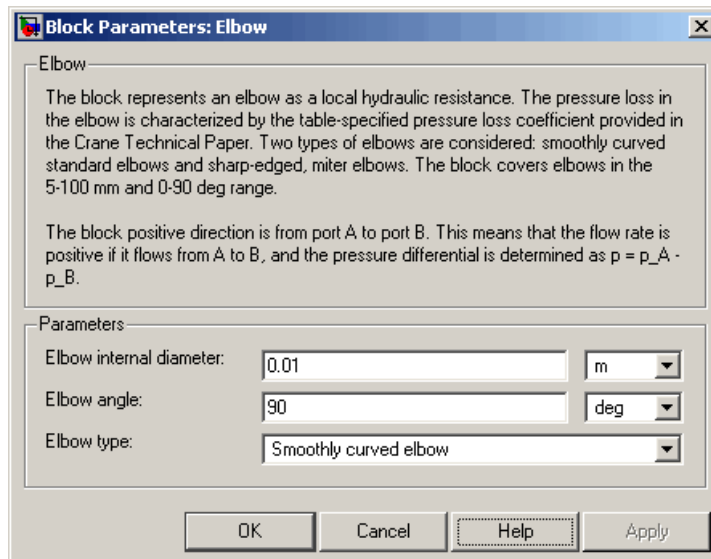
The formulas used in the Elbow block are very approximate, especially in the laminar and transient flow regions. For more accurate results, use the Local Resistance block with a table-specified  $K=f(Re)$  relationship.

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- Fluid inertia is not taken into account.
- The flow is assumed to be completely turbulent and fully developed along the pipe length.
- The elbow is assumed to be made of a clean commercial steel pipe.

## 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.

## Global Parameters

### Fluid density

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

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the elbow inlet.

B

Hydraulic conserving port associated with the elbow outlet.

## References

[1] *Flow of Fluids Through Valves, Fittings, and Pipe*, Crane Valves North America, Technical Paper No. 410M

[2] George R. Keller, *Hydraulic System Analysis*, Published by the Editors of Hydraulics & Pneumatics Magazine, 1970

## See Also

Gradual Area Change

Local Resistance

Pipe Bend

# Elbow

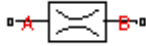
---

Sudden Area Change  
T-junction

**Purpose** Simulate hydraulic orifice with constant cross-sectional area

**Library** Hydraulic Elements

## 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  $p = p_A - p_B$ .

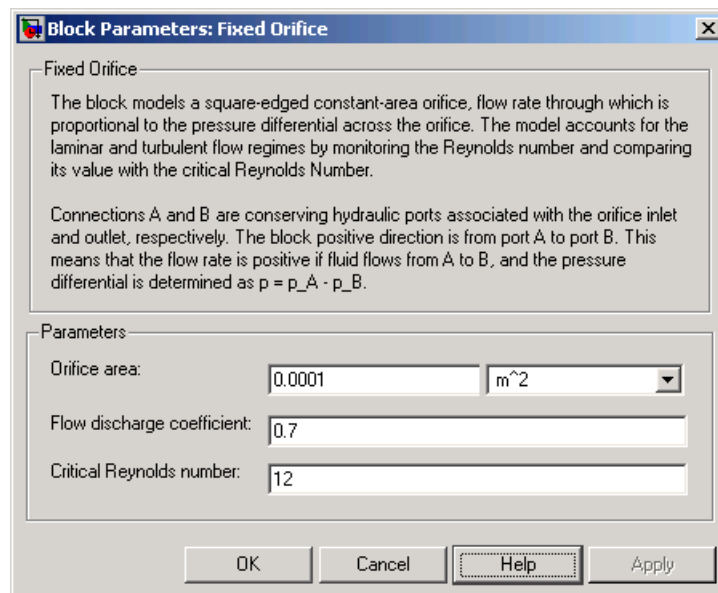
## Basic Assumptions and Limitations

The model is based on the following assumptions:

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



## Dialog Box and Parameters



### Orifice area

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

### Flow discharge coefficient

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

### Critical Reynolds number

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12, which corresponds to a round orifice in thin material with sharp edges.

# Fixed Orifice

---

## Global Parameters

### Fluid density

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

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the orifice inlet.

B

Hydraulic conserving port associated with the orifice outlet.

## See Also

Annular Orifice

Orifice with Variable Area Round Holes

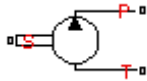
Orifice with Variable Area Slot

Variable Orifice

**Purpose** Simulate fixed-displacement hydraulic pump

**Library** Pumps and Motors

## Description



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

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

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

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

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

$$p = p_P - p_T$$

where

$q$	Pump delivery
$p$	Pressure differential across the pump
$p_P, p_T$	Gauge pressures at the block terminals
$T$	Torque at the pump driving shaft
$D$	Pump displacement
$\omega$	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

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

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

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

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

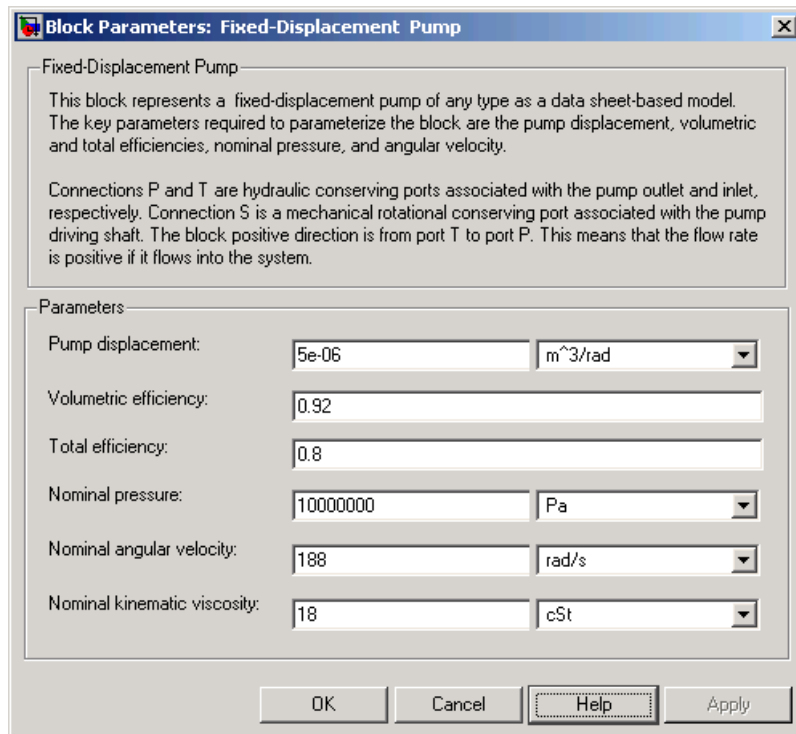
## **Basic Assumptions and Limitations**

The model is based on the following assumptions:

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

# Fixed-Displacement Pump

## Dialog Box and Parameters



### Pump displacement

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

### Volumetric efficiency

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

### Total efficiency

Pump total efficiency, which is determined as a ratio between the hydraulic power at the pump outlet and mechanical power at the driving shaft at nominal pressure, angular velocity, and fluid viscosity. The default value is 0.8.

## Nominal pressure

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

## Nominal angular velocity

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

## Nominal kinematic viscosity

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

## Global Parameters

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

T

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

P

Hydraulic conserving port associated with the pump outlet.

S

Mechanical rotational conserving port associated with the pump driving shaft.

## Examples

The Power Unit with Fixed-Displacement Pump demo (sh\_power\_unit\_fxd\_dspl\_pump) contains a fixed-displacement pump, which is driven by a motor through a compliant transmission, a pressure-relief valve, and a variable orifice, which simulates system fluid consumption. The motor model is represented as an Ideal Angular Velocity Source block, which rotates the shaft at 188 rad/s at zero torque. The load on the shaft decreases the velocity with a slip coefficient of 1.2

# Fixed-Displacement Pump

---

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

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

## See Also

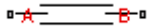
Variable-Displacement Pressure-Compensated Pump



**Purpose** Simulate pressure differential across tube or channel due to change in fluid velocity

**Library** Hydraulic Elements

**Description** The Fluid Inertia block models pressure differential, due to change in fluid velocity, across a fluid passage of constant cross-sectional area. The pressure differential is determined according to the following equation:



$$p = \rho \frac{L}{A} \frac{dq}{dt}$$

where

$\rho$	Pressure differential
$\rho$	Fluid density
$L$	Passage length
$A$	Passage area
$q$	Flow rate
$t$	Time

Use this block in various pipe or channel models that require fluid inertia to be accounted for.

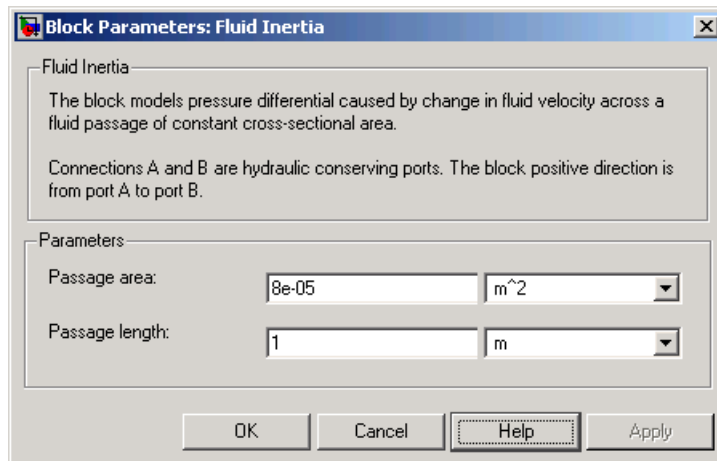
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B.

**Assumptions and Limitations** The model is based on the following assumptions:

- Fluid density remains constant.

# Fluid Inertia

## Dialog Box and Parameters



### Passage area

Fluid passage cross-sectional area. The default value is  $8e-5 \text{ m}^2$ .

### Passage length

Length of the fluid passage. The default value is 1 m.

## Global Parameters

### Fluid density

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the passage inlet.

B

Hydraulic conserving port associated with the passage outlet.

**Purpose** Simulate hydraulic accumulator with gas as compressible medium

**Library** Accumulators

## Description



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

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

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

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

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

where

# Gas-Charged Accumulator

---

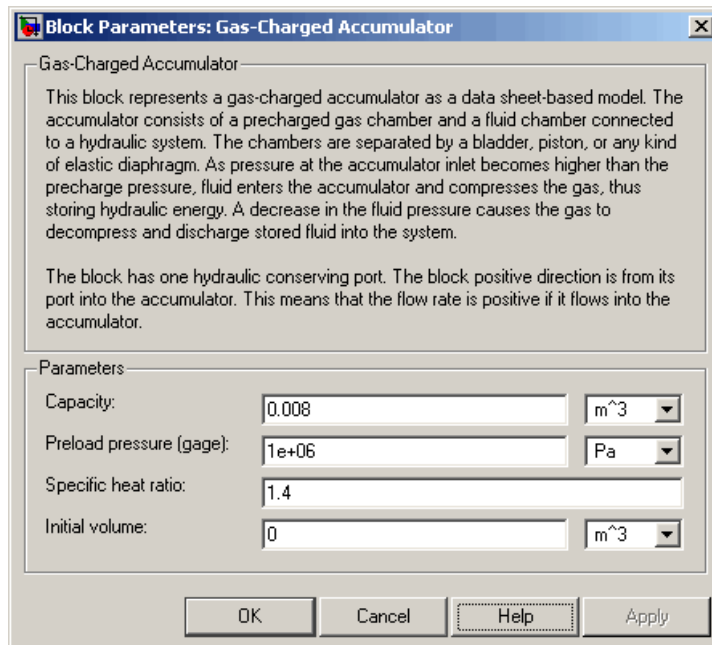
$V_F$	Fluid volume
$V_A$	Accumulator capacity
$p$	Inlet gauge pressure
$p_{pr}$	Precharge pressure
$k$	Specific heat ratio
$q$	Volumetric flow rate
$t$	Time

## Basic Assumptions and Limitations

The model is based on the following assumptions:

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

## Dialog Box and Parameters



### Accumulator capacity

Accumulator capacity. The default value is 0.008 m<sup>3</sup>.

### Preload pressure

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

### Specific heat ratio

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

### Initial volume

Initial volume of fluid in the accumulator. This parameter specifies the initial condition for use in computing the block's initial state at the beginning of a simulation run. For more information, see "Computing Initial Conditions". The default value is 0.

# Gas-Charged Accumulator

---

## **Global Parameters**

### **Atmospheric pressure**

Absolute pressure of the environment. The default value is 101325 Pa.

## **Ports**

The block has one hydraulic conserving port associated with the accumulator inlet.

The flow rate is positive if fluid flows into the accumulator.

## **See Also**

Spring-Loaded Accumulator

**Purpose** Simulate gear boxes in mechanical systems

**Library** Mechanisms

**Description**



The Gear Box block represents an ideal, nonplanetary, fixed gear ratio gear box. The gear ratio is determined as the ratio of the input shaft angular velocity to that of the output shaft.

The gear box is described with the following equations:

$$\omega_1 = N \cdot \omega_2$$

$$T_2 = N \cdot T_1$$

$$P_1 = \omega_1 \cdot T_1$$

$$P_2 = -\omega_2 \cdot T_2$$

where

$\omega_1$  Input shaft angular velocity

$\omega_2$  Output shaft angular velocity

$N$  Gear ratio

$T_1$  Torque on the input shaft

$T_2$  Torque on the output shaft

$P_1$  Power on the input shaft

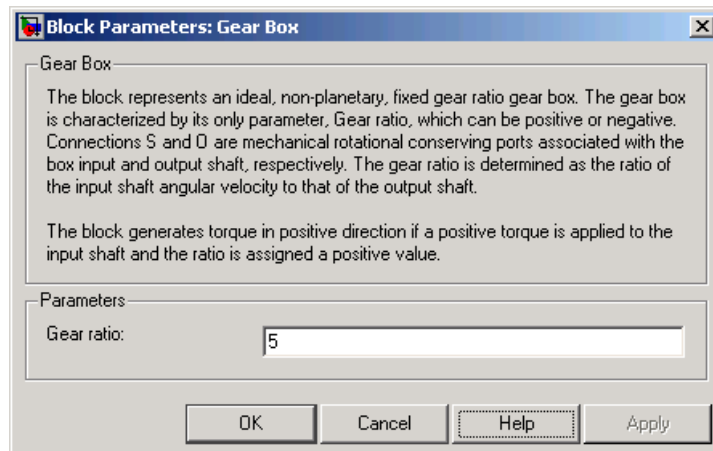
$P_2$  Power on the output shaft. Notice the minus sign in computing  $P_2$ . One of the network rules is that the power flowing through a conserving port is positive if it is removed (dissipated) from the circuit, and is negative if the component generates power into the system.

# Gear Box

---

Connections S and O are mechanical rotational conserving ports associated with the box input and output shaft, respectively. The block positive directions are from S to the reference point and from the reference point to O.

## Dialog Box and Parameters



### Gear ratio

The ratio of the input shaft angular velocity to that of the output shaft. You can specify both positive and negative values. The default value is 5.

### Ports

The block has the following ports:

S

Mechanical rotational conserving port associated with input shaft.

O

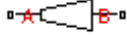
Mechanical rotational conserving port associated with the output shaft.



**Purpose** Simulate gradual enlargement or contraction

**Library** Local Hydraulic Resistances

## Description



The Gradual Area Change block represents a local hydraulic resistance, such as a gradual cross-sectional area change. The resistance represents a gradual enlargement (diffuser) if fluid flows from inlet to outlet, or a gradual contraction if fluid flows from outlet to inlet. The block is based on the Local Resistance block. It determines the pressure loss coefficient and passes its value to the Local Resistance block. The block offers two methods of parameterization: by applying semi-empirical formulas 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 A.H. Gibson equations (see [1] and [2]):

$$K_{GE} = \begin{cases} K_{SE} \cdot 2.6 \sin \frac{\alpha}{2} & \text{for } 0 < \alpha \leq 45^\circ \\ K_{SE} & \text{for } 45^\circ < \alpha \leq 180^\circ \end{cases}$$

$$K_{GC} = \begin{cases} K_{SC} \cdot 1.6 \sin \frac{\alpha}{2} & \text{for } 0 < \alpha \leq 45^\circ \\ K_{SC} \cdot \sqrt{\sin \frac{\alpha}{2}} & \text{for } 45^\circ < \alpha \leq 180^\circ \end{cases}$$

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

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

# Gradual Area Change

---

where

$K_{GE}$	Pressure loss coefficient for the gradual enlargement, which takes place if fluid flows from inlet to outlet
$K_{GC}$	Pressure loss coefficient for the gradual contraction, which takes place if fluid flows from outlet to inlet
$K_{SE}$	Pressure loss coefficient for a sudden enlargement
$K_{SC}$	Pressure loss coefficient for a sudden contraction
$K_{cor}$	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 Local Resistance block, which computes the pressure loss according to the formulas explained in the reference documentation for that block.

The Gradual Area Change block is bidirectional and computes pressure loss for both the direct flow (gradual enlargement) and return flow (gradual contraction). If the loss coefficient is specified by a table, the table must cover both the positive and the negative flow regions.

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

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

## Basic Assumptions and Limitations

The model is based on the following assumption:

- Fluid inertia is not taken into account.
- If you select parameterization by semi-empirical formulas, the flow is assumed to be completely turbulent and fully developed along the pipe length.

## Dialog Box and Parameters

**Block Parameters: Gradual Area Change**

Gradual Area Change

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

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

Parameters

Small diameter:  m

Large diameter:  m

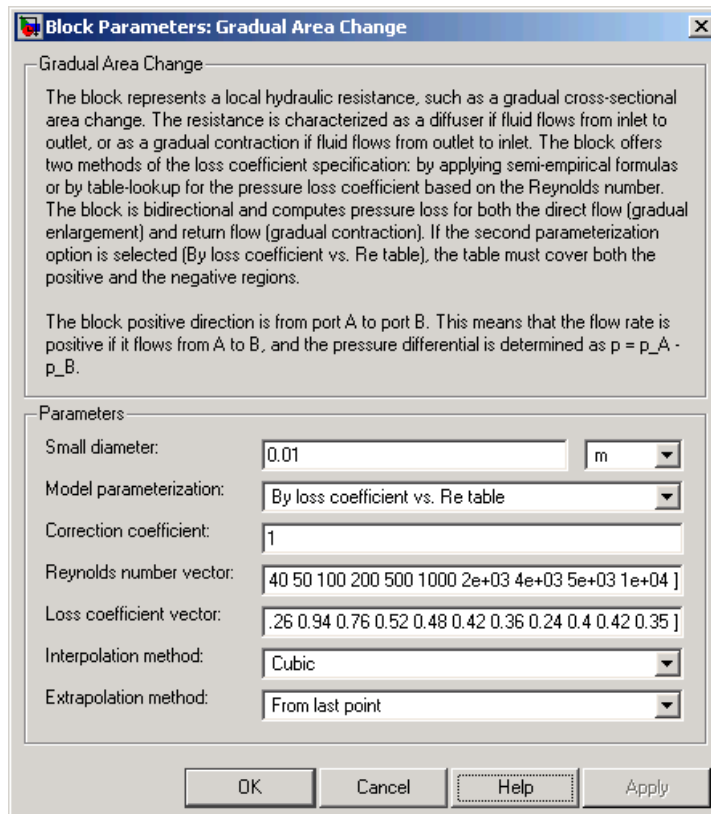
Cone angle:  deg

Model parameterization:

Correction coefficient:

OK Cancel Help Apply

# Gradual 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.

## 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.

## Reynolds number vector

Specify the vector of input values for Reynolds numbers as a tabulated 1-by-m array. The input values vector must be strictly monotonically increasing. The values can be nonuniformly spaced. You must provide at least three values. The default values are [-4000, -3000, -2000, -1000, -500, -200, -100, -50, -40, -30, -20, -15, -10, 10, 20, 30, 40, 50, 100, 200, 500, 1000, 2000, 4000, 5000, 10000]. This parameter is used if **Model parameterization** is set to By loss coefficient vs. Re table.

## Loss coefficient vector

Specify the vector of output values for the loss coefficient as a tabulated 1-by-m array. The loss coefficient vector must be the same size as the Reynolds numbers vector. The default values are [0.25, 0.3, 0.65, 0.9, 0.65, 0.75, 0.90, 1.15,

# Gradual Area Change

---

1.35, 1.65, 2.3, 2.8, 3.10, 5, 2.7, 1.8, 1.46, 1.3, 0.9, 0.65, 0.42, 0.3, 0.20, 0.40, 0.42, 0.25]. This parameter is used if **Model parameterization** is set to By loss coefficient vs. Re table.

## Interpolation method

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

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

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to By loss coefficient vs. Re table.

## Extrapolation method

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

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

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to By loss coefficient vs. Re table.

## Global Parameters

### Fluid density

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

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the resistance inlet.

B

Hydraulic conserving port associated with the resistance outlet.

## References

[1] *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

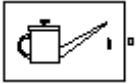
# Hydraulic Fluid

---

**Purpose** Set working fluid properties by selecting from list of predefined fluids

**Library** Hydraulic Utilities

## Description



The Hydraulic Fluid block lets you specify the type of hydraulic fluid used in a loop of hydraulic blocks. It provides the hydraulic fluid properties, such as kinematic viscosity, density, and bulk modulus, for all the hydraulic blocks in the loop. These fluid properties are assumed to be constant during simulation time. The density is determined by the type of fluid, while kinematic viscosity additionally requires that the temperature is specified. The bulk modulus is determined by the type of fluid and by the relative amount of entrained, nondissolved gas in the fluid.

The block offers a selection of predefined fluids. Once you select a fluid name, you can also specify the temperature of the fluid and the relative amount of entrained, nondissolved gas. Based on these parameters, SimHydraulics computes the three global fluid properties: kinematic viscosity, density, and bulk modulus.

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

Each topologically distinct hydraulic loop in a diagram requires exactly one Hydraulic Fluid block or Custom Hydraulic Fluid block to be connected to it. Therefore, there must be as many Hydraulic Fluid blocks (or Custom Hydraulic Fluid blocks) as there are loops in the system.



## Dialog Box and Parameters

**Block Parameters: Hydraulic Fluid**

Hydraulic Fluid  
Set working fluid properties

Parameters

Hydraulic fluid: Skydrol LD-4

Relative amount of trapped air: 0.005

System temperature (C): 75

Fluid Properties

Density: 9.501537e+002 kg/m<sup>3</sup>

Viscosity: 5.501634e-006 cSt

Bulk modulus: 1.139178e+009 Pa

OK Cancel Help Apply

### Hydraulic fluid

Hydraulic fluid type. Select one of the predefined fluids:

- Skydrol LD-4
- 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

# Hydraulic Fluid

---

- Oil SAE-30
- Oil SAE-50
- Transmission fluid ATF (Dexron III)
- ISO VG 32 (ESSO UNIVIS N 32)
- Gasoline
- Diesel fuel
- Jet fuel
- Water-Glycol 60/40

## **Relative amount of trapped air**

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

## **System temperature**

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

## **Ports**

The block has one hydraulic conserving port.

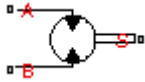
## **See Also**

Custom Hydraulic Fluid

**Purpose** Simulate fixed-displacement hydraulic motor

**Library** Pumps and Motors

## Description



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

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

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

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

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

$$p = p_A - p_B$$

where

$q$	Flow rate through the motor
$p$	Pressure differential across the motor
$p_A, p_B$	Gauge pressures at the block terminals
$T$	Torque at the motor output shaft
$D$	Motor displacement
$\omega$	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

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

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

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

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

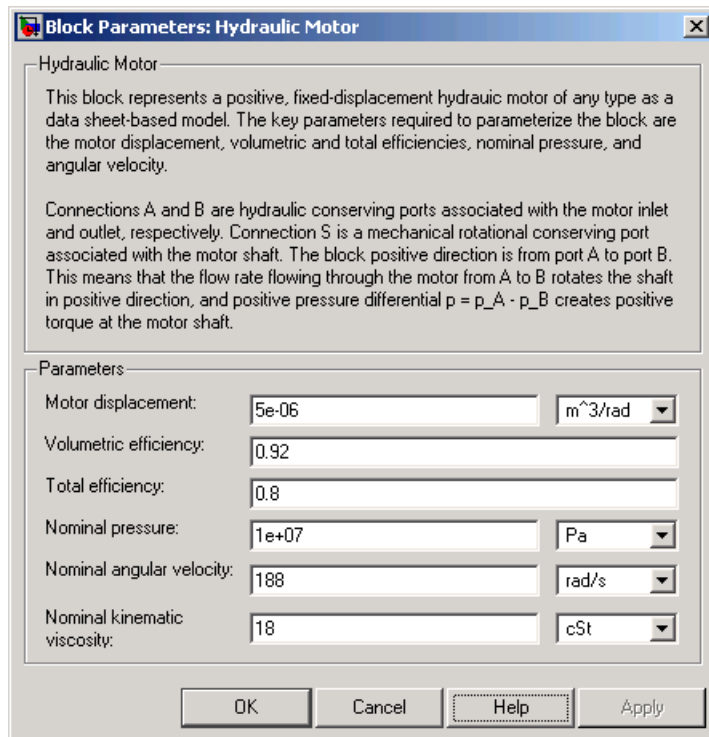
## **Basic Assumptions and Limitations**

The model is based on the following assumptions:

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

# 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  $1 \times 10^7$  Pa.

**Nominal angular velocity**

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

**Nominal kinematic viscosity**

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

**Global Parameters****Fluid kinematic viscosity**

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

**Ports**

The block has the following ports:

A

Hydraulic conserving port associated with the motor inlet.

B

Hydraulic conserving port associated with the motor outlet.

S

Mechanical rotational conserving port associated with the motor output shaft.

**See Also**

Variable-Displacement Motor

# Hydraulic Pipeline

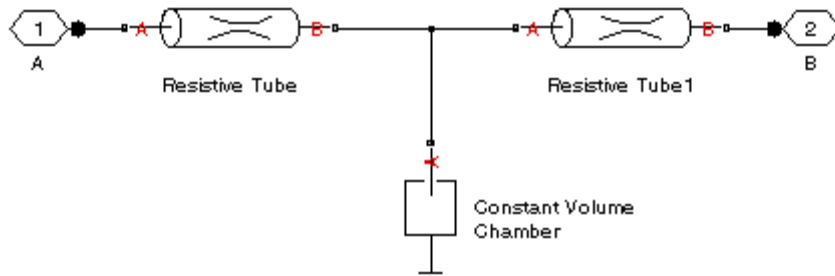
**Purpose** Simulate hydraulic pipeline with resistive and fluid compressibility properties

**Library** Pipelines

**Description** The Hydraulic Pipeline block models hydraulic pipelines with circular and non-circular cross sections. The block accounts for friction loss along the pipe length and for fluid compressibility. The block does not account for fluid inertia and cannot be used for predicting effects like water hammer or changes in pressure caused by fluid acceleration.



The model is built of SimHydraulics building blocks and its schematic diagram is shown below.



The Resistive Tube blocks account for friction losses, while the Constant Volume Chamber block accounts for fluid compressibility. By using the block parameters, you can set the model to simulate pipeline with rigid or compliant walls, including simulation of hydraulic hoses with elastic and viscoelastic properties.

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

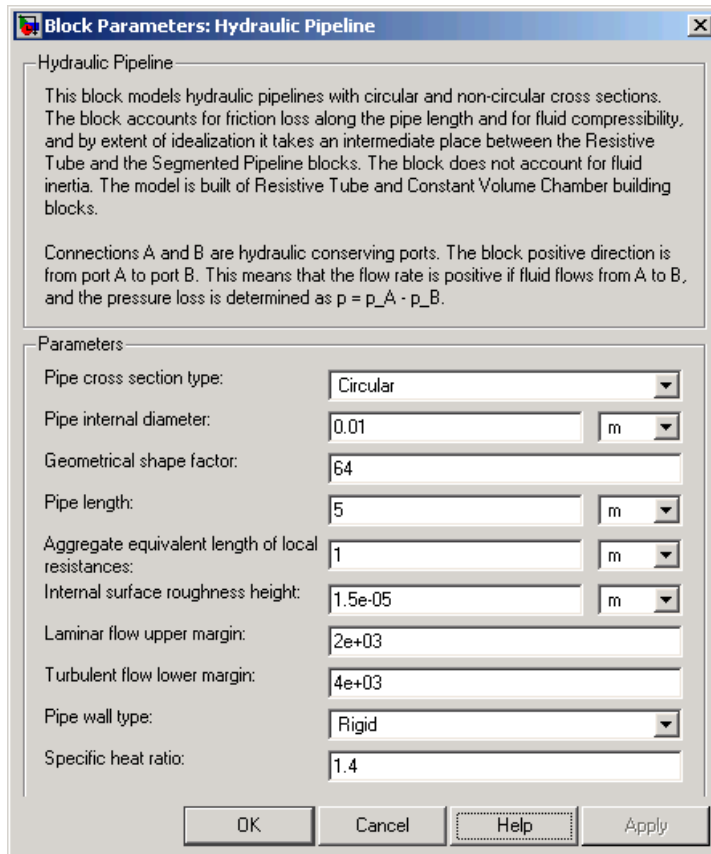


## Basic Assumptions and Limitations

The model is based on the following assumptions:

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

## Dialog Box and Parameters



**Block Parameters: Hydraulic Pipeline**

Hydraulic Pipeline

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

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

Parameters

Pipe cross section type:	Circular
Pipe internal diameter:	0.01 m
Geometrical shape factor:	64
Pipe length:	5 m
Aggregate equivalent length of local resistances:	1 m
Internal surface roughness height:	1.5e-05 m
Laminar flow upper margin:	2e+03
Turbulent flow lower margin:	4e+03
Pipe wall type:	Rigid
Specific heat ratio:	1.4

OK Cancel Help Apply

# Hydraulic Pipeline

**Block Parameters: Hydraulic Pipeline** [X]

Hydraulic Pipeline

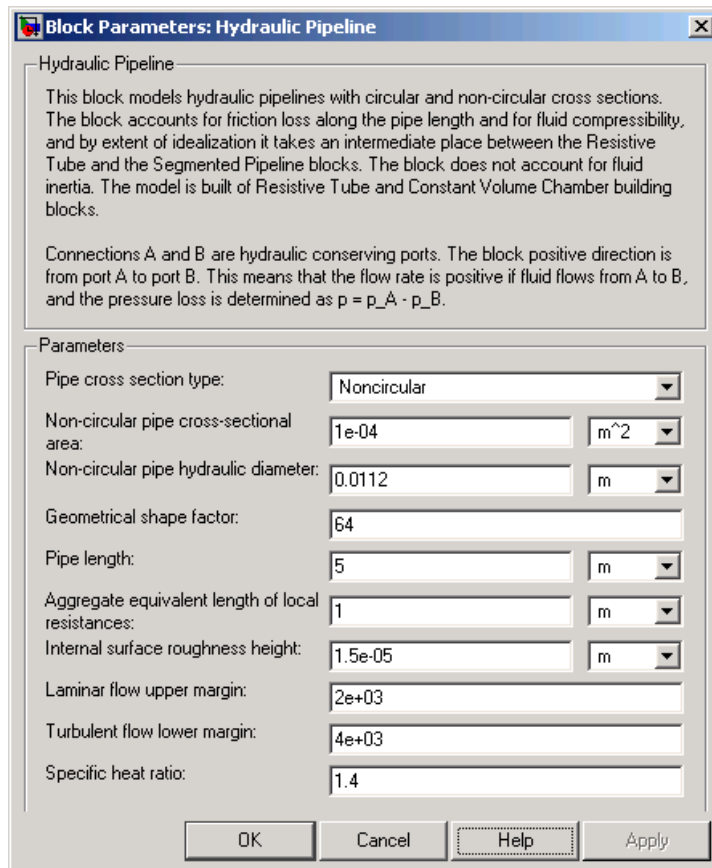
This block models hydraulic pipelines with circular and non-circular cross sections. The block accounts for friction loss along the pipe length and for fluid compressibility, and by extent of idealization it takes an intermediate place between the Resistive Tube and the Segmented Pipeline blocks. The 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-05"/>	<input type="text" value="m"/>
Laminar flow upper margin:	<input type="text" value="2e+03"/>	
Turbulent flow lower margin:	<input type="text" value="4e+03"/>	
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"/>	

OK Cancel Help Apply



## Pipe cross section type

The parameter can have one of two values: Circular or Noncircular. For a circular pipe, you need to specify its internal diameter. For a noncircular pipe, you need to specify its hydraulic diameter and pipe cross-sectional area. The default value of the parameter is Circular.

# Hydraulic Pipeline

---

## **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.

## **Non-circular 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>.

## **Non-circular pipe hydraulic diameter**

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

## **Geometrical shape factor**

The parameter is used for computing friction factor at laminar flow and depends of the shape of the pipe cross section. For a pipe with noncircular cross section, you must set the factor to an appropriate value, for example, 56 for a square, 96 for concentric annulus, 62 for rectangle (2:1), and so on (see [1]). The default value is 64, which corresponds to a pipe with a circular cross section.

## **Pipe length**

Pipe geometrical length. The default value is 5 m.

## **Aggregate equivalent length of local resistances**

This parameter represents total equivalent length of all local resistances associated with the pipe. You can account for the pressure loss caused by local resistances, such as bends, fittings, armature, inlet/outlet losses, and so on, by adding to the pipe geometrical length an aggregate equivalent length of all the local resistances. The default value is 1 m.

## **Internal surface roughness height**

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

## **Laminar flow upper margin**

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

## **Turbulent flow lower margin**

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

## **Pipe wall type**

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

## **Static pressure-diameter coefficient**

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

## **Viscoelastic process time constant**

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

# Hydraulic Pipeline

---

## Specific heat ratio

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

## Global Parameters

### Fluid density

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

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the pipe inlet.

B

Hydraulic conserving port associated with the pipe outlet.

## References

[1] White, F.M., *Viscous Fluid Flow*, McGraw-Hill, 1991

## See Also

Resistive Tube

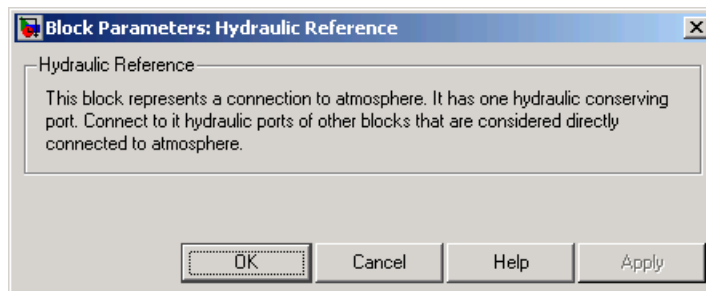
**Purpose** Simulate connection to atmospheric pressure

**Library** Hydraulic Elements

**Description** The Hydraulic Reference block represents a connection to atmospheric pressure. Hydraulic conserving ports of all the blocks that are referenced to atmosphere (for example, suction ports of hydraulic pumps, or return ports of valves, cylinders, pipelines, if they are considered directly connected to atmosphere) must be connected to a Hydraulic Reference block.



## Dialog Box and Parameters



The Hydraulic Reference block has no parameters.

**Ports** The block has one hydraulic conserving port.

**See Also** Mechanical Rotational Reference  
Mechanical Translational Reference

# Ideal Angular Velocity Source

**Purpose** Simulate ideal angular velocity source in mechanical rotational systems

**Library** Mechanical Sensors and Sources

## Description

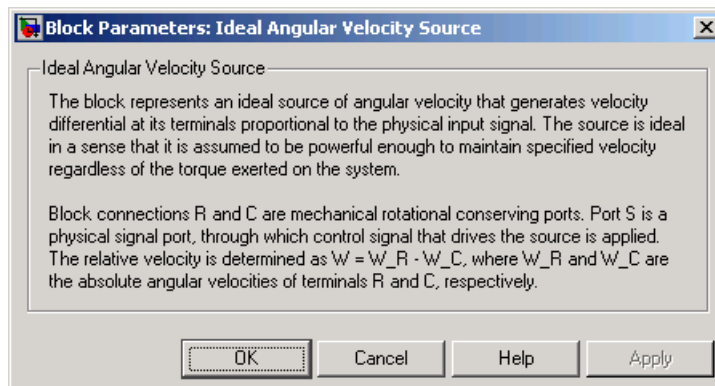


The Ideal Angular Velocity Source block represents an ideal source of angular velocity that generates velocity differential at its terminals proportional to the input physical signal. The source is ideal in a sense that it is assumed to be powerful enough to maintain specified velocity regardless of the torque exerted on the system.

Connections R and C are mechanical rotational conserving ports. Port S is a physical signal port, through which the control signal that drives the source is applied. The relative velocity (velocity differential) across the source is directly proportional to the signal at the control port S. The entire variety of Simulink signal sources can be used to generate the desired velocity variation profile.

The block positive direction is from port R to port C. This means that the velocity is measured as  $\omega = \omega_R - \omega_C$ , where  $\omega_R$ ,  $\omega_C$  are the absolute angular velocities at ports R and C, respectively, and torque through the source is positive if it is directed from R to C. The power generated by the source is negative if the source delivers energy to port R.

## Dialog Box and Parameters



The block has no parameters.



## Ports

The block has the following ports:

R

Mechanical rotational conserving port.

C

Mechanical rotational conserving port associated with the source reference point (case).

S

Physical signal input port, through which the control signal that drives the source is applied.

## See Also

[Ideal Translational Velocity Source](#)

[Ideal Torque Source](#)

[Ideal Force Source](#)

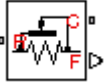
# Ideal Force Sensor

---

**Purpose** Simulate force sensor in mechanical translational systems

**Library** Mechanical Sensors and Sources

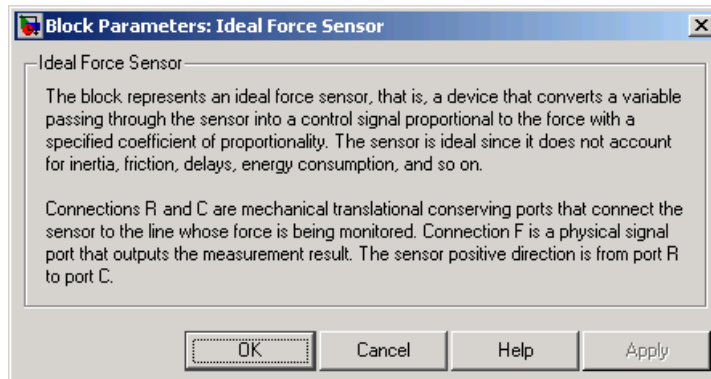
**Description** The Ideal Force Sensor block represents a device that converts a variable passing through the sensor into a control signal proportional to the force. The sensor is ideal since it does not account for inertia, friction, delays, energy consumption, and so on.



Connections R and C are mechanical translational conserving ports that connect the block to the line where force is being monitored. Connection F is a physical signal port that outputs the measurement result.

The block positive direction is from port R to port C. This means that positive force applied to port R (the sensor positive probe) generates a positive output signal.

## Dialog Box and Parameters



The block has no parameters.

**Ports** The block has the following ports:

R  
Mechanical translational conserving port associated with the sensor positive probe.

- C Mechanical translational conserving port associated with the sensor negative (reference) probe.
- F Physical signal output port for force.

## **See Also**

Ideal Torque Sensor

Ideal Rotational Motion Sensor

Ideal Translational Motion Sensor

# Ideal Force Source

**Purpose** Simulate ideal source of mechanical energy that generates force proportional to the input signal

**Library** Mechanical Sensors and Sources

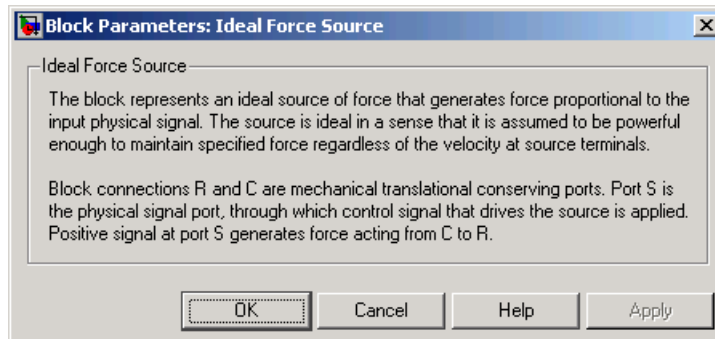
**Description** The Ideal Force Source block represents an ideal source of mechanical energy that generates force proportional to the input physical signal. The source is ideal in a sense that it is assumed to be powerful enough to maintain specified force at its output regardless of the velocity at source terminals.



Connections R and C are mechanical translational conserving ports. Port S is a physical signal port, through which the control signal that drives the source is applied. You can use the entire variety of Simulink signal sources to generate the desired force variation profile. Positive signal at port S generates force acting from C to R. The force generated by the source is directly proportional to the signal at the control port S.

The block positive direction is from port C to port R. This means that the force is positive if it acts in the direction from C to R. The relative velocity is determined as  $v = v_R - v_C$ , where  $v_R$ ,  $v_C$  are the absolute velocities at ports R and C, respectively, and it is negative if velocity at port R is greater than that at port C. The power generated by the source is negative if the source delivers energy to port R.

## Dialog Box and Parameters



The block has no parameters.

## Ports

The block has the following ports:

R

Mechanical translational conserving port.

C

Mechanical translational conserving port associated with the source reference point (case).

S

Physical signal input port, through which the control signal that drives the source is applied.

## See Also

[Ideal Translational Velocity Source](#)

[Ideal Angular Velocity Source](#)

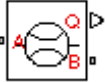
[Ideal Torque Source](#)

# Ideal Hydraulic Flow Rate Sensor

**Purpose** Simulate ideal flow meter

**Library** Hydraulic Sensors and Sources

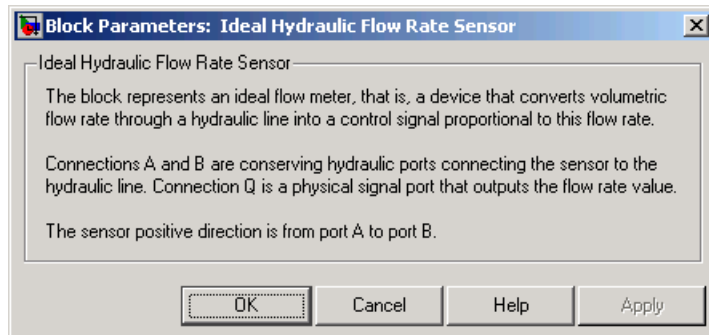
## Description



The Ideal Hydraulic Flow Rate Sensor block represents an ideal flow meter, that is, a device that converts volumetric flow rate through a hydraulic line into a control signal proportional to this flow rate. The sensor is ideal because it does not account for inertia, friction, delays, pressure loss, and so on.

Connections A and B are conserving hydraulic ports connecting the sensor to the hydraulic line. Connection Q is a physical signal port that outputs the flow rate value. The sensor positive direction is from A to B. This means that the flow rate is positive if it flows from A to B.

## Dialog Box and Parameters



The block has no parameters.

## Ports

The block has the following ports:

- A Hydraulic conserving port associated with the sensor positive probe.
- B Hydraulic conserving port associated with the sensor negative (reference) probe.

# Ideal Hydraulic Flow Rate Sensor

---

Q

Physical signal port that outputs the flow rate value.

## **See Also**

Ideal Hydraulic Pressure Sensor

Ideal Hydraulic Flow Rate Source

# Ideal Hydraulic Flow Rate Source

**Purpose** Simulate ideal source of hydraulic energy, characterized by flow rate

**Library** Hydraulic Sensors and Sources

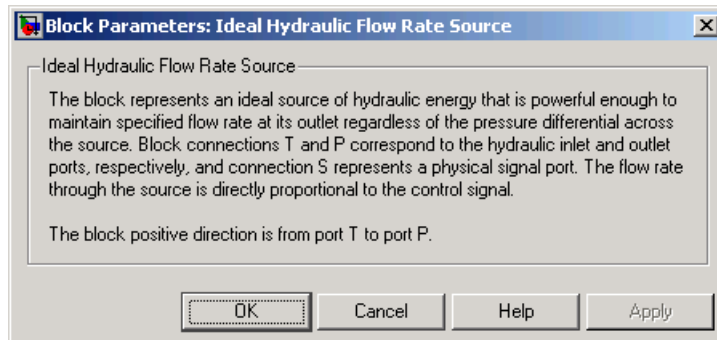
## Description



The Ideal Hydraulic Flow Rate Source block represents an ideal source of hydraulic energy that is powerful enough to maintain specified flow rate at its outlet regardless of the pressure differential across the source. Block connections T and P correspond to the hydraulic inlet and outlet ports, respectively, and connection S represents a control signal port. The flow rate through the source is directly proportional to the signal at the control port S. The entire variety of Simulink signal sources can be used to generate desired flow rate variation profile.

The block positive direction is from port T to port P. This means that the flow rate is positive if it flows from T to P. The pressure differential is determined as  $p = p_T - p_P$  and is negative if pressure at the source outlet is greater than pressure at its inlet. The power generated by the source is negative if the source delivers energy to port P.

## Dialog Box and Parameters



The block has no parameters.

## Ports

The block has the following ports:

T

Hydraulic conserving port associated with the source inlet.



# Ideal Hydraulic Flow Rate Source

---

- P Hydraulic conserving port associated with the source outlet.
- S Control signal port.

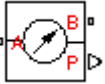
**See Also**      Ideal Hydraulic Pressure Source

# Ideal Hydraulic Pressure Sensor

**Purpose** Simulate ideal pressure sensing device

**Library** Hydraulic Sensors and Sources

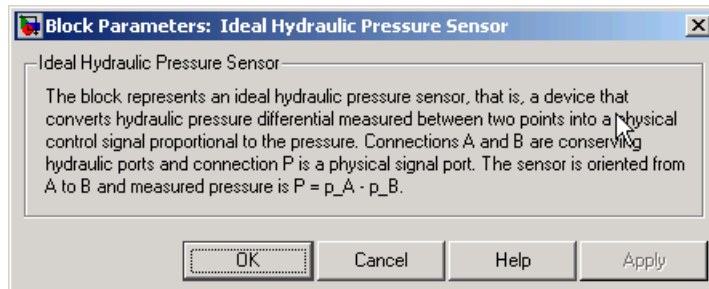
## Description



The Ideal Hydraulic Pressure Sensor block represents an ideal hydraulic pressure sensor, that is, a device that converts hydraulic pressure differential measured between two points into a control signal proportional to this pressure. The sensor is ideal because it does not account for inertia, friction, delays, pressure loss, and so on.

Connections A and B are conserving hydraulic ports connecting the sensor to the hydraulic line. Connection P is a physical signal port that outputs the pressure value. The sensor positive direction is from A to B. This means that the flow rate is positive if it flows from A to B.

## Dialog Box and Parameters



The block has no parameters.

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the sensor positive probe.

B

Hydraulic conserving port associated with the sensor negative (reference) probe.

# Ideal Hydraulic Pressure Sensor

---

P

Physical signal port that outputs the pressure value.

## **See Also**

Ideal Hydraulic Flow Rate Sensor

Ideal Hydraulic Pressure Source

# Ideal Hydraulic Pressure Source

**Purpose** Simulate ideal source of hydraulic energy, characterized by pressure

**Library** Hydraulic Sensors and Sources

**Description** The Ideal Hydraulic Pressure Source block represents an ideal source of hydraulic energy that is powerful enough to maintain specified pressure at its outlet regardless of the flow rate consumed by the system. Block connections T and P correspond to the hydraulic inlet and outlet ports, respectively, and connection S represents a control signal port. The pressure differential across the source

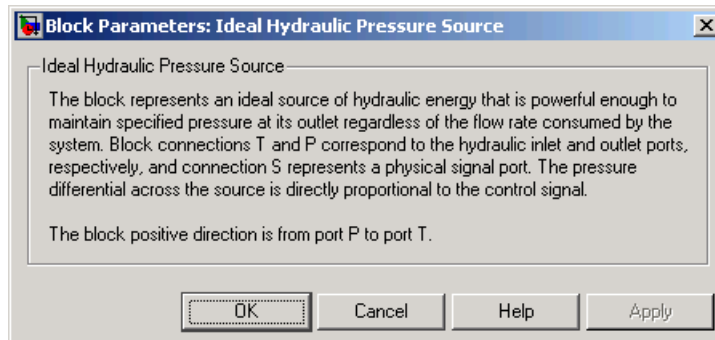


$$p = p_P - p_T$$

where  $p_P$ ,  $p_T$  are the gauge pressures at the source ports, is directly proportional to the signal at the control port S. The entire variety of Simulink signal sources can be used to generate desired pressure variation profile.

The block positive direction is from port P to port T. This means that the flow rate is positive if it flows from P to T. The power generated by the source is negative if the source delivers energy to port P.

## Dialog Box and Parameters



The block has no parameters.

**Ports** The block has the following ports:

# Ideal Hydraulic Pressure Source

---

- P Hydraulic conserving port associated with the source inlet.
- T Hydraulic conserving port associated with the source outlet.
- S Control signal port.

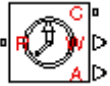
**See Also** [Ideal Hydraulic Flow Rate Source](#)

# Ideal Rotational Motion Sensor

**Purpose** Simulate motion sensor in mechanical rotational systems

**Library** Mechanical Sensors and Sources

## Description



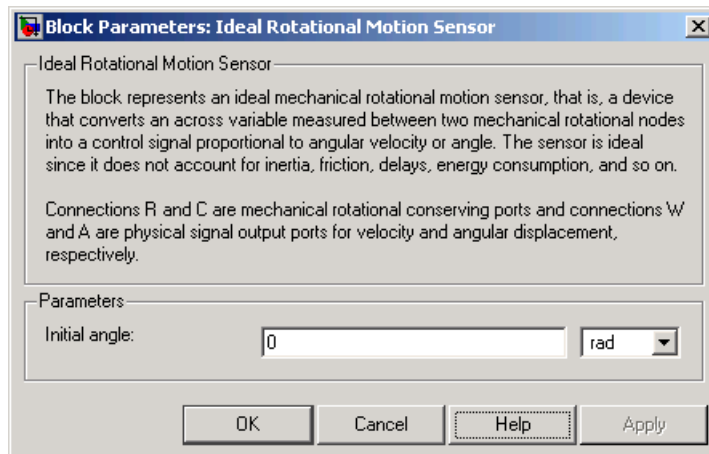
The Ideal Rotational Motion Sensor block represents an ideal mechanical rotational motion sensor, that is, a device that converts an across variable measured between two mechanical rotational nodes into a control signal proportional to angular velocity or angle. You can specify the initial angular position (offset) as a block parameter.

The sensor is ideal since it does not account for inertia, friction, delays, energy consumption, and so on.

Connections R and C are mechanical rotational conserving ports that connect the block to the nodes whose motion is being monitored. Connections W and A are physical signal output ports for velocity and angular displacement, respectively.

The block positive direction is from port R to port C. This means that the velocity is measured as  $\omega = \omega_R - \omega_C$ , where  $\omega_R$ ,  $\omega_C$  are the absolute angular velocities at ports R and C, respectively.

## Dialog Box and Parameters



**Initial angle**

Sensor initial angle, or offset (rad). The default value is 0.

**Ports**

The block has the following ports:

R

Mechanical rotational conserving port associated with the sensor positive probe.

C

Mechanical rotational conserving port associated with the sensor negative (reference) probe.

W

Physical signal output port for angular velocity.

A

Physical signal output port for angular displacement.

**See Also**

Ideal Translational Motion Sensor

Ideal Force Sensor

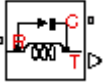
Ideal Torque Sensor

# Ideal Torque Sensor

**Purpose** Simulate torque sensor in mechanical rotational systems

**Library** Mechanical Sensors and Sources

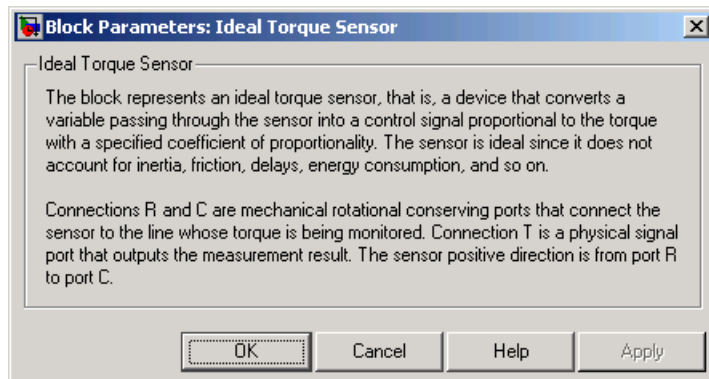
**Description** The Ideal Torque Sensor block represents a device that converts a variable passing through the sensor into a control signal proportional to the torque. The sensor is ideal since it does not account for inertia, friction, delays, energy consumption, and so on.



Connections R and C are mechanical rotational conserving ports that connect the block to the line where torque is being monitored. Connection T is a physical signal port that outputs the measurement result.

The block positive direction is from port R to port C.

## Dialog Box and Parameters



The block has no parameters.

**Ports** The block has the following ports:

R  
Mechanical rotational conserving port associated with the sensor positive probe.



- C Mechanical rotational conserving port associated with the sensor negative (reference) probe.
- T Physical signal output port for torque.

## **See Also**

Ideal Rotational Motion Sensor  
Ideal Translational Motion Sensor  
Ideal Force Sensor

# Ideal Torque Source

## Purpose

Simulate ideal source of mechanical energy that generates torque proportional to the input signal

## Library

Mechanical Sensors and Sources

## Description

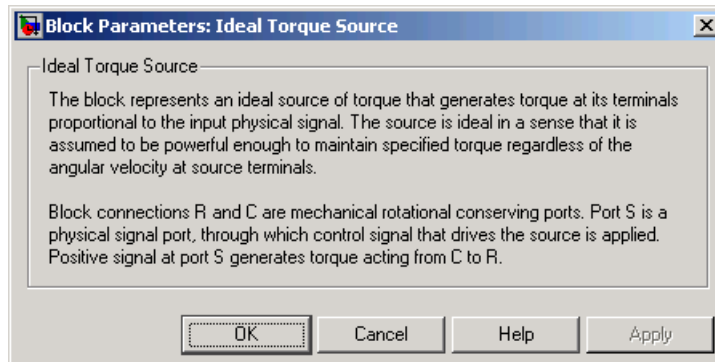


The Ideal Torque Source block represents an ideal source of mechanical energy that generates torque proportional to the input physical signal. The source is ideal in a sense that it is assumed to be powerful enough to maintain specified torque regardless of the angular velocity at source terminals.

Connections R and C are mechanical rotational conserving ports. Port S is a physical signal port, through which the control signal that drives the source is applied. You can use the entire variety of Simulink signal sources to generate the desired torque variation profile. Positive signal at port S generates torque acting from C to R. The torque generated by the source is directly proportional to the signal at the control port S.

The block positive direction is from port C to port R. This means that the torque is positive if it acts in the direction from C to R. The relative velocity is determined as  $\omega = \omega_R - \omega_C$ , where  $\omega_R$ ,  $\omega_C$  are the absolute angular velocities at ports R and C, respectively, and it is negative if velocity at port R is greater than that at port C. The power generated by the source is negative if the source delivers energy to port R.

## Dialog Box and Parameters



The block has no parameters.

## Ports

The block has the following ports:

R

Mechanical rotational conserving port.

C

Mechanical rotational conserving port associated with the source reference point (case).

S

Physical signal input port, through which the control signal that drives the source is applied.

## See Also

[Ideal Angular Velocity Source](#)

[Ideal Translational Velocity Source](#)

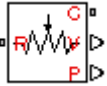
[Ideal Force Source](#)

# Ideal Translational Motion Sensor

**Purpose** Simulate motion sensor in mechanical translational systems

**Library** Mechanical Sensors and Sources

**Description** The Ideal Translational Motion Sensor block represents a device that converts an across variable measured between two mechanical translational nodes into a control signal proportional to velocity or position. You can specify the initial position (offset) as a block parameter.

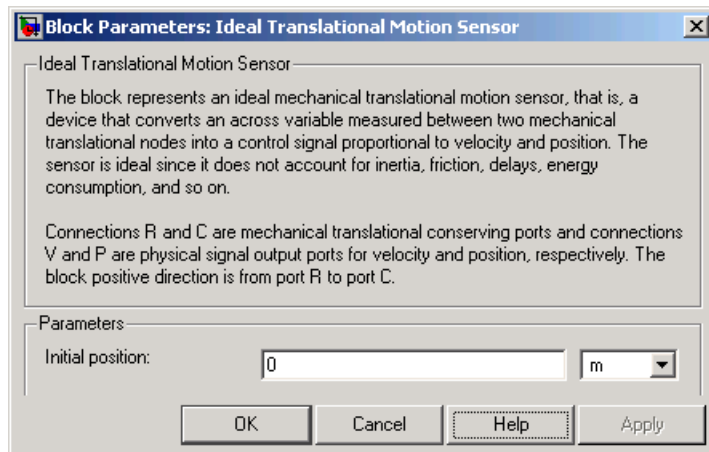


The sensor is ideal since it does not account for inertia, friction, delays, energy consumption, and so on.

Connections R and C are mechanical translational conserving ports that connect the block to the nodes whose motion is being monitored. Connections V and P are physical signal output ports for velocity and position, respectively.

The block positive direction is from port R to port C. This means that the velocity is measured as  $v = v_R - v_C$ , where  $v_R, v_C$  are the absolute velocities at ports R and C, respectively.

## Dialog Box and Parameters



# Ideal Translational Motion Sensor

---

## **Initial position**

Sensor initial position, or offset (m). The default value is 0.

## **Ports**

The block has the following ports:

R

Mechanical translational conserving port associated with the sensor positive probe.

C

Mechanical translational conserving port associated with the sensor negative (reference) probe.

V

Physical signal output port for velocity.

P

Physical signal output port for position.

## **See Also**

Ideal Rotational Motion Sensor

Ideal Force Sensor

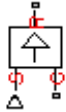
Ideal Torque Sensor

# Ideal Translational Velocity Source

**Purpose** Simulate ideal velocity source in mechanical translational systems

**Library** Mechanical Sensors and Sources

## Description

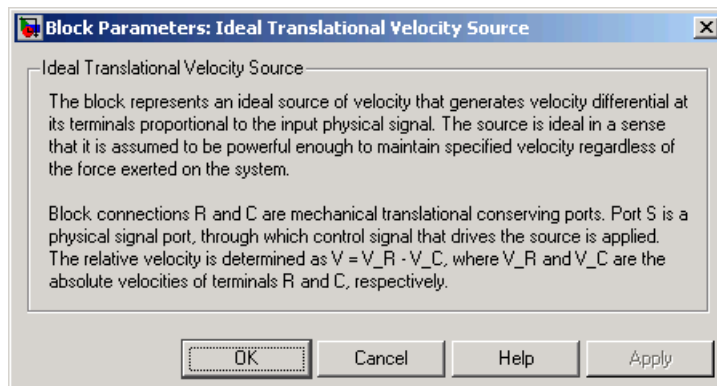


The Ideal Translational Velocity Source block represents an ideal source of velocity that generates velocity differential at its terminals proportional to the input physical signal. The source is ideal in a sense that it is assumed to be powerful enough to maintain specified velocity regardless of the force exerted on the system.

Connections R and C are mechanical translational conserving ports. Port S is a physical signal port, through which the control signal that drives the source is applied. The relative velocity (velocity differential) across the source is directly proportional to the signal at the control port S. The entire variety of Simulink signal sources can be used to generate the desired velocity variation profile.

The block positive direction is from port R to port C. This means that the velocity is measured as  $v = v_R - v_C$ , where  $v_R$ ,  $v_C$  are the absolute velocities at ports R and C, respectively, and force through the source is negative if it acts from C to R. The power generated by the source is negative if the source delivers energy to port R.

## Dialog Box and Parameters



The block has no parameters.

# Ideal Translational Velocity Source

---

## Ports

The block has the following ports:

R

Mechanical translational conserving port.

C

Mechanical translational conserving port associated with the source reference point (case).

S

Physical signal input port, through which the control signal that drives the source is applied.

## See Also

[Ideal Angular Velocity Source](#)

[Ideal Torque Source](#)

[Ideal Force Source](#)

# Inertia

**Purpose** Simulate inertia in mechanical rotational systems

**Library** Mechanical Rotational Elements

**Description** The Inertia block represents an ideal mechanical rotational inertia, described with the following equation:



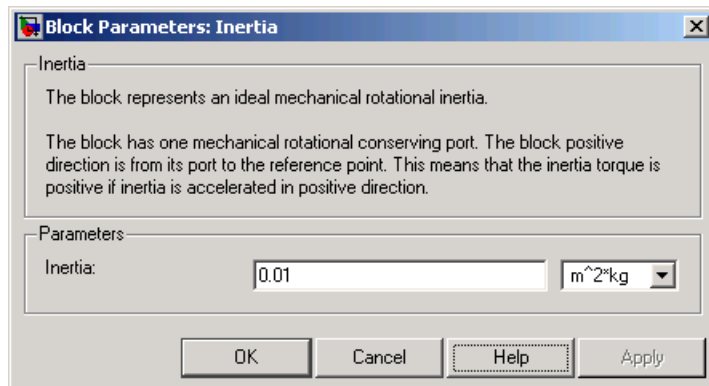
$$T = J \frac{d\omega}{dt}$$

where

- $T$  Inertia torque
- $J$  Inertia
- $\omega$  Angular velocity
- $t$  Time

The block has one mechanical rotational conserving port. The block positive direction is from its port to the reference point. This means that the inertia torque is positive if inertia is accelerated in positive direction.

## Dialog Box and Parameters





**Inertia**

Inertia. The default value is  $0.001 \text{ kg}\cdot\text{m}^2$ .

**Ports**

The block has one mechanical rotational conserving port, associated with the inertia connection to the system.

**See Also**

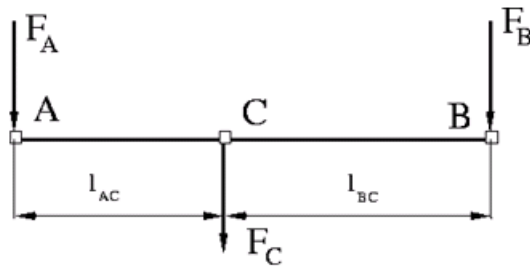
Mass

# Lever

**Purpose** Simulate lever in mechanical systems

**Library** Mechanisms

**Description** The Lever block represents a mechanical lever in its generic form, known as a free or summing lever, shown in the following schematic.



The summing lever equations are derived with the assumption of small angle deviation from initial position:

$$v_C = K_{AC} \cdot v_A + K_{BC} \cdot v_B$$

$$F_A = K_{AC} \cdot F_C$$

$$F_B = K_{BC} \cdot F_C$$

$$K_{AC} = \frac{l_{BC}}{l_{AC} + l_{BC}}$$

$$K_{BC} = \frac{l_{AC}}{l_{AC} + l_{BC}}$$

where

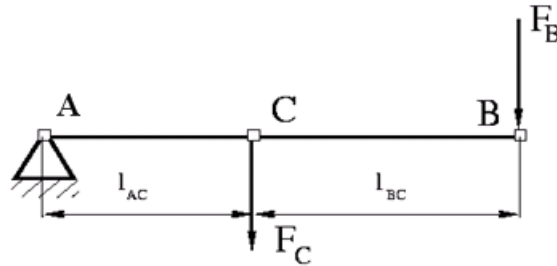
$v_A, v_B, v_C$  Lever joints velocities

$F_A, F_B, F_C$  Lever joints forces

$l_{AC}, l_{BC}$  Arm lengths

The above equations were derived with the assumption that the lever sums forces and motions at node C. The assumption was arbitrary and does not impose any limitations on how the forces or motions are applied to the lever. In other words, any of the lever nodes can be “input” or “output” nodes, depending on the value of the force. Moreover, any of the block nodes can be connected to the reference point, thus converting a three-node lever into a first-class lever, with the fulcrum at the end, or a second-class lever, with the fulcrum in the middle.

The following illustration shows a schematic of a two-node first-class lever, with the fulcrum at node A.



It is described with the following equations:

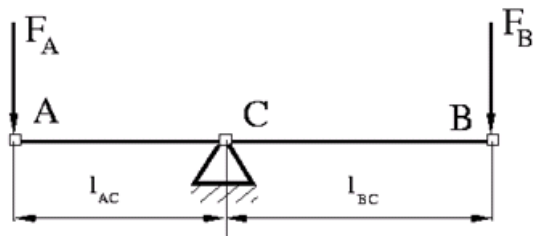
$$v_C = K_{BC} \cdot v_B$$

$$F_B = K_{BC} \cdot F_C$$

The next illustration shows a schematic of a second-class lever, with the fulcrum in the middle.

# Lever

---



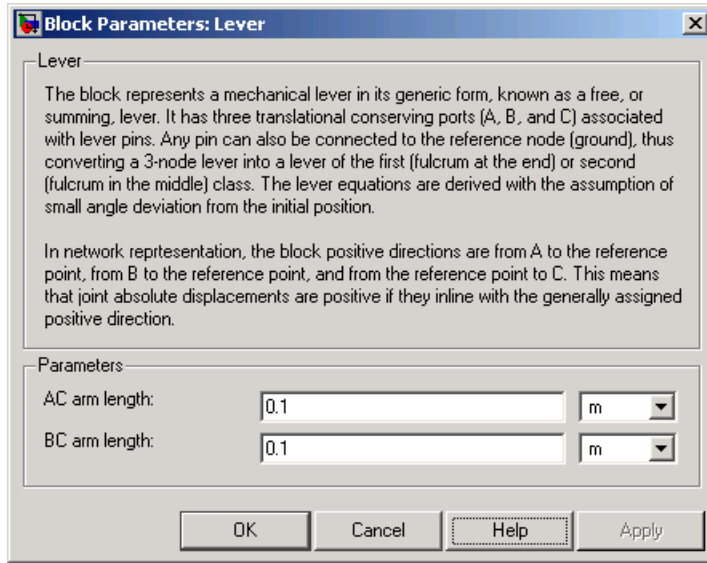
It is described with the following equations:

$$v_A = -\frac{l_{AC}}{l_{BC}} \cdot v_B$$

$$F_B = -\frac{l_{AC}}{l_{BC}} \cdot F_A$$

As far as the block directionality is concerned, the joints' absolute displacements are positive if they are in line with the globally assigned positive direction.

## Dialog Box and Parameters



### AC arm length

Arm length between nodes A and C. The default value is 0.1 m.

### BC arm length

Arm length between nodes B and C. The default value is 0.1 m.

## Ports

The block has the following ports:

A

Mechanical translational conserving port associated with the node A of the lever.

B

Mechanical translational conserving port associated with the node B of the lever.

C

Mechanical translational conserving port associated with the node C of the lever.

# Lever

---

## Examples

The Linkage Mechanism demo (sh\_linkage\_mechanism) illustrates the use of the Lever block in three different modes. Linkages L\_1 and L\_4 simulate first-class levers with the fulcrum at the end. Linkage L\_2 represents a summing lever. Linkage L\_3 simulates a second-class lever with the fulcrum in the middle.

## Purpose

Simulate hydraulic pipeline with linear resistance losses

## Library

Hydraulic Elements

## Description

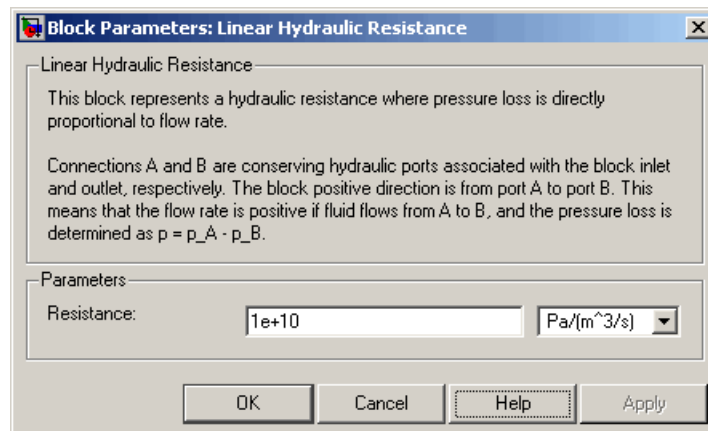


The Linear Hydraulic Resistance block represents a hydraulic resistance where pressure loss is directly proportional to flow rate. This block can be useful at preliminary stages of development, or as a powerful means to speed up the simulation, especially if the flow rate varies insignificantly with respect to the operating point.

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  $p = p_A - p_B$ .

## Dialog Box and Parameters



## Resistance

The linear resistance coefficient. The default value is  $10e9$  Pa/m<sup>3</sup>/s.

## Ports

The block has the following ports:

# Linear Hydraulic Resistance

---

- A Hydraulic conserving port associated with the resistance inlet.
- B Hydraulic conserving port associated with the resistance outlet.

**See Also** Resistive Tube



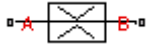
## Purpose

Simulate all kinds of hydraulic resistances specified by loss coefficient

## Library

Local Hydraulic Resistances

## Description



The Local Resistance block represents a generic local hydraulic resistance, such as a bend, elbow, fitting, filter, local change in the flow cross section, and so on. The pressure loss caused by resistance is computed 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 versus Reynolds number.

The pressure loss is determined according to the following equations:

$$p = K \frac{\rho}{2A^2} q | q |$$

$$p = p_A - p_B$$

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

$$\text{Re} = \frac{q \cdot d}{A \cdot v}$$

where

$q$  Flow rate

$\rho$  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

$\text{Re}$  Reynolds number

# Local Resistance

---

$A$	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.
- By table-specified  $K=f(Re)$  relationship — The pressure loss coefficient is specified as function of the Reynolds number.

The resistance can be symmetrical or asymmetrical. In symmetrical resistances, the pressure loss practically does not depend on flow direction and one value of the coefficient is used for both the direct and reverse flow. For asymmetrical resistances, the separate coefficients are provided for each flow direction. If the loss coefficient is specified by a table, the table must cover both the positive and the negative flow regions.

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

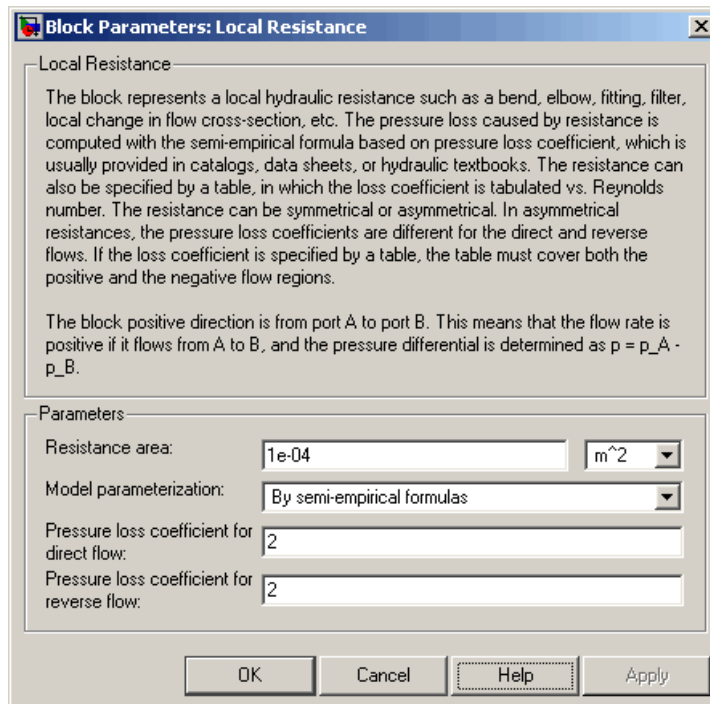
The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure loss is determined as  $p = p_A - p_B$ .

## Basic Assumptions and Limitations

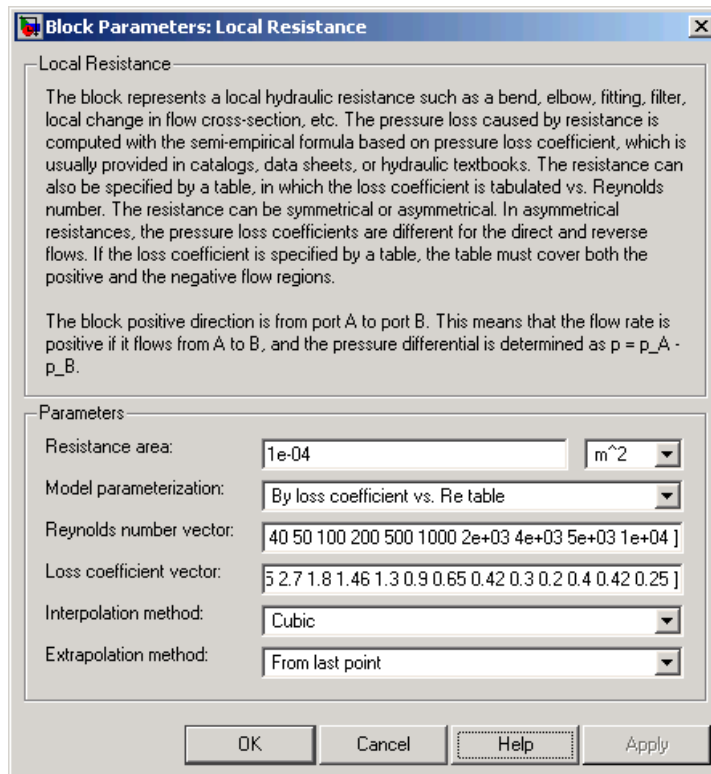
The model is based on the following assumption:

- Fluid inertia is not taken into account.
- If you select parameterization by semi-empirical formulas, the flow is assumed to be completely turbulent and fully developed along the pipe length.

## Dialog Box and Parameters



# 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`.

### **Reynolds number vector**

Specify the vector of input values for Reynolds numbers as a tabulated 1-by-m array. The input values vector must be strictly monotonically increasing. The values can be non-uniformly spaced. You must provide at least three values. The default values are [-4000, -3000, -2000, -1000, -500, -200, -100, -50, -40, -30, -20, -15, -10, 10, 20, 30, 40, 50, 100, 200, 500, 1000, 2000, 4000, 5000, 10000]. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

### **Loss coefficient vector**

Specify the vector of output values for the loss coefficient as a tabulated 1-by-m array. The loss coefficient vector must be the same size as the Reynolds numbers vector. The default values are [0.25, 0.3, 0.65, 0.9, 0.65, 0.75, 0.90, 1.15, 1.35, 1.65, 2.3, 2.8, 3.10, 5, 2.7, 1.8, 1.46, 1.3,

# Local Resistance

---

0.9, 0.65, 0.42, 0.3, 0.20, 0.40, 0.42, 0.25]. This parameter is used if **Model parameterization** is set to By loss coefficient vs. Re table.

## Interpolation method

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

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

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to By loss coefficient vs. Re table.

## Extrapolation method

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

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

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to By loss coefficient vs. Re table.

## Global Parameters

### Fluid density

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

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the resistance inlet.

B

Hydraulic conserving port associated with the resistance outlet.

## References

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

## See Also

Elbow

Gradual Area Change

Pipe Bend

Sudden Area Change

T-junction

# Mass

**Purpose** Simulate mass in mechanical translational systems

**Library** Mechanical Translational Elements

**Description** The Mass block represents an ideal mechanical translational mass, described with the following equation:



$$F = m \frac{dv}{dt}$$

where

$F$  Inertia force

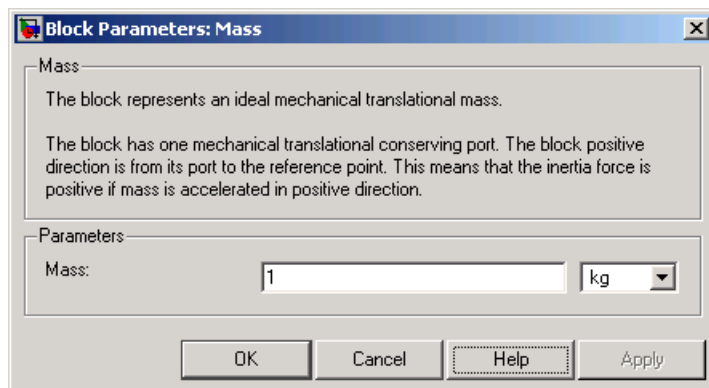
$m$  Mass

$v$  Velocity

$t$  Time

The block has one mechanical translational conserving port. The block positive direction is from its port to the reference point. This means that the inertia force is positive if mass is accelerated in positive direction.

## Dialog Box and Parameters





**Mass**

Mass. The default value is 1 kg.

**Ports**

The block has one mechanical translational conserving port, associated with the mass connection to the system.

**See Also**

Inertia

# Mechanical Rotational Reference

---

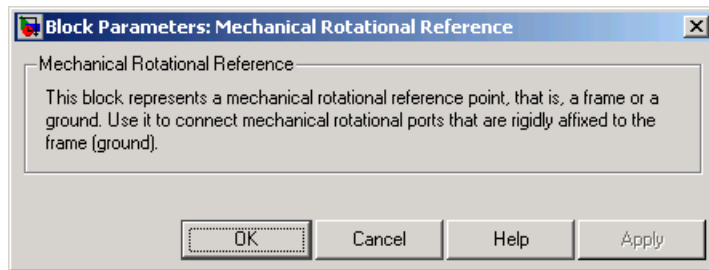
**Purpose** Simulate reference for mechanical rotational ports

**Library** Mechanical Rotational Elements

**Description** The Mechanical Rotational Reference block represents a reference point, or frame, for all mechanical rotational ports. All rotational ports that are rigidly clamped to the frame (ground) must be connected to a Mechanical Rotational Reference block.



## Dialog Box and Parameters



The Mechanical Rotational Reference block has no parameters.

**Ports** The block has one mechanical rotational port.

**See Also** Mechanical Translational Reference  
Hydraulic Reference

# Mechanical Translational Reference

**Purpose** Simulate reference for mechanical translational ports

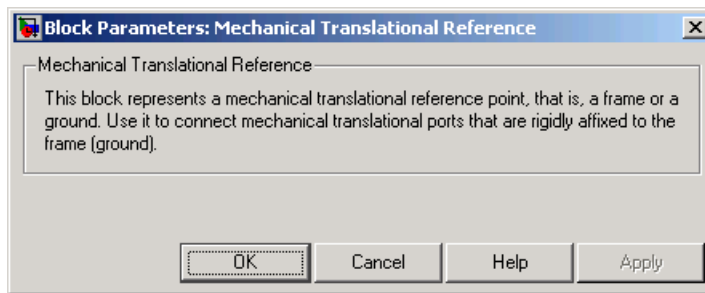
**Library** Mechanical Translational Elements

## Description



The Mechanical Translational Reference block represents a reference point, or frame, for all mechanical translational ports. All translational ports that are rigidly clamped to the frame (ground) must be connected to a Mechanical Translational Reference block.

## Dialog Box and Parameters



The Mechanical Translational Reference block has no parameters.

**Ports** The block has one mechanical translational port.

**See Also** Mechanical Rotational Reference

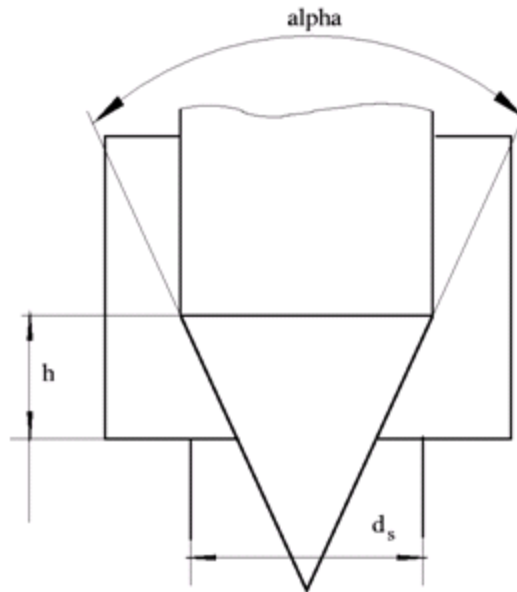
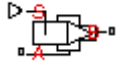
Hydraulic Reference

# Needle Valve

**Purpose** Simulate hydraulic needle valve

**Library** Flow Control Valves

**Description** The Needle Valve block models a variable orifice created by a conical needle and a round sharp-edged orifice in thin material.



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

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

$$h = x_0 + x$$

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

$$p = p_A - p_B$$

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

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

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

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

where

$q$  Flow rate

$\rho$  Pressure differential

$p_A, p_B$  Gauge pressures at the block terminals

# Needle Valve

---

$C_D$	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
$x_0$	Initial opening
$x$	Needle displacement from initial position
$h$	Valve opening
$d_s$	Orifice diameter
$\alpha$	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

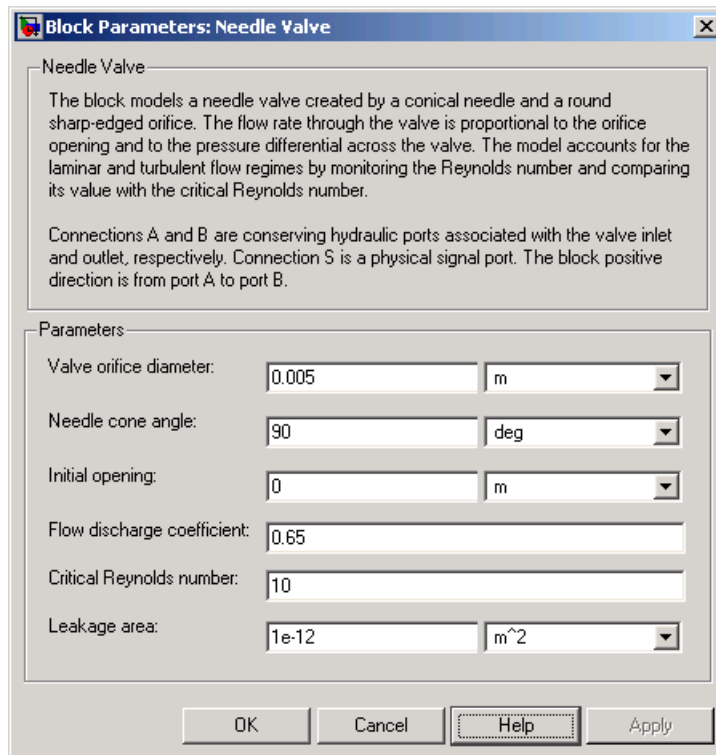
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as  $p = p_A - p_B$ . Positive signal at the physical signal port S opens the valve.

## Basic Assumptions and Limitations

The model is based on the following assumptions:

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

## Dialog Box and Parameters



### Variable orifice diameter

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

### Valve 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. The parameter value must be nonnegative. The default value is 0.

# Needle 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 0.65.

## **Critical Reynolds number**

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

## **Leakage area**

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is  $1e-12 \text{ m}^2$ .

## **Global Parameters**

### **Fluid density**

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

### **Fluid kinematic viscosity**

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

## **Ports**

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.



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

## **See Also**

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

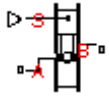
# Orifice with Variable Area Round Holes

---

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

**Library** Hydraulic Elements

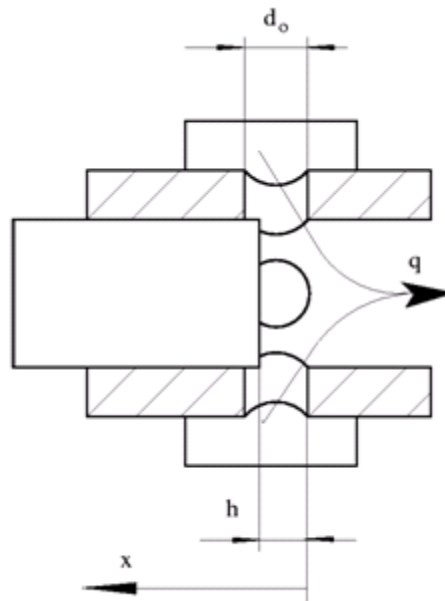
## Description



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

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

# Orifice with Variable Area Round Holes



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

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

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

# Orifice with Variable Area Round Holes

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

$$p = p_A - p_B$$

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

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

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

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

where

- $q$  Flow rate
- $p$  Pressure differential
- $p_A, p_B$  Gauge pressures at the block terminals
- $C_D$  Flow discharge coefficient
- $A(h)$  Instantaneous orifice passage area
- $d_0$  Hole diameter
- $z$  Number of holes
- $x_0$  Initial opening

# Orifice with Variable Area Round Holes

$x$	Spool displacement from initial position
$h$	Orifice opening
$or$	Orifice orientation indicator. The variable assumes +1 value if a spool displacement in the globally assigned positive direction opens the orifice, and $-1$ if positive motion decreases the opening.
$\rho$	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**.

## Basic Assumptions and Limitations

The model is based on the following assumptions:

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

# Orifice with Variable Area Round Holes

## Dialog Box and Parameters

**Block Parameters: Orifice with Variable Area Round Holes**

Orifice with Variable Area Round Holes

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

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

Parameters

Diameter of round holes: 0.005 m

Number of round holes: 6

Flow discharge coefficient: 0.6

Initial opening: 0 m

Orifice orientation: Opens in positive direction

Critical Reynolds number: 12

Leakage area: 1e-15 m<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.

### **Flow discharge coefficient**

Semi-empirical parameter for orifice capacity characterization.

Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets.

The default value is 0.6.

# Orifice with Variable Area Round Holes

---

## **Initial opening**

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

## **Orifice orientation**

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

## **Critical Reynolds number**

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

## **Leakage area**

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is  $1e-15 \text{ m}^2$ .

# Orifice with Variable Area Round Holes

---

## Global Parameters

### Fluid density

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

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the orifice inlet.

B

Hydraulic conserving port associated with the orifice outlet.

S

Physical signal port to control spool displacement.

The flow rate is positive if fluid flows from port A to port B. Positive signal at the physical signal port S opens or closes the orifice depending on the value of the parameter **Orifice orientation**.

## See Also

Annular Orifice

Fixed Orifice

Orifice with Variable Area Slot

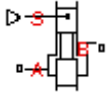
Variable Orifice



**Purpose** Simulate hydraulic variable orifice shaped as rectangular slot

**Library** Hydraulic Elements

## Description



The block models a variable orifice created by a cylindrical sharp-edged spool and a rectangular slot in a sleeve. The flow rate through the orifice is proportional to the orifice opening and to the pressure differential across the orifice. The model accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number ( $Re$ ) and comparing its value with the critical Reynolds number ( $Re_{cr}$ ). The flow rate is determined according to the following equations:

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

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

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

$$p = p_A - p_B$$

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

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

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

# Orifice with Variable Area Slot

---

where

$q$	Flow rate
$p$	Pressure differential
$p_A, p_B$	Gauge pressures at the block terminals
$C_D$	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
$b$	Width of the orifice slot
$x_0$	Initial opening
$x$	Spool displacement from initial position
$h$	Orifice opening
$or$	Orifice orientation indicator. The variable assumes +1 value if a spool displacement in the globally assigned positive direction opens the orifice, and $-1$ if positive motion decreases the opening.
$\rho$	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

The model is based on the following assumptions:

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

## Dialog Box and Parameters

**Block Parameters: Orifice with Variable Area Slot**

Orifice with Variable Area Slot

The block models a variable orifice created by a cylindrical sharp-edged spool and a rectangular slot in a sleeve. The flow rate through the orifice is proportional to the orifice opening and to the pressure differential across the orifice.

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

Parameters:

Orifice width:	0.01	m
Flow discharge coefficient:	0.7	
Initial opening:	0	m
Orifice orientation:	Opens in positive direction	
Critical Reynolds number:	12	
Leakage area:	1e-12	m <sup>2</sup>

OK Cancel Help Apply

### Orifice width

The width of the rectangular slot. The default value is 1 e-2 m.

### Flow discharge coefficient

Semi-empirical parameter for orifice capacity characterization.

Its value depends on the geometrical properties of the orifice, and

# Orifice with Variable Area Slot

---

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

## **Initial opening**

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

## **Orifice orientation**

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

## **Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

## **Leakage area**

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is  $1e-12 \text{ m}^2$ .

## Global Parameters

### Fluid density

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

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the orifice inlet.

B

Hydraulic conserving port associated with the orifice outlet.

S

Physical signal port to control spool displacement.

The flow rate is positive if fluid flows from port A to port B. Positive signal at the physical signal port S opens or closes the orifice depending on the value of the parameter **Orifice orientation**.

## See Also

Annular Orifice

Fixed Orifice

Orifice with Variable Area Round Holes

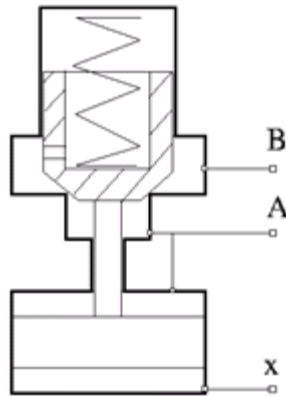
Variable Orifice

# Pilot-Operated Check Valve

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

**Library** Directional Valves

**Description** The Pilot-Operated Check Valve block represents a hydraulic pilot-operated check valve as a data-sheet-based model. The purpose of the check valve is to permit flow in one direction and block it in the opposite direction, as shown in the following figure.



Unlike a conventional check valve, the pilot-operated check valve can be opened by inlet pressure  $p_A$ , pilot pressure  $p_X$ , or both. The force acting on the poppet is determined as

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

where

$p_A, p_B$  Gauge pressures at the valve terminals

$p_X$  Gauge pressure at the pilot terminal

$A_A$  Area of the spool in the A chamber

$A_B$  Area of the spool in the B chamber

$A_X$  Area of the pilot chamber

This equation is commonly used in a slightly modified form

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

where  $k_p = A_X/A_A$  is usually referred to as pilot ratio and  $p_e$  is the equivalent pressure differential across the poppet. The valve remains closed while this pressure differential across the valve is lower than the valve cracking pressure. When cracking pressure is reached, the valve control member (spool, ball, poppet, etc.) is forced off its seat, thus creating a passage between the inlet and outlet. If the flow rate is high enough and pressure continues to rise, the area is further increased until the control member reaches its maximum. At this moment, the valve passage area is at its maximum. The valve maximum area and the cracking and maximum pressures are generally provided in the catalogs and are the three key parameters of the block.

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

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

# Pilot-Operated Check Valve

---

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

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

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

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

$$p = p_A - p_B$$

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

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

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

where

- $q$  Flow rate through the valve
- $p$  Pressure differential across the valve
- $p_e$  Equivalent pressure differential across the control member



$p_A, p_B$	Gauge pressures at the valve terminals
$p_X$	Gauge pressure at the pilot terminal
$k_p$	Pilot ratio, $k_p = p_X/p_A$
$k$	Valve gain coefficient
$C_D$	Flow discharge coefficient
$A(\rho)$	Instantaneous orifice passage area
$A_{max}$	Fully open valve passage area
$A_{leak}$	Closed valve leakage area
$p_{crack}$	Valve cracking pressure
$p_{max}$	Pressure needed to fully open the valve
$D_H$	Instantaneous orifice hydraulic diameter
$\rho$	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  $p = p_A - p_B$ .

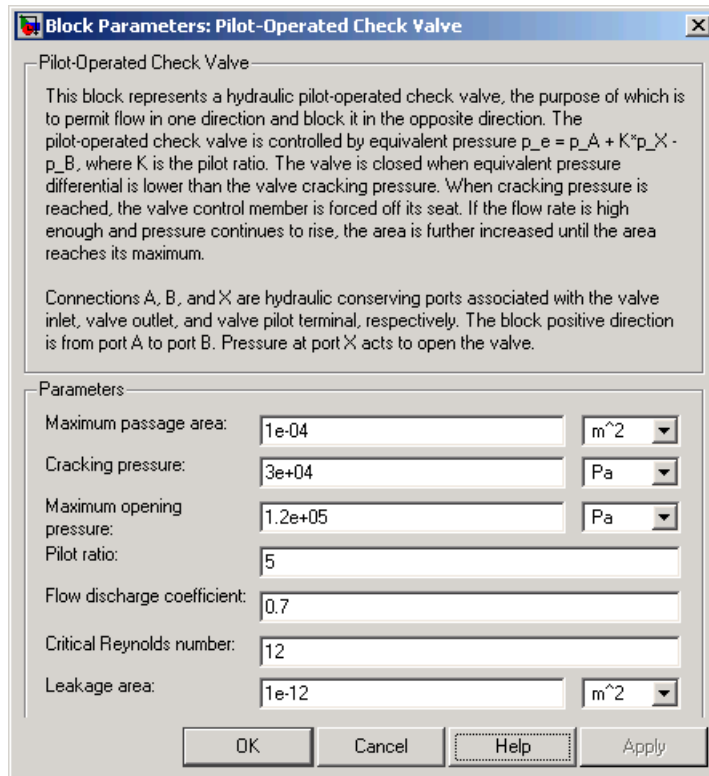
## Basic Assumptions and Limitations

The model is based on the following assumptions:

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

# Pilot-Operated Check Valve

## Dialog Box and Parameters



### Maximum passage area

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

### Cracking pressure

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

### Maximum opening pressure

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

**Pilot ratio**

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

**Flow discharge coefficient**

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

**Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

**Leakage area**

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is  $1e-12 \text{ m}^2$ .

**Global Parameters****Fluid density**

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

**Fluid kinematic viscosity**

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

**Ports**

The block has the following ports:

# Pilot-Operated Check Valve

---

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

**See Also** Check Valve

**Purpose** Simulate hydraulic resistance in pipe bend

**Library** Local Hydraulic Resistances

## Description



The Pipe Bend block represents a pipe bend as a local hydraulic resistance. The pressure loss in the bend is assumed to consist of

- Loss in the straight pipe
- Loss due to curvature

The loss in a straight pipe is simulated with the Resistive Tube block. The loss due to curvature is simulated with the Local Resistance block, and the pressure loss coefficient is determined in accordance with the Crane Co. recommendations (see [1], p. A-29).

The pressure loss due to curvature 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

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

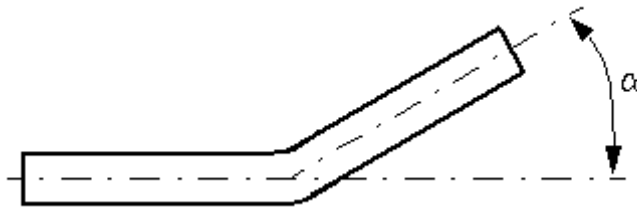
$$K = f(r, d, \alpha)$$

where

# Pipe Bend

---

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



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

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

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

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

## Warning

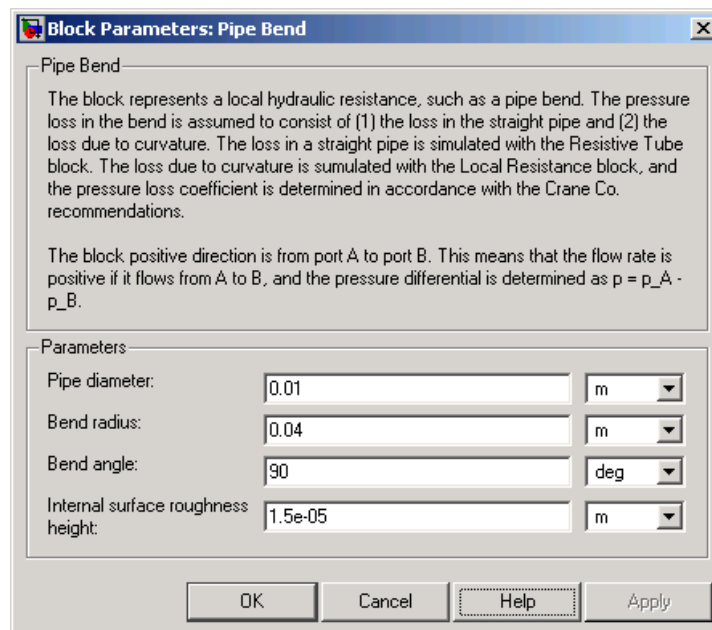
**The formulas used in the Pipe Bend block are very approximate, especially in the laminar and transient flow regions. For more accurate results, use the Local Resistance block with a table-specified  $K=f(Re)$  relationship.**

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- Fluid inertia, fluid compressibility, and wall compliance are not taken into account.
- The flow is assumed to be completely turbulent and fully developed along the pipe length.
- The bend is assumed to be made of a clean commercial steel pipe.

## Dialog Box and Parameters



### Pipe diameter

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

### Bend radius

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

# Pipe Bend

---

## **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.

## **Global Parameters**

### **Fluid density**

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

### **Fluid kinematic viscosity**

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

## **Ports**

The block has the following ports:

A

Hydraulic conserving port associated with the bend inlet.

B

Hydraulic conserving port associated with the bend outlet.

## **References**

[1] *Flow of Fluids Through Valves, Fittings, and Pipe*, Crane Valves North America, Technical Paper No. 410M

[2] George R. Keller, *Hydraulic System Analysis*, Published by the Editors of Hydraulics & Pneumatics Magazine, 1970

## **See Also**

Elbow

Gradual Area Change

Local Resistance

Resistive Tube



Sudden Area Change

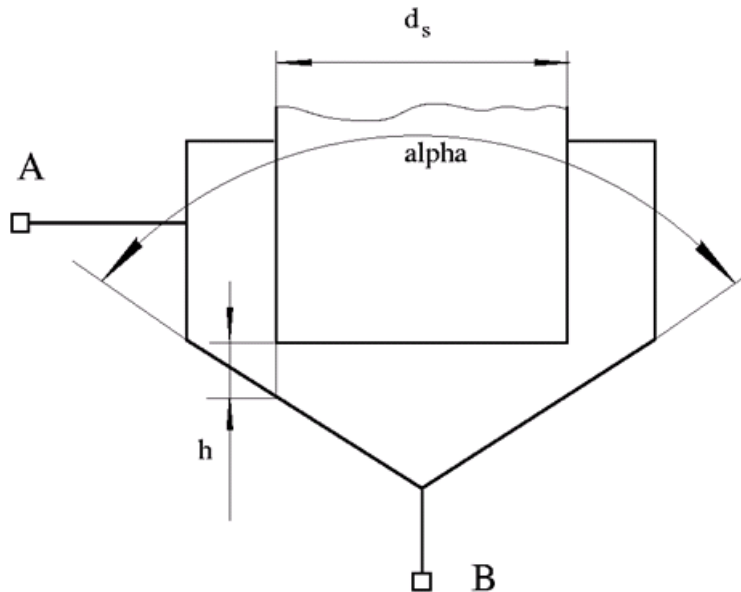
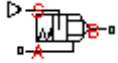
T-junction

# Poppet Valve

**Purpose** Simulate hydraulic poppet valve

**Library** Flow Control Valves

**Description** The Poppet Valve block models a variable orifice created by a cylindrical sharp-edged stem and a conical seat.



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

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

$$h = x_0 + x$$

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

$$p = p_A - p_B$$

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

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

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

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

where

$q$  Flow rate

$\rho$  Pressure differential

$p_A, p_B$  Gauge pressures at the block terminals

# Poppet Valve

---

$C_D$	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

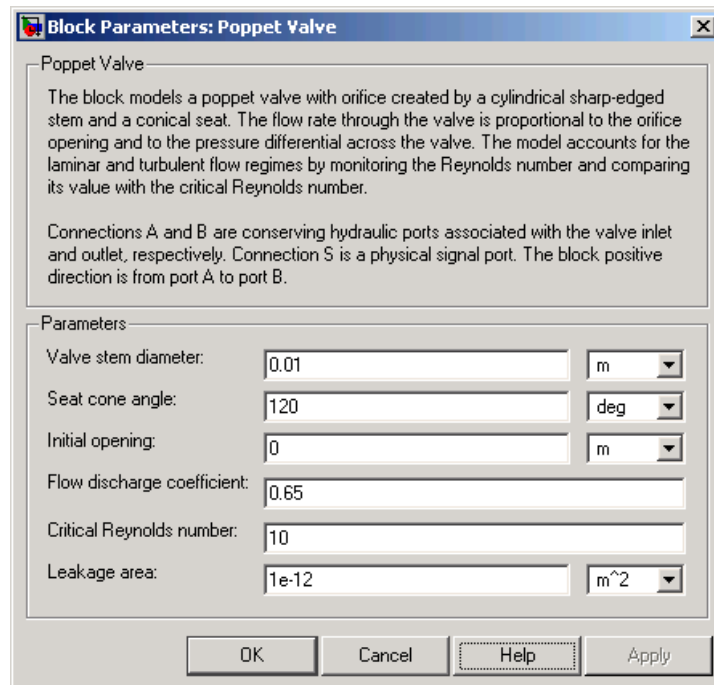
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as  $p = p_A - p_B$ . Positive signal at the physical signal port S opens the valve.

## Basic Assumptions and Limitations

The model is based on the following assumptions:

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

## Dialog Box and Parameters



### Variable 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 failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is  $1e-12 \text{ m}^2$ .

## **Global Parameters**

### **Fluid density**

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

### **Fluid kinematic viscosity**

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

## **Ports**

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

S

Physical signal port to control spool displacement.

## **See Also**

Ball Valve

Needle Valve

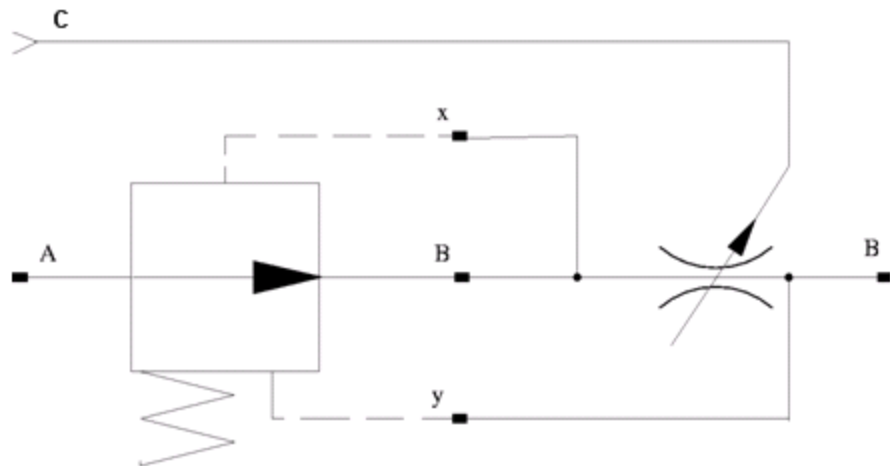
Pressure-Compensated Flow Control Valve

# Pressure-Compensated Flow Control Valve

**Purpose** Simulate hydraulic pressure compensating valve

**Library** Flow Control Valves

**Description** The Pressure-Compensated Flow Control Valve block represents a pressure-compensated flow control valve as a data-sheet based model. The valve is based on a Pressure Compensator block installed upstream from a Variable Orifice block, as shown in the following illustration.



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

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



# Pressure-Compensated Flow Control Valve

---

In the first case, the passage area is assumed to be linearly dependent on the control member displacement, that is, the orifice is assumed to be closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. In the second case, the passage area is determined by one-dimensional interpolation from the table  $A=A(h)$ . In both cases, a small leakage area is assumed to exist even after the orifice is completely closed. Physically, it represents a possible clearance in the closed valve, but the main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation.

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

## **Assumptions and Limitations**

The model is based on the following assumption:

- Fluid inertia is not taken into account.

# Pressure-Compensated Flow Control Valve

## Dialog Box and Parameters

**Block Parameters: Pressure-Compensated Flow Control Valve**

Pressure-Compensated Flow Control Valve

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

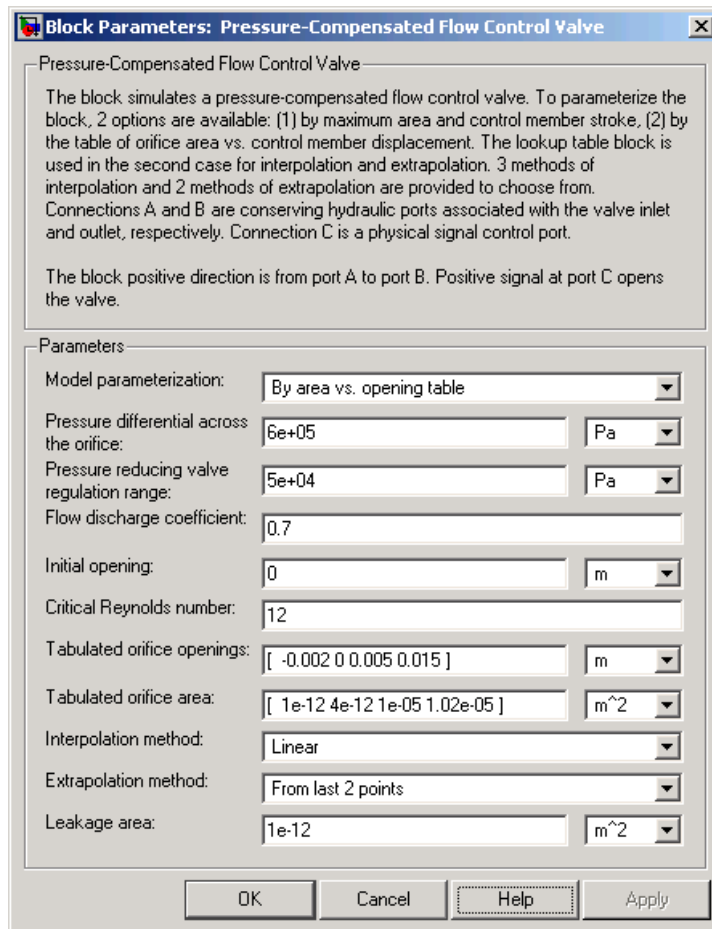
The block positive direction is from port A to port B. Positive signal at port C opens the valve.

Parameters

Model parameterization:	<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 tabulated 1-by- $m$  array. The input values vector must be strictly monotonically increasing. The values can be non-uniformly spaced. You must provide at least three values. The default values, in meters, are  $[-2e-3, 0, 5e-3, 15e-3]$ . This parameter is used if **Model parameterization** is set to **By area vs. opening table**. **Tabulated orifice openings** values will be used together with **Tabulated orifice area** values for one-dimensional table lookup.

## **Tabulated orifice area**

Specify the vector of output values for orifice area as a tabulated 1-by- $m$  array. The orifice area vector must be the same size as the orifice openings vector. All the values must be positive. The default values, in  $\text{m}^2$ , are  $[1e-12, 4e-12, 1.e-5, 1.02e-5]$ . This

# Pressure-Compensated Flow Control Valve

---

parameter is used if **Model parameterization** is set to By area vs. opening table.

## Interpolation method

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

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

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

## Extrapolation method

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

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

# Pressure-Compensated Flow Control Valve

---

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

## **Pressure differential across the orifice**

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

## **Pressure reducing valve regulation range**

Pressure increase over the preset level needed to fully close the valve. The default value is  $5e4$  Pa.

## **Flow discharge coefficient**

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

## **Initial opening**

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

## **Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

## **Leakage area**

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is  $1e-12$  m<sup>2</sup>.

# Pressure-Compensated Flow Control Valve

---

## Global Parameters

### Fluid density

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

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

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

Ball Valve

Needle Valve

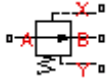
Poppet Valve

# Pressure Compensator

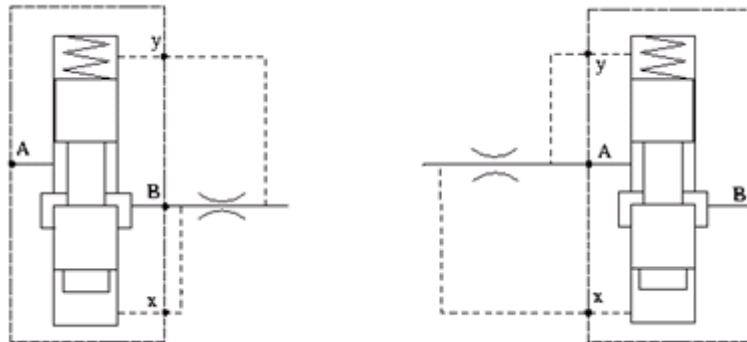
**Purpose** Simulate hydraulic pressure compensating valve

**Library** Hydraulic Elements

## Description



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

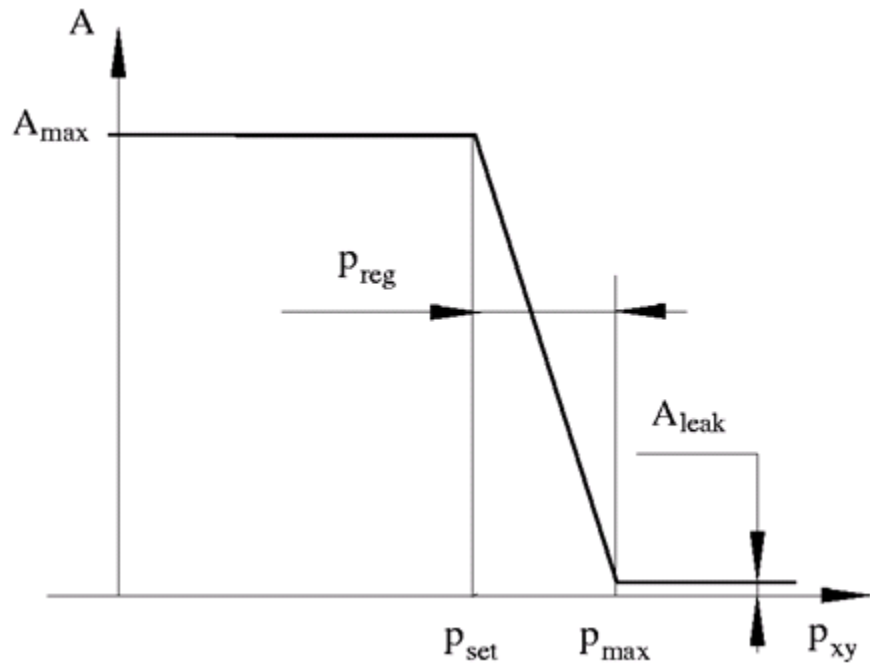


The block is implemented as a data-sheet-based model, based on parameters usually provided in the manufacturer's catalogs or data sheets.

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



# Pressure Compensator



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

The main parameters of the block are the valve maximum area and regulation range. In addition, you need to specify the leakage area of the valve. Physically, it represents a possible clearance in the closed valve, but the main purpose of the parameter is to maintain numerical

# Pressure Compensator

---

integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation.

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

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

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

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

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

$$p = p_A - p_B$$

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

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

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

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

where

$q$	Flow rate
$p$	Pressure differential across the valve
$p_{xy}$	Pressure differential across valve control terminals
$p_A, p_B$	Gauge pressures at the valve main terminals
$p_x, p_y$	Gauge pressures at the valve control terminals
$p_{set}$	Valve preset pressure
$p_{max}$	Pressure needed to fully close the orifice
$p_{reg}$	Regulation range
$A(h)$	Instantaneous orifice passage area
$A_{max}$	Orifice maximum area
$C_D$	Flow discharge coefficient
$\rho$	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

# Pressure Compensator

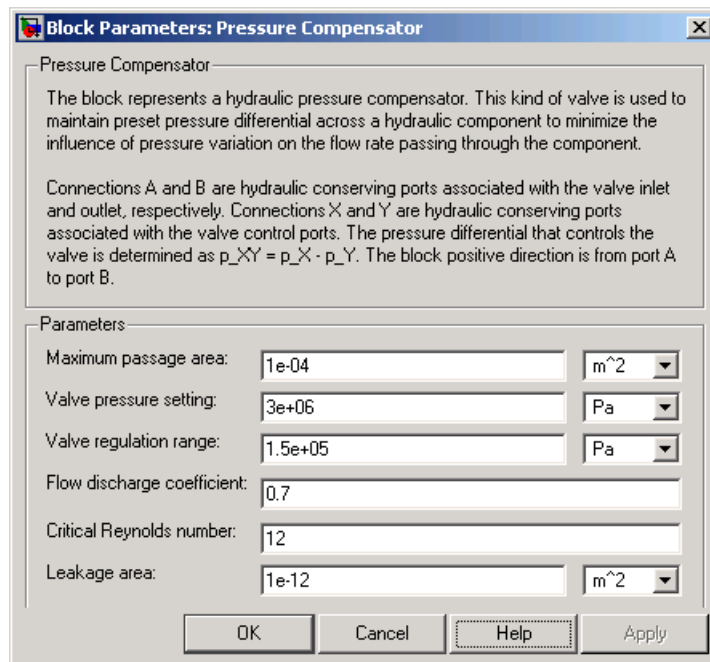
measured as  $P_{xy} = P_x - P_y$ , and it creates a force acting against the spring preload.

## Assumptions and Limitations

The model is based on the following assumptions:

- Valve opening is linearly proportional to the pressure differential.
- No loading on the valve, such as inertia, friction, spring, and so on, is considered.
- Flow consumption associated with the spool motion is neglected.

## Dialog Box and Parameters



### Maximum passage area

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

**Valve pressure setting**

Pressure difference that must be maintained across an element connected to ports X and Y. At this pressure the valve orifice starts to close. The default value is  $3e6$  Pa.

**Valve regulation range**

Length of the fluid passage. The default value is  $1.5e5$  Pa.

**Flow discharge coefficient**

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

**Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

**Leakage area**

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is  $1e-12$  m<sup>2</sup>.

**Global Parameters****Fluid density**

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

# Pressure Compensator

---

## **Fluid kinematic viscosity**

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

## **Ports**

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

X

Hydraulic conserving port associated with the pressure control terminal that opens the orifice.

Y

Hydraulic conserving port associated with the pressure control terminal that closes the orifice.

## **See Also**

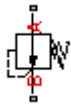
Pressure Relief Valve

# Pressure Reducing Valve

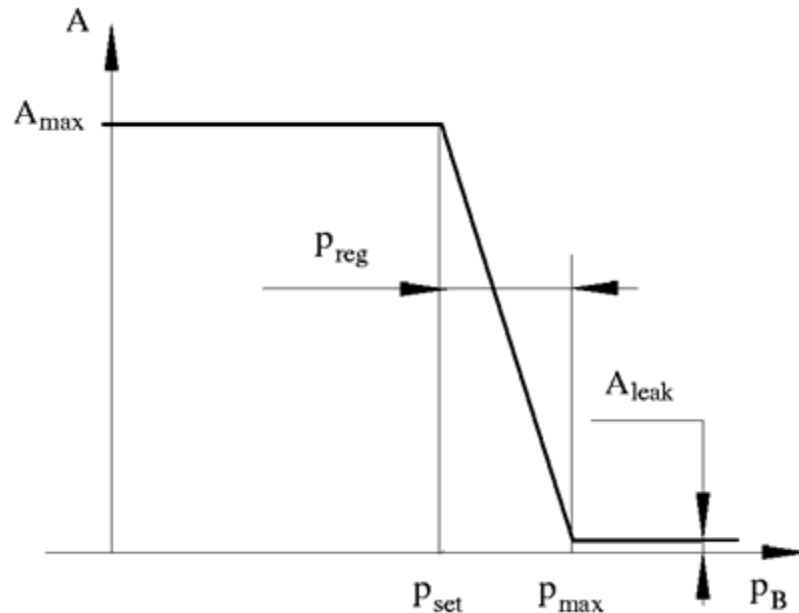
**Purpose** Simulate pressure control valve maintaining reduced pressure in portion of system

**Library** Pressure Control Valves

## Description



The Pressure Reducing Valve block represents a hydraulic pressure-reducing valve as a data-sheet-based model. Pressure-reducing valves are used to maintain reduced pressure in a portion of a system. The following figure shows the typical dependency between the valve passage area  $A$  and the pressure  $p_B$  downstream from the valve.



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

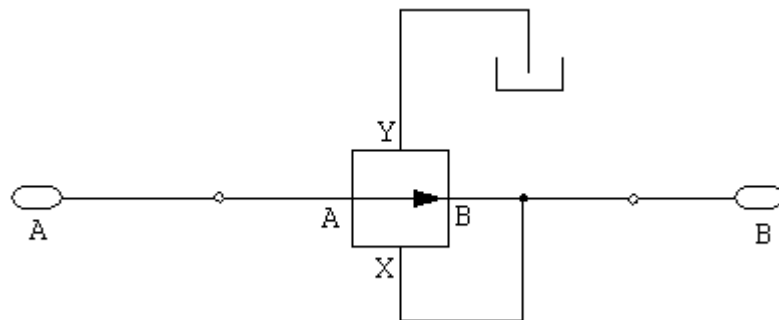
# Pressure Reducing Valve

---

trying to maintain outlet pressure at preset level. Any further increase in the outlet pressure causes the control member to close the orifice even more until the point when the orifice is fully closed. The pressure increase that is necessary to close the valve is referred to as regulation range, and is generally provided in the catalogs, along with the valve maximum area. The valve maximum area and regulation range are the key parameters of the block.

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

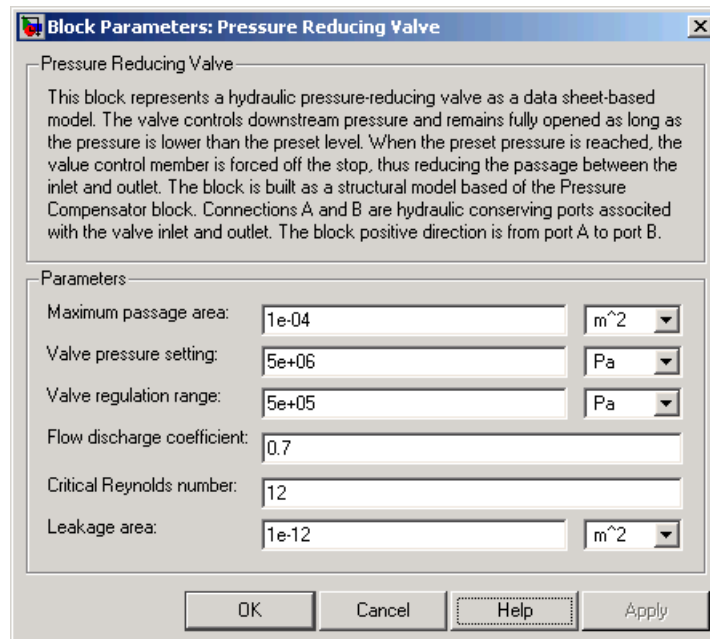
The block is built as a structural model based on the Pressure Compensator block, as shown in the following schematic.



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



## Dialog Box and Parameters



### Maximum passage area

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

### Valve pressure setting

Preset pressure level, at which the orifice of the valve starts to close. The default value is  $5e6 \text{ Pa}$ .

### Valve regulation range

Pressure increase over the preset level needed to fully close the valve. 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.

# Pressure Reducing 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.

## Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is  $1e-12m^2$ .

## Global Parameters

### Fluid density

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

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

## Examples

The Power Unit with Pressure Reducing Valve demo (sh\_power\_unit\_pressure\_red\_valve) illustrates the use of the

Pressure Reducing Valve block in hydraulic systems. The pressure reducing valve is set to  $20e5$  Pa and maintains this pressure downstream, as long as the upstream pressure is higher than this setting.

## **See Also**

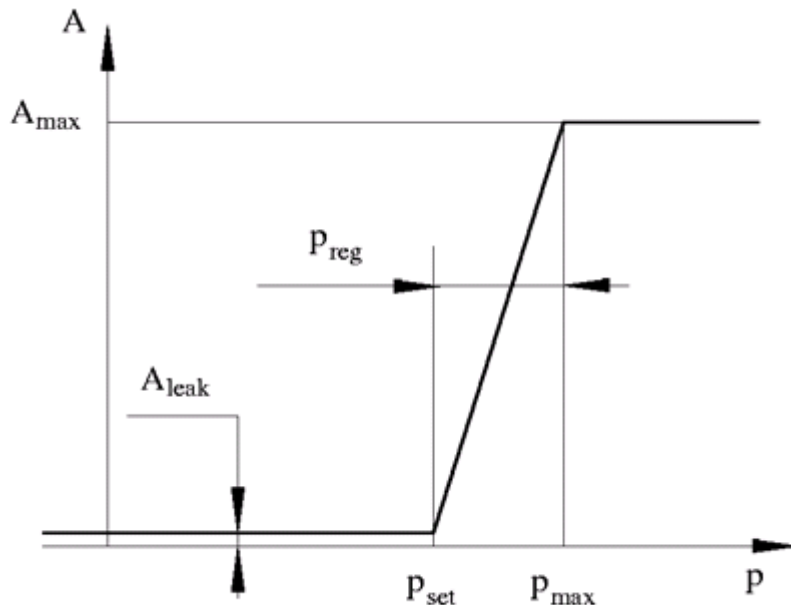
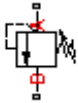
Pressure Compensator

# Pressure Relief Valve

**Purpose** Simulate pressure control valve maintaining preset pressure in system

**Library** Pressure Control Valves

**Description** The Pressure Relief Valve block represents a hydraulic pressure relief valve as a data-sheet-based model. The following figure shows the typical dependency between the valve passage area  $A$  and the pressure differential  $p$  across the valve.



The valve remains closed while pressure at the valve inlet is lower than the valve preset pressure. When the preset pressure is reached, the valve control member (spool, ball, poppet, etc.) is forced off its seat, thus creating a passage between the inlet and outlet. Some fluid is diverted to a tank through this orifice, thus reducing the pressure at the inlet. If

this flow rate is not enough and pressure continues to rise, the area is further increased until the control member reaches its maximum. At this moment, the maximum flow rate is passing through the valve. The value of a maximum flow rate and the pressure increase over the preset level to pass this flow rate are generally provided in the catalogs. The pressure increase over the preset level is frequently referred to as valve steady state error, or regulation range. The valve maximum area and regulation range are the key parameters of the block.

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

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

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

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

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

# Pressure Relief Valve

---

$$p = p_A - p_B$$

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

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

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

where

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

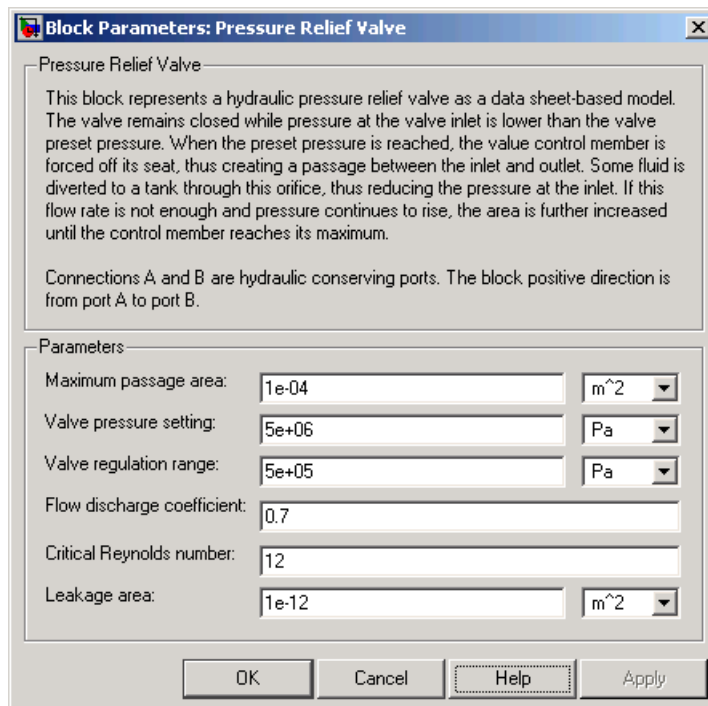
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as  $p = p_A - p_B$ .

## Basic Assumptions and Limitations

The model is based on the following assumptions:

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

## Dialog Box and Parameters



**Block Parameters: Pressure Relief Valve**

Pressure Relief Valve

This block represents a hydraulic pressure relief valve as a data sheet-based model. The valve remains closed while pressure at the valve inlet is lower than the valve preset pressure. When the preset pressure is reached, the valve control member is forced off its seat, thus creating a passage between the inlet and outlet. Some fluid is diverted to a tank through this orifice, thus reducing the pressure at the inlet. If this flow rate is not enough and pressure continues to rise, the area is further increased until the control member reaches its maximum.

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

Parameters

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

OK Cancel Help Apply

# Pressure Relief Valve

---

## **Maximum passage area**

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

## **Valve pressure setting**

Preset pressure level, at which the orifice of the valve starts to open. The default value is  $50e5 \text{ Pa}$ .

## **Valve regulation range**

Pressure increase over the preset level needed to fully open the valve. The default value is  $5e5 \text{ Pa}$ .

## **Flow discharge coefficient**

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

## **Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

## **Leakage area**

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is  $1e-12 \text{ m}^2$ .



## Global Parameters

### Fluid density

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

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

## Examples

The Power Unit with Fixed-Displacement Pump demo (sh\_power\_unit\_fxd\_dspl\_pump) illustrates the use of the Pressure Relief Valve block in hydraulic systems. The valve is set to  $75 \times 10^5$  Pa and starts diverting fluid to tank as soon as the pressure at its inlet reaches this value.

# Proportional and Servo-Valve Actuator

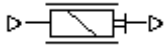
## Purpose

Simulate continuous valve driver with output proportional to input signal

## Library

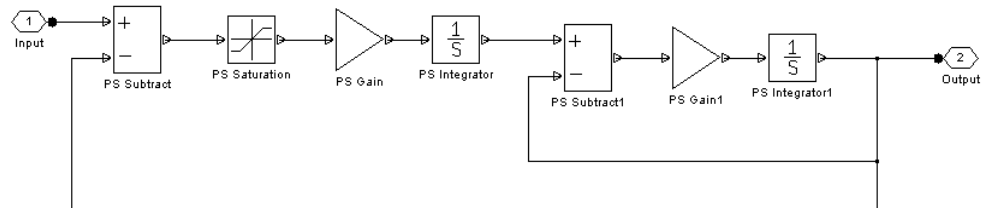
Valve Actuators

## Description



The Proportional and Servo-Valve Actuator block represents an electromagnetic actuator that is used in proportional and servo-valves to drive a spool or other working member. The block is intended to work with one of the directional valve models to form a desirable configuration of a proportional or servo-valve. The block is implemented as a data-sheet-based model and reproduces only the input/output relationship, or the actuator's transient response, as presented in the catalog or data sheet.

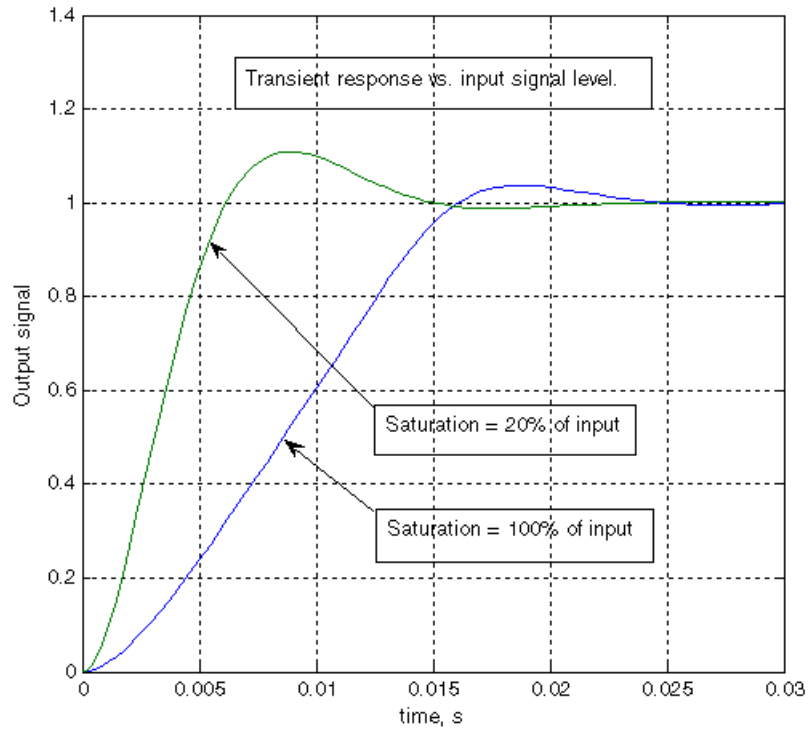
The Proportional and Servo-Valve Actuator block is built using the blocks from the Physical Signals library. Both the input and the output of the block are physical signals. The block diagram of the model is shown in the following figure.



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

The typical transient responses of a servo-valve or a high-quality proportional valve are shown in the following figure. The only difference between the two responses in the figure is the value of the saturation. The response that corresponds to 100% of the input signal is considerably slower than that with the 20% saturation.

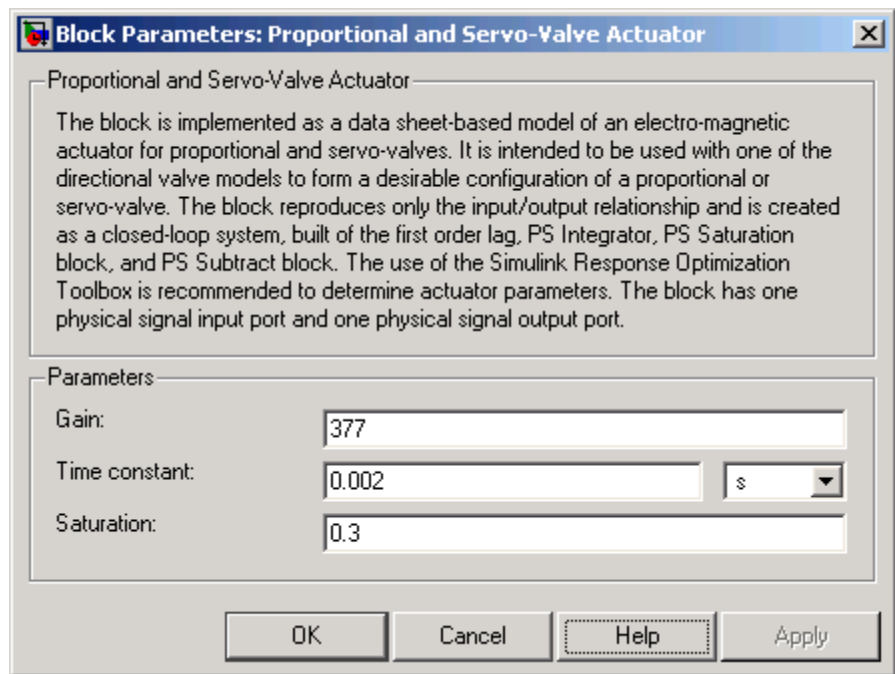
# Proportional and Servo-Valve Actuator



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

# Proportional and Servo-Valve Actuator

## Dialog Box and Parameters



### Gain

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

### Time constant

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

### Saturation

Saturation level of the Saturation block in the actuator model. The default value is 0.3.

## Ports

The block has one physical signal input port and one physical signal output port.

# Proportional and Servo-Valve Actuator

---

## **Examples**

The Closed-Loop Electrohydraulic Actuator with Proportional Valve demo (sh\_closed\_loop\_actuator) illustrates the use of the Proportional and Servo-Valve Actuator block in hydraulic systems.

## **See Also**

2-Position Valve Actuator

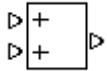
3-Position Valve Actuator

# PS Add

**Purpose** Add two physical signal inputs

**Library** Physical Signals/Functions

**Description** The PS Add block outputs the sum of two input physical signals:



$$y = u_1 + u_2$$

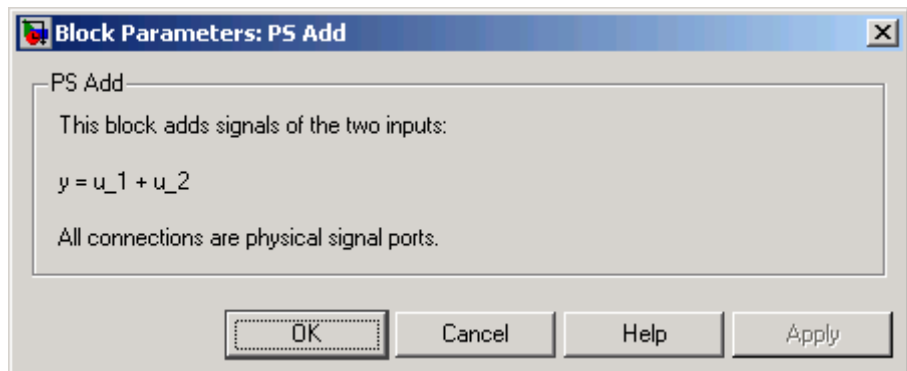
where

$u_1$  Physical signal at the first input port

$u_2$  Physical signal at the second input port

$y$  Physical signal at the output port

## Dialog Box and Parameters



The PS Add block has no parameters.

**Ports** The block has two physical signal input ports and one physical signal output port.

**See Also** PS Divide  
PS Gain

PS Product

PS Subtract

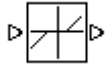
# PS Dead Zone

---

**Purpose** Provide region of zero output for physical signals

**Library** Physical Signals/Nonlinear Operators

**Description**



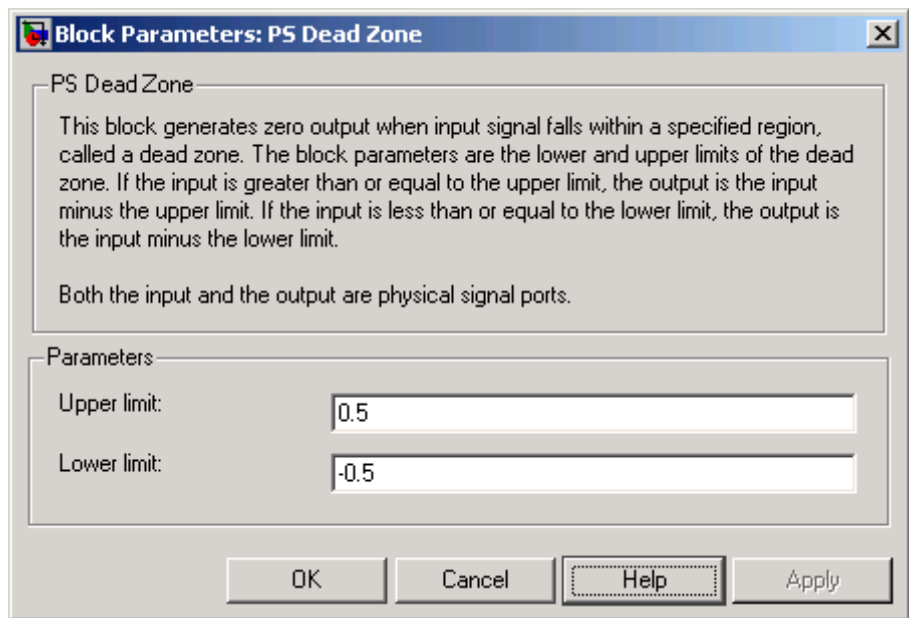
The PS Dead Zone block generates zero output when input signal falls within a specified region, called a dead zone. You can specify the lower and upper limits of the dead zone as block parameters. The block output depends on the input and dead zone:

- If the input is within the dead zone (greater than the lower limit and less than the upper limit), the output is zero.
- If the input is greater than or equal to the upper limit, the output is the input minus the upper limit.
- If the input is less than or equal to the lower limit, the output is the input minus the lower limit.

Both the input and the output are physical signals.



## Dialog Box and Parameters



### Upper limit

The upper limit, or end, of the dead zone. The default value is 0.5.

### Lower limit

The lower limit, or start, of the dead zone. The default value is -0.5.

## Ports

The block has one physical signal input port and one physical signal output port.

## See Also

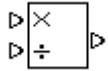
PS Saturation

# PS Divide

**Purpose** Compute simple division of two input physical signals

**Library** Physical Signals/Functions

**Description** The PS Divide block divides one physical signal input by another and outputs the difference:



$$y = u_1 \div u_2$$

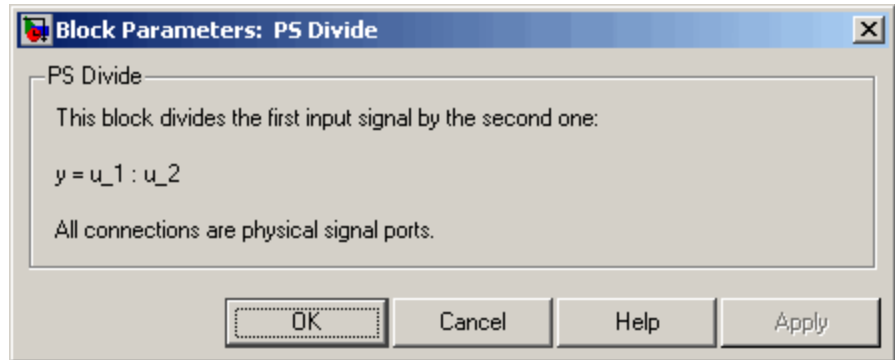
where

$u_1$  Physical signal at the first input port (marked with the x sign)

$u_2$  Physical signal at the second input port (marked with the ÷ sign)

$y$  Physical signal at the output port

## Dialog Box and Parameters



The PS Divide block has no parameters.

**Ports** The block has two physical signal input ports and one physical signal output port.

## **See Also**

PS Add

PS Gain

PS Product

PS Subtract

# PS Gain

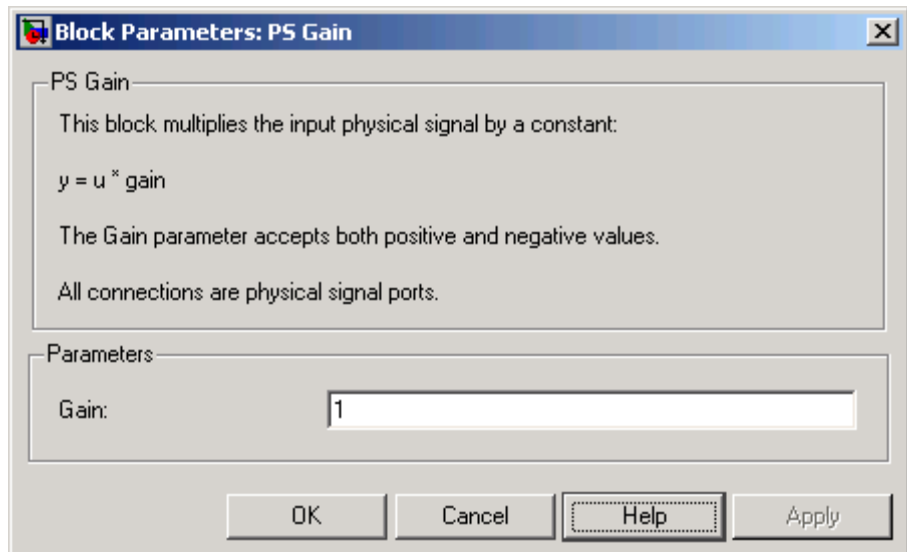
**Purpose** Multiply input physical signal by constant

**Library** Physical Signals/Functions

**Description** The PS Gain block multiplies the input physical signal by a constant value (gain). You specify the value of the gain as the **Gain** parameter.



## Dialog Box and Parameters



**Gain** The multiplication coefficient. You can specify both positive and negative values.

**Ports** The block has one physical signal input port and one physical signal output port.

## See Also

PS Add

PS Divide

PS Product

PS Subtract

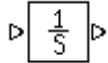
# PS Integrator

---

**Purpose** Integrate physical signal

**Library** Physical Signals/Linear Operators

**Description** The PS Integrator block outputs the integral of its input at the current time step. The following equation represents the output of the block:



$$y(t) = \int_{t_0}^t u(t)dt + y_0$$

where

- $u$  Physical signal at the input port
- $y_0$  Initial condition
- $y$  Physical signal at the output port
- $t$  Time

The PS Integrator block is a dynamic system with one state, its output. The PS Integrator block's input is the state's time derivative:

$$x = y(t)$$

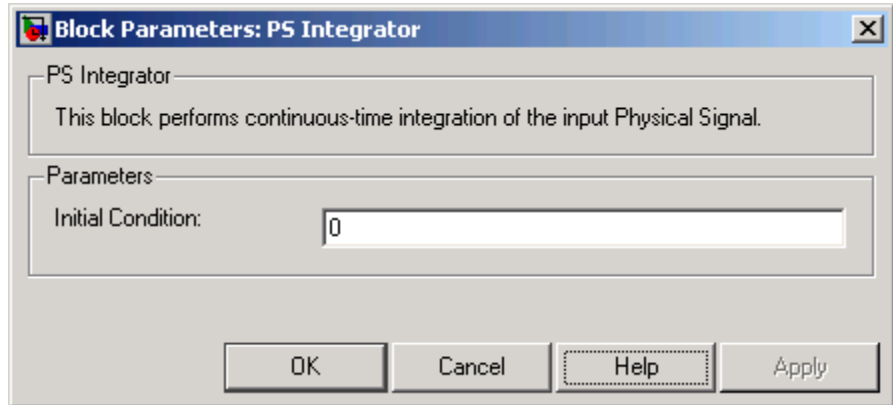
$$x_0 = y_0$$

$$\dot{x} = u(t)$$

The solver computes the output of the PS Integrator block at the current time step, using the current input value and the value of the state at the previous time step. To support this computational model, the PS Integrator block saves its output at the current time step for use by the solver to compute its output at the next time step. The block also provides the solver with an initial condition for use in computing the block's initial state at the beginning of a simulation run. The default

value of the initial condition is 0. You can specify another value for the initial condition as a parameter on the block dialog box.

## Dialog Box and Parameters



### Initial Condition

Specify the initial condition for use in computing the block's initial state at the beginning of a simulation run. For more information, see "Computing Initial Conditions". The default value is 0.

## Ports

The block has one physical signal input port and one physical signal output port.

# PS Lookup Table (1D)

---

**Purpose** Approximate one-dimensional function using specified lookup method

**Library** Physical Signals/Lookup Tables

**Description** The PS Lookup Table (1D) block computes an approximation to some function  $y=f(x)$  given data vectors  $x$  and  $y$ . Both the input and the output are physical signals.



---

**Note** To map two physical signal inputs to an output, use the PS Lookup Table (2-D) block.

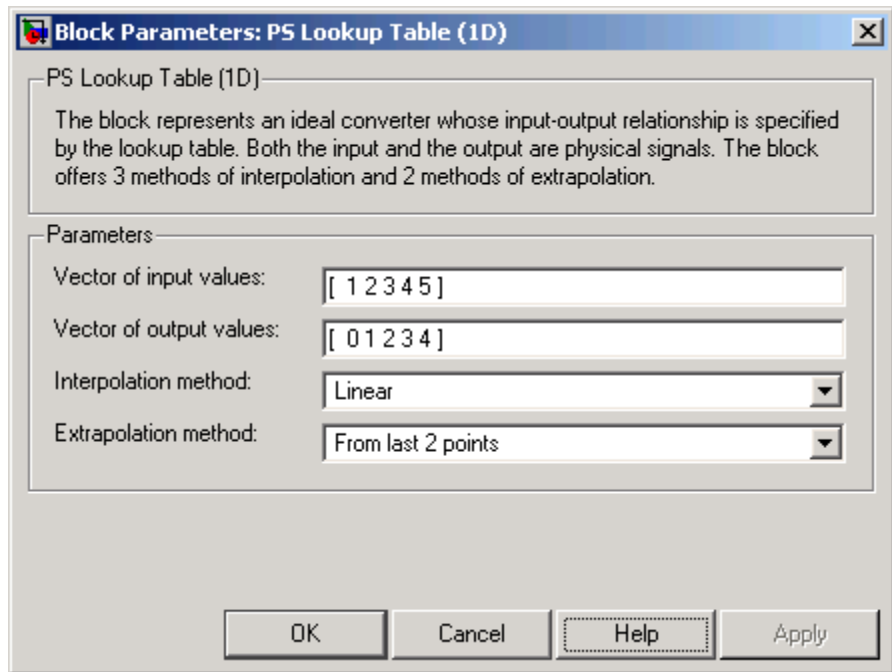
---

The length of the  $x$  and  $y$  data vectors provided to this block must match. Also, the  $x$  data vector must be *strictly monotonically increasing* (i.e., the value of the next element in the vector is greater than the value of the preceding element).

You define the lookup table by specifying the **Vector of input values** parameter as a 1-by- $n$  vector and the **Vector of output values** parameter as a 1-by- $n$  vector. The block generates output based on the input values using the selected interpolation and extrapolation methods. You have a choice of three interpolation methods and two extrapolation methods.



## Dialog Box and Parameters



### Vector of input values

Specify the vector of input values as a tabulated 1-by-n array. The input values vector must be strictly monotonically increasing. The values can be non-uniformly spaced.

### Vector of output values

Specify the vector of output values as a tabulated 1-by-n array. The output values vector must be the same size as the input values vector.

### 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 function.

# PS Lookup Table (1D)

---

- **Cubic**—Uses the Piecewise Cubic Hermite Interpolation Polynomial (PCHIP). For more information, see [1] and the `pchip` MATLAB function.
- **Spline**—Uses the cubic spline interpolation algorithm described in [2].

## **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.

## **Ports**

The block has one physical signal input port and one physical signal output port.

## **References**

- [1] D.Kahaner, Cleve Moler, Stephen Nash, *Numerical Methods and Software*, Prentice Hall, 1988
- [2] W.H.Press, B.P.Flannery, S.A.Teulkolsky, W.T.Wetterling, *Numerical Recipes in C: The Art of Scientific Computing*, Cambridge University Press, 1992

## **See Also**

PS Lookup Table (2D)

**Purpose** Approximate two-dimensional function using specified lookup method

**Library** Physical Signals/Lookup Tables

## Description



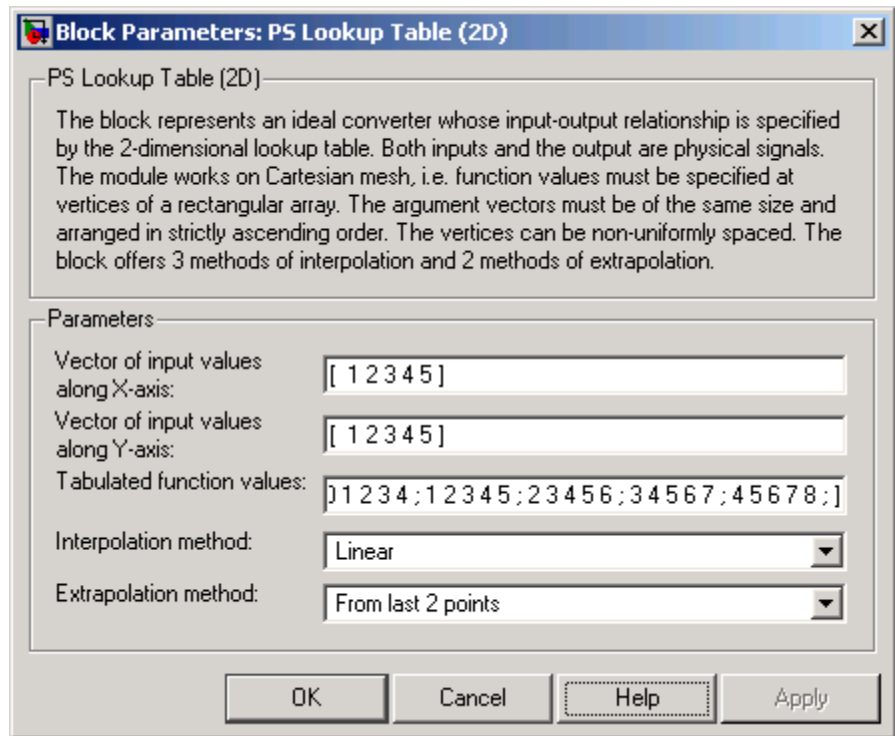
The PS Lookup Table (2D) block computes an approximation to some function  $z=f(x,y)$  given the  $x$ ,  $y$ ,  $z$  data points. The two inputs and the output are physical signals.

The  $x$  and  $y$  data vectors must be *strictly monotonically increasing* (i.e., the value of the next element in the vector is greater than the value of the preceding element). The matrix size of the tabulated function values must match the dimensions defined by the input vectors.

You define the lookup table by specifying the **Vector of input values along X-axis** parameter as a 1-by- $m$  vector of  $x$  data points, the **Vector of input values along Y-axis** parameter as a 1-by- $n$  vector of  $y$  data points, and the **Tabulated function values** as an  $m$ -by- $n$  matrix of  $z$  data points. The block works on Cartesian mesh, i.e., function values must be specified at vertices of a rectangular array. The block generates output based on the input grid lookup using the selected interpolation and extrapolation methods. You have a choice of three interpolation methods and two extrapolation methods.

# PS Lookup Table (2D)

## Dialog Box and Parameters



### Vector of input values along X-axis

Specify the vector of input values along the  $x$ -axis as a tabulated 1-by- $m$  array. The input values vector must be strictly monotonically increasing. The values can be non-uniformly spaced.

### Vector of input values along Y-axis

Specify the vector of input values along the  $y$ -axis as a tabulated 1-by- $n$  array. The input values vector must be strictly monotonically increasing. The values can be non-uniformly spaced.

## Tabulated function values

Specify the output values as a tabulated  $m$ -by- $n$  matrix, defining the function values at the input grid vertices. The matrix size must match the dimensions defined by the input vectors.

## 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. The method performs linear interpolation first in  $x$ -direction and then in  $y$ -direction.
- **Cubic**—Uses the bicubic interpolation algorithm described in [1].
- **Spline**—Uses the bicubic spline interpolation algorithm described in [1].

## 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 grid location, similar to PS Lookup Table (1D) block.
- **From last point**—Uses the last specified output value at the appropriate grid location, similar to PS Lookup Table (1D) block..

## Ports

The block has two physical signal input ports and one physical signal output port.

## PS Lookup Table (2D)

---

### References

[1] W.H.Press, B.P.Flannery, S.A.Teulkolsky, W.T.Wetterling, *Numerical Recipes in C: The Art of Scientific Computing*, Cambridge University Press, 1992

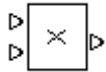
### See Also

PS Lookup Table (1D)

**Purpose** Multiply two physical signal inputs

**Library** Physical Signals/Functions

**Description** The PS Product block outputs the product of two input physical signals:



$$y = u_1 \cdot u_2$$

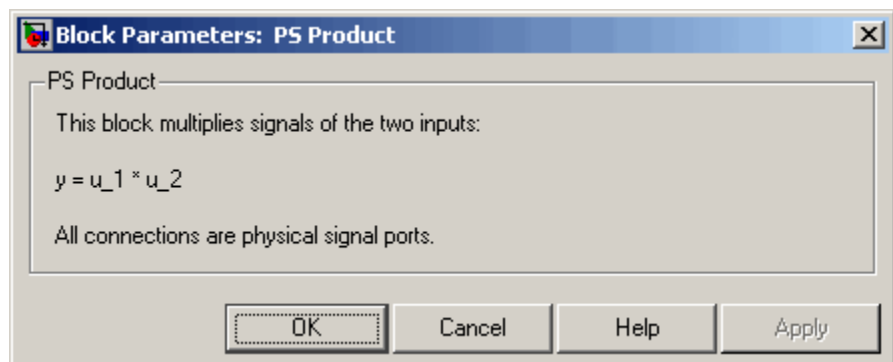
where

$u_1$  Physical signal at the first input port

$u_2$  Physical signal at the second input port

$y$  Physical signal at the output port

## Dialog Box and Parameters



The PS Product block has no parameters.

**Ports** The block has two physical signal input ports and one physical signal output port.

**See Also** PS Add  
PS Divide

# PS Product

---

PS Gain

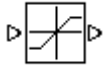
PS Subtract



**Purpose** Limit range of physical signal

**Library** Physical Signals/Nonlinear Operators

## Description

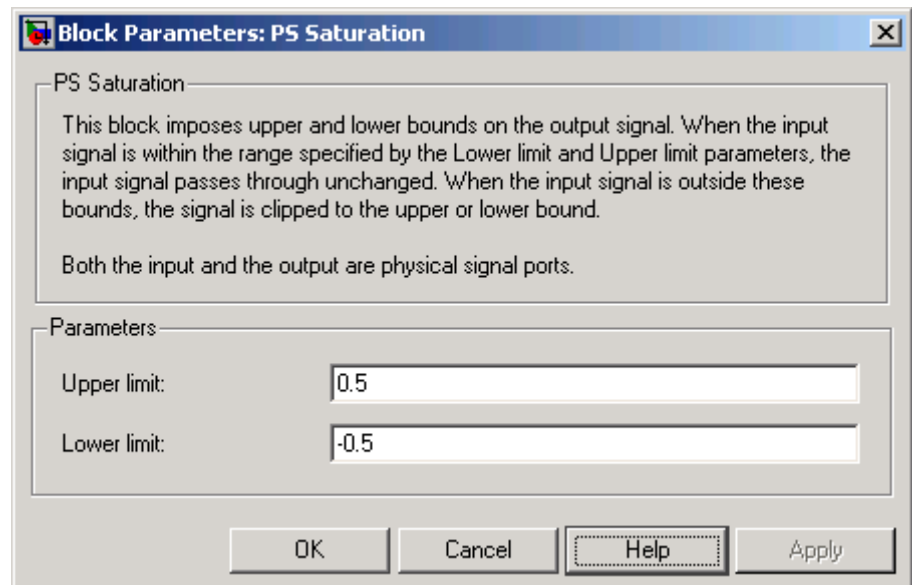


The PS Saturation block imposes upper and lower bounds on a physical signal. When the input signal is within the range specified by the **Lower limit** and **Upper limit** parameters, the input signal passes through unchanged. When the input signal is outside these bounds, the signal is clipped to the upper or lower bound.

When the **Lower limit** and **Upper limit** parameters are set to the same value, the block outputs that value.

Both the input and the output are physical signals.

## Dialog Box and Parameters



# PS Saturation

---

## **Upper limit**

The upper bound on the input signal. When the input signal to the Saturation block is above this value, the output of the block is clipped to this value. The default is 0.5.

## **Lower limit**

The lower bound on the input signal. When the input signal to the Saturation block is below this value, the output of the block is clipped to this value. The default is -0.5.

## **Ports**

The block has one physical signal input port and one physical signal output port.

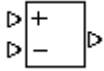
## **See Also**

PS Dead Zone

**Purpose** Compute simple subtraction of two input physical signals

**Library** Physical Signals/Functions

**Description** The PS Subtract block subtracts one physical signal input from another and outputs the difference:



$$y = u_1 - u_2$$

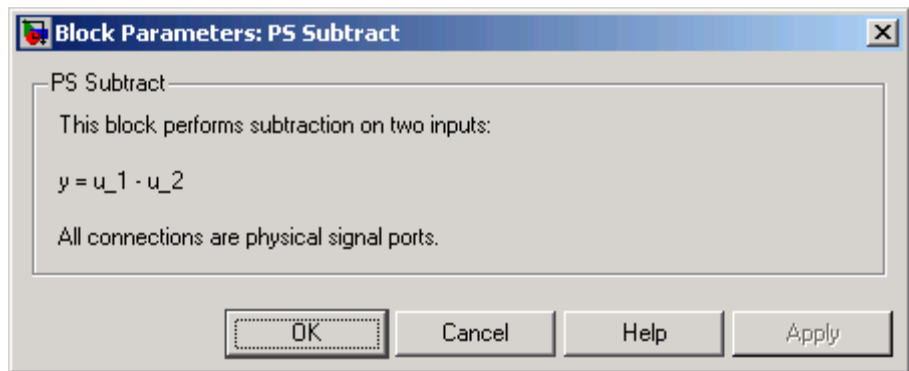
where

$u_1$  Physical signal at the first input port (marked with the plus sign)

$u_2$  Physical signal at the second input port (marked with the minus sign)

$y$  Physical signal at the output port

## Dialog Box and Parameters



The PS Subtract block has no parameters.

**Ports** The block has two physical signal input ports and one physical signal output port.

# PS Subtract

---

## **See Also**

PS Add

PS Divide

PS Gain

PS Product

**Purpose** Convert physical signal into Simulink output signal

**Library** Utilities

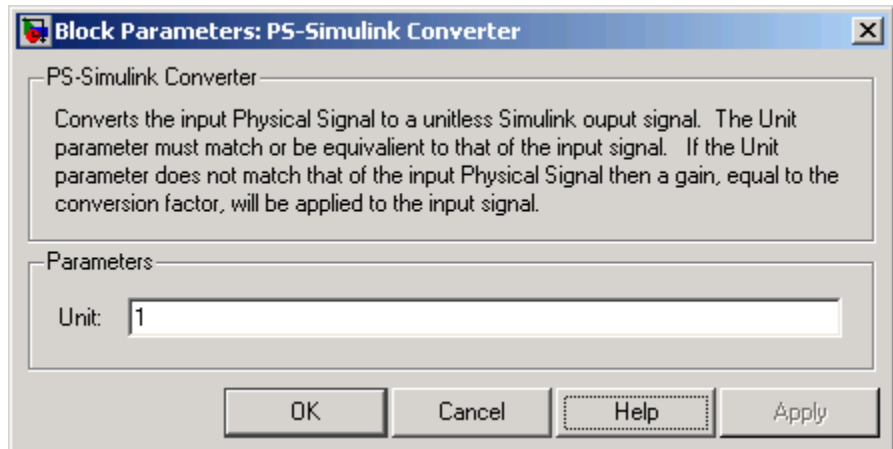
## Description



The PS-Simulink Converter block converts a physical signal, in specified units, into a unitless Simulink signal. You specify the desired units as the **Unit** parameter.

Use this block to connect outputs of a Physical Network diagram to Simulink scopes or other Simulink blocks.

## Dialog Box and Parameters



### Unit

Units for the physical signal. You can type the desired unit name, such as rpm, or a valid expression, such as rad/s. For more information and a list of unit abbreviations, see “Working with Physical Units”. The system compares the units you specifies with the actual units of the input physical signal and applies a gain equal to the conversion factor before outputting the Simulink signal. The default value is 1, which means that the unit is not specified. If you do not specify a unit, or if the unit matches the actual units of the input physical signal, no gain is applied.

# PS-Simulink Converter

---

**Ports**

The block has a physical signal input port, located on its left-hand side, and a Simulink output port, located on its right-hand side (in the block default orientation).

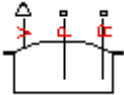
**See Also**

Simulink-PS Converter

**Purpose** Simulate pressurized hydraulic reservoir

**Library** Hydraulic Utilities

## Description



The Reservoir block represents a pressurized hydraulic reservoir, in which fluid is stored under a specified pressure. The pressure remains constant regardless of volume change. The block accounts for pressure loss in the return line that can be caused by a filter, fittings, or some other local resistance. The loss is specified with the pressure loss coefficient. The block computes the volume of fluid in the tank and exports it outside through the physical signal port V.

## Dialog Box and Parameters

**Block Parameters: Reservoir**

Reservoir

This block represents a pressurized hydraulic reservoir, in which fluid is stored under a specified pressure. The pressure remains constant regardless of volume change. The block accounts for pressure loss in the return line that can be caused by a filter, fittings, or some other local resistance. The loss is specified with the pressure loss coefficient. The block computes volume of fluid in a tank and exports it outside through the physical signal port V.

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

Parameters

Pressurization level:	<input type="text" value="0"/>	Pa
Initial fluid volume:	<input type="text" value="0.02"/>	m <sup>3</sup>
Return line diameter:	<input type="text" value="0.02"/>	m
Pressure loss coefficient in return line:	<input type="text" value="1"/>	

OK Cancel Help Apply

### Pressurization level

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

# Reservoir

---

## **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. The default value is 1.

## **Ports**

The block has the following ports:

P

Hydraulic conserving port associated with the pump line.

R

Hydraulic conserving port associated with the return line.

V

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

## **See Also**

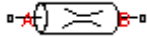
Hydraulic Reference



**Purpose** Simulate hydraulic pipeline which accounts for friction losses only

**Library** Hydraulic Elements

## Description



The Resistive Tube block models hydraulic pipelines with circular and non-circular 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, all the resistances are converted into their equivalent lengths, and then the total length of all the resistances is added 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:

$$p = f \frac{(L + L_{eq})}{D_H} \frac{\rho}{2A^2} q \cdot |q|$$

# Resistive Tube

---

$$f = \begin{cases} K_s / Re & \text{for } Re \leq Re_L \\ f_L + \frac{f_T - f_L}{Re_T - Re_L} (Re - Re_L) & \text{for } Re_L < Re < Re_T \\ \frac{1}{\left( -1.8 \log_{10} \left( \frac{6.9}{Re} + \left( \frac{r/D_H}{3.7} \right)^{1.11} \right) \right)^2} & \text{for } Re \geq Re_T \end{cases}$$

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

where

$\rho$  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

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

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- Flow is assumed to be fully developed along the pipe length.
- Fluid inertia, fluid compressibility, and wall compliance are not taken into account.

# Resistive Tube

## Dialog Box and Parameters

**Block Parameters: Resistive Tube**

Resistive Tube

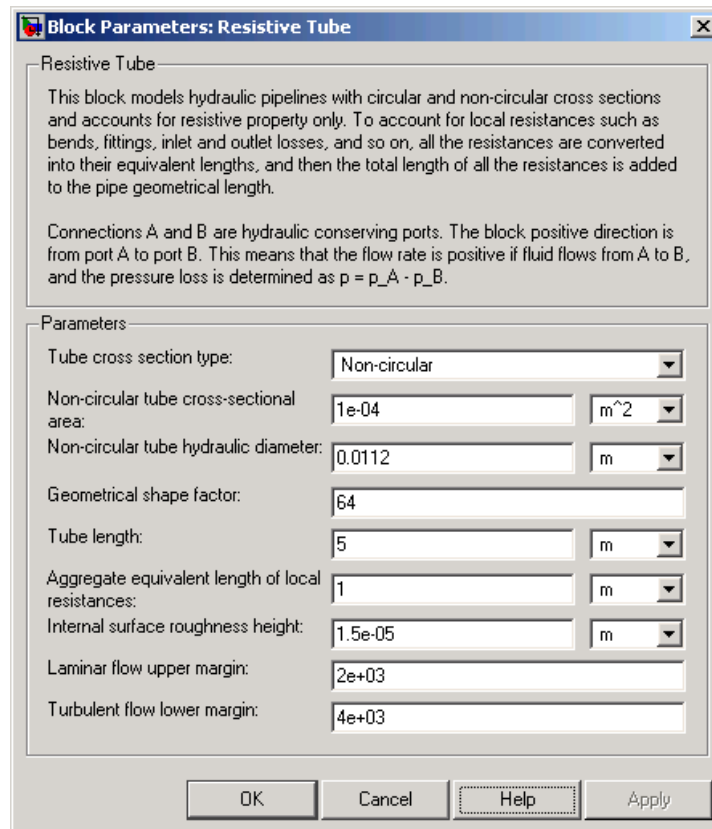
This block models hydraulic pipelines with circular and non-circular cross sections and accounts for resistive property only. To account for local resistances such as bends, fittings, inlet and outlet losses, and so on, all the resistances are converted into their equivalent lengths, and then the total 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

Tube cross section type:	<input type="text" value="Circular"/>
Tube internal diameter:	<input type="text" value="0.01"/> <input type="text" value="m"/>
Geometrical shape factor:	<input type="text" value="64"/>
Tube 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-05"/> <input type="text" value="m"/>
Laminar flow upper margin:	<input type="text" value="2e+03"/>
Turbulent flow lower margin:	<input type="text" value="4e+03"/>

OK Cancel Help Apply



## Tube cross section type

The parameter can have one of two values: **Circular** or **Non-circular**. For a circular tube, you need to specify its internal diameter. For a non-circular tube, you need to specify its hydraulic diameter and tube cross-sectional area. The default value of the parameter is **Circular**.

## Tube internal diameter

Tube internal diameter. The parameter is used if **Tube cross section type** is set to **Circular**. The default value is 0.01 m.

# Resistive Tube

---

## **Non-circular tube cross-sectional area**

Tube cross-sectional area. The parameter is used if **Tube cross section type** is set to Non-circular. The default value is  $1e-4 \text{ m}^2$ .

## **Non-circular tube hydraulic diameter**

Hydraulic diameter of the tube cross section. The parameter is used if **Tube cross section type** is set to Non-circular. The default value is 0.0112 m.

## **Geometrical shape factor**

The parameter is used for computing friction factor at laminar flow and depends of the shape of the tube cross section. For a tube with non-circular cross section, you must set the factor to an appropriate value, for example, 56 for a square, 96 for concentric annulus, 62 for rectangle (2:1), and so on (see [1]). The default value is 64, which corresponds to a tube with a circular cross section.

## **Tube length**

Tube 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 tube. 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 tube 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, 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.

## **Global Parameters**

### **Fluid density**

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

### **Fluid kinematic viscosity**

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

## **Ports**

The block has the following ports:

A

Hydraulic conserving port associated with the tube inlet.

B

Hydraulic conserving port associated with the tube outlet.

## **References**

[1] White, F.M., *Viscous Fluid Flow*, McGraw-Hill, 1991

## **See Also**

Hydraulic Pipeline

# Rotational Damper

**Purpose** Simulate viscous damper in mechanical rotational systems

**Library** Mechanical Rotational Elements

**Description** The Rotational Damper block represents an ideal mechanical rotational viscous damper described with the following equations:



$$T = D \cdot \omega$$

$$\omega = \omega_R - \omega_C$$

where

$T$  Torque transmitted through the spring

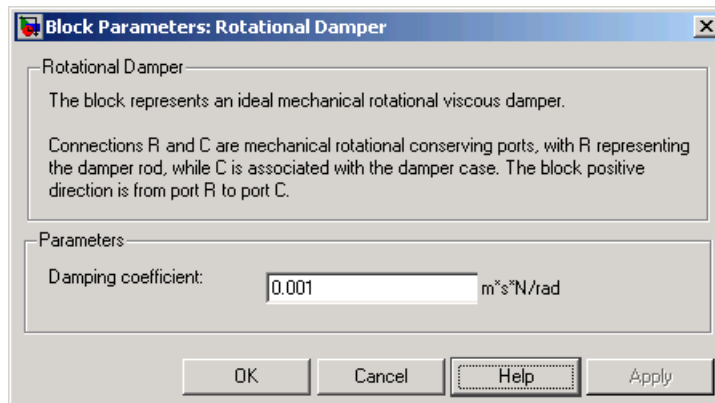
$D$  Damping (viscous friction) coefficient

$\omega$  Relative angular velocity

$\omega_R, \omega_C$  Absolute angular velocities of terminals R and C, respectively

The block positive direction is from port R to port C. This means that the torque is positive if it acts in the direction from R to C.

## Dialog Box and Parameters





**Damping coefficient**

Damping coefficient, defined by viscose friction. The default value is 0.001 N\*m/(rad/s).

**Ports**

The block has the following ports:

R

Mechanical rotational conserving port.

C

Mechanical rotational conserving port.

**See Also**

Rotational Friction

Rotational Hard Stop

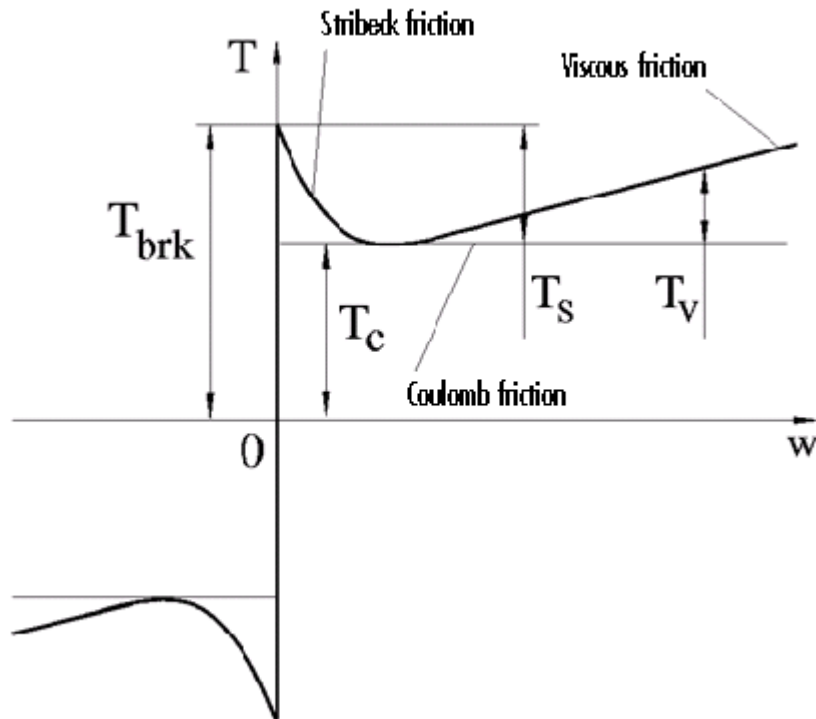
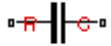
Rotational Spring

# Rotational Friction

**Purpose** Simulate friction in contact between rotating bodies

**Library** Mechanical Rotational Elements

**Description** The Rotational Friction block represents friction in contact between rotating bodies. The friction torque is simulated as a function of relative velocity and is assumed to be the sum of Stribeck, Coulomb, and viscous components, as shown in the following figure.



The Stribeck friction is the negatively sloped characteristics taking place at low velocities (see [1]). The Coulomb friction results in a

constant torque at any velocity. The viscous friction opposes motion with the torque directly proportional to the relative velocity. The sum of the Coulomb and Stribeck frictions at the vicinity of zero velocity is often referred to as the breakaway friction. The friction is approximated with the following equations:

$$T = (T_C + (T_{brk} - T_C) \cdot \exp(-c_v |\omega|)) \text{sign}(\omega) + f\omega$$

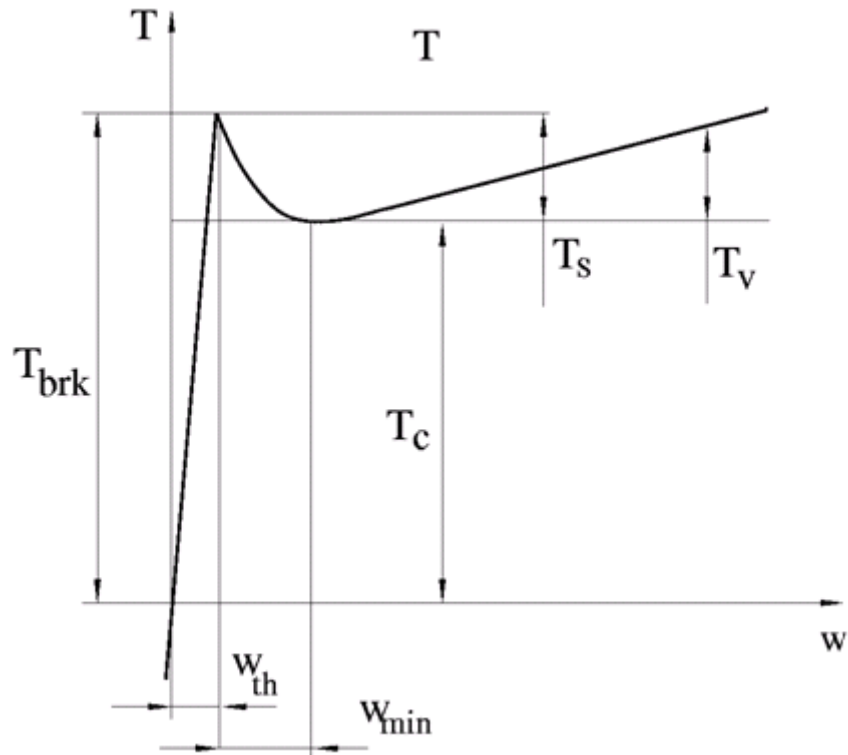
$$\omega = \omega_R - \omega_C$$

where

$T$	Friction torque
$T_C$	Coulomb friction torque
$T_{brk}$	Breakaway friction torque
$c_v$	Coefficient
$\omega$	Relative velocity
$\omega_R, \omega_C$	Absolute angular velocities of terminals R and C, respectively
$f$	Viscous friction coefficient

The approximation above is too idealistic and has a substantial drawback. The characteristic is discontinuous at  $\omega = 0$ , which creates considerable computational problems. It has been proven that the discontinuous friction model is a nonphysical simplification in the sense that the mechanical contact with distributed mass and compliance cannot exhibit an instantaneous change in torque (see [1]). There are numerous models of friction without discontinuity. The Rotational Friction block implements one of the simplest versions of continuous friction models. The friction torque-relative velocity characteristic of this approximation is shown in the following figure.

# Rotational Friction



The discontinuity is eliminated by introducing a very small, but finite, region in the zero velocity vicinity, within which friction torque is assumed to be linearly proportional to velocity, with the proportionality coefficient  $T_{brk}/\omega_{th}$ , where  $\omega_{th}$  is the velocity threshold. It has been proven experimentally that the velocity threshold in the range between  $10^{-3}$  and  $10^{-5}$  rad/s is a good compromise between the accuracy and computational robustness and effectiveness. Notice that friction torque computed with this approximation does not actually stop relative motion when an acting torque drops below breakaway friction level. The bodies will creep relative to each other at a very small velocity proportional to acting torque.

As a result of introducing the velocity threshold, the block equations are slightly modified:

- If  $|\omega| \geq \omega_{th}$ ,

$$T = (T_C + (T_{brk} - T_C) \cdot \exp(-c_v |\omega|)) \text{sign}(\omega) + f\omega$$

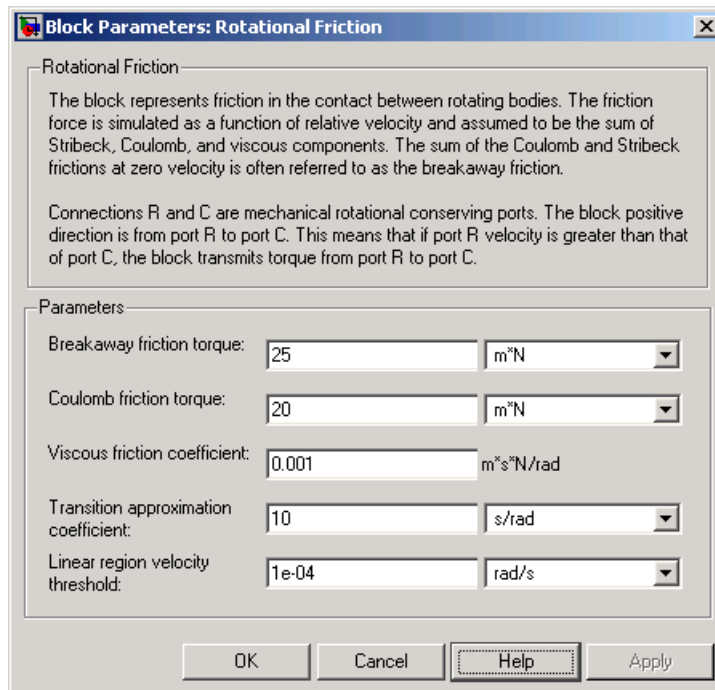
- If  $|\omega| < \omega_{th}$ ,

$$T = \omega \frac{(f\omega_{th} + (T_C + (T_{brk} - T_C) \cdot \exp(-c_v \omega_{th})))}{\omega_{th}}$$

The block positive direction is from port R to port C. This means that if the port R velocity is greater than that of port C, the block transmits torque from R to C.

# Rotational Friction

## Dialog Box and Parameters



### Breakaway friction torque

Breakaway friction torque, which is the sum of the Coulomb and the static frictions. It must be greater than or equal to the Coulomb friction torque value. The default value is 25 N\*m.

### Coulomb friction torque

Coulomb friction torque, which is the friction that opposes rotation with a constant torque at any velocity. The default value is 20 N\*m.

### Viscous friction coefficient

Proportionality coefficient between the friction torque and the relative angular velocity. The parameter value must be greater than or equal to zero. The default value is 0.001 N\*m/(rad/s).

## Transition approximation coefficient

The parameter sets the value of coefficient  $c_v$ , which is used for the approximation of the transition between the static and the Coulomb frictions. Its value is assigned based on the following considerations: the static friction component reaches approximately 95% of its steady-state value at velocity  $3/c_v$ , and 98% at velocity  $4/c_v$ , which makes it possible to develop an approximate relationship  $c_v \sim 4/\omega_{min}$ , where  $\omega_{min}$  is the relative velocity at which friction torque has its minimum value. By default,  $c_v$  is set to 10 rad/s, which corresponds to a minimum friction at velocity of about 0.4 s/rad.

## Linear region velocity threshold

The parameter sets the small vicinity near zero velocity, within which friction torque is considered to be linearly proportional to the relative velocity. The MathWorks recommends that you use values in the range between  $1e-5$  and  $1e-3$  rad/s. The default value is  $1e-4$  rad/s.

## Ports

The block has the following ports:

R

Mechanical rotational conserving port.

C

Mechanical rotational conserving port.

## Examples

The Mechanical Rotational System with Stick-Slip Motion demo (sh\_rot\_system\_stick\_slip) illustrates the use of the Rotational Friction block in mechanical systems. The friction element is installed between the load and the velocity source, and there is a difference between the breakaway and the Coulomb frictions. As a result, stick-slip motion is developed in the regions of constant velocities.

## References

[1] B. Armstrong, C.C. de Wit, *Friction Modeling and Compensation*, The Control Handbook, CRC Press, 1995

# Rotational Friction

---

## **See Also**

Rotational Damper

Rotational Hard Stop

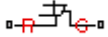
Rotational Spring



**Purpose** Simulate double-sided rotational hard stop

**Library** Mechanical Rotational Elements

## Description



The Rotational Hard Stop block represents a double-sided mechanical rotational hard stop that restricts motion of a body between upper and lower bounds. Both ports of the block are of mechanical rotational type. The impact interaction between the slider and the stops is assumed to be elastic. The stop is implemented as a spring that comes into contact with the slider as the gap is cleared. The spring opposes slider penetration into the stop with the force linearly proportional to this penetration. To account for energy dissipation and nonelastic effects, the damping is introduced as a block parameter, thus making it possible to account for energy loss.

The hard stop is described with the following equations:

$$T = \begin{cases} K_p \cdot \delta + D_p (\omega_R - \omega_C) & \text{for } \delta \geq g_p \\ 0 & \text{for } g_n < \delta < g_p \\ K_n \cdot \delta + D_n (\omega_R - \omega_C) & \text{for } \delta \leq g_n \end{cases}$$

$$\delta = \varphi_R - \varphi_C$$

$$\omega_R = \frac{d\varphi_R}{dt}$$

$$\omega_C = \frac{d\varphi_C}{dt}$$

where

- $T$  Interaction torque between the slider and the case
- $\delta$  Relative angular displacement between the slider and the case

# Rotational Hard Stop

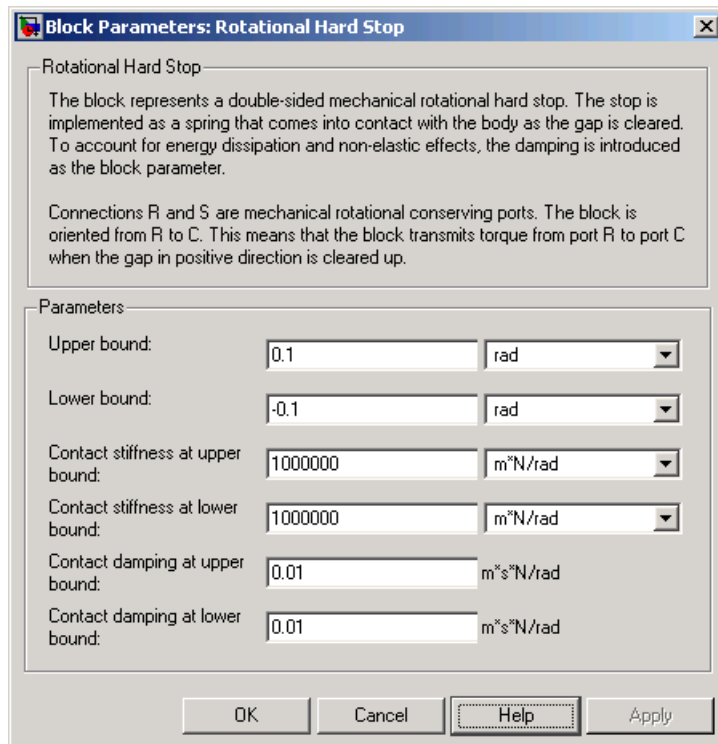
---

$g_p$	Gap between the slider and the case in positive direction
$g_n$	Gap between the slider and the case in negative direction
$\omega_R, \omega_C$	Absolute angular velocities of terminals R and C, respectively
$\phi_R, \phi_C$	Absolute angular displacements of terminals R and C, respectively
$K_p$	Contact stiffness at positive restriction
$K_n$	Contact stiffness at negative restriction
$D_p$	Damping coefficient at positive restriction
$D_n$	Damping coefficient at negative restriction
$t$	Time

The equations are derived with respect to the local coordinate system whose axis is directed clockwise from port R to port C. The terms “positive” and “negative” in the variable descriptions refer to this coordinate system, and the gap in negative direction must be specified with negative value. If the local coordinate system is not aligned with the globally assigned positive direction, the gaps interchange their values with respective sign adjustment.

The block is oriented from R to C. This means that the block transmits torque from port R to port C when the gap in positive direction is cleared up.

## Dialog Box and Parameters



### Upper bound

Gap between the slider and the upper bound. The direction is specified with respect to the local coordinate system, with the slider located in the origin. A positive value of the parameter specifies the gap between the slider and the upper bound. A negative value sets the slider as penetrating into the upper bound. The default value is 0.1 rad.

### Lower bound

Gap between the slider and the lower bound. The direction is specified with respect to the local coordinate system, with the slider located in the origin. A negative value of the parameter specifies the gap between the slider and the lower bound. A

# Rotational Hard Stop

---

positive value sets the slider as penetrating into the lower bound. The default value is  $-0.1$  rad.

## **Contact stiffness at upper bound**

The parameter specifies the elastic property of colliding bodies when the slider hits the upper bound. The greater the value of the parameter, the less the bodies penetrate into each other, the more rigid the impact becomes. Lesser value of the parameter makes contact softer, but generally improves convergence and computational efficiency. The default value is  $1e6$  N\*m/rad.

## **Contact stiffness at lower bound**

The parameter specifies the elastic property of colliding bodies when the slider hits the lower bound. The greater the value of the parameter, the less the bodies penetrate into each other, the more rigid the impact becomes. Lesser value of the parameter makes contact softer, but generally improves convergence and computational efficiency. The default value is  $1e6$  N\*m/rad.

## **Contact damping at upper bound**

The parameter specifies dissipating property of colliding bodies when the slider hits the upper bound. At zero damping, the impact is close to an absolutely elastic one. The greater the value of the parameter, the more energy dissipates during an interaction. Keep in mind that damping affects slider motion as long as the slider is in contact with the stop, including the period when slider is pulled back from the contact. For computational efficiency and convergence reasons, The MathWorks recommends that you assign a nonzero value to this parameter. The default value is  $0.01$  N\*m\*s/rad.

## **Contact damping at lower bound**

The parameter specifies dissipating property of colliding bodies when the slider hits the lower bound. At zero damping, the impact is close to an absolutely elastic one. The greater the value of the parameter, the more energy dissipates during an interaction. Keep in mind that damping affects slider motion as long as the slider is in contact with the stop, including the period when slider is pulled back from the contact. For computational efficiency

and convergence reasons, The MathWorks recommends that you assign a nonzero value to this parameter. The default value is  $0.01 \text{ N}\cdot\text{m}\cdot\text{s}/\text{rad}$ .

## Ports

The block has the following ports:

R

Mechanical rotational conserving port associated with the slider that travels between stops installed on the case.

C

Mechanical rotational conserving port associated with the case.

## See Also

Rotational Damper

Rotational Friction

Rotational Spring

# Rotational Spring

---

**Purpose** Simulate ideal spring in mechanical rotational systems

**Library** Mechanical Rotational Elements

**Description** The Rotational Spring block represents an ideal mechanical rotational linear spring, described with the following equations:



$$T = K \cdot \phi$$

$$\phi = \phi_{init} + \phi_R - \phi_C$$

$$\omega = \frac{d\phi}{dt}$$

where

$T$  Torque transmitted through the spring

$K$  Spring rate

$\phi$  Relative displacement angle (spring deformation)

$\phi_{init}$  Spring preliminary winding (spring offset)

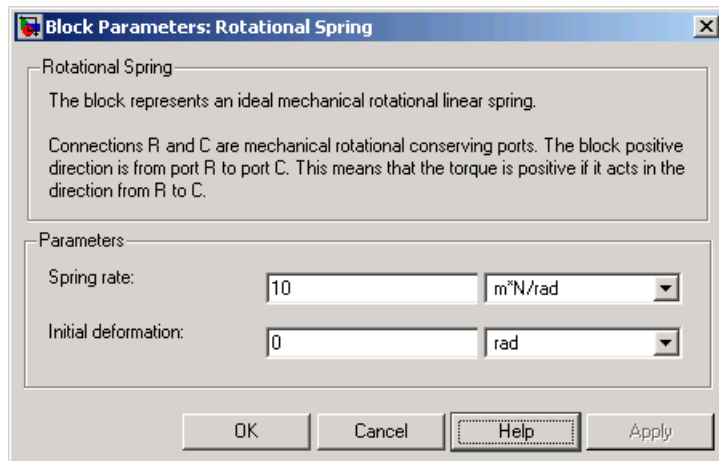
$\phi_R, \phi_C$  Absolute angular displacements of terminals R and C, respectively

$\omega$  Relative angular velocity

$t$  Time

The block positive direction is from port R to port C. This means that the torque is positive if it acts in the direction from R to C.

## Dialog Box and Parameters



### Spring rate

Spring rate. The default value is 10 N\*m/rad.

### Initial deformation

Spring initial deformation, or offset, in angular units. The deformation is determined as  $\phi = \phi_{\text{init}} + \phi_R - \phi_C$ , where  $\phi_{\text{init}}$  is the initial deformation, and  $\phi_R, \phi_C$  are the absolute angular displacements of terminals R and C in the globally assigned positive direction. The spring can be initially compressed ( $\phi_{\text{init}} > 0$ ) or stretched ( $\phi_{\text{init}} < 0$ ). This parameter specifies the initial condition for use in computing the block's initial state at the beginning of a simulation run. For more information, see "Computing Initial Conditions". The default value is 0.

## Ports

The block has the following ports:

- R  
Mechanical rotational conserving port.
- C  
Mechanical rotational conserving port.

# Rotational Spring

---

## **See Also**

Rotational Damper

Rotational Friction

Rotational Hard Stop



# Segmented Pipeline

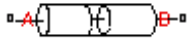
## Purpose

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

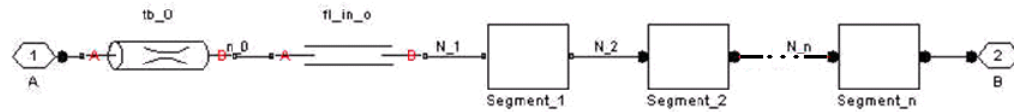
## Library

Pipelines

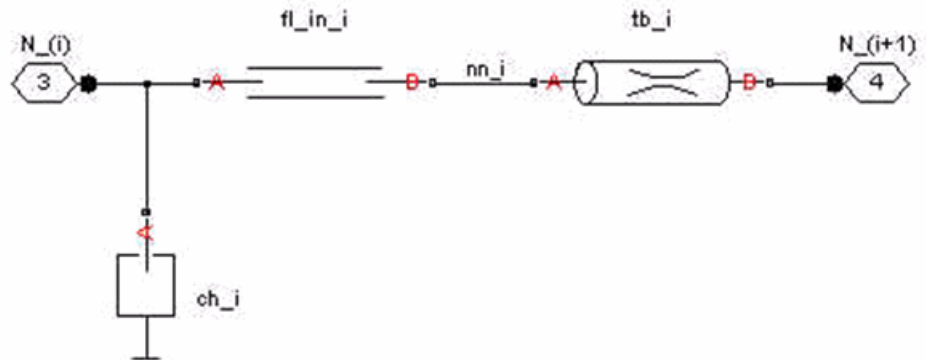
## Description



The Segmented Pipeline block models hydraulic pipelines with circular cross sections. Hydraulic pipelines, which are inherently distributed parameter elements, are represented with sets of identical, connected in series, lumped parameter segments. It is assumed that the larger the number of segments, the closer the lumped parameter model becomes to its distributed parameter counterpart. The equivalent circuit of a pipeline adopted in the block is shown below, along with the segment configuration.



## Pipeline Equivalent Circuit



## Segment Configuration

# Segmented Pipeline

---

The model contains as many Constant Chamber blocks as there are segments. The chamber lumps fluid volume equal to

$$V = \frac{\pi \cdot d^2}{4} \frac{L}{N}$$

where

- $V$  Fluid volume
- $d$  Pipe diameter
- $L$  Pipe length
- $N$  Number of segments

The Constant Chamber block is placed between two branches, each consisting of a Resistive Tube block and a Fluid Inertia block. Every Resistive Tube block lumps  $(L+L_{ad}) / (N+1)$ -th portion of the pipe length, while Fluid Inertia block has  $L / (N+1)$  length ( $L_{ad}$  denotes additional pipe length equal to aggregate equivalent length of pipe local resistances, such as fitting, elbows, bends, and so on).

The nodes to which Constant Chamber blocks are connected are assigned names  $N_1, N_2, \dots, N_n$  ( $n$  is the number of segments). Pressures at these nodes are assumed to be equal to average pressure of the segment. Intermediate nodes between Resistive Tube and Fluid Inertia blocks are assigned names  $nn_0, nn_1, nn_2, \dots, nn_n$ . The Constant Chamber blocks are named  $ch_1, ch_2, \dots, ch_n$ , Resistive Tube blocks are named  $tb_0, tb_1, tb_2, \dots, tb_n$ , and Fluid Inertia blocks are named  $fl\_in_0, fl\_in_1, fl\_in_2, \dots, fl\_in_n$ .

The number of segments is the block parameter. In determining the number of segments needed, you have to find a compromise between the accuracy and computational burden for a particular application. It is practically impossible to determine analytically how many elements are necessary to get the results with a specified accuracy. The golden rule is to use as many elements as possible based on computational considerations, and an experimental assessment is perhaps the only

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

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

where

- $N$  Number of segments
- $L$  Pipe length
- $c$  Speed of sound in the fluid
- $\omega$  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  $p = p_A - p_B$ .

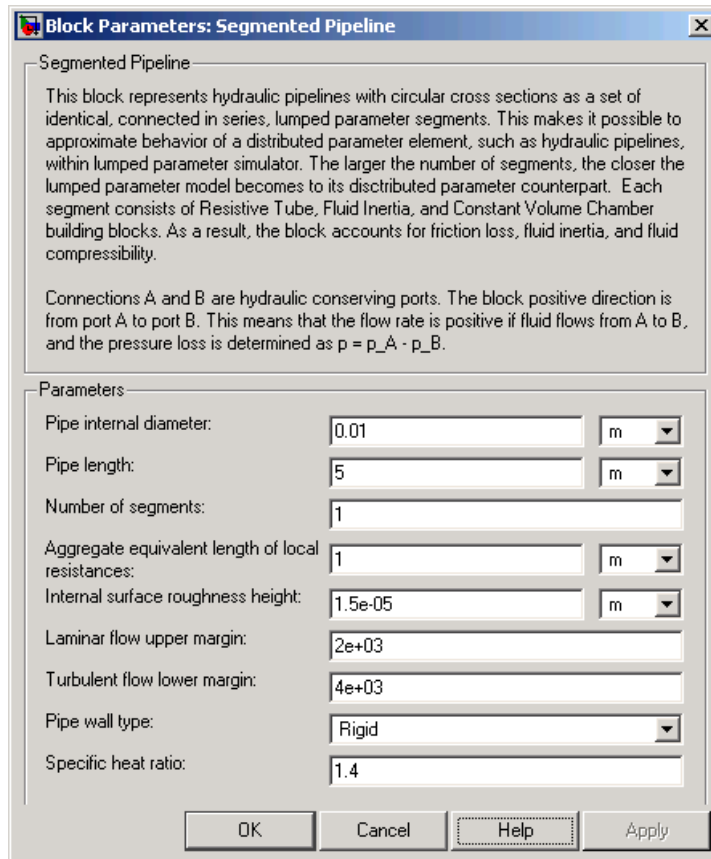
# Segmented Pipeline

## Basic Assumptions and Limitations

## Dialog Box and Parameters

The model is based on the following assumption:

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



### Pipe internal diameter

Pipe internal diameter. The parameter is used if **Pipe cross section type** is set to Circular. The default value is 0.01 m.

**Pipe length**

Pipe geometrical length. The default value is 5 m.

**Number of segments**

Number of lumped parameter segments in the pipeline model.  
The default value is 1.

**Aggregate equivalent length of local resistances**

This parameter represents total equivalent length of all local resistances associated with the pipe. You can account for the pressure loss caused by local resistances, such as bends, fittings, armature, inlet/outlet losses, and so on, by adding to the pipe geometrical length an aggregate equivalent length of all the local resistances. The default value is 1 m.

**Internal surface roughness height**

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

**Laminar flow upper margin**

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

**Turbulent flow lower margin**

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

**Pipe wall type**

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

# Segmented Pipeline

---

## **Static pressure-diameter coefficient**

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

## **Viscoelastic process time constant**

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

## **Specific heat ratio**

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

## **Global Parameters**

### **Fluid density**

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

### **Fluid kinematic viscosity**

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

## **Ports**

The block has the following ports:

A

Hydraulic conserving port associated with the pipe inlet.

B

Hydraulic conserving port associated with the pipe outlet.

## **See Also**

Resistive Tube

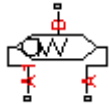
## Purpose

Simulate hydraulic valve that allows flow in one direction only

## Library

Directional Valves

## Description



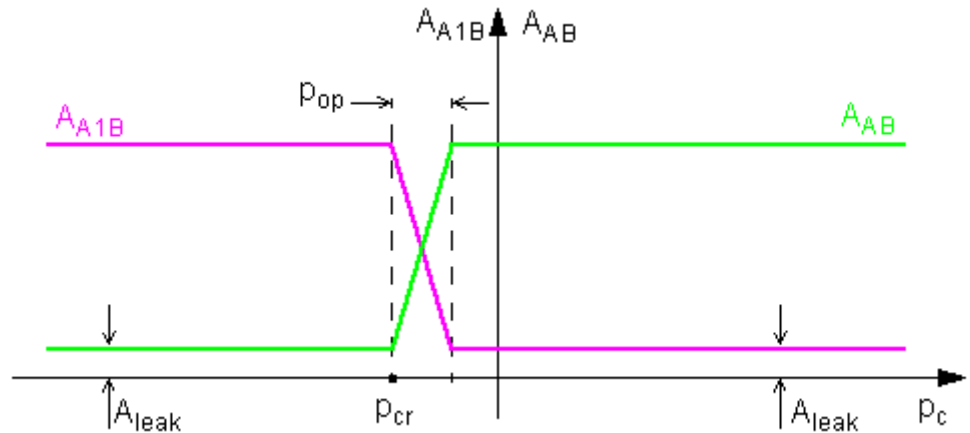
The Shuttle Valve block represents a hydraulic shuttle valve as a data-sheet-based model. The valve has two inlet ports (A and A1) and one outlet port (B). The valve is controlled by pressure differential

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

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

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

# Shuttle Valve



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

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

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



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

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

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

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

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

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

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

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

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

# Shuttle Valve

---

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

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

where

$q_{AB}, q_{A1B}$	Flow rates through the AB and A1B orifices
$p_{AB}, p_{A1B}$	Pressure differentials across the AB and A1B orifices
$p_A, p_{A1}, p_B$	Gauge pressures at the block terminals
$C_D$	Flow discharge coefficient
$A_{AB}, A_{A1B}$	Instantaneous orifice AB and A1B passage areas
$A_{max}$	Fully open orifice passage area
$A_{leak}$	Closed valve leakage area
$p_{cr}$	Valve cracking pressure differential
$p_{opx}$	Pressure differential needed to fully shift the valve
$D_{HAB}, D_{HA1B}$	Instantaneous orifice hydraulic diameters
$\rho$	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

The model is based on the following assumptions:

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

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

## Dialog Box and Parameters

**Block Parameters: Shuttle Valve**

Shuttle Valve

This block represents hydraulic shuttle valve as a data sheet-based model. The valve has two inlet ports (A and A1) and one outlet port (B). The valve is controlled by pressure differential  $P_c = P_a - P_{a1}$ . The valve permits flow either between ports A and B or ports A1 and B depending on pressure differential  $P_c$ . Initially, path A-B is assumed to be opened. To open A1-B path (and close A-B at the same time), pressure differential must be greater than the valve cracking pressure ( $P_{cr} <= 0$ ).

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

Parameters

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

OK Cancel Help Apply

### Maximum passage area

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

### Cracking pressure

Pressure differential level at which the orifice of the valve starts to open. The default value is  $-1e4 \text{ Pa}$ .

### Opening pressure

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

# Shuttle 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.

## **Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

## **Leakage area**

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is  $1e-12 \text{ m}^2$ .

## **Global Parameters**

### **Fluid density**

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

### **Fluid kinematic viscosity**

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

## **Ports**

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

- A1 Hydraulic conserving port associated with the valve inlet.
- B Hydraulic conserving port associated with the valve outlet.

## **See Also**

Check Valve  
Pilot-Operated Check Valve

# Simulink-PS Converter

**Purpose** Convert Simulink input signal into physical signal

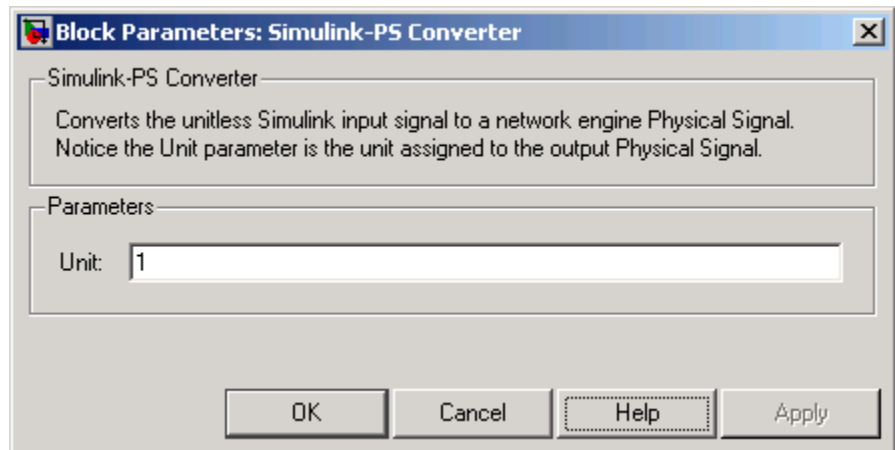
**Library** Utilities

**Description** The Simulink-PS Converter block converts the input Simulink signal into a physical signal, in specified units. You specify the desired units as the **Unit** parameter. If you leave the block unitless, with the **Unit** parameter set to 1, then the physical signal units are inferred from the destination block. The default destination block units are meter-kilogram-second or MKS (SI).



Use this block to connect Simulink sources or other Simulink blocks to the inputs of a Physical Network diagram.

## Dialog Box and Parameters



**Unit** Units to be assigned to the physical signal. You can type the desired unit name, such as rpm, or a valid expression, such as rad/s. For more information and a list of unit abbreviations, see “Working with Physical Units”. The default value is 1, which means that the units of the physical signal at the block output match the expected default units of the destination block input.

**Ports** The block has a Simulink input port, located on its left-hand side, and a physical signal output port, located on its right-hand side (in the block default orientation).

**See Also** PS-Simulink Converter





# Single-Acting Hydraulic Cylinder

---

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

The energy through port A is directed to the Translational Hydro-Mechanical Converter block and the Variable Volume Chamber block. The converter transforms hydraulic energy into mechanical energy, while the chamber accounts for the fluid compressibility in the cylinder chamber. The rod motion is limited with the mechanical Translational Hard Stop block in such a way that the rod can travel only between cylinder caps. The Ideal Translational Motion Sensor block in the schematic is introduced to determine an instantaneous piston position, which is necessary for the Variable Volume Chamber block.

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

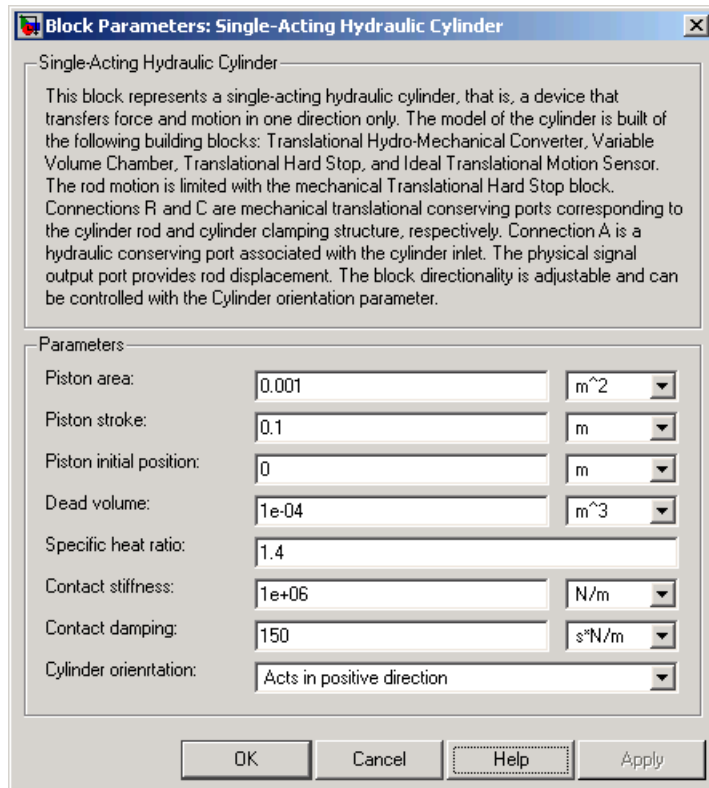
## Basic Assumptions and Limitations

The model is based on the following assumptions:

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

# Single-Acting Hydraulic Cylinder

## Dialog Box and Parameters



### Piston area

Effective piston area. The default value is 0.001 m<sup>2</sup>.

### Piston stroke

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

### Piston initial position

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

**Dead volume**

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

**Specific heat ratio**

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

**Contact stiffness**

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

**Contact damping**

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

**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

---

## Global Parameters

### Fluid bulk modulus

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the cylinder inlet.

R

Mechanical translational conserving port associated with the cylinder rod.

C

Mechanical translational conserving port associated with the cylinder clamping structure.

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

## See Also

Double-Acting Hydraulic Cylinder

Ideal Translational Motion Sensor

Translational Hard Stop

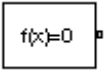
Translational Hydro-Mechanical Converter

Variable Volume Chamber

**Purpose** Represent Physical Networks environment and solver configuration

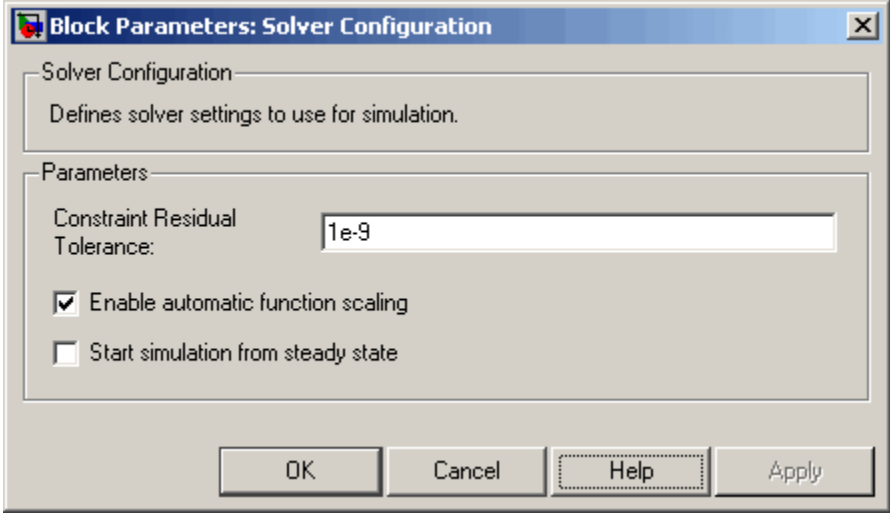
**Library** Utilities

**Description** Each hydro-mecahnical device represented by a connected SimHydraulics block diagram requires global environment information for simulation. The Solver Configuration block specifies this global information and connects the solver that your model needs before you can begin simulation.



Each topologically distinct SimHydraulics block diagram requires exactly one Solver Configuration block to be connected to it.

### Dialog Box and Parameters



**Constraint Residual Tolerance**  
Specify the relative tolerance. The default value is 1e-9.

**Enable automatic function scaling**  
Enables automatic scaling of equations at the time of transient initialization. This makes the equations more stable numerically, but it may slow down the simulation.

# Solver Configuration

---

## **Start simulation from steady state**

When this box is selected, the solver attempts to find the steady state that would result if the inputs to the system were held constant for a sufficiently large time, starting from the initial state obtained from the initial conditions computation. For more information, see “Computing Initial Conditions”. Simulation then starts from this steady state.

## **Ports**

The block has one conserving port. You can add this block anywhere on a physical network circuit by creating a branching point and connecting it to the only port of the Solver Configuration block.

**Purpose** Simulate hydraulic accumulator with spring used for energy storage

**Library** Accumulators

## Description



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

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

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

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

where

$p$  Pressure at the accumulator inlet

$q$  Flow rate into accumulator

$V_{max}$  Accumulator capacity (maximum volume)

$V_F$  Instantaneous volume of fluid in the accumulator

$p_{pr}$  Preload pressure

$p_{max}$  Pressure needed to fully fill the accumulator

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

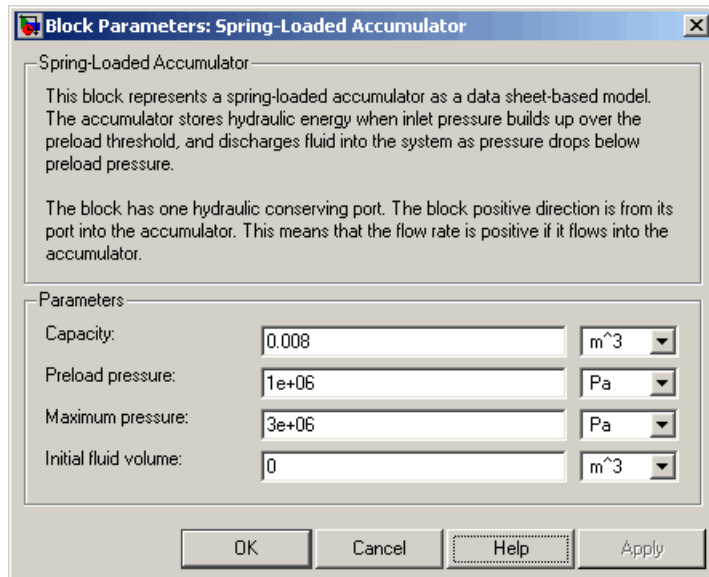
# Spring-Loaded Accumulator

## Basic Assumptions and Limitations

The model is based on the following assumptions:

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

## Dialog Box and Parameters



### Accumulator 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.

### Accumulator maximum pressure

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



**Initial fluid volume**

Initial volume of fluid in the accumulator. This parameter specifies the initial condition for use in computing the block's initial state at the beginning of a simulation run. For more information, see "Computing Initial Conditions". The default value is 0.

**Ports**

The block has one hydraulic conserving port associated with the accumulator inlet.

The flow rate is positive if fluid flows into the accumulator.

**See Also**

Gas-Charged Accumulator

# Sudden Area Change

---

**Purpose** Simulate sudden enlargement or contraction

**Library** Local Hydraulic Resistances

**Description**



The Sudden Area Change block represents a local hydraulic resistance, such as a sudden cross-sectional area change. The resistance represents a sudden enlargement if fluid flows from inlet to outlet, or a sudden contraction if fluid flows from outlet to inlet. The block is based on the Local Resistance block. It determines the pressure loss coefficient and passes its value to the Local Resistance block. The block offers two methods of parameterization: by applying semi-empirical formulas or by table lookup for the pressure loss coefficient based on the Reynolds number.

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

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

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

where

$K_{SE}$  Pressure loss coefficient for the sudden enlargement, which takes place if fluid flows from inlet to outlet

$K_{SC}$  Pressure loss coefficient for the sudden contraction, which takes place if fluid flows from outlet to inlet

$K_{cor}$  Correction factor

$A_S$  Small area

$A_L$  Large area

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

The pressure loss coefficient, determined by either of the two methods, is then passed to the Local Resistance block, which computes the pressure loss according to the formulas explained in the reference documentation for that block.

The Sudden Area Change block is bidirectional and computes pressure loss for both the direct flow (sudden enlargement) and return flow (sudden contraction). If the loss coefficient is specified by a table, the table must cover both the positive and the negative flow regions.

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

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

## Basic Assumptions and Limitations

The model is based on the following assumption:

- Fluid inertia is not taken into account.
- If you select parameterization by semi-empirical formulas, the flow is assumed to be completely turbulent and fully developed along the pipe length.

# Sudden Area Change

## 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

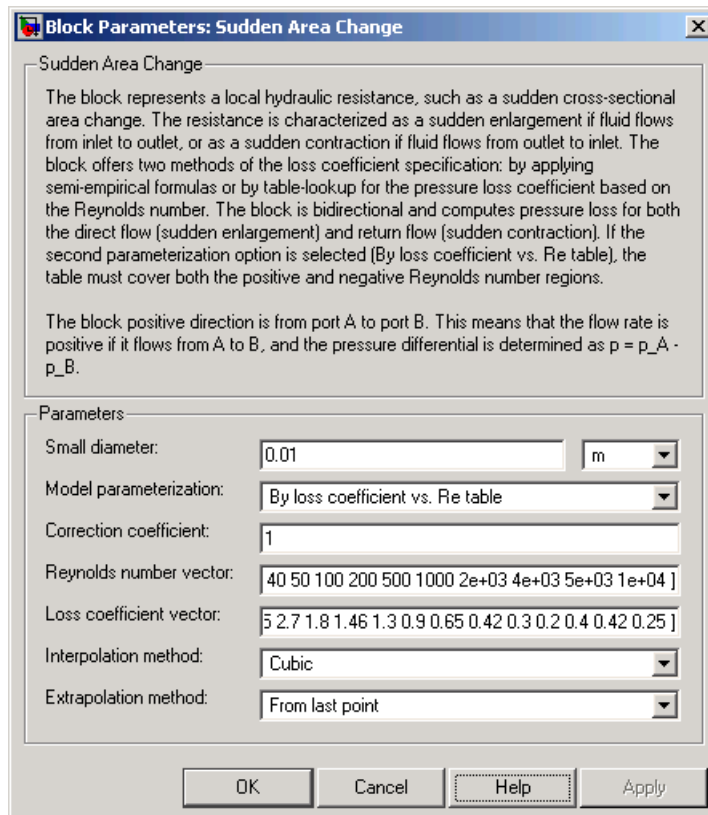
Small diameter:  m

Large diameter:  m

Model parameterization:

Correction coefficient:

OK Cancel Help Apply



## 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:

# Sudden Area Change

---

- 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.

## **Reynolds number vector**

Specify the vector of input values for Reynolds numbers as a tabulated 1-by-m array. The input values vector must be strictly monotonically increasing. The values can be non-uniformly spaced. You must provide at least three values. The default values are [-4000, -3000, -2000, -1000, -500, -200, -100, -50, -40, -30, -20, -15, -10, 10, 20, 30, 40, 50, 100, 200, 500, 1000, 2000, 4000, 5000, 10000]. This parameter is used if **Model parameterization** is set to By loss coefficient vs. Re table.

## **Loss coefficient vector**

Specify the vector of output values for the loss coefficient as a tabulated 1-by-m array. The loss coefficient vector must be the same size as the Reynolds numbers vector. The default values are [0.25, 0.3, 0.65, 0.9, 0.65, 0.75, 0.90, 1.15, 1.35, 1.65, 2.3, 2.8, 3.10, 5, 2.7, 1.8, 1.46, 1.3, 0.9, 0.65, 0.42, 0.3, 0.20, 0.40, 0.42, 0.25]. This parameter is used if **Model parameterization** is set to By loss coefficient vs. Re table.

## Interpolation method

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

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

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

## Extrapolation method

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

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

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

# Sudden Area Change

---

## Global Parameters

### Fluid density

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

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the resistance inlet.

B

Hydraulic conserving port associated with the resistance outlet.

## References

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

## See Also

Elbow

Gradual Area Change

Local Resistance

Pipe Bend

T-junction



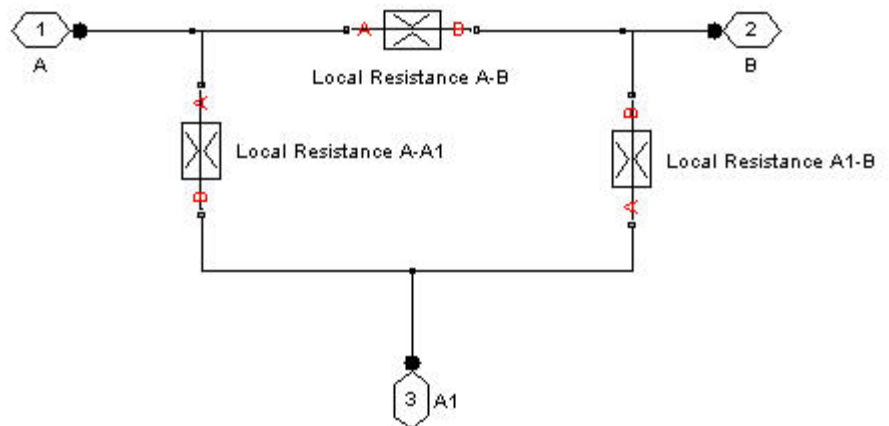
**Purpose** Simulate hydraulic resistance of T-junction in pipe

**Library** Local Hydraulic Resistances

## Description



The T-junction block represents a T-junction (wye connection) consisting, in general, of a main run and a branch merging to the main run. The junction as a hydraulic resistance is built of three Local Resistance blocks, as shown in the following diagram.



To specify pressure loss for all possible flow directions, you have to provide six pressure loss coefficients.

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

# T-junction

## Basic Assumptions and Limitations

The model is based on the following assumption:

- Fluid inertia is not taken into account.
- The flow is assumed to be completely turbulent and fully developed along the pipe length.

## 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"/>	

OK Cancel Help Apply

### 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.

**Global Parameters****Fluid density**

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

**Fluid kinematic viscosity**

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

**Ports**

The block has the following ports:

# T-junction

---

- A Hydraulic conserving port associated with the main run inlet.
- B Hydraulic conserving port associated with the main run outlet.
- A1 Hydraulic conserving port associated with the branch inlet.

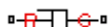
## See Also

- Elbow
- Gradual Area Change
- Local Resistance
- Pipe Bend
- Sudden Area Change

**Purpose** Simulate viscous damper in mechanical translational systems

**Library** Mechanical Translational Elements

**Description** The Translational Damper block represents an ideal mechanical translational viscous damper, described with the following equations:



$$F = Dv$$

$$v = v_R - v_C$$

where

$F$  Force transmitted through the damper

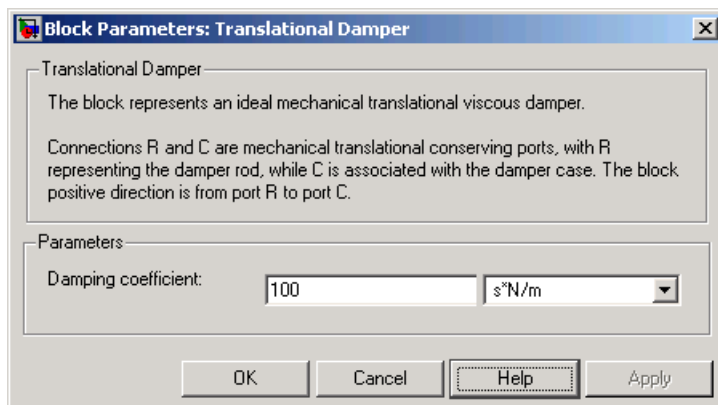
$D$  Damping (viscous friction) coefficient

$v$  Relative velocity

$v_R, v_C$  Absolute velocities of terminals R and C, respectively

The block positive direction is from port R to port C. This means that the force is positive if it acts in the direction from R to C.

## Dialog Box and Parameters



# Translational Damper

---

## **Damping coefficient**

Damping coefficient, defined by viscose friction. The default value is 100 N/(m/s).

## **Ports**

The block has the following ports:

R

Mechanical translational conserving port associated with the damper rod.

C

Mechanical translational conserving port associated with the damper case.

## **See Also**

Translational Friction

Translational Hard Stop

Translational Spring

# Translational Friction

## Purpose

Simulate friction in contact between moving bodies

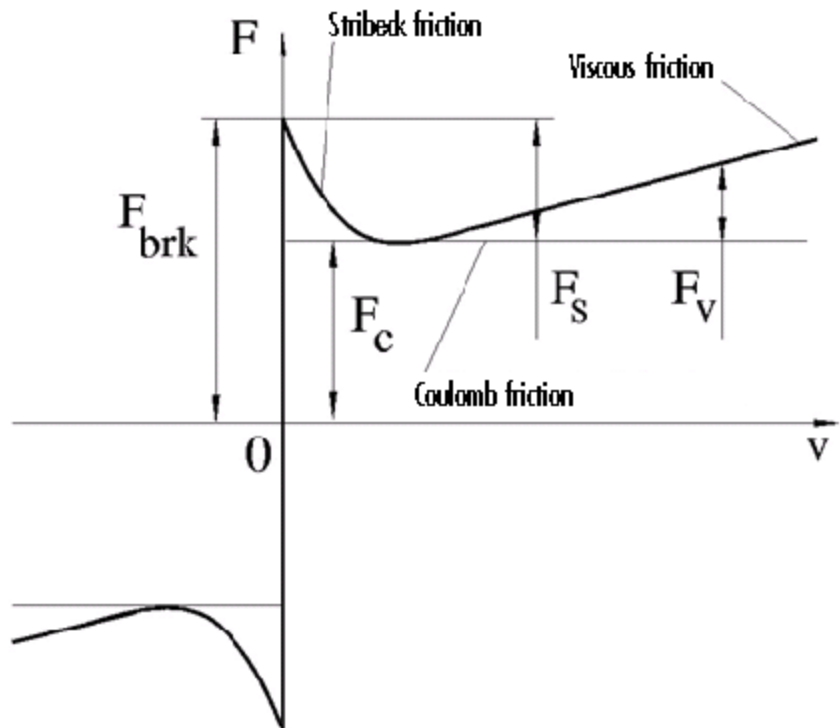
## Library

Mechanical Translational Elements

## Description



The Translational Friction block represents friction in contact between moving bodies. The friction force is simulated as a function of relative velocity and is assumed to be the sum of Stribeck, Coulomb, and viscous components, as shown in the following figure.



The Stribeck friction is the negatively sloped characteristics taking place at low velocities (see [1]). The Coulomb friction results in a

# Translational Friction

---

constant force at any velocity. The viscous friction opposes motion with the force directly proportional to the relative velocity. The sum of the Coulomb and Stribeck frictions at the vicinity of zero velocity is often referred to as the breakaway friction. The friction is approximated with the following equations:

$$F = (F_C + (F_{brk} - F_C) \cdot \exp(-c_v |v|)) \text{sign}(v) + fv$$

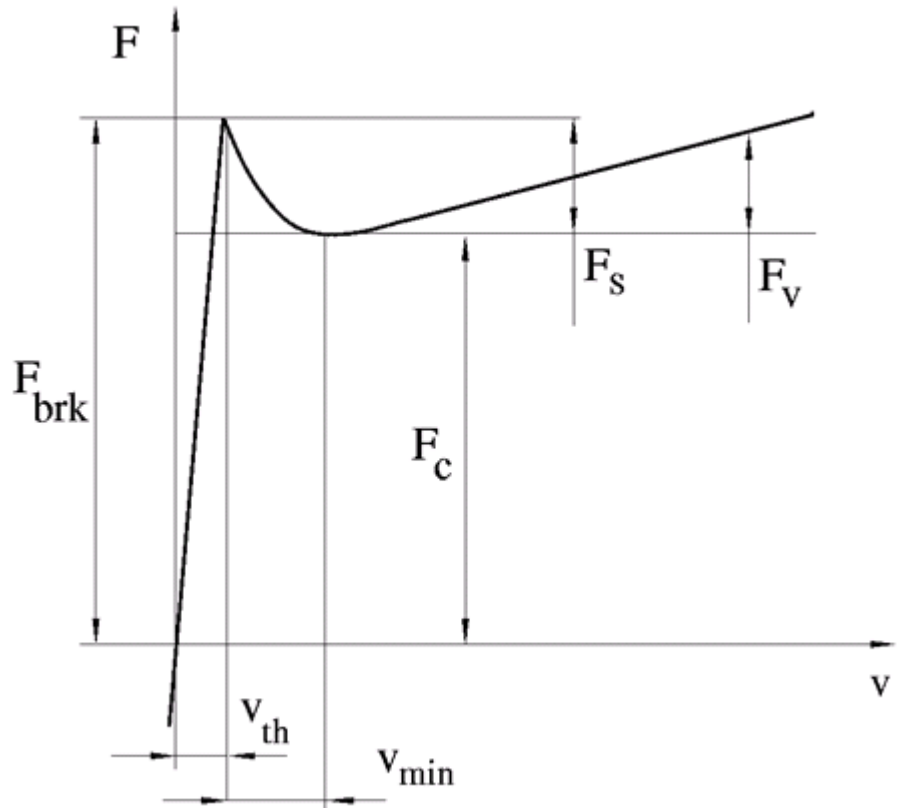
$$v = v_R - v_C$$

where

$F$	Friction force
$F_C$	Coulomb friction
$F_{brk}$	Breakaway friction
$c_v$	Coefficient
$v$	Relative velocity
$v_R, v_C$	Absolute velocities of terminals R and C, respectively
$f$	Viscous friction coefficient

The approximation above is too idealistic and has a substantial drawback. The characteristic is discontinuous at  $v = 0$ , which creates considerable computational problems. It has been proven that the discontinuous friction model is a nonphysical simplification in the sense that the mechanical contact with distributed mass and compliance cannot exhibit an instantaneous change in force (see [1]). There are numerous models of friction without discontinuity. The Translational Friction block implements one of the simplest versions of continuous friction models. The friction force-relative velocity characteristic of this approximation is shown in the following figure.





The discontinuity is eliminated by introducing a very small, but finite, region in the zero velocity vicinity, within which friction force is assumed to be linearly proportional to velocity, with the proportionality coefficient  $F_{brk}/v_{th}$ , where  $v_{th}$  is the velocity threshold. It has been proven experimentally that the velocity threshold in the range between  $10^{-4}$  and  $10^{-6}$  m/s is a good compromise between the accuracy and computational robustness and effectiveness. Notice that friction force computed with this approximation does not actually stop relative

# Translational Friction

---

motion when an acting force drops below breakaway friction level. The bodies will creep relative to each other at a very small velocity proportional to acting force.

As a result of introducing the velocity threshold, the block equations are slightly modified:

- If  $|v| \geq v_{th}$ ,

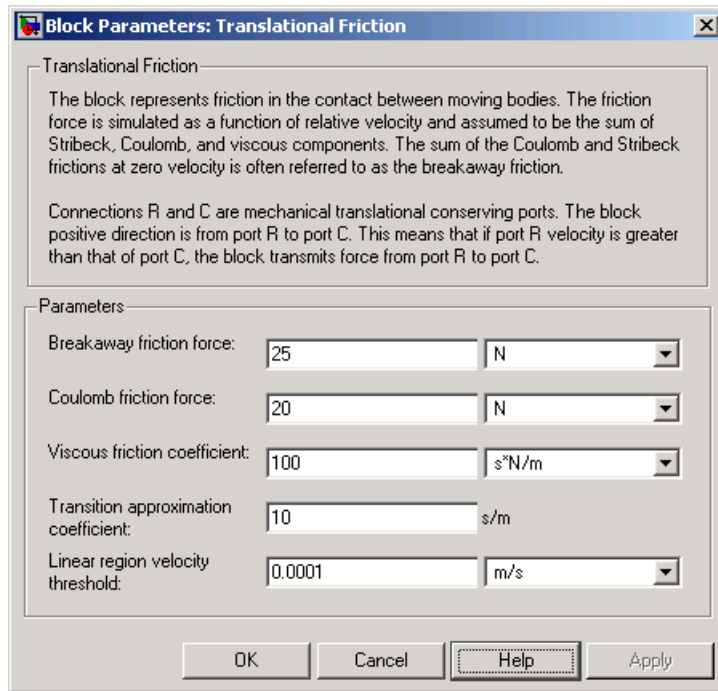
$$F = (F_C + (F_{brk} - F_C) \cdot \exp(-c_v |v|)) \text{sign}(v) + fv$$

- If  $|v| < v_{th}$ ,

$$F = v \frac{(fv_{th} + (F_C + (F_{brk} - F_C) \cdot \exp(-c_v v_{th})))}{v_{th}}$$

The block positive direction is from port R to port C. This means that if the port R velocity is greater than that of port C, the block transmits force from R to C.

## Dialog Box and Parameters



### Breakaway friction force

Breakaway friction force, which is the sum of the Coulomb and the static frictions. It must be greater than or equal to the Coulomb friction force value. The default value is 25 N.

### Coulomb friction force

Coulomb friction force, which is the friction that opposes motion with a constant force at any velocity. The default value is 20 N.

### Viscous friction coefficient

Proportionality coefficient between the 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).

# Translational Friction

---

## Transition approximation coefficient

The parameter sets the value of coefficient  $c_v$ , which is used for the approximation of the transition between the static and the Coulomb frictions. Its value is assigned based on the following considerations: the static friction component reaches approximately 95% of its steady-state value at velocity  $3/c_v$ , and 98% at velocity  $4/c_v$ , which makes it possible to develop an approximate relationship  $c_v \approx 4/v_{min}$ , where  $v_{min}$  is the relative velocity at which friction force has its minimum value. By default,  $c_v$  is set to 10 s/m, which corresponds to a minimum friction at velocity of about 0.4 m/s.

## Linear region velocity threshold

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

## Ports

The block has the following ports:

R

Mechanical translational conserving port.

C

Mechanical translational conserving port.

## References

[1] B. Armstrong, C.C. de Wit, *Friction Modeling and Compensation*, The Control Handbook, CRC Press, 1995

## See Also

Translational Damper

Translational Hard Stop

Translational Spring

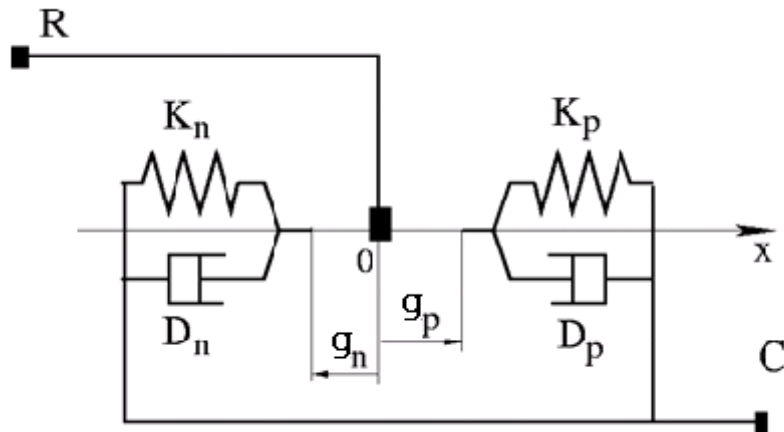
**Purpose** Simulate double-sided translational hard stop

**Library** Mechanical Translational Elements

**Description**



The Translational Hard Stop block represents a double-sided mechanical translational hard stop that restricts motion of a body between upper and lower bounds. Both ports of the block are of mechanical translational type. The impact interaction between the slider and the stops is assumed to be elastic. This means that the stop is represented as a spring that comes into contact with the slider as the gap is cleared and opposes slider penetration into the stop with the force linearly proportional to this penetration. To account for energy dissipation and nonelastic effects, the damping is introduced as the block's parameter, thus making it possible to account for energy loss. The following schematic shows the idealization of the mechanical translational hard stop adopted in the block:



The hard stop is described with the following equations:

# Translational Hard Stop

---

$$F = \begin{cases} K_p \cdot \delta + D_p (v_R - v_C) & \text{for } \delta \geq g_p \\ 0 & \text{for } g_n < \delta < g_p \\ K_n \cdot \delta + D_n (v_R - v_C) & \text{for } \delta \leq -g_n \end{cases}$$

$$\delta = x_R - x_C$$

$$v_R = \frac{dx_R}{dt}$$

$$v_C = \frac{dx_C}{dt}$$

where

$F$	Interaction force between the slider and the case
$\delta$	Relative displacement between the slider and the case
$g_p$	Gap between the slider and the case in positive direction
$g_n$	Gap between the slider and the case in negative direction
$v_R, v_C$	Absolute velocities of terminals R and C, respectively
$x_R, x_C$	Absolute displacements of terminals R and C, respectively
$K_p$	Contact stiffness at positive restriction
$K_n$	Contact stiffness at negative restriction
$D_p$	Damping coefficient at positive restriction
$D_n$	Damping coefficient at negative restriction
$t$	Time

The equations are derived with respect to the local coordinate system whose axis is directed from port R to port C. The terms “positive” and “negative” in the variable descriptions refer to this coordinate system, and the gap in negative direction must be specified with negative value.

If the local coordinate system is not aligned with the globally assigned positive direction, the gaps interchange their values with respective sign adjustment.

The block is oriented from R to C. This means that the block transmits force from port R to port C when the gap in positive direction is cleared up.

## Dialog Box and Parameters

**Block Parameters: Translational Hard Stop**

Translational Hard Stop

The block represents a double-sided mechanical translational hard stop that restricts motion of a body between upper and lower bounds. The stop is implemented as a spring that comes into contact with the slider as the gap is cleared. To account for energy dissipation and non-elastic effects, the damping is introduced as the block parameter, thus making it possible to account for energy loss.

Connections R and S are mechanical translational conserving ports. The block is oriented from R to C. This means that the block transmits force from port R to port C when the gap in positive direction is cleared up.

Parameters

Upper bound:	<input type="text" value="0.1"/>	<input type="text" value="m"/>
Lower bound:	<input type="text" value="-0.1"/>	<input type="text" value="m"/>
Contact stiffness at upper bound:	<input type="text" value="1000000"/>	<input type="text" value="N/m"/>
Contact stiffness at lower bound:	<input type="text" value="1000000"/>	<input type="text" value="N/m"/>
Contact damping at upper bound:	<input type="text" value="150"/>	<input type="text" value="s*N/m"/>
Contact damping at lower bound:	<input type="text" value="150"/>	<input type="text" value="s*N/m"/>

OK Cancel Help Apply

### Upper bound

Gap between the slider and the upper bound. The direction is specified with respect to the local coordinate system, with the slider located in the origin. A positive value of the parameter

# Translational Hard Stop

---

specifies the gap between the slider and the upper bound. A negative value sets the slider as penetrating into the upper bound. The default value is 0.005 m.

## **Lower bound**

Gap between the slider and the lower bound. The direction is specified with respect to the local coordinate system, with the slider located in the origin. A negative value of the parameter specifies the gap between the slider and the lower bound. A positive value sets the slider as penetrating into the lower bound. The default value is -0.005 m.

## **Contact stiffness at upper bound**

The parameter specifies the elastic property of colliding bodies when the slider hits the upper bound. 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 10e6 N/m.

## **Contact stiffness at lower bound**

The parameter specifies the elastic property of colliding bodies when the slider hits the lower bound. 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 10e6 N/m.

## **Contact damping at upper bound**

The parameter specifies dissipating property of colliding bodies when the slider hits the upper bound. At zero damping, the impact is close to an absolutely elastic one. The greater the value of the parameter, the more energy dissipates during an interaction. Keep in mind that damping affects slider motion as long as the slider is in contact with the stop, including the period when slider is pulled back from the contact. For computational efficiency and convergence reasons, The MathWorks recommends that you assign a nonzero value to this parameter. The default value is 150 N\*s/m.



## Contact damping at lower bound

The parameter specifies dissipating property of colliding bodies when the slider hits the lower bound. At zero damping, the impact is close to an absolutely elastic one. The greater the value of the parameter, the more energy dissipates during an interaction. Keep in mind that damping affects slider motion as long as the slider is in contact with the stop, including the period when slider is pulled back from the contact. For computational efficiency and convergence reasons, The MathWorks recommends that you assign a nonzero value to this parameter. The default value is 150 N\*s/m.

## Ports

The block has the following ports:

R

Mechanical translational conserving port associated with the slider that travels between stops installed on the case.

C

Mechanical translational conserving port associated with the case.

## Examples

The Mechanical System with Translational Hard Stop demo (`sh_mechanical_system_translational_hardstop`) illustrates the use of the Translational Hard Stop block in mechanical systems. Two masses are interacting through a hard stop. The mass on the left is driven by an ideal velocity source. Plotting the displacement of the second mass against the displacement of the first mass produces a typical hysteresis curve.

## See Also

Translational Damper

Translational Friction

Translational Spring

# Translational Hydro-Mechanical Converter

---

**Purpose** Simulate single chamber of hydraulic cylinder as building block for various cylinder models

**Library** Hydraulic Elements

**Description** The Translational Hydro-Mechanical Converter block models an ideal transducer that converts hydraulic energy into mechanical energy in the form of translational motion of the converter output member. Using this block as a basic element, you can build a large variety of hydraulic cylinder models by adding application-specific effects, such as fluid compressibility, leakage, friction, hard stops, and so on.



The converter is simulated according to the following equations:

$$q = A(v_R - v_C) \cdot or$$

$$F = A \cdot p \cdot or$$

where

- |       |                                                                                                                                                                                                                                                       |
|-------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| $q$   | Flow rate due to fluid compressibility                                                                                                                                                                                                                |
| $A$   | Effective piston area                                                                                                                                                                                                                                 |
| $v_R$ | Converter rod velocity                                                                                                                                                                                                                                |
| $v_C$ | Converter body velocity                                                                                                                                                                                                                               |
| $F$   | Force developed by the converter                                                                                                                                                                                                                      |
| $p$   | Gauge pressure of fluid in the converter chamber                                                                                                                                                                                                      |
| $or$  | Converter orientation with respect to the globally assigned positive direction. If pressure applied at port A exerts force in positive direction, $or$ equals 1. If pressure applied at port A exerts force in negative direction, $or$ equals $-1$ . |

Port A is a hydraulic conserving port associated with the converter inlet. Ports R and C are translational mechanical conserving ports associated with the rod and the body of the converter, respectively.

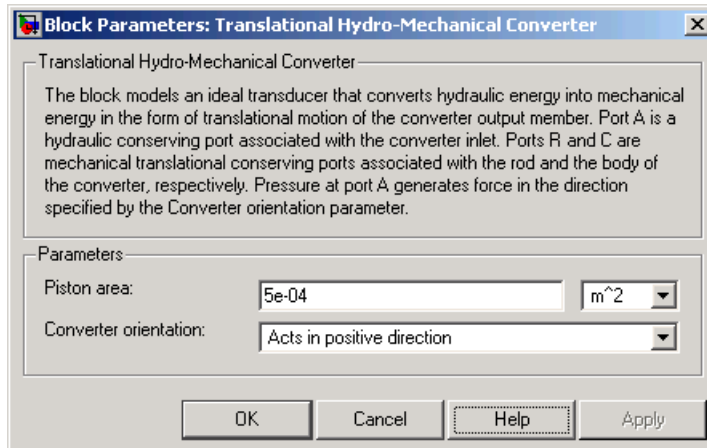
# Translational Hydro-Mechanical Converter

## Basic Assumptions and Limitations

The model is based on the following assumption:

- The block simulates an ideal converter, with only the transduction property considered. No inertia, friction, leakage, or other effects are taken into account.

## Dialog Box and Parameters



### Piston area

Effective piston area. The default value is  $5e-4 \text{ m}^2$ .

### Converter orientation

Specifies converter orientation with respect to the globally assigned positive direction. The converter 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.

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the converter inlet.

# Translational Hydro-Mechanical Converter

---

R  
Mechanical translational conserving port associated with the rod of the converter.

C  
Mechanical translational conserving port associated with the body of the converter.

## Examples

The Closed-Loop Circuit with 4-Way Valve and Custom Cylinder demo (`sh_closed_loop_circuit_4_way_valve_cust_cyl`) and the Hydraulic Flapper-Nozzle Amplifier demo (`sh_hydraulic_flapper_nozzle_amplifier`) illustrate the use of the Translational Hydro-Mechanical Converter block in hydraulic systems.

## See Also

Variable Volume Chamber

## Purpose

Simulate ideal spring in mechanical translational systems

## Library

Mechanical Translational Elements

## Description



The Translational Spring block represents an ideal mechanical linear spring, described with the following equations:

$$F = Kx$$

$$x = x_{init} + x_R - x_C$$

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

where

$F$  Force transmitted through the spring

$K$  Spring rate

$x$  Relative displacement (spring deformation)

$x_{init}$  Spring initial displacement (spring offset)

$x_R, x_C$  Absolute displacements of terminals R and C, respectively

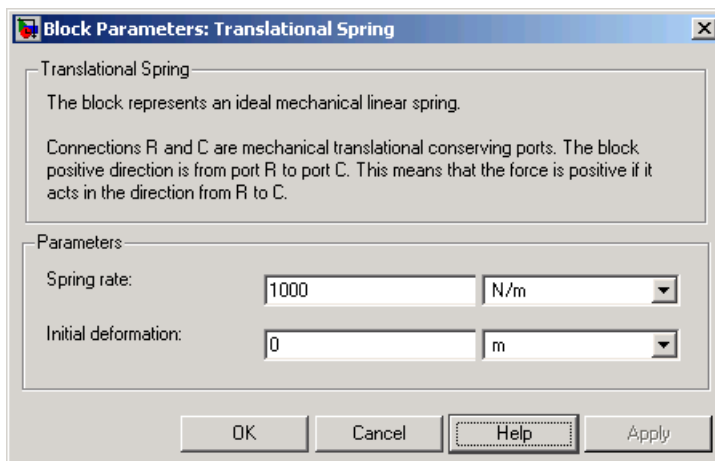
$v$  Relative velocity

$t$  Time

The block positive direction is from port R to port C. This means that the force is positive if it acts in the direction from R to C.

# Translational Spring

## Dialog Box and Parameters



### Spring rate

Spring rate. The default value is 1000 N/m.

### Initial deformation

Spring initial deformation, or offset, in length units. The deformation is determined as  $x = x_{init} + x_R - x_C$ , where  $x_{init}$  is the initial deformation, and  $x_R$ ,  $x_C$  are the absolute displacements of terminals R and C in the globally assigned positive direction. The spring can be initially compressed ( $x_{init} > 0$ ) or stretched ( $x_{init} < 0$ ). This parameter specifies the initial condition for use in computing the block's initial state at the beginning of a simulation run. For more information, see "Computing Initial Conditions". The default value is 0.

### Ports

The block has the following ports:

- R  
Mechanical translational conserving port.
- C  
Mechanical translational conserving port.

**See Also**

Translational Damper

Translational Friction

Translational Hard Stop

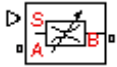
# Variable Orifice

---

**Purpose** Simulate generic hydraulic variable orifice

**Library** Hydraulic Elements

**Description** The block represents a variable orifice of any type as a data-sheet-based model. Depending on data listed in the manufacturer's catalogs or data sheets for your particular orifice, you can choose one of the following model parameterization options:



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

In the first case, the passage area is assumed to be linearly dependent on the control member displacement, that is, the orifice is assumed to be closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. In the second case, the passage area is determined by one-dimensional interpolation from the table  $A=A(h)$ . In both cases, a small leakage area is assumed to exist even after the orifice is completely closed. Physically, it represents a possible clearance in the closed valve, but the main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation.

In the first and second cases, the model accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number ( $Re$ ) and comparing its value with the critical Reynolds number ( $Re_{cr}$ ). After the



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

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

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

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

$$p = p_A - p_B$$

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

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

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

where

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

# Variable Orifice

---

$A_{max}$	Orifice maximum area
$h_{max}$	Control member maximum displacement
$x_0$	Initial opening
$x$	Control member displacement from initial position
$h$	Orifice opening
$or$	Orifice orientation indicator. The variable assumes +1 value if the control member displacement in the globally assigned positive direction opens the orifice, and -1 if positive motion decreases the opening.
$\rho$	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 monotonically increasing. The vertices can be non-uniformly spaced. You have a choice of three interpolation methods and two extrapolation methods.

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

port S opens or closes the orifice depending on the value of the orifice orientation indicator.

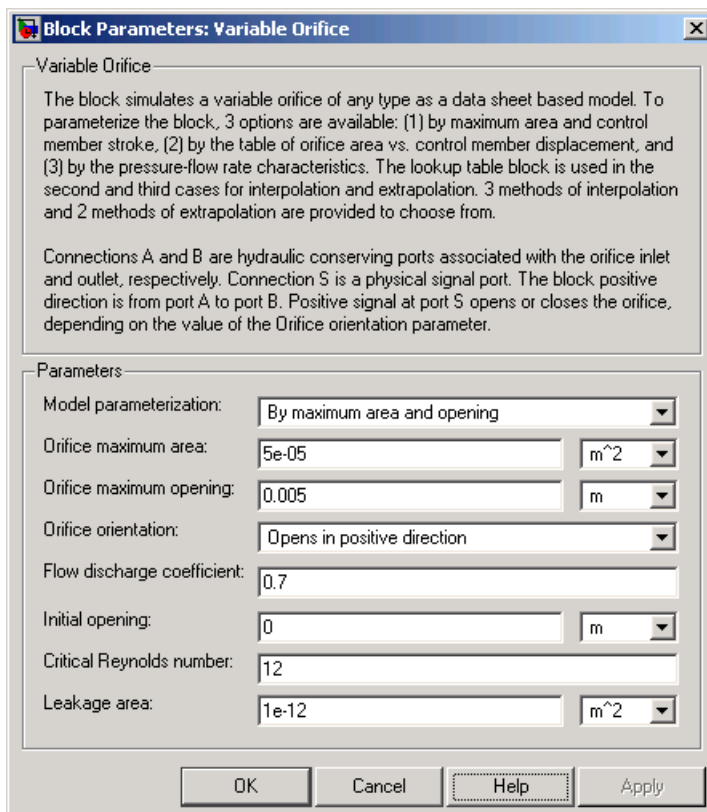
## Basic Assumptions and Limitations

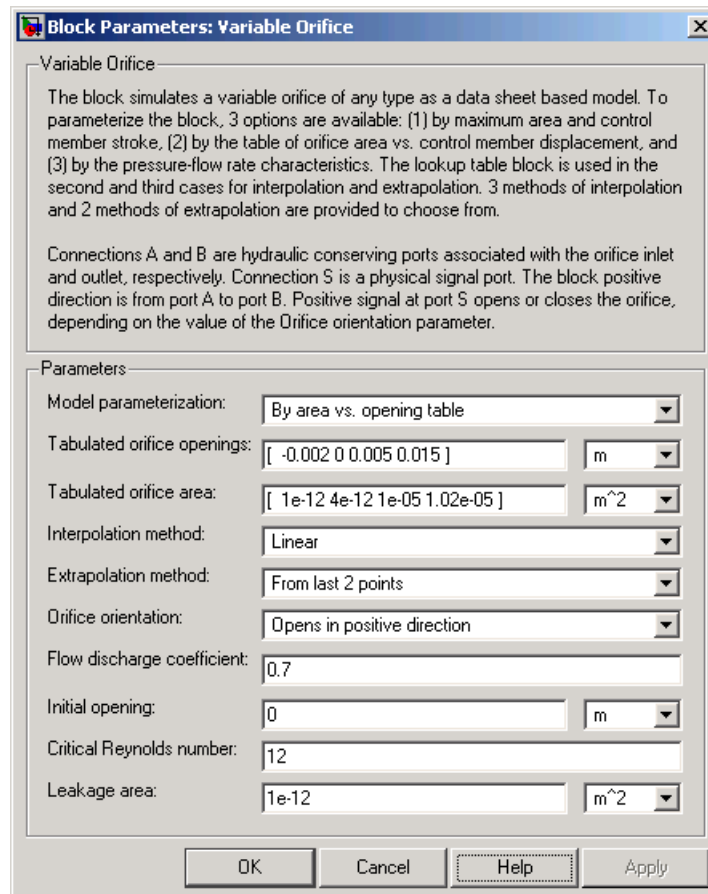
The model is based on the following assumptions:

- Fluid inertia is not taken into account.
- For orifices specified by the passage area (the first two parameterization options), the transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at  $Re=Re_{cr}$ .
- For orifices specified by pressure-flow characteristics (the third parameterization option), the model does not explicitly account for the flow regime or leakage flow rate, because the tabulated data is assumed to account for these characteristics.

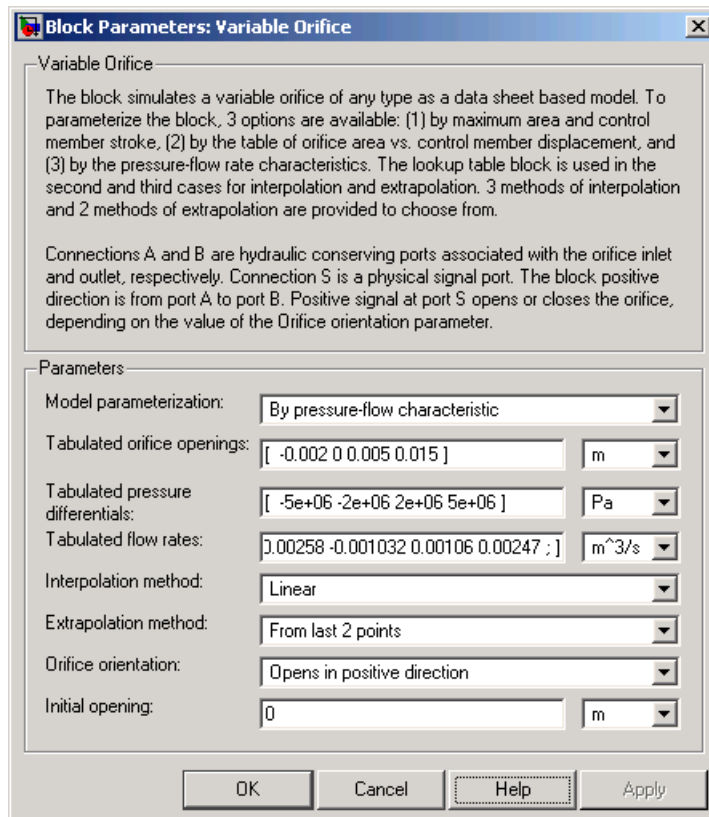
# Variable Orifice

## Dialog Box and Parameters





# Variable Orifice



## Model parameterization

Select one of the following methods for specifying the orifice:

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

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

### **Orifice maximum area**

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

### **Orifice maximum opening**

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

### **Tabulated orifice openings**

Specify the vector of input values for orifice openings as a tabulated 1-by-m array. The input values vector must be strictly monotonically increasing. The values can be non-uniformly spaced. You must provide at least three values. The default values, in meters, are  $[-2e-3, 0, 5e-3, 15e-3]$ . If **Model parameterization** is set to By area vs. opening table, the **Tabulated orifice openings** values will be used together with **Tabulated orifice area** values for one-dimensional table lookup. If **Model parameterization** is set to By pressure-flow characteristic, the **Tabulated orifice openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

# Variable Orifice

---

## Tabulated orifice area

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

## Tabulated pressure differentials

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

## Tabulated flow rates

Specify the output values for flow rates as a tabulated m-by-n matrix, defining the function values at the input grid vertices. Each value in the matrix specifies flow rate taking place at a specific combination of orifice opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in  $m^3/s$ , are:

```
[ -1e-12,  -0.6e-13,   0.6e13,    0.1e12; ...  
  -0.8e-7, -0.4e-7,   0.4e7,     0.8e7; ...  
  -2.5e-3, -1.03e-3,  1.e-3,     2.4e-3; ...  
  -2.58e-3, -1.032e-3, 1.06e-3,  2.47e-3]
```

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

## Interpolation method

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



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

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

### **Extrapolation method**

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

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

# Variable Orifice

---

specified input argument, and the first specified output value for all input values less than the first specified input argument.

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

## **Orifice orientation**

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

## **Flow discharge coefficient**

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

## **Initial opening**

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

## **Critical Reynolds number**

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on orifice geometrical profile, and the recommendations on the parameter value can be found in hydraulic textbooks. The default value is 12.

## Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is  $1e-12 \text{ m}^2$ .

## Global Parameters

### Fluid density

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

### Fluid kinematic viscosity

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

## Ports

The block has the following ports:

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

The flow rate is positive if fluid flows from port A to port B. Positive signal at the physical signal port S opens or closes the orifice depending on the value of the parameter **Orifice orientation**.

## Examples

The Hydraulic Flapper-Nozzle Amplifier demo (`sh_hydraulic_flapper_nozzle_amplifier`) illustrates the use of the Variable Orifice block in hydraulic systems.

# Variable Orifice

---

## **See Also**

Annular Orifice

Fixed Orifice

Orifice with Variable Area Round Holes

Orifice with Variable Area Slot

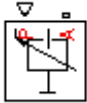
PS Lookup Table (1D)

PS Lookup Table (2D)

**Purpose** Simulate hydraulic capacity of variable volume with compressible fluid

**Library** Hydraulic Elements

**Description**



The Variable Volume Chamber block models fluid compressibility in variable volume chambers, such as hydraulic cylinder cavities. The fluid is considered to be a mixture of liquid and a small amount of entrained, nondissolved gas. Use this block together with the Translational Hydro-Mechanical Converter block.

---

**Note** The Variable Volume Chamber block takes into account only the flow rate caused by fluid compressibility. The fluid volume consumed to create piston velocity is accounted for in the Translational Hydro-Mechanical Converter block.

---

The chamber is simulated according to the following equations:

$$q = \frac{V_0 + A \cdot x}{E} \cdot \frac{dp}{dt}$$

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

where

- $q$  Flow rate due to fluid compressibility
- $V_0$  Initial volume of fluid in the chamber
- $A$  Effective piston area
- $x$  Piston displacement from initial position

# Variable Volume Chamber

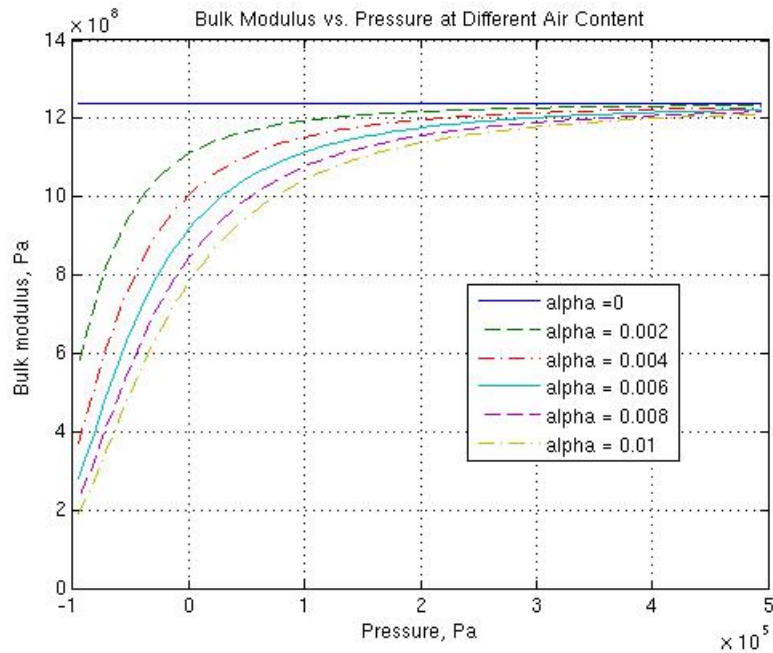
---

$or$	Chamber orientation with respect to the globally assigned positive direction. If displacement in positive direction increases the volume of the chamber, $or$ equals 1. If displacement in positive direction decreases the volume of the chamber, $or$ equals $-1$ .
$E$	Fluid bulk modulus
$E_l$	Pure liquid bulk modulus
$p$	Gauge pressure of fluid in the chamber
$p_a$	Atmospheric pressure
$\alpha$	Relative gas content at atmospheric pressure, $\alpha = V_g/V_L$
$V_g$	Gas volume at atmospheric pressure
$V_L$	Volume of liquid
$n$	Gas-specific heat ratio

The main objective of representing fluid as a mixture of liquid and gas is to introduce an approximate model of cavitation, which takes place in a chamber if pressure drops below fluid vapor saturation level. As it is seen in the graph below, the bulk modulus of a mixture decreases at  $p \rightarrow p_a$ , thus considerably slowing down further pressure change.

At high pressure,  $p \gg p_a$ , a small amount of nondissolved gas has practically no effect on the system behavior.

# Variable Volume Chamber



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

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

Port A is a hydraulic conserving port associated with the chamber inlet. Port P is a physical signal port that controls piston displacement.

The block positive direction is from port A to the reference point. This means that the flow rate is positive if it flows into the chamber.

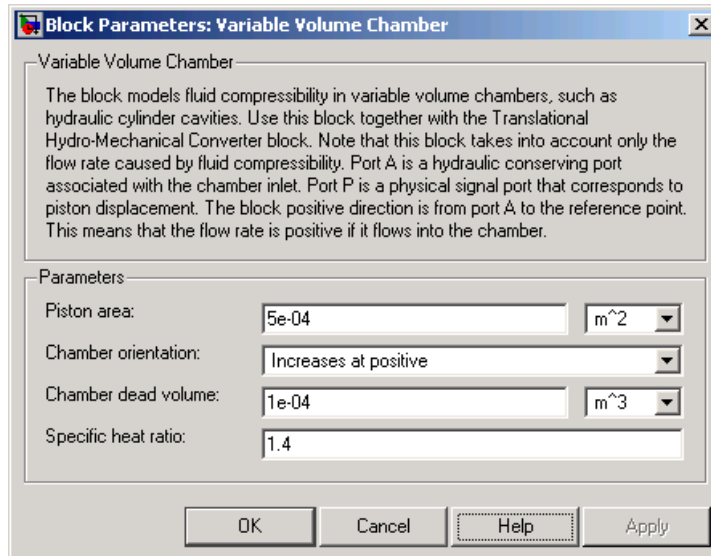
# Variable Volume Chamber

## Basic Assumptions and Limitations

The model is based on the following assumptions:

- Fluid density remains constant.
- Fluid fills the entire chamber volume.

## Dialog Box and Parameters



### Piston area

Effective piston area. The default value is  $5e-4 \text{ m}^2$ .

### Chamber orientation

Specifies chamber orientation with respect to the globally assigned positive direction. The chamber can be installed in two different ways, depending upon whether the piston motion in the positive direction increases or decreases the volume of the chamber. If piston motion in the positive direction decreases the chamber volume, set the parameter to *Decreases at positive*. The default value is *Increases at positive*.



## Chamber dead volume

Volume of fluid in the chamber at initial piston position. The default value is  $1e-4 \text{ m}^3$ .

## Specific heat ratio

Gas-specific heat ratio. The default value is 1.4.

## Global Parameters

### Fluid bulk modulus

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

### Nondissolved gas ratio

Nondissolved gas relative content determined as a ratio of gas volume to the liquid volume. The parameter is determined by the type of working fluid selected for the system under design. Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

## Ports

The block has the following ports:

A

Hydraulic conserving port associated with the chamber inlet.

P

Physical signal port that controls piston displacement.

## Examples

The Variable Volume Chamber Test Rig demo (`sh_variable_chamber_test_rig`) is specifically designed to demonstrate the Variable Volume Chamber block behavior at different regimes. Each chamber is installed between two equal fixed-area orifices. The top chamber is set to increase its volume with the piston motion in positive direction, while the bottom chamber volume decreases at the same time.

## See Also

Constant Volume Chamber

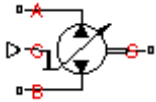
Translational Hydro-Mechanical Converter

# Variable-Displacement Motor

**Purpose** Simulate variable-displacement reversible hydraulic motor

**Library** Pumps and Motors

## Description



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

Two block parameterization options are available:

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

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

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

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

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

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

# Variable-Displacement Motor

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

$$p = p_A - p_B$$

where

$q$	Motor flow rate
$p$	Pressure differential across the motor
$p_A, p_B$	Gauge pressures at the block terminals
$D$	Motor instantaneous displacement
$D_{max}$	Motor maximum displacement
$x$	Control member displacement
$x_{max}$	Control member maximum stroke
$T$	Torque at the motor output shaft
$\omega$	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

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

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

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

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

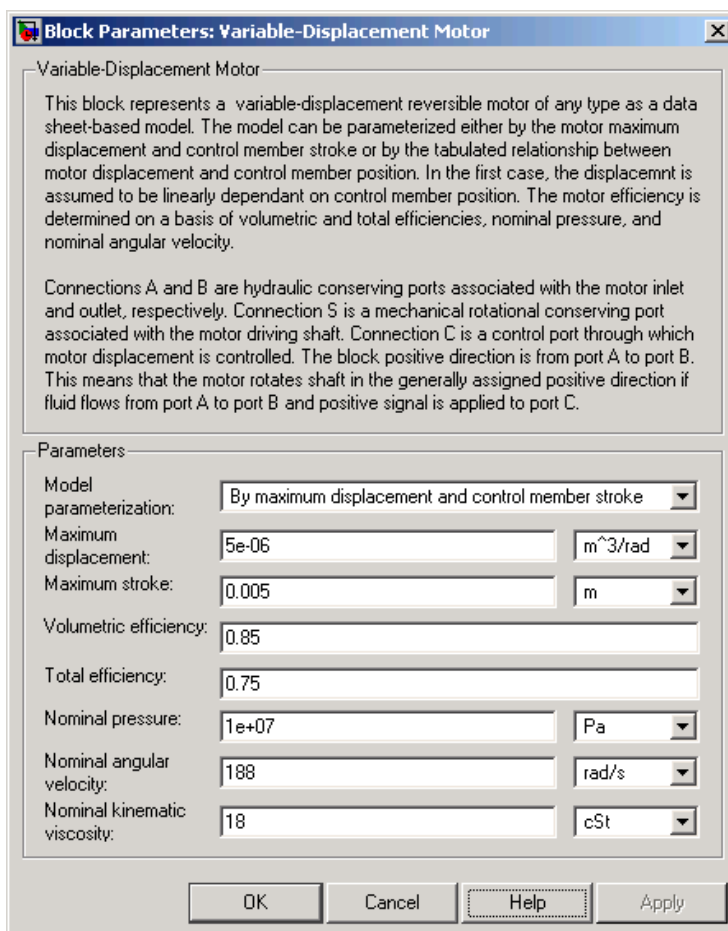
## **Basic Assumptions and Limitations**

The model is based on the following assumptions:

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

# Variable-Displacement Motor

## Dialog Box and Parameters



**Block Parameters: Variable-Displacement Motor**

Variable-Displacement Motor

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

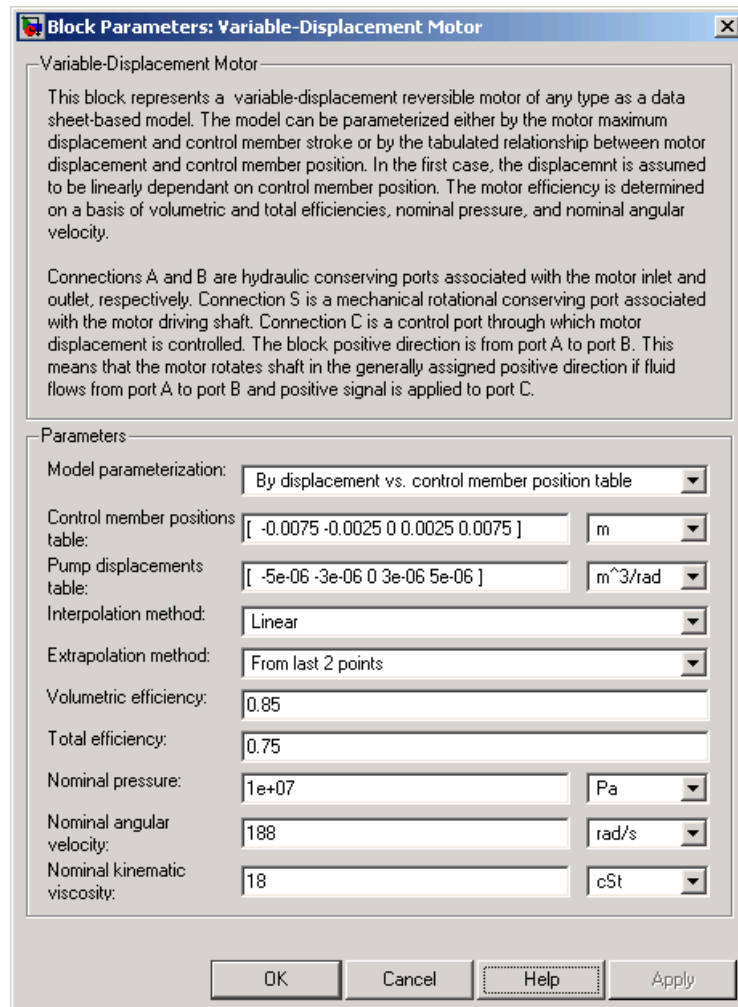
Connections A and B are hydraulic conserving ports associated with the motor inlet and outlet, respectively. Connection S is a mechanical rotational conserving port associated with the motor driving shaft. Connection C is a control port through which motor displacement is controlled. The block positive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

Parameters

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

OK Cancel Help Apply

# Variable-Displacement Motor



**Block Parameters: Variable-Displacement Motor**

Variable-Displacement Motor

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

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

Parameters

Model parameterization: By displacement vs. control member position table

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

Pump displacements table: [ -5e-06 -3e-06 0 3e-06 5e-06 ] m<sup>3</sup>/rad

Interpolation method: Linear

Extrapolation method: From last 2 points

Volumetric efficiency: 0.85

Total efficiency: 0.75

Nominal pressure: 1e+07 Pa

Nominal angular velocity: 188 rad/s

Nominal kinematic viscosity: 18 cSt

OK Cancel Help Apply

## Model parameterization

Select one of the following block parameterization options:

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

# Variable-Displacement Motor

---

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. This parameter is used if **Model parameterization** is set to By maximum displacement and control member stroke.

## Maximum stroke

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

## Control member positions table

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

## Pump displacements table

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



## Interpolation method

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

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

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to By displacement vs. control member position table.

## Extrapolation method

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

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

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

# Variable-Displacement Motor

---

## **Volumetric efficiency**

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

## **Total efficiency**

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

## **Nominal pressure**

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

## **Nominal angular velocity**

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

## **Nominal kinematic viscosity**

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

## **Global Parameters**

### **Fluid kinematic viscosity**

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

## **Ports**

The block has the following ports:

- A Hydraulic conserving port associated with the motor inlet.
- B Hydraulic conserving port associated with the motor outlet.
- C Physical signal port that controls motor displacement.

S

Mechanical rotational conserving port associated with the motor output shaft.

**See Also**

Hydraulic Motor

# Variable-Displacement Pressure-Compensated Pump

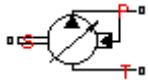
## Purpose

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

## Library

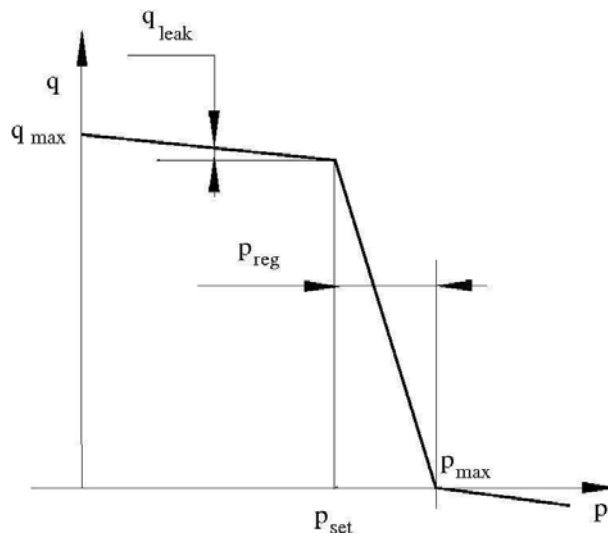
Pumps and Motors

## Description



The Variable-Displacement Pressure-Compensated Pump block represents a positive, variable-displacement, pressure-compensated pump of any type as a data-sheet-based model. The key parameters required to parameterize the block are the pump maximum displacement, regulation range, volumetric and total efficiencies, nominal pressure, and angular velocity. All these parameters are generally provided in the data sheets or catalogs.

The following figure shows the delivery-pressure characteristic of the pump.



The pump tries to maintain preset pressure at its outlet by adjusting its delivery flow in accordance with the system requirements. If pressure differential across the pump is less than the setting pressure, the pump outputs its maximum delivery corrected for internal leakage. After

# Variable-Displacement Pressure-Compensated Pump

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

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

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

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

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

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

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

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

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

$$p = p_P - p_T$$

where

# Variable-Displacement Pressure-Compensated Pump

---

$q$	Pump delivery
$p$	Pressure differential across the pump
$p_p, p_T$	Gauge pressures at the block terminals
$D$	Pump instantaneous displacement
$D_{max}$	Pump maximum displacement
$p_{set}$	Pump setting pressure
$p_{max}$	Maximum pressure, at which the pump displacement is zero
$T$	Torque at the pump driving shaft
$\omega$	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,  $\mu = \nu \cdot \rho$

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

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

which provides the formula to determine the Hagen-Poiseuille coefficient

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

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

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

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

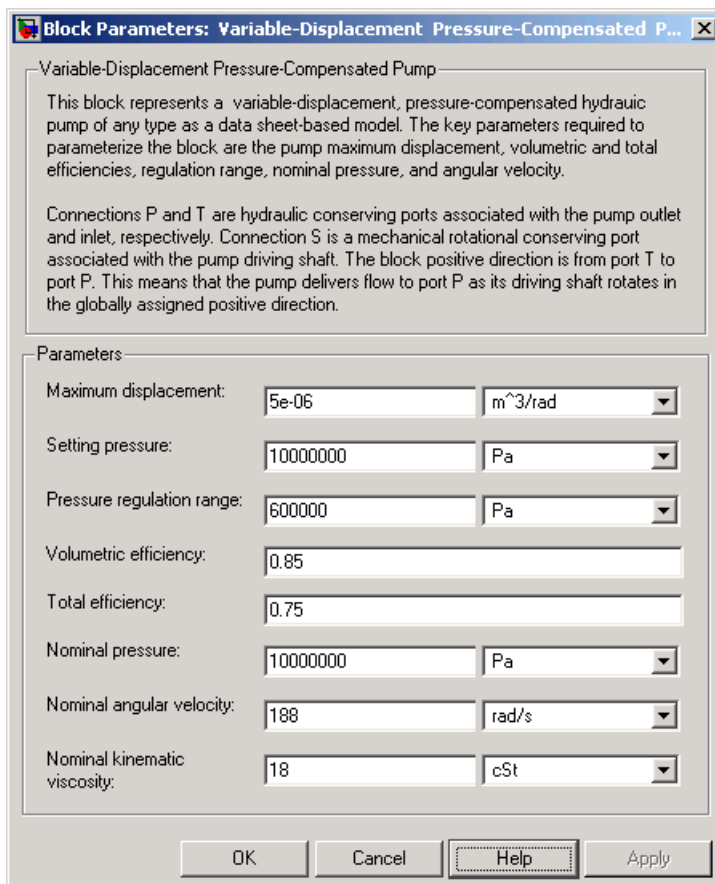
## Basic Assumptions and Limitations

The model is based on the following assumptions:

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

# Variable-Displacement Pressure-Compensated Pump

## Dialog Box and Parameters



### Maximum displacement

Pump displacement. The default value is  $5e-6$  m<sup>3</sup>/rad.

### Setting pressure

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

### Pressure regulation range

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



# Variable-Displacement Pressure-Compensated Pump

---

## **Volumetric efficiency**

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

## **Total efficiency**

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

## **Nominal pressure**

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

## **Nominal angular velocity**

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

## **Nominal kinematic viscosity**

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

## **Global Parameters**

### **Fluid kinematic viscosity**

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

## **Ports**

The block has the following ports:

T

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

P

Hydraulic conserving port associated with the pump outlet.

S

Mechanical rotational conserving port associated with the pump driving shaft.

# Variable-Displacement Pressure-Compensated Pump

---

## **Examples**

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

## **See Also**

Fixed-Displacement Pump

Variable-Displacement Pump

## Purpose

Simulate variable-displacement reversible hydraulic pump

## Library

Pumps and Motors

## Description



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

Two block parameterization options are available:

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

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

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

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

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

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

# Variable-Displacement Pump

---

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

$$p = p_P - p_T$$

where

$q$	Pump delivery
$p$	Pressure differential across the pump
$p_P, p_T$	Gauge pressures at the block terminals
$D$	Pump instantaneous displacement
$D_{max}$	Pump maximum displacement
$x$	Control member displacement
$x_{max}$	Control member maximum stroke
$T$	Torque at the pump driving shaft
$\omega$	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  $\nu = \nu_{nom}$  can be determined from the catalog data

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

which provides the formula to determine the Hagen-Poiseuille coefficient

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

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

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

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

# Variable-Displacement Pump

## Basic Assumptions and Limitations

The model is based on the following assumptions:

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

## Dialog Box and Parameters

**Block Parameters: Variable-Displacement Pump**

Variable-Displacement Pump

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

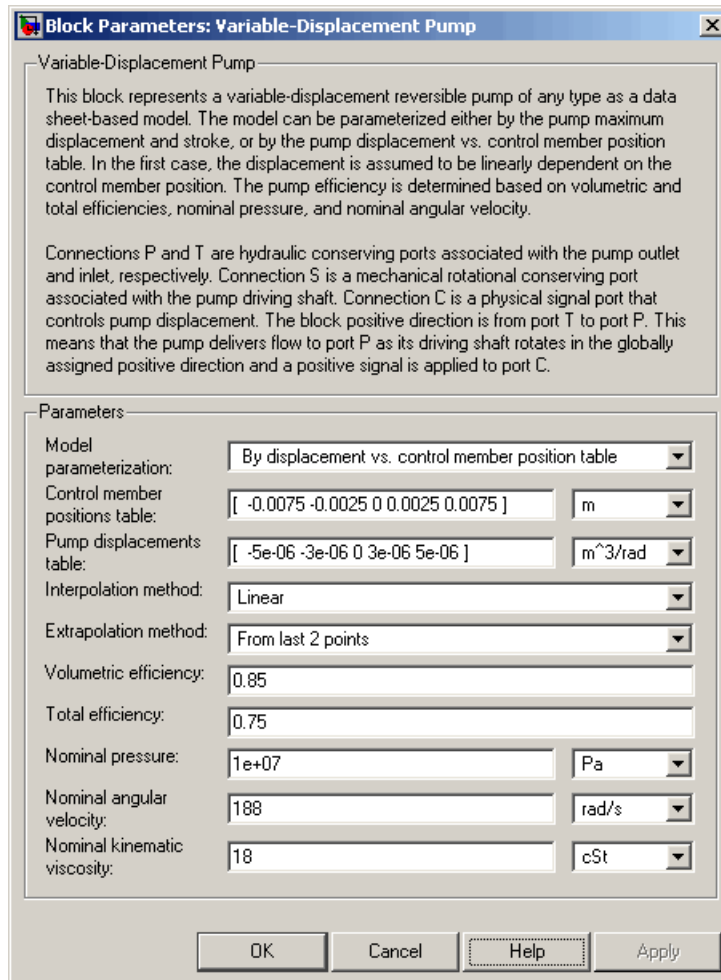
Connections P and T are hydraulic conserving ports associated with the pump outlet and inlet, respectively. Connection S is a mechanical rotational conserving port associated with the pump driving shaft. Connection C is a physical signal port that controls pump displacement. The block positive direction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

Parameters

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

OK Cancel Help Apply

# Variable-Displacement Pump



**Block Parameters: Variable-Displacement Pump**

Variable-Displacement Pump

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

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

Parameters

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

OK Cancel Help Apply

## 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

# Variable-Displacement Pump

---

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. This parameter is used if **Model parameterization** is set to By maximum displacement and control member stroke.

## Maximum stroke

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

## Control member positions table

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

## Pump displacements table

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



## Interpolation method

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

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

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to By displacement vs. control member position table.

## Extrapolation method

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

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

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

# Variable-Displacement Pump

---

## **Volumetric efficiency**

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

## **Total efficiency**

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

## **Nominal pressure**

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

## **Nominal angular velocity**

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

## **Nominal kinematic viscosity**

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

## **Global Parameters**

### **Fluid kinematic viscosity**

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

## **Ports**

The block has the following ports:

T

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

P

Hydraulic conserving port associated with the pump outlet.

C

Physical signal port that controls pump displacement.

S

Mechanical rotational conserving port associated with the pump driving shaft.

## **See Also**

Fixed-Displacement Pump

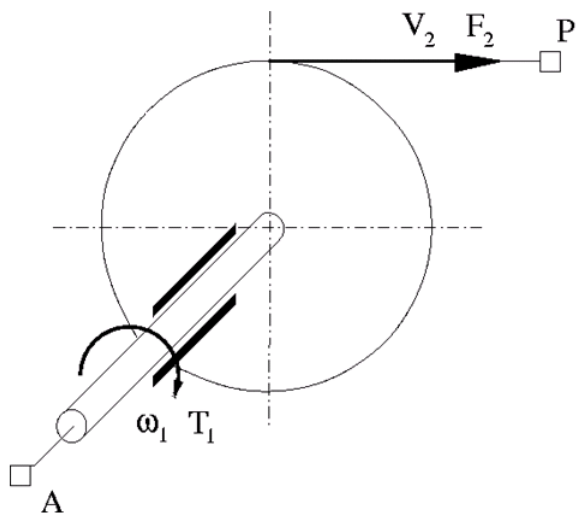
Variable-Displacement Pressure-Compensated Pump

# Wheel and Axle

**Purpose** Simulate wheel and axle mechanism in mechanical systems

**Library** Mechanisms

**Description** The Wheel and Axle block represents a wheel and axle mechanism shown in the following schematic.



The wheel and the axle have the same axis, and the axis is assumed to be rigidly connected to the frame, thus making this mechanism an ideal converter of mechanical rotational into mechanical translational motion. The mechanism has two connections: a mechanical rotational port A, which corresponds to the axle, and a mechanical translational port P, which corresponds to the wheel periphery. The mechanism is described with the following equations:

$$T = r \cdot F \cdot \cos \theta$$

$$v = r \cdot \omega \cdot \cos \theta$$

where

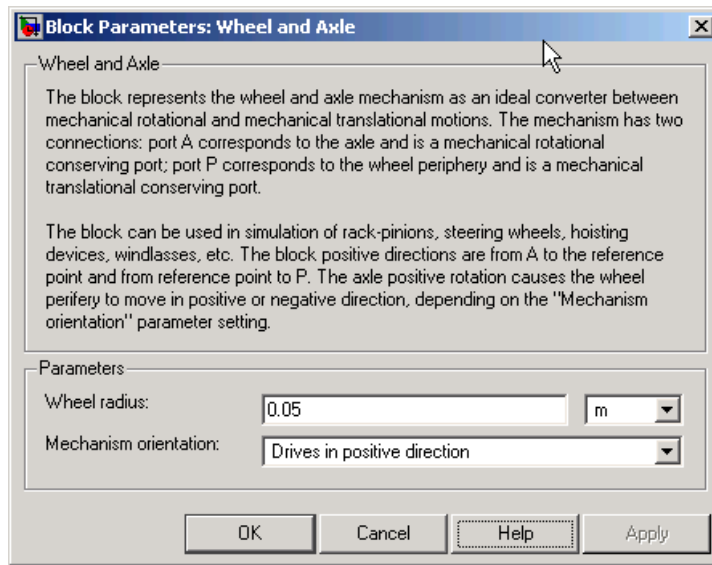
$T$	Torque on the axle
$F$	Force on the wheel periphery
$\omega$	Angular velocity
$v$	Linear velocity on the wheel periphery
$r$	Wheel radius
$or$	Mechanism orientation indicator. The variable assumes +1 value if axle rotation in the globally assigned positive direction is converted into translational motion in positive direction, and -1 if positive rotation results in translational motion in negative direction.

The block can be used in simulation of rack-pinions, steering wheels, hoisting devices, windlasses, and so on.

The block positive directions are from A to the reference point and from the reference point to P.

# Wheel and Axle

## Dialog Box and Parameters



### Wheel radius

Radius of the wheel. The default value is 0.05 m.

### Mechanism orientation

The parameter can be set to one of two options: `Drives in positive direction` or `Drives in negative direction`. The value `Drives in positive direction` specifies a mechanism where axle rotation in the globally assigned positive direction is converted into translational motion in positive direction. The value `Drives in negative direction` specifies a mechanism where axle rotation in the globally assigned positive direction is converted into translational motion in negative direction. The default value is `Drives in positive direction`.

### Ports

The block has the following ports:

A

Mechanical rotational conserving port associated with the axle.

P

Mechanical translational conserving port associated with the wheel periphery.

## Examples

The Simple Mechanical System demo (`sh_simple_mechanical_system`) illustrates the use of the Wheel and Axle block in mechanical systems.

## See Also

Gear Box





# Commands — Alphabetical List

---

`pm_adddimension`

Add new dimension to unit registry

`pm_addunit`

Add new unit to unit registry

`pm_getunits`

Get information about all units in unit registry

# pm\_adddimension

---

**Purpose** Add new dimension to unit registry

**Syntax** `pm_adddimension(dimension, unitname)`

**Description** `pm_adddimension(dimension, unitname)` adds a new dimension named `dimension` with a fundamental unit, `unitname`. `dimension` may be any string. `unitname` must be a valid unit name, that is, it must begin with a letter and contain only letters and numbers.

**Examples** Add a new unit dimension, `length`, with a fundamental unit of meter, `m`:

```
pm_adddimension('length', 'm');
```

**See Also** `pm_addunit`, `pm_getunits`

**Purpose** Add new unit to unit registry

**Syntax** `pm_addunit(unitname, conversion, unitexpression)`

**Description** `pm_addunit(unitname, conversion, unitexpression)` introduces a new unit, `unitname`, defined as `conversion * unitexpression`. `conversion` must be a positive real scalar, and `unitexpression` must be a valid unit expression in terms of units already defined in the unit registry.

The new unit name, `unitname`, must be a valid unit name, that is, it must begin with a letter and contain only letters and numbers.

The following operators are supported in the unit mathematical expressions:

- \* Multiplication
- / Division
- ^ Power
- +, - Plus, minus — for exponents only
- () Brackets to specify evaluation order

**Examples** Add a new unit centimeter, `cm`, in terms of meter, `m`:

```
pm_addunit('cm', 0.01, 'm');
```

Add a new unit newton, `N`, in terms of kilograms, meters, and seconds:

```
pm_addunit('N', 1, 'kg*m/s^2');
```

**See Also** `pm_adddimension`, `pm_getunits`

# pm\_getunits

---

**Purpose** Get information about all units in unit registry

**Syntax** [units, conversions, expressions] = pm\_getunits

**Description** [units, conversions, expressions] = pm\_getunits returns all units in the registry in a cell array, units. Their corresponding conversions and base expressions are returned in conversions and expressions, respectively. For fundamental units, the conversion is 1.0 and the base expression is the unit itself.

**Examples** List all units currently defined in the registry:

```
pm_getunits
```

```
ans =
```

```
'm'  
'kg'  
's'  
'c'  
'K'  
'rad'  
'cm'  
'mm'  
'km'  
'in'  
'ft'  
'mi'  
'yd'  
'l'  
'gal'  
'g'  
'mg'  
'lbm'  
'oz'  
'slug'  
'N'
```

'lbf'  
'dyn'  
'lb'  
'min'  
'hr'  
'deg'  
'rev'  
'mph'  
'fpm'  
'fps'  
'rpm'  
'J'  
'Btu'  
'W'  
'HP'  
'V'  
'A'  
'F'  
'H'  
'Ohm'  
'Pa'  
'bar'  
'psi'  
'atm'  
'lpm'  
'gpm'  
'Poise'  
'cP'  
'reyn'  
'St'  
'cSt'  
'Newt'

**See Also** `pm_adddimension`, `pm_addunit`



2-Position Valve Actuator block 2-2  
2-Way Directional Valve block 2-8  
3-Position Valve Actuator block 2-18  
3-Way Directional Valve block 2-23  
4-Way Directional Valve block 2-34

## A

accumulators  
    gas-charged 2-105  
    spring-loaded 2-317  
Annular Orifice block 2-47

## B

Ball Valve block 2-51

## C

Check Valve block 2-57  
commands  
    pm\_adddimension 3-2  
    pm\_addunit 3-3  
    pm\_getunits 3-4  
Connection Port block 2-63  
Constant Volume Chamber block 2-65  
Custom Hydraulic Fluid block 2-74  
Cylinder Friction block 2-76

## D

Double-Acting Hydraulic Cylinder block 2-82

## E

Elbow block 2-88

## F

Fixed Orifice block 2-93  
Fixed-Displacement Pump block 2-97

Fluid Inertia block 2-103

## G

Gas-Charged Accumulator block 2-105  
Gear Box block 2-109  
Gradual Area Change block 2-111

## H

Hydraulic Fluid block 2-118  
Hydraulic Motor block 2-121  
Hydraulic Pipeline block 2-126  
Hydraulic Reference block 2-133

## I

Ideal Angular Velocity Source block 2-134  
Ideal Force Sensor block 2-136  
Ideal Force Source block 2-138  
Ideal Hydraulic Flow Rate Sensor block 2-140  
Ideal Hydraulic Flow Rate Source block 2-142  
Ideal Hydraulic Pressure Sensor block 2-144  
Ideal Hydraulic Pressure Source block 2-146  
Ideal Rotational Motion Sensor block 2-148  
Ideal Torque Sensor block 2-150  
Ideal Torque Source block 2-152  
Ideal Translational Motion Sensor block 2-154  
Ideal Translational Velocity Source  
    block 2-156  
Inertia block 2-158

## L

Lever block 2-160  
Linear Hydraulic Resistance block 2-165  
Local Resistance block 2-167

## M

Mass block 2-174

Mechanical Rotational Reference block 2-176  
Mechanical Translational Reference  
block 2-177

## **N**

Needle Valve block 2-178

## **O**

Orifice with Variable Area Round Holes  
block 2-184  
Orifice with Variable Area Slot block 2-191

## **P**

Pilot-Operated Check Valve block 2-196  
Pipe Bend block 2-203  
pm\_adddimension command 3-2  
pm\_addunit command 3-3  
pm\_getunits command 3-4  
Poppet Valve block 2-208  
Pressure Compensator block 2-222  
Pressure Reducing Valve block 2-229  
Pressure Relief Valve block 2-234  
Pressure-Compensated Flow Control Valve  
block 2-214  
Proportional and Servo-Valve Actuator  
block 2-240  
PS Add block 2-244  
PS Dead Zone block 2-246  
PS Divide block 2-248  
PS Gain block 2-250  
PS Integrator block 2-252  
PS Lookup Table (1D) block 2-254  
PS Lookup Table (2D) block 2-257  
PS Product block 2-261  
PS Saturation block 2-263  
PS Subtract block 2-265  
PS-Simulink Converter block 2-267

## **R**

Reservoir block 2-269  
Resistive Tube block 2-271  
Rotational Damper block 2-278  
Rotational Friction block 2-280  
Rotational Hard Stop block 2-287  
Rotational Spring block 2-292

## **S**

Segmented Pipeline block 2-295  
Shuttle Valve block 2-301  
Simulink-PS Converter block 2-308  
Single-Acting Hydraulic Cylinder block 2-310  
Solver Configuration block 2-315  
Spring-Loaded Accumulator block 2-317  
Sudden Area Change block 2-320

## **T**

T-junction block 2-327  
Translational Damper block 2-331  
Translational Friction block 2-333  
Translational Hard Stop block 2-339  
Translational Hydro-Mechanical Converter  
block 2-344  
Translational Spring block 2-347

## **V**

Variable Orifice block 2-350  
Variable Volume Chamber block 2-363  
Variable-Displacement Motor block 2-368  
Variable-Displacement Pressure-Compensated  
Pump block 2-378  
Variable-Displacement Pump block 2-385

## **W**

Wheel and Axle block 2-394