

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 of The MathWorks, Inc.

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

Patents

The MathWorks products are protected by one or more U.S. patents. Please see www.mathworks.com/patents for more information.

Revision History

March 2006 Online only New for Version 1.0 (Release 2006a+)

Blocks – By Category

1

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

Blocks – Alphabetical List

2

Index

Blocks – By Category

This section lists all SimHydraulics blocks arranged by library. The subsequent pages contain reference information for all blocks in SimHydraulics, arranged in alphabetical order by block name.

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. The Physical Networks library consists of the following libraries:

Foundation Library (p. 1-2)	Contains basic hydraulic, mechanical, and physical signal blocks
SimHydraulics Library (p. 1-8)	Contains advanced hydraulic diagram blocks, such as valves, cylinders, pipelines, pumps, and accumulators
Utilities Library (p. 1-12)	Contains essential environment blocks for creating Physical Networks models

These libraries contain sublibraries of blocks, organized into categories according to their behavior. Use the Simulink Library Browser or the Physical Networks library to access the blocks directly, guided by this hierarchical library list.

Foundation Library

The Foundation library contains the following sublibraries:

Hydraulic Library (p. 1-2)	Contains basic hydraulic diagram blocks, such as orifices, chambers, sensors and sources, and utilities
Mechanical Library (p. 1-4)	Contains mechanical elements for rotational and translational motion, as well as mechanical sensors and sources
Physical Signals Library (p. 1-6)	Contains blocks for transmitting physical control signals

Hydraulic Library

The Hydraulic library contains the following sublibraries:

Hydraulic Elements Library (p. 1-2)	Simulate hydraulic building blocks, such as orifices, chambers, and hydro-mechanical converters
Hydraulic Sensors and Sources Library (p. 1-3)	Simulate hydraulic sensors and sources
Hydraulic Utilities Library (p. 1-4)	Simulate basic hydraulic environment blocks, such as custom hydraulic fluid

Hydraulic Elements Library

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 Library

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 Library

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

Mechanical Library

The Mechanical library contains the following sublibraries:

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

Mechanical Sensors and Sources Library

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 Library

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 Library

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 Library

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 Library

The Physical Signals library contains the following sublibraries:

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

Functions Library

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

Linear Operators Library

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

Lookup Tables Library

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

Nonlinear Operators Library

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

SimHydraulics Library

The SimHydraulics library contains the following sublibraries:

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

Accumulators Library

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 Library

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 Library

Hydraulic Fluid	Set working fluid properties by selecting from list of predefined fluids
-----------------	--

Local Hydraulic Resistances Library

Local Resistance	Simulate all kinds of hydraulic resistances specified by a loss coefficient
------------------	---

Pipelines Library

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 Library

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

Valves Library

The Valves library contains the following sublibraries:

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

Directional Valves Library

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

Flow Control Valves Library

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 Library

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

Valve Actuators Library

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 the output proportional to the input signal

Utilities Library

Connection Port

Create a Physical Modeling connector port for a subsystem

PS-Simulink Converter

Convert physical signal into Simulink output signal

Simulink-PS Converter

Convert Simulink input signal into physical signal

Solver Configuration

Represent the Physical Networks environment and solver configuration

Blocks – Alphabetical List

2-Position Valve Actuator

Purpose Simulate actuator for two-position valves

Library Valve Actuators

Description

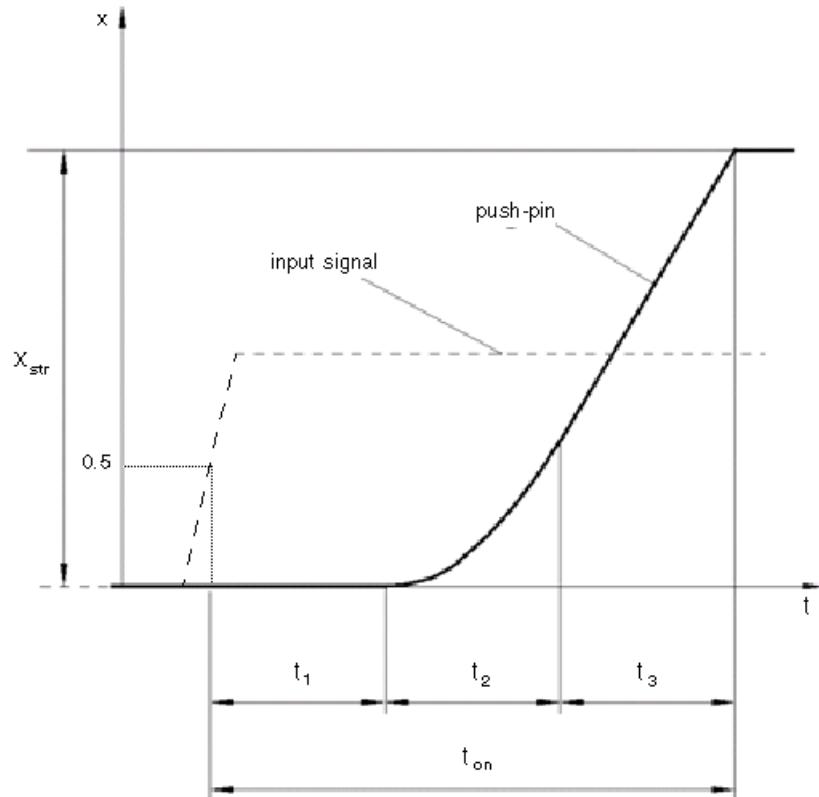


The 2-Position Valve Actuator block represents an actuator that you can use with directional valves to control their position. This actuator can drive a two-position valve. The block is developed as a data sheet-based model and all its parameters are generally provided in catalogs or data sheets. The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

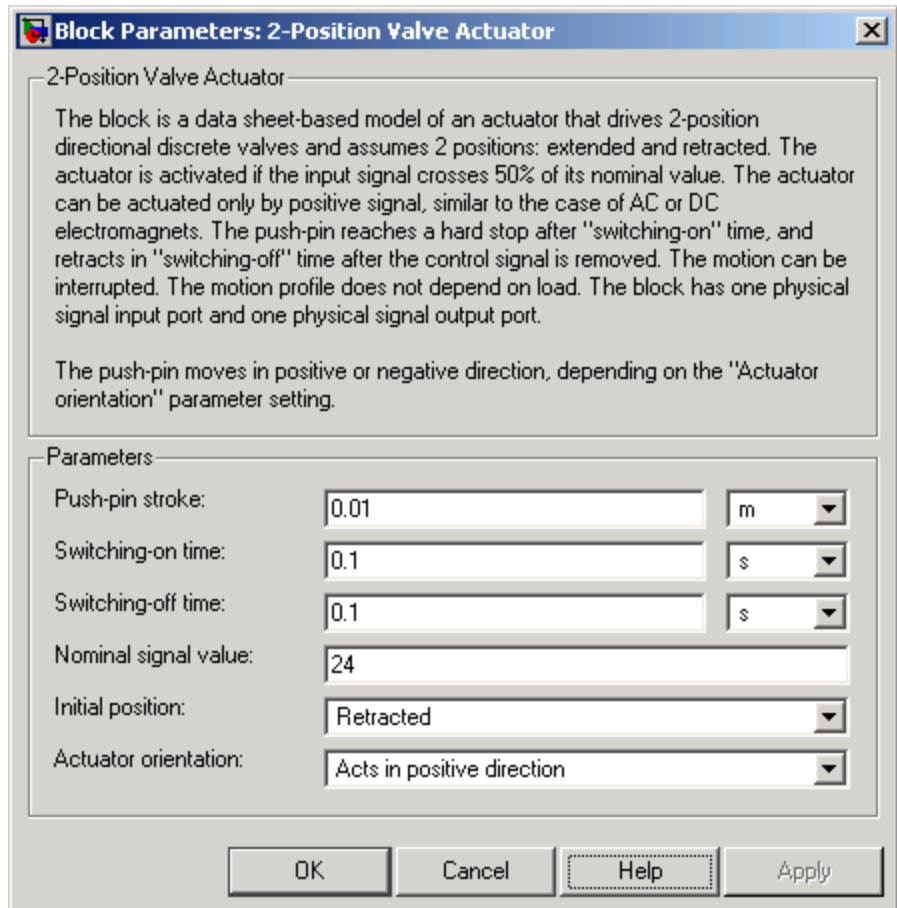
The push-pin is actuated only by positive signal, similar to the AC or DC electromagnets. The direction of push-pin motion is controlled by the **Actuator orientation** parameter, which can have one of two values: Acts in positive direction or Acts in negative direction.

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.

2-Position Valve Actuator

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

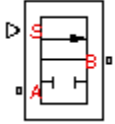
Proportional and Servo-Valve Actuator

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 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

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

2-Way Directional Valve

Block Parameters: 2-Way Directional Valve [X]

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:

Tabulated valve openings:

Tabulated valve passage area:

Interpolation method:

Extrapolation method:

Flow discharge coefficient:

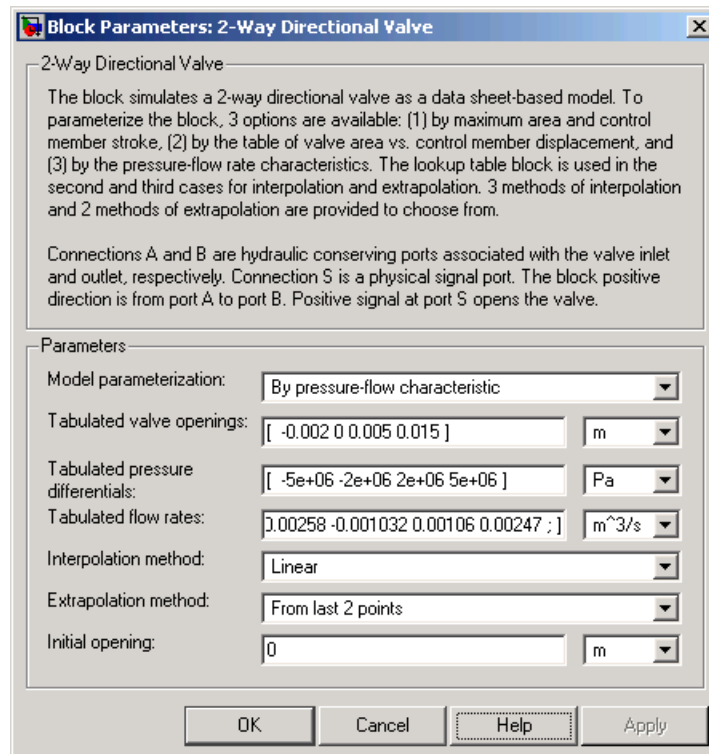
Initial opening:

Critical Reynolds number:

Leakage area:

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 non-uniformly spaced. You must provide at least 3 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 non-uniformly spaced. You must provide at least 3 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

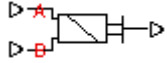
- 3-Way Directional Valve
- 4-Way Directional Valve

3-Position Valve Actuator

Purpose Simulate actuator for three-position valves

Library Valve Actuators

Description

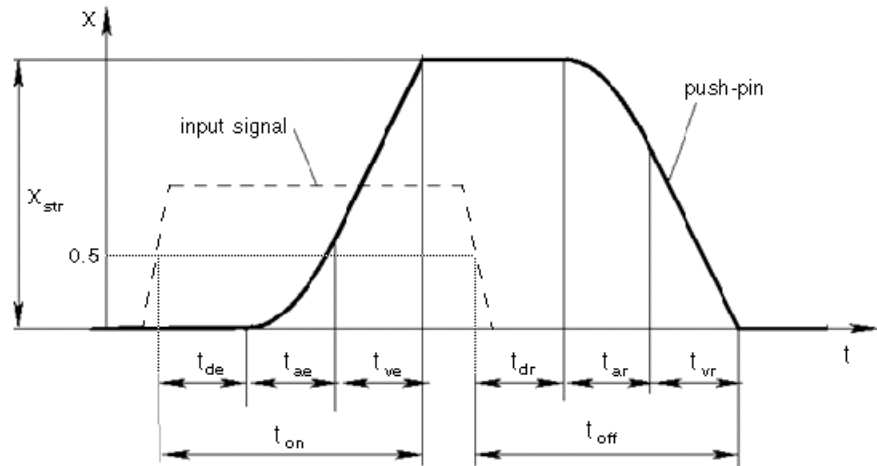


The 3-Position Valve Actuator block represents an actuator that you can use with directional valves to control their position. This actuator can drive a three-position valve. The block is developed as a data sheet-based model and all its parameters are generally provided in catalogs or data sheets. The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

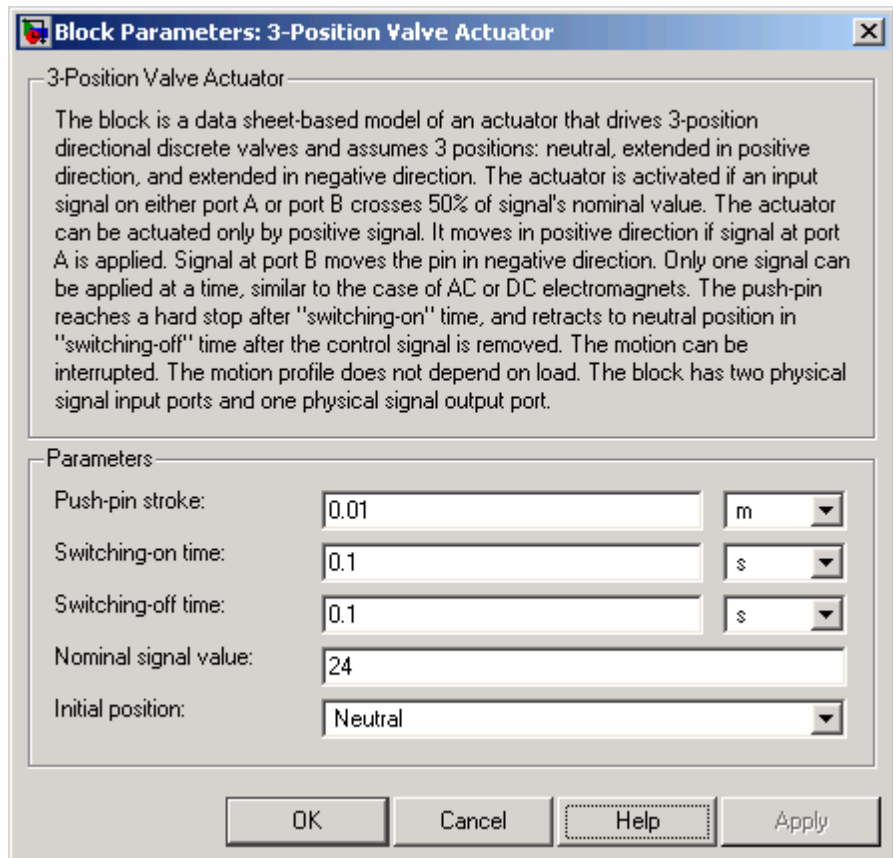
Only positive signals activate the actuator. In other words, negative signals at ports A and B have no effect on the actuator, which is similar to the behavior of electromagnetically-controlled 3-position directional valves.

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

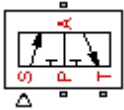
Proportional and Servo-Valve Actuator

3-Way Directional Valve

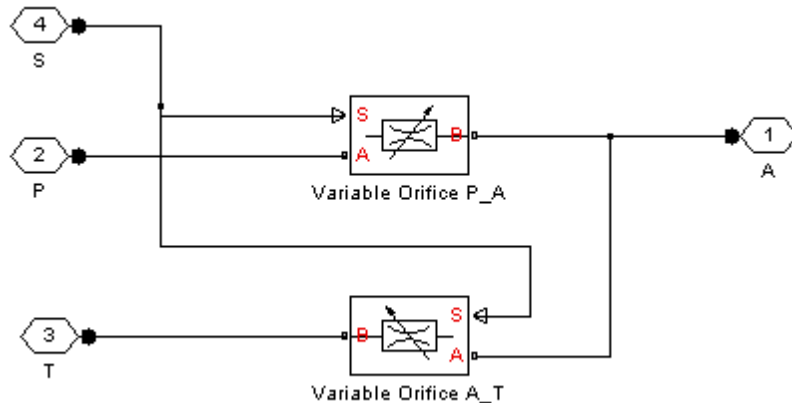
Purpose Simulate hydraulic continuous 3-way directional valve

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 non-uniformly 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

Block Parameters: 3-Way Directional Valve

3-Way Directional Valve

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

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

Parameters

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

OK Cancel Help Apply

3-Way Directional Valve

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] m²

Interpolation method: Linear

Extrapolation method: From last 2 points

Flow discharge coefficient: 0.7

Orifice P-A initial opening: 0 m

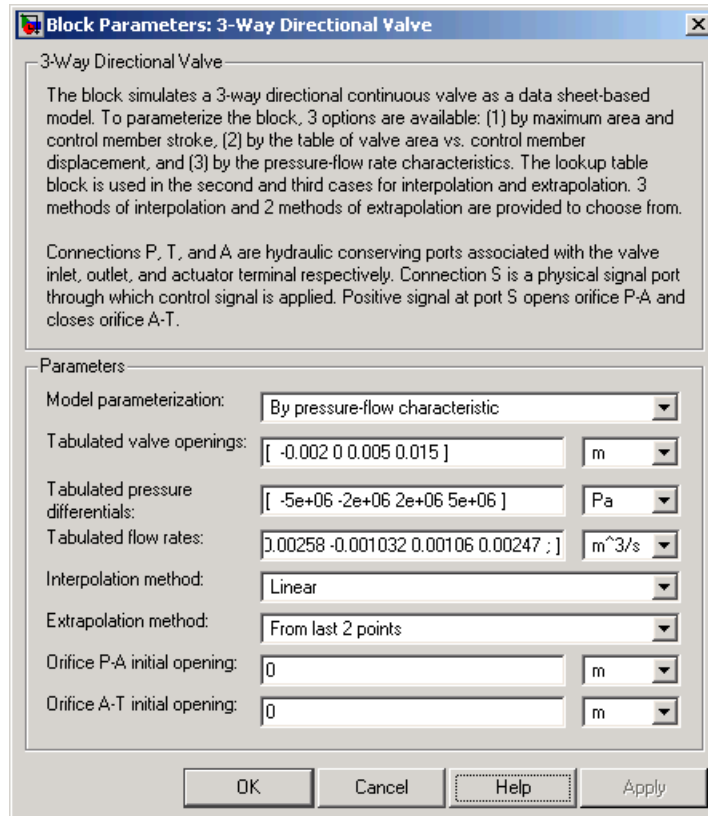
Orifice A-T initial opening: 0 m

Critical Reynolds number: 12

Leakage area: 1e-12 m²

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 non-uniformly spaced. You must provide at least 3 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 non-uniformly spaced. You must provide at least 3 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

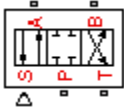
4-Way Directional Valve

4-Way Directional Valve

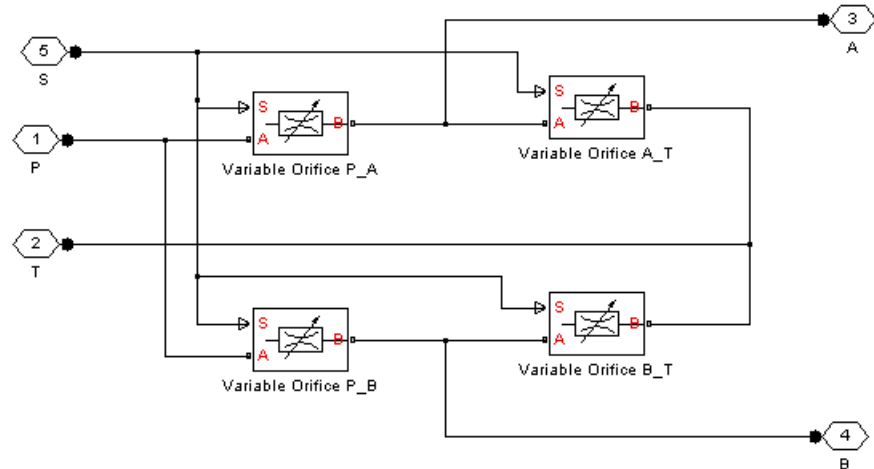
Purpose Simulate hydraulic continuous 4-way directional valve

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 non-uniformly 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.

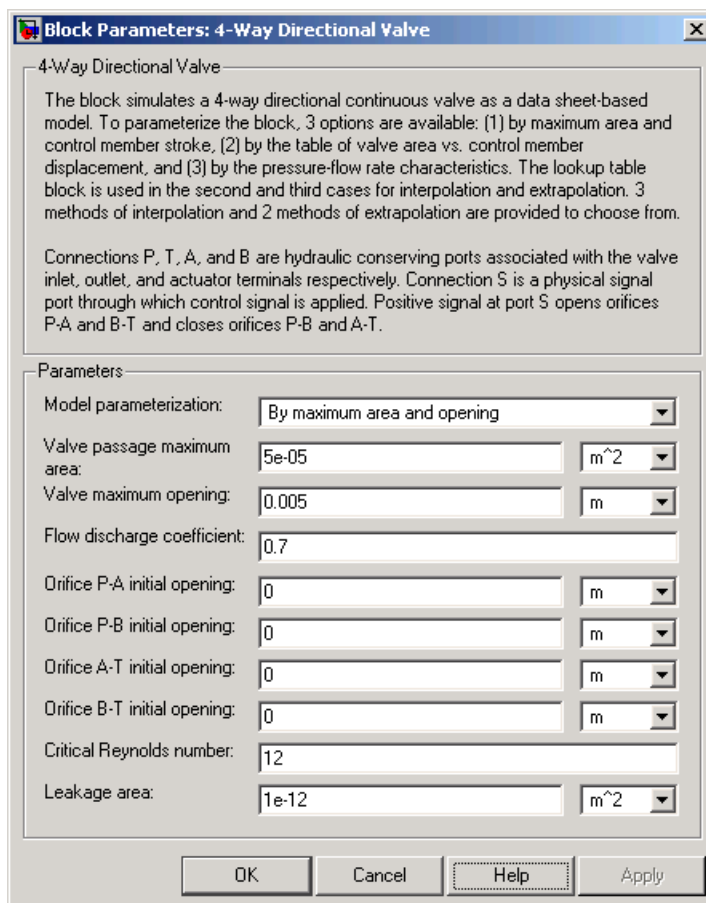
Basic Assumptions and Limitations

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 [X]

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:

Tabulated valve openings:

Tabulated valve passage area:

Interpolation method:

Extrapolation method:

Flow discharge coefficient:

Orifice P-A initial opening:

Orifice P-B initial opening:

Orifice A-T initial opening:

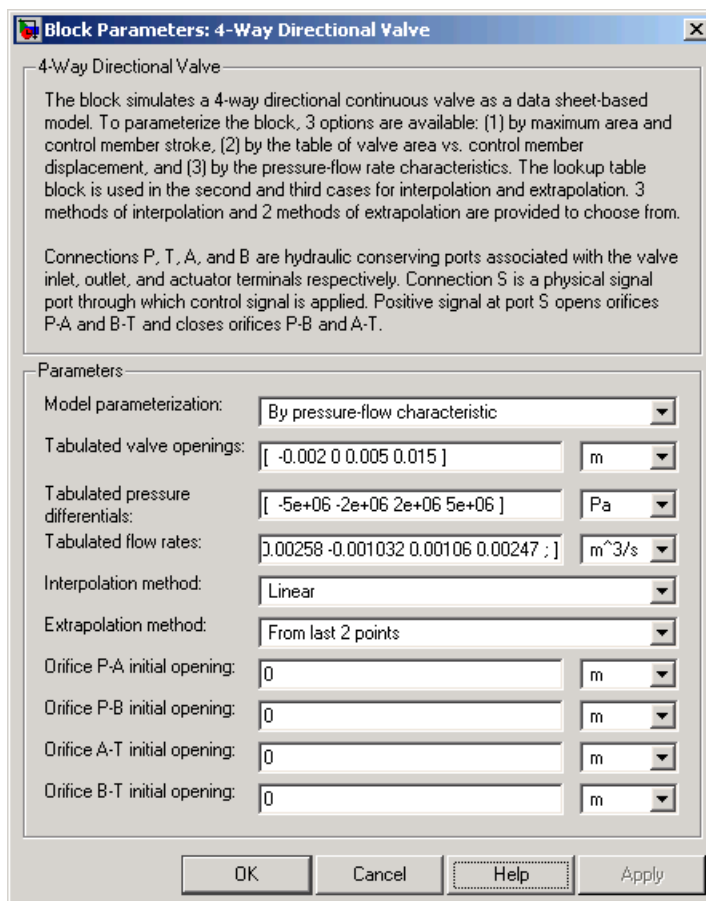
Orifice B-T initial opening:

Critical Reynolds number:

Leakage area:

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 non-uniformly spaced. You must provide at least 3 values. The default values, in meters, are $[-2e-3, 0, 5e-3, 15e-3]$. If **Model parameterization** is set to By area vs. opening table, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to By pressure-flow characteristic, the **Tabulated valve openings** values will

4-Way Directional Valve

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

Tabulated valve passage area

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

Tabulated pressure differentials

Specify the vector of input values for pressure differentials as a tabulated 1-by-n array. The input values vector must be strictly monotonically increasing. The values can be non-uniformly spaced. You must provide at least 3 values. The default values, in Pa, are [-5e6, -2e6, 2e6, 5e6]. This parameter is used if **Model parameterization** is set to By pressure-flow characteristic.

Tabulated flow rates

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

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

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

Interpolation method

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

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

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

Extrapolation method

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

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

4-Way Directional Valve

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

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

Flow discharge coefficient

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

Orifice P-A initial opening

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

Orifice P-B initial opening

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

Orifice A-T initial opening

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

Orifice B-T initial opening

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

Critical Reynolds number

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

Leakage area

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

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

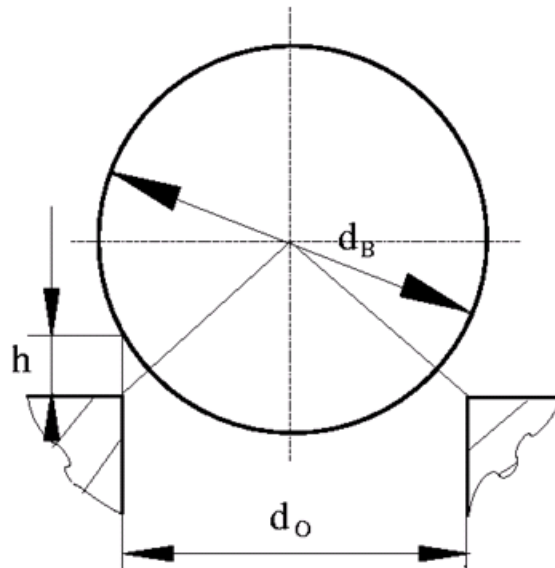
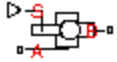
2-Way Directional Valve

3-Way Directional Valve

Purpose Simulate hydraulic 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	Gage pressures at the block terminals
C_D	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
x_0	Initial opening
x	Ball displacement from initial position
h	Valve opening
d_o	Orifice diameter
r_o	Orifice radius
d_B	Ball diameter
r_B	Ball radius
ρ	Fluid density
D_H	Valve instantaneous hydraulic diameter
ν	Fluid kinematic viscosity
A_{leak}	Closed valve leakage area
A_{max}	Maximum valve open area

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

Basic Assumptions and Limitations

The model is based on the following assumptions:

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

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 m²

OK Cancel Help Apply

Valve ball diameter

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

Orifice diameter

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

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.

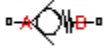
See Also

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

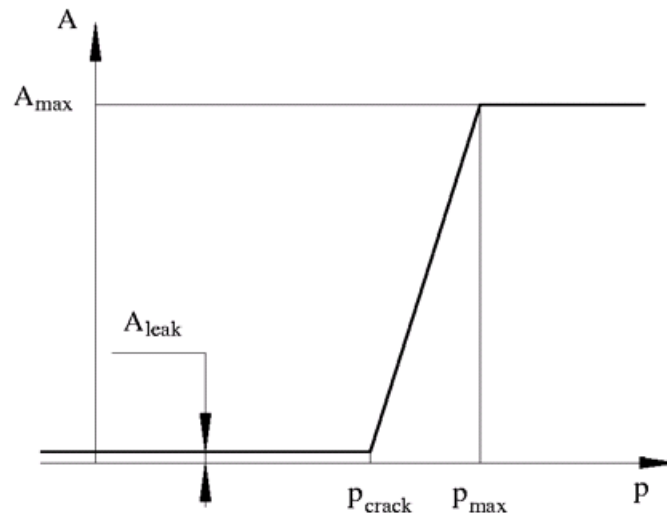
Purpose Simulate hydraulic valve that allows flow in one direction only

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:

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

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

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

$$p = p_A - p_B$$

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

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

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

where

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

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

Check Valve

Basic Assumptions and Limitations

The model is based on the following assumptions:

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

Dialog Box and Parameters

Block Parameters: Check Valve

Check Valve

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

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

Parameters

Maximum passage area:	<input type="text" value="0.0001"/>	<input type="text" value="m^2"/>
Cracking pressure:	<input type="text" value="30000"/>	<input type="text" value="Pa"/>
Maximum opening pressure:	<input type="text" value="120000"/>	<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 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 a Physical Modeling connector port for a subsystem

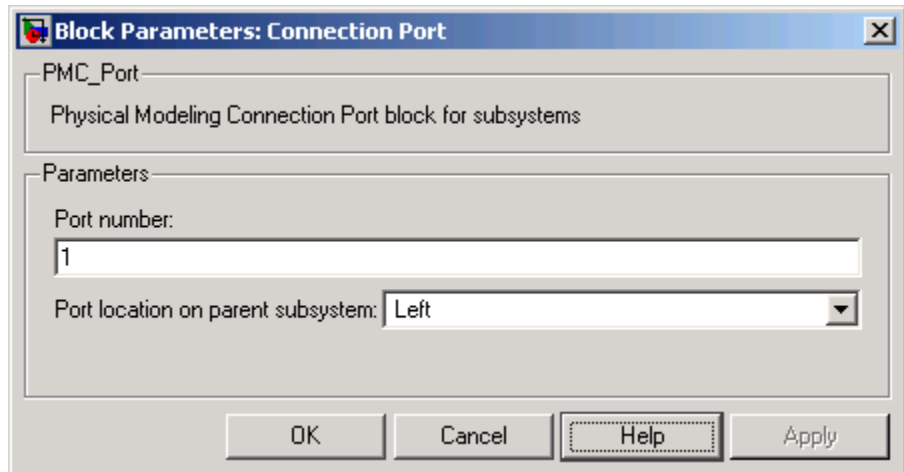
Library Utilities

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

p Gage 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, non-dissolved gas. The mixture bulk modulus is determined as:

Constant Volume Chamber

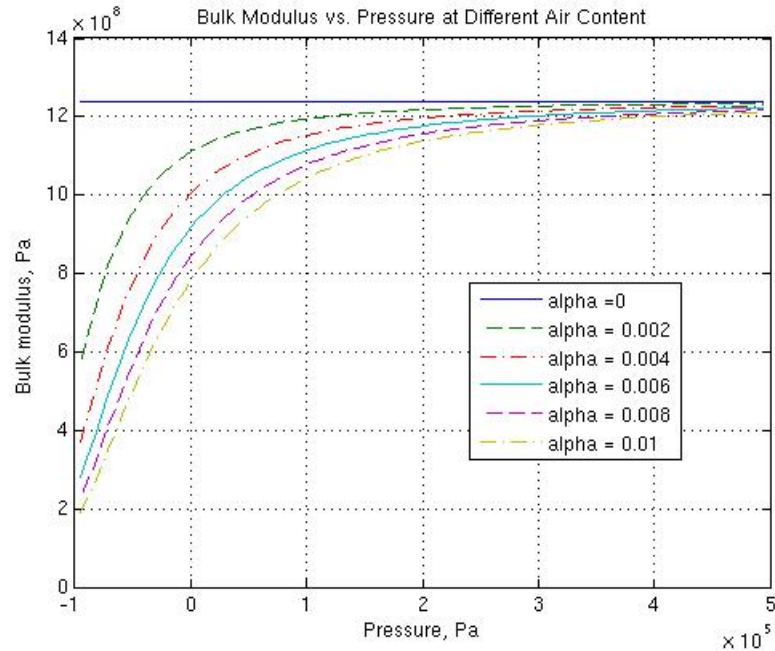
$$E = E_l \frac{1 + \alpha \left(\frac{p_a}{p_a + p} \right)^{1/n}}{1 + \alpha \frac{p_a^{1/n}}{n \cdot (p_a + p)^{\frac{n+1}{n}}} E_l}$$

where

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

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

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



Cavitation is an inherently thermodynamic process, requiring consideration of multiple-phase fluids, heat transfers, etc., and as such can not 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)
τ	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
	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.

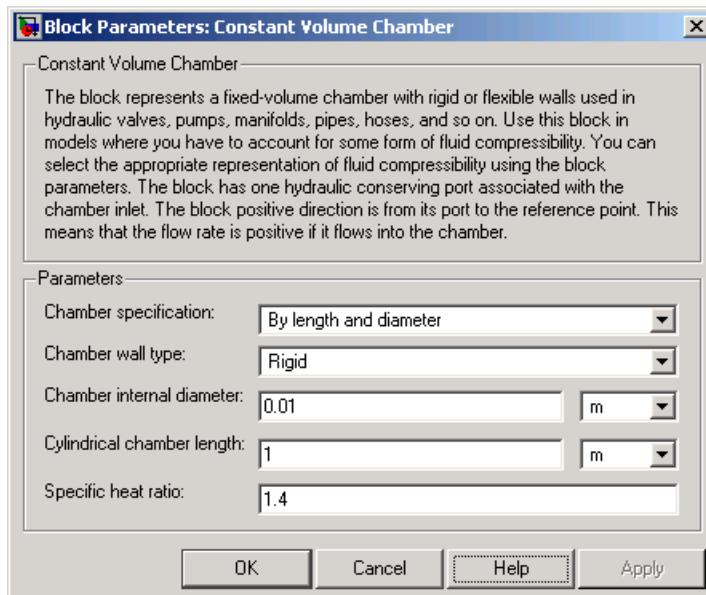
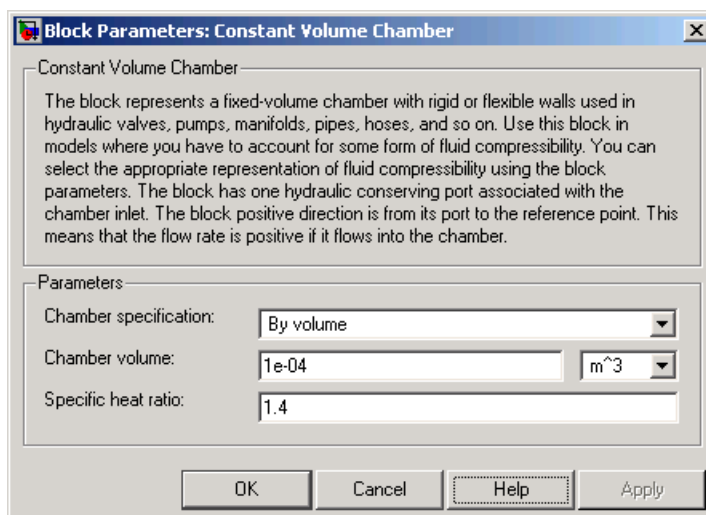
Basic Assumptions and Limitations

The model is based on the following assumptions:

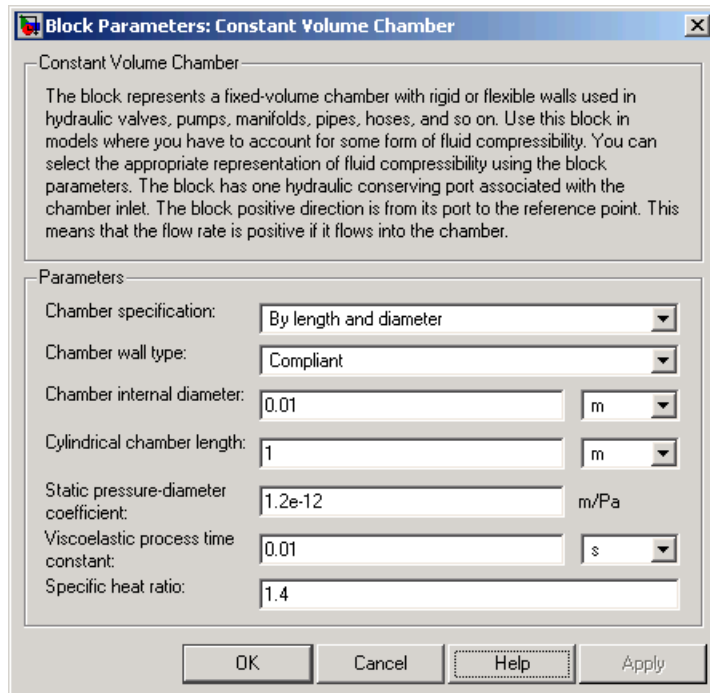
- No inertia associated with pipe walls is taken into account.
- Chamber with compliant walls is assumed to have a cylindrical shape. Chamber with rigid wall can have any shape.

Constant Volume Chamber

Dialog Box and Parameters



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.

Non-dissolved gas ratio

Non-dissolved 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

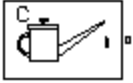
Variable Volume Chamber

Custom Hydraulic Fluid

Purpose Set working fluid properties by specifying parameter values

Library Hydraulic Utilities

Description



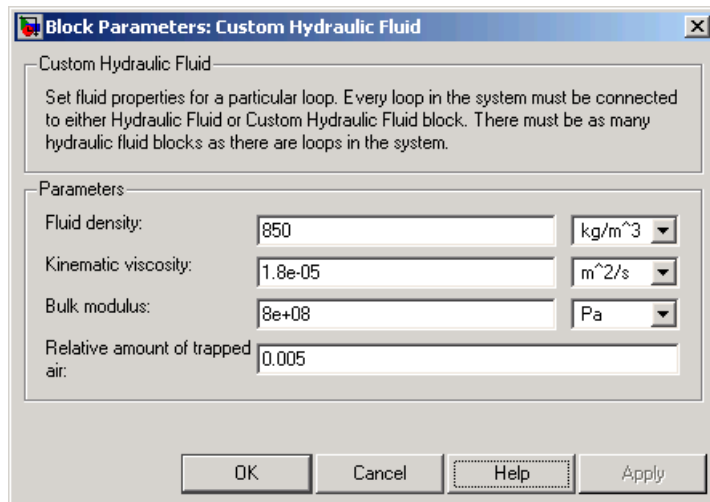
The Custom Hydraulic Fluid block lets you specify the type of hydraulic fluid used in a loop of hydraulic blocks. It provides the hydraulic fluid properties, such as kinematic viscosity, density, and bulk modulus, for all the hydraulic blocks in the loop. These fluid properties are assumed to be constant 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 kg/m³.

Kinematic viscosity

Kinematic viscosity of the working fluid. The default value is 1.8e-5 m²/s.

Bulk modulus

Bulk modulus of the working fluid. The default value is 8e8 Pa.

Relative amount of trapped air

Amount of entrained, non-dissolved 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

Double-Acting Hydraulic Cylinder

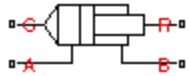
Purpose

Simulate hydraulic actuator exerting force in both directions

Library

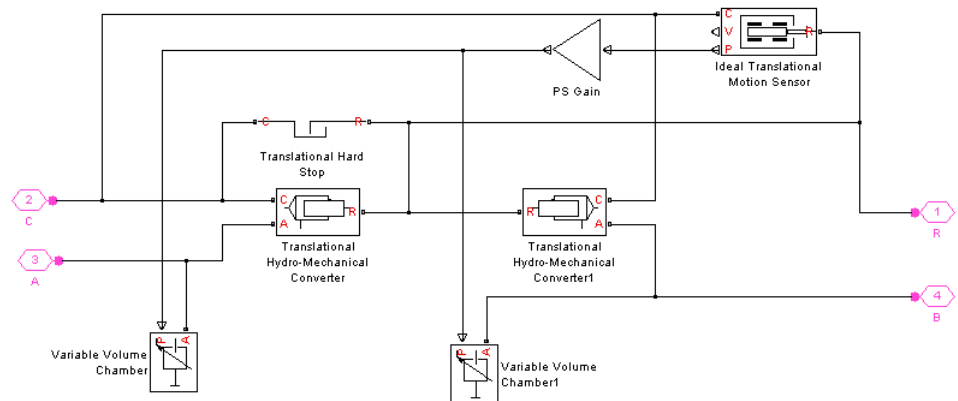
Cylinders

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.

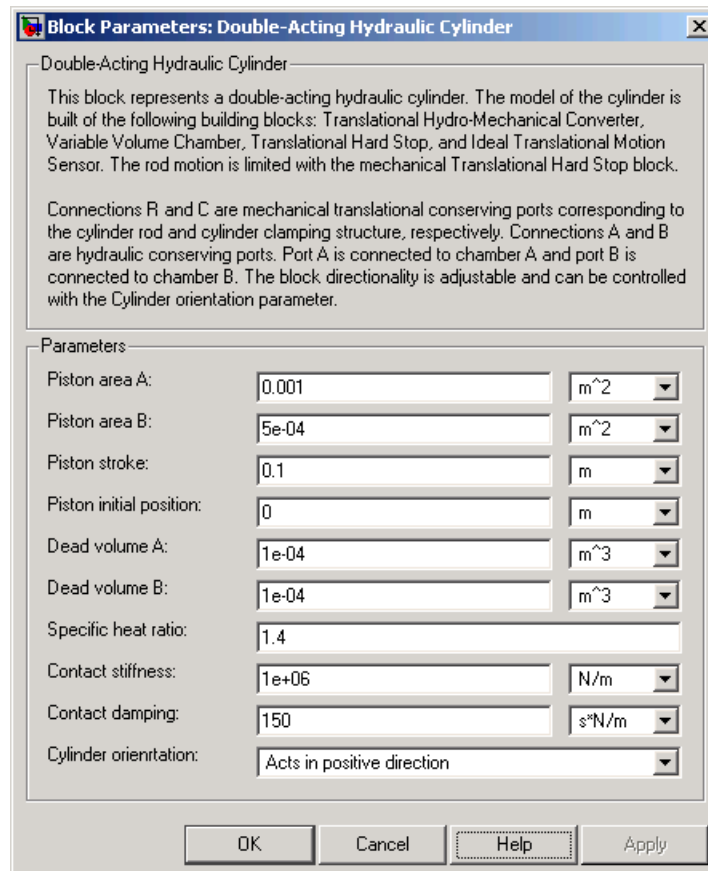
Basic Assumptions and Limitations

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 m².

Chamber B piston area

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

Piston stroke

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

Double-Acting Hydraulic Cylinder

Piston initial position

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

Dead volume A

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

Dead volume B

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

Specific heat ratio

Gas specific heat ratio for the 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 $1e6 \text{ N/m}$.

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, it is recommended that you assign a nonzero value to this parameter. The default value is $150 \text{ N}^*\text{s/m}$.

Cylinder orientation

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

Double-Acting Hydraulic Cylinder

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.

See Also

`Single-Acting Hydraulic Cylinder`
`Ideal Translational Motion Sensor`

Double-Acting Hydraulic Cylinder

Translational Hard Stop

Translational Hydro-Mechanical Converter

Variable Volume Chamber

Fixed Orifice

Purpose Simulate hydraulic orifice with constant cross-sectional area

Library Hydraulic Elements

Description



The 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 Gage pressures at the block terminals

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

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

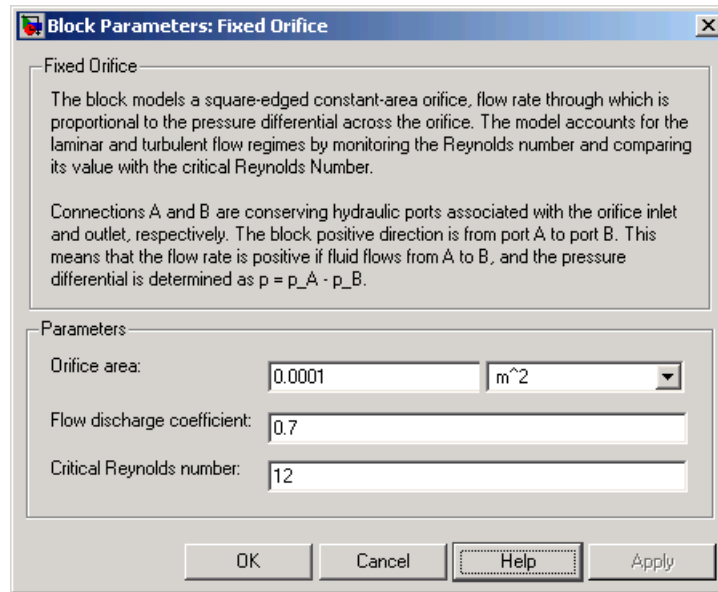
Basic Assumptions and Limitations

The model is based on the following assumptions:

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

Fixed Orifice

Dialog Box and Parameters



Orifice area

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

Flow discharge coefficient

Semi-empirical parameter for orifice capacity characterization.

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

The default value is 0.7.

Critical Reynolds number

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

Global Parameters

Fluid density

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

Fluid kinematic viscosity

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

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the orifice inlet.

B

Hydraulic conserving port associated with the orifice outlet.

See Also

Orifice with Variable Area Round Holes

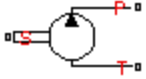
Orifice with Variable Area Slot

Fixed-Displacement Pump

Purpose Simulate fixed-displacement hydraulic pump

Library Pumps and Motors

Description



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

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

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

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

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

$$p = p_P - p_T$$

where

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

k_{HP}	Hagen-Poiseuille coefficient
η_V	Pump volumetric efficiency
η_{mech}	Pump mechanical efficiency
ν	Fluid kinematic viscosity
ρ	Fluid density
p_{nom}	Pump nominal pressure
ω_{nom}	Pump nominal angular velocity
ν_{nom}	Nominal fluid kinematic viscosity

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

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

where

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

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

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

Fixed-Displacement Pump

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

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

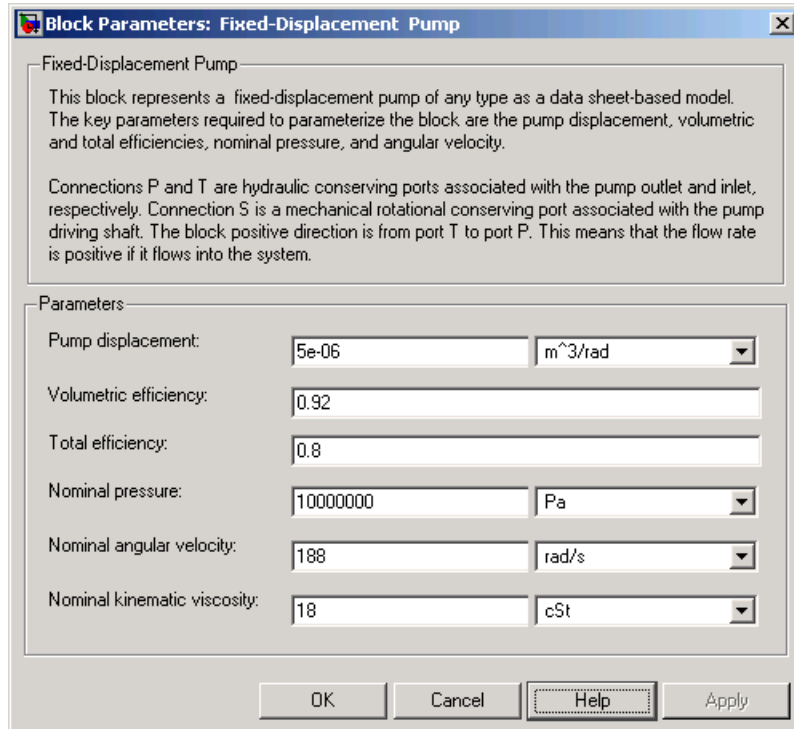
The block positive direction is from port T to port P. This means that the pump transfers fluid from T to P provided that the shaft S rotates in the positive direction. The pressure differential across the pump is determined as $p = p_P - p_T$.

Basic Assumptions and Limitations

The model is based on the following assumptions:

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

Dialog Box and Parameters



Pump displacement

Pump displacement. The default value is $5e-6$ m³/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.

Fixed-Displacement Pump

Nominal pressure

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

Nominal angular velocity

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

Nominal kinematic viscosity

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

Global Parameters

Fluid kinematic viscosity

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

Ports

The block has the following ports:

T

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

P

Hydraulic conserving port associated with the pump outlet.

S

Mechanical rotational conserving port associated with the pump driving shaft.

Examples

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

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

The simulation starts with the variable orifice open, which results in a low system pressure and the maximum flow rate going to the system. The orifice starts closing at 0.5 s, and is closed completely at 3 s. The output pressure builds up until it reaches the pressure setting of the relief valve (75×10^5 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

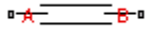
Variable-Displacement Pressure-Compensated Pump

Fluid Inertia

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

p	Pressure differential
ρ	Fluid density
L	Passage length
A	Passage area
q	Flow rate
t	Time

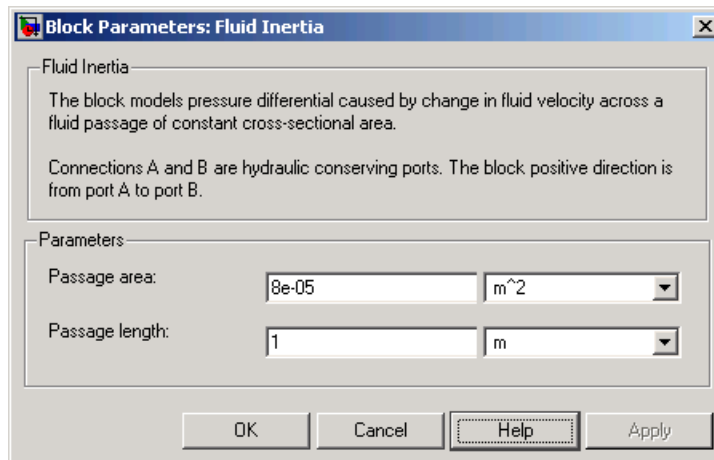
Use this block in various pipe or channel models that require fluid inertia to be accounted for.

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.

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.

Gas-Charged Accumulator

Purpose Simulate hydraulic accumulator with gas as compressible medium

Library Accumulators

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



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

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

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

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

where

V_F	Fluid volume
V_A	Accumulator capacity
ρ	Inlet gage pressure
ρ_{pr}	Precharge pressure
k	Specific heat ratio
q	Volumetric flow rate
t	Time

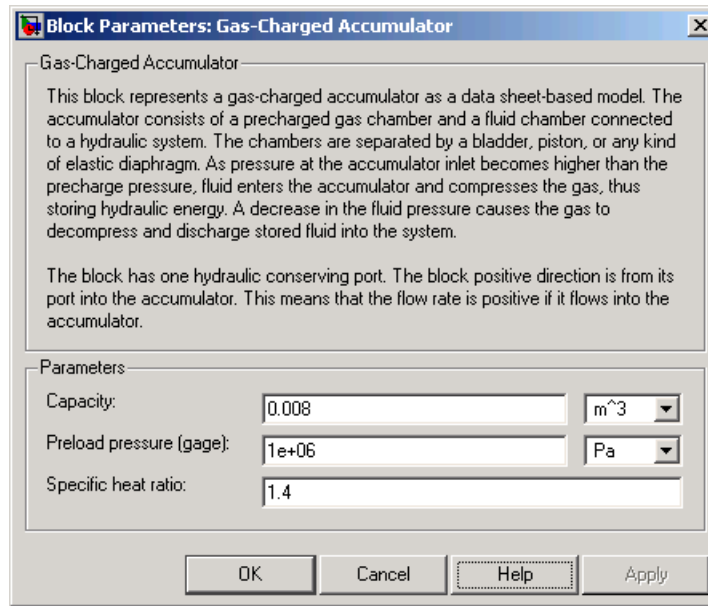
Basic Assumptions and Limitations

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.

Gas-Charged Accumulator

Dialog Box and Parameters



Accumulator capacity

Accumulator capacity. The default value is 0.008 m³.

Preload pressure

Precharge gage 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).

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

Gear Box

Purpose Simulate gear boxes in mechanical systems

Library Mechanisms

Description The Gear Box block represents an ideal, non-planetary, 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

ω_1 Input shaft angular velocity

ω_2 Output shaft angular velocity

N Gear ratio

T_1 Torque on the input shaft

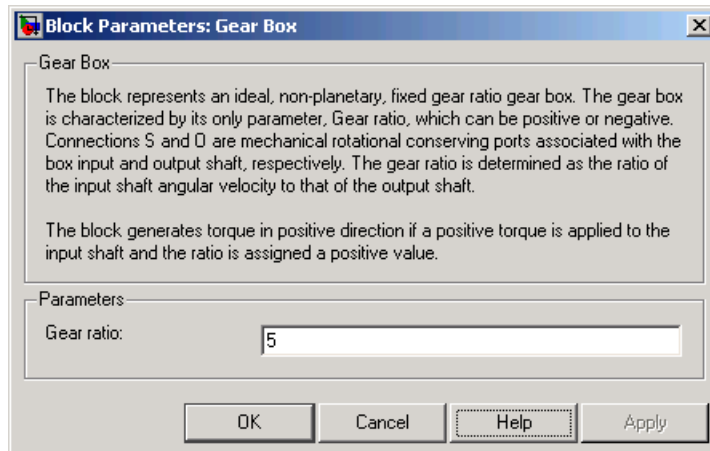
T_2 Torque on the output shaft

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.

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

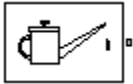
Mechanical rotational conserving port associated with the output shaft.

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, non-dissolved 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, non-dissolved 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/m³

Viscosity: 5.501634e-006 cSt

Bulk modulus: 1.139178e+009 Pa

OK Cancel Help Apply

Hydraulic fluid

Hydraulic fluid type. Select one of the predefined fluids:

- Skydrol LD-4
- 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, non-dissolved 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.

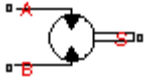
See Also

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	Gage pressures at the block terminals
T	Torque at the motor output shaft
D	Motor displacement
ω	Output shaft angular velocity
k_{leak}	Leakage coefficient

Hydraulic Motor

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

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

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

where

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

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

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

which provides the formula to determine the Hagen-Poiseuille coefficient

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

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

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

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

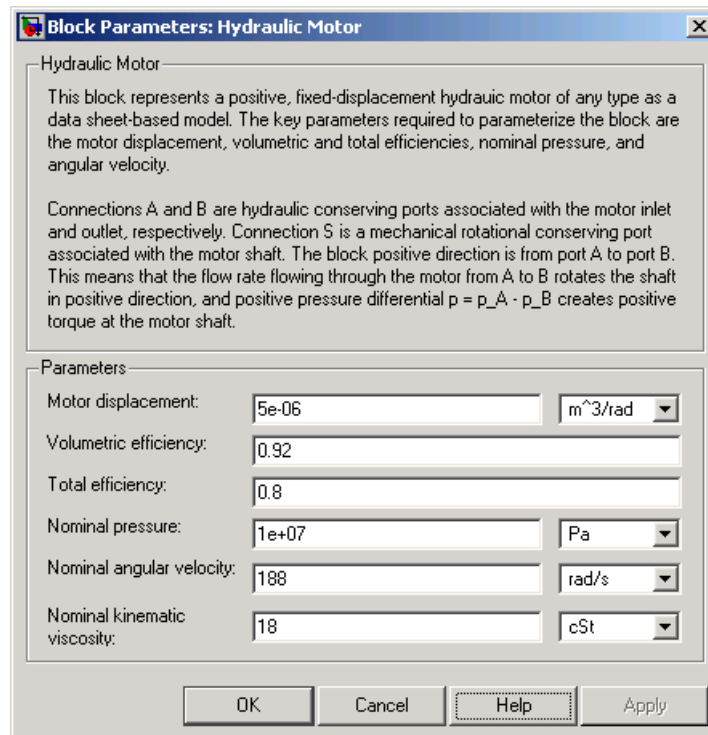
Basic Assumptions and Limitations

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 $1e7$ Pa.

Nominal angular velocity

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

Nominal kinematic viscosity

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

Global Parameters**Fluid kinematic viscosity**

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

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the motor inlet.

B

Hydraulic conserving port associated with the motor outlet.

S

Mechanical rotational conserving port associated with the motor output shaft.

Hydraulic Pipeline

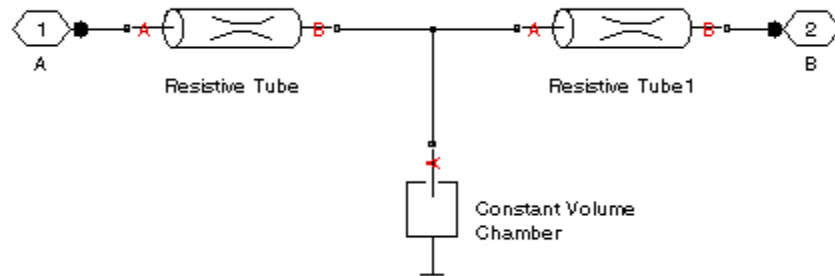
Purpose Simulate hydraulic pipeline with resistive and fluid compressibility properties

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 can not 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.

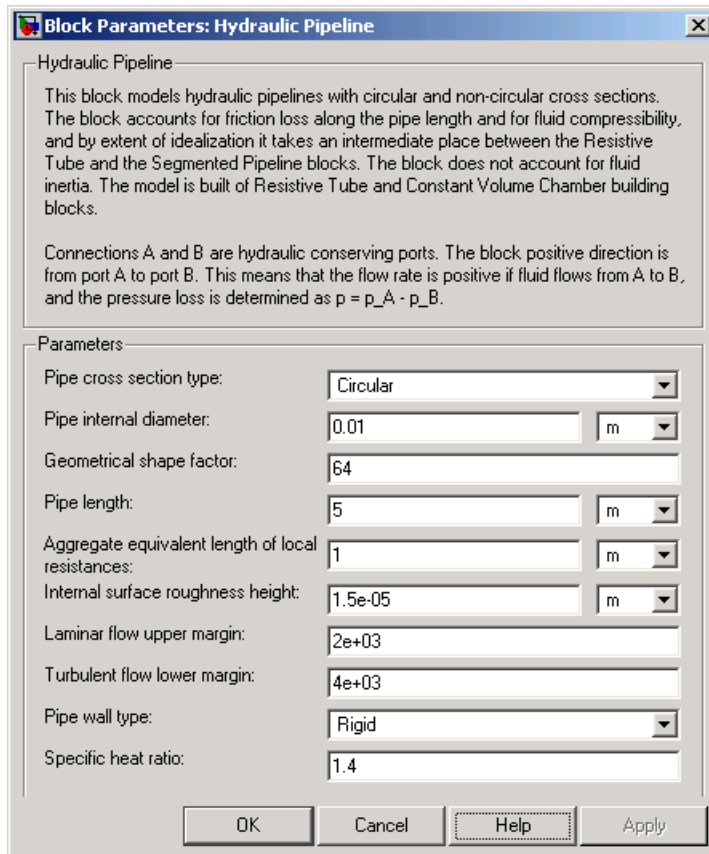
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

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

Hydraulic Pipeline

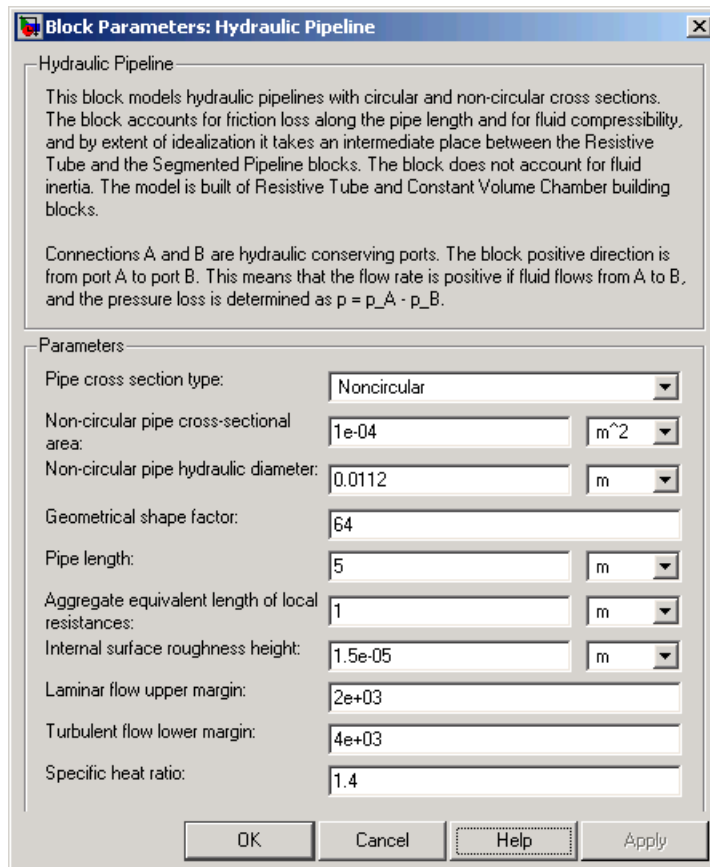
This block models hydraulic pipelines with circular and non-circular cross sections. The block accounts for friction loss along the pipe length and for fluid compressibility, and by extent of idealization it takes an intermediate place between the Resistive Tube and the Segmented Pipeline blocks. The 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 $1e-4$ m².

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

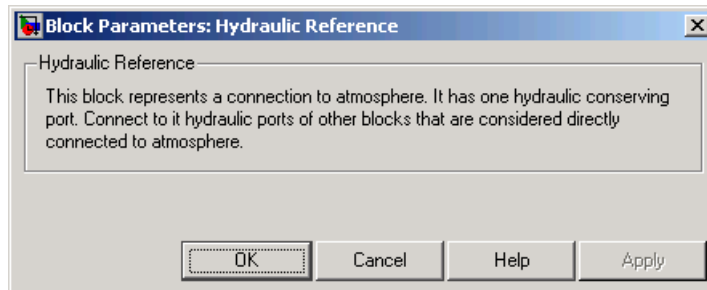
Library Hydraulic Elements

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

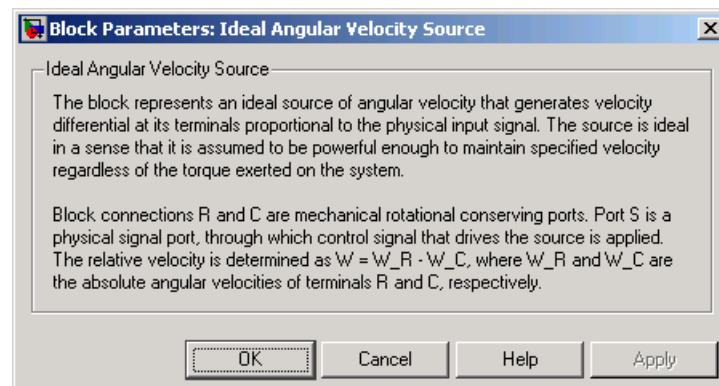


The Ideal Angular Velocity Source block represents an ideal source of angular velocity that generates velocity differential at its terminals proportional to the input physical signal. The source is ideal in a sense that it is assumed to be powerful enough to maintain specified velocity regardless of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

Connections R and C are mechanical rotational conserving ports. Port S is a physical signal port, through which 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 ω_R , ω_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.

Ideal Angular Velocity Source

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 control signal that drives the source is applied.

See Also

[Ideal Translational Velocity Source](#)

[Ideal Torque Source](#)

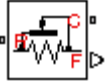
[Ideal Force Source](#)

Ideal Force Sensor

Purpose Simulate force sensor in mechanical translational systems

Library Mechanical Sensors and Sources

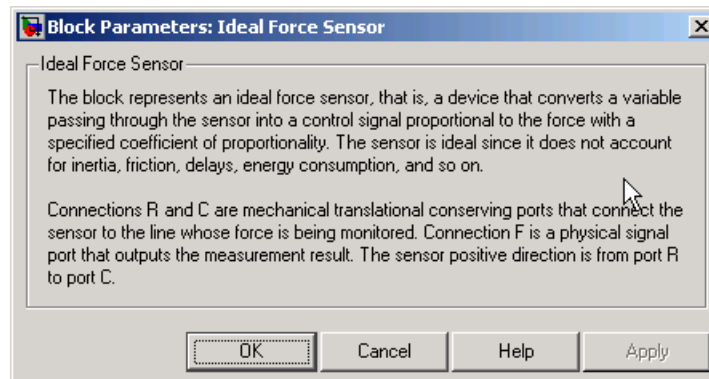
Description The Ideal Force Sensor block represents a device that converts a variable passing through the sensor into a control signal proportional to the force. The sensor is ideal since it does not account for inertia, friction, delays, energy consumption, and so on.



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.

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

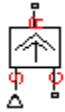
Ideal Torque Sensor
Ideal Rotational Motion Sensor
Ideal Translational Motion Sensor

Ideal Force Source

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

Library Mechanical Sensors and Sources

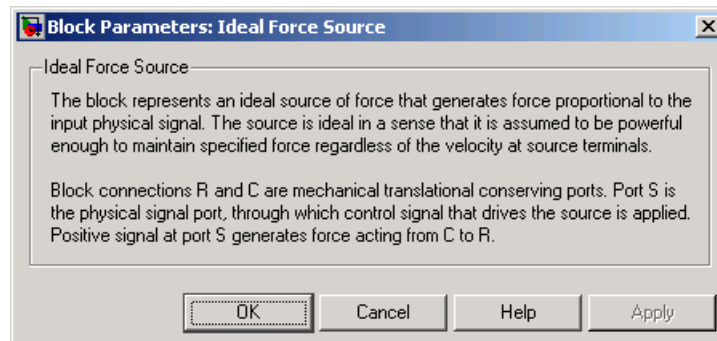
Description The Ideal Force Source block represents an ideal source of mechanical energy that generates force proportional to the input physical signal. The source is ideal in a sense that it is assumed to be powerful enough to maintain specified force at its output regardless of the velocity at source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



Connections R and C are mechanical translational conserving ports. Port S is a physical signal port, through which 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 control signal that drives the source is applied.

See Also

Ideal Translational Velocity Source

Ideal Angular Velocity Source

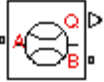
Ideal Torque Source

Ideal Hydraulic Flow Rate Sensor

Purpose Simulate ideal flow meter

Library Hydraulic Sensors and Sources

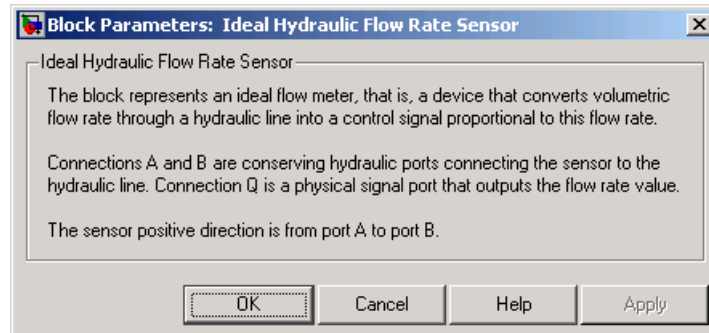
Description



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

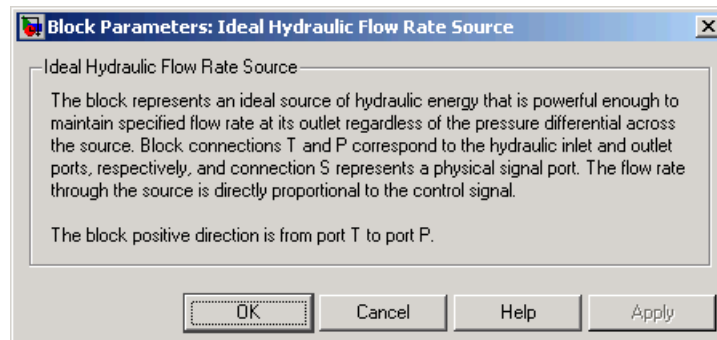
Description



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.

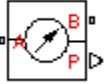
See Also Ideal Hydraulic Pressure Source

Ideal Hydraulic Pressure Sensor

Purpose Simulate ideal pressure sensing device

Library Hydraulic Sensors and Sources

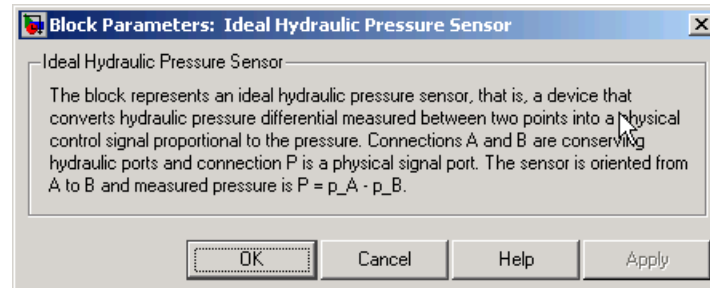
Description



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

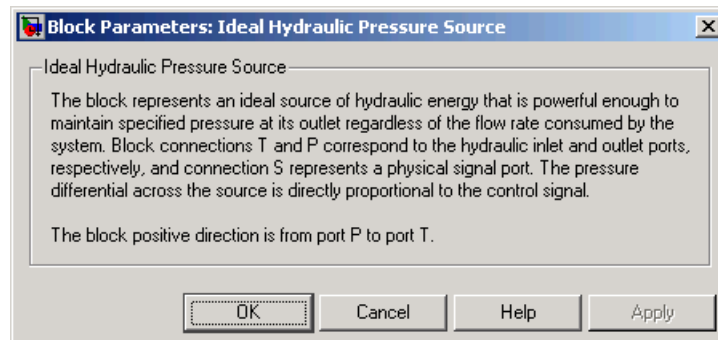


$$p = p_P - p_T$$

where p_P , p_T are the gage 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.

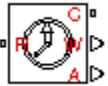
See Also Ideal Hydraulic Flow Rate Source

Ideal Rotational Motion Sensor

Purpose Simulate motion sensor in mechanical rotational systems

Library Mechanical Sensors and Sources

Description



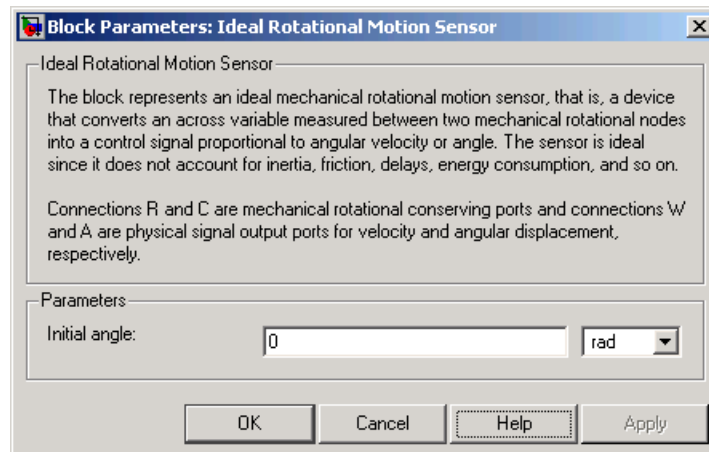
The Ideal Rotational Motion Sensor block represents an ideal mechanical rotational motion sensor, that is, a device that converts an across variable measured between two mechanical rotational nodes into a control signal proportional to angular velocity or angle. You can specify the initial angular position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

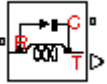
Ideal Torque Sensor

Ideal Torque Sensor

Purpose Simulate torque sensor in mechanical rotational systems

Library Mechanical Sensors and Sources

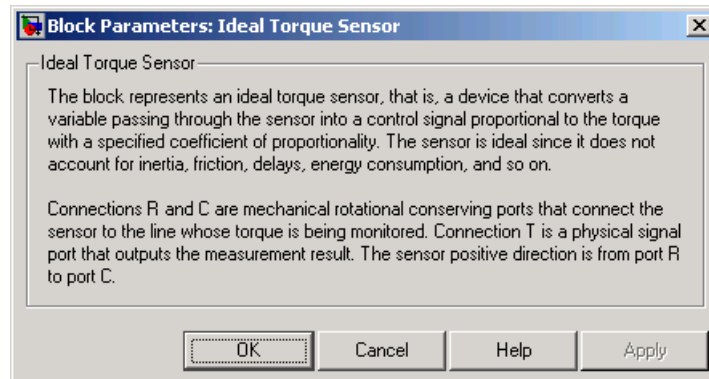
Description The Ideal Torque Sensor block represents a device that converts a variable passing through the sensor into a control signal proportional to the torque. The sensor is ideal since it does not account for inertia, friction, delays, energy consumption, and so on.



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

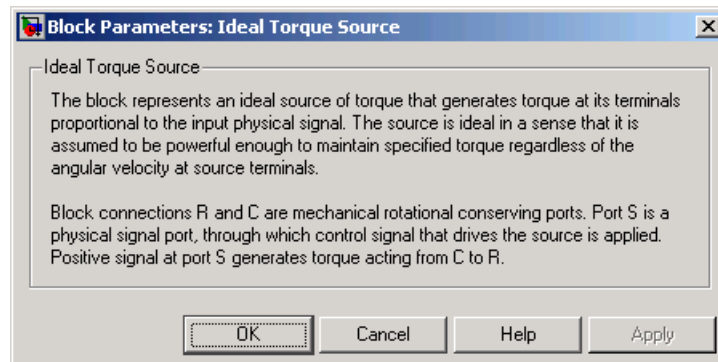
Description The Ideal Torque Source block represents an ideal source of mechanical energy that generates torque proportional to the input physical signal. The source is ideal in a sense that it is assumed to be powerful enough to maintain specified torque regardless of the angular velocity at source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



Connections R and C are mechanical rotational conserving ports. Port S is a physical signal port, through which 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 ω_R , ω_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 control signal that drives the source is applied.

See Also

Ideal Angular Velocity Source

Ideal Translational Velocity Source

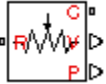
Ideal Force Source

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.

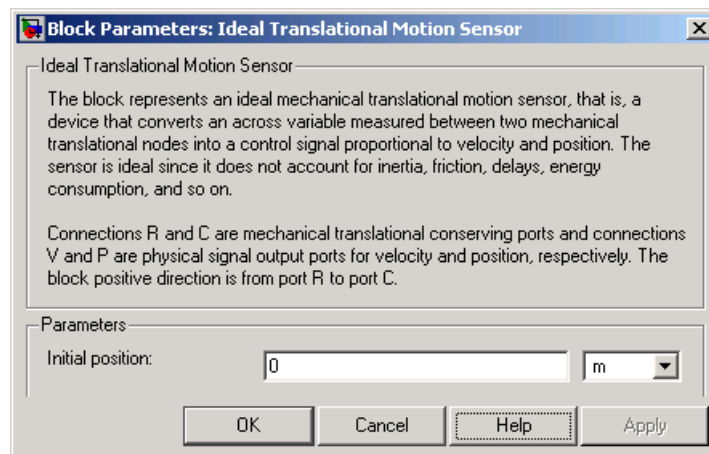


The sensor is ideal since it does not account for inertia, friction, delays, energy consumption, and so on.

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

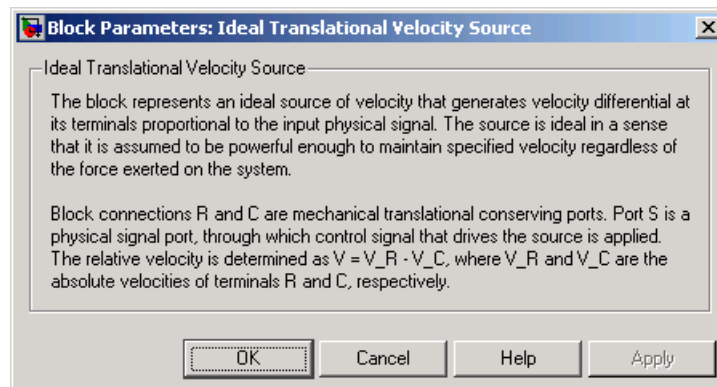


The Ideal Translational Velocity Source block represents an ideal source of velocity that generates velocity differential at its terminals proportional to the input physical signal. The source is ideal in a sense that it is assumed to be powerful enough to maintain specified velocity regardless of the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

Connections R and C are mechanical translational conserving ports. Port S is a physical signal port, through which 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 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:



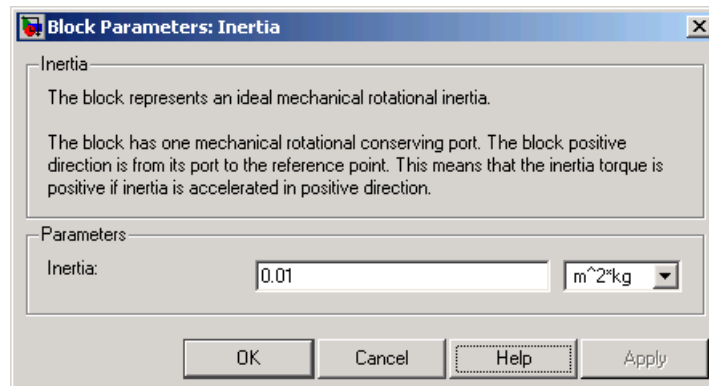
$$T = J \frac{d\omega}{dt}$$

where

T	Inertia torque
J	Inertia
ω	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

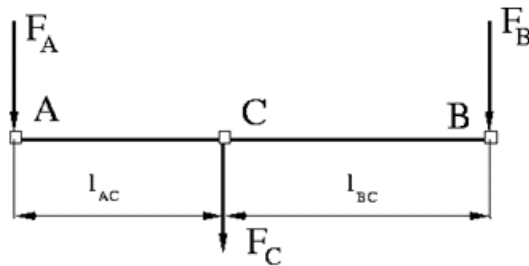
Mass

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

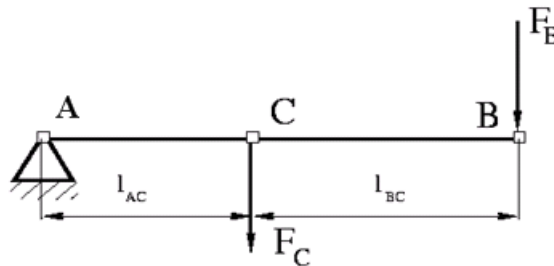
v_A, v_B, v_C Lever joints velocities

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.



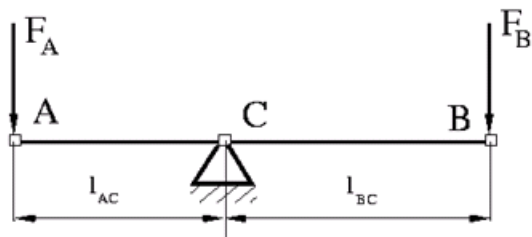
It is described with the following equations:

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

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

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

Lever



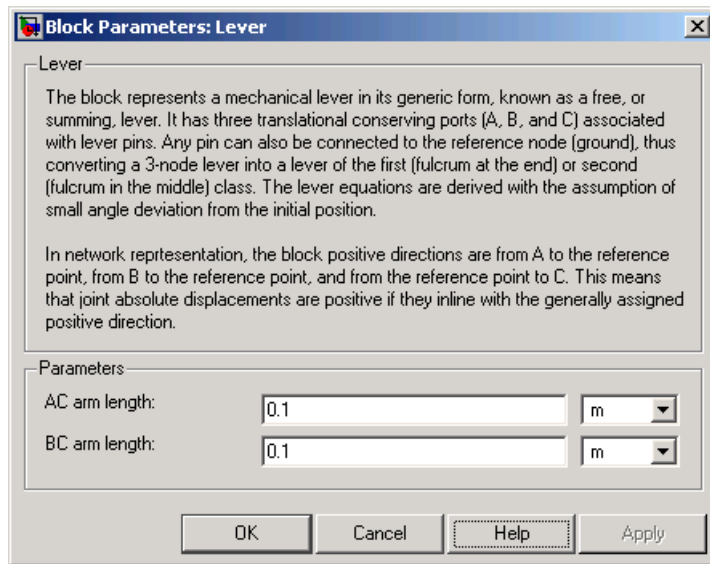
It is described with the following equations:

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

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

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

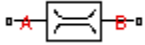
Examples

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 fulcrum at the end. Linkage L_2 represents a summing lever. Linkage L_3 simulates a second class lever with fulcrum in the middle.

Purpose Simulate hydraulic pipeline with linear resistance losses

Library Hydraulic Elements

Description

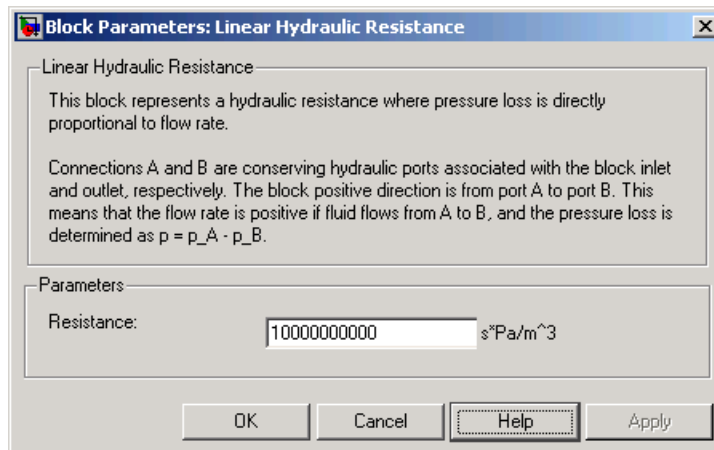


The Linear Hydraulic Resistance block represents a hydraulic resistance where pressure loss is directly proportional to flow rate. This block can be useful at preliminary stages of development, or as a powerful means to speed up the simulation, especially if the flow rate varies insignificantly with 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/m³/s.

Ports The block has the following ports:

Linear Hydraulic Resistance

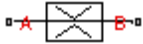
- A Hydraulic conserving port associated with the resistance inlet.
- B Hydraulic conserving port associated with the resistance outlet.

See Also Resistive Tube

Purpose Simulate all kinds of hydraulic resistances specified by a loss coefficient

Library Local Hydraulic Resistances

Description



The Local Resistance block represents a local hydraulic resistance, such as a bend, elbow, fitting, filter, local change in flow cross-section, and so on. The resistances of this type are also often referred to as minor losses. The resistance induces pressure loss caused by the change in fluid velocity profile and mainly depends on the element configuration and fluid properties. The block computes pressure loss by using semi-empirical formula based on the pressure loss coefficient. This coefficient is usually provided in catalogs, data sheets, or hydraulic textbooks as a scalar or as a table-specified relationship between the coefficient and the Reynolds number (for example, see [1]).

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.

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 pressure loss is determined according to the following equations:

$$p = \begin{cases} K \frac{\rho}{2A^2} q |q| & \text{for } Re \geq Re_{cr} \\ K \frac{Re_{cr} \cdot \nu \cdot \rho}{2d \cdot A} q & \text{for } Re < Re_{cr} \end{cases}$$

$$p = p_A - p_B$$

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

where

Local Resistance

q	Flow rate
p	Pressure loss
p_A, p_B	Gage pressures at the block terminals
K	Pressure loss coefficient
A	Passage area
ρ	Fluid density
ν	Fluid kinematic viscosity

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

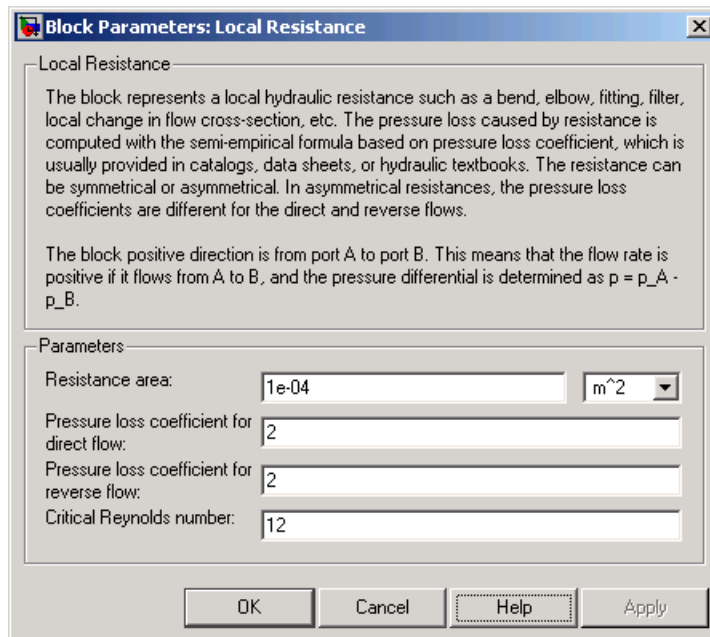
The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

Basic Assumptions and Limitations

The model is based on the following 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



Resistance area

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

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 [2]). The default value is 2.

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.

Local Resistance

Critical Reynolds number

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

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

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

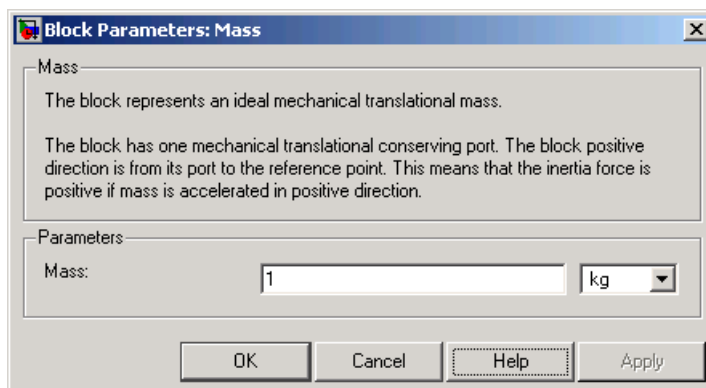
m Mass

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

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

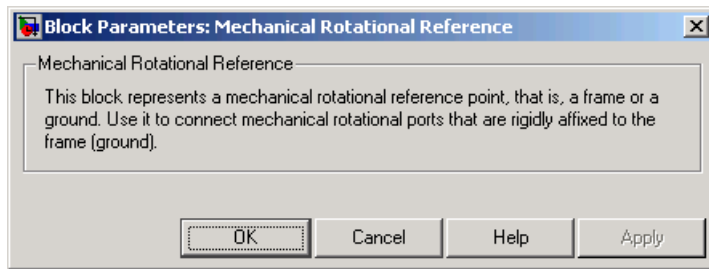
Library Mechanical Rotational Elements

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

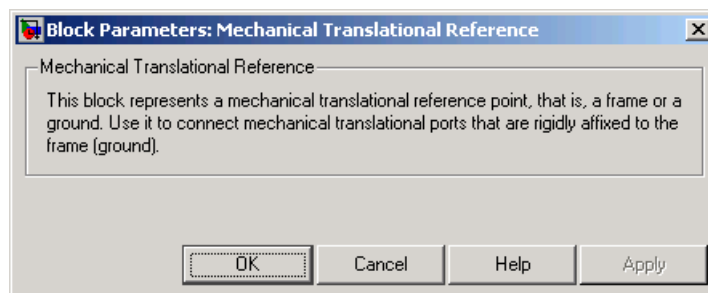
Purpose Simulate reference for mechanical translational ports

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.

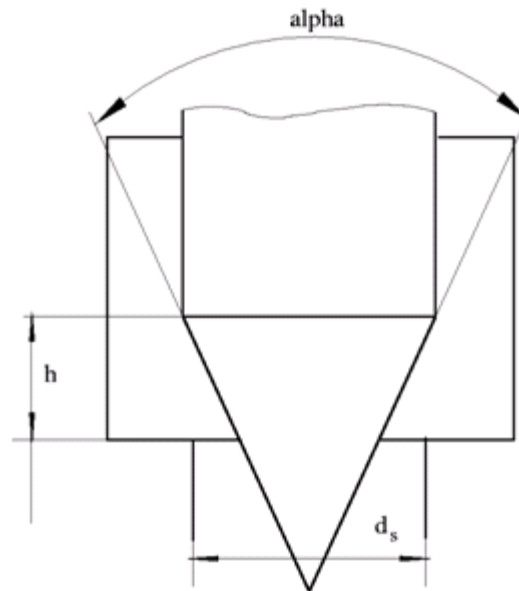
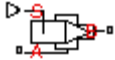
Ports The block has one mechanical translational port.

See Also Mechanical Rotational Reference
Hydraulic Reference

Purpose Simulate hydraulic needle valve

Library Flow Control Valves

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



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

Needle Valve

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

$$h = x_0 + x$$

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

$$p = p_A - p_B$$

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

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

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

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

where

q Flow rate

ρ Pressure differential

p_A, p_B Gage pressures at the block terminals

C_D	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
x_0	Initial opening
x	Needle displacement from initial position
h	Valve opening
d_s	Orifice diameter
α	Needle angle
ρ	Fluid density
D_H	Valve instantaneous hydraulic diameter
ν	Fluid kinematic viscosity
A_{leak}	Closed valve leakage area
A_{max}	Maximum valve open area

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

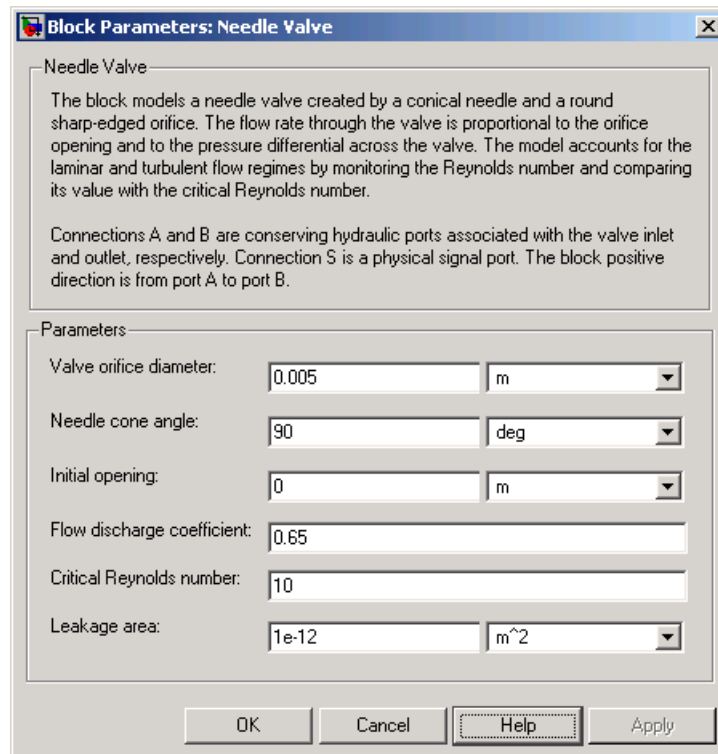
Basic Assumptions and Limitations

The model is based on the following assumptions:

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

Needle Valve

Dialog Box and Parameters



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.

Flow discharge coefficient

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

Critical Reynolds number

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

Leakage area

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

Global Parameters

Fluid density

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

Fluid kinematic viscosity

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

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

Needle Valve

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

See Also

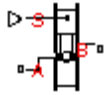
- Ball Valve
- Poppet Valve
- Pressure-Compensated Flow Control Valve

Orifice with Variable Area Round Holes

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

Library Hydraulic Elements

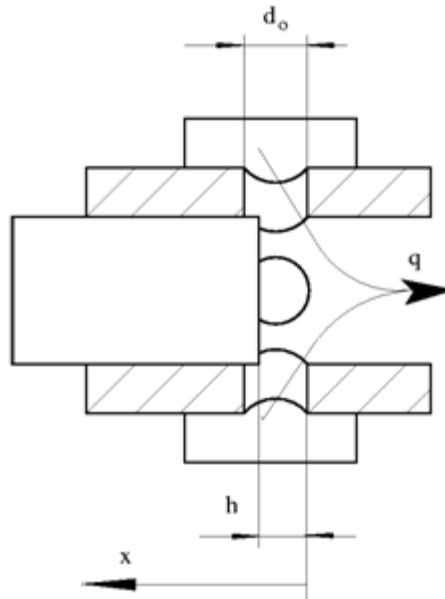
Description



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_o Orifice hole diameter

Orifice with Variable Area Round Holes



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

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

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

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 Gage pressures at the block terminals
- C_D Flow discharge coefficient
- $A(h)$ Instantaneous orifice passage area
- d_0 Hole diameter
- z Number of holes
- x_0 Initial opening

Orifice with Variable Area Round Holes

x	Spool displacement from initial position
h	Orifice opening
or	Orifice orientation indicator. The variable assumes +1 value if a spool displacement in the globally assigned positive direction opens the orifice, and -1 if positive motion decreases the opening.
ρ	Fluid density
D_H	Instantaneous orifice hydraulic diameter
ν	Fluid kinematic viscosity
A_{leak}	Closed orifice leakage area
A_{max}	Fully-open orifice passage area

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

Basic Assumptions and Limitations

The model is based on the following assumptions:

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

Orifice with Variable Area Round Holes

Dialog Box and Parameters

Block Parameters: Orifice with Variable Area Round Holes

Orifice with Variable Area Round Holes

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

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

Parameters

Diameter of round holes:

Number of round holes:

Flow discharge coefficient:

Initial opening:

Orifice orientation:

Critical Reynolds number:

Leakage area:

OK Cancel Help Apply

Diameter of round holes

Diameter of the orifice holes. The default value is 5×10^{-3} m.

Number of round holes

Number of holes. The default value is 6.

Flow discharge coefficient

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

Orifice with Variable Area Round Holes

Initial opening

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

Orifice orientation

The parameter is introduced to specify the effect of the orifice control member motion on the valve opening. The parameter can be set to one of two options: Opens in positive direction or Opens in negative direction. The value Opens in positive direction specifies an orifice whose control member opens the valve when it is shifted in the globally assigned positive direction. The parameter is extremely useful when multi-orifice valve is built 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

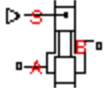
Orifice with Variable Area Slot

Orifice with Variable Area Slot

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	Gage pressures at the block terminals
C_D	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
b	Width of the orifice slot
x_0	Initial opening
x	Spool displacement from initial position
h	Orifice opening
or	Orifice orientation indicator. The variable assumes +1 value if a spool displacement in the globally assigned positive direction opens the orifice, and -1 if positive motion decreases the opening.
ρ	Fluid density
D_H	Instantaneous orifice hydraulic diameter
ν	Fluid kinematic viscosity
A_{leak}	Closed orifice leakage area

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

Orifice with Variable Area Slot

Basic Assumptions and Limitations

The model is based on the following assumptions:

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

Dialog Box and Parameters

Block Parameters: Orifice with Variable Area Slot

Orifice with Variable Area Slot:

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

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

Parameters:

Orifice width: 0.01 m

Flow discharge coefficient: 0.7

Initial opening: 0 m

Orifice orientation: Opens in positive direction

Critical Reynolds number: 12

Leakage area: 1e-12 m²

OK Cancel Help Apply

Orifice width

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

Flow discharge coefficient

Semi-empirical parameter for orifice capacity characterization.

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

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

Initial opening

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

Orifice orientation

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

Critical Reynolds number

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

Leakage area

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

Orifice with Variable Area Slot

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

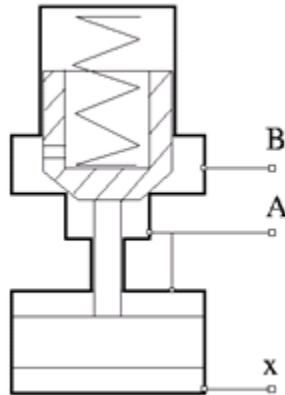
Orifice with Variable Area Round Holes

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 Gage pressures at the valve terminals

p_X Gage pressure at the pilot terminal

A_A Area of the spool in the A chamber

Pilot-Operated Check Valve

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:

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

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

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

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

$$p = p_A - p_B$$

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

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

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

where

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

Pilot-Operated Check Valve

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

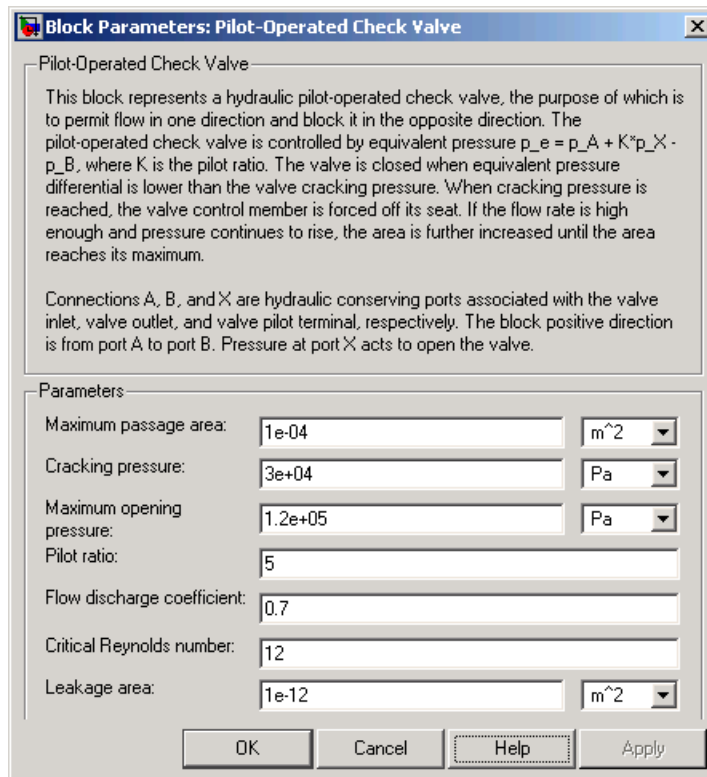
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as $p = p_A - p_B$.

Basic Assumptions and Limitations

The model is based on the following assumptions:

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

Dialog Box and Parameters



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-Operated Check Valve

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.

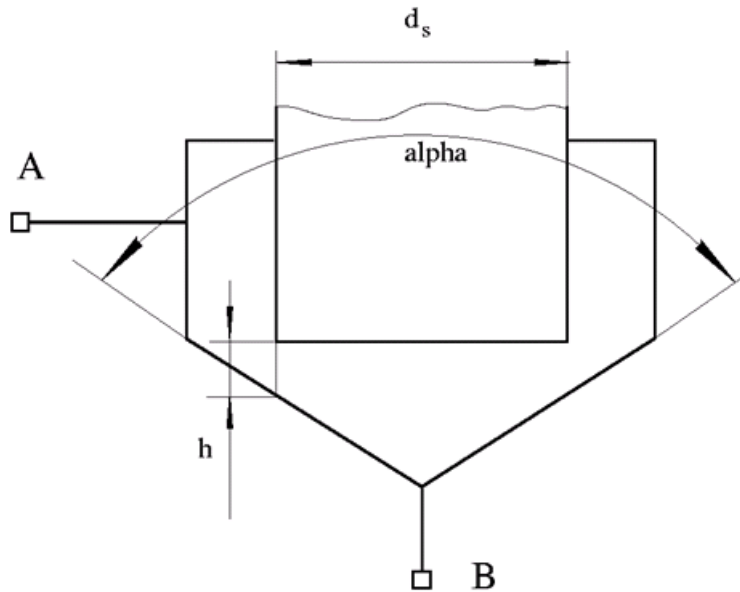
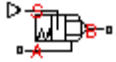
See Also Check Valve

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

ρ Pressure differential

p_A, p_B Gage 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
α	Cone angle
ρ	Fluid density
D_H	Valve instantaneous hydraulic diameter
ν	Fluid kinematic viscosity
A_{leak}	Closed valve leakage area
A_{max}	Maximum valve open area

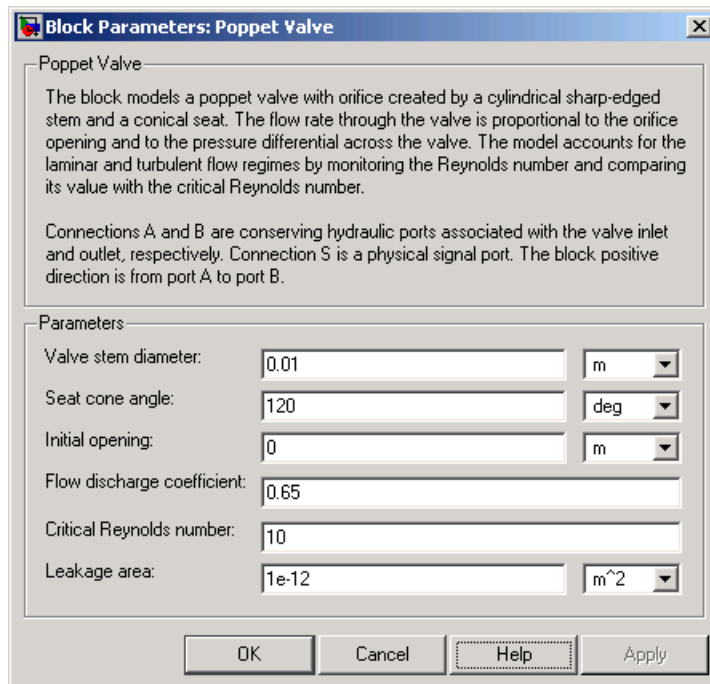
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as $p = p_A - p_B$. Positive signal at the physical signal port S opens the valve.

Basic Assumptions and Limitations

The model is based on the following assumptions:

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

Dialog Box and Parameters



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

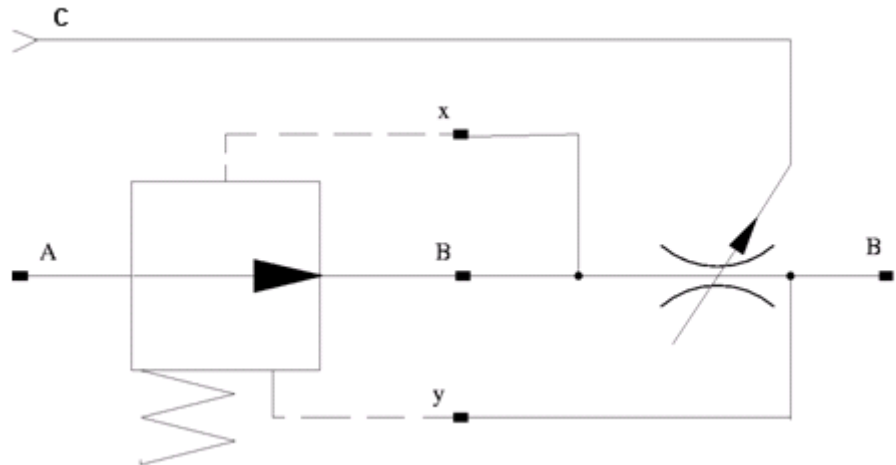
Pressure-Compensated Flow Control Valve

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.

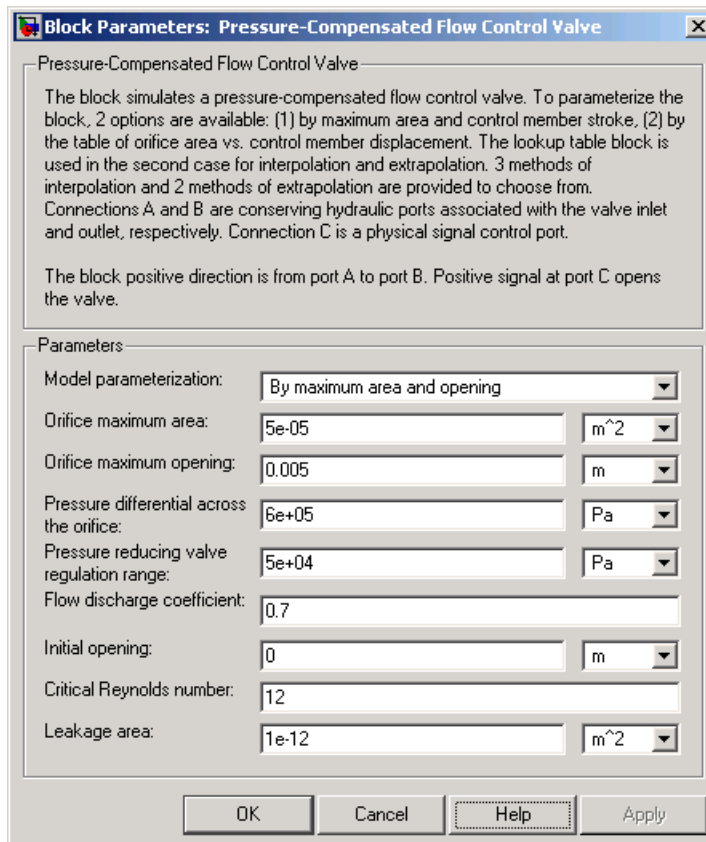
Assumptions and Limitations

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.

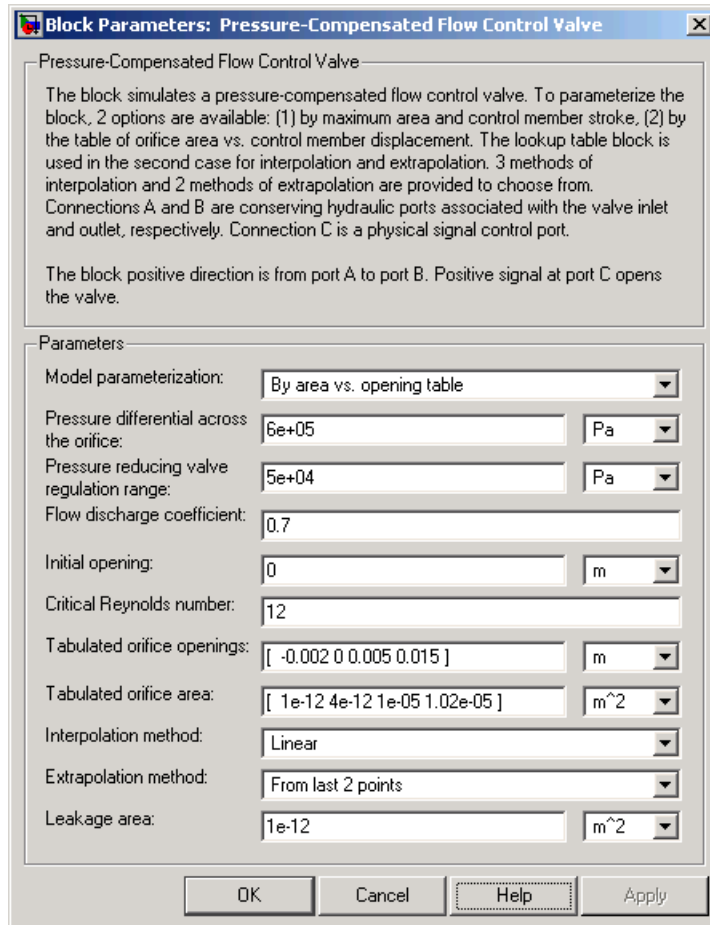
The block positive direction is from port A to port B. Positive signal at port C opens the valve.

Parameters:

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

OK Cancel Help Apply

Pressure-Compensated Flow Control Valve



Model parameterization

Select one of the following methods for specifying the orifice:

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

Pressure-Compensated Flow Control Valve

of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.

- By area vs. opening table—Provide tabulated data of orifice openings and corresponding orifice areas. The passage area is determined by one-dimensional table lookup. You have a choice of three interpolation methods and two extrapolation methods.

Orifice maximum area

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

Orifice maximum opening

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

Tabulated orifice openings

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

Tabulated orifice area

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

Pressure-Compensated Flow Control Valve

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

Interpolation method

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

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

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

Extrapolation method

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

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

Pressure-Compensated Flow Control Valve

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

Pressure differential across the orifice

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

Pressure reducing valve regulation range

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

Flow discharge coefficient

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

Initial opening

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

Critical Reynolds number

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

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Extreme caution should be exercised if the parameter is set to 0. The default value is $1e-12$ m².

Pressure-Compensated Flow Control Valve

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

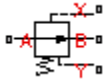
Poppet Valve

Pressure Compensator

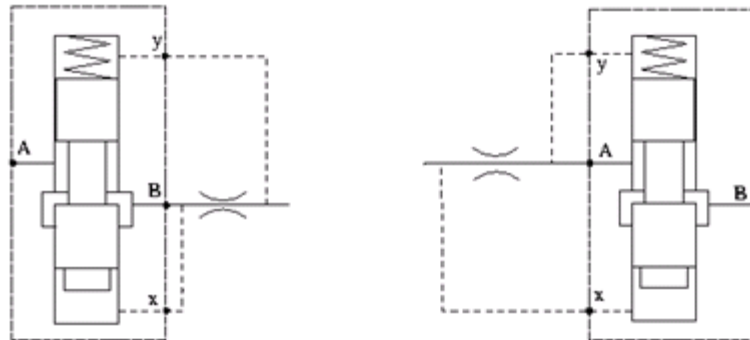
Purpose Simulate hydraulic pressure compensating valve

Library Hydraulic Elements

Description



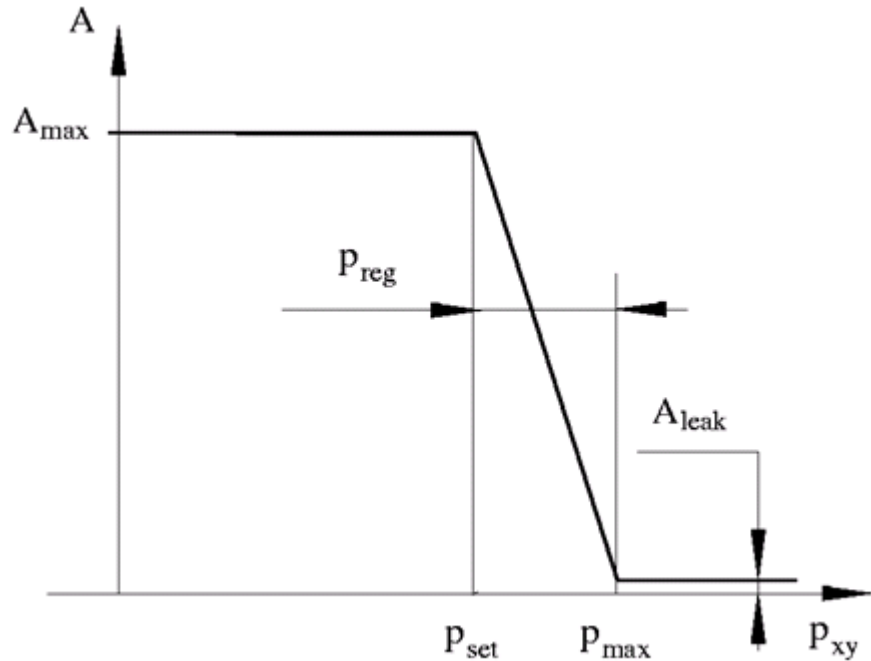
The Pressure Compensator block represents a hydraulic pressure compensating valve, or pressure compensator. Pressure compensators are used to maintain preset pressure differential across a hydraulic component to minimize the influence of pressure variation on a flow rate passing through the component.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

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

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

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

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

$$p = p_A - p_B$$

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

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

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

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

where

q	Flow rate
p	Pressure differential across the valve
p_{xy}	Pressure differential across valve control terminals
p_A, p_B	Gage pressures at the valve main terminals
p_x, p_y	Gage pressures at the valve control terminals
p_{set}	Valve preset pressure
p_{max}	Pressure needed to fully close the orifice
p_{reg}	Regulation range
$A(h)$	Instantaneous orifice passage area
A_{max}	Orifice maximum area
C_D	Flow discharge coefficient
ρ	Fluid density
D_H	Instantaneous orifice hydraulic diameter
ν	Fluid kinematic viscosity
A_{leak}	Closed orifice leakage area

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

Pressure Compensator

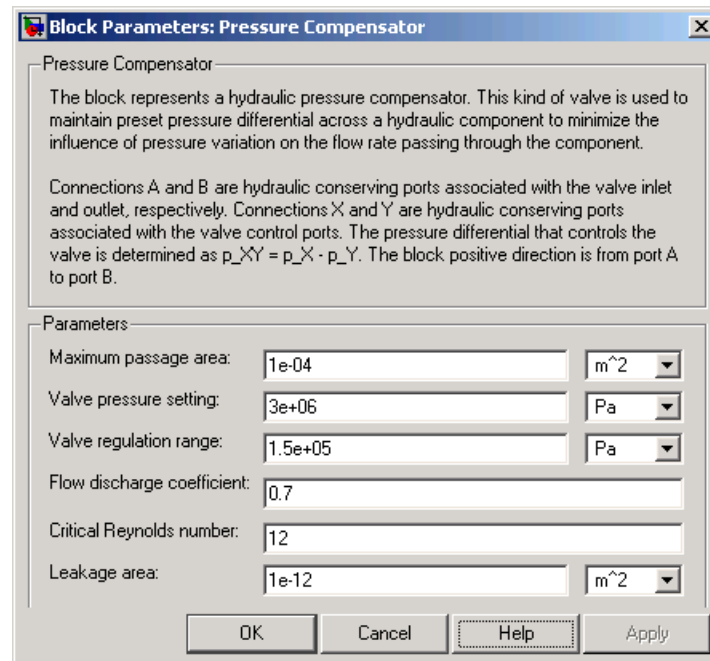
measured as $P_{xy} = P_x - P_y$, and it creates a force acting against the spring preload.

Assumptions and Limitations

The model is based on the following assumptions:

- Valve opening is linearly proportional to the pressure differential.
- No loading on the valve, such as inertia, friction, spring, and so on, is considered.
- Flow consumption associated with the spool motion is neglected.

Dialog Box and Parameters



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

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.

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

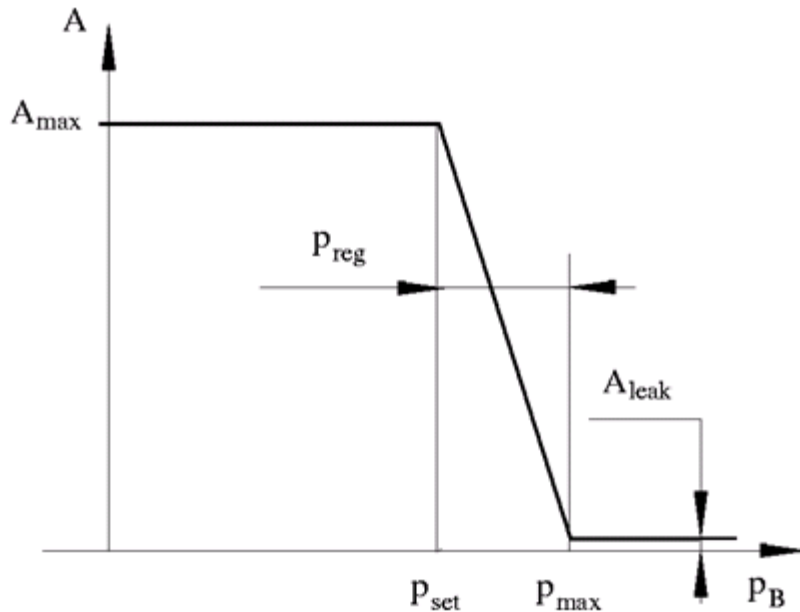
Purpose Simulate pressure control valve maintaining reduced pressure in a portion of system

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.



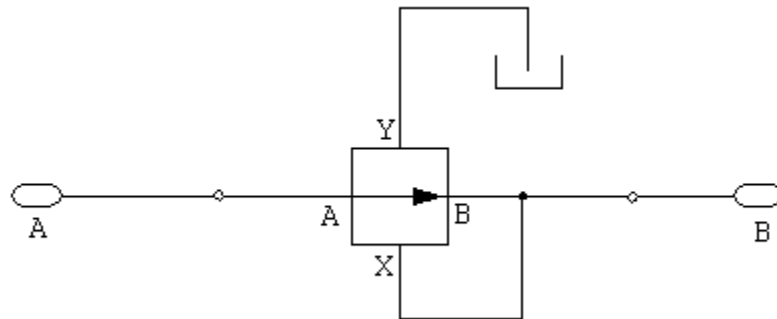
The pressure-reducing valve is a normally-open valve and it remains fully open while outlet pressure is lower than the valve preset pressure. When the preset pressure is reached, the valve control member (spool, ball, poppet, etc.) is forced off its stop and starts closing the orifice, thus

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.

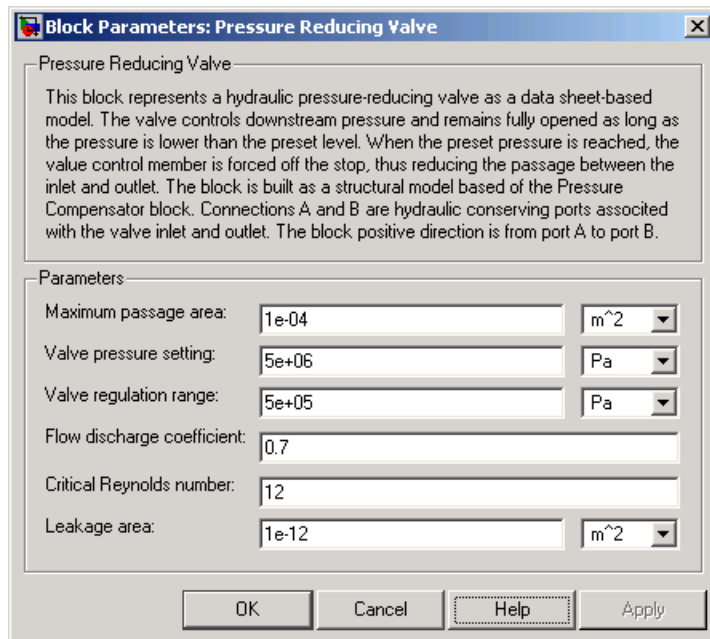
In addition to the maximum area, the leakage area is also required to characterize the valve. The main purpose of the parameter is not to account for possible leakage, even though this is also important, but to maintain numerical integrity of the circuit by preventing a portion of the system from getting 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 20×10^5 Pa and maintains this pressure downstream, as long as the upstream pressure is higher than this setting.

See Also

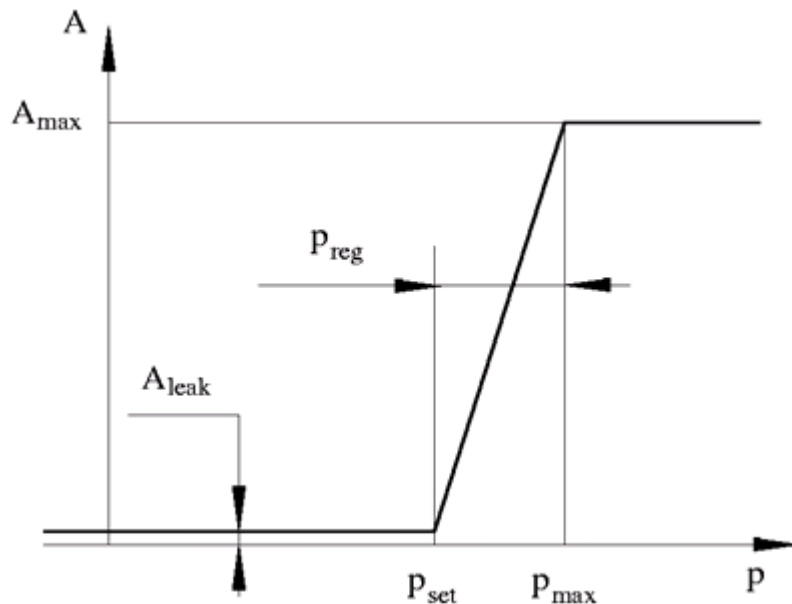
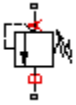
Pressure Compensator

Pressure Relief Valve

Purpose Simulate pressure control valve maintaining preset pressure in a 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:

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

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

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

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	Gage pressures at the block terminals
C_D	Flow discharge coefficient
$A(p)$	Instantaneous orifice passage area
A_{max}	Fully-open valve passage area
A_{leak}	Closed valve leakage area
p_{reg}	Regulation range
p_{set}	Valve preset pressure
p_{max}	Valve pressure at maximum opening
D_H	Instantaneous orifice hydraulic diameter
ρ	Fluid density
ν	Fluid kinematic viscosity

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

Basic Assumptions and Limitations

The model is based on the following assumptions:

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

Dialog Box and Parameters

Block Parameters: Pressure Relief Valve

Pressure Relief Valve

This block represents a hydraulic pressure relief valve as a data sheet-based model. The valve remains closed while pressure at the valve inlet is lower than the valve preset pressure. When the preset pressure is reached, the valve control member is forced off its seat, thus creating a passage between the inlet and outlet. Some fluid is diverted to a tank through this orifice, thus reducing the pressure at the inlet. If this flow rate is not enough and pressure continues to rise, the area is further increased until the control member reaches its maximum.

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

Parameters

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

OK Cancel Help Apply

Pressure Relief Valve

Maximum passage area

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

Valve pressure setting

Preset pressure level, at which the orifice of the valve starts to open. The default value is $50e5 \text{ Pa}$.

Valve regulation range

Pressure increase over the preset level needed to fully open the valve. The default value is $5e5 \text{ Pa}$.

Flow discharge coefficient

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

Critical Reynolds number

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

Leakage area

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

Global Parameters

Fluid density

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

Fluid kinematic viscosity

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

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

Examples

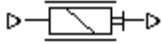
The Power Unit with Fixed-Displacement Pump demo (sh_power_unit_fxd_dsp1_pump) illustrates the use of the Pressure Relief Valve block in hydraulic systems. The valve is set to $75e5$ Pa and starts diverting fluid to tank as soon as the pressure at its inlet reaches this value.

Proportional and Servo-Valve Actuator

Purpose Simulate continuous valve driver with the output proportional to the input signal

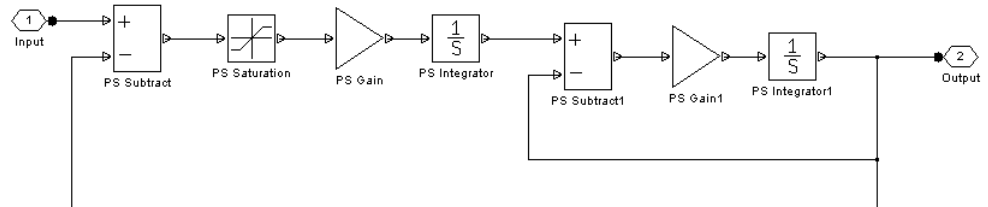
Library Valve Actuators

Description



The Proportional and Servo-Valve Actuator block represents an electromagnetic actuator that is used in proportional and servo-valves to drive a spool or other working member. The block is intended to work with one of the directional valve models to form a desirable configuration of a proportional or servo-valve. The block is implemented as a data sheet-based model and reproduces only the input/output relationship, or the actuator's transient response, as presented in the catalog or data sheet.

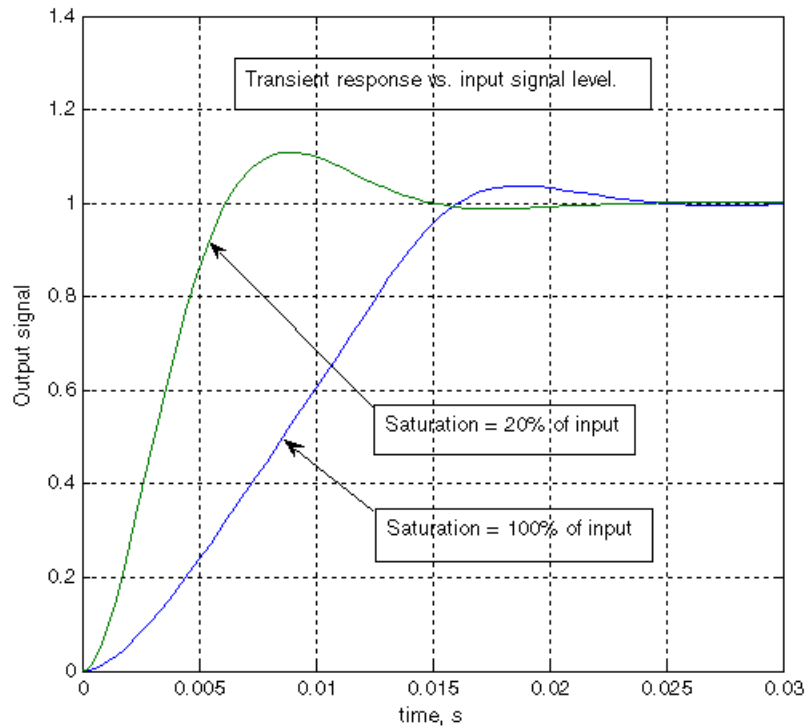
The Proportional and Servo-Valve Actuator block is built using the blocks from the Physical Signals library. Both the input and the output of the block are physical signals. The block diagram of the model is shown in the following figure.



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

The typical transient responses of a servo-valve or a high-quality proportional valve are shown in the following figure. The only difference between the two responses in the figure is the value of the saturation. The response that corresponds to 100% of the input signal is considerable slower than that with the 20% saturation.

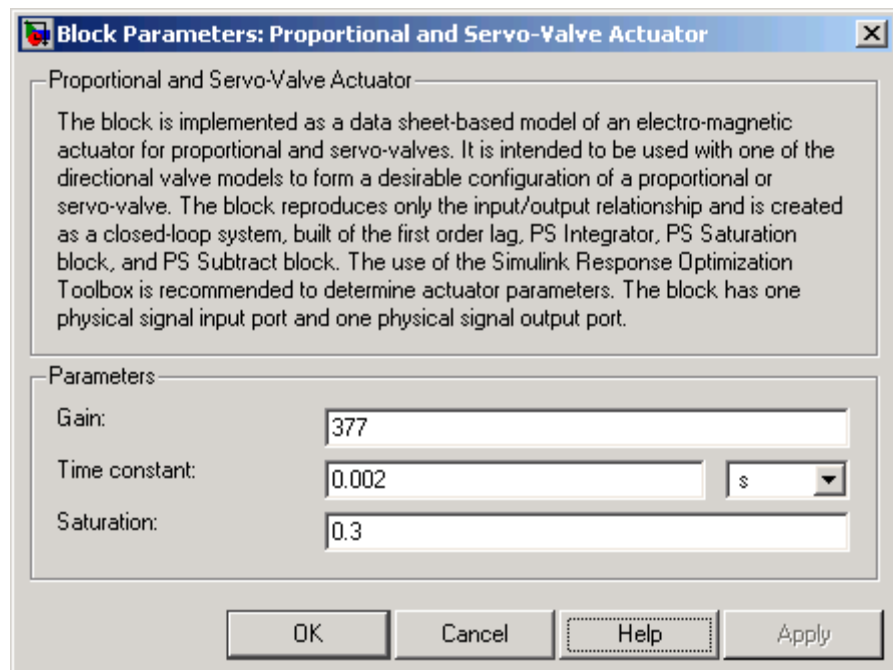
Proportional and Servo-Valve Actuator



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

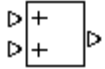
3-Position Valve Actuator

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:



$$y = u_1 + u_2$$

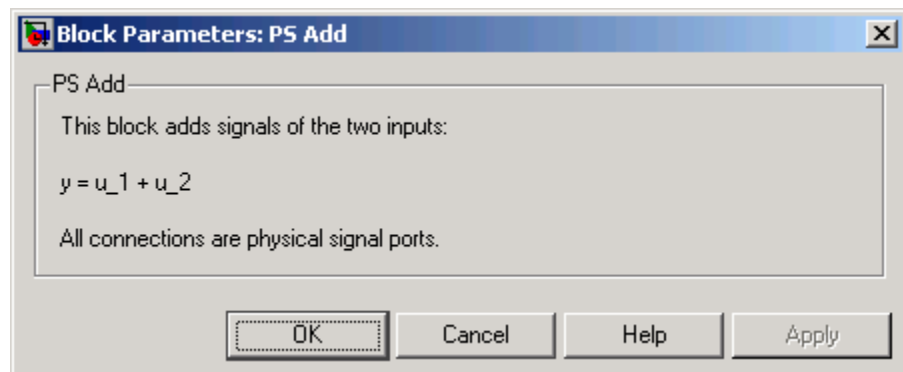
where

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.

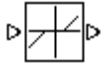
Ports The block has two physical signal input ports and one physical signal output port.

See Also PS Subtract

Purpose Provide a region of zero output for physical signals

Library Physical Signals/Nonlinear Operators

Description



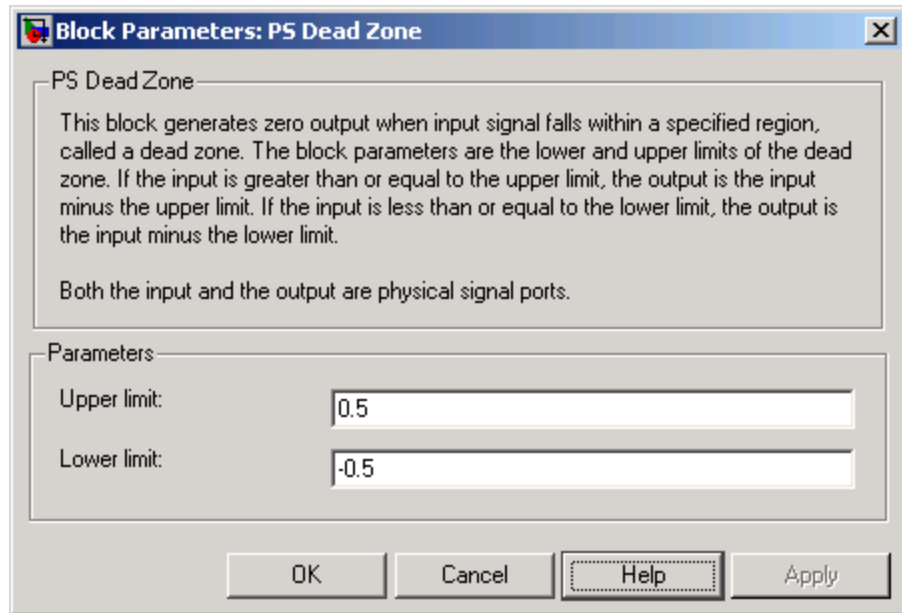
The PS Dead Zone block generates zero output when input signal falls within a specified region, called a dead zone. You can specify the lower and upper limits of the dead zone as block parameters. The block output depends on the input and dead zone:

- 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.

PS Dead Zone

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

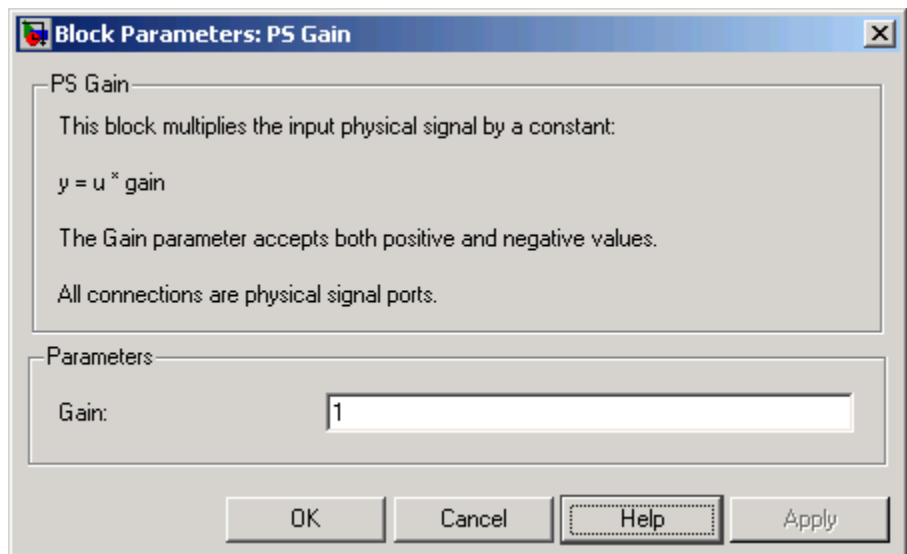
Purpose Multiply the physical signal input by a constant

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.

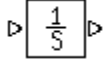
Ports The block has one physical signal input port and one physical signal output port.

PS Integrator

Purpose Integrate a 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:

$$x = y(t)$$

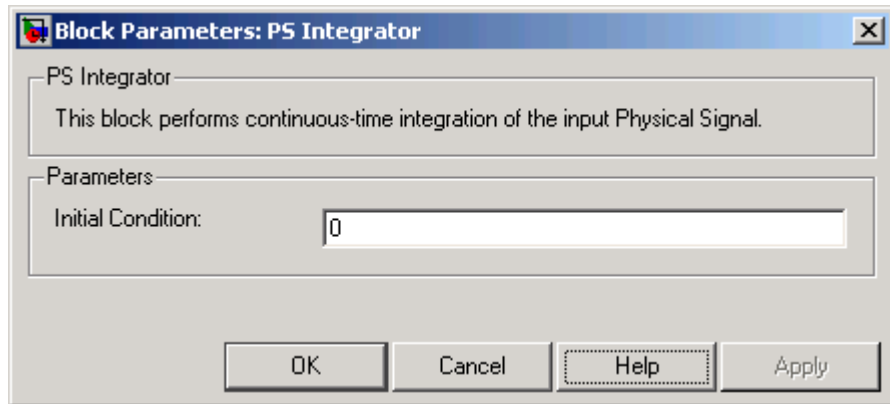
$$x_0 = y_0$$

$$\dot{x} = u(t)$$

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. The default value is 0.

Ports

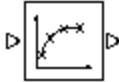
The block has one physical signal input port and one physical signal output port.

PS Lookup Table (1D)

Purpose Approximate one-dimensional function using the 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.

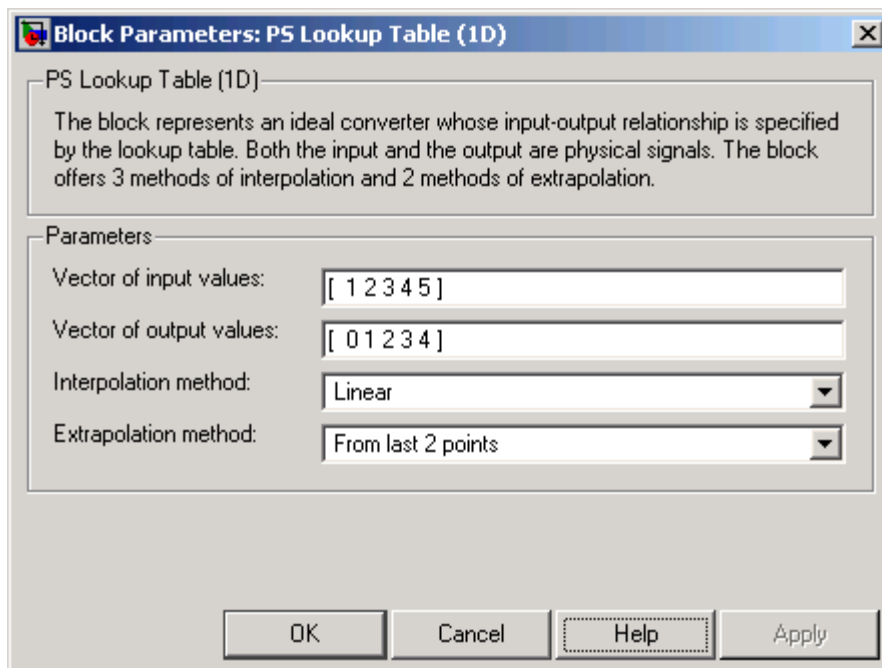


Note To map two physical signal inputs to an output, use the PS Lookup Table (2-D) block.

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 the 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.

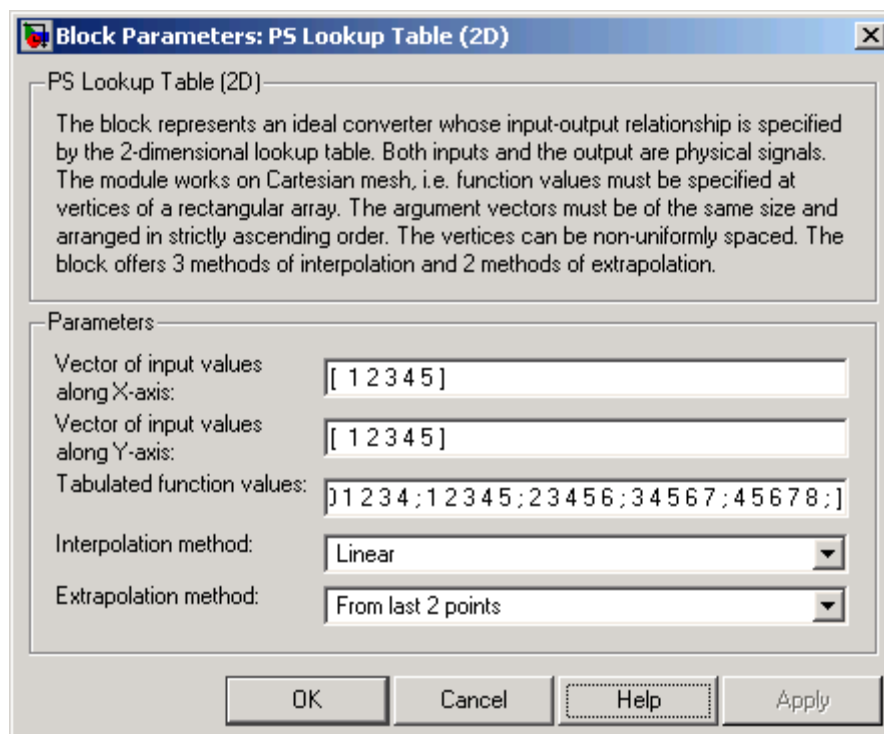


The x and y data vectors must be *strictly monotonically increasing* (i.e., the value of the next element in the vector is greater than the value of the preceding element). The matrix size of the tabulated function values must match the dimensions defined by the input vectors.

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

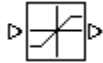
See Also

PS Lookup Table (1D)

Purpose Limit the range of a physical signal

Library Physical Signals/Nonlinear Operators

Description

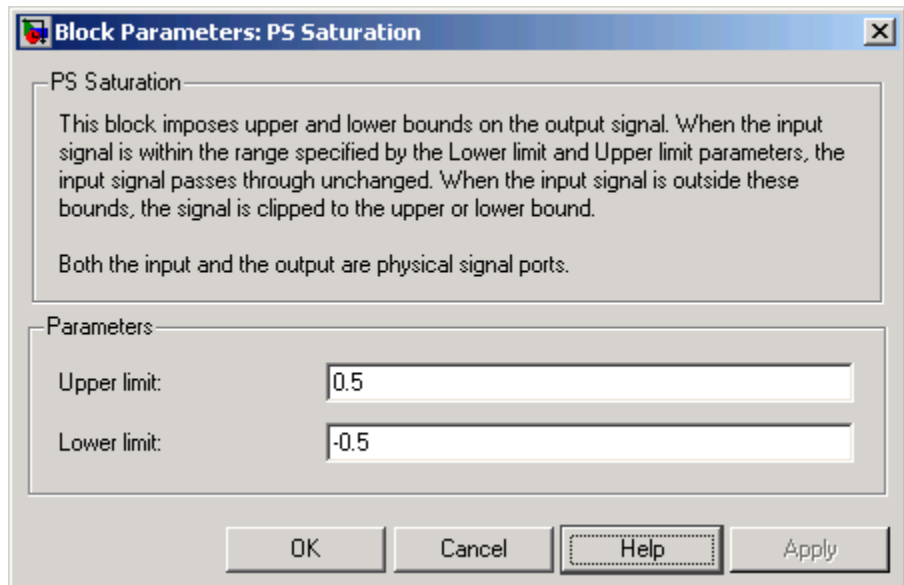


The PS Saturation block imposes upper and lower bounds on a physical signal. When the input signal is within the range specified by the **Lower limit** and **Upper limit** parameters, the input signal passes through unchanged. When the input signal is outside these bounds, the signal is clipped to the 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.

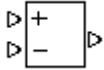
See Also

PS Dead Zone

Purpose Compute the simple subtraction of the two input physical signals

Library Physical Signals/Functions

Description The PS Subtract block subtracts one physical signal input from another and outputs the difference:



$$y = u_1 - u_2$$

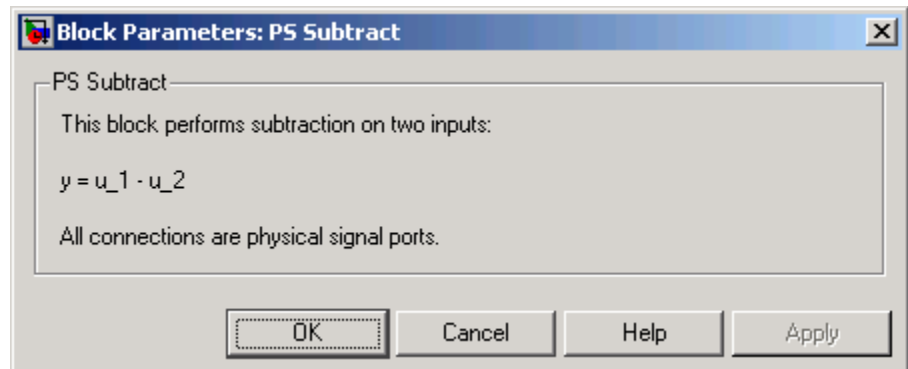
where

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.

See Also PS Add

PS-Simulink Converter

Purpose Convert physical signal into Simulink output signal

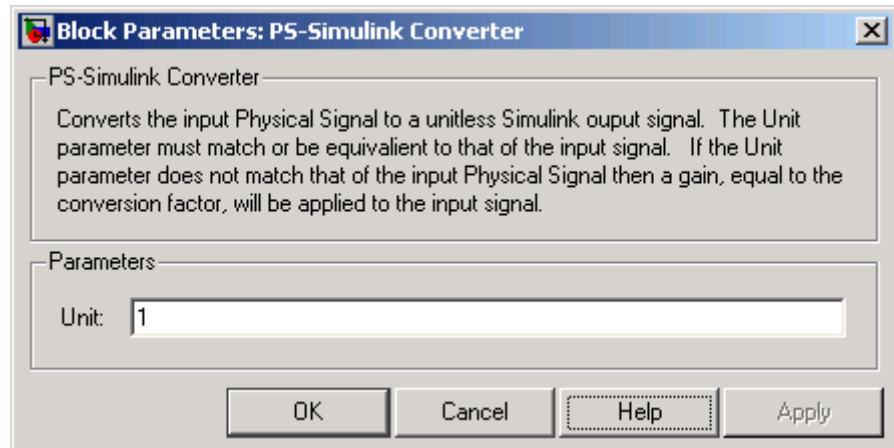
Library Utilities

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. The system compares the units you specify 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.

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).

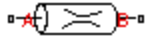
See Also Simulink-PS Converter

Resistive Tube

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 can not 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:

$$p = f \frac{(L + L_{eq})}{D_H} \frac{\rho}{2A^2} q \cdot |q|$$

$$f = \begin{cases} K_s / Re & \text{for } Re \leq Re_L \\ f_L + \frac{f_T - f_L}{Re_T - Re_L} (Re - Re_L) & \text{for } Re_L < Re < Re_T \\ \frac{1}{\left(-1.8 \log_{10} \left(\frac{6.9}{Re} + \left(\frac{r/D_H}{3.7} \right)^{1.11} \right) \right)^2} & \text{for } Re \geq Re_T \end{cases}$$

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

where

- ρ 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

Resistive Tube

r Height of the roughness on the pipe internal surface

ν Fluid kinematic viscosity

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

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.

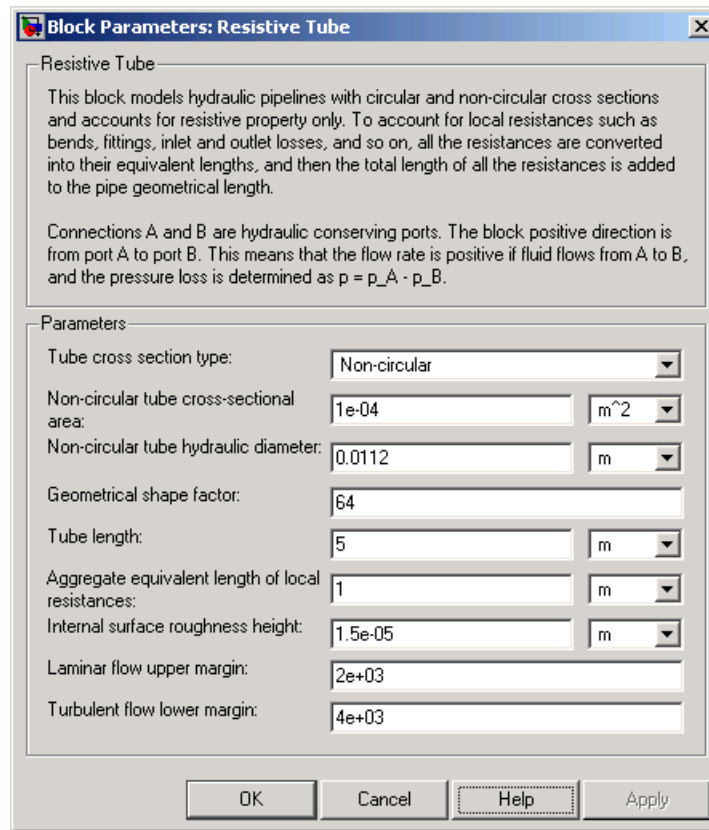
Connections A and B are hydraulic conserving ports. The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

Parameters

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

OK Cancel Help Apply

Resistive Tube



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.

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.

Resistive Tube

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

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

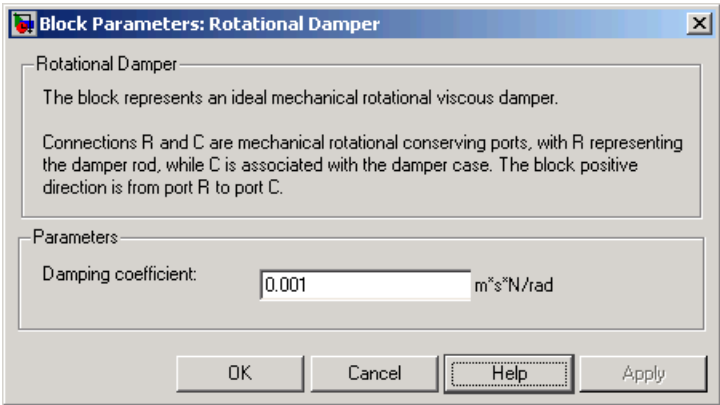
D Damping (viscous friction) coefficient

ω Relative angular velocity

ω_R, ω_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



Rotational Damper

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

Purpose

Simulate friction in contact between rotating bodies

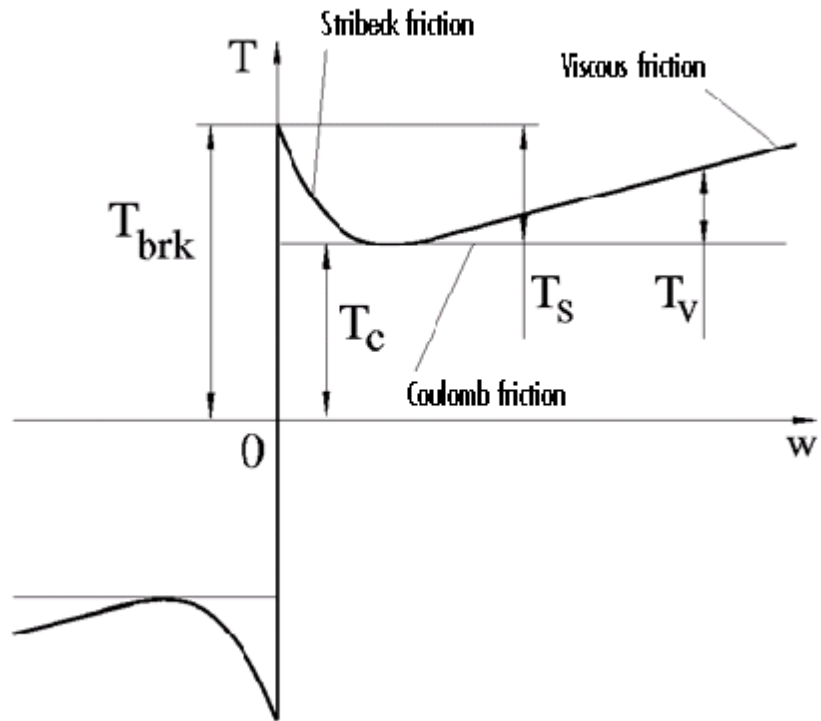
Library

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

Rotational Friction

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:

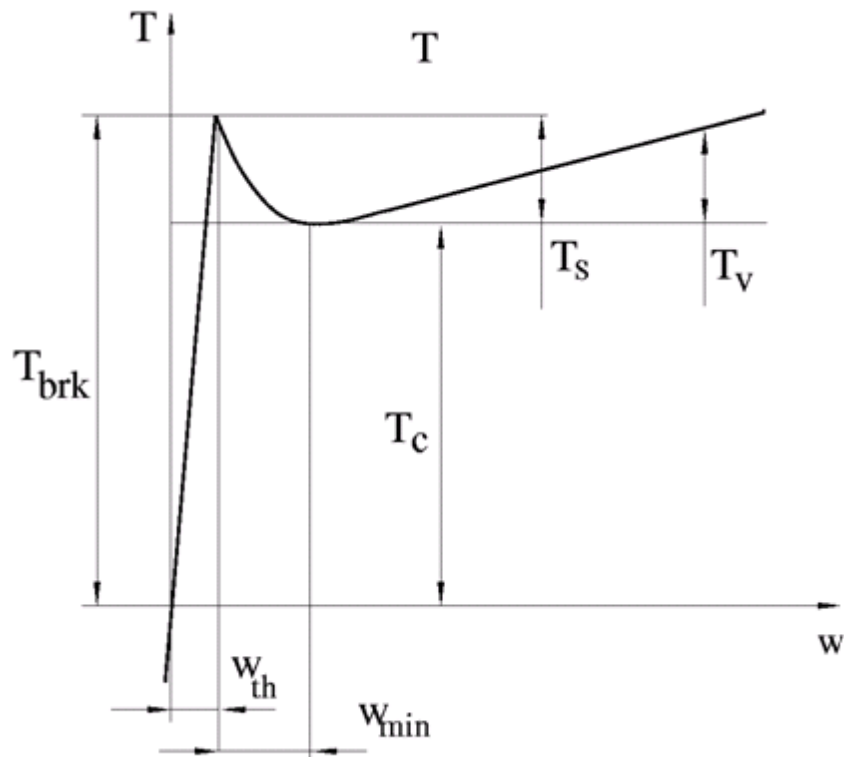
$$T = (T_C + (T_{brk} - T_C) \cdot \exp(-c_v |\omega|)) \text{sign}(\omega) + f\omega$$

$$\omega = \omega_R - \omega_C$$

where

T	Friction torque
T_C	Coulomb friction torque
T_{brk}	Breakaway friction torque
c_v	Coefficient
ω	Relative velocity
ω_R, ω_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 can not 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.



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}/ω_{th} , where ω_{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 very small velocity proportional to acting torque.

Rotational Friction

As a result of introducing the velocity threshold, the block equations are slightly modified:

- If $|\omega| \geq \omega_{th}$,

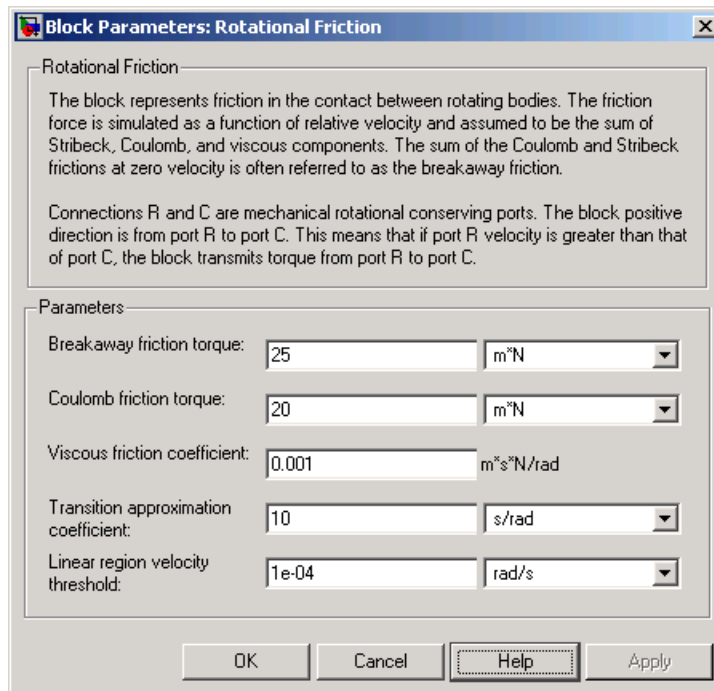
$$T = (T_C + (T_{brk} - T_C) \cdot \exp(-c_v |\omega|)) \text{sign}(\omega) + f\omega$$

- 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.

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).

Rotational 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 $c_v \approx 4/\omega_{min}$, where ω_{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. It is recommended to 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

See Also

Rotational Damper

Rotational Hard Stop

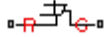
Rotational Spring

Rotational Hard Stop

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 non-elastic 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
Relative angular displacement between the slider and the case

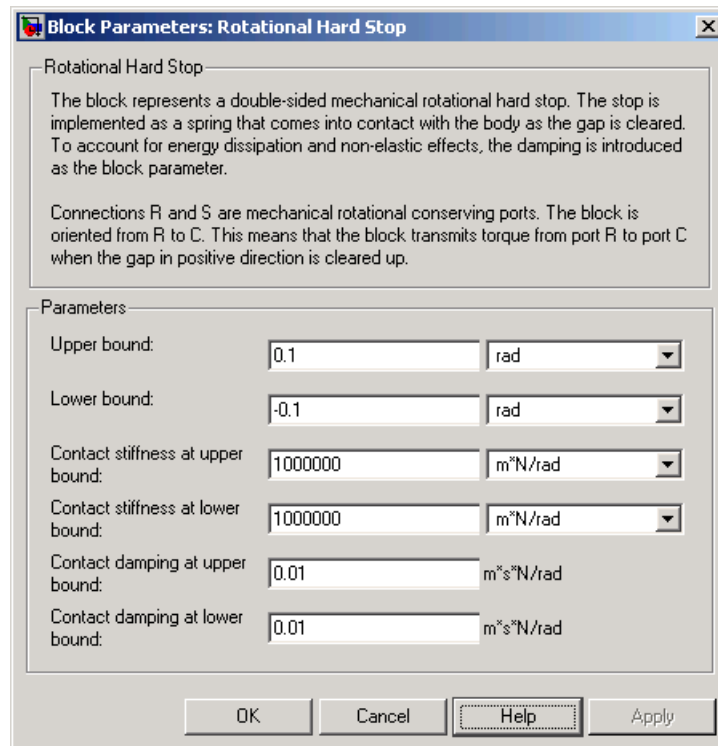
g_p	Gap between the slider and the case in positive direction
g_n	Gap between the slider and the case in negative direction
ω_R, ω_C	Absolute angular velocities of terminals R and C, respectively
ϕ_R, ϕ_C	Absolute angular displacements of terminals R and C, respectively
K_p	Contact stiffness at positive 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.

Rotational Hard Stop

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

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, it is recommended 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

Rotational Hard Stop

convergence reasons, it is recommended 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

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:



$$T = K \cdot \phi$$

$$\phi = \phi_{init} + \phi_R - \phi_C$$

$$\omega = \frac{d\phi}{dt}$$

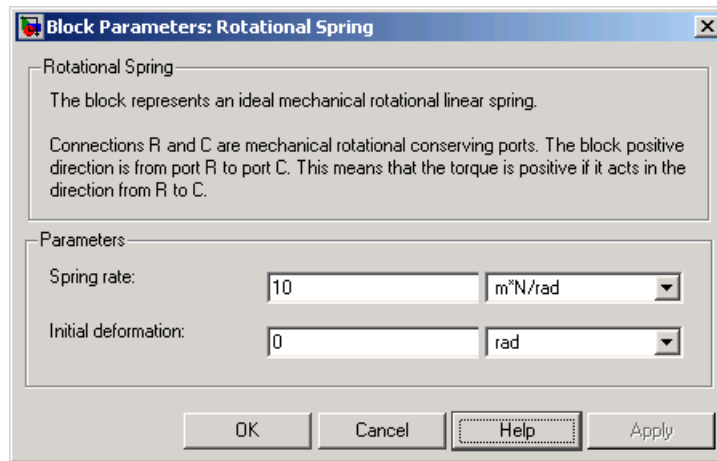
where

- T Torque transmitted through the spring
- K Spring rate
- ϕ Relative displacement angle (spring deformation)
- ϕ_{init} Spring preliminary winding (spring offset)
- ϕ_R, ϕ_C Absolute angular displacements of terminals R and C, respectively
- ω 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.

Rotational Spring

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 $\phi = \phi_{init} + \phi_R - \phi_C$, where ϕ_{init} is the initial deformation, and ϕ_R, ϕ_C are the absolute angular displacements of terminals R and C in the globally assigned positive direction. The spring can be initially compressed ($\phi_{init} > 0$) or stretched ($\phi_{init} < 0$). The default value is 0.

Ports

The block has the following ports:

R

Mechanical rotational conserving port.

C

Mechanical rotational conserving port.

See Also

Rotational Damper

Rotational Friction

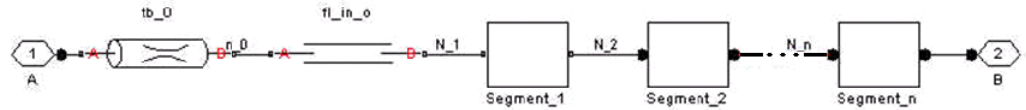
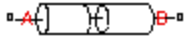
Rotational Hard Stop

Segmented Pipeline

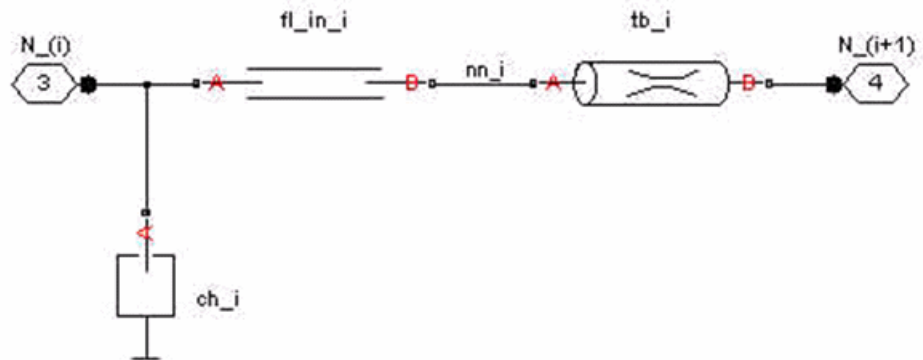
Purpose Simulate hydraulic pipeline with resistive, fluid inertia, and fluid compressibility properties

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

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

Segmented Pipeline

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

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

where

- N Number of segments
- L Pipe length
- c Speed of sound in the fluid
- ω Maximum frequency to be observed in the pipe response

The table below contains an example of simulation of a pipeline where the first 4 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 8-segmented version is used.

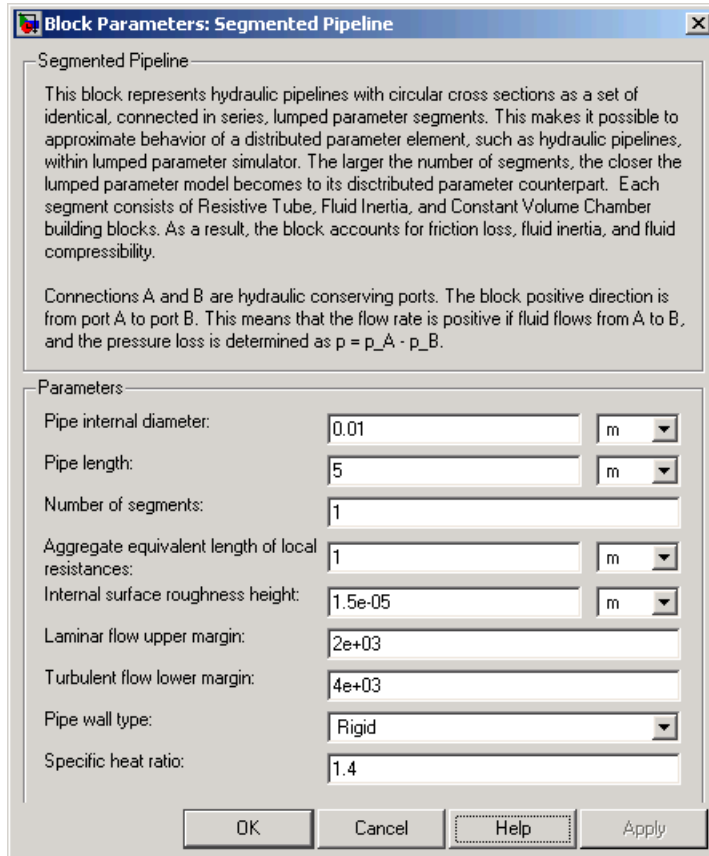
The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

Basic Assumptions and Limitations

The model is based on the following assumption:

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

Dialog Box and Parameters



Block Parameters: Segmented Pipeline

Segmented Pipeline

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

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

Parameters

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

OK Cancel Help Apply

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.

Segmented Pipeline

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.

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

Simulink-PS Converter

Purpose Convert Simulink input signal into physical signal

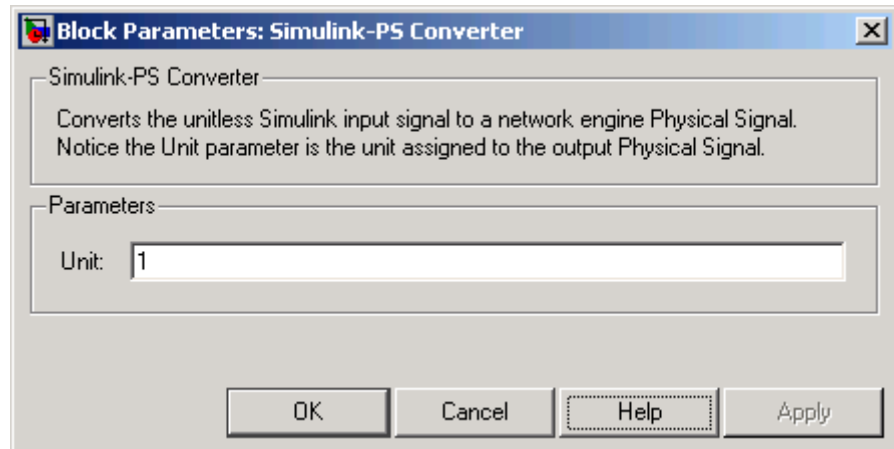
Library Utilities

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. 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).

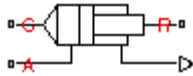
See Also PS-Simulink Converter

Single-Acting Hydraulic Cylinder

Purpose Simulate hydraulic actuator exerting force in one direction

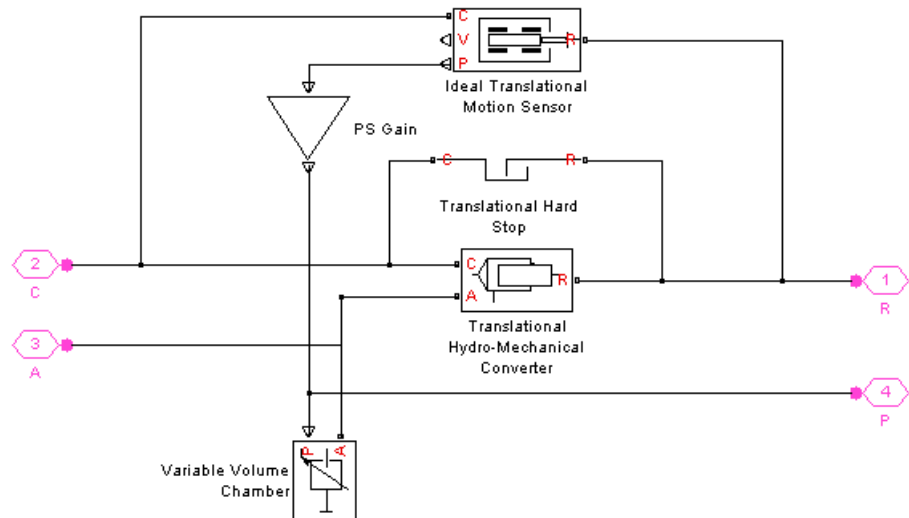
Library Cylinders

Description



The Single-Acting Hydraulic Cylinder block models a device that converts hydraulic energy into mechanical energy in the form of translational motion. Hydraulic fluid pumped under pressure into the cylinder chamber forces the piston to move and exert force on the cylinder rod. Single-acting cylinders transfer force and motion in one direction only. Use an external device, such as a spring, weight, or another opposite installed cylinder, to move the rod in opposite direction.

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



Connections R and C are mechanical translational conserving ports corresponding to the cylinder rod and cylinder clamping structure, respectively. Connection A is a hydraulic conserving port associated

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

The energy through port A is directed to the Translational Hydro-Mechanical Converter block and the Variable Volume Chamber block. The converter transforms hydraulic energy into mechanical energy, while the chamber accounts for the fluid compressibility in the cylinder chamber. The rod motion is limited with the mechanical Translational Hard Stop block in such a way that the rod can travel only between cylinder caps. The Ideal Translational Motion Sensor block in the schematic is introduced to determine an instantaneous piston position, which is necessary for the Variable Volume Chamber block.

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

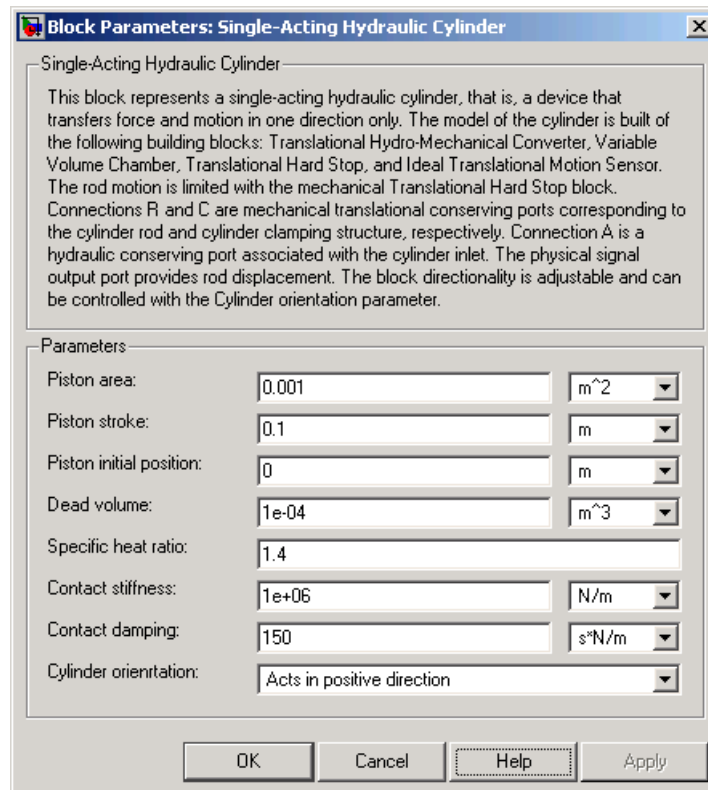
Basic Assumptions and Limitations

The model is based on the following assumptions:

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

Single-Acting Hydraulic Cylinder

Dialog Box and Parameters



Piston area

Effective piston area. The default value is 0.001 m².

Piston stroke

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

Piston initial position

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

Single-Acting Hydraulic Cylinder

Dead volume

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

Specific heat ratio

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

Contact stiffness

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

Contact damping

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

Cylinder orientation

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

Single-Acting Hydraulic Cylinder

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

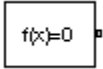
Translational Hydro-Mechanical Converter

Variable Volume Chamber

Purpose Represent the Physical Networks environment and solver configuration

Library Utilities

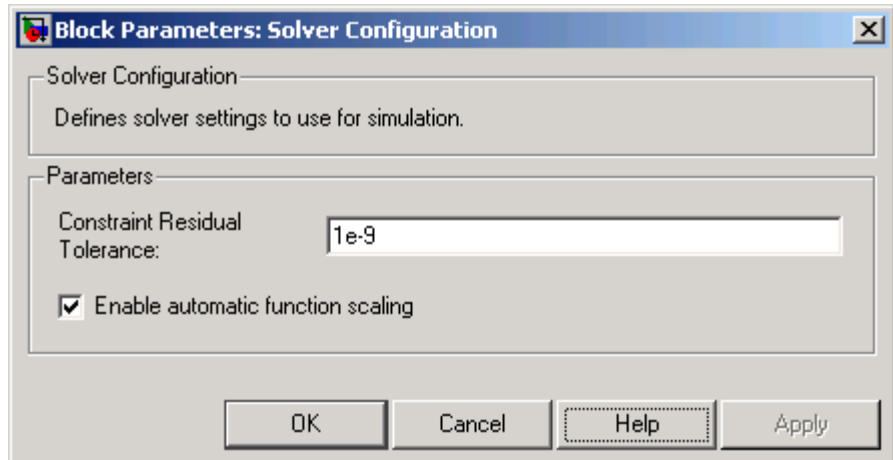
Description



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

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.

Spring-Loaded Accumulator

Purpose Simulate hydraulic accumulator with spring used for energy storage

Library Accumulators

Description



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

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

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

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

where

p Pressure at the accumulator inlet

q Flow rate into accumulator

V_{max} Accumulator capacity (maximum volume)

V_F Instantaneous volume of fluid in the accumulator

p_{pr} Preload pressure

p_{max} Pressure needed to fully fill the accumulator

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

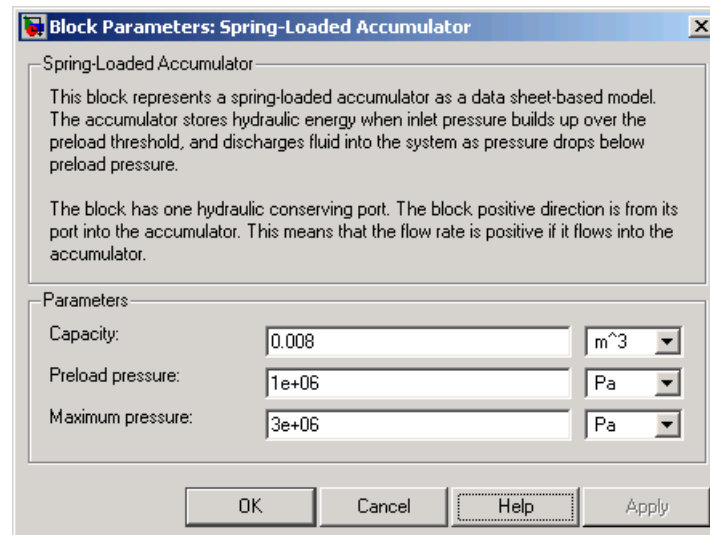
Spring-Loaded Accumulator

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 m³.

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.

Spring-Loaded Accumulator

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

Translational Damper

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

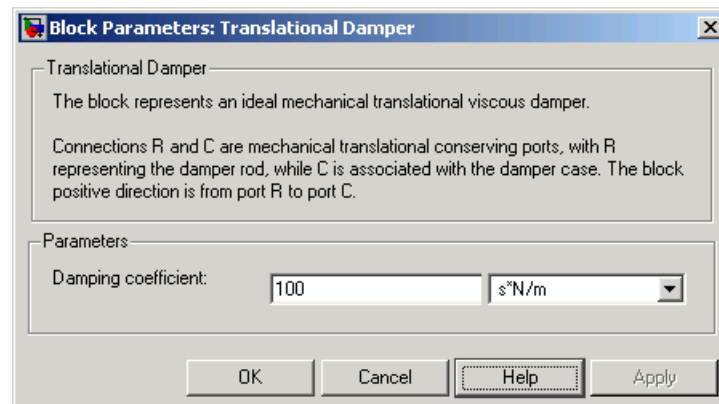
D Damping (viscous friction) coefficient

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



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

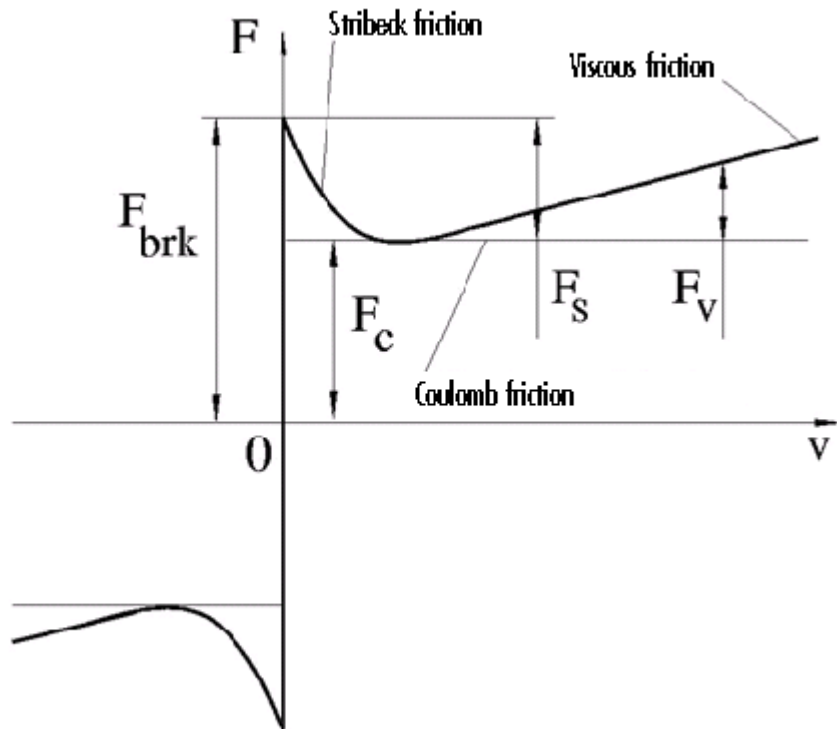
Translational Friction

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.

□ $F = -c$ □



The Stribeck friction is the negatively-sloped characteristics taking place at low velocities (see [1]). The Coulomb friction results in a

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:

$$F = (F_C + (F_{brk} - F_C) \cdot \exp(-c_v |v|)) \text{sign}(v) + fv$$

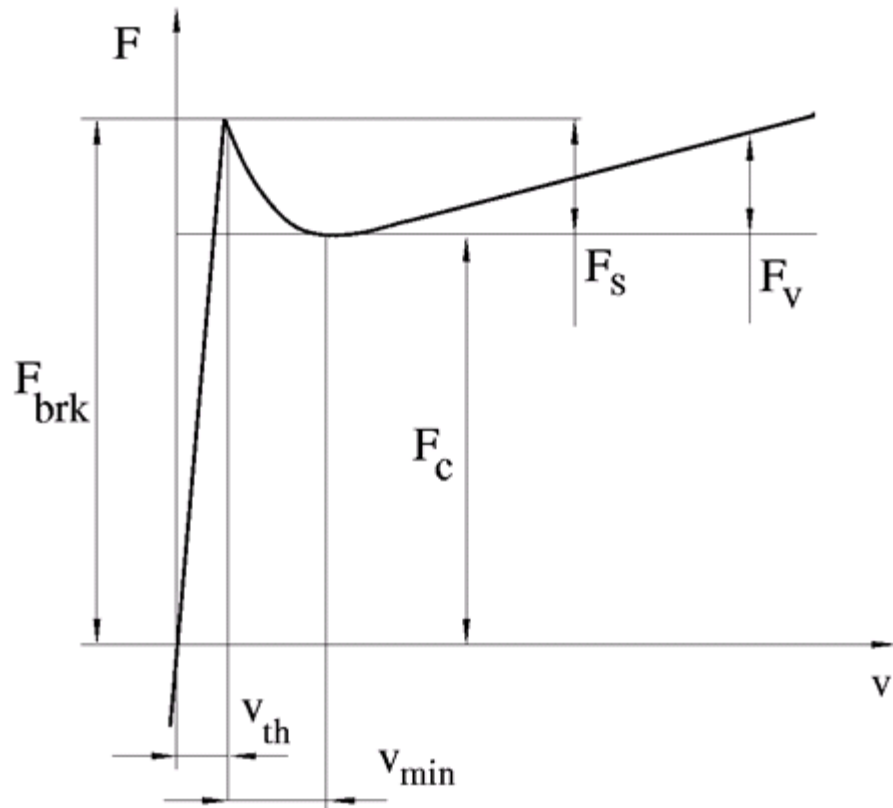
$$v = v_R - v_C$$

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 can not 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.

Translational Friction



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

motion when an acting force drops below breakaway friction level. The bodies will creep relative to each other at very small velocity proportional to acting force.

As a result of introducing the velocity threshold, the block equations are slightly modified:

- If $|v| \geq v_{th}$,

$$F = (F_C + (F_{brk} - F_C) \cdot \exp(-c_v |v|)) \text{sign}(v) + fv$$

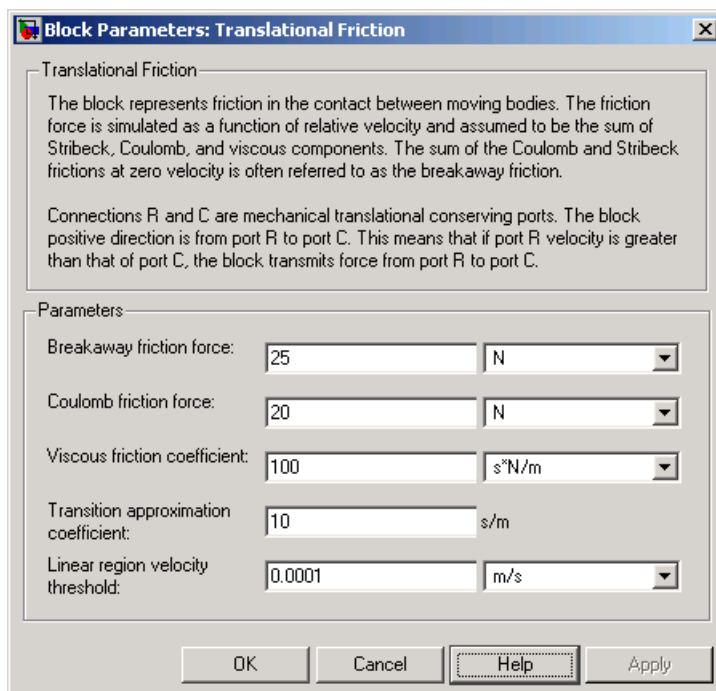
- If $|v| < v_{th}$,

$$F = v \frac{(fv_{th} + (F_C + (F_{brk} - F_C) \cdot \exp(-c_v v_{th})))}{v_{th}}$$

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.

Translational Friction

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).

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. It is recommended to 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

Translational Hard Stop

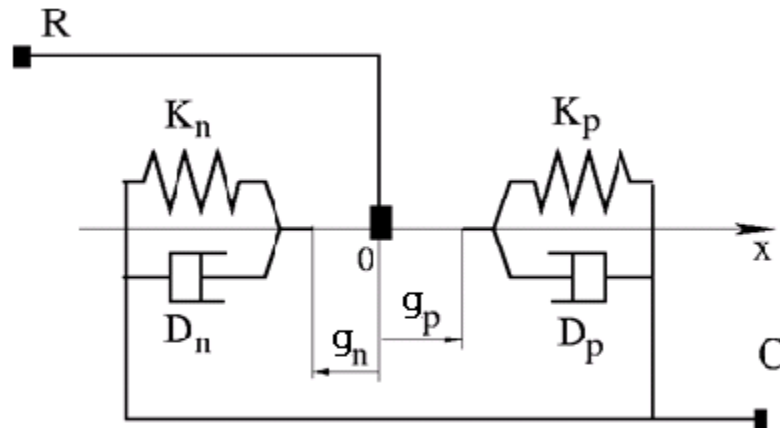
Purpose Simulate double-sided translational hard stop

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 non-elastic 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:

$$F = \begin{cases} K_p \cdot \delta + D_p (v_R - v_C) & \text{for } \delta \geq g_p \\ 0 & \text{for } g_n < \delta < g_p \\ K_n \cdot \delta + D_n (v_R - v_C) & \text{for } \delta \leq g_n \end{cases}$$

$$\delta = x_R - x_C$$

$$v_R = \frac{dx_R}{dt}$$

$$v_C = \frac{dx_C}{dt}$$

where

F	Interaction force between the slider and the case
δ	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

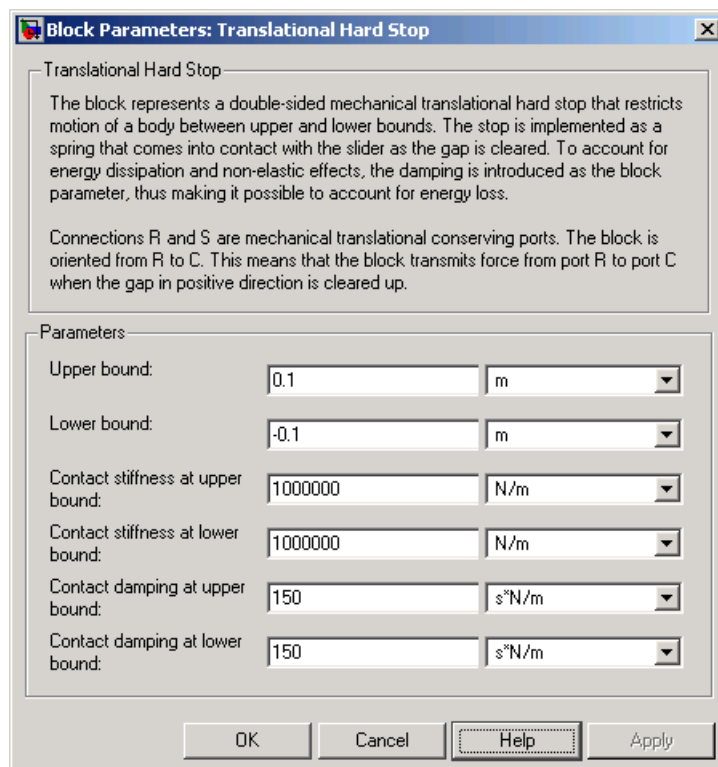
The equations are derived with respect to the local coordinate system whose axis is directed from port R to port C. The terms “positive” and “negative” in the variable descriptions refer to this coordinate system, and the gap in negative direction must be specified with negative value.

Translational Hard Stop

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



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 0.005 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 10^6 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 10^6 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, it is recommended that you assign a nonzero value to this parameter. The default value is 150 N*s/m.

Translational Hard Stop

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, it is recommended 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	Gage 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.

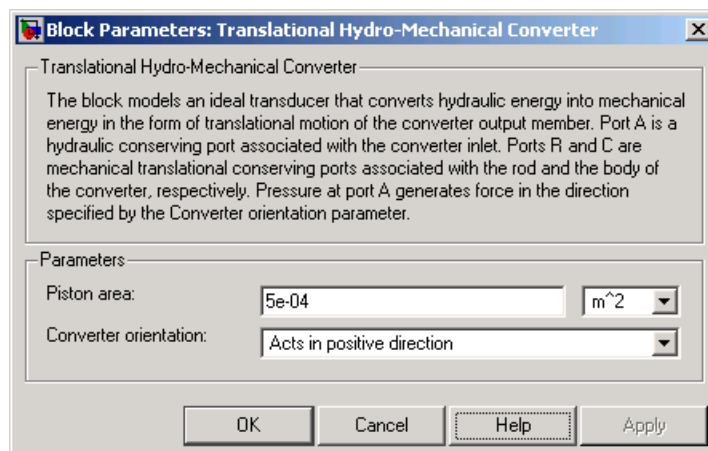
Translational Hydro-Mechanical Converter

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

Translational Spring

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)

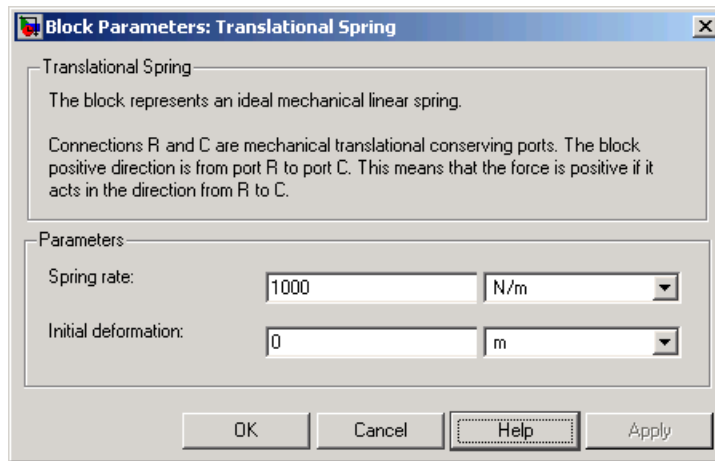
x_R, x_C Absolute displacements of terminals R and C, respectively

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.

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 ($x_{init} < 0$). 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

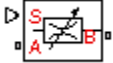
Translational Spring

Translational Hard Stop

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

Variable Orifice

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

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

$$h = x_0 + x \cdot \sigma r$$

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

$$p = p_A - p_B$$

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

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

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

where

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

A_{max}	Orifice maximum area
h_{max}	Control member maximum displacement
x_0	Initial opening
x	Control member displacement from initial position
h	Orifice opening
or	Orifice orientation indicator. The variable assumes +1 value if the control member displacement in the globally assigned positive direction opens the orifice, and -1 if positive motion decreases the opening.
ρ	Fluid density
D_H	Instantaneous orifice hydraulic diameter
ν	Fluid kinematic viscosity
A_{leak}	Closed orifice leakage area

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

Variable Orifice

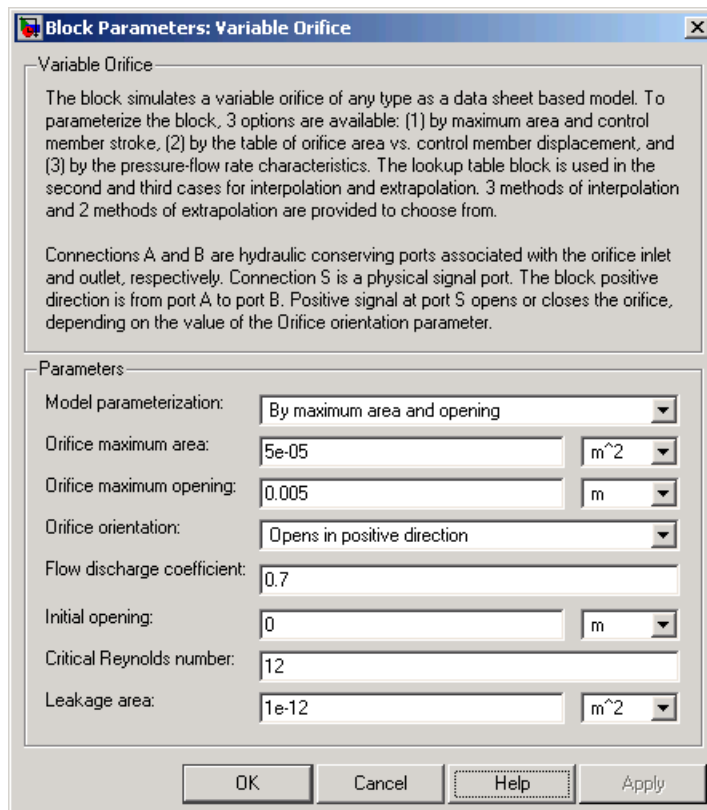
port S opens or closes the orifice depending on the value of the orifice orientation indicator.

Basic Assumptions and Limitations

The model is based on the following assumptions:

- Fluid inertia is not taken into account.
- For orifices specified by the passage area (the first two parameterization options), the transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.
- For orifices specified by pressure-flow characteristics (the third parameterization option), the model does not explicitly account for the flow regime or leakage flow rate, because the tabulated data is assumed to account for these characteristics.

Dialog Box and Parameters



Block Parameters: Variable Orifice

Variable Orifice

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

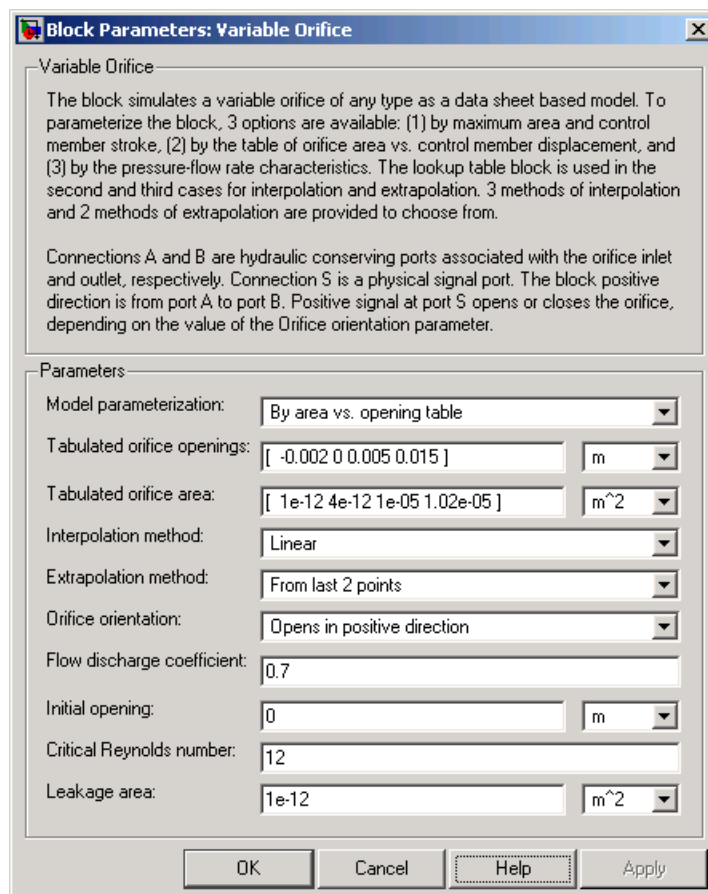
Connections A and B are hydraulic conserving ports associated with the orifice inlet and outlet, respectively. Connection S is a physical signal port. The block positive direction is from port A to port B. Positive signal at port S opens or closes the orifice, depending on the value of the Orifice orientation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

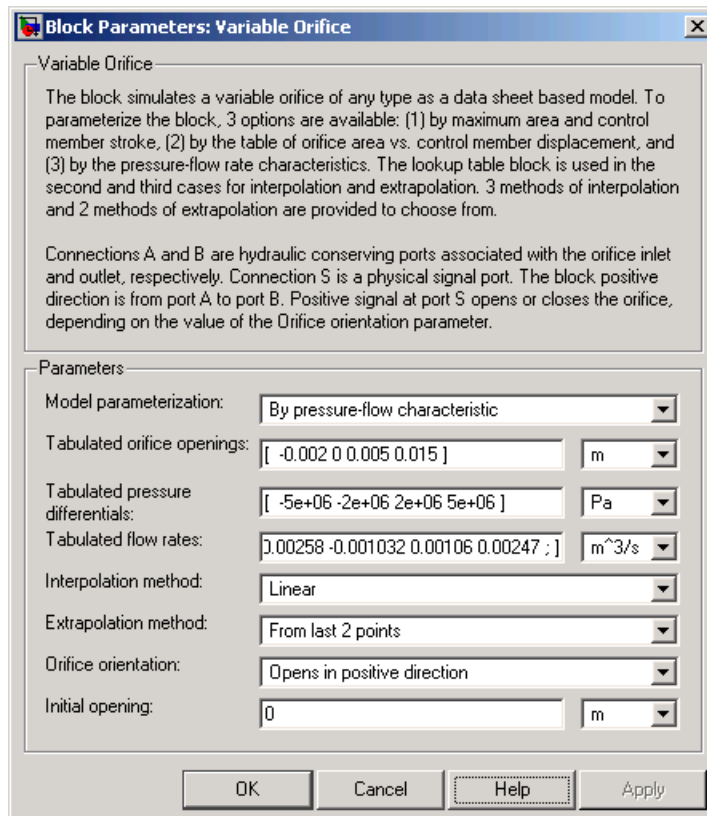
Parameters

Model parameterization:	By maximum area and opening	
Orifice maximum area:	5e-05	m ²
Orifice maximum opening:	0.005	m
Orifice orientation:	Opens in positive direction	
Flow discharge coefficient:	0.7	
Initial opening:	0	m
Critical Reynolds number:	12	
Leakage area:	1e-12	m ²

OK Cancel Help Apply

Variable Orifice





Model parameterization

Select one of the following methods for specifying the orifice:

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

Variable Orifice

- **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 3 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.

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 3 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.

Variable Orifice

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

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

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

Extrapolation method

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

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

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

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 when multi-orifice valve is built 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

Variable Orifice

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.

See Also

Orifice with Variable Area Round Holes

Orifice with Variable Area Slot

PS Lookup Table (1D)

PS Lookup Table (2D)

Variable Volume Chamber

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, non-dissolved 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:

$$q = \frac{V_0 + A \cdot x \text{ or } dp}{E} \cdot \frac{dp}{dt}$$
$$E = E_l \frac{1 + \alpha \left(\frac{p_a}{p_a + p} \right)^{1/n}}{1 + \alpha \frac{p_a^{1/n}}{n \cdot (p_a + p)^{\frac{n+1}{n}}} E_l}$$

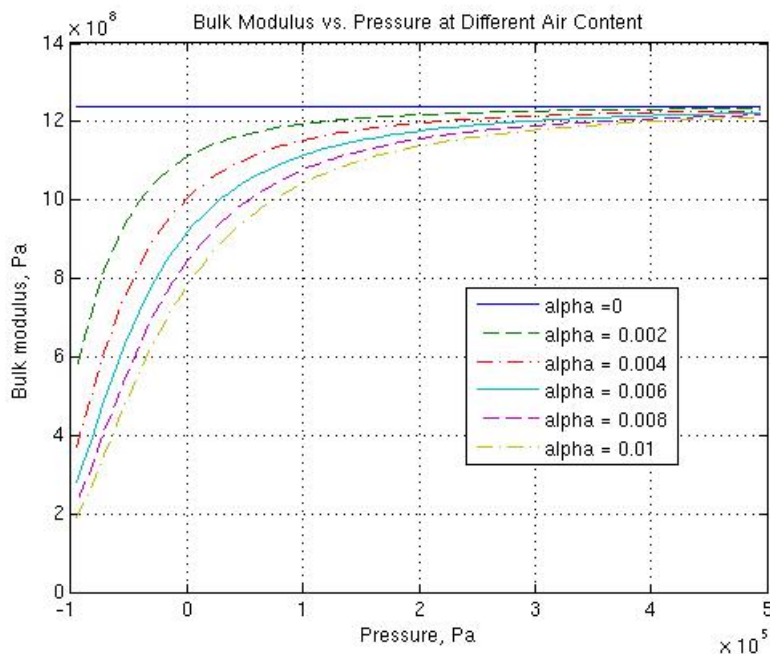
where

- 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

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	Gage pressure of fluid in the chamber
p_α	Atmospheric pressure
α	Relative gas content at atmospheric pressure, $\alpha = V_g/V_L$
V_g	Gas volume at atmospheric pressure
V_L	Volume of liquid
n	Gas specific heat ratio

The main objective of representing fluid as a mixture of liquid and gas is to introduce an approximate model of cavitation, which takes place in a chamber if pressure drops below fluid vapor saturation level. As it is seen in the graph below, the bulk modulus of a mixture decreases at $p \rightarrow p_\alpha$, thus considerably slowing down further pressure change. At high pressure, $p \gg p_\alpha$, a small amount of non-dissolved 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 can not 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.

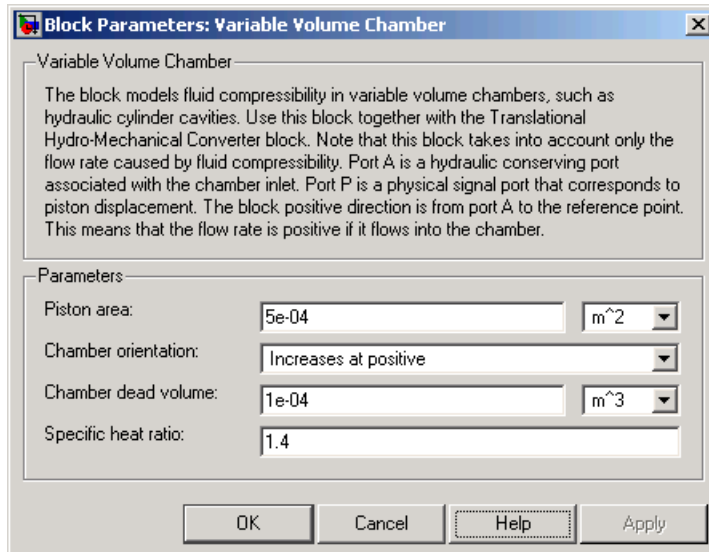
The block positive direction is from port A to the reference point. This means that the flow rate is positive if it flows into the chamber.

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.

Variable Volume Chamber

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.

Non-dissolved gas ratio

Non-dissolved 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 Pressure-Compensated Pump

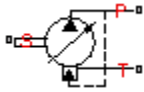
Purpose

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

Library

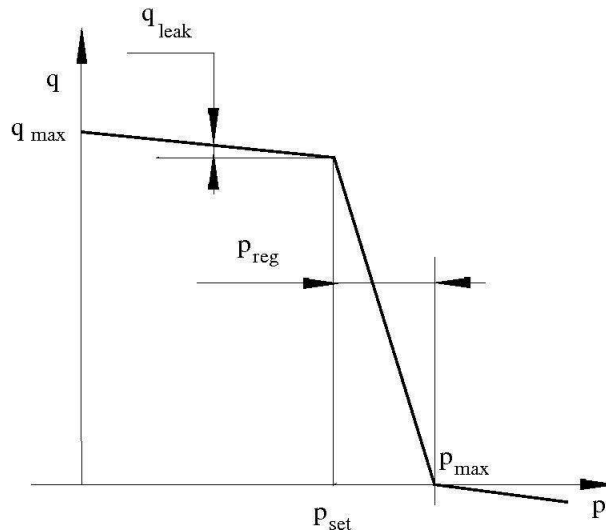
Pumps and Motors

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	Gage pressures at the block terminals
D	Pump instantaneous displacement
D_{max}	Pump maximum displacement
p_{set}	Pump setting pressure
p_{max}	Maximum pressure, at which the pump displacement is zero
T	Torque at the pump driving shaft
ω	Pump angular velocity
k_{leak}	Leakage coefficient
k_{HP}	Hagen-Poiseuille coefficient
η_V	Pump volumetric efficiency
η_{mech}	Pump mechanical efficiency
ν	Fluid kinematic viscosity
ρ	Fluid density
p_{nom}	Pump nominal pressure
ω_{nom}	Pump nominal angular velocity
ν_{nom}	Nominal fluid kinematic viscosity

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

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

where

Variable-Displacement Pressure-Compensated Pump

q_{leak} Leakage flow

d, l Geometric parameters of the leakage path

μ Fluid dynamic viscosity, $\mu = \nu \rho$

The leakage flow at $p = p_{nom}$ and $v = v_{nom}$ can be determined from the catalog data

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

which provides the formula to determine the Hagen-Poiseuille coefficient

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

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

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

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

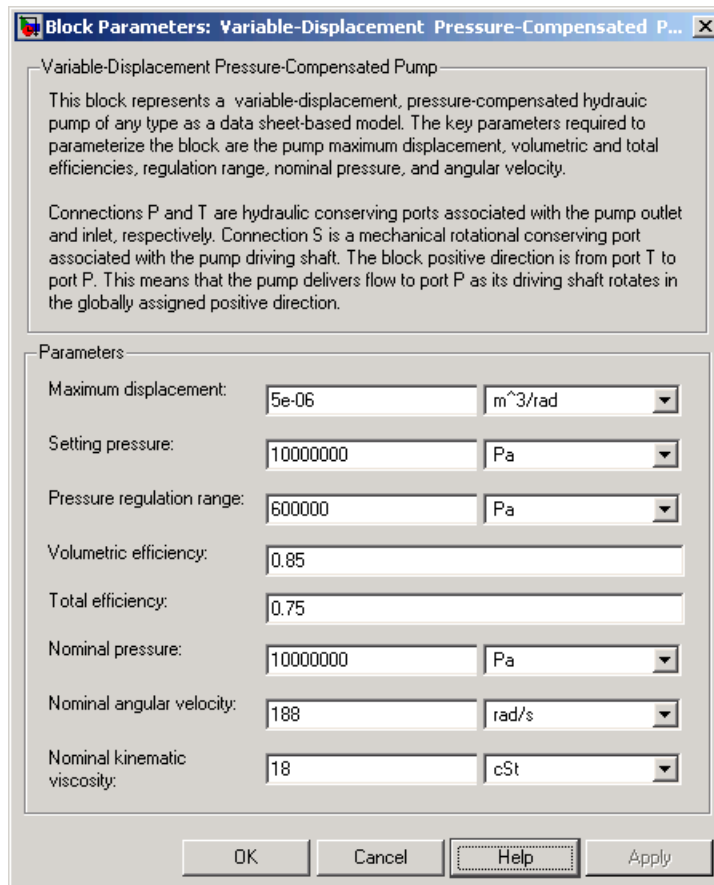
Basic Assumptions and Limitations

The model is based on the following assumptions:

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

Variable-Displacement Pressure-Compensated Pump

Dialog Box and Parameters



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

Variable-Displacement Pressure-Compensated Pump

This block represents a variable-displacement, pressure-compensated hydraulic pump of any type as a data sheet-based model. The key parameters required to parameterize the block are the pump maximum displacement, volumetric and total efficiencies, regulation range, nominal pressure, and angular velocity.

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

Parameters

Maximum displacement:	<input type="text" value="5e-06"/>	<input type="text" value="m^3/rad"/>
Setting pressure:	<input type="text" value="10000000"/>	<input type="text" value="Pa"/>
Pressure regulation range:	<input type="text" value="600000"/>	<input type="text" value="Pa"/>
Volumetric efficiency:	<input type="text" value="0.85"/>	
Total efficiency:	<input type="text" value="0.75"/>	
Nominal pressure:	<input type="text" value="10000000"/>	<input type="text" value="Pa"/>
Nominal angular velocity:	<input type="text" value="188"/>	<input type="text" value="rad/s"/>
Nominal kinematic viscosity:	<input type="text" value="18"/>	<input type="text" value="cSt"/>

OK Cancel Help Apply

Maximum displacement

Pump displacement. The default value is $5e-6$ m³/rad.

Setting pressure

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

Pressure regulation range

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

Variable-Displacement Pressure-Compensated Pump

Volumetric efficiency

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

Total efficiency

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

Nominal pressure

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

Nominal angular velocity

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

Nominal kinematic viscosity

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

Global Parameters

Fluid kinematic viscosity

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

Ports

The block has the following ports:

T

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

P

Hydraulic conserving port associated with the pump outlet.

S

Mechanical rotational conserving port associated with the pump driving shaft.

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

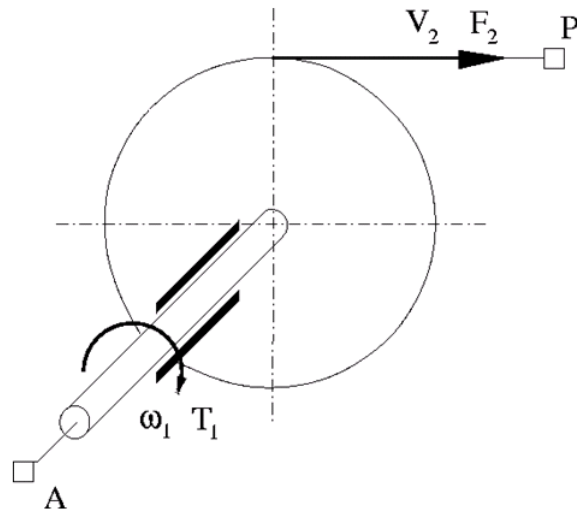
Fixed-Displacement Pump

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

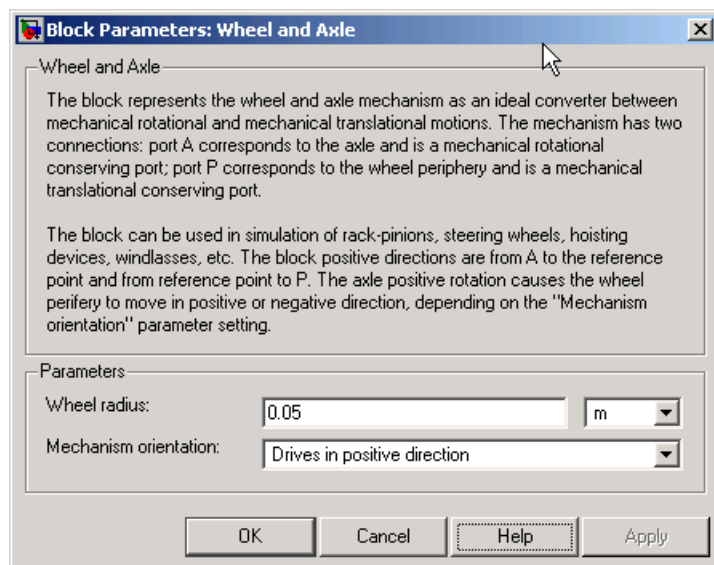
T	Torque on the axle
F	Force on the wheel periphery
ω	Angular velocity
v	Linear velocity on the wheel periphery
r	Wheel radius
or	Mechanism orientation indicator. The variable assumes +1 value if axle rotation in the globally assigned positive direction is converted into translational motion in positive 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

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 2-90
 spring-loaded 2-271

B

Ball Valve block 2-47

C

Check Valve block 2-53
Connection Port block 2-59
Constant Volume Chamber block 2-61
Custom Hydraulic Fluid block 2-70

D

Double-Acting Hydraulic Cylinder block 2-72

F

Fixed Orifice block 2-78
Fixed-Displacement Pump block 2-82
Fluid Inertia block 2-88

G

Gas-Charged Accumulator block 2-90
Gear Box block 2-94

H

Hydraulic Fluid block 2-96
Hydraulic Motor block 2-99

Hydraulic Pipeline block 2-104
Hydraulic Reference block 2-111

I

Ideal Angular Velocity Source block 2-112
Ideal Force Sensor block 2-114
Ideal Force Source block 2-116
Ideal Hydraulic Flow Rate Sensor block 2-118
Ideal Hydraulic Flow Rate Source block 2-120
Ideal Hydraulic Pressure Sensor block 2-122
Ideal Hydraulic Pressure Source block 2-124
Ideal Rotational Motion Sensor block 2-126
Ideal Torque Sensor block 2-128
Ideal Torque Source block 2-130
Ideal Translational Motion Sensor block 2-132
Ideal Translational Velocity Source block 2-134
Inertia block 2-136

L

Lever block 2-138
Linear Hydraulic Resistance block 2-143
Local Resistance block 2-145

M

Mass block 2-149
Mechanical Rotational Reference block 2-151
Mechanical Translational Reference
 block 2-152

N

Needle Valve block 2-153

O

Orifice with Variable Area Round Holes
 block 2-159
Orifice with Variable Area Slot block 2-166

P

Pilot-Operated Check Valve block 2-171
Poppet Valve block 2-178
Pressure Compensator block 2-192
Pressure Reducing Valve block 2-199
Pressure Relief Valve block 2-204
Pressure-Compensated Flow Control Valve
block 2-184
Proportional and Servo-Valve Actuator
block 2-210
PS Add block 2-214
PS Dead Zone block 2-215
PS Gain block 2-217
PS Integrator block 2-218
PS Lookup Table (1D) block 2-220
PS Lookup Table (2D) block 2-223
PS Saturation block 2-227
PS Subtract block 2-229
PS-Simulink Converter block 2-230

R

Resistive Tube block 2-232
Rotational Damper block 2-239
Rotational Friction block 2-241
Rotational Hard Stop block 2-248
Rotational Spring block 2-253

S

Segmented Pipeline block 2-256
Simulink-PS Converter block 2-262
Single-Acting Hydraulic Cylinder block 2-264
Solver Configuration block 2-269
Spring-Loaded Accumulator block 2-271

T

Translational Damper block 2-274
Translational Friction block 2-276
Translational Hard Stop block 2-282
Translational Hydro-Mechanical Converter
block 2-287
Translational Spring block 2-290

V

Variable Orifice block 2-293
Variable Volume Chamber block 2-306
Variable-Displacement Pressure-Compensated
Pump block 2-311

W

Wheel and Axle block 2-318