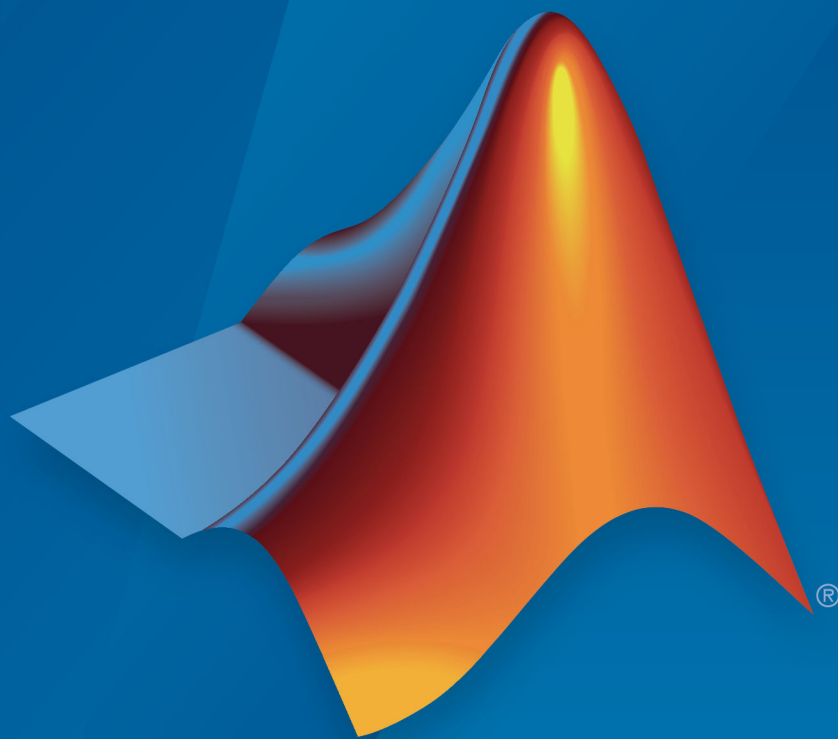


Simscape™ Fluids™

User's Guide



MATLAB® & SIMULINK®

R2018a



How to Contact MathWorks



Latest news: www.mathworks.com
Sales and services: www.mathworks.com/sales_and_services
User community: www.mathworks.com/matlabcentral
Technical support: www.mathworks.com/support/contact_us



Phone: 508-647-7000



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

Simscape™ Fluids™ User's Guide

© COPYRIGHT 2006–2018 by The MathWorks, Inc.

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

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

Trademarks

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

Patents

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

Revision History

March 2006	Online only	New for Version 1.0 (Release 2006a+)
September 2006	Online only	Revised for Version 1.1 (Release 2006b)
March 2007	Online only	Revised for Version 1.2 (Release 2007a)
September 2007	Online only	Revised for Version 1.2.1 (Release 2007b)
March 2008	Online only	Revised for Version 1.3 (Release 2008a)
October 2008	Online only	Revised for Version 1.4 (Release 2008b)
March 2009	Online only	Revised for Version 1.5 (Release 2009a)
September 2009	Online only	Revised for Version 1.6 (Release 2009b)
March 2010	Online only	Revised for Version 1.7 (Release 2010a)
September 2010	Online only	Revised for Version 1.8 (Release 2010b)
April 2011	Online only	Revised for Version 1.9 (Release 2011a)
September 2011	Online only	Revised for Version 1.10 (Release 2011b)
March 2012	Online only	Revised for Version 1.10.1 (Release 2012a)
September 2012	Online only	Revised for Version 1.11 (Release 2012b)
March 2013	Online only	Revised for Version 1.12 (Release 2013a)
September 2013	Online only	Revised for Version 1.13 (Release 2013b)
March 2014	Online only	Revised for Version 1.14 (Release 2014a)
October 2014	Online only	Revised for Version 1.15 (Release 2014b)
March 2015	Online only	Revised for Version 1.16 (Release 2015a)
September 2015	Online only	Revised for Version 1.17 (Release 2015b)
March 2016	Online only	Revised for Version 2.0 (Release 2016a) (Renamed from <i>SimHydraulics@User's Guide</i>)
September 2016	Online only	Revised for Version 2.1 (Release R2016b)
March 2017	Online only	Revised for Version 2.2 (Release R2017a)
September 2017	Online only	Revised for Version 2.3 (Release R2017b)
March 2018	Online only	Revised for Version 2.4 (Release R2018a)

1 Getting Started

Simscape Fluids Product Description	1-2
Key Features	1-2
Simscape Fluids Applications	1-3
Start a New Simscape Fluids Model	1-4
Simulating Hydraulic Models	1-5
Simulation Basics	1-5
Selecting a Solver	1-5
Specifying Simulation Accuracy/Speed Tradeoff	1-6
Nonphysical Values During Simulation	1-6
Troubleshooting Hydraulic Models	1-7
Sharing Models with Simscape Users	1-9

2 Modeling Hydraulic Systems

Simscape Fluids Modeling Principles	2-2
Block Diagrams	2-2
Connection and Signal Lines	2-3
Conserving and Signal Ports	2-4
Signal Conversions	2-6
Model Equations	2-7
Example Model	2-8
Specifying the Working Fluid	2-11
Specifying Fluid Properties	2-11

Selecting a Fluid Settings Block	2-11
Working with Multiple Fluid Networks	2-12
Omitting Fluid Properties in a Network	2-12
Fluid Volumes in Thermal Liquid Networks	2-13
Fluid Volumes	2-13
Component Dynamics	2-13
Grounding Rules	2-14
Initial Conditions	2-15
Dynamic Components	2-15
Energy Flows in Thermal Liquid Networks	2-16
Upwind Energy Scheme	2-16
Numerical Smoothing	2-18
Total Energy Flow Rate	2-19
Model a Thermal Liquid Pipeline	2-21
Model Overview	2-21
Start a New Model	2-22
Model the Pipeline	2-23
Model the Thermal Conduction	2-24
Model the Pump Torque	2-25
Add Thermal Liquid Sensors	2-28
Simulate the Pipeline Model	2-31
Model an Isothermal Hydraulic Actuator	2-34
Model Overview	2-34
Start a New Model	2-36
Model the Actuator	2-37
Model the Actuator Load	2-39
Add Sources and Scopes	2-40
Simulate the Actuator Model	2-43
Modeling Power Units	2-46
Modeling Directional Valves	2-50
Types of Directional Valves	2-50
4-Way Directional Valve Configurations	2-51
Building Blocks and How to Use Them	2-56
Build a Custom Directional Valve	2-60

Modeling Low-Pressure Fluid Transportation Systems	2-64
How Fluid Transportation Systems Differ from Power and Control Systems	2-64
Available Blocks and How to Use Them	2-66
Low-Pressure Fluid Transportation System	2-68
Pump and Motor Characteristic Curves	2-70
What Are Characteristic Curves?	2-70
Supported Blocks and Block Variants	2-70
Performance, Efficiency, and Loss Curves	2-71
How Characteristic Curves Are Generated	2-74
Example 1: Plot Performance Curves	2-75
Example 2: Plot Efficiency Curves	2-77

Getting Started

- “Simscape Fluids Product Description” on page 1-2
- “Simscape Fluids Applications” on page 1-3
- “Start a New Simscape Fluids Model” on page 1-4
- “Simulating Hydraulic Models” on page 1-5
- “Sharing Models with Simscape Users” on page 1-9

Simscape Fluids Product Description

Model and simulate fluid systems

Simscape Fluids provides component libraries for modeling and simulating fluid systems. It includes models of hydraulic pumps, valves, actuators, pipelines, and heat exchangers. You can use these components to develop fluid power systems such as front-loader, power steering, and landing gear actuation systems. Simscape Fluids also enables you to develop engine cooling, gearbox lubrication, and fuel supply systems. You can integrate mechanical, electrical, thermal, and other physical systems into your model using components from the Simscape family of products.

Simscape Fluids helps you develop control systems and test system-level performance. You can create custom component models with the MATLAB® based Simscape language, which enables text-based authoring of physical modeling components, domains, and libraries. You can parameterize your models using MATLAB variables and expressions, and design control systems for your hydraulic system in Simulink®. To deploy models to other simulation environments, including hardware-in-the-loop (HIL) systems, Simscape Fluids supports C-code generation.

Key Features

- Pump models, including centrifugal, jet, and axial-piston pumps
- Valve models, including check valves, directional valves, and pressure relief valves
- Rotational and translational actuator models, including optional friction and centrifugal forces
- Heat exchanger models for modeling heating and cooling systems
- Tank and pipe models with elevation effects for modeling fluid transportation systems
- Customizable library of common fluids, with ability to import properties from common databases
- MATLAB based Simscape language for creating custom component models
- Support for C-code generation (with Simulink Coder™)

Simscape Fluids Applications

Simscape Fluids software enables you to model a variety of thermal and isothermal liquid systems. Key application areas include fluid transport, power generation, as well as heating and cooling. Simscape Fluids blocks are organized into three libraries spanning two Simscape liquid domains, Hydraulics and Thermal Liquid. The libraries include:

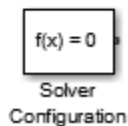
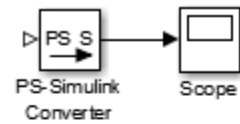
- Hydraulics — Model liquid systems at constant temperature, without regard for heat transfer. The fluid properties are constant functions of temperature. This library is suitable for systems where temperature fluctuations are small, as in hydraulic actuators operating at nearly constant temperature.
- Thermal Liquid — Model liquid systems at variable temperature, with thermal effects due to convective and conductive heat transfer. The fluid properties are functions of pressure and temperature. This library is suitable for systems where temperature fluctuations can be large, as in engine oil coolers.
- Fluid Network Interfaces — Model heat exchange and other physical interactions between otherwise distinct liquid networks. The library includes convenience blocks such as Interface (TL-IL) that you can use to expand any existing Hydraulics models with Thermal Liquid components.

Start a New Simscape Fluids Model

You can use the Simscape model template as a starting point for your Simscape Fluids models. The template includes the required Solver Configuration block and the commonly used Simulink-PS Converter and PS-Simulink Converter blocks.

To open the Simscape model template, at the MATLAB command prompt, enter `ssc_new`. Simscape software opens the model template along with the Foundation library. Enter `SimscapeFluids_lib` to open the Simscape Fluids block library.

The figure shows the Simscape model template. Drag blocks from the Simscape and Simscape Fluids libraries to represent the various system components. Use specialized blocks from physical modeling products such as Simscape Multibody™ and Simscape Driveline™ to model multiphysics systems.



Simulating Hydraulic Models

In this section...

“Simulation Basics” on page 1-5

“Selecting a Solver” on page 1-5

“Specifying Simulation Accuracy/Speed Tradeoff” on page 1-6

“Nonphysical Values During Simulation” on page 1-6

“Troubleshooting Hydraulic Models” on page 1-7

Simulation Basics

Simscape Fluids software gives you multiple ways to simulate and analyze hydraulic power and control systems in the Simulink environment. Running a hydraulic simulation is similar to running a simulation of any other Simscape model. See “Desktop Simulation” (Simscape) for a discussion of the following topics:

- Explanation of how Simscape Fluids software validates and simulates a model
- Specifics of using Simulink linearization commands in Simscape Fluids models
- Generating code for Simscape Fluids models
- Restrictions and limitations on using Simulink tools in Simscape Fluids models

All these aspects of simulating Simscape Fluids models are exactly the same as for Simscape models.

Selecting a Solver

Simscape Fluids software supports all of the continuous-time solvers that Simscape supports. For more information, see “Setting Up Solvers for Physical Models” (Simscape).

You can select any of the supported solvers for running a Simscape Fluids simulation. The variable-step solvers, `ode23t` and `ode15s`, are recommended for most applications because they run faster and work better for systems with a range of both fast and slow dynamics.

To use Simulink Coder software to generate standalone C or C++ code from your model, you must use the `ode14x` solver. For more information about code generation, see “Code Generation” (Simscape).

Specifying Simulation Accuracy/Speed Tradeoff

To trade off accuracy and simulation time, adjust one or more of the following parameters:

- **Relative tolerance** (in the Configuration Parameters dialog box)
- **Absolute tolerance** (in the Configuration Parameters dialog box)
- **Max step size** (in the Configuration Parameters dialog box)
- **Constraint Residual Tolerance** (in the Solver Configuration block dialog box)

In most cases, the default tolerance values produce accurate results without sacrificing unnecessary simulation time. The parameter value that is most likely to be inappropriate for your simulation is **Max step size**, because the default value, `auto`, depends on the simulation start and stop times rather than on the amount by which the signals are changing during the simulation. If you are concerned about the solver missing significant behavior, change the parameter to prevent the solver from taking too large a step.

The Simulink documentation describes the following parameters in more detail and provides tips on how to adjust them:

- “Relative tolerance” (Simulink)
- “Absolute tolerance” (Simulink)
- “Max step size” (Simulink)

The Solver Configuration block reference page explains when to adjust the **Constraint Residual Tolerance** parameter value.

Nonphysical Values During Simulation

Sometimes Simscape Fluids models may display physically unattainable values during simulation because of an irregularity within the model. In general, simulation does not stop if one or more variables assume nonphysical values, such as gauge pressures below -1 bar, negative value of fluid volume in the reservoir, and so on. The rationale is that, at the end of simulation, the user knows the extent of the irregularity rather than just the fact that the reservoir is short of fluid or there is a pressure drop below vaporization level.

If you see that your model displays physically unattainable values during simulation, you must analyze your model and iteratively modify your design to smooth out the irregularities.

You can set your models to either warn or stop simulating with an error when absolute pressure in a hydraulic chamber falls below absolute zero. The default behavior is to stop simulation with an error. You can change this by using the Hydraulic Fluid block or the Custom Hydraulic Fluid block connected to the loop, and have the simulation continue with a warning. See the block reference pages for details.

Troubleshooting Hydraulic Models

Simscape Fluids simulations can stop before completion with one or more error messages. For a discussion of generic error types and error-fixing strategies, see “Troubleshooting Simulation Errors” (Simscape). The following troubleshooting techniques are specific to Hydraulic models:

- Review the model configuration. If your error message contains a list of blocks, look at these blocks first. Also look for:
 - Wrong connections — Verify that the model makes sense as a hydraulic system. For example, look for accumulators connected to the pump outlet without check valves; cylinders connected against each other, so that they try to move in opposite directions; directional valves bypassed by a huge orifice, and so on.
 - Wrong use of hydraulic elements — Simscape Fluids blocks model their respective hydraulic units within certain limits. For example, an Ideal Pressure Source block can simulate a pump only when the pressure remains constant (see “Modeling Power Units” on page 2-46). Similarly, the Pressure Relief Valve block is a steady-state representation of a real valve. A block may exhibit wrong behavior if it is placed in the wrong environment. Always check the validity of the model for a particular environment and the simulation objectives.
- Avoid portions of the system getting isolated from the main system. An isolated or "hanging" part of the system could affect computational efficiency and even cause failure of computation. Use the **Leakage Area** parameter, introduced specifically for this purpose, to maintain numerical integrity of the circuit. This parameter is present in all the directional valve blocks, pressure control and flow control valve blocks, and most of the variable orifices.
- Avoid "dry" nodes in a hydraulic system. By adding a hydraulic chamber to a node, you can considerably improve the convergence and computational efficiency of a model. Adding a chamber adds a degree of freedom. By adding a chamber, you replace a complex algebraic constraint (the dry node) by a dynamic constraint. In the Mechanical domain, this is like replacing an ideal gearbox with a gearbox that includes compliance (spring and damper). The hydraulic chamber is represented by

the Constant Volume Hydraulic Chamber block in the Simscape Hydraulic Elements library.

MathWorks recommends that you build, simulate, and test your model incrementally. Start with an idealized, simplified model of your system, simulate it, verify that it works the way you expected. Then incrementally make your model more realistic, factoring in effects such as fluid compressibility, fluid inertia, and the other things that describe real-world phenomena. Simulate and test your model at every incremental step. Use subsystems to capture the model hierarchy, and simulate and test your subsystems separately before testing the whole model configuration. This approach helps you keep your models well organized and makes it easier to troubleshoot them.

Sharing Models with Simscape Users

The Simscape Editing Mode functionality lets you open, simulate, and save models that contain blocks from add-on products, including Simscape Fluids blocks, in Restricted mode, without checking out add-on product licenses, as long as the products are installed on your machine.

This functionality allows a user, model developer, to build a model that uses Simscape and Simscape Fluids blocks and share that model with other users, model users. When building the model in Full mode, the model developer must have both a Simscape license and a Simscape Fluids license. Once the model is built, model users need only to check out a Simscape license to simulate the model and fine-tune its parameters in Restricted mode. As long as no structural changes are made to the model, model users can work in Restricted mode and do not need to check out Simscape Fluids licenses.

Another workflow lets multiple users, who all have Simscape licenses, share a small number of Simscape Fluids licenses by working mostly in Restricted mode, and temporarily switching models to Full mode only when they need to perform a specific design task that requires being in Full mode.

For a complete description of this functionality, see “Add-On Product License Management” (Simscape).

Modeling Hydraulic Systems

- “Simscape Fluids Modeling Principles” on page 2-2
- “Specifying the Working Fluid” on page 2-11
- “Fluid Volumes in Thermal Liquid Networks” on page 2-13
- “Energy Flows in Thermal Liquid Networks” on page 2-16
- “Model a Thermal Liquid Pipeline” on page 2-21
- “Model an Isothermal Hydraulic Actuator” on page 2-34
- “Modeling Power Units” on page 2-46
- “Modeling Directional Valves” on page 2-50
- “Modeling Low-Pressure Fluid Transportation Systems” on page 2-64
- “Pump and Motor Characteristic Curves” on page 2-70

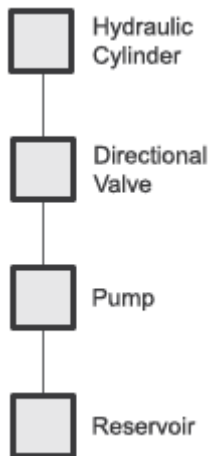
Simscape Fluids Modeling Principles

In this section...

- “Block Diagrams” on page 2-2
- “Connection and Signal Lines” on page 2-3
- “Conserving and Signal Ports” on page 2-4
- “Signal Conversions” on page 2-6
- “Model Equations” on page 2-7
- “Example Model” on page 2-8

Block Diagrams

Simscape Fluids block diagrams are like other Simscape models. They represent fluid systems by means of blocks and connection lines. Each block represents a physical component and its salient attributes. Each line represents a physical connection or interaction between components—including any exchanges in mass, energy, and momentum. Lines connect through ports representing inlets, outlets, and other component features.



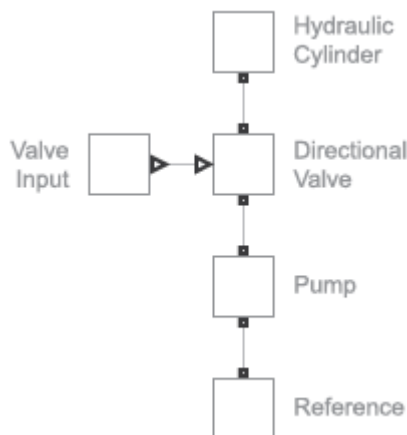
Blocks as Components and Lines as Connections

You can model fluid components using blocks from the Simscape Foundation or Simscape Fluids libraries. Simscape Foundation blocks represent idealized components such as local restrictions, mechanical converters, and constant-volume chambers. Simscape Fluids blocks represent specialized components with detailed component physics, such as directional valves, single-acting actuators, and gas-charged accumulators.

Like the systems they represent, Simscape Fluids models are often multi-domain in nature. The hydraulic actuator model on page 2-2 is one example. The model contains a hydraulic network representing the hydraulic actuation system and a Mechanical Translational network representing the actuator load. To model components from different domains, you can use blocks from the various Simscape Foundation libraries or from the several specialized physical modeling products.

Connection and Signal Lines

Lines can join blocks through physical conserving ports, \square , or physical signal ports, \triangle . A line between physical conserving ports represents a physical connection bound by physical conservation principles. A line between physical signal ports represents a component input, such as a change in valve position, or sensing output, such as a fluid pressure measurement.








Conserving and Physical Signal Ports

Physical connection lines are nondirectional. They apply constraints between the Across and Through variables of the connected conserving ports, but do not transmit numerical

values in any particular direction. Physical signal lines are directional. They pass numerical signals and associated physical units in a specific direction from one block to another, often as block inputs or outputs.

Blocks and physical connection lines are color-coded by physical domain. The various colors help to differentiate the physical domains comprising a Simscape model. Use them as visual guides when deciding if a physical connection between components is valid. All physical connection lines must join conserving ports of the same domain. The figure shows the color legend for the physical domains associated with Simscape Fluids blocks.

Color	Domain
	Hydraulic
	Thermal Liquid
	Thermal
	Mechanical Rotational
	Mechanical Translational

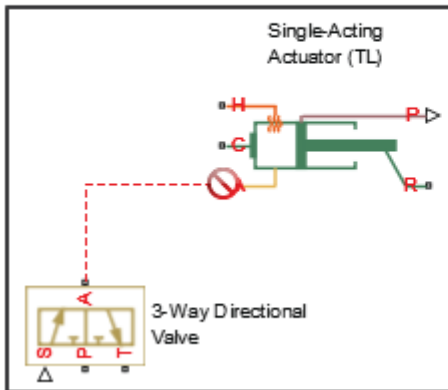
Simscape Fluids Color Coding

For more information, see “Domain-Specific Line Styles” (Simscape).

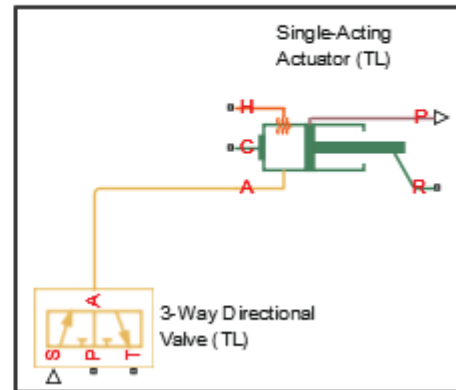
Conserving and Signal Ports

Ports always connect to like ports. A physical conserving port connects only to other physical conserving port—and only if they belong to the same physical domain. Conserving ports are associated with specific Across and Through variables and cannot connect to conserving ports associated with different variables. You can directly connect a Hydraulic conserving port to another Hydraulic conserving port but not to a Thermal Liquid conserving port.

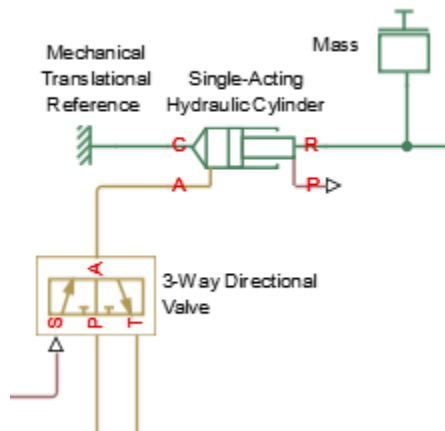
**Hydraulic-Thermal Liquid
Invalid Connection**



**Thermal Liquid-Thermal Liquid
Valid Connection**



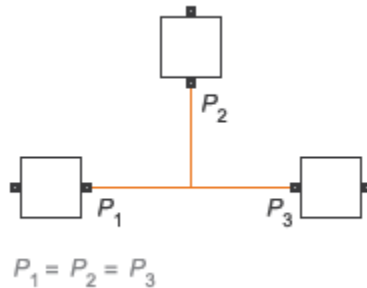
To interconnect different physical domains, you must use multi-domain blocks with ports representing different domains. The Single-Acting Hydraulic Cylinder block, shown in the figure, is one such block. A Hydraulic conserving port connects to a Hydraulic network, shown in yellow. A Mechanical Translational conserving port connects to a Mechanical Translational network, shown in green.



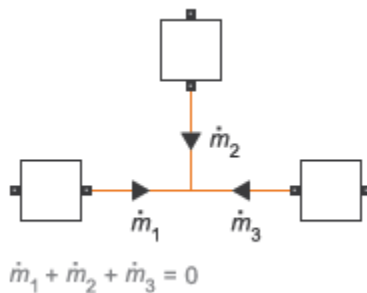
Physical signal ports similarly connect only to other physical signal ports. Physical signals can arise from blocks representing physical components, such as 2-Position Valve

general circuit analysis principles, often referred to as Kirchhoff's rules. These rules constrain the Across and Through variables so that:

- The Across variables of all directly connected conserving ports always have the same value.



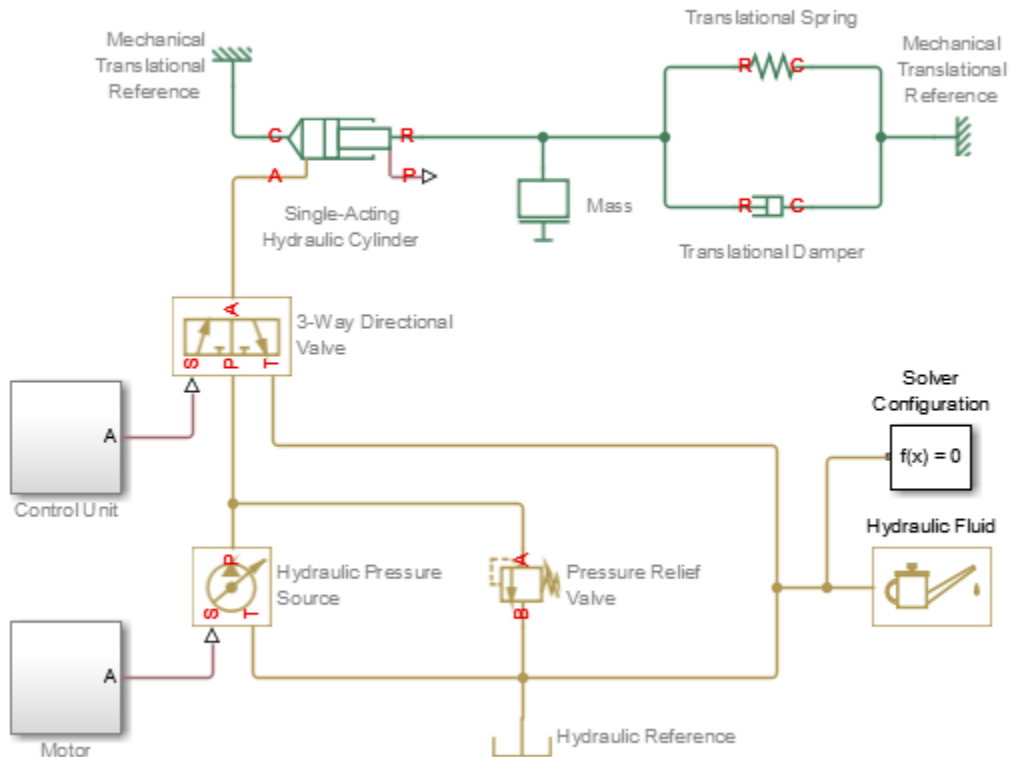
- The Through variables at a junction between two or more connection lines always sum to zero.



For more information on how Simscape software assembles and solves the governing equations of a model, see “How Simscape Simulation Works” (Simscape)

Example Model

The figure shows a Simscape Fluids model of a simple hydraulic actuator. The model has two physical networks—one hydraulic, indicated in yellow, and one Mechanical Translational, indicated in green. The Hydraulic network models the hydraulic power system that drives the actuator. The Mechanical Translational network models a damped mass-spring oscillator that the actuator pushes against.



Hydraulic Actuator Simscape Fluids Model

The Hydraulic blocks represent the components of the hydraulic power system—a hydraulic pressure source, a directional valve, a relief valve, and the hydraulic fluid. The Mechanical Translational blocks represent the components of the damped mass-spring oscillator—a mass, a translational spring, and a translational damper.

The connection lines represent the physical connections between the components. For example, the connection line between the Hydraulic Pressure Source and 3-Way Directional Valve blocks indicates a direct connection between the actual components. This connection enables the flow of mass, energy, and momentum between the source and valve during simulation.

The physical connection lines join conserving ports of the same type. Hydraulic conserving ports connect to other Hydraulic conserving ports. Mechanical Translational conserving ports connect to other Mechanical Translational conserving ports. Hydraulic and Mechanical Translational components connect through a multi-domain component represented by the Single-Acting Hydraulic Cylinder block.

The physical signal lines represent Simscape block inputs. The physical signal line between the Control Unit subsystem and the 3-Way Directional Valve blocks specifies the valve position as a function of time. The subsystem block generates the position signal with Simulink blocks that a Simulink-PS Converter block then converts into a physical signal.

For more information about Simscape modeling rules, see “Basic Principles of Modeling Physical Networks” (Simscape). For a tutorial showing how to model an isothermal hydraulic actuator, see “Model an Isothermal Hydraulic Actuator” on page 2-34.

Specifying the Working Fluid

In this section...

“Specifying Fluid Properties” on page 2-11

“Selecting a Fluid Settings Block” on page 2-11

“Working with Multiple Fluid Networks” on page 2-12

“Omitting Fluid Properties in a Network” on page 2-12

Specifying Fluid Properties

Fluid properties impact your model dynamics and are required parameters for simulation. To specify them, select a fluid settings block from the appropriate physical domain and connect it anywhere on the target fluid network. The fluid settings block propagates the specified fluid properties to all components in the attached fluid network.

Selecting a Fluid Settings Block

- To set the fluid properties in a Thermal Liquid network, use the Thermal Liquid Settings (TL) block from the Simscape Foundation library. The block takes as inputs the necessary fluid properties, each specified as a tabulated function of pressure and temperature.

For an alternative way of specifying the fluid properties in a Thermal Liquid network, use the Thermal Liquid Properties (TL) block from the Simscape Fluids Thermal Liquid library. The block requires as input little more than the name of the fluid, selected from a dropdown list that includes water, seawater, and various mixtures with uses in cooling and deicing.

- To set the fluid properties in a Hydraulic network, use the Custom Hydraulic Fluid block from the Simscape Foundation library. The block takes as inputs the necessary fluid properties, each specified as a constant.

For an alternative way of specifying the fluid properties in a Hydraulic network, use the Hydraulic Fluid block from the Simscape Fluids Hydraulic (Isothermal Liquid) library. The block requires as input little more than the name of the fluid, selected from a dropdown list that includes water, gasoline, jet fuel, and others with application in hydraulic systems.

- To set the fluid properties in a Gas network, use the Gas Properties (G) block from the Simscape Foundation library. The block takes as inputs the necessary fluid properties, these specific to the type of gas modeled (perfect, semi-perfect, or real) and variously specified as constants or tabulated functions (of temperature and/or pressure).

Working with Multiple Fluid Networks

Simscape Fluids models can have multiple fluid networks, each representing a single-fluid system. To avoid model compilation errors, use only one fluid settings block per network. Two fluid networks are treated as distinct if they belong to different Simscape domains or if they share no direct connection line.

Omitting Fluid Properties in a Network

If a fluid network has no fluid settings block, that network reverts to the default properties defined in the Simscape domain source files. The defaults correspond to water in the Thermal Liquid domain and to a typical hydraulic fluid in the Hydraulic domain.

- For the default thermal liquid properties, see “Thermal Liquid Domain” (Simscape).
- For the default hydraulic fluid properties, see “Hydraulic Domain” (Simscape).

Fluid Volumes in Thermal Liquid Networks

In this section...

“Fluid Volumes” on page 2-13

“Component Dynamics” on page 2-13

“Grounding Rules” on page 2-14

“Initial Conditions” on page 2-15

“Dynamic Components” on page 2-15

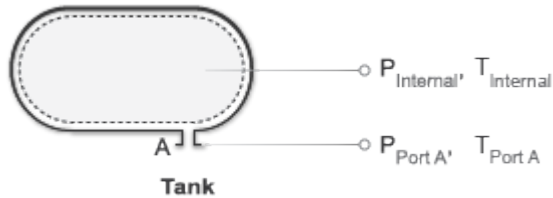
Fluid Volumes

Thermal Liquid components fall into two groups—those modeled with internal fluid volume and those not. A component has internal fluid volume if that volume is substantial and the internal fluid dynamics relevant. Components with fluid volume include pipes, tanks, and actuators. Components without fluid volume include valves, orifices, and sudden area changes. Fluid volumes enable you to:

- Model internal component dynamics.
- Satisfy the Simscape grounding rules.
- Specify initial conditions in a model.

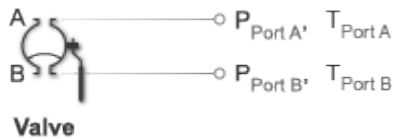
Component Dynamics

Components with fluid volume have internal states, defined as differential variables, that can evolve over time. The internal states capture the transient effects of dynamic compressibility, fluid inertia, and wall compliance, when supported and enabled. State changes occur gradually, over the course of characteristic response times determined from component geometries and fluid properties. Such components are called *dynamic*.



Dynamic Component

Components without fluid volume have no internal states. Algebraic equations constrain the states at the ports, but transient changes due to internal component dynamics are ignored. State changes occur instantaneously, without time lag. The components respond to dynamic changes just outside their ports, but so rapidly that they are, at each time step, at steady state with their neighbors. Such components are called *quasi-steady*.



Quasi-Steady Component

Grounding Rules

Dynamic components are required in Thermal Liquid models. These components have internal nodes at known pressure and temperature and therefore serve as state references in Thermal Liquid networks. You can satisfy the Simscape grounding rules by connecting at least one dynamic component to each Thermal Liquid network in your model.

Initial Conditions

To arrive at meaningful numerical solutions, the differential equations of dynamic components require initial conditions as inputs. The block dialog boxes of dynamic components provide initial condition parameters that you can set to change the initial configuration of your model. Quasi-steady components have no internal states and therefore no initial states to set.

The number of initial conditions should equal the number of degrees of freedom in your model. The number of degrees of freedom is the difference between the number of differential variables and the number of dynamic constraints. You can obtain both number using Simscape Statistics Viewer. You can view the initial conditions specified in a model using Simscape Variable Viewer.

Dynamic Components

The table lists the dynamic components that you can use, organized by library.

Library	Dynamic Components
Simscape Fluids > Thermal Liquid	Single-Acting Actuator (TL)
	Double-Acting Actuator (TL)
	Pipe (TL)
	Tank (TL)
	Gas-Charged Accumulator (TL)
Simscape Fluids > Fluid Network Interfaces	E-NTU Heat Exchanger (TL-TL)
	E-NTU Heat Exchanger (TL)
	Heat Exchanger Interface (TL)
Simscape > Foundation > Thermal Liquid	Pipe (TL)
	Constant Volume Chamber (TL)
	Controlled Reservoir (TL)
	Reservoir (TL)
	Rotational Mechanical Converter (TL)
	Translational Mechanical Converter (TL)

Energy Flows in Thermal Liquid Networks

In this section...

“Upwind Energy Scheme” on page 2-16

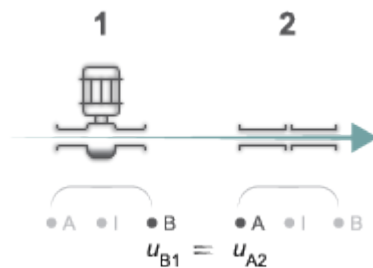
“Numerical Smoothing” on page 2-18

“Total Energy Flow Rate” on page 2-19

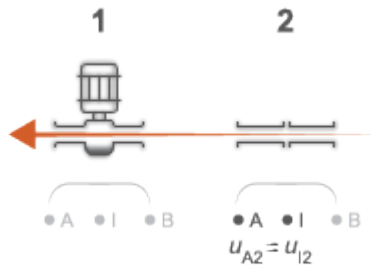
Upwind Energy Scheme

Thermal energy advection should always match the direction of flow. To satisfy this condition, the Thermal Liquid domain employs a modeling framework known as the upwind scheme. This scheme works by matching the specific internal energy at a port to that just upwind of the port:

- If the flow at a port is directed into the component, the specific internal energy at that port is equal to that carried by the incoming fluid. In the figure, the specific internal energy at port A of component **2** is equal to that at port B of component **1**.



- If the fluid flow at a port is directed out of the component, the specific internal energy at that port is equal to that in the component’s internal fluid volume. In the figure, the specific internal energy at port A of component **2** is equal to that in the internal fluid volume of the same component.



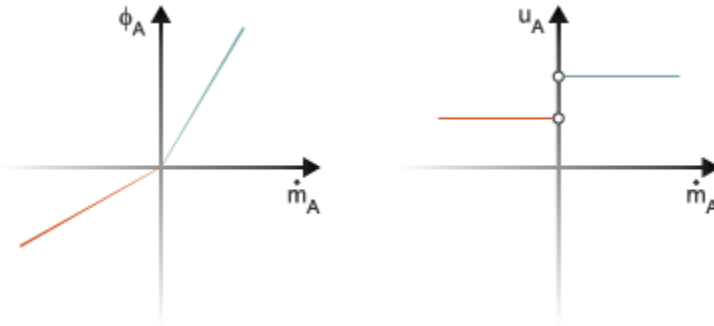
The upwind scheme describes the advection thermal energy flow rate using a conditional expression that, for port A of some component, reads:

$$\Phi_A^{Ad} = \begin{cases} \dot{m}_A u_A, & \dot{m}_A > 0 \\ \dot{m}_A u_I, & \dot{m}_A \leq 0 \end{cases}$$

where:

- Φ_A^{Ad} is the thermal energy flow rate due to advection through port A.
- \dot{m}_A is the mass flow rate through port A.
- u_A is the specific internal energy upwind of port A.
- u_I is the specific internal energy in the internal fluid volume.

Without numerical smoothing, this expression introduces a number of computational challenges. It adds a slope discontinuity to the thermal energy flux at zero mass flow rates that makes the model more prone to simulation errors. It also adds a jump discontinuity to the specific internal energy and makes its value ill-defined during flow reversals.



Upwind Energy Scheme without Smoothing

Numerical Smoothing

The Thermal Liquid domain smoothes the numerical discontinuities introduced by the upwind energy scheme by adding to the thermal energy flow rate a thermal conduction term. At port A of some component:

$$\Phi_A^{Th} = \begin{cases} \dot{m}_A u_A + G(u_A - u_I), & \dot{m}_A > 0 \\ \dot{m}_A u_I + G(u_A - u_I), & \dot{m}_A \leq 0 \end{cases}$$

where:

- Φ_A^{Th} is the thermal energy flow rate due to advection through port A and conduction between port A and the internal fluid volume.
- G is a thermal conductance coefficient computed from the fluid properties and component geometry.

Rewriting the modified expression in a more convenient form:

$$\Phi_A^{Th} = \begin{cases} (\dot{m}_A + G)u_A - Gu_I, & \dot{m}_A > 0 \\ Gu_A + (\dot{m}_A - G)u_I, & \dot{m}_A \leq 0 \end{cases}$$

Using min and max functions to collapse the conditional expression:

$$\Phi_A^{Th} = [\max(\dot{m}_A, 0) + G]u_A + [\min(\dot{m}_A, 0) - G]u_I$$

Approximating the collapsed expression as a numerically smooth expression:

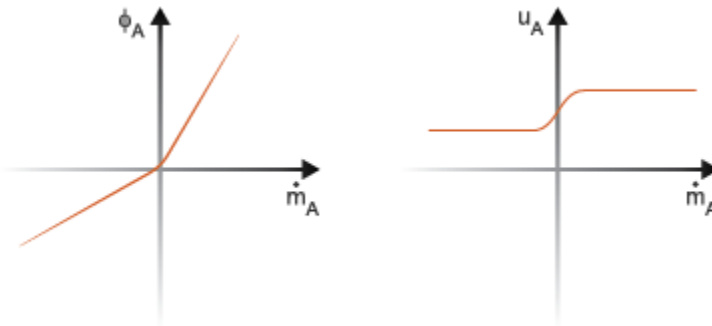
$$\Phi_A^{Th} \approx \frac{\dot{m}_A + \sqrt{\dot{m}_A^2 + 4G^2}}{2} u_A + \frac{\dot{m}_A - \sqrt{\dot{m}_A^2 + 4G^2}}{2} u_I$$

The end expression removes the slope discontinuity from the thermal energy flow rate curve and the jump discontinuity from the specific internal energy curve. The thermal conductance coefficient determines the amount of smoothing applied in both cases:

$$G = \frac{kS}{c_v L},$$

where:

- k is the thermal conductivity defined in the Thermal Liquid Settings (TL) block.
- S is the flow cross-sectional area at the port considered.
- c_v is the specific heat defined in the Thermal Liquid Settings (TL) block.
- L is a characteristic distance between the port and the component's fluid volume.



Upwind Energy Scheme with Smoothing

Total Energy Flow Rate

The Through variable of the Thermal Liquid domain is the total energy flow rate. This variable is defined in terms of the smoothed upwind thermal energy flow rate as:

$$\Phi_A = \Phi_A^{Th} + \frac{P}{\rho},$$

where:

- Φ_A is the total energy flow rate through port A.
- Φ_A^{Th} is the thermal energy flow rate through port A computed from the smoothed upwind energy scheme.
- p is the absolute pressure at port A.
- ρ is the fluid density at port A.

The kinetic energy contribution to the total energy flow rate is assumed negligible in Thermal Liquid networks and is ignored in this domain.

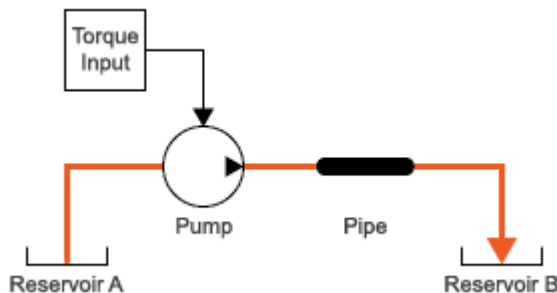
Model a Thermal Liquid Pipeline

In this section...

- “Model Overview” on page 2-21
- “Start a New Model” on page 2-22
- “Model the Pipeline” on page 2-23
- “Model the Thermal Conduction” on page 2-24
- “Model the Pump Torque” on page 2-25
- “Add Thermal Liquid Sensors” on page 2-28
- “Simulate the Pipeline Model” on page 2-31

Model Overview

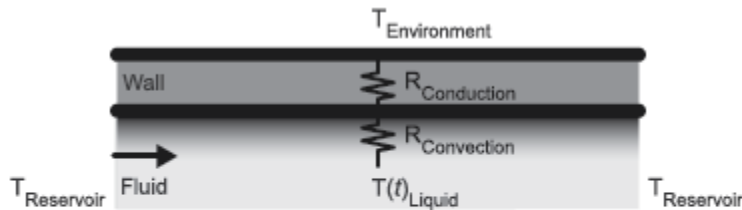
This tutorial shows how to model a pump-driven thermal liquid pipeline. The pipeline system comprises a pump, a pipe, and two thermal liquid reservoirs. The pump provides power generation and the pipe fluid transport between the reservoirs. The reservoirs provide the pressure and temperature boundary conditions for the pipeline system.



Thermal Liquid Pipeline Schematic

The reservoir conditions are fixed at room temperature and atmospheric pressure. The outer pipe surface temperature is fixed at a temperature of 275 K. The pipe fluid volume is initially at room temperature but, through the combined effects of viscous heating and conductive cooling, gradually shifts toward a new steady-state value determined by system dynamics.

The thermal resistance of the pipe is the result of convective and conductive individual resistances. A Pipe (TL) block models the convective thermal resistance at the inner pipe surface and the viscous heating in the thermal liquid. A Conductive Heat Transfer block models the conductive thermal resistance between the inner and outer surfaces.



$$T_{\text{Reservoir}} = T(0)_{\text{Liquid}} = 293.15 \text{ K}$$
$$T_{\text{Environment}} = 275 \text{ K}$$

Pipe Thermal Resistances

Start a New Model

- 1 At the MATLAB command prompt, enter `ssc_new`. MATLAB opens the Simscape model template. This template provides a starting point for your Simscape Fluids model. Save the model often as you add new blocks.
- 2 From the **Simscape Foundation** > **Thermal Liquid** > **Utilities** library, drag a Thermal Liquid Settings (TL) block to the model canvas. This block defines the physical properties of the thermal liquid, including its viscosity, thermal conductivity, and bulk modulus.
- 3 Connect the Solver Configuration and Thermal Liquid Settings (TL) blocks as shown in the figure. The Solver Configuration block provides Simscape solver settings for your model. Each topologically distinct physical network requires one such block.



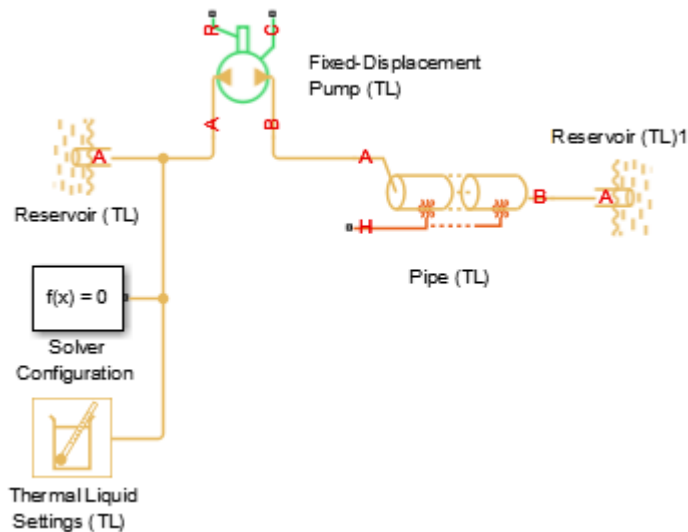
Model the Pipeline

- 1 From the **Simscape Foundation > Thermal Liquid > Elements** library, drag two Reservoir (TL) blocks. The reservoir blocks set the pressure and temperature boundary conditions for the pipeline model.
- 2 From the **Simscape Fluids > Thermal Liquid** library drag these blocks.

Block	Library
Pipe (TL)	Pipes & Fittings
Fixed-Displacement Pump (TL)	Pumps & Motors

The pipe block provides a conduit for thermal liquid transport between the reservoirs. The pump block provides a power source to generate a positive mass flow rate through the pipe.

- 3 Connect the blocks as shown in the figure. The left reservoir serves as a thermal liquid source. The right reservoir serves as a thermal liquid sink.



The component colors indicate the physical domains they represent—yellow for Thermal Liquid, orange for Thermal, and green for Mechanical Rotational. The pump and pipe blocks are multidomain blocks that you can use to interface different physical domains.

- 4 In the Pipe (TL) block, set the **Pipe length** parameter to 1000 m. This length makes more evident the thermal liquid heating due to viscous friction losses.

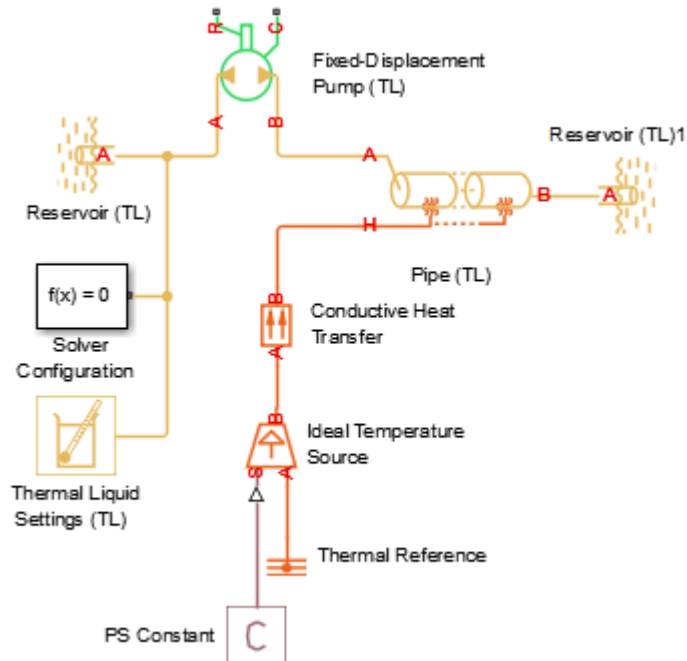
Model the Thermal Conduction

- 1 From the Simscape Foundation library, drag these blocks.

Block	Library
Conductive Heat Transfer	Thermal > Thermal Elements
Thermal Reference	Thermal > Thermal Elements
Temperature Source	Thermal > Thermal Sources

The heat transfer block models the conductive heat losses through the pipe wall to the surrounding environment. The ideal temperature source sets the environment temperature through a physical signal input. The thermal reference block provides a reference against which to specify the temperature physical signal input.

- From the **Simscape** > **Foundation** library, drag a PS Constant block. This block enables you to specify the numerical value of the environment temperature.
- Connect the blocks as shown in the figure. The new blocks representing the conductive heat losses belong to the Simscape thermal domain and connect only to thermal conserving ports.



- In the PS Constant block dialog box, set the **Constant** parameter to 275. This value is the environment temperature in the default temperature units of Kelvin.
- In the Conductive Heat Transfer block, set the **Area** parameter to $\pi \cdot 0.1128 \cdot 1000$. This expression gives the pipe surface area in terms of the pipe hydraulic diameter (0.1128 m) and length (1000 m). The default **Thermal conductivity** value is that of copper, a highly conductive metal.

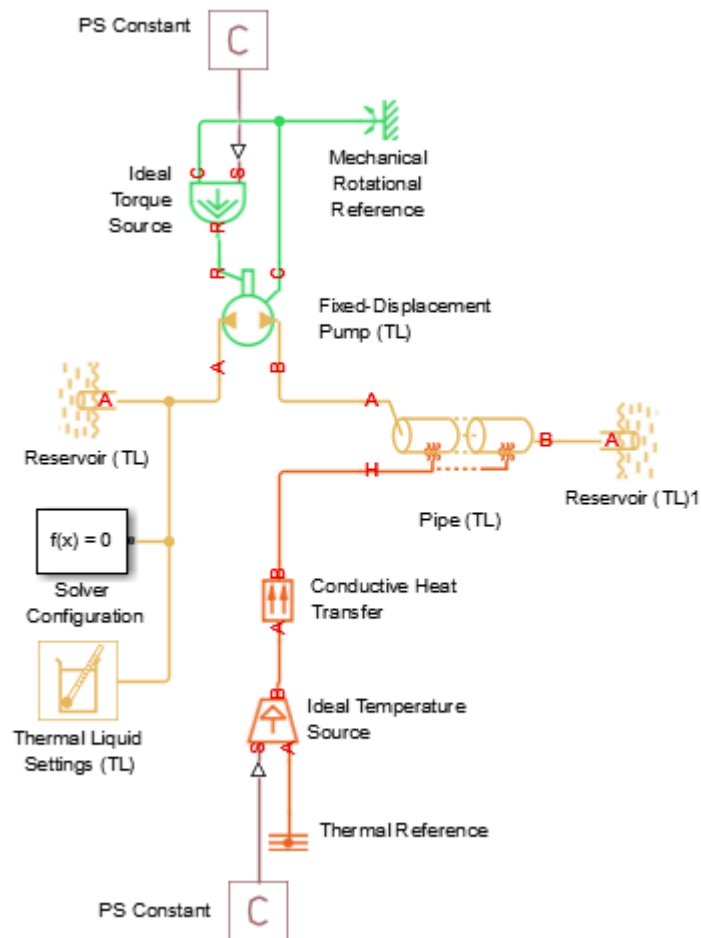
Model the Pump Torque

- From the Simscape Foundation library, drag these blocks.

Block	Library
Ideal Torque Source	Mechanical > Rotational Sources
Mechanical Rotational Reference	Mechanical > Rotational Elements

The ideal torque source block sets the pump driving torque through a physical signal input. The Mechanical Rotational reference block provides a reference against which to specify the torque physical signal input.

- 2 From the **Simscape > Utilities** library, drag a PS Constant block. This block specifies the numerical value of the pump torque.
- 3 Connect the blocks as shown in the figure. The new blocks representing the pump power source belong to the Mechanical Rotational domain and connect only to Mechanical Rotational conserving ports.



Port R of the pump block represents the rotating shaft and connects to the ideal torque source block. Port C represents the pump casing and connects to the mechanical rotational reference.

- 4 In the PS Constant block dialog box, set the **Constant** parameter to 50. This value specifies the pump torque in the default torque units of N*m.

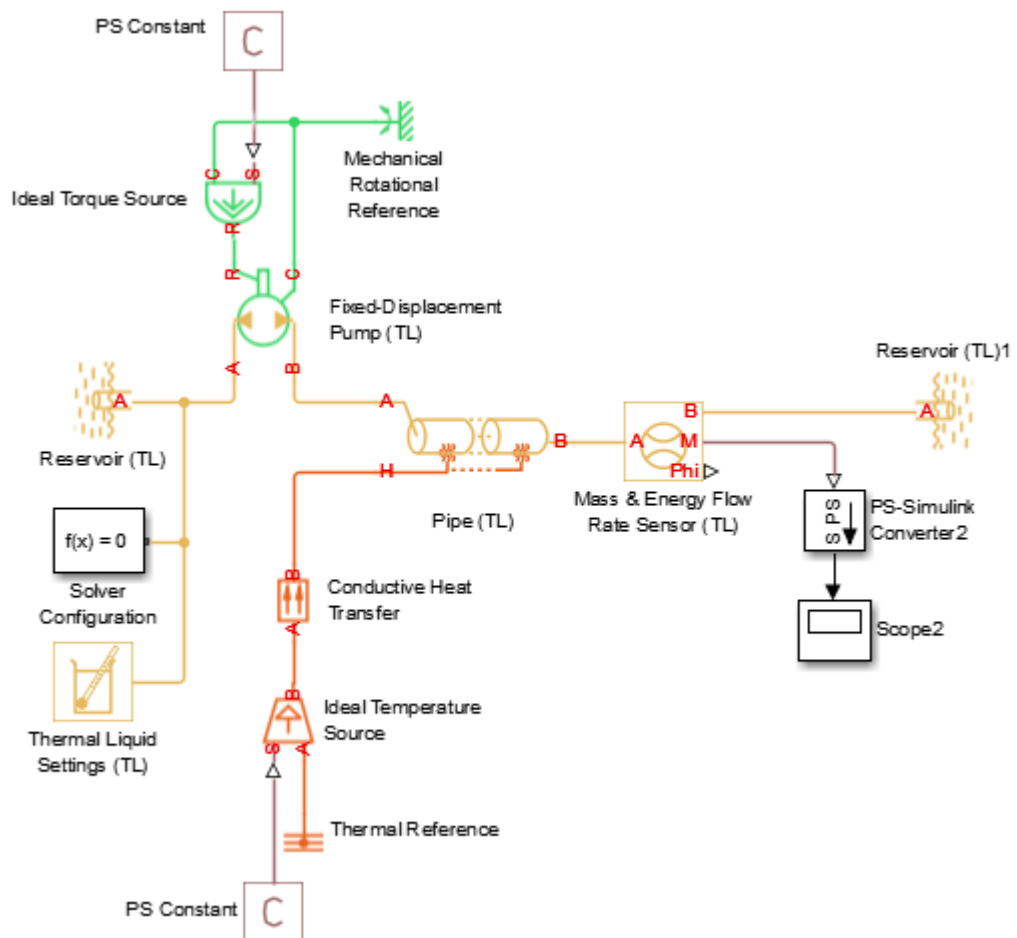
Add Thermal Liquid Sensors

- 1 Drag these blocks to the model.

Block	Library
Mass & Energy Flow Rate Sensor (TL)	Simscape Foundation > Thermal Liquid > Sensors
PS-Simulink Converter	Simscape > Utilities
Scope	Simulink > Sinks

The sensor block measures the Through variables of the Thermal Liquid domain and outputs them as physical signals. The converter block transforms the physical signal into a Simulink signal for plotting in a Scope block.

- 2 Connect the blocks as shown in the figure. Arrange the sensor block so that both of its conserving ports connect to the Thermal Liquid branch being probed. The sensor output is the measured flow rate in the direction of port A to port B.



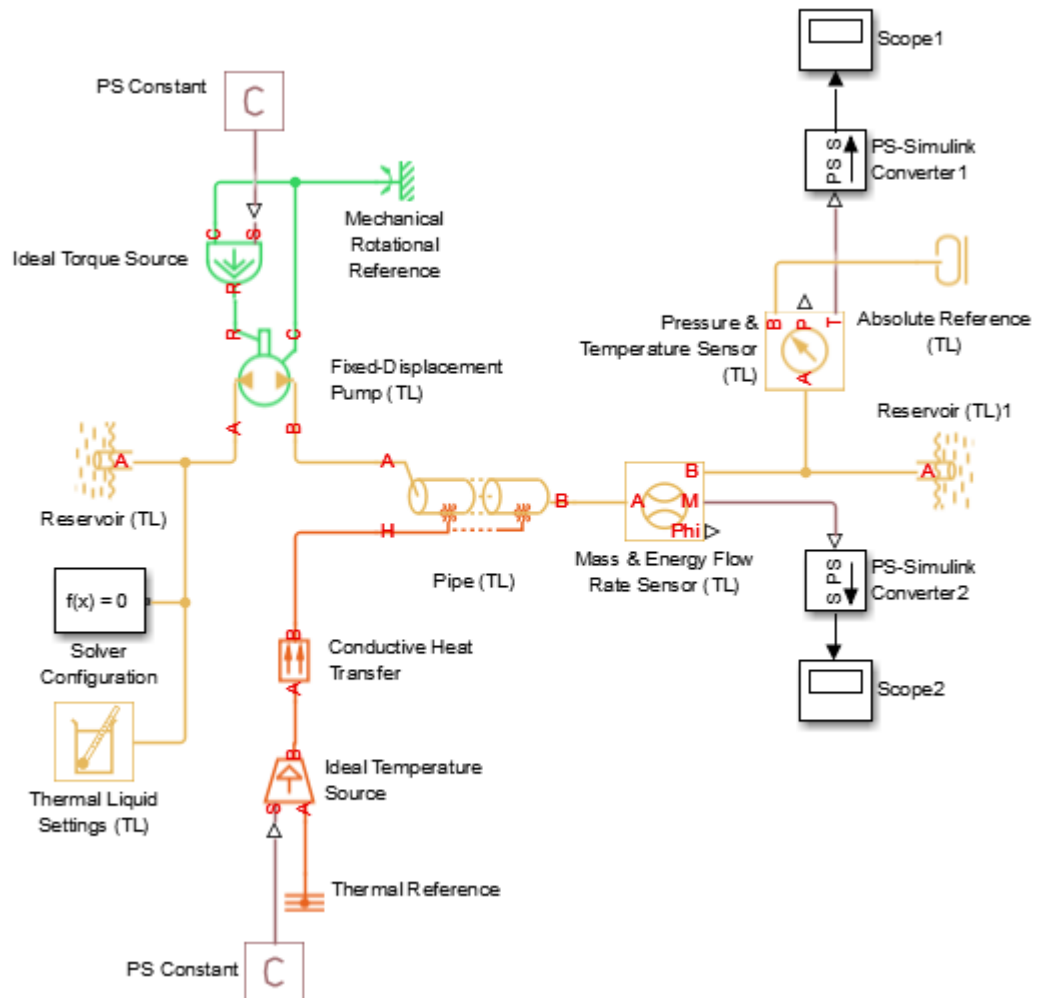
- 3 Drag these blocks to the model.

Block	Library
Pressure & Temperature Sensor (TL)	Simscape Foundation > Thermal Liquid > Sensors
Absolute Reference (TL)	Simscape Foundation > Thermal Liquid > Elements
PS-Simulink Converter	Simscape > Utilities

Block	Library
Scope	Simulink > Sinks

The sensor block measures the Across variables of the Thermal Liquid domain and outputs them as physical signals. The converter block transforms the physical signal into a Simulink signal for plotting in a Scope block.

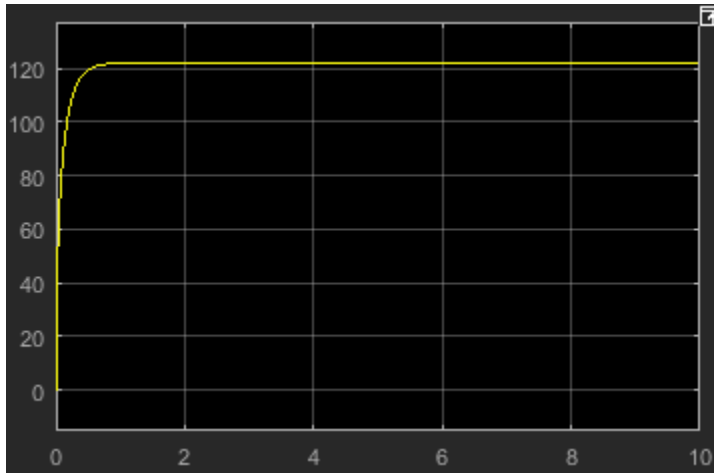
- 4 Connect the blocks as shown in the figure. Arrange the sensor block so that one conserving port connects to the Thermal Liquid branch being probed and the other to a known pressure and temperature reference, such as that provided by an Absolute Reference (TL) block. The sensor output is the difference between the measured and reference quantities, taken from port A to port B.



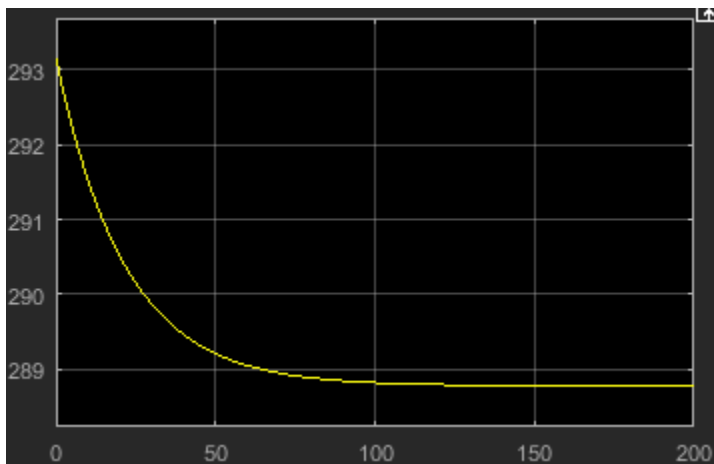
Simulate the Pipeline Model

The Simscape model template specifies suitable solver settings for this model. Run the simulation, for example, by clicking the  button in the Simulink toolstrip. Open the

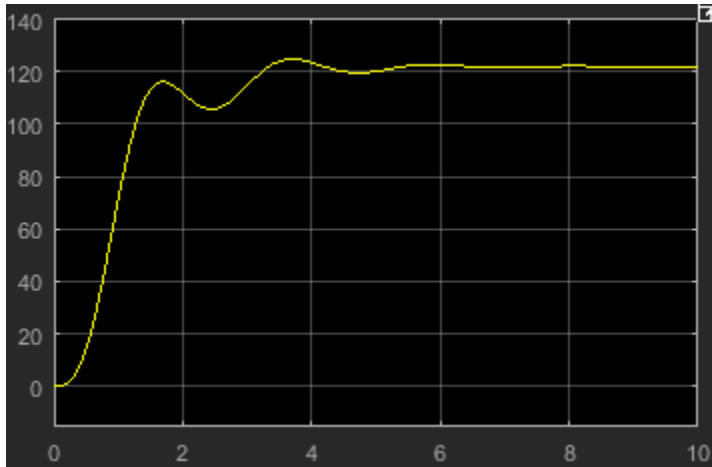
mass flow rate Scope block. The plot shows a gradual change in flow rate as the thermal liquid transitions from a state of rest to a new steady-state velocity.



The time scale for temperature changes is substantially greater than the 10-second simulation time. To see the new steady-state temperature, change the simulation stop time to 200 s. Then, simulate the model and open the temperature Scope block. The plot shows the gradual cooling at the pipe outlet as the thermal liquid transitions to a new steady-state temperature.



Experiment with the block settings and input signals. Try setting the Pipe (TL) block **Fluid inertia** parameter to 0n. The figure shows the combined effects of fluid dynamic compressibility and fluid inertia on the mass flow rate plot. The simulation stop time is 10 seconds for this plot.



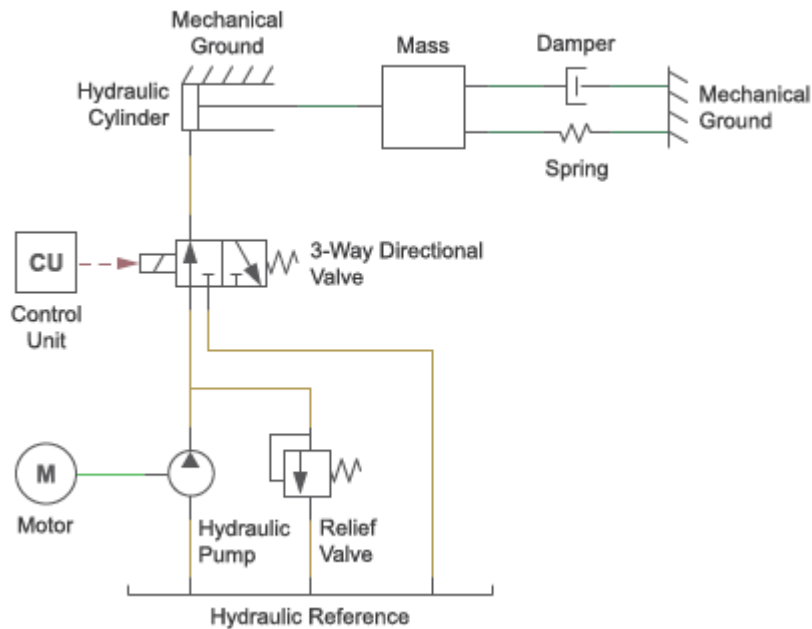
Model an Isothermal Hydraulic Actuator

In this section...
“Model Overview” on page 2-34
“Start a New Model” on page 2-36
“Model the Actuator” on page 2-37
“Model the Actuator Load” on page 2-39
“Add Sources and Scopes” on page 2-40
“Simulate the Actuator Model” on page 2-43

Model Overview

This tutorial shows how to model an isothermal hydraulic actuator using Simscape Fluids blocks. The actuator comprises a pump, a valve, and a cylinder. The pump provides power generation, the valve flow regulation, and the cylinder power conversion into mechanical translational form. A lumped mass-spring-damper system provides a simple model of the actuator load.

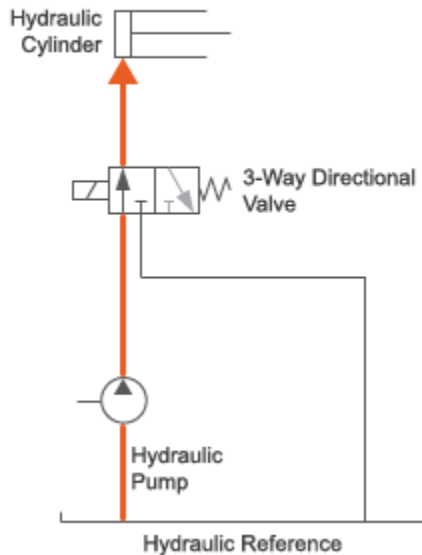
The figure shows a schematic of the complete actuator system. The model makes some simplifying assumptions. An ideal pressure source replaces the pump as a power source. Simple physical signal inputs replace the motor component and the control unit algorithm. The relief valve is unnecessary detail in this example and is ignored.



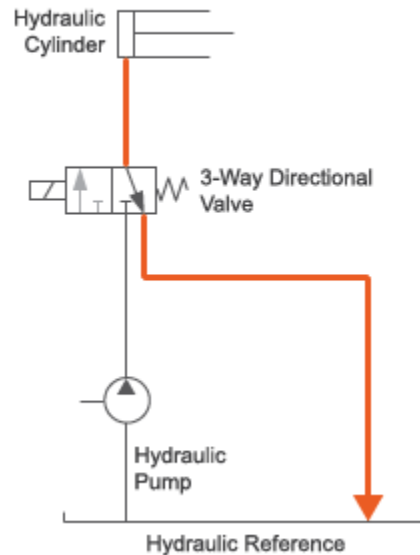
Hydraulic Actuator Schematic

The 3-way directional valve provides three ports and two control positions. The ports represent the cylinder, pump, and hydraulic reference inlets. The control position varies with the valve actuator input. Above 50% of the valve actuator nominal signal value, the valve opens the pump-cylinder flow path. The cylinder shaft extends against the actuator load. Below 50% of the valve actuator nominal signal value, the valve opens the cylinder-reservoir flow path. The cylinder shaft retracts under the action of the actuator load.

Positive Valve Control Signal



Negative Valve Control Signal



Directional Valve Positions

Start a New Model

- 1 At the MATLAB command prompt, enter `ssc_new`. MATLAB opens the Simscape model template. This template provides a starting point for your Simscape Fluids model. Save the model often as you add new blocks.
- 2 From the **Hydraulics (Isothermal) > Hydraulic Utilities** library, drag a Hydraulic Fluid block to the model canvas. This block defines the physical properties of the hydraulic fluid, including its density, viscosity, and bulk modulus.
- 3 Connect the Solver Configuration and Hydraulic Fluid blocks as shown in the figure. The Solver Configuration block provides Simscape solver settings for your model. Each topologically distinct physical network requires one such block.



Model the Actuator

- 1 From the **Simscape > Foundation** library, drag these blocks to the model canvas.

Block	Sublibrary
Hydraulic Constant Pressure Source	Hydraulic > Hydraulic Sources
Hydraulic Reference	Hydraulic > Hydraulic Elements

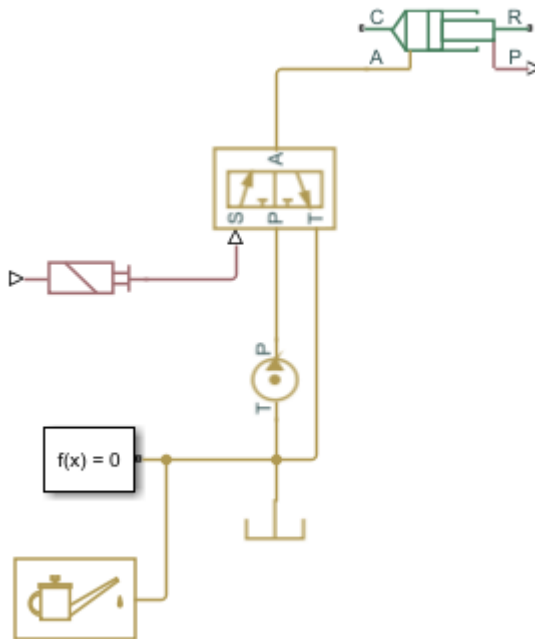
The pressure source represents the pump. The hydraulic reference represents the reservoir that the pump draws fluid from.

- 2 From the **Simscape > Fluids** library, drag these blocks to the model canvas.

Block	Sublibrary
Single-Acting Hydraulic Cylinder	Hydraulics (Isothermal) > Hydraulic Cylinders
3-Way Directional Valve	Hydraulics (Isothermal) > Valves
2-Position Valve Actuator	Hydraulics (Isothermal) > Valves

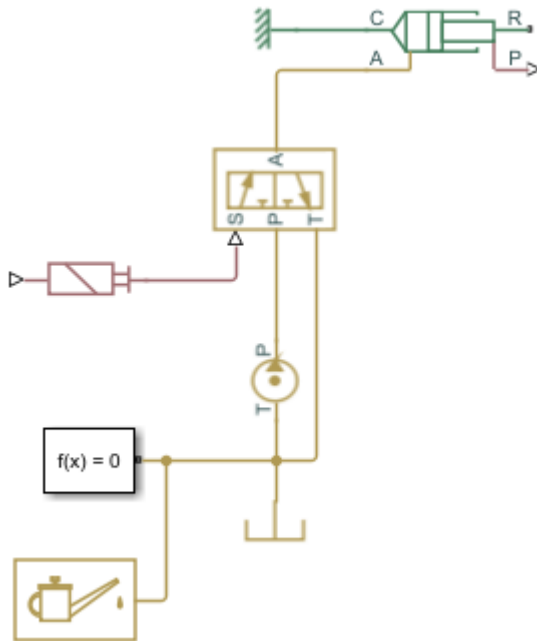
The hydraulic cylinder converts the hydraulic power generated by the pressure source into a mechanical translational form. The directional valve and valve actuator regulate the flow to the cylinder. In so doing, they control the mechanical power generated at the cylinder.

- 3 Connect the blocks as shown in the figure. The 3-Way Directional Valve and Hydraulic Constant Pressure Source blocks must both connect to the Hydraulic Reference block.



The block connections mirror the physical connections between the actual components. The cylinder connects to the valve, which connects to the pump, which in turn connects to the fluid reservoir.

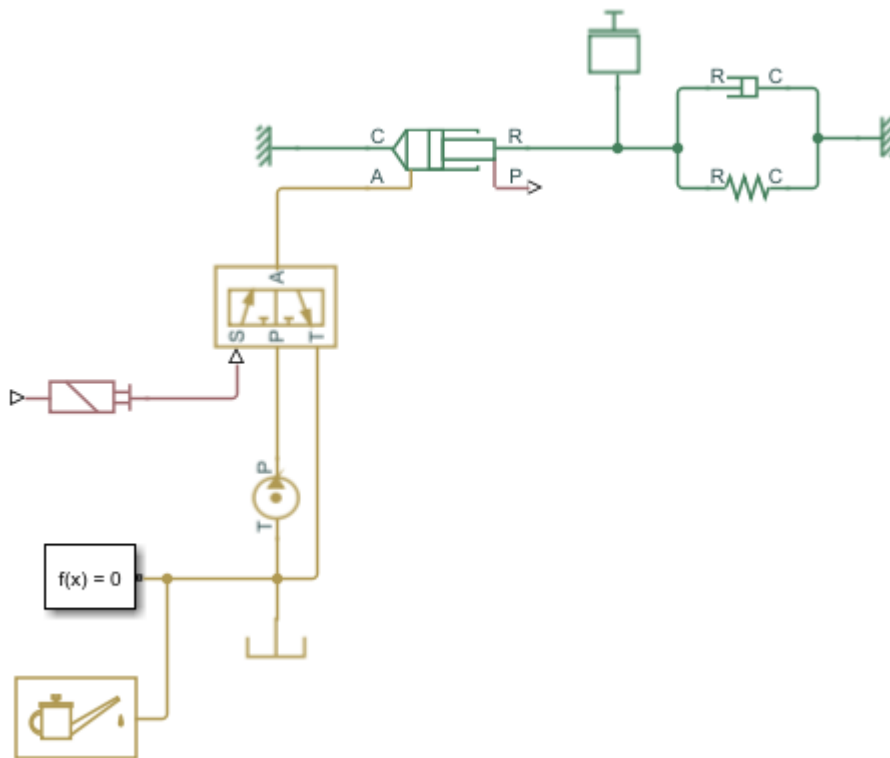
- 4 In the 3-Way Directional Valve block dialog box, click the **Valve Opening Offsets** tab and set the **Between A-T** parameter to 0.01 m.
- 5 From the **Simscape > Foundation > Mechanical > Translational** library, add a Mechanical Translational Reference block and connect it as shown in the figure.



The connection line fixes the hydraulic cylinder casing, represented by port c, to a mechanical ground so that only the shaft, represented by port R, is capable of motion.

Model the Actuator Load

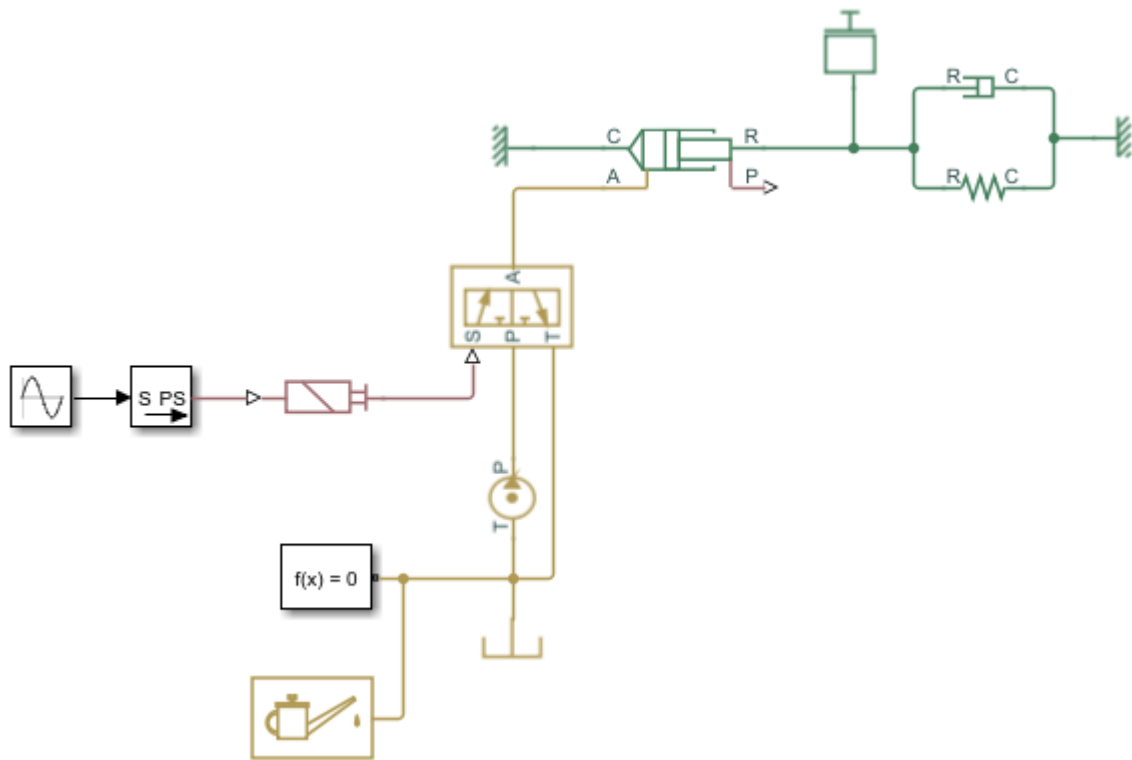
- 1 From the **Simscape** > **Foundation** > **Mechanical** > **Translational Elements** library, drag these blocks to the model canvas:
 - Mass
 - Translational Spring
 - Translational Damper
 - Mechanical Translational Reference
- 2 Connect the blocks as shown in the figure.



The mass, spring, and damper elements provide a simple model of the actuator load. The mass accounts for kinetic energy storage, the spring for potential energy storage, and the damper for viscous energy dissipation due to motion.

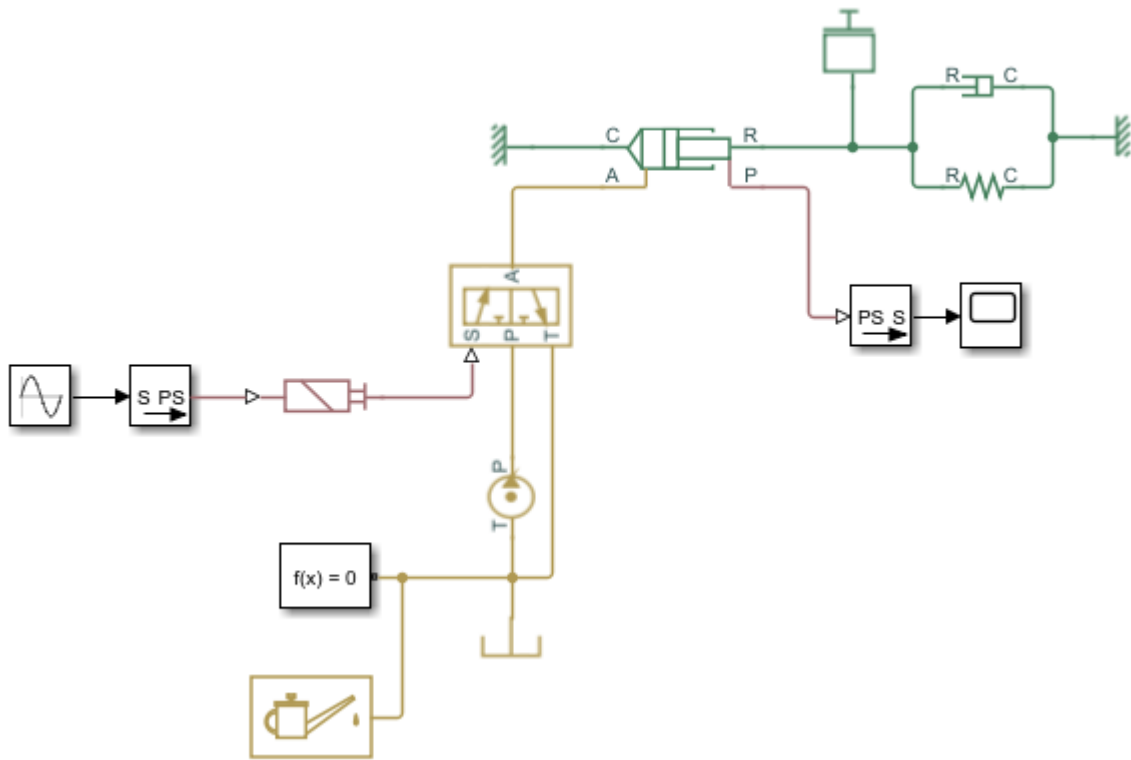
Add Sources and Scopes

- 1 From the Simulink Sources library, drag a Sine Wave and two Scope blocks.
- 2 From the Simscape Utilities library, drag one Simulink-PS Converter and two PS-Simulink Converter blocks.
- 3 Connect the Sine Wave, Simulink-PS Converter, and 2-Position Valve Actuator blocks in series as shown in the figure.



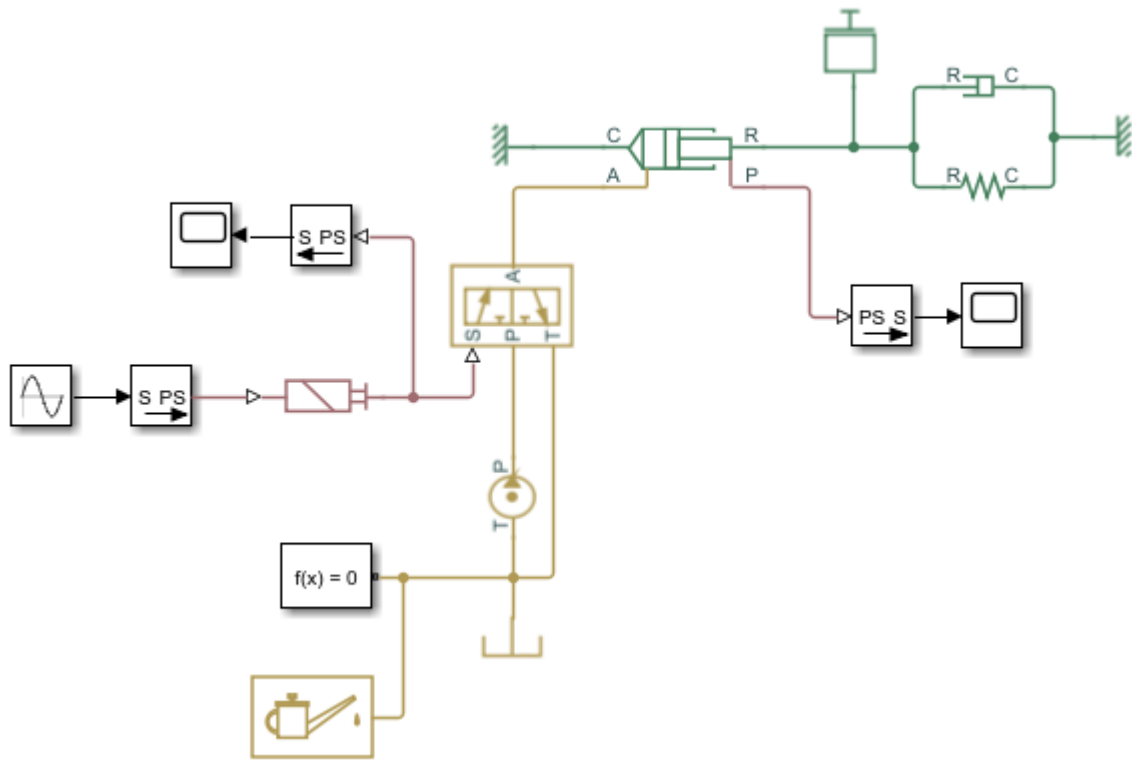
The Sine Wave block provides a sinusoidal control input to the 2-Position Valve Actuator block. The Simulink-PS Converter block converts the sinusoidal input from a Simulink signal into a physical signal compatible with Simscape Multibody blocks.

- 4 In the Sine Wave block dialog box, set the **Amplitude** parameter to 24. This value corresponds to the default nominal signal value of the 2-Position Valve Actuator block.
- 5 Connect the Single-Acting Hydraulic Cylinder block, a PS-Simulink Converter block, and a Scope block in series as shown in the figure.



Port **P** of the Single-Acting Hydraulic Cylinder block outputs a physical signal with the measured cylinder shaft position. The PS-Simulink Converter block converts the physical signal into a Simulink signal for plotting with the Simulink Scope block.


- 6 Connect the output side of the 2-Position Valve Actuator block, a PS-Simulink Converter block, and a Scope block in series as shown in the figure.



The output side of the 2-Position Valve Actuator block outputs a physical signal with the measured valve opening. The PS-Simulink Converter block converts the physical signal into a Simulink signal for plotting with the Simulink Scope block.

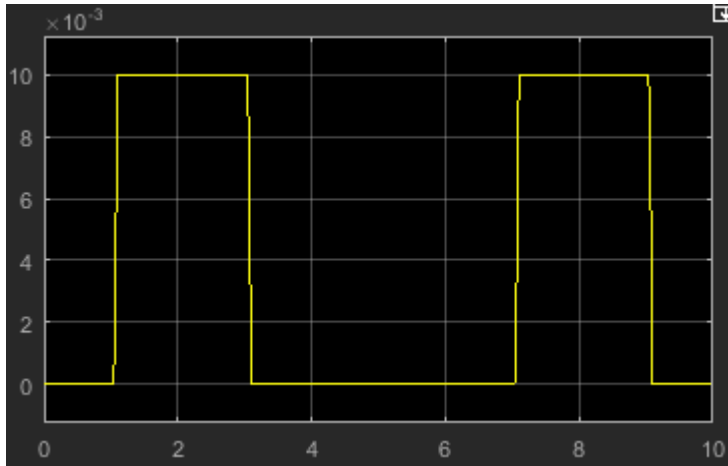
Simulate the Actuator Model

1

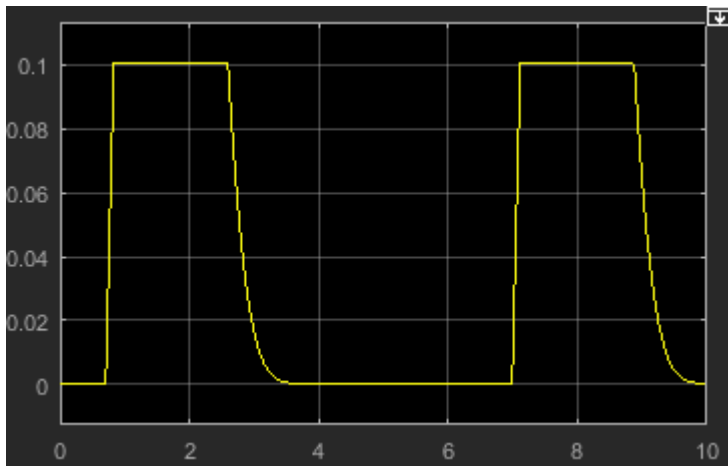
Run the simulation. You can do this from the Simulink menu bar, by clicking the  button. Simscape software evaluates the model, determines the initial conditions, and simulates the model. For more information, see “How Simscape Simulation Works” (Simscape).

2 Open the Scope blocks.

The valve opening plot shows the dynamic response of the valve to the sinusoidal wave input signal.



The cylinder position plot shows the dynamic response of the cylinder shaft to the pressure changes in the cylinder chamber.



The valve is initially closed. When the valve control signal reaches 50% of the actuator nominal signal, the valve gradually opens to allow flow between the pump and the

cylinder. The cylinder shaft moves in the positive direction until it reaches the maximum position allowed by the piston stroke.

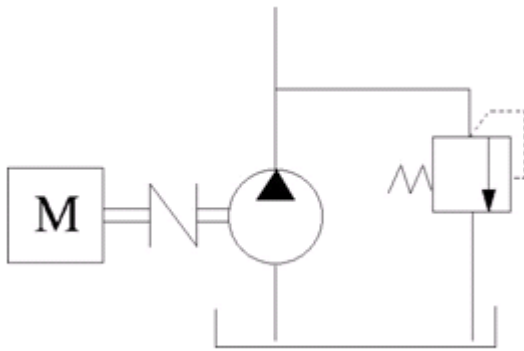
When the valve signal goes below 50% of the actuator nominal signal, the valve gradually closes to prevent flow from the pump to the cylinder. The spring element in the actuator load pushes back against the cylinder shaft, forcing it to return to its retracted position.

Experiment with different valve inputs and block parameter values. Simulate the model under different conditions to see the effect of different inputs and parameters on the valve opening dynamics and cylinder shaft displacement.

Modeling Power Units

The power unit is perhaps the most prevalent unit in hydraulic systems. Its main function is to supply the required amount of fluid under specified pressure. There is a wide variety of power unit designs varying by the amount and type of pumps, prime movers, valves, tanks, etc. The set of blocks available in the Simscape Fluids libraries allows you to simulate practically any of these configurations. This section considers basic approaches in simulating power units and examples of typical schematics.

A typical power unit of a hydraulic system, as shown in the following illustration, consists of a fixed-displacement or variable-displacement pump, reservoir, pressure-relief valve, and a prime mover that drives the hydraulic pump.



Typical Hydraulic Power Unit

In developing a model of a power unit, you must reach a compromise between the robustness, speed of simulation, and accuracy, meaning that the model should be as simple as possible to provide acceptable accuracy within the working range of variable parameters.

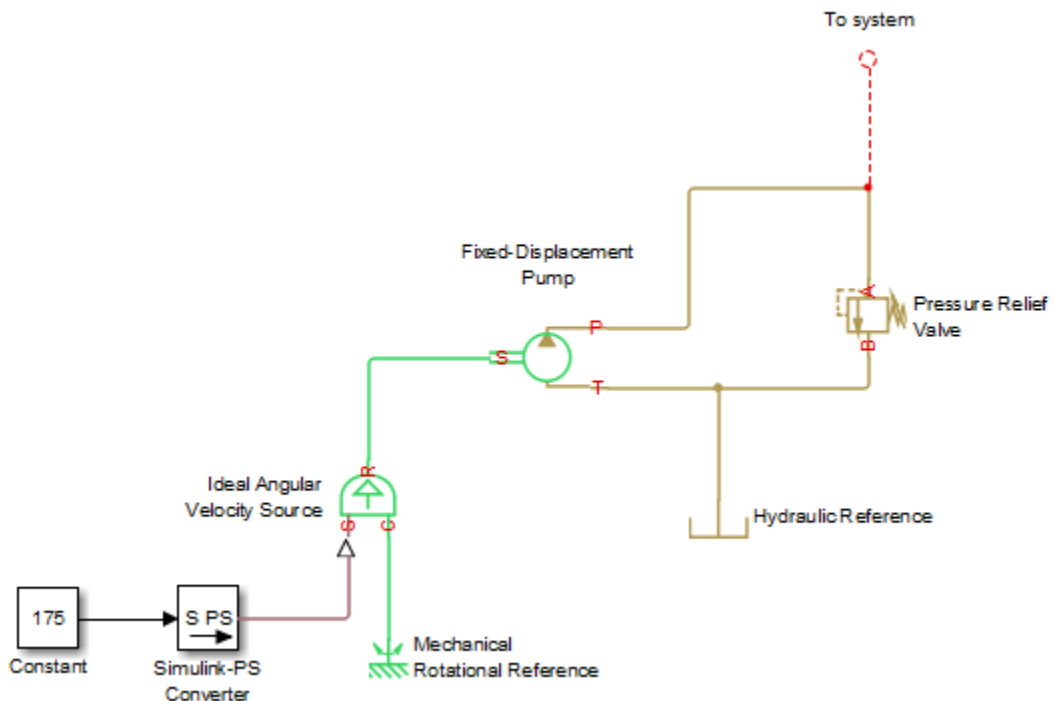
The first option is to simulate a power unit literally, as it is, reproducing all its components. This approach is illustrated in the Closed-Loop Pump Control with Flexible Drive Shaft example. The power unit consists of a fixed-displacement pump, which is driven by a motor through a compliant transmission, a pressure-relief valve, and a variable orifice, which simulates system fluid consumption. The motor model is represented as a source of angular velocity rotating 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 opened, which results in a low system pressure and the maximum flow rate going to the system. The orifice starts closing at 0.5 s, and is closed completely at 3 s. The output pressure builds up until it reaches the pressure setting of the relief valve ($75e5$ Pa) and is maintained at this level by the valve. At 3 s, the variable orifice starts opening, thus returning system to its initial state.

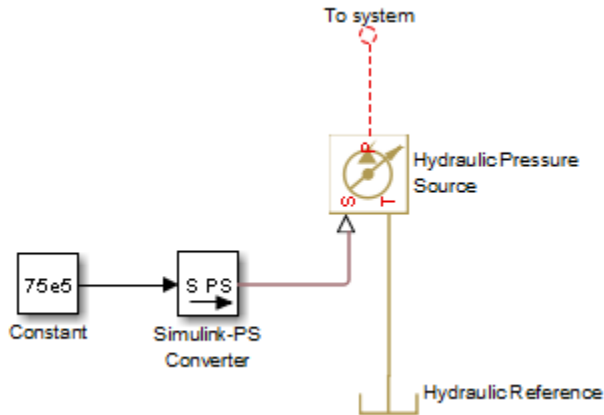
You can implement a considerably more complex model of a prime mover by following the pattern used in the example. For instance, the shaft can be represented with multiple segments and intermediate bearings. The model of a prime mover can be more comprehensive, accounting for its type (DC or AC electric motor, diesel or gasoline engine), characteristics, control type, and so on. In addition, a complex mechanical transmission driven by a diesel or gasoline internal combustion engine modeled using Simscape Driveline software can be combined with the Simscape Fluids model of the hydraulic portion of a power unit.

Depending on your particular application, you may be able to simplify the model of a power unit practically without a loss in accuracy. The main factors to be considered in this process are the driving shaft angular velocity variation magnitude and the system pressure variation range. If the prime mover angular velocity remains practically constant during simulated time or varies insignificantly with respect to its steady-state value, the entire driving shaft subsystem can be replaced with the Ideal Angular Velocity Source block, whose output is set to the steady-state value, as it is shown in the following illustration.



Using the Ideal Angular Velocity Source Block in Modeling Power Units

Furthermore, if pump delivery exceeds the system's fluid requirements at all times, the pump output pressure remains practically constant and close to the pressure setting of the pressure-relief valve. If this assumption is true and acceptable, the entire power unit can be reduced to an ideal Hydraulic Pressure Source block, as shown in the next illustration.



Using the Hydraulic Pressure Source Block in Modeling Power Units

The two previous examples demonstrate that the use of ideal sources is a powerful means of reducing the complexity of models. However, you should exercise extreme caution every time you use an ideal source instead of a real pump. The substitution is possible only if there is an assurance that the controlled parameter (angular velocity in the first example, and pressure in the second example) remains constant. If this is not the case, the power unit represented with an ideal source will generate considerably more power than its simulated physical counterpart, thus making the simulation results incorrect.

Modeling Directional Valves

In this section...

“Types of Directional Valves” on page 2-50

“4-Way Directional Valve Configurations” on page 2-51

“Building Blocks and How to Use Them” on page 2-56

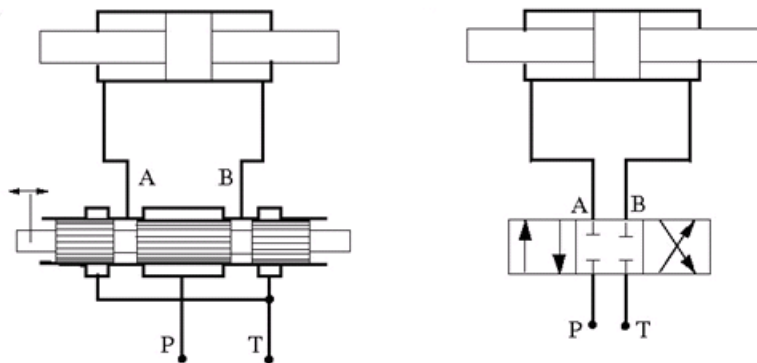
“Build a Custom Directional Valve” on page 2-60

Types of Directional Valves

The main function of directional valves in hydraulic systems is to direct and distribute flow between consumers. As far as the valve modeling is concerned, the valves are classified by the following main characteristics:

- Number of external paths (connecting ports) — One-way, two-way, three-way, four-way, multiple-way
- Number of positions a control member of the valve can assume — Two-position, three-position, multiple-position, continuous (can assume any position within working range)
- Control member type — Spool, poppet, sliding flat spool, and so on

As an example, the following illustration shows a portion of a hydraulic system with a 4-way, 3-position directional valve controlling a double-acting cylinder, next to its schematic diagram.



Throughout Simscape Fluids libraries, Hydraulic (isothermal liquid) ports are identified with the following symbols:

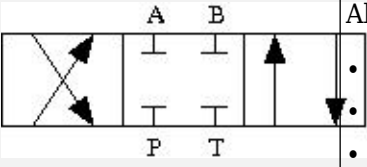
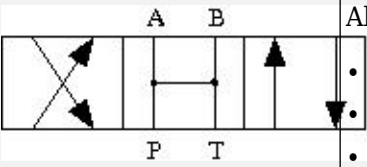
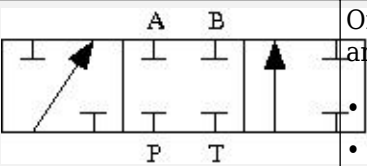
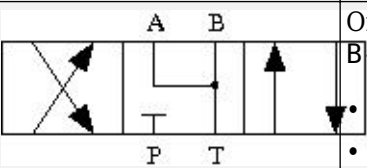
- P — Pressure port
- T — Return (tank) port
- A, B — Actuator ports
- X, Y — Pilot or control ports

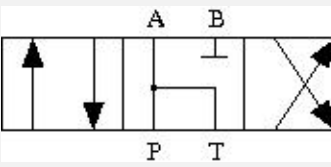
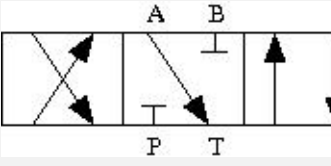
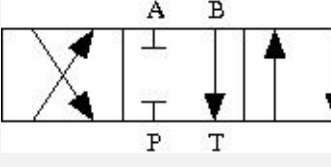
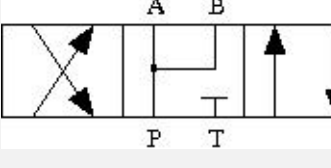
4-Way Directional Valve Configurations

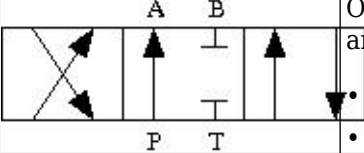
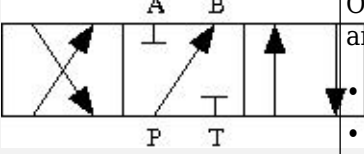
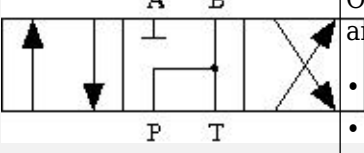
4-way directional valves are available in multiple configurations, depending on the port connections in three distinctive valve positions: leftmost, neutral, and rightmost. Each configuration is characterized by the number of variable orifices, the way the orifices are connected, and initial openings of the orifices. Ten 4-way directional valve blocks in Simscape Fluids libraries represent twenty most typical valve configurations. Configurations that differ only by the values of initial openings are covered by the same model.

The basic 4-Way Directional Valve block lets you model eleven most popular configurations by changing the initial openings of the orifices, as shown in the following table.

Basic 4-Way Directional Valve Configurations

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

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

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

The other nine configurations are covered by the remaining 4-way directional valve blocks (A through K), as shown in the next table.

Other 4-Way Directional Valve Blocks

Block Name	Configuration	Description
4-Way Directional Valve A		<p>Contains two additional normally-open, sequentially-located orifices. Valve displacement to the left or to the right closes the path to tank.</p>
4-Way Directional Valve B		<p>Ports P and A are permanently connected through fixed orifice. Orifices P-B and B-T are initially open (underlapped).</p>
4-Way Directional Valve C		<p>Ports P and B are permanently connected through fixed orifice. Orifices P-A, A-T and B-T are initially open (underlapped).</p>
4-Way Directional Valve D		<p>Two orifices are installed in the P-A link. Port A never connects to port T.</p>
4-Way Directional Valve E		<p>Two orifices are installed in the P-B link. Port B never connects to port T.</p>
4-Way Directional Valve F		<p>Two parallel orifices in the P-A arm and two series orifices in the A-T arm.</p>

Block Name	Configuration	Description
4-Way Directional Valve G		Two parallel orifices in the P-B arm and two series orifices in the B-T arm.
4-Way Directional Valve H		Two parallel orifices in the P-B arm and two series orifices in the P-T arm.
4-Way Directional Valve K		Two parallel orifices in the P-A arm and two series orifices in the P-T arm.

Building Blocks and How to Use Them

The Directional Valves library offers several prebuilt directional valve models. As indicated in their descriptions, all of them are symmetrical, continuous valves. In other words, the control member in 2-way, 3-way, 4-way, and 6-way valves can assume any position, controlled by the physical signal port S. The valves are symmetrical in that all the orifices the valve is built of are of the same type and size. The only possible difference between orifices is the orifice initial opening.

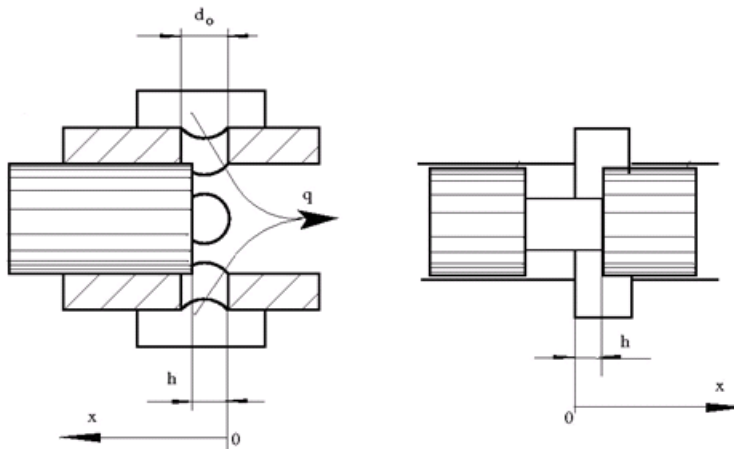
These configurations cover a substantial portion of real valves, but the directional valves family is so diverse as to make it practically impossible to have a library model for every member. Instead, Simscape Fluids libraries offer a set of building blocks that is comprehensive enough to build a model for any real configuration. This section describes the rules of building a custom model of a directional valve.

All directional valve models are built of variable orifices. In Simscape Fluids libraries, the following variable orifice models are available:

- Annular Orifice
- Orifice with Variable Area Round Holes

- Orifice with Variable Area Slot
- Variable Orifice
- Ball Valve
- Needle Valve
- Poppet Valve

To simplify the way variable orifices are combined in a model, their instantaneous opening is computed in the same way for all types of orifices. The orifice opening is always computed in the direction the spool, or any other control member, opens the orifice. In other words, positive value of the opening corresponds to open orifice, while negative value denotes overlapped, or closed, orifice. The origin always corresponds to zero-lap position, when the edge of the control member coincides with the edge of the orifice. In the illustration below, origins are marked with θ for the orifice with variable area round holes (schematic on the left) and for the orifice with variable slot (schematic on the right). The x arrow denotes the direction in which orifice opening is measured in both cases.



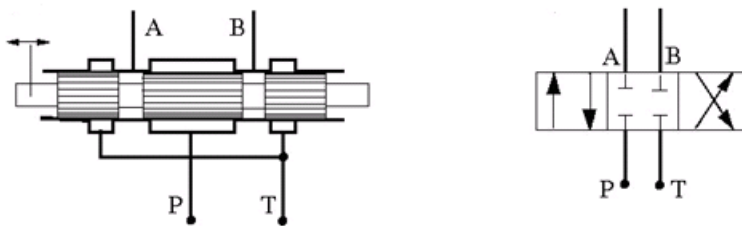
The instantaneous value of the orifice opening is determined as

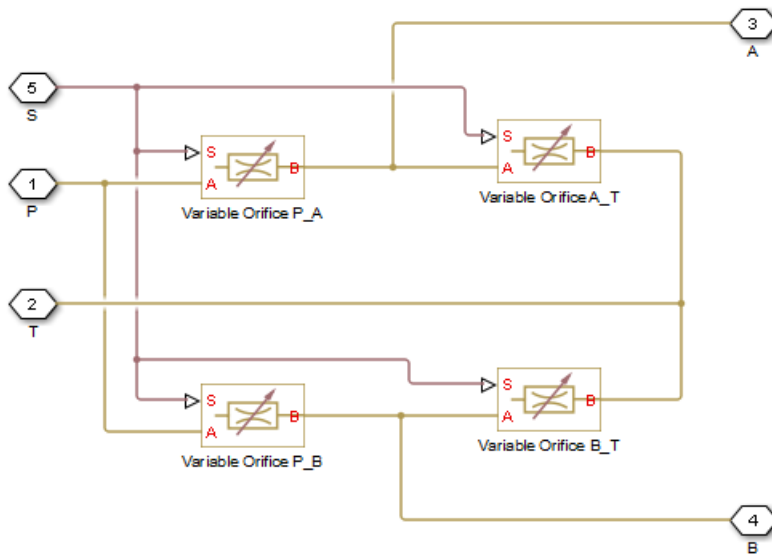
$$h = x_0 + x_{sp} \cdot \text{OR}$$

where

h	Instantaneous orifice opening.
x_{θ}	Initial opening. The initial opening value is positive for initially open (underlapped) orifices and negative for overlapped orifices.
x_{sp}	Spool (or other control member) displacement from initial position, which controls the orifice.
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.

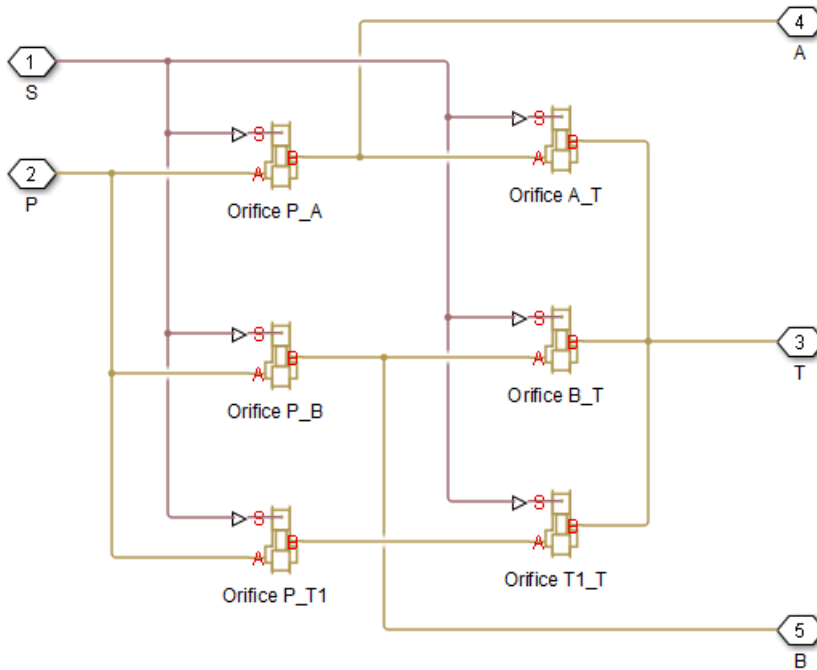
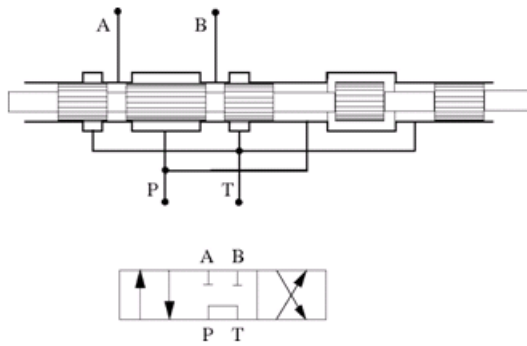
The number of variable orifices and the way they are connected are determined by the valve design. Usually, the model of a valve mimics the physical layout of a real valve. The illustration below shows an example of a 4-way valve, its symbol, and an equivalent circuit of its Simscape Fluids model.





The 4-way valve in its simplest form is built of four variable orifices. In the equivalent circuit, they are named P_A, P_B, A_T, and B_T. The Variable Orifice block, which is the most generic model of a variable orifice in the Simscape Fluids libraries, is used in this particular example. You can use any other variable orifice blocks if the real valve design employs a configuration backed by a stock model, such as an orifice with round holes or rectangular slots, poppet, ball, or needle. In general, all orifices in the model can be simulated with different blocks or with the same block, but with different way of parameterization. For instance, two orifices can be represented by their pressure-flow characteristics, while two others can be simulated with the table-specified area variation option (for details, see the Variable Orifice block reference page).

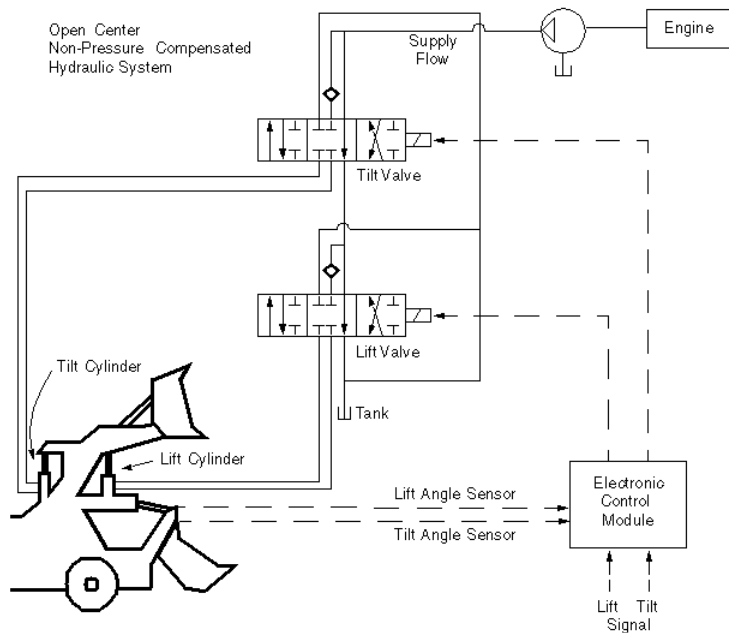
The next example shows another configuration of a 4-way directional valve. This valve unloads the pump in neutral position and requires six variable orifice blocks. The Orifice with Variable Area Round Holes blocks have been used as a variable orifice in this model. Port T1 corresponds to an intermediate point between ports P and T.



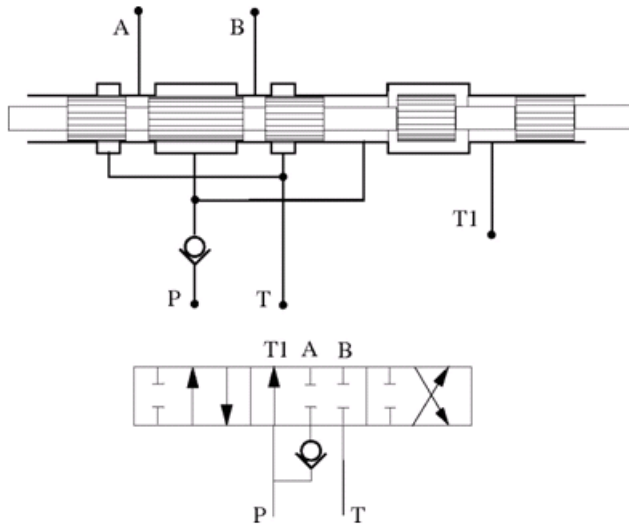
Build a Custom Directional Valve

Finally, let us consider a more complex directional valve example. The figure below shows basic elements of a front loader hydraulic system. Both the lift and the tilt cylinders are

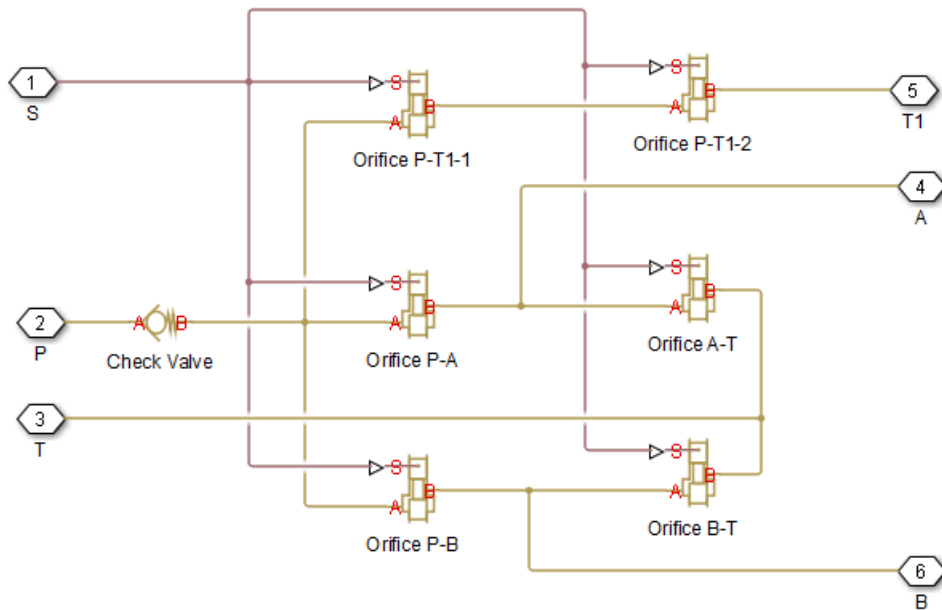
controlled by custom 3-position, 5-way valves, developed for this particular application. The valves are designed in such a way that the pump delivery is diverted to tank (unloaded) if both cylinders are commanded to be in neutral position. The pump is disconnected from the tank if either of the two control valves is shifted from neutral position.



To develop a model, the physical version of the valve must be created first. The following illustration shows one of the possible configurations of the valve.



The Simscape Fluids model, shown below, is an exact copy of the physical valve configuration.



All the orifices in the model are closed (overlapped) in valve neutral position, except orifices P_T1_1 and P_T1_2. These two orifices should be set open to an extent that allows pump delivery to be discharged at low pressure.

Modeling Low-Pressure Fluid Transportation Systems

In this section...

“How Fluid Transportation Systems Differ from Power and Control Systems” on page 2-64

“Available Blocks and How to Use Them” on page 2-66

“Low-Pressure Fluid Transportation System” on page 2-68

How Fluid Transportation Systems Differ from Power and Control Systems

In hydraulics, the steady uniform flow in a component with one entrance and one exit is characterized by the following energy equation

$$-\frac{\dot{W}_s}{\dot{m}g} = \frac{V_2^2 - V_1^2}{2g} + \frac{p_2 - p_1}{\rho g} + z_2 - z_1 + h_L$$

where

\dot{W}_s	Work rate performed by fluid
\dot{m}	Mass flow rate
V_2	Fluid velocity at the exit
V_1	Fluid velocity at the entrance
p_1, p_2	Static pressure at the entrance and the exit, respectively
g	Gravity acceleration
ρ	Fluid density
z_1, z_2	Elevation above a reference plane (datum) at the entrance and the exit, respectively
h_L	Hydraulic loss

Subscripts 1 and 2 refer to the entrance and exit, respectively. All the terms in “Equation 2-1” on page 2-64 have dimensions of height and are named kinematic head, piezometric head, geometric head, and loss head, respectively. For a variety of reasons,

analysis of hydraulic power and control systems is performed with respect to pressures, rather than to heads, and “Equation 2-1” on page 2-64 for a typical passive component is presented in the form

$$\frac{\rho}{2}V_1^2 + p_1 + \rho gz_1 = \frac{\rho}{2}V_2^2 + p_2 + \rho gz_2 + p_L$$

where

V_1, p_1, z_1	Velocity, static pressure, and elevation at the entrance, respectively
V_2, p_2, z_2	Velocity, static pressure, and elevation at the exit, respectively
p_L	Pressure loss

Term $\frac{\rho}{2}V^2$ is frequently referred to as kinematic, or dynamic, pressure, and ρgz as piezometric pressure. Dynamic pressure terms are usually neglected because they are very small, and “Equation 2-2” on page 2-65 takes the form

$$p_1 + \rho gz_1 = p_2 + \rho gz_2 + p_L$$

The size of a typical power and control system is usually small and rarely exceeds 1.5 - 2 m. To add to this, these systems operate at pressures in the range 50 - 300 bar.

Therefore, ρgz terms are negligibly small compared to static pressures. As a result, Simscape Fluids components (with the exception of the ones designed specifically for low-pressure simulation, described in “Available Blocks and How to Use Them” on page 2-66) have been developed with respect to static pressures, with the following equations

$$p = p_L = f(q)$$

$$q = f(p_1, p_2)$$

where

p	Pressure difference between component ports
q	Flow rate through the component

Fluid transportation systems usually operate at low pressures (about 2-4 bar), and the difference in component elevation with respect to reference plane can be very large.

Therefore, geometrical head becomes an essential part of the energy balance and must be accounted for. In other words, the low-pressure fluid transportation systems must be

simulated with respect to piezometric pressures $p_{pz} = p + \rho gz$, rather than static pressures. This requirement is reflected in the component equations

$$p = p_L = f(q, z_1, z_2)$$

$$q = f(p_1, p_2, z_1, z_2)$$

Equations in the form “Equation 2-5” on page 2-66 must be applied to describe a hydraulic component with significant difference between port elevations. In hydraulic systems, there is only one type of such components: hydraulic pipes. The models of pipes intended to be used in low pressure systems must account for difference in elevation of their ports. The dimensions of the rest of the components are too small to contribute noticeably to energy balance, and their models can be built with the constant elevation assumption, like all the other Simscape Fluids blocks. To sum it up:

- You can build models of low-pressure systems with difference in elevations of their components using regular Simscape Fluids blocks, with the exception of the pipes. Use low-pressure pipes, described in “Available Blocks and How to Use Them” on page 2-66 .
- When modeling low-pressure systems, you must use low-pressure pipe blocks to connect all nodes with difference in elevation, because these are the only blocks that provide information about the vertical locations of the system parts. Nodes connected with any other blocks, such as valves, orifices, actuators, and so on, will be treated as if they have the same elevation.

Available Blocks and How to Use Them

When modeling low-pressure hydraulic systems, use the pipe blocks from the Low-Pressure Blocks library instead of the regular pipe blocks. These blocks account for the port elevation above reference plane and differ in the extent of idealization, just like their high-pressure counterparts:

- Resistive Pipe LP — Models hydraulic pipe with circular and noncircular cross sections and accounts for friction loss only, similar to the Resistive Tube block, available in the Simscape Foundation library.
- Resistive Pipe LP with Variable Elevation — Models hydraulic pipe with circular and noncircular cross sections and accounts for friction losses and variable port elevations.

Use this block for low-pressure system simulation in which the pipe ends change their positions with respect to the reference plane.

- Hydraulic Pipe LP — Models hydraulic pipe with circular and noncircular cross sections and accounts for friction loss along the pipe length and for fluid compressibility, similar to the Hydraulic Pipeline block in the Pipelines library.
- Hydraulic Pipe LP with Variable Elevation — Models hydraulic pipe with circular and noncircular cross sections and accounts for friction loss along the pipe length and for fluid compressibility, as well as variable port elevations. Use this block for low-pressure system simulation in which the pipe ends change their positions with respect to the reference plane.
- Segmented Pipe LP — Models circular hydraulic pipe and accounts for friction loss, fluid compressibility, and fluid inertia, similar to the Segmented Pipe block in the Pipelines library.

Use these low-pressure pipe blocks to connect all Hydraulic nodes in your model with difference in elevation, because these are the only blocks that provide information about the vertical location of the ports. Nodes connected with any other blocks, such as valves, orifices, actuators, and so on, will be treated as if they have the same elevation.

The additional models of pressurized tanks available for low-pressure system simulation include:

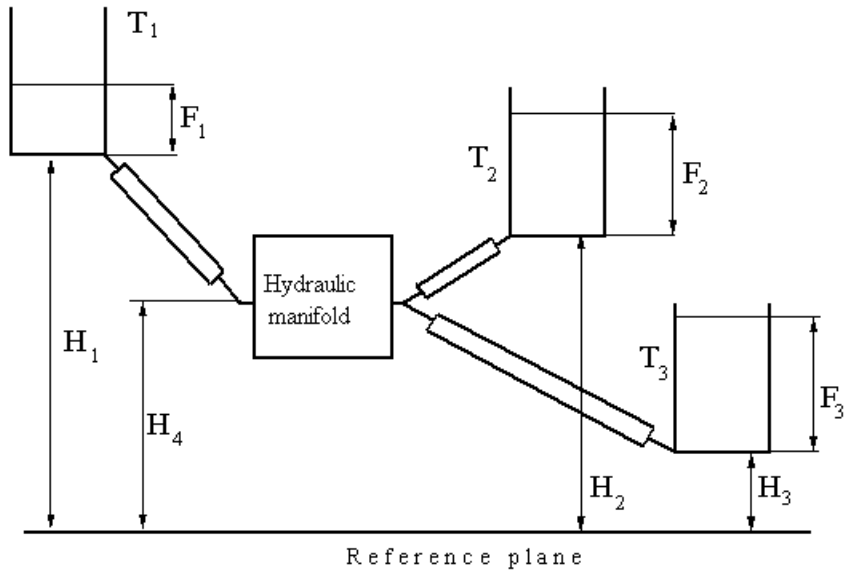
- Constant Head Tank — Represents a pressurized hydraulic reservoir, in which fluid is stored under a specified pressure. The size of the tank is assumed to be large enough to neglect the pressurization and fluid level change due to fluid volume. The block accounts for the fluid level elevation with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

of volume change. The block accounts for the fluid level change caused by the volume variation, as well as for pressure loss in the connecting pipes that can be caused by a filter, fittings, or some other local resistance. The loss is specified with the pressure loss coefficient at each outlet. The block computes the volume of fluid in the tank and exports it outside through the physical signal port V .

- Variable Head Three-Arm Tank — Represents a three-arm pressurized tank, in which fluid is stored under a specified pressure. The pressurization remains constant regardless of volume change. The block accounts for the fluid level change caused by the volume variation, as well as for pressure loss in the connecting pipes that can be caused by a filter, fittings, or some other local resistance. The loss is specified with the pressure loss coefficient at each outlet. The block computes the volume of fluid in the tank and exports it outside through the physical signal port V .

Low-Pressure Fluid Transportation System

The following illustration shows a simple system consisting of three tanks whose bottom surfaces are located at heights H_1 , H_2 , and H_3 , respectively, from the reference plane. The tanks are connected by pipes to a hydraulic manifold, which may contain any hydraulic elements, such as valves, orifices, pumps, accumulators, other pipes, and so on, but these elements have one feature in common - their elevations are all the same and equal to H_4 .



The models of tanks account for the fluid level heights F_1 , F_2 , and F_3 , respectively, and represent pressure at their bottoms as

$$p_i = \rho g F_i \quad \text{for } i = 1, 2, 3$$

The components inside the manifold can be simulated with regular Simscape Fluids blocks, like you would use for hydraulic power and control systems simulation. The pipes must be simulated with one of the low-pressure pipe models: Resistive Pipe LP, Hydraulic Pipe LP, or Segmented Pipe LP, depending on the required extent of idealization. Use the Constant Head Tank or Variable Head Tank blocks to simulate the tanks. For details of implementation, see the Water Supply System and the Three Constant Head Tanks examples.

Pump and Motor Characteristic Curves

In this section...

“What Are Characteristic Curves?” on page 2-70

“Supported Blocks and Block Variants” on page 2-70

“Performance, Efficiency, and Loss Curves” on page 2-71

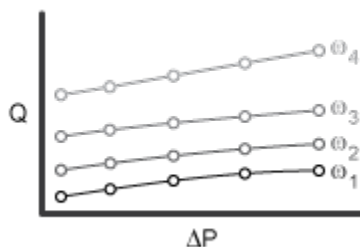
“How Characteristic Curves Are Generated” on page 2-74

“Example 1: Plot Performance Curves” on page 2-75

“Example 2: Plot Efficiency Curves” on page 2-77

What Are Characteristic Curves?

A characteristic curve is a component-specific plot that shows how a change in one dynamic variable might affect others. Such curves capture the performance of a component over a range of operating conditions and are commonly used in component selection, sizing, and analysis. To aid in these tasks, Simscape Fluids software enables you to plot the characteristic curves of select components.



Supported Blocks and Block Variants

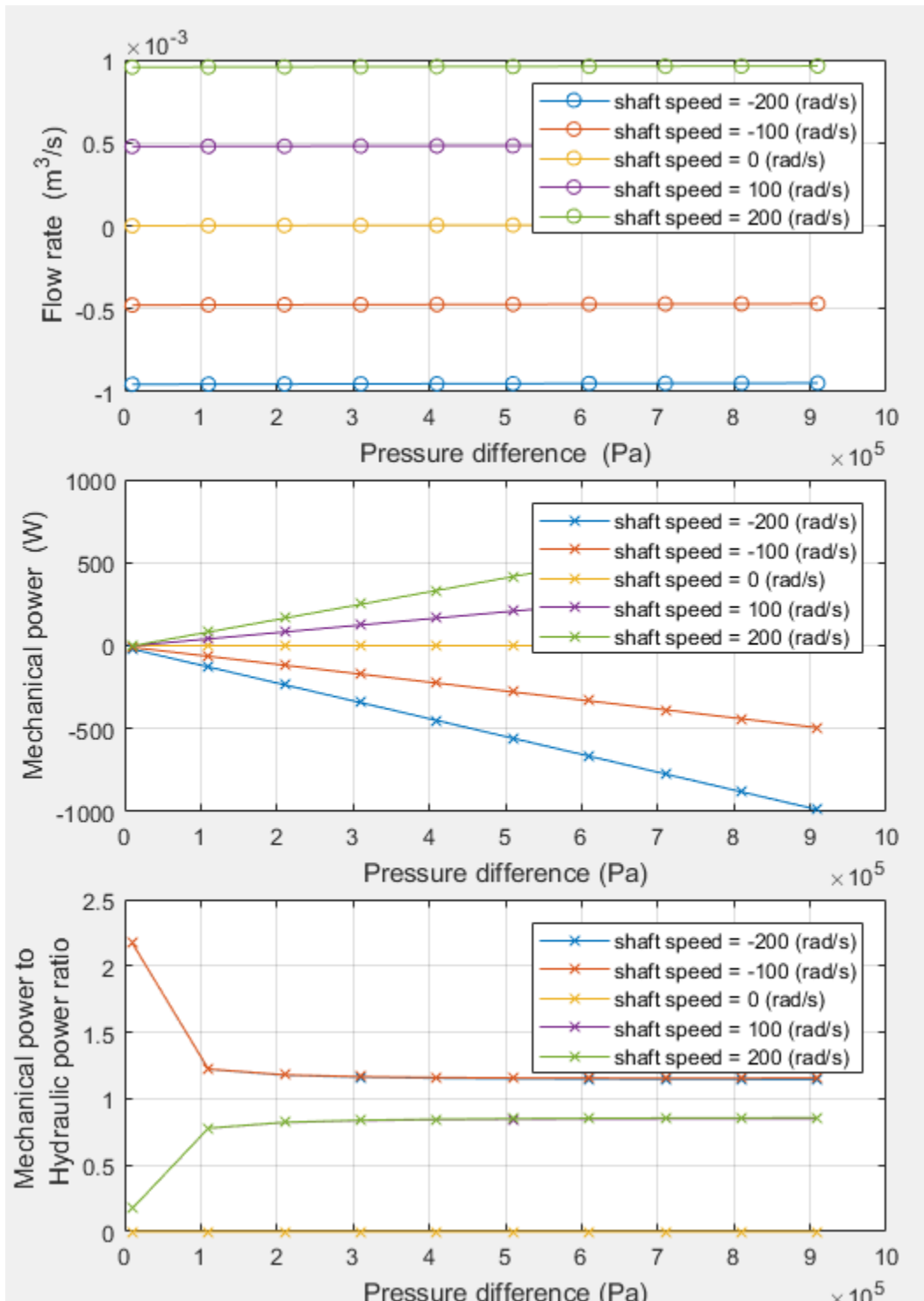
You can plot the characteristic curves of the following blocks from the Hydraulics (Isothermal) library:

- Fixed-Displacement Pump
- Fixed-Displacement Motor

These components have variants that you can select from. Only one variant enables you to plot characteristic curves—the default **Analytical** or **tabulated data**. To check or change the block variant, right-click the block to open its context-sensitive menu and select **Simscape > Block Choices**. Selecting the **Analytical** or **tabulated data** variant adds a **Fluids > Plot characteristic** option to the context-sensitive menu.

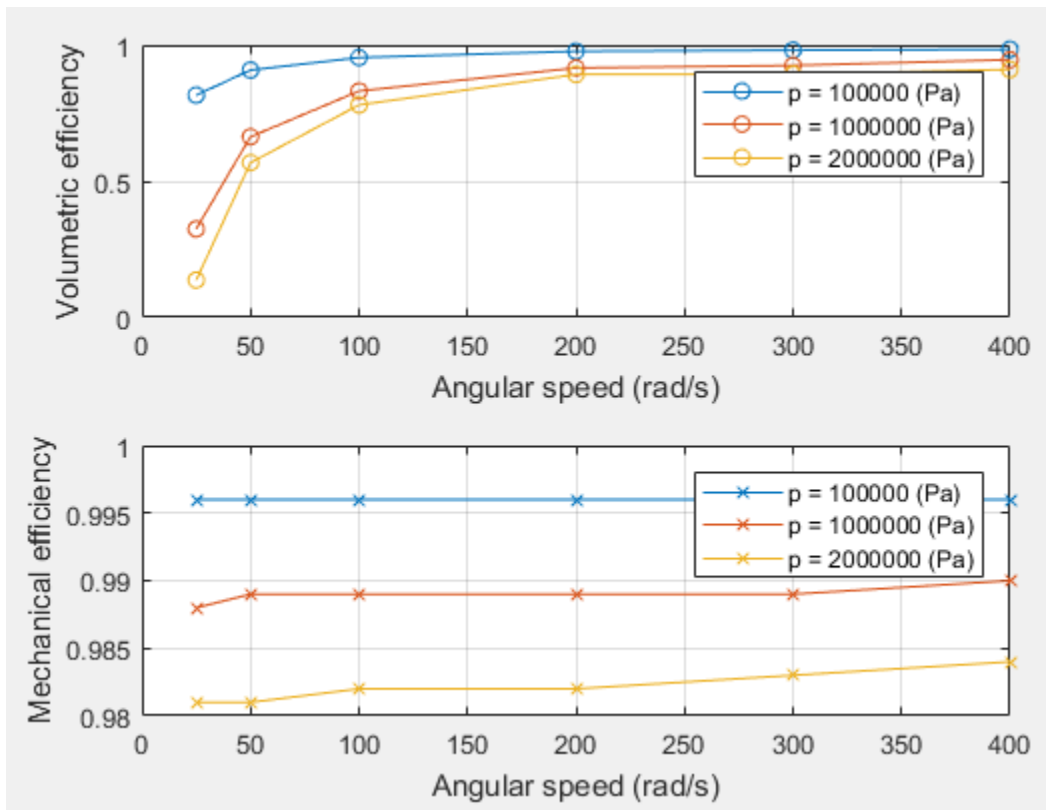
Performance, Efficiency, and Loss Curves

The types of characteristic curves that you can visualize depend on the block parameterization that you select. Three performance curves are always shown—one of the volumetric flow rate, one of the mechanical power, and one of the mechanical-hydraulic power ratio. The pressure difference and output shaft speed serve as the independent variables in each case. The pressure difference is shown on the horizontal axis. The shaft speed is shown through a series of curves, one for each value of the shaft speed.



If the **Leakage and friction parameterization** parameter is set to a Tabulated data option, two additional curves are shown. These curves are of the volumetric and mechanical efficiencies for the Tabulated data – volumetric and mechanical efficiencies parameterization and of the volumetric and mechanical losses for the Tabulated data – volumetric and mechanical losses parameterization.

Here, too, the shaft speed and pressure difference serve as the independent variables. However, the shaft speed is now the variable on the horizontal axis. The pressure difference is now shown through a series of curves, one for each value of the pressure difference. The values plotted are those specified directly in the block dialog boxes. The purpose of these curves is to provide a means to visually check your tabulated data inputs for inadvertent errors.



Efficiency Curves of a Fixed-Displacement Motor in the Default Configuration

The table summarizes the types of curves that Simscape Fluids generates for the different block parameterizations. Notice that this table applies only to the Fixed-Displacement Motor and Fixed-Displacement Pump blocks in the **Hydraulics > Pumps and Motors** library.

		Characteristic Curves		
		Performance	Efficiency	Loss
Parameterization	Analytical	✓		
	Tabulated data - volumetric and mechanical efficiencies	✓	✓	
	Tabulated data - volumetric and mechanical losses	✓		✓

How Characteristic Curves Are Generated

Simscape Fluids generates the characteristic curves of a component from a test harness for that component. The test harness comprises a simple model with blocks representing the component of interest, the hydraulic operating conditions, the mechanical operating conditions, and the fluid properties. You open the test harness from the block context-sensitive menu. Right-click the block and select **Fluids > Plot Characteristic**.

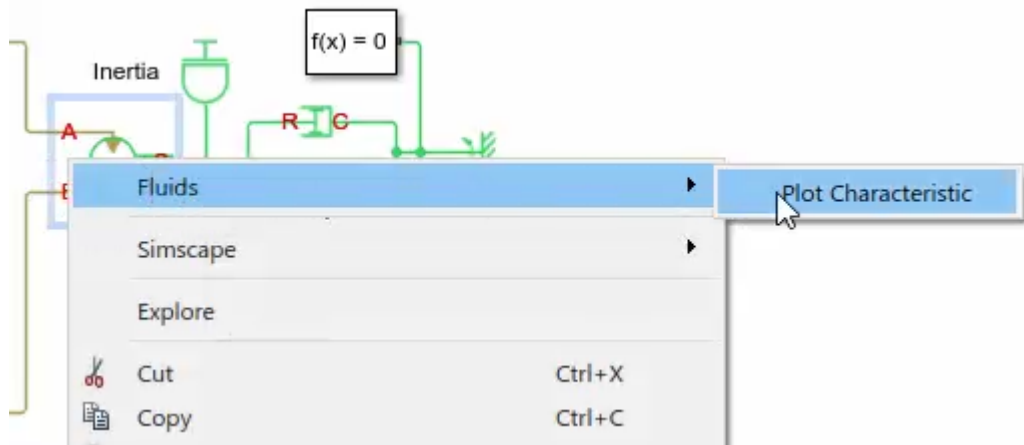
The model simulates under a variety of operating conditions—at different shaft speeds and pressure differences—with the simulation results providing the data need to plot the performance curves. You can change the fluid properties and block parameters in the source model or in the test harness before simulation. If you change block parameters in the source model, you must regenerate the test harness by right-clicking the block and selecting **Fluids > Plot characteristic** again.

Changes that you make to the block parameters do not carry over to the source model, even after you close the test harness. If you intend to keep any changes to the test block, consider making those changes in the source model directly.

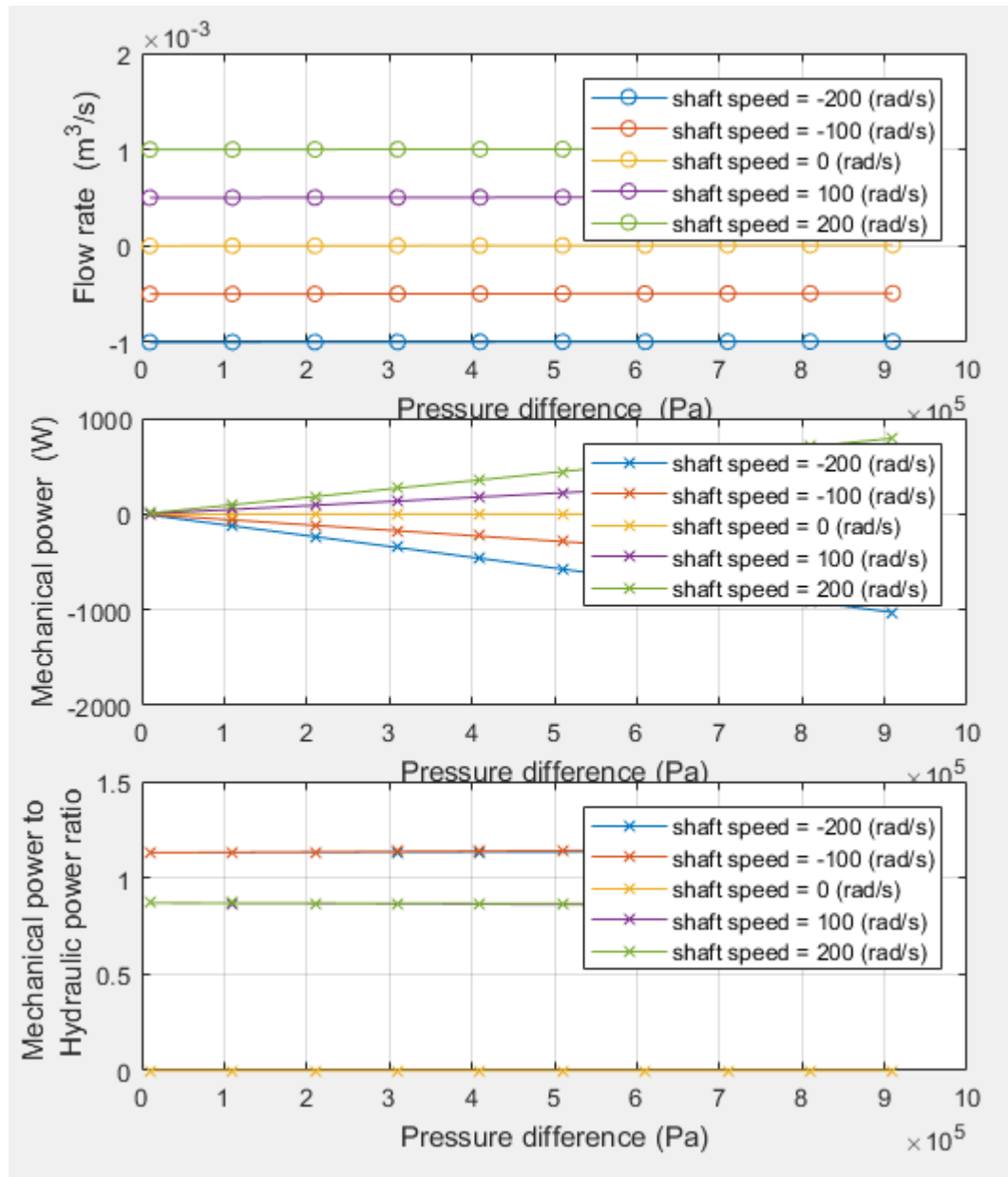
Example 1: Plot Performance Curves

As an example, plot the performance curves of a fixed-displacement motor in a hydrostatic transmission model. You can use these curves to analyze the performance of the motor under different operating conditions.

- 1 At the MATLAB command prompt, enter `sh_hydrostatic_trans`. A Simscape Fluids model of a hydrostatic transmission system opens. The model contains a Fixed-Displacement Motor block, named Motor, with the **Leakage and friction parameterization** parameter set to Analytical.
- 2 Right-click the Motor block and select **Fluids > Plot Characteristic**. A test harness opens with a list of steps needed to generate the characteristic curves. Step 2, **Define range of pressure and angular velocity**, provides the range of operating conditions under which to simulate the motor.



- 3 Click step 4, **Generate plots**. Simscape Fluids simulates the motor component at various shaft speeds and pressure differences and uses the simulation results to generate the characteristic curves. The shaft speeds and pressure differences are the defaults specified in step 2, **Define range of pressure and angular velocity**.



Notice that the characteristic curves generated in the Analytical parameterization do not include efficiency or loss curves. The purpose of such curves is to visually check any input efficiency or loss data, and this data is reduced to a constant in the Analytical parameterization.

Example 2: Plot Efficiency Curves

Specify the motor efficiency in tabulated form and visually check the specified data for errors by plotting the efficiency curves. This example is based on the same `sh_hydrostatic_trans` model used to plot the characteristic curves of a fixed-displacement motor in the Analytical parameterization.

- 1 In the dialog box of the Fixed-Displacement Motor block named Motor, set the **Leakage and friction parameterization** parameter to `Tabulated data – volumetric and mechanical efficiencies`. The data ranges of the **Pressure drop vector for efficiencies, dp** and **Shaft angular velocity vector for efficiencies, w** parameters replace those entered in step 2 of the motor test harness for the Analytical parameterization.
- 2 In the field for the **Volumetric efficiency table, e_v(dp,w)** parameter, enter the matrix shown below. This matrix is similar to the default efficiency matrix but contains an intentional numerical error.


```
[0.816, 0.908, 0.954, 0.977, 0.981, 0.984;...
 0.325, 0.663, 0.831, 0.916, 0.925, 0.0946;...
 0.137, 0.568, 0.78, 0.892, 0.893, 0.91]
```
- 3 In the block diagram, right-click the Motor block and select **Fluids > Plot Characteristic**. A new test harness opens for the motor component.
- 4 Click step 4, **Generate plots**. Simscape Fluids simulates the motor component and generates two sets of curves—the performance curves, similar to those obtained in “Example 1: Plot Performance Curves” on page 2-75, and the efficiency curves showing the efficiency tabular data.

The figure shows the volumetric efficiency curves. The curve corresponding to a pressure difference of 1,000,000 Pa reveals that the data point corresponding to a shaft speed of 400 rad/s is a likely numerical error and should be fixed before simulation.

