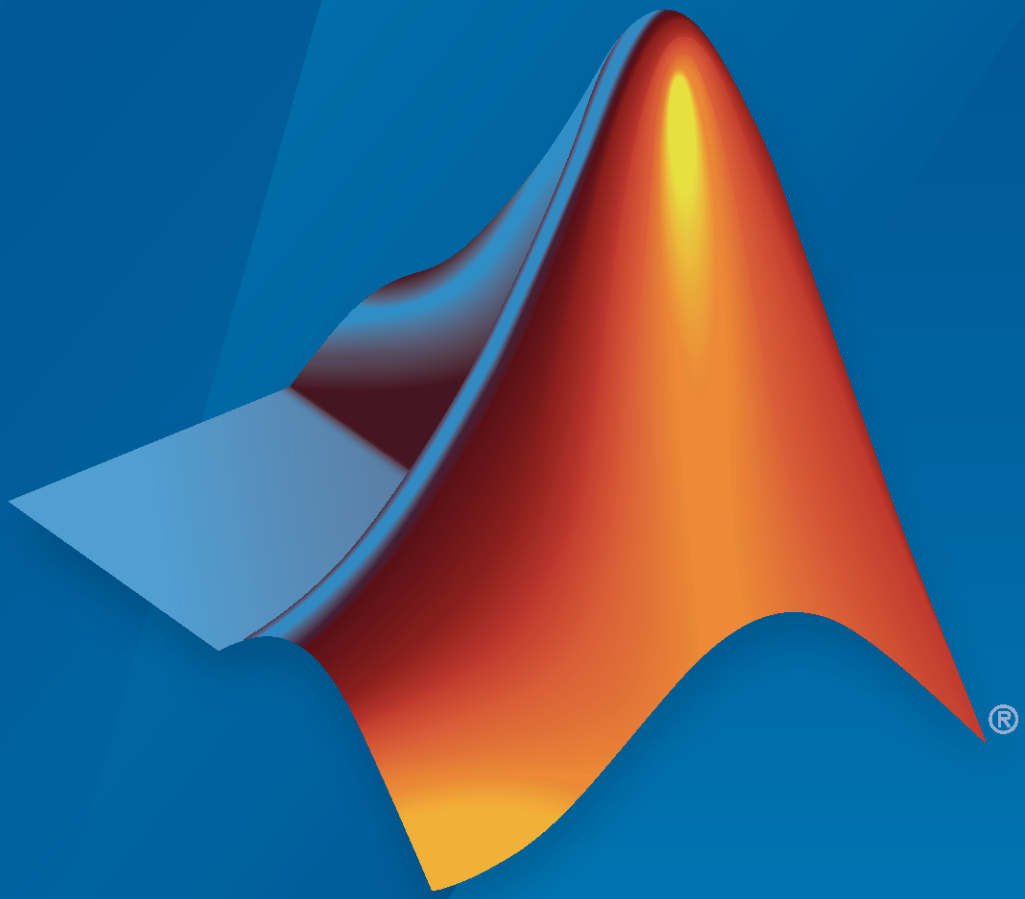
Simscape™ Fluids™

User's Guide



MATLAB® & SIMULINK®

R2021b

MathWorks®

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.
1 Apple Hill Drive
Natick, MA 01760-2098

Simscape™ Fluids™ User's Guide

© COPYRIGHT 2006–2021 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 *SimHydraulics® User's Guide*)
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)
September 2018	Online only	Revised for Version 2.5 (Release R2018b)
March 2019	Online only	Revised for Version 2.6 (Release R2019a)
September 2019	Online only	Revised for Version 2.7 (Release R2019b)
March 2020	Online only	Revised for Version 3.0 (Release R2020a)
September 2020	Online only	Revised for Version 3.1 (Release R2020b)
March 2021	Online only	Revised for Version 3.2 (Release R2021a)
September 2021	Online only	Revised for Version 3.3 (Release R2021b)

Contents

1 Get Started

Simscape Fluids Product Description **1-2**
Key Features **1-2**

Select the Working Fluid **1-3**
Specify Fluid Properties **1-4**

Modeling and Simulation of Fluid Networks **1-6**
Parameterizing a Block **1-6**
Start and Simulate Your Model **1-7**
Modifying Parameters **1-7**
Balancing Accuracy, Speed, and Efficiency in Your Model **1-8**

Networks and Network Connections **1-9**
Grounding Rules **1-9**

Considerations For Building Fluids Models **1-14**

2 Modeling Fluid Dynamic Systems

Block Equations and the Simscape Numerical Scheme **2-2**
System Equations **2-2**
Solving the Network: The Simscape Numerical Scheme **2-2**
Smoothing in the Simscape Numerical Scheme **2-3**

Model a Thermal Liquid Pipeline **2-6**
Start a New Model **2-7**
Model the Pipeline **2-7**
Introduce Thermal Conduction **2-8**
Model the Pump Torque **2-11**
Add Thermal Liquid Sensors **2-12**
Simulate the Pipeline Model **2-15**

Model an Isothermal Actuation System **2-16**
Model Schematic **2-16**
Start a New Model **2-17**
Model the Actuator Subsystem **2-18**
Add a Load Model **2-20**
Add a Control System and Monitoring **2-21**
Run the Simulation **2-23**

Modeling Power Units **2-28**

Modeling Valves in Simscape Fluids **2-31**
Valve Control with Valve Actuator Blocks **2-31**
Four-Way Directional Valve Schematic **2-31**
Parameterizing Four-Way Directional Valve Blocks **2-33**

Building a Custom Valve **2-37**
Fundamental Directional Valve Composite Structures **2-37**
Custom Directional Valve for a Front Loader **2-39**

Modeling Hydraulic Directional Valves **2-42**
Classifying Directional Valves **2-42**
4-Way Directional Valve Schematic **2-42**
Building a Custom Directional Valve **2-47**

Modeling Low-Pressure Fluid Transportation Systems **2-52**
How Fluid Transportation Systems Differ from Power and Control Systems **2-52**
Available Blocks and How to Use Them **2-53**
Low-Pressure Fluid Transportation System **2-55**

Numerical Smoothing **2-56**
Zero-Crossings **2-56**
Bounded Variables **2-56**

3 Verification and Validation

Validating Flow Response With the Simulation Data Inspector **3-2**
Build a Test Harness **3-3**
Collect Simulation Data **3-3**
Assess Model Results **3-4**

4 How To

Upgrading Simscape Fluids Models Containing Hydraulics (Isothermal) Blocks **4-2**
Special Pre-Conversion Considerations **4-2**
Upgrading Your Models **4-4**
Fixing Post-Conversion Warning Messages **4-12**
Block Conversions **4-16**

5 Simscape Fluids Examples

Gerotor Pump 5-2

Pressure Control Servo Valve 5-5

Pressure Intensifier 5-10

Water Hammer Effect 5-13

Brayton Cycle (Gas Turbine) 5-18

4-Way 3-Position Valve Parameterization 5-26

Electro-Hydraulic Limited Slip Differential 5-28

Pressure Control Solenoid 5-32

Lubrication System 5-36

Priority Valve Controlling Two Hydraulic Motors 5-40

Front-Loader Actuation System 5-42

Drill-Ream Actuator 5-45

Electric Vehicle Thermal Management 5-51

Power-Assisted Steering Mechanism 5-69

Hydraulic Clutch System 5-77

Secondary Cylinder 5-80

Single-Stage Primary Cylinder 5-82

ABS open loop test bench 5-84

Flow Divider 5-88

Anti-Lock Braking System (ABS) 5-90

Sequencing Circuit for Two Rotary Actuators 5-94

Hydraulic Axial-Piston Pump with Load-Sensing and Pressure-Limiting Control 5-98

Hydrostatic Transmission 5-108

Closed-Circuit Hydraulic Actuator 5-113

Diesel Engine In-Line Injection System 5-117

Hydraulic Flow Rectifier Circuit **5-130**

Hydraulic Actuator with Dual Counterbalance Valves **5-133**

Partially Filled Pipe **5-138**

Servo-Valve Controlling Hydraulic Actuator **5-142**

Air Liquefaction **5-147**

Washing Machine Fault Analysis **5-152**

Vacuum Booster **5-155**

Tandem Primary Cylinder **5-159**

Vacuum Boosted Tandem Primary Cylinder **5-165**

Residential Refrigeration Unit **5-169**

Single-Acting Cylinder with 3-Way Valve **5-179**

Closed-Loop Actuator with Variable-Displacement Pressure-Compensated Pump **5-182**

Closed-Loop Hydraulic Actuator Model for Real-Time Simulation **5-185**

Closed-Loop Actuator with 2-Way Valve **5-191**

Hydraulic Differential Cylinder with 3-Way Valve **5-193**

Hydraulic Cylinder with Flexible Mount **5-196**

Closed-Circuit Hydraulic Actuator **5-198**

Hydraulic Actuator with Counterbalance Valve **5-201**

Hydraulic Actuator with Dual Counterbalance Valves **5-205**

Custom Hydraulic Cylinder **5-211**

Hydraulic Cylinder with Two-Chamber Snubbers **5-213**

Hydraulic Actuator with Telescopic Cylinder **5-218**

Digital Hydraulic Actuator **5-221**

Rotating Hydraulic Actuator **5-224**

Oscillating Hydraulic Mechanism **5-227**

Sequencing Circuit for Two Rotary Actuators **5-229**

Hydrostatic Transmission **5-232**

Hydrostatic Transmission with Secondary Control **5-234**

Hydrostatic Transmission with Shuttle Valve **5-236**

2-Position Valve Actuator **5-238**

3-Position Valve Actuator **5-240**

Diesel Engine In-Line Injection System **5-242**

Injection Molding Actuation System **5-253**

Drill-Ream Actuator **5-261**

Front-Loader Actuation System **5-267**

Power-Assisted Steering Mechanism **5-270**

Lubrication System **5-274**

Closed-Loop Pump Control with Flexible Drive Shaft **5-278**

Pump with Pressure-Reducing Valve and Two Loads **5-280**

Hydraulic Motor Driven by Load-Sensing Pump **5-282**

Hydraulic Motor Driven by Load-Sensing Variable-Displacement Pump **5-284**

Hydraulic Axial-Piston Pump with Load-Sensing and Pressure-Limiting Control **5-287**

Hydraulic Flapper-Nozzle Amplifier **5-297**

Servo-Valve Controlling Hydraulic Actuator **5-299**

Pilot-Operated Check Valve Test Rig **5-304**

Priority Valve Controlling Two Hydraulic Motors **5-306**

Direct-Acting Pressure Relief Valve Test Rig **5-308**

Pressure-Compensated Flow Control Test Rig **5-310**

Hydraulic Flow Rectifier Circuit **5-312**

Aircraft Fuel Supply System with Three Tanks **5-314**

Aircraft Fuel Supply System with Three Tanks **5-322**

Water Supply System **5-328**

Three Constant Head Tanks **5-331**

Well with Jet Pump **5-333**

Flow Divider Test Rig **5-336**

Segmented Pipeline Test Rig **5-338**

Gas-Charged Accumulator Test Rig **5-340**

Spring-Loaded Accumulator Test Rig **5-342**

Constant Volume Chamber Test Rig **5-344**

Engine Cooling System **5-346**

Hydraulic Oil System with Thermal Control **5-354**

House Heating System **5-360**

Reciprocal Actuator with Differential Cylinders **5-367**

4-Way Directional Valve with Mechanical Effects **5-372**

Antagonistic McKibben Muscle Actuator **5-378**

Simple CPU Cooling System **5-383**

Pipe Fluid Vaporization and Condensation **5-388**

EV Battery Cooling System **5-394**

Condenser and Evaporator Heat Transfer **5-404**

Closed-Circuit Hydraulic Actuator **5-408**

Photovoltaic Thermal (PV/T) Hybrid Solar Panel **5-412**

Get Started

- “Simscape Fluids Product Description” on page 1-2
- “Select the Working Fluid” on page 1-3
- “Modeling and Simulation of Fluid Networks” on page 1-6
- “Networks and Network Connections” on page 1-9
- “Considerations For Building Fluids Models” on page 1-14

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

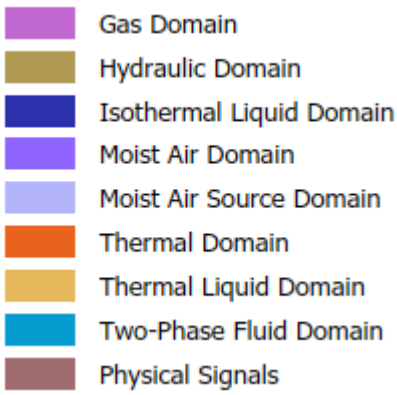
- Pump models, including centrifugal, jet, and axial-piston pumps
- Valve models, including check valves, directional valves, and pressure relief valves
- Rotational and translational actuator models, including optional friction and centrifugal forces
- Heat exchanger models for modeling heating and 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™)

Select the Working Fluid

The Simscape Fluids software enables you to model a variety of isothermal, thermal, and multi-phase systems. Key application areas include fluid transport, power generation, as well as heating and cooling. Simscape Fluids blocks are organized by domain:

- Isothermal Liquid: Use this domain to model adiabatic liquid systems. This library is based on mass flow rates and uniformly accounts for density as a function of pressure. This domain is best suited for models with little or no variation in temperature, or where changes in fluid viscosity do not significantly influence network dynamics, even if the network is at an elevated or reduced temperature.
- Hydraulic (Isothermal Liquid): Use this domain to model adiabatic liquid systems. This domain is best suited for models with little or no variation in temperature, or where changes in fluid viscosity do not significantly influence network dynamics, even if the network is at an elevated or reduced temperature.
- Thermal Liquid: Use this domain to model liquid systems with varying temperatures and viscosities, including systems with convective or conductive heat transfer. The fluid properties are functions of pressure and temperature.
- Gas: Use this domain to model a perfect, semiperfect, or real gas system. This domain is best suited for applications with one predominant network gas.
- Two-Phase Fluids: Use this domain to model systems where the working fluid may be both a liquid and a vapor. This domain does not support more than one fluid in the network. The model is based on the average fluid density in each phase; as a result, opalescence and the effect of gravity are not taken into account.
- Moist Air: Use this domain to model air-based systems with water vapor present. This domain is best suited for models where condensation or CO_2 fluctuations contribute to the dynamics of the system.

The connecting lines in each network are colored according to its domain:



Simscape Fluids Color Coding

If there is a functionality in another library that you would like to use in your model, consider using an interface block in the Fluid Network Interfaces library. These blocks allow you to connect two different networks if conditions are constant over the connection. You can also reduce the simulation time of a two-phase fluid model with an interface connection by making use of the gas and thermal

liquid domains in regions where the conditions are constant and have a vapor quality of 1 or 0, respectively.

For example, if you would like to model a pipe bend in a thermal liquid network, and if the temperature does not vary in this region of the model, you can use an Interface (TL-IL) block to connect to a Pipe Bend (IL) block, and then another Interface (TL-IL) block to connect back to the thermal liquid network.

You can also model heat exchange and mass transfer within and between domains using blocks from the Fluid Network Interfaces library.

Specify Fluid Properties

A *fluid properties* block is a block that defines the fluid properties of a network. The Simscape Fluids software uses lookup tables to set these fluid properties. If you do not attach a fluid properties block, the domain defaults apply. For the default properties of the different domains, see:

- “Isothermal Liquid Domain”
- “Hydraulic Domain”
- “Thermal Liquid Domain”
- “Gas Domain”
- “Two-Phase Fluid Domain”
- “Moist Air Domain”

To set the fluid properties in an isothermal liquid network using a fluid properties block, you can either:

- Specify your own properties by connecting an Isothermal Liquid Properties (IL) block from the Simscape Foundation library. This block lets you specify whether the liquid bulk modulus is constant or pressure-dependent. You can also specify whether the fluid contains entrained air, and, if present, whether its amount is constant or pressure-dependent. The default working fluid is water, with constant bulk modulus and zero entrained air.
- Connect an Isothermal Liquid Predefined Properties (IL) block from the Simscape Fluids Isothermal Liquid library and select one of the predefined liquids. You can also visualize fluid properties as a function of pressure with this block.

To set the fluid properties in a Hydraulic (Isothermal Liquid) network, you can either:

- Specify your own properties by connecting a Custom Hydraulic Fluid block from the Simscape Foundation library.
- Connect a Hydraulic Fluid block from the Simscape Fluids Hydraulic (Isothermal Liquid) library and select one of the predefined liquids.

To set the fluid properties in a Thermal Liquid network, you can either:

- Specify your own properties by connecting a Thermal Liquid Settings (TL) block at any point in the network. This block is in the Simscape Foundation library.
- Connect a Thermal Liquid Properties (TL) block from the Simscape Fluids Thermal Liquid library and select one of the predefined liquids.

To set the properties in a Two-Phase Fluid network, you can either:

- Specify your own properties by connecting a Two-Phase Fluid Properties (2P) block from the Simscape Foundation library. This block lets you define the properties of liquid and vapor separately, each as a tabulated function of pressure and temperature.
  - You can provide tabulated data for use in the Two-Phase Fluid Properties (2P) block with the `twoPhaseFluidTables` function. This function also allows you to view the properties of the fluid. See “Manually Generate Fluid Property Tables” for more information.
- Connect a Two-Phase Fluid Predefined Properties (2P) block from the Simscape Fluids Two-Phase Liquid library and select one of the predefined liquids. You can also visualize fluid properties as a function of specific internal energy or specific enthalpy and pressure with this block.

To set the properties in a moist air network, you can specify your own properties by connecting a Moist Air Properties (MA) block from the Simscape Foundation library. You can provide tabulated data for dry air and water vapor and specify the fluid temperature and pressure ranges.

To set the fluid properties in a gas network, you can specify your own properties by connecting a Gas Properties (G) block from the Simscape Foundation library. You can use this block to model a perfect, semi-perfect or real gas.

See Also

“Simscape Block Libraries” | “Manually Generate Fluid Property Tables”

More About

- “Modeling Moist Air Systems”
- “Modeling Thermal Liquid Systems”
- “Modeling Gas Systems”
- “Modeling Isothermal Liquid Systems”

Modeling and Simulation of Fluid Networks

In this section...
“Parameterizing a Block” on page 1-6
“Start and Simulate Your Model” on page 1-7
“Modifying Parameters” on page 1-7
“Balancing Accuracy, Speed, and Efficiency in Your Model” on page 1-8

Simscape Fluids blocks and connections represent one-dimensional flow paths. This means that internal component dynamics, such as the pressure variation along a pump blade, are not modeled in detail.

As with electric circuits, all network components have an associated Across and Through variable. Unlike an electric circuit, some Simscape Fluids domains have more than one Across and Through variable. For example, in the thermal liquid domain, mass flow rate and energy flow rate are block Through variables, and pressure and temperature are block Across variables. Blocks in the gas, moist air, and two-phase domains also have multiple Through and Across variables. For more information on block connections, ports, and variable types, see “Basic Principles of Modeling Physical Networks”.

In Fluids networks, flow is supported in either direction along connection lines. Many Simscape Fluids blocks also support reversed flows. Additionally, pressure and temperature are always absolute values. Regardless of flow direction, the variable values remain positive.

It is essential that your network is grounded. Simscape Fluids blocks with internal volumes, called *dynamic components*, can act as a network ground due to their internally-defined states. Furthermore, dynamic components within the network set the model initial conditions from the specified block parameters.

Due to these internal volumes, fluid variables are calculated at internal nodes as part of the momentum balance over the block. You may observe different values if you choose to measure a variable at an internal node or at a port. See for more information.

Parameterizing a Block

Some blocks can be parameterized by different methods. You can parameterize these blocks by:

- An analytical formulation
- Data from a data sheet
- Data from measurements

Data tables may be 2-D or 3-D and are composed of independent vectors, such as from a test matrix. For example, in the Variable-Displacement Pump (IL) block, the **Volumetric loss table, q_loss(dp,w,D)** parameter is populated with the volumetric losses at each test point for pressure loss, shaft speed, and displacement volume, over the range of each parameter.

You can plot and check the data in a lookup table with the PS Lookup Table (1D) and PS Lookup Table (2D) blocks. See “Plot Lookup Tables” for more information.

Start and Simulate Your Model

The Simscape Fluids software provides multiple ways to simulate and analyze multidomain fluid networks in the Simulink environment.

- To open the Simscape model template, enter `ssc_new` at the MATLAB command line. You can view the Simscape Fluids block library by entering `SimscapeFluids_lib` on the MATLAB command line. Drag blocks from the Simscape Library Browser to the Simulink Editor to construct your model.
- To view and inspect your network variables before, during, and after simulation, use the “Variable Viewer”. To learn more about initializing models and setting variable priorities, see “Block-Level Variable Initialization”. Initial conditions for Simscape Fluids models are domain-dependent:
  - In the isothermal liquid domain, the initial pressure must be provided.
  - In the thermal liquid domain, the initial pressure and temperature must be provided.
  - In the gas domain, the initial pressure, and either the density or temperature must be provided.
  - In the moist air domain, the initial pressure, temperature, humidity quality, and trace gas amount must be provided.
  - In the two-phase domain, the initial pressure and either the phase quality, temperature, enthalpy, or internal energy must be provided.
- You can view the results of a simulation by attaching a **Scope** block to a sensor, or by enabling data logging and viewing the results in the Results Explorer. Exporting data from the Results Explorer to the Data Inspector allows you to compare data between simulations. For more information, see “About the Simscape Results Explorer” and Simulation Data Inspector.
  - Scope blocks plot Simulink signals. Convert a Simscape physical signal with a **PS-Simulink Converter** block to connect a physical signal to a Scope block. See “Connecting Simscape Diagrams to Simulink Sources and Scopes” for more information.
  - To turn on data logging, from the Simulink toolstrip, select **Modeling > Model Settings > Model Settings**. In the left pane of the Configuration Parameters dialog box, select Simscape. In the Simscape pane, under **Data Logging**, set **Log simulation data** to `All` and select **Open viewer after simulation**.
- To learn more about selecting and configuring the right solver for your system, see “Setting Up Solvers for Physical Models”. The variable-step solvers `ode23t` and `ode15s` are versatile solvers that are suitable for a range of dynamics. To use Simulink Coder to generate code from your model, use the `ode14x` or `ode1be` solvers. For more information about code generation, see “Code Generation”.

See “Essential Steps for Constructing a Physical Model” for a detailed overview of a Simscape modeling workflow.

Modifying Parameters

You can modify parameter values during simulation if they are run-time configurable. To learn more about configurable parameters and settings, see “About Simscape Run-Time Parameters”.

Balancing Accuracy, Speed, and Efficiency in Your Model

To change the accuracy, speed, and efficiency of your simulation, you can adjust one or more of the following parameters in the Configuration Parameters dialog box:

- **Relative tolerance**
- **Absolute tolerance**
- **Max step size**

You can also adjust the **Constraint Residual Tolerance** in the Solver Configuration block dialog box.

In most cases, the default tolerance values will produce a good balance between the accuracy and speed of your model. However, in some cases you may wish to adjust the **Max step size** from its default setting of `auto`, which is calculated based on the simulation duration and not on the simulation dynamics. If you are concerned that the solver may be missing significant behavior, you can change this value to a smaller value.

For more information about the solver parameters, see:

- “Relative tolerance”
- “Absolute tolerance”
- “Max step size”

See Solver Configuration to learn more about when to adjust the **Constraint Residual Tolerance** parameter value.

See Also

More About

- “Basic Principles of Modeling Physical Networks”
- “Essential Physical Modeling Techniques”
- “Modeling Best Practices”

Networks and Network Connections

Simscape Fluids models can include multiple fluid networks, each representing a single-fluid system. Only use one fluid properties block per network to avoid model compilation errors. Two fluid networks are distinct if they belong to different Simscape domains or if they are not directly connected.

You can use the blocks in the Fluid Network Interfaces library to connect different domains. For example, you can use the Condenser-Evaporator 2P-MA block to model heat exchange between a two-phase and moist air network.

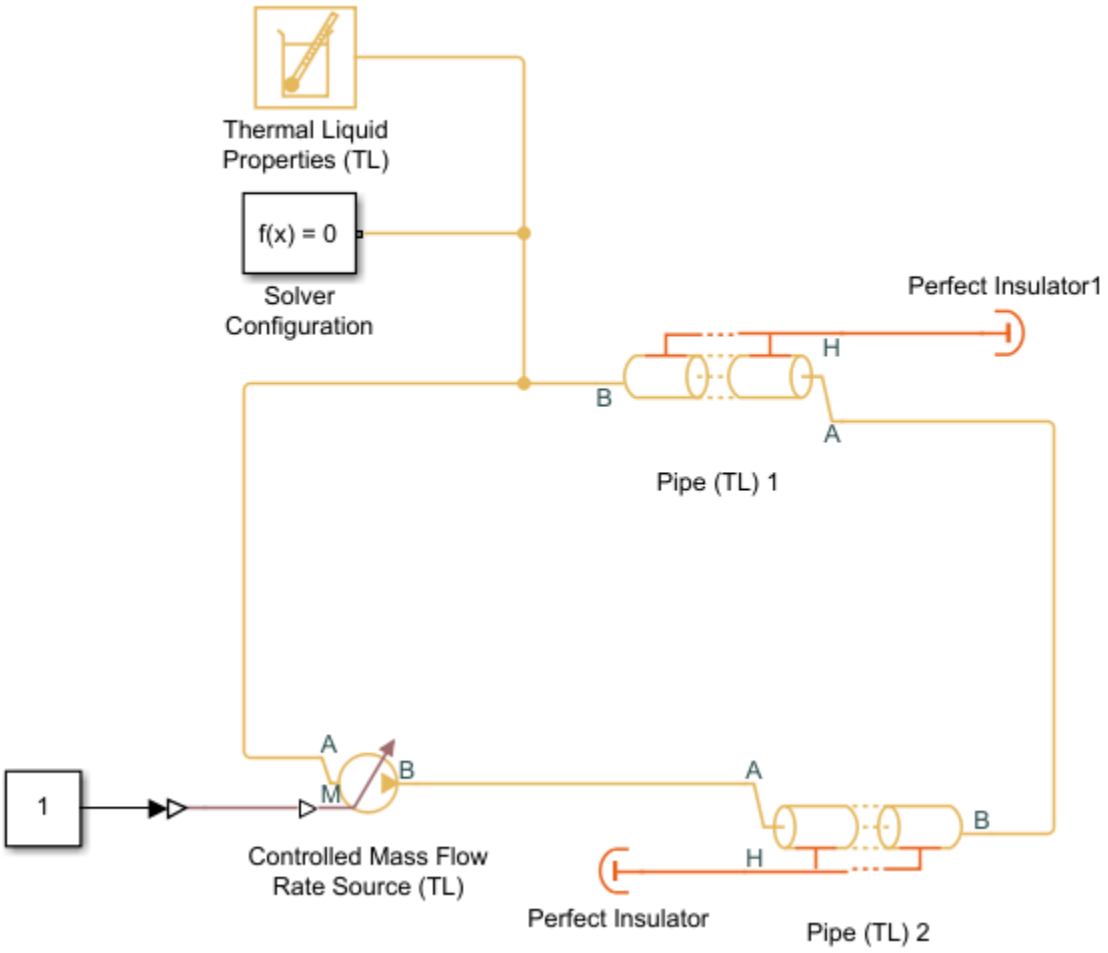
Grounding Rules

The absence of a reference block, such as a Reservoir (TL) block, or dynamic component in a non-isothermal liquid domain may result in variable drift in your model. The internal node of a reference or dynamic component provides a state reference during momentum balance calculations. See for more information on internal volumes in dynamic components.

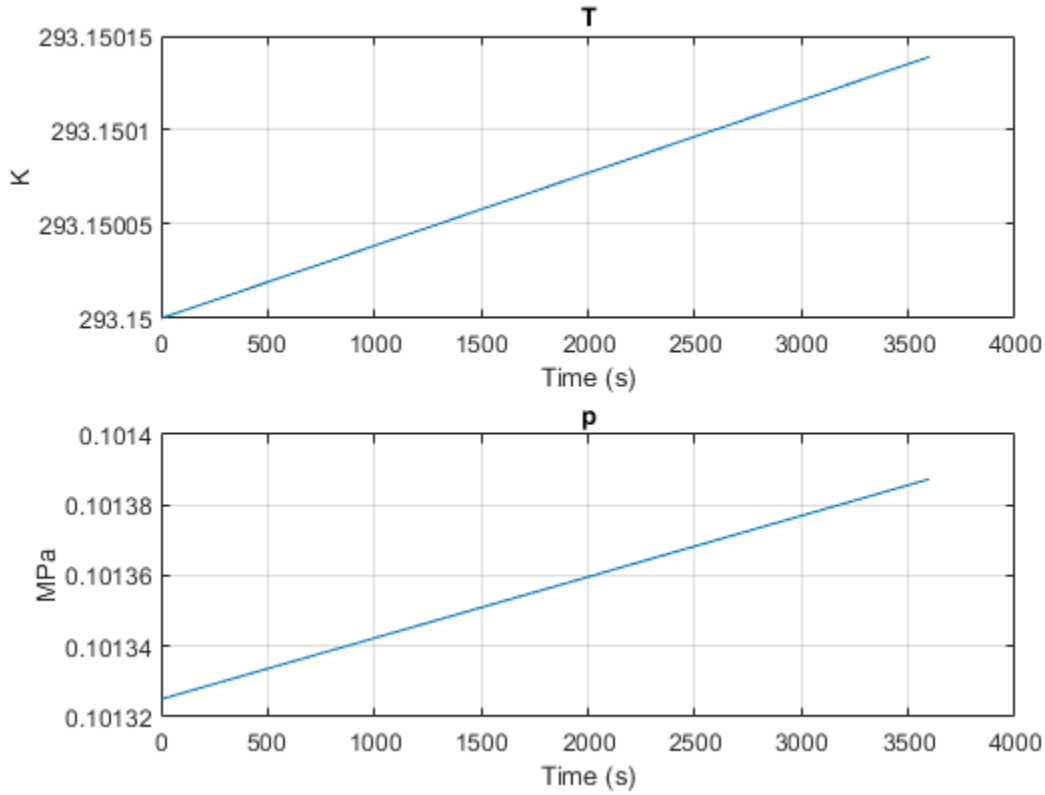
If you observe variable drift during your simulation, add grounding to your model.

Ground a Thermal Liquid Closed Loop

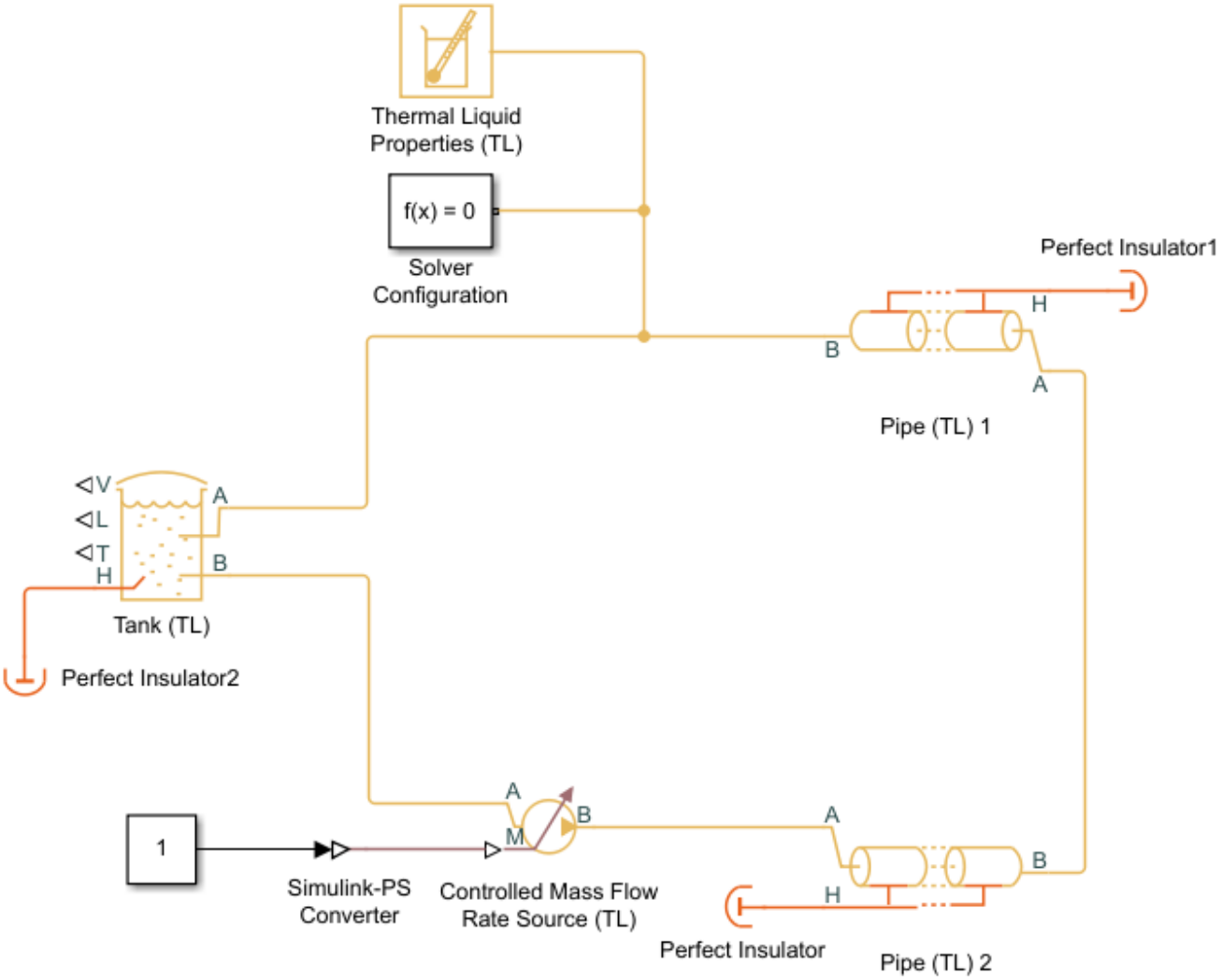
Simulating the example network below, consisting of two pipes connected to a Controlled Mass Flow Rate Source (TL) block, will result in a net increase in pipe temperature and pressure.



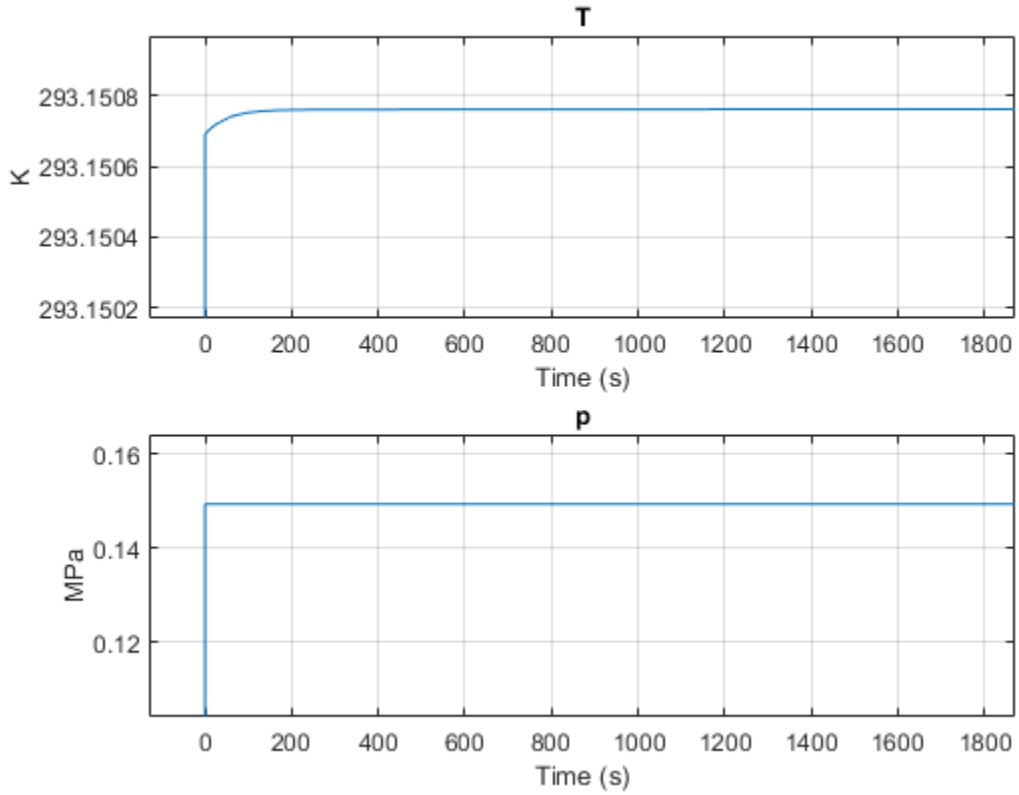
Despite the lack of explicit heat transfer, the absence of thermal grounding in the network results in a net increase of energy due to energy conservation. The increase of energy in the system takes the form of a rise in temperature.



You can mitigate this phenomenon by adding a dynamic component to your network. Add a Tank (TL) block just before the controlled Mass Flow Rate Source (TL). Right-click on the block and select **Simscape > Block choices > Two inlets**.



In this scenario, the pipe pressure and temperature reach steady-state. The differential equations associated with the Tank (TL) internal volume reduce the stiffness of the model solution, removing the condition of gradually increasing system temperature and pressure.



See Also

More About

- “Modeling Best Practices”

Considerations For Building Fluids Models

For general Simscape troubleshooting techniques, see “Troubleshooting Simulation Errors”.

If your model returns an error message that identifies specific model blocks, inspect these blocks for wrong connections or unintended use. For individual blocks, check:

- Block Use — Check the block documentation for model adherence to any assumptions and limitations listed. For example, an Ideal Pressure Source block can simulate a pump only when the pressure is constant.
- Absolute Variables — Pressure and temperature are absolute values and should always be positive. There are no restrictions on the sign of the change in pressure or the change in temperature of a component. Note that some blocks in the Hydraulic (Isothermal Liquid) library return pressure as gauge values.
- Block Parameters — For blocks with a **Leakage Area** parameter, ensure the parameter has a nonzero value. Isolation of certain parts of the system can occur if a small amount of fluid is not present in all components of the network throughout the simulation.
- Dynamic Compressibility — Fluid dynamic compressibility may be relevant for components with internal volumes if the time constant associated with the component is much smaller than the time constant associated with system transient events. In the Isothermal Liquid library, the dynamic compressibility default is `On` for blocks that support it.
- Scope Measurement — Note that sensors and scopes take measurements at nodes. Nodes do not have internal volumes, and will record variable changes instantaneously. This may be discontinuous with the expected response of a component with an internal volume. Consider inspecting results in the Results Explorer.
- Scope Units — Be aware of Scope units when converting between Simulink and Simscape signals. Scopes record the mass flow rate changes in variables without user adjustment, but you may need to apply an affine conversion to variables with absolute values, such as temperature. The default Simscape unit of temperature is Kelvin. For more information on applying affine conversions, see “How to Apply Affine Conversion”.
- Block Connections — Verify that the model makes sense as a system. For example, look for:

  - Accumulators connected to a pump outlet without check valves.
  - Actuators connected against each other, causing motion in opposite directions.
  - Directional valves with a large bypass.

  Note that connections between blocks do not represent physical volumes and values are passed along these connections instantaneously.

In the network, check:

- Variables — Use the Variable Viewer to inspect variable values, units, and initialization status.
- Solver Choice — Verify that you are using the best solver for your model. For thermal systems, conduction in pipes can create a stiff system of differential equations. Adjust the **Max step size** parameter in the Configuration Parameters dialog box so that each solver iteration can capture the dynamics of your system.
- Network Domain — See “Select the Working Fluid” on page 1-3 to verify that your network fluid is appropriate for your application.
- Model Fidelity — Maintain the same level of refinement over multiple networks. Avoid mixing ideal and real networks in the same system.

- Dry Nodes — If too many quasi-steady components are clustered in your network (such as valves, where calculations only occur at block ports), your model may be at higher risk for dry nodes during simulation. Dry nodes may slow or stop your simulation. You can improve simulation convergence in these scenarios by adding blocks with internal volumes, which add dynamic components that can respond to dynamic shifts in the network. You can check for dry nodes with the Model Advisor by running **Check for dry hydraulic nodes** in **By Product > Simscape**. This check will apply only to blocks in the Hydraulic (Isothermal Liquid) domain.
- Heat Transfer — In thermal fluid systems, heat can drive flow. Block port sizes can also significantly impact upwind conduction in the network. Check your system for unexpected heat transfer.
- Loop Continuity — In the absence of a network ground (see “Grounding Rules”), closed-loop systems in thermal or two-phase domains may experience baseline drift. This is caused by an imbalance in the mass and energy conservation equations over the network. For example, if only some blocks in your network account for the effect of gravity, your model may experience a gradual temperature rise. If the effect of gravity in this scenario is not relevant to network dynamics, you can set the gravitational acceleration parameter to 0.

It is good practice to build and test models incrementally. Start with simulating an idealized, simplified system and verify that it works as expected. Once verified, change default settings and add complexity incrementally. Use subsystems to capture the model hierarchy and test these subsystems separately before testing the entire model.

See Also

“Troubleshooting Simulation Errors”

More About

- “Modeling Best Practices”
- “Plot Lookup Tables”

Modeling Fluid Dynamic Systems

- “Block Equations and the Simscape Numerical Scheme” on page 2-2
- “Model a Thermal Liquid Pipeline” on page 2-6
- “Model an Isothermal Actuation System” on page 2-16
- “Modeling Power Units” on page 2-28
- “Modeling Valves in Simscape Fluids” on page 2-31
- “Building a Custom Valve” on page 2-37
- “Modeling Hydraulic Directional Valves” on page 2-42
- “Modeling Low-Pressure Fluid Transportation Systems” on page 2-52
- “Numerical Smoothing” on page 2-56

Block Equations and the Simscape Numerical Scheme

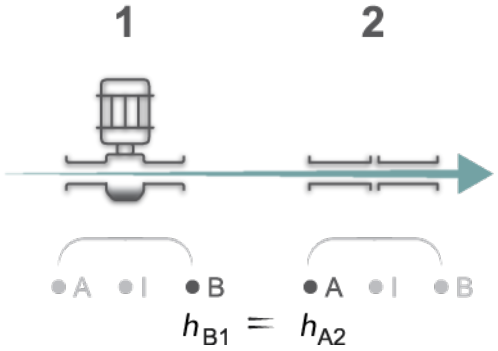
System Equations

Simscape Fluids blocks are a combination of differential equations, which represent the one-dimensional component dynamics, and algebraic equations, which represent the mass and energy continuity over the component. The system of equations, which comprises all blocks over all Simscape domains in the model, are assembled into an exact square matrix that is solved iteratively at each time step until convergence. The number of equations being solved and any zero-crossing events influence the speed and quality of model convergence. For more information on how Simscape software assembles and solves the governing equations of a model, see “How Simscape Simulation Works”.

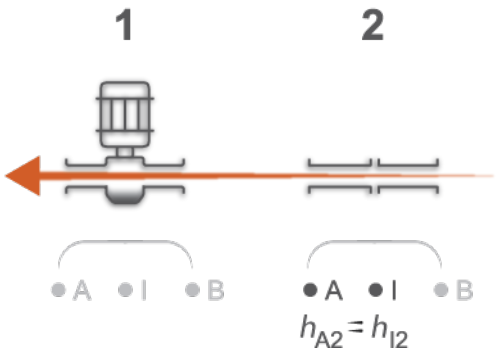
Solving the Network: The Simscape Numerical Scheme

Calculations are computed at each node of your fluid network, such as at block ports, connector intersections, and at internal points in dynamic components. To solve a network, the fluid properties at each node propagate according to the *upwind* numerical scheme. This means that the fluid properties at a given node are calculated based on the fluid properties at that node at the previous solver iteration and the current value of the fluid properties that precede it in space. Flow into an element is considered positive.

The following diagram shows two representations of blocks. Each block has a port A and B and an internal node I. The specific total enthalpy h_{A2} at port A in block 2 is calculated according to the flow direction and is based on the specific total enthalpy, h_{B1}, at port B of block 1:



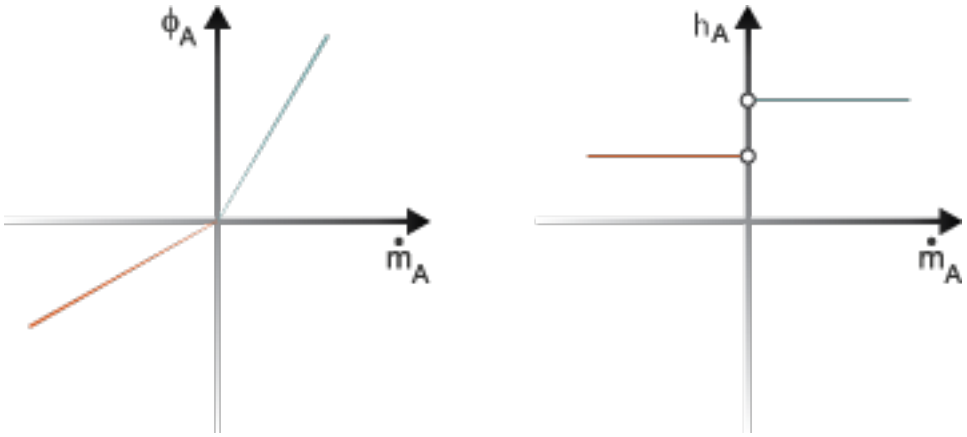
In the same configuration, flow in the reverse direction would mean that the specific total enthalpy at port A of block 2 is calculated based on the specific total enthalpy of the internal node, I, of the same component:



This means that you may observe differences in measured variable values depending on the flow direction and interrogated node. See for an example of variable measurement in a network with flow reversal.

Smoothing in the Simscape Numerical Scheme

In a flow reversal region, as defined by the total energy flow rate and block thresholds, an energy flow rate without smoothing results in a jump discontinuity of the specific total enthalpy:



Upwind Scheme Without Smoothing

This can result in differences in measured variables between adjacent nodes and may influence the solver performance.

To avoid these discontinuities, smoothing is applied to the calculations and fluid values are calculated in terms of energy flows. For example, in the thermal liquid domain, the smoothed energy flux at A is the sum of the energy flow rate and the specific total enthalpy conduction between port A and internal node I. Conduction contributes to numerical smoothing while contributing a negligible quantity to the energy flow rate under nominal block conditions.

Quasi-steady components also account for smoothing to the energy flow rate. Although the block itself does not have an internal volume, the difference between internal enthalpy and enthalpy at the block ports is still calculated. The total energy flow rate at each node is determined by the fluid flow direction:

$$\Phi_A = G(h_A - h_I) + \begin{cases} \dot{m}_A h_A & \dot{m}_A > 0 \\ \dot{m}_A h_I & \dot{m}_A \le 0 \end{cases},$$

where:

- Φ_A is the energy flow rate through port **A** and conduction between port **A** and the internal node **I**.
- G is the thermal conductance coefficient, which for thermal liquid and moist air domains is calculated as:

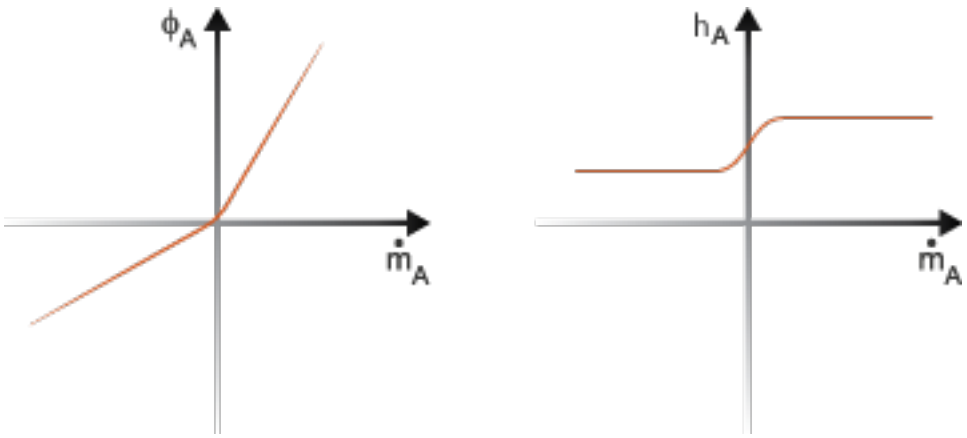
  $$G = \frac{kS}{c_v L},$$

  where:

  - k is the thermal conductivity.
  - S is the port cross-sectional area.
  - c_v is the fluid specific heat.
  - L is the characteristic length between the port and internal node.

For two-phase and gas domains, G is a general tuning parameter and is set by the threshold parameters defined for the block. For example, increasing the value of the **Mach number for flow reversal** parameter in a Two-Phase library block increases the amount of smoothing applied to the energy flow. Similarly, increasing the value of the **Dynamic pressure threshold for flow reversal** parameter in a Gas library block increases the region of smoothing applied.

In a flow reversal region, as defined by the total energy flow rate and block thresholds, the specific total enthalpy changes according to a gradual hyperbolic tangential:



Upwind Energy Scheme with Smoothing

See Also

"How Simscape Simulation Works"

More About

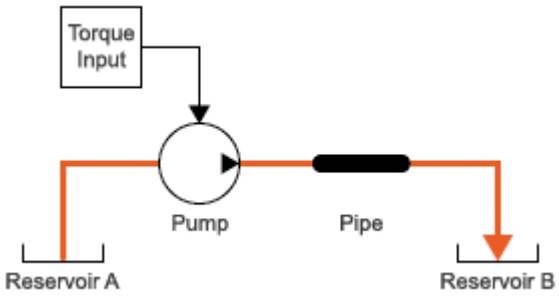
- "Important Concepts and Choices in Physical Simulation"

- “Making Optimal Solver Choices for Physical Simulation”

Model a Thermal Liquid Pipeline

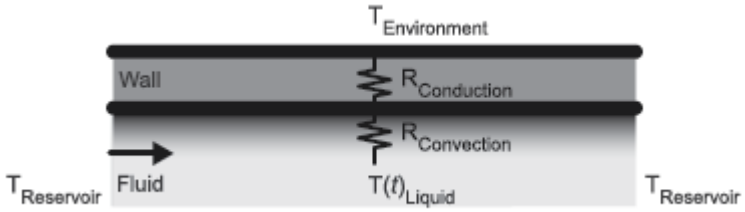
In this section...
“Start a New Model” on page 2-7
“Model the Pipeline” on page 2-7
“Introduce Thermal Conduction” on page 2-8
“Model the Pump Torque” on page 2-11
“Add Thermal Liquid Sensors” on page 2-12
“Simulate the Pipeline Model” on page 2-15

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 generates power and the pipe transfers fluid between the reservoirs. The reservoirs act as pressure and temperature boundary conditions for the pipeline system. Note that "thermal" refers to the properties of the solid system, while "thermal liquid" refers to the properties of the fluid system.



Thermal Liquid Pipeline Schematic

The reservoir conditions are fixed at room temperature, 293.15 K, and atmospheric pressure, 0.101325 MPa. 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 heating due to friction and conductive cooling, gradually shifts toward a new steady-state value. The thermal resistance of the pipe is the combination of these convective and conductive resistances.



Pipe Thermal Resistances

A Pipe (TL) block models the convective thermal resistance at the inner pipe surface and the heating due to friction in the thermal liquid. A Conductive Heat Transfer block models the conductive thermal resistance between the inner and outer pipe surfaces.

Start a New Model

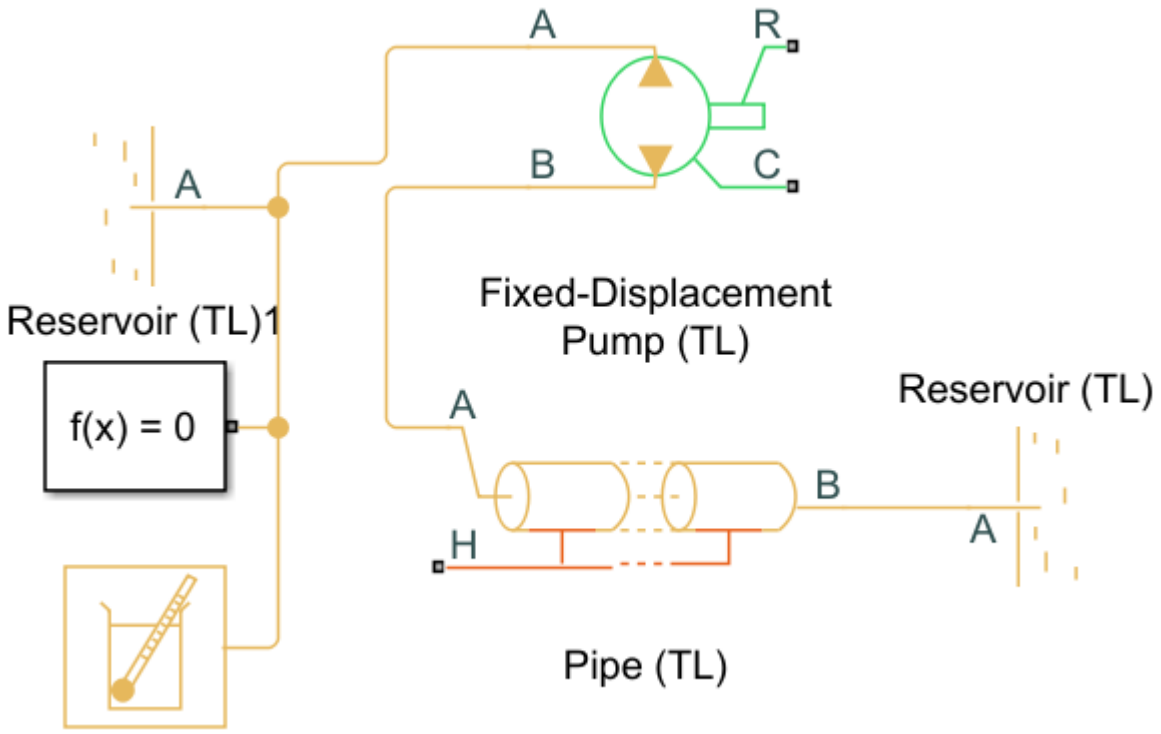
1. At the MATLAB command prompt, enter `ssc_new`. MATLAB opens the Simscape model template. This template provides a starting point for your Simscape Fluids model, and includes a Solver Configuration block, a Simulink-PS converter block, a PS-Simulink converter block, and a Scope block, which enables you to view the results of your simulation. You will not need the Simulink-PS Converter block in this example.
2. Save the model often as you add new blocks.
3. In the **Simulation** tab, click **Library Browser**. In the pop-up window, select **Simscape > Foundation Library > Thermal Liquid > Utilities** library and 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.
4. 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 physical network requires a Solver Configuration block.

Model the Pipeline

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

Block	Library	Purpose
Pipe (TL)	**Pipes & Fittings**	Conduit for thermal liquid between reservoirs
Fixed-Displacement Pump (TL)	**Pumps & Motors**	Power generation source

3. Connect the blocks and verify your connections in the figure below. Note that blocks may be rotated.
   a. Port **A** of the Pipe (TL) block connects to port **B** of the Fixed-Displacement Pump (TL) block.
   b. Port **B** of the Pipe (TL) block connects to one Reservoir (TL) block. This serves as a thermal liquid sink.
   c. Port **A** of the Fixed-Displacement Pump (TL) block connects to the second Reservoir (TL) block, the Solver Configuration block, and the Thermal Liquid Settings (TL) block. The Reservoir block serves as a thermal liquid source.



The component colors indicate the physical domains they represent: yellow for Thermal Liquid, red for Thermal, and green for Mechanical Rotational. The pump and pipe blocks are multi-domain 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 results in observable pressure losses due to dissipation and heat transfer by friction at the pipe walls.

Introduce Thermal Conduction

1 Open the **Library Browser** and insert these blocks from the **Simscape > Foundation Library**.

Block	Library	Purpose
Conductive Heat Transfer	**Thermal > Thermal Elements**	Models the conductive heat losses through the pipe wall to the surrounding environment
Thermal Reference	**Thermal > Thermal Elements**	Provides a temperature reference for the physical signal input
Controlled Temperature Source	**Thermal > Thermal Sources**	Acts as an ideal energy source and maintains temperature regardless of heat flow rate
PS Constant	**Physical Signals > Sources**	Specifies environmental temperature

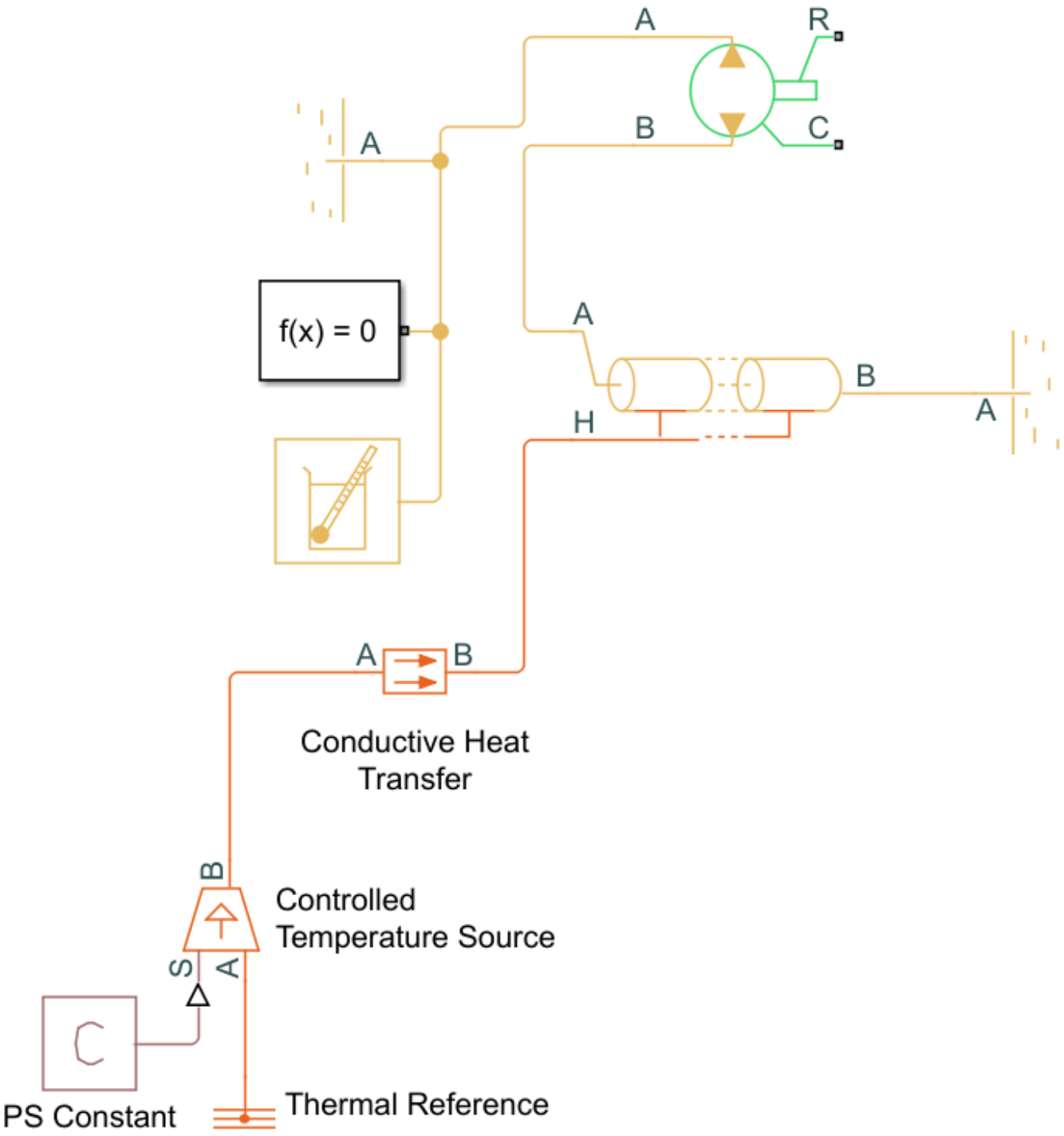
2. Connect the blocks and verify your connections in the figure below. Note that blocks may be rotated.

   a. Connect port **B** of the Conductive Heat Transfer block to port **H** of the Pipe (TL) block.

   b. Connect port **B** of the Controlled Temperature Source block to port **A** of the Conductive Heat Transfer block.

   c. Connect the PS Constant block to port **S** of the Controlled Temperature Source block.

   d. Connect the Thermal Reference block to port **A** of the Controlled Temperature Source block.



3. In the PS Constant Block Parameters Window, set the **Constant** parameter to `275 K`. This is the environment temperature.
4. In the Conductive Heat Transfer block, set the **Area** parameter to `pi*0.1128*1000` m^2. This is the pipe surface area in terms of the hydraulic diameter, `0.1128` m, and length, `1000` m. The default **Thermal conductivity** value is for copper, `401 W/(mK)` .

Model the Pump Torque

1 Open the **Library Browser** and insert these blocks from **SimscapeFoundation Library**.

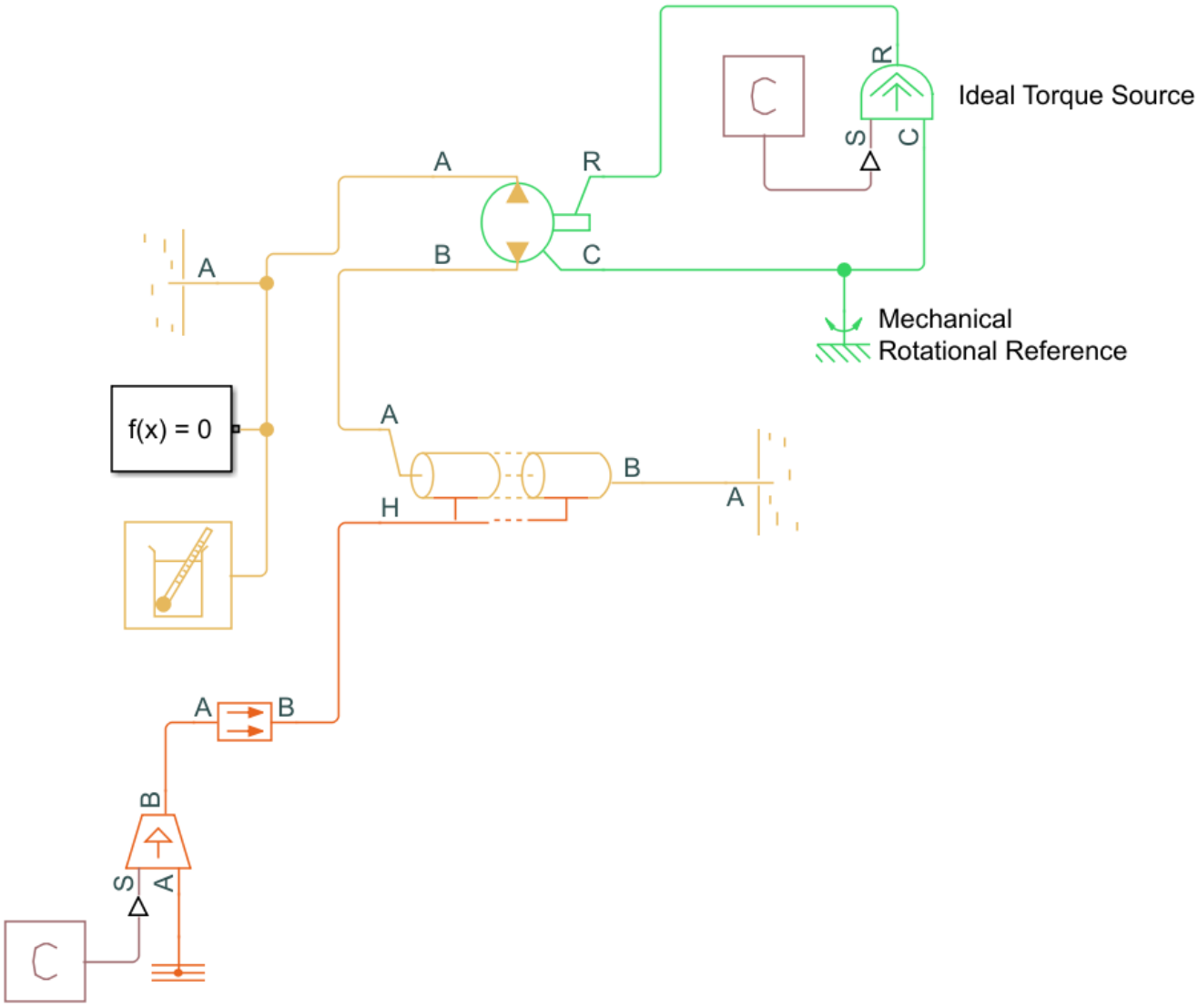
Block	Library	Purpose
Ideal Torque Source	**Mechanical** > **Mechanical Sources**	Sets the pump torque rate as a physical signal
Mechanical Rotational Reference	**Mechanical** > **Rotational Elements**	Acts as a reference for the Ideal Torque Source and maintains a physical signal at a specified value
PS Constant	Double-click on the browser white space, type PS Constant and hit **Enter**	Specifies the pump torque

2 Connect the blocks and verify your connections in the figure. Note that blocks may be rotated.

   a Connect port **R** of the Fixed-displacement Pump (TL) block, which represents the rotating pump shaft, to port **R** of the Ideal Torque Source block.

   b Connect port **C** of the Ideal Torque Source block, which represents pump casing, to port **C** of the Fixed-Displacement Pump (TL) block and the Mechanical Rotational Reference block.

   c Connect port **S** of the Ideal Torque Source block to the PS Constant block.



3 In the PS Constant Block Parameters window, set the **Constant** parameter to `50 N*m`.

Add Thermal Liquid Sensors

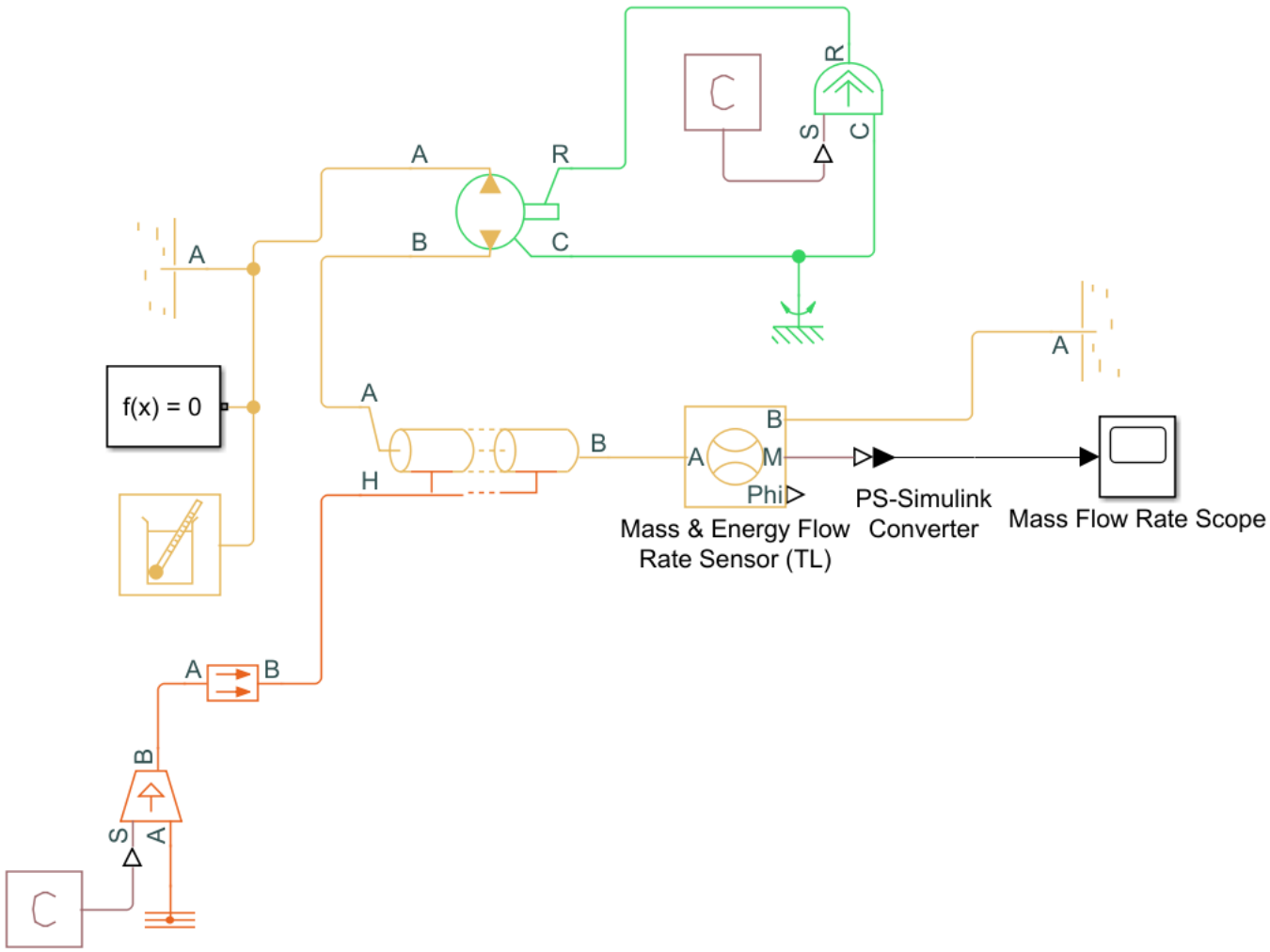
1 Open the **Library Browser** and add these blocks to the model.

Block	Library	Purpose
Mass & Energy Flow Rate Sensor (TL)	**Simscape** > **Foundation Library** > **Thermal Liquid** > **Sensors**	Measures the Through variables of the Thermal Liquid domain and returns a physical signal
PS-Simulink Converter	**Simscape** > **Utilities**	Transforms a physical signal into a Simulink signal

Block	Library	Purpose
Scope	**Simulink > Sinks**	Plots Simulink signals

2 Connect the blocks and verify your connections in the figure. Note that blocks may be rotated.

- **a** Your Pipe (TL) block is connected to a Reservoir from Section 1.5.4, which provides pressure and temperature boundary conditions for the system. You will now add a way to measure pipe flow. Disconnect port **B** of the Pipe (TL) block from the Reservoir block and connect it to port **A** of the Mass & Energy Flow Rate Sensor (TL) block.
- **b** Connect port **B** of the Mass & Energy Flow Rate Sensor block to the Reservoir block.
- **c** Connect port **M** of the Mass & Energy flow Rate Sensor (TL) block to the PS-Simulink Converter block.
- **d** Connect the PS-Simulink Converter block to the Scope block. Click the Scope block and name it "Mass Flow Rate Scope."

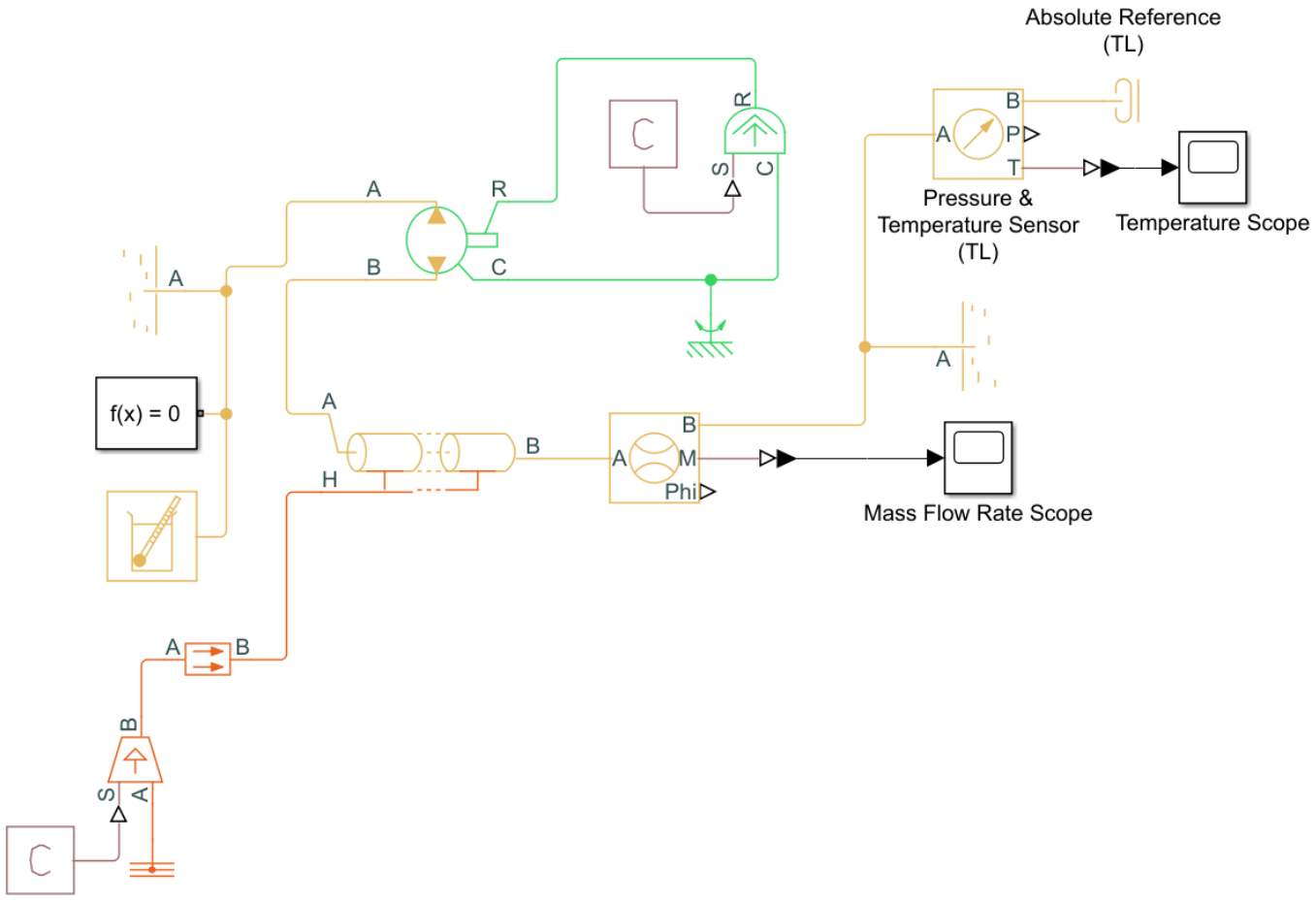


3 Open the **Library Browser** and add these blocks to the model. You will also use the PS-Simulink Converter and Scope blocks that were included when you opened your new model.

Block	Library	Purpose
Pressure & Temperature Sensor (TL)	**Simscape > Foundation Library > Thermal Liquid > Sensors**	Measures the Across variables of the Thermal Liquid domain and returns a physical signal
Absolute Reference (TL)	**Simscape > Foundation Library > Thermal Liquid > Elements**	Provides a control reference where pressure and temperature are equal to zero

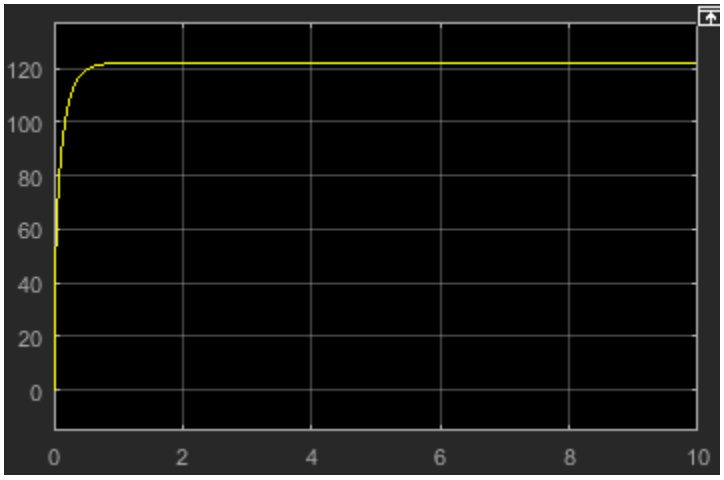
4 Connect the blocks and verify your connections in the figure. Note that blocks may be rotated.

- **a** Create a branch from port **B** of the Mass & Energy Flow Rate Sensor (TL) block and connect it to port **A** of the Pressure & Temperature Sensor (TL) block.
- **b** Connect port **B** of the Pressure & Temperature Sensor(TL) block to the Absolute Reference block.
- **c** Connect port **T** of the Pressure & Temperature Sensor (TL) block to the PS-Simulink Converter block.
- **d** If the PS-Simulink Converter is not already connected to the remaining Scope block, connect the two. Click this Scope block and name it "Temperature Scope."

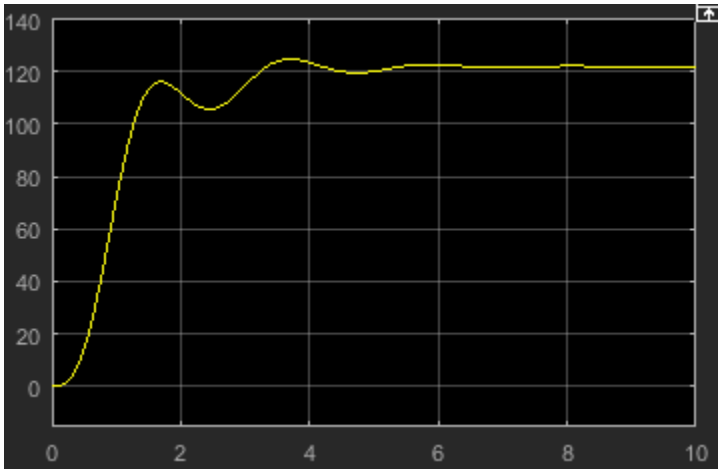


Simulate the Pipeline Model

The Simscape model template specifies suitable solver settings for this model. You can run the simulation by opening the **Simulation** tab and clicking **Run > Run**. Once the model has compiled, open the Mass Flow Rate Scope. Your plot should show a quick change in flow rate as the thermal liquid transitions from a state of rest to a new steady-state velocity.



You can experiment with other block settings and input signals. Try setting the Pipe (TL) block **Fluid inertia** parameter to `On`, then run your model. The figure shows the combined effects of fluid dynamic compressibility and fluid inertia on the mass flow rate plot.



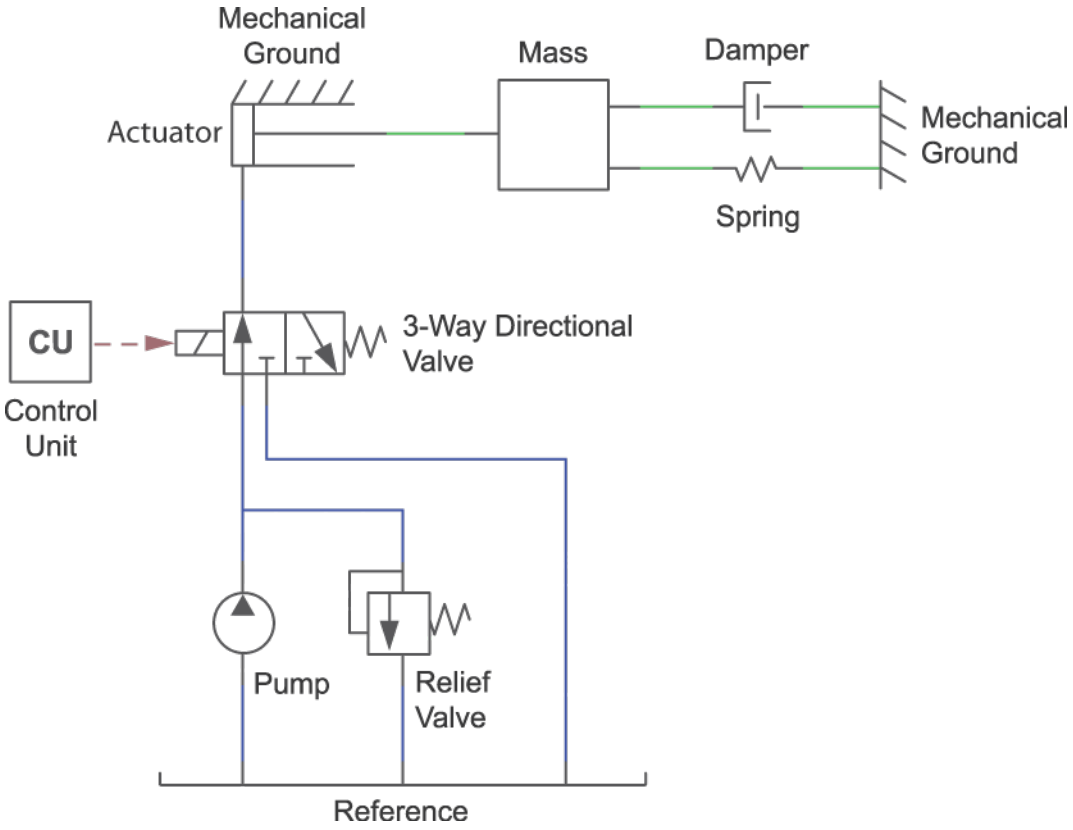
See Also

Model an Isothermal Actuation System

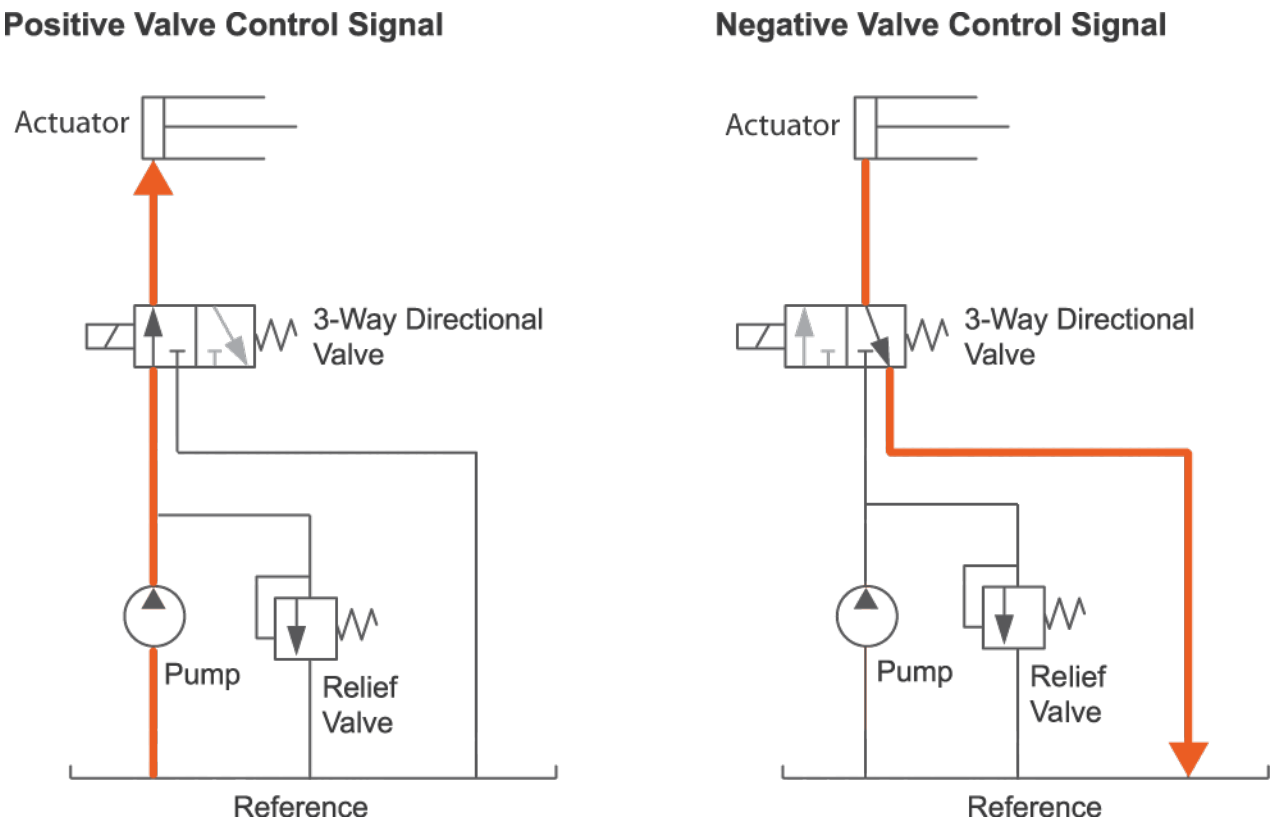
In this section...
“Model Schematic” on page 2-16 “Start a New Model” on page 2-17 “Model the Actuator Subsystem” on page 2-18 “Add a Load Model” on page 2-20 “Add a Control System and Monitoring” on page 2-21 “Run the Simulation” on page 2-23

Model Schematic

This tutorial shows how to model an isothermal actuation system using Simscape Fluids blocks. The system consists of a pump, which is modeled by an idealized pressure source, a control unit, a flow regulation valve, and a mechanical load.



The control unit represents an incoming signal for actuation. The three-way valve represents flow regulation into the actuator. Depending on the position received from the control unit, the three-way valve allows fluid to enter the actuator chamber or redirects it to the environment ("Reference"). In the 3-Way Directional Valve (IL) block, there is a single spool that adjusts the two valve positions - a flow path between ports **P** and **A** and a flow path between ports **A** and **T**. A change in spool position results in a change in flow path through the valve.

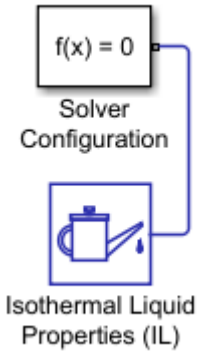


Directional Valve Positions

A relief valve vents to the environment if the pressure between the pump and valve becomes too high.

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. You will use blocks from both the Simscape Foundation and Simscape Fluids product libraries.
2. From the Simulation tab, click on the **Library Browser**. From the **Simscape > Foundation Library > Isothermal Liquid > Utilities** library, drag an Isothermal Liquid Properties (IL) block to the model canvas. Use this block to define the physical properties of the network fluid, including its density, viscosity, and bulk modulus. Use the Isothermal Liquid Predefined Properties (IL) block from the **Fluids > Isothermal Liquid > Utilities** library to select a fluid with these properties already defined.
3. Connect the Solver Configuration block to the Isothermal Liquid Properties (IL) block as shown in the figure. The Solver Configuration block provides Simscape solver settings for your model. See “Setting Up Solvers for Physical Models” for more information on solver selection.



Each topologically distinct physical network requires a solver block.

Model the Actuator Subsystem

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

Block	Sublibrary
Pressure Source (IL)	**Isothermal Liquid > Sources**
Reservoir (IL)	**Isothermal Liquid > Elements**

Alternatively, double-click on the model canvas and type the block name. Click on the block name in the drop-down list that appears to add the block to the canvas.

Specify the model configuration in each block dialog by double-clicking on the block icon:

2 A pressure source represents a constant-pressure pump. In the Pressure Source (IL) block dialog, set **Source type** to `Constant`. Input `10e5` Pa for **Pressure differential**.

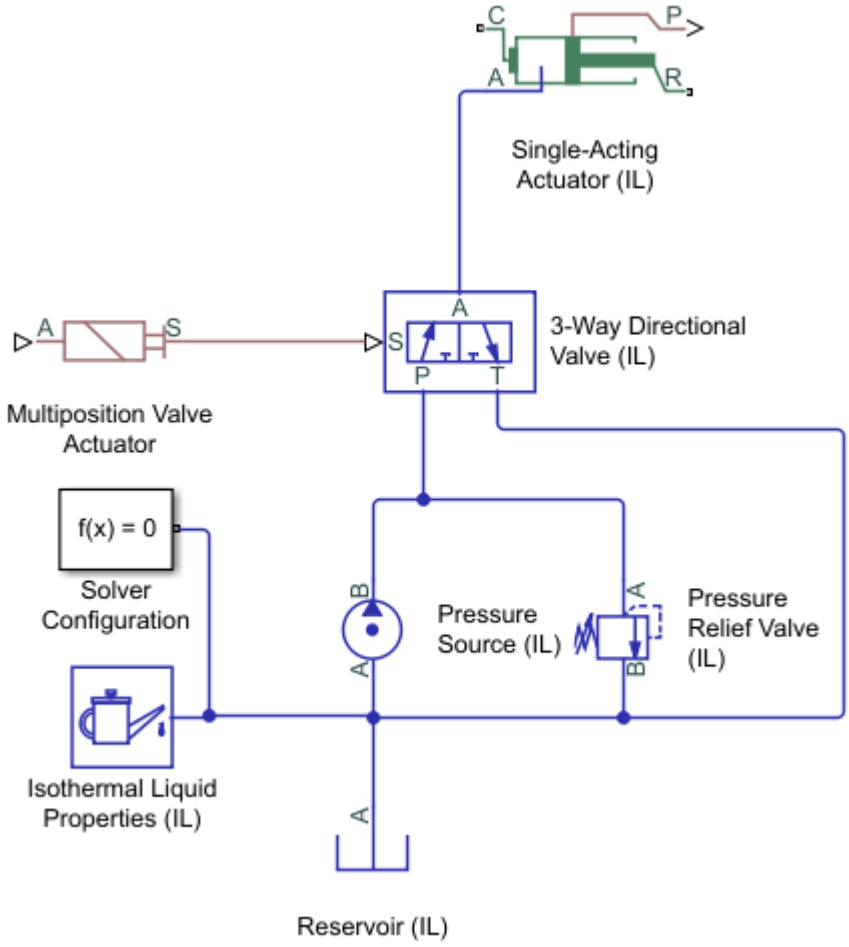
3 A reservoir sets the environmental boundary conditions. In the Reservoir (IL) block, set **Reservoir pressure specification** to `Specified pressure` and **Reservoir pressure** to `0.2` MPa.

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

Block	Sublibrary
Single-Acting Actuator (IL)	**Isothermal Liquid > Actuators**
3-Way Directional Valve (IL)	**Isothermal Liquid > Valves & Orifices > Directional Control Valves**
Multiposition Valve Actuator	**Isothermal Liquid > Valve Actuators**
Pressure Relief Valve (IL)	**Isothermal Liquid > Valves & Orifices > Pressure Control Valves**

The actuator converts the power generated by the pressure source into a translating force. The directional valve and valve actuator regulate the flow to the actuator.

5 Connect the blocks as shown in the figure. Note that the 3-Way Directional Valve (IL), Pressure Source (IL), and Pressure Relief Valve (IL) blocks are grounded by their connection to the Reservoir (IL) block.



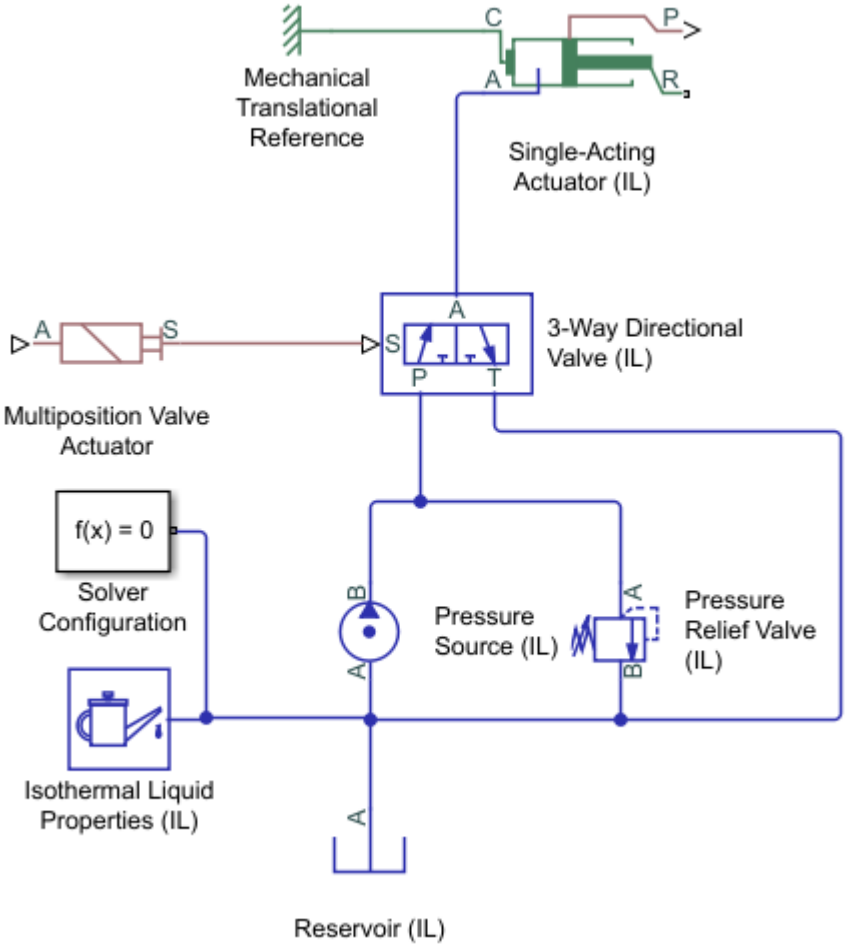
6 Set the operating pressures of the Pressure Relief Valve (IL). In the **Set pressure differential** parameter, enter `50e5` Pa. This is the pressure at which the valve begins to open. In the **Pressure regulation range** parameter, which describes the operational range of the valve, enter `5e5` Pa. The relief valve maximum pressure is the sum of the set pressure and the regulation range.

7 In the 3-Way Directional Valve (IL) block dialog, on the **P-A Orifice** tab, set **Spool position at maximum P-A orifice area** to `0.01` m. This is the position of the valve spool when the P-A flow path is fully open. On the **A-T Orifice** tab, set **Spool position at maximum A-T orifice area** to `0` m.

   The **Spool travel between closed and open orifice** parameter on the **Model parameterization** tab sets the distance over which both orifices open and close. At this value, both orifices are closed.

8 In the Single-Acting Actuator (IL) block, set the **Piston cross-sectional area** to `0.001` m^2.

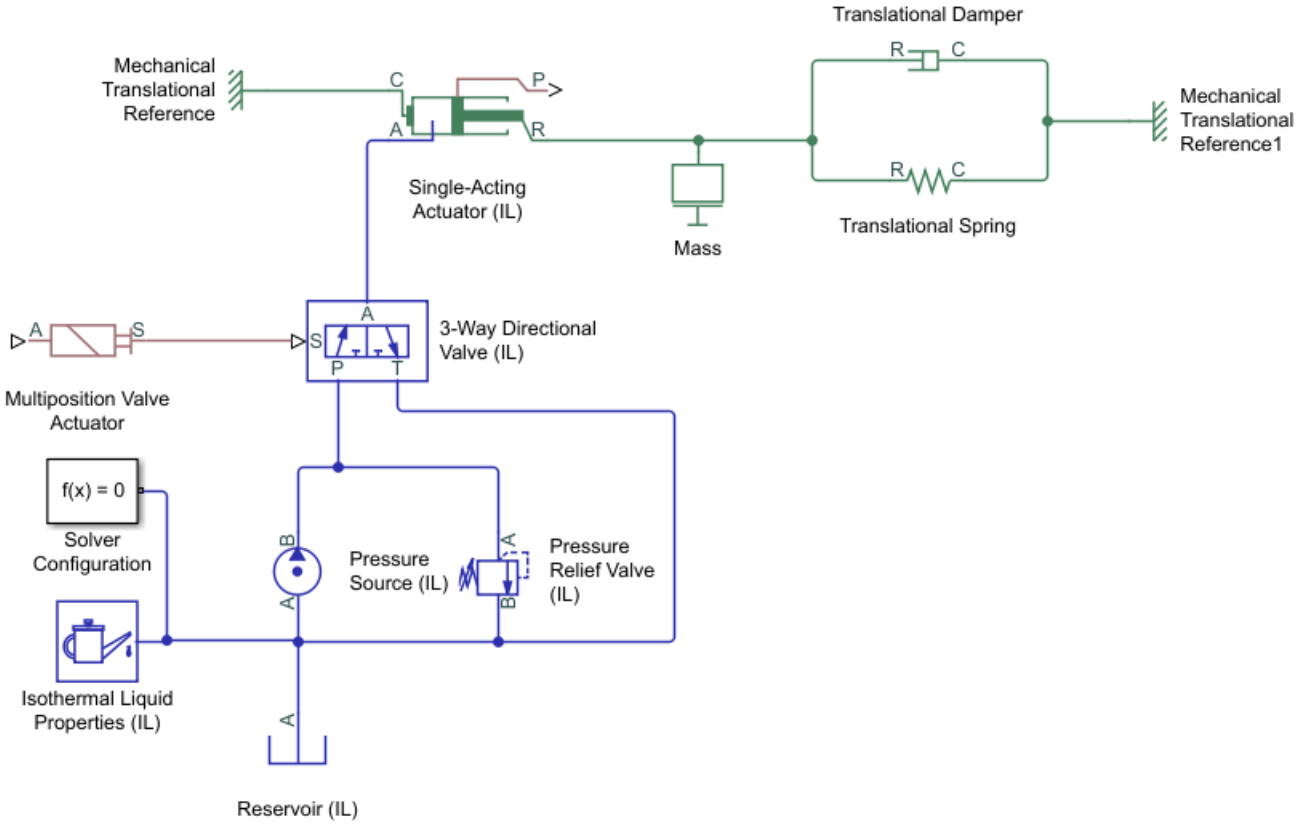
9 From the **Simscape > Foundation Library > Mechanical > Translational Elements** library, add a Mechanical Translational Reference block and connect it as shown in the figure.



This fixes the mechanical casing of the Single-Acting Actuator (IL), which is represented by port **C**. The actuator shaft, represented by port **R**, moves relative to this reference.

Add a Load Model

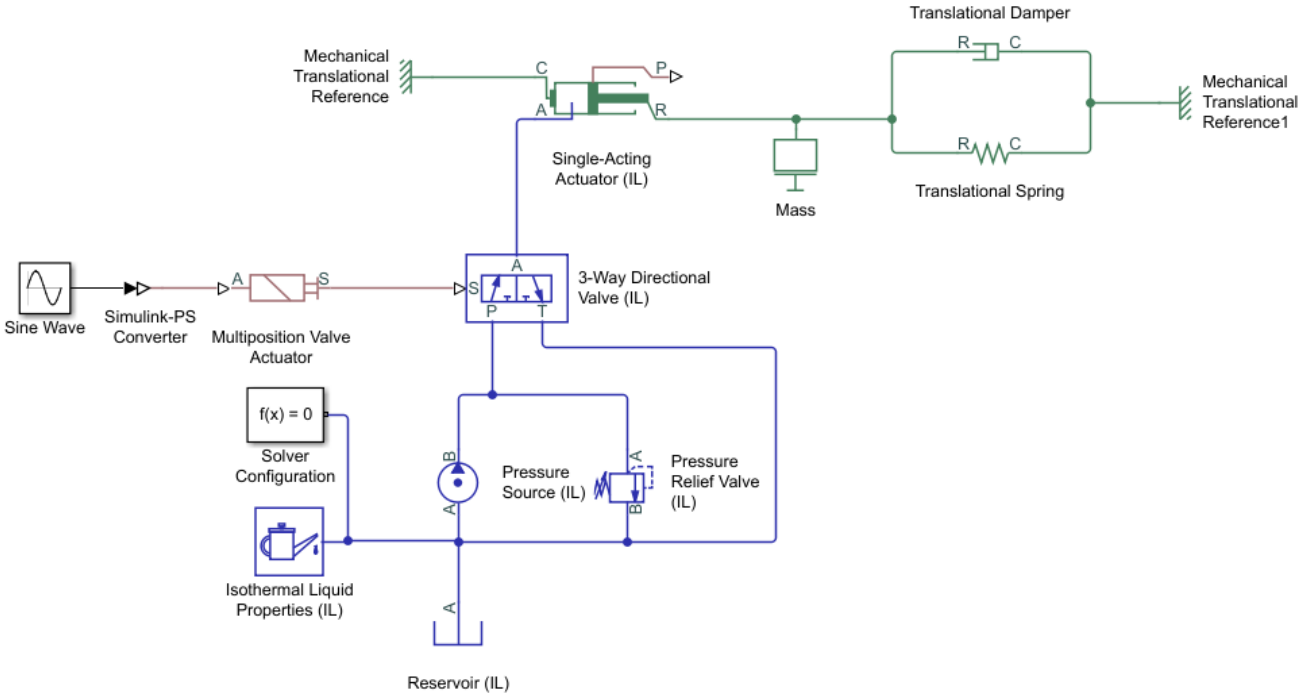
1. From the **Simscape > Foundation Library > Mechanical > Translational Elements** library, drag these blocks to the model canvas:
   - Mass
   - Translational Spring
   - Translational Damper
   - Mechanical Translational Reference
2. Connect the blocks as shown in the figure.



The mass, spring, and damper elements provide a basic model of a load with forcing, inertia, and damping.

Add a Control System and Monitoring

1. From the **Simulink** > **Sources** library, drag a Sine Wave to the canvas. This represents a varying signal to the control system, which in turn regulates the flow received by the actuator.
2. From the **Simscape** > **Utilities** library, drag one PS-Simulink Converter block. To convert between Simscape physical signals, which are bidirectional, and Simulink signals, which are unidirectional, you will need to use a Simulink-PS Converter or PS-Simulink Converter block.
3. Connect the Sine Wave block, the Simulink-PS Converter block that is already on the canvas, and Multiposition Valve Actuator block in series, as shown in the figure.

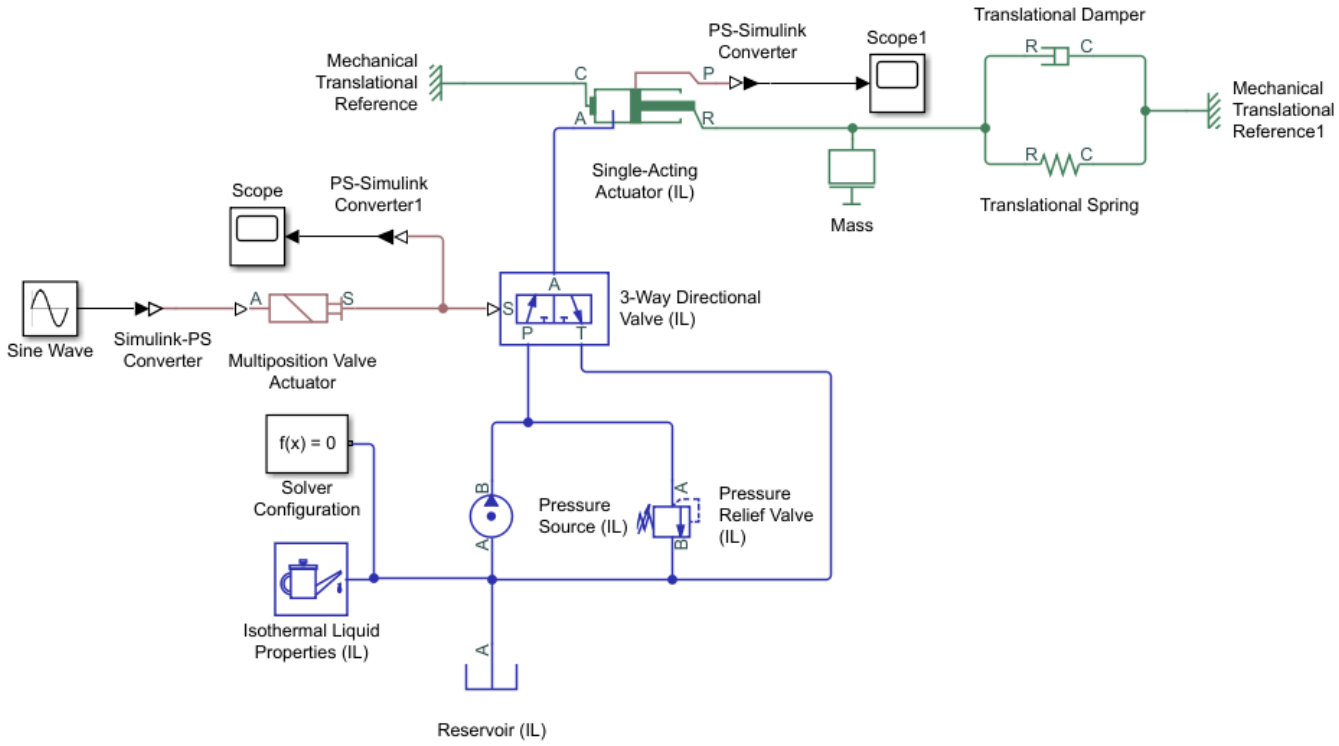


4 In the Sine Wave block dialog, set the **Amplitude** parameter to 24. This value corresponds to the default nominal signal value of the Multiposition Valve Actuator block.

5 Drag a Scope block from the Simulink Sources library. The Scope block allows you to quickly inspect model results.

6 Connect the Single-Acting Actuator (IL) block, a PS-Simulink Converter block, and a Scope block in series, as shown in the figure.

7 Connect the output side of the Multiposition Valve Actuator block, the other PS-Simulink Converter block, and a Scope block in series, as shown in the figure.



Port **P** of the Single-Acting Actuator (IL) block outputs a physical signal of the actuator shaft position, relative to the **Piston initial displacement**.

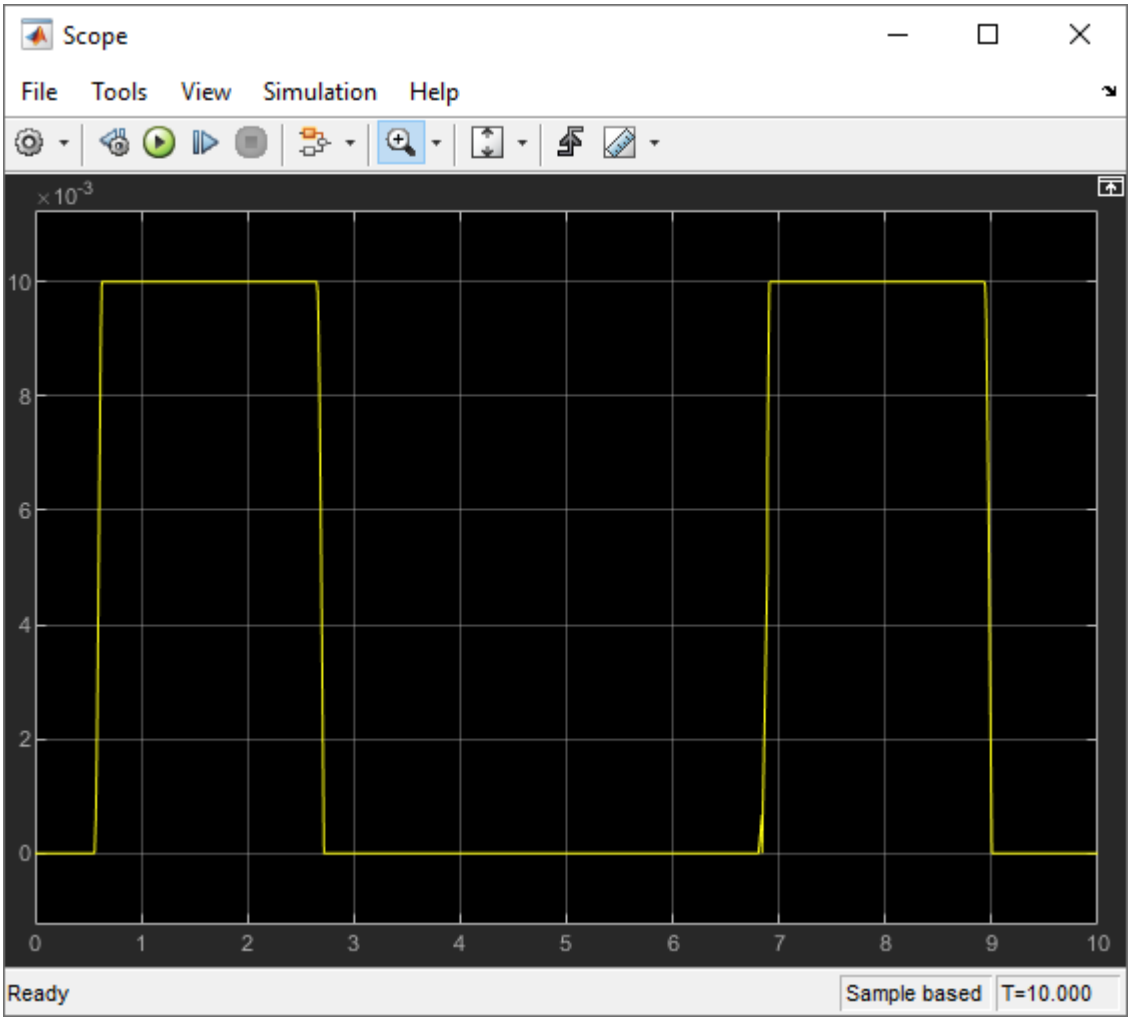
The Multiposition Valve Actuator block outputs a physical signal of the valve opening distance, relative to the **Initial position** and the **Push-pin stroke**.

Run the Simulation

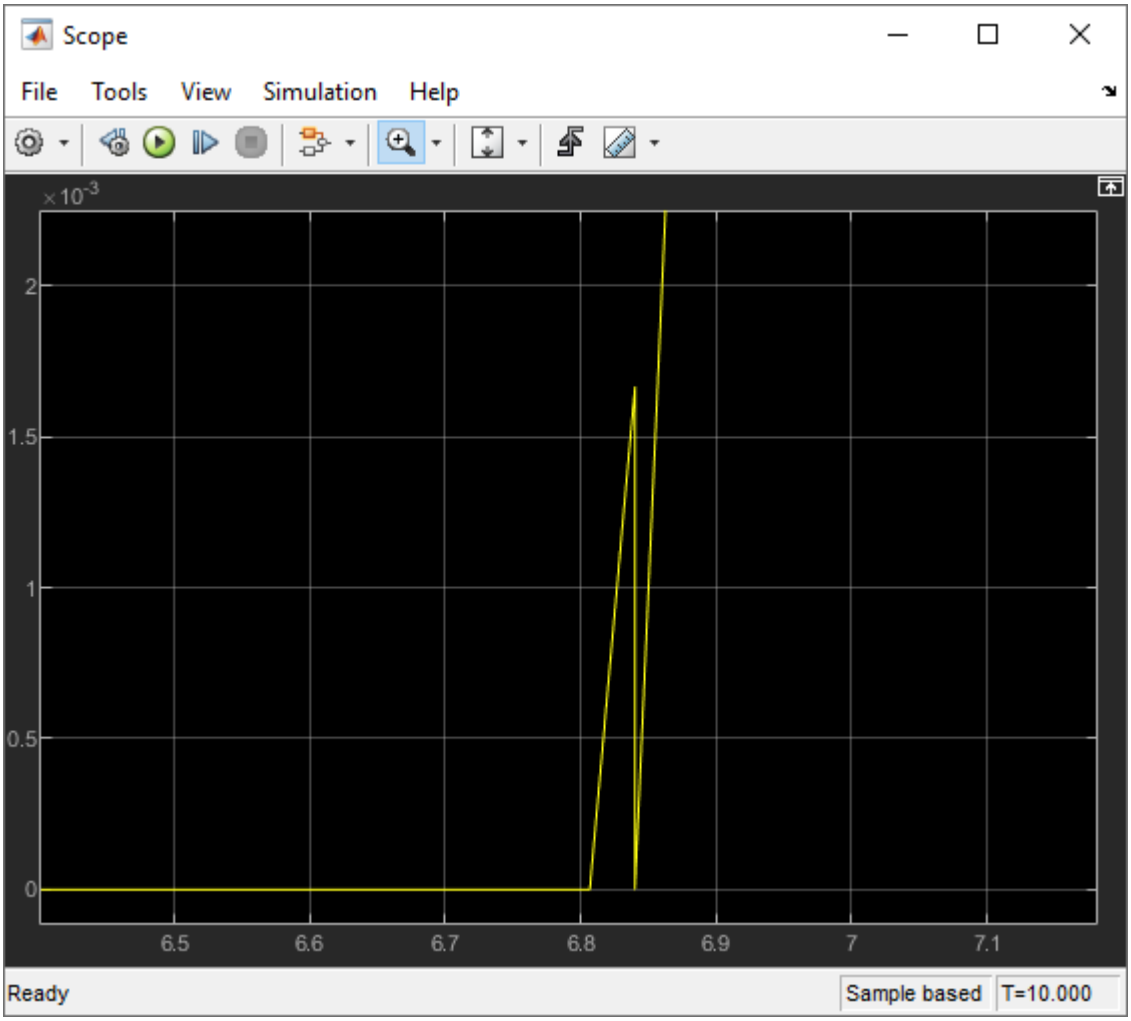
When you run the model, the Simscape software evaluates the model, determines the simulation initial conditions, and simulates the model. To learn more about this process, see “How Simscape Simulation Works”.

1. Click on any block in the model, for example the 3-Way Directional Valve (IL) block. In the **Simscape Block** tab, click **Log Data** and select **Log all Simscape data**. This enables you to explore the simulation variable values in the Results Explorer.
2. Click **Run**.
3. In the Debug tab, select **Simscape > Variable Viewer**. The “Variable Viewer” shows the initialization values and priorities of all of the model variables. You can set variable targets and priorities during initialization for blocks with a Variables tab in the block dialog. Verify that the status of all variables is green.
4. You can inspect simulation results in multiple ways. For a quick assessment of the model, use Scope blocks.

   Double-click on the Scope block connected to the Multiposition Valve Actuator. This shows the valve spool position, which is set by the Multiposition Valve Actuator signal.



Zooming in, there is a small spike in the signal at the beginning of the second peak.



This can be resolved by limiting the size of the solver time step:

a Open the Configuration Parameter dialog box. On the **Modeling** tab, select **Model Settings > Model Settings** and click **Solver** in the left-hand pane.

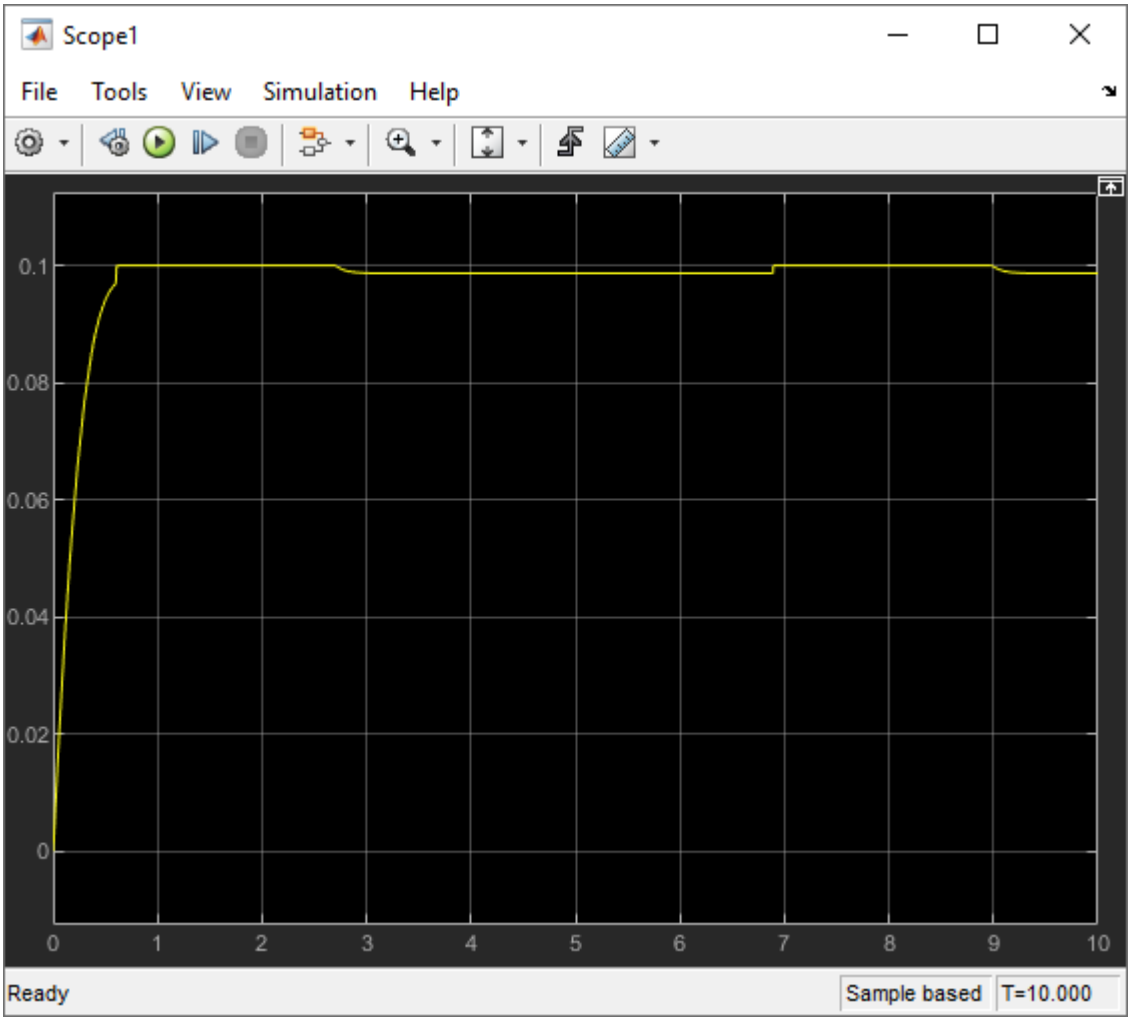
b Expand the Solver details option.

c Change **Max step size** from `auto` to `0.01`.

d Click **OK**.

This configuration will be used the next time you run the model.

Double-click on the Scope block connected to the Single-Acting Actuator (IL) block. The Scope plot shows the shaft position as a function of the pressure in the actuator chamber.



It appears that the actuator is not fully responding to the signal received from the control unit.

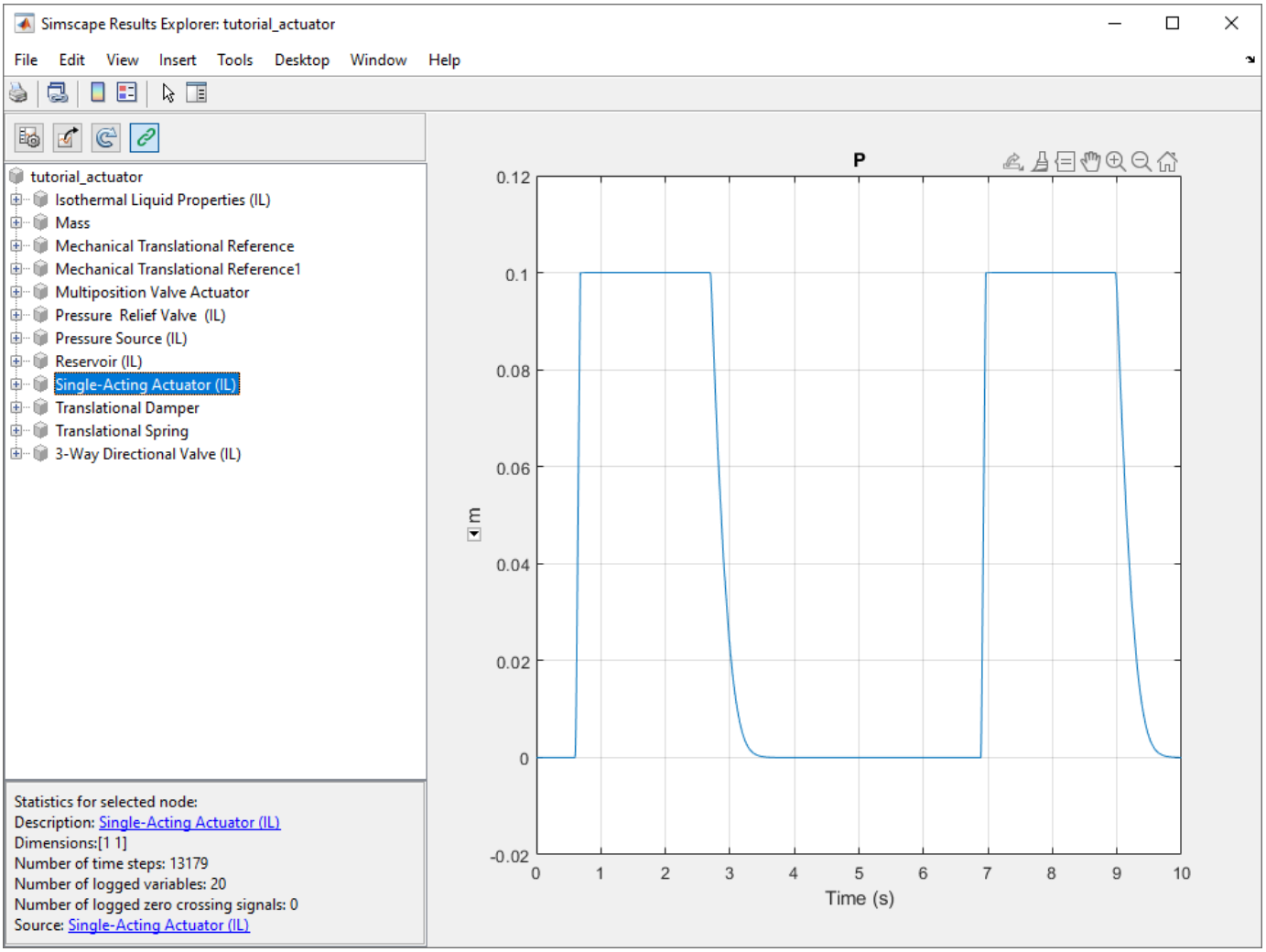
5. The environmental pressure is specified in three locations: in the Isothermal Liquid Properties (IL) block, in the Reservoir (IL) block, and in the Single-Acting Actuator (IL) block. In the model, the Reservoir (IL) block is set to 0.2 MPa and the other two are set to 0.101 MPa. In the Single-Acting Actuator (IL) block, set the **Environment pressure specification** to `Specified pressure` and set **Environment pressure** to `0.2` MPa.
6. Run the model.
7. Check the results with the Results Explorer by clicking on any block, such as the Multiposition Valve Actuator block, and selecting **Results Explorer** in the **Simscape Block** tab.
8. The Results Explorer shows every variable in the model, including internal variables, which cannot be reported by a Scope block. Click on the **Single-Acting Actuator (IL)** option in the left-hand pane. This shows a plot of the physical signal at port **P**, or the position of the actuator shaft. Click the plus next to the option to see the variables logged for the Single-Acting Actuator (IL) block.
9. The plot shows the shaft position for the last 3.5 seconds of the simulation. To see the shaft position over the entire simulation, turn data log limiting off:

a Open the Configuration Parameter dialog box. On the **Modeling** tab in the Simulink toolstrip, select **Model Settings > Model Settings** and click **Simscape** in the left-hand pane.

b Clear **Limit data points**.

c Run the model.

The actuator shaft position now follows the control signal.



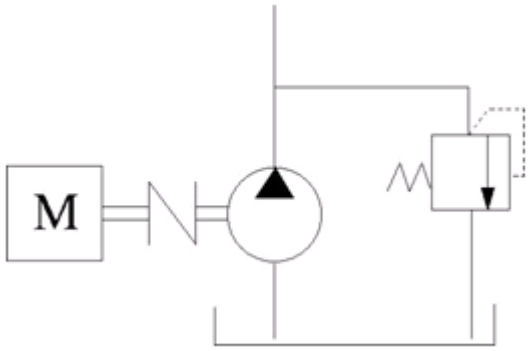
See Also

“Essential Steps for Constructing a Physical Model” | “Essential Physical Modeling Techniques”

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.

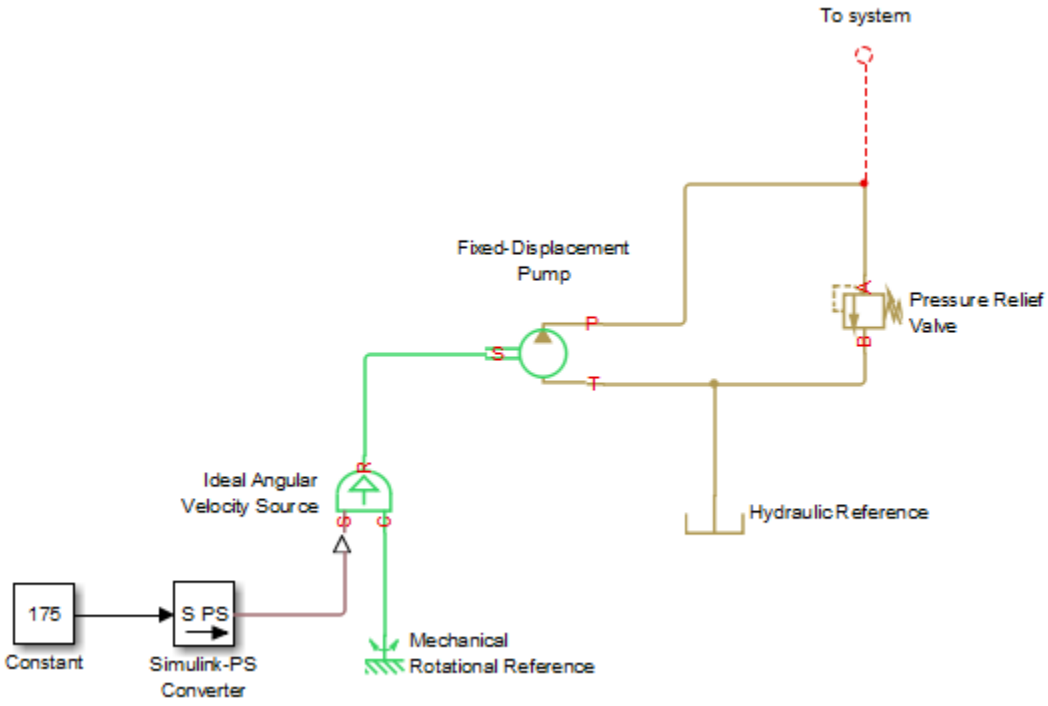
The first option is to simulate a power unit literally, as it is, reproducing all its components. This approach is illustrated in the “Closed-Loop Pump Control with Flexible Drive Shaft” on page 5-278 example. The power unit consists of a fixed-displacement pump, which is driven by a motor through a 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

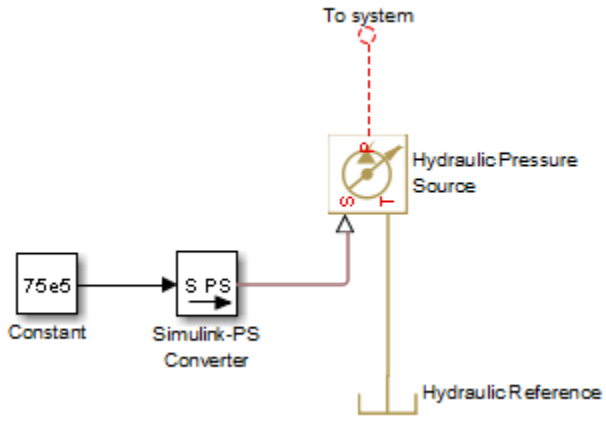
engine modeled using Simscape Driveline™ software can be combined with the Simscape Fluids model of the hydraulic portion of a power unit.

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.

See Also

Modeling Valves in Simscape Fluids

In this section...
“Valve Control with Valve Actuator Blocks” on page 2-31
“Four-Way Directional Valve Schematic” on page 2-31
“Parameterizing Four-Way Directional Valve Blocks” on page 2-33

You can model flow control, pressure control, and directional flow control with Fluids library valve blocks. Flow-controlling valves, such as a gate valve, limit the volumetric flow rate through a valve. Pressure-controlling valves limit valve operational pressures by opening or closing the valve based on pressure, or by venting excess flow.

Directional valves connect different flow paths based on the displacement of a central control member. They can be proportional-area, such as a four-way directional valve, or proportional-pressure, such as a check valve. Directional valves such as a shuttle valve can also control flow switching.

Proportional directional valves are configured based on the flow paths that open and close when the control member displaces, and are classified by:

- The number of external paths or connecting ports
- The number of positions the valve can assume
- Continuous or discrete movement

The directional valves in the Isothermal Liquid, Thermal Liquid, and Gas libraries comprise two-way, three-way, and four-way directional valves with a single, continuous spool control member. The block ports are named to correspond with common connections:

- P — Pump or pressure port
- T — Tank or return port
- A, B — Load ports
- X, Y — Pilot ports

Additionally, you can add mechanical effects to a valve by connecting blocks modeling damping, friction, and a hard stop to the valve actuator signal, such as in the “4-Way Directional Valve with Mechanical Effects” on page 5-372 example.

Valve Control with Valve Actuator Blocks

You can model servo or solenoid valve control with blocks from the Valve Actuator sublibrary, such as the Proportional Valve Actuator or the Pilot Valve Actuator (IL). To model solenoid control, you can also use a Solenoid block from the Simscape Electrical library.

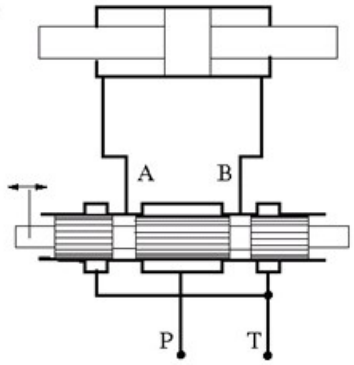
The Multiposition Valve Actuator block can be paired with directional valves to control the valve spool position according to input signals or switching time.

Four-Way Directional Valve Schematic

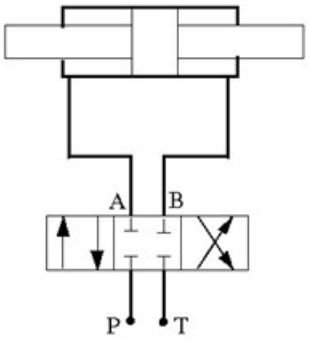
A proportional flow directional valve is characterized by its initial open flow path, the flow paths that open or close with positive or negative control member displacement, and the orifice opening offset.

The relationship between the control member stroke and the opening identifies whether an orifice is overlapped, underlapped, or neutral.

When a four-way, three-position directional valve has all ports closed in the neutral position, the valve, connected to a double-acting actuator, looks like this:

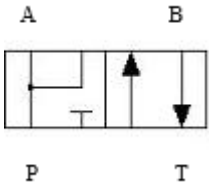


When the control member displaces in the positive direction, to the right in the valve diagram, flow paths P-A and B-T open. When the control member displaces in the negative direction, to the left in the valve diagram, flow paths P-B and A-T open. This configuration for positive and negative displacement is schematically represented as:



where the box on the left represents the positive spool displacement connections, and the box on the right represents the negative spool displacement connections.

In a four-way, two-position valve, additional flow configurations may open only during transition. For example, setting the 4-Way 2-Position Directional Valve (IL) block to this valve configuration



where:

- **Positive spool position open connections** is set to `P-A and P-B`.
- **Negative spool position open connections** is set to `P-A and B-T`.

results in open P-A, P-B, and B-T flow paths when the spool transitions between the positive and negative positions.

Note In the 4-Way 3-Position Directional Valve (IL) and 4-Way 2-Position Directional Valve (IL) blocks, you can:

- Select a pre-parameterization based on manufacturer data sheets. This can be set in the description section of the block dialog.
- Visualize the opening characteristics of each orifice in the valve. Right-click on the block and select **Fluids > Plot Valve Characteristics**.

Parameterizing Four-Way Directional Valve Blocks

In the 4-Way 3-Position Directional Valve (IL) and 4-Way 2-Position Directional Valve (IL) blocks, the *spool position at maximum orifice area* parameter indicates the absolute spool position at which that orifice is fully open. For example, when the valve is configured for a P-A flow path, the P-A orifice is open at the value specified in the **Spool position at maximum P-A orifice area** parameter. Orifices identified in the **Positive spool position open connections** parameter must have a positive spool position at the maximum opening. Orifices identified in the **Negative spool position open connections** parameter must have a negative spool position at the maximum opening. If these valves are not also identified in the **Neutral spool position open connections** parameter, they must be closed at a spool position of 0.

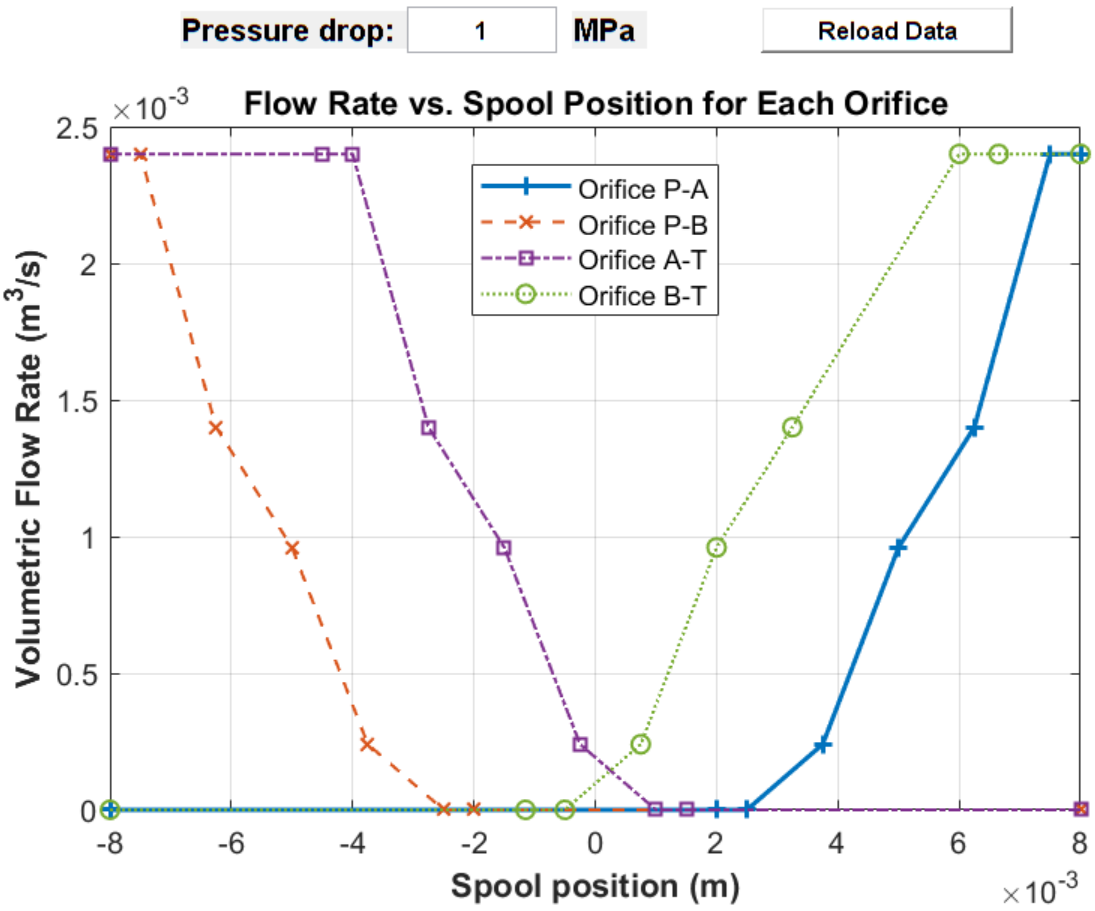
The *spool travel between closed and open orifice* parameter is the stroke that opens the orifice. If **Area characteristics** is set to `Different for each flow path`, the spool travel between the closed and open orifice can be defined differently for each flow path.

To check that the defined spool positions for each orifice opening and closing position comprise a consistent configuration, the **Consistency check for neutral spool position open connections** parameter can notify you if parameterization values are inconsistent when the spool is in the neutral position. A warning or error is generated when you plot or refresh the plot of the valve configuration.

In the 4-Way Directional Valve (TL) and 4-Way Directional Valve (G) blocks, the *maximum valve opening* parameter is the orifice stroke, which corresponds to the *spool travel between closed and open orifice* parameter in the isothermal liquid blocks. The parameters under **Valve Opening Offsets**, such as **Between ports P and A**, identify the spool displacement from neutral at which the flow path begins to open for each orifice, such as the one between ports **P** and **A**. These parameters correspond to the spool position at the orifice *minimum* opening.

Visualizing a Valve Configuration

In the 4-Way 3-Position Directional Valve (IL) and 4-Way 2-Position Directional Valve (IL) blocks, you can visualize the orifice area for each flow path as a function of spool position:



This visualization shows a four-way, three-position valve configured with:

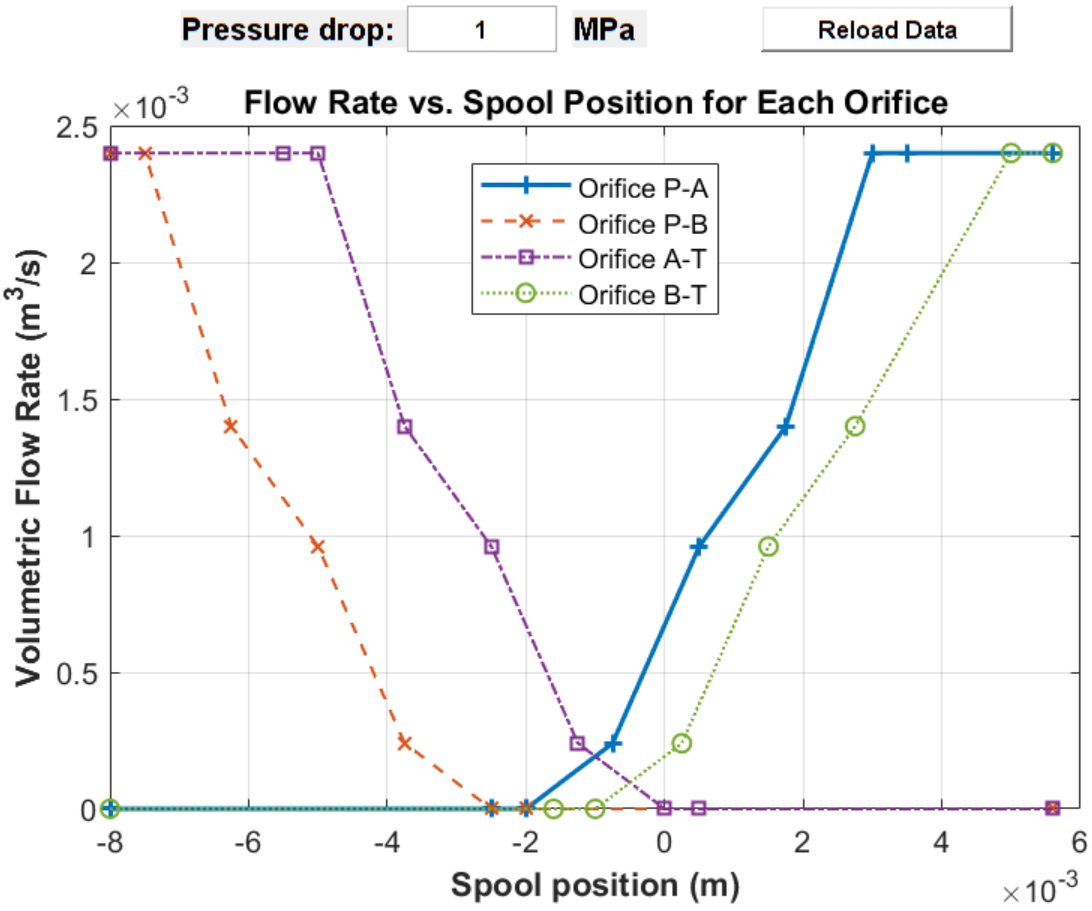
- **Positive spool position open connections** set to `P-A and B-T`
- **Neutral spool position open connections** set to `A-T and B-T`
- **Negative spool position open connections** set to `P-B and A-T`
- **Area characteristics** set to `Different for each flow path`
- **Orifice parameterization** set to `Tabulated data - Volumetric flow rate vs. spool travel and pressure drop`

With the following values changed from the block defaults:

- **Spool position at maximum A-T orifice area** set to `-0.004` m
- **Spool position at maximum B-T orifice area** set to `0.006` m
- **B-T orifice spool travel vector, ds** set to `[0, .00125, .0025, .00375, .0065]` m.

The visualization shows that orifices A-T and B-T have a nonzero flow rate, or are open, when the spool position is 0. Orifice B-T is fully closed when the spool displaces to -0.0005 m; it is fully open at 0.006 m and has an opening length of 0.0065 m. Orifice A-T is fully closed at 0.001 m; it is fully open at -0.004 m and has an opening length of 0.005 m.

Changing **Spool position at maximum P-A orifice area** to `0.003` m results in an open P-A orifice when the spool is in the neutral position, which violates the valve configuration settings. This is because the last element of the **P-A orifice spool travel vector, ds**, is larger than the **Spool position at maximum P-A orifice area**, meaning that the valve must start opening when the spool is still in the negative position:



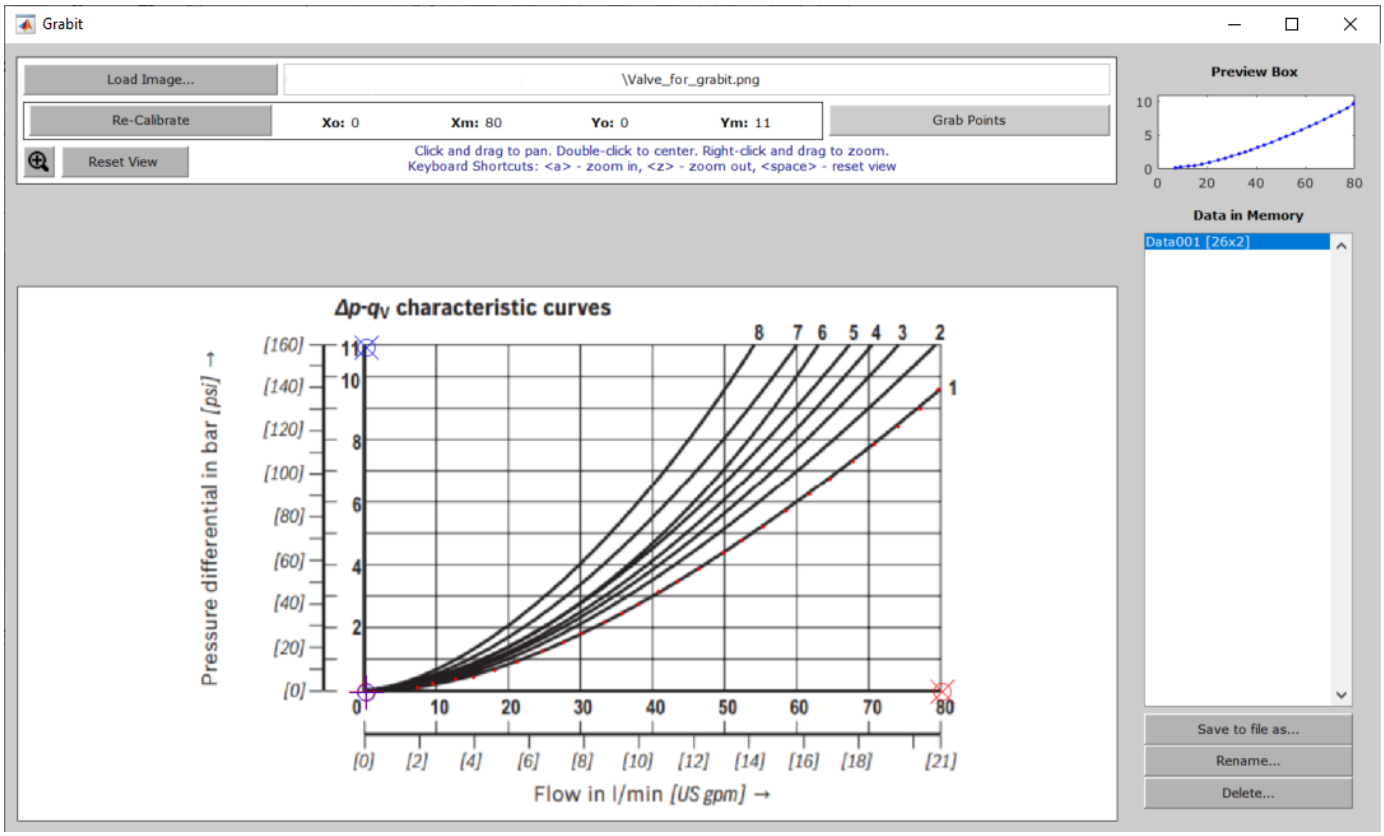
Parameterization with a Datasheet

Manufacturer data sheets provide flow rate–pressure differential curves for the different flow paths of a directional valve. You can parameterize a directional valve block with data from the manufacturer by setting **Orifice parameterization** to `Tabulated data - Volumetric flow rate vs. spool travel and pressure drop`.

If you do not want to manually input data from a data sheet, you can use the MATLAB File Exchange function `grabit` to extract data points from a plot image:

1. Enter `grabit()` at the MATLAB command line.
2. From the GUI that appears, click **Load Image** to load a valve curve in PNG, TIF, JPG, GIF, or BMP format.
3. Calibrate the image by selecting the axes ranges and entering the range values. Start by clicking on the *x*-axis origin, entering `0` in the box that appears, and pressing **Enter**.

4 Click **Grab Points** to begin collecting data by clicking along the desired curve. To collect another set of data, press **Enter** and click **Grab Points** again.



5 Click **Save to file as** to save the dataset as `P_A_pressure_drop.mat` in your working directory.

6 Enter the saved variables as the parameter setting:

See Also

Multiposition Valve Actuator | 4-Way 3-Position Directional Valve (IL) | 4-Way 2-Position Directional Valve (IL) | 4-Way Directional Valve (G) | 4-Way Directional Valve (TL)

Related Examples

- “Building a Custom Valve” on page 2-37
- “Front-Loader Actuation System” on page 5-42
- “4-Way Directional Valve with Mechanical Effects” on page 5-372
- “4-Way 3-Position Valve Parameterization” on page 5-26

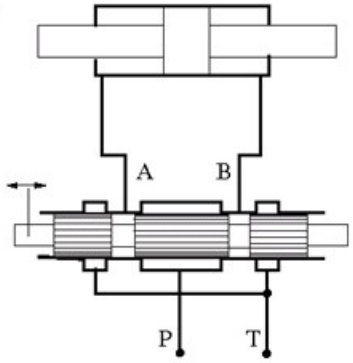
Building a Custom Valve

Valves can be modeled with one or more orifice blocks in series or parallel. For valves that open or close in response to pressure or flow rate, such as a pressure relief valve, you can model your own by connecting a pressure sensor or flow rate sensor to an orifice. Additionally, you can add mechanical effects such as damping and friction, as shown in “4-Way Directional Valve with Mechanical Effects” on page 5-372.

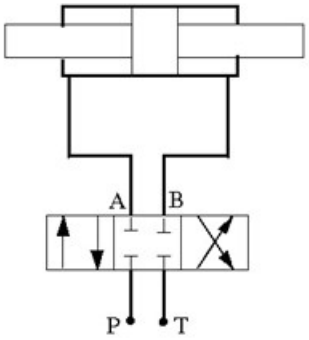
You can use the Solenoid block to model a solenoid-controlled valve. For more information, see “Valve Control with Valve Actuator Blocks” on page 2-31.

Fundamental Directional Valve Composite Structures

When a four-way, three-position directional valve has all ports closed in the neutral position, the valve, connected to a double-acting actuator, looks like this:



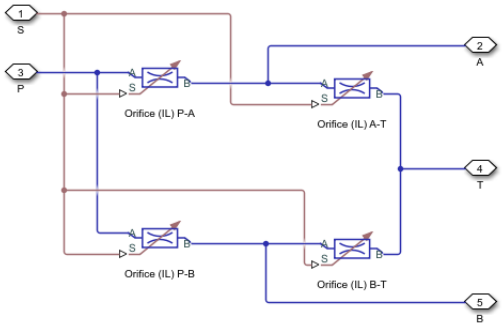
When the control member displaces in the positive direction, flow paths P-B and A-T open. When the control member displaces in the negative direction, flow paths A-P and B-T open. This is schematically represented as:



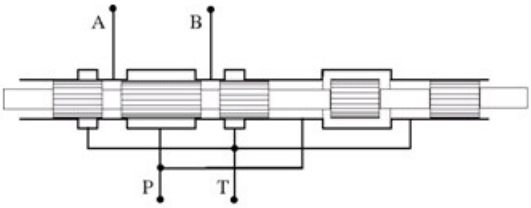
This valve can be modeled as a collection of orifice blocks. In the isothermal liquid domain, this four-way, three-position valve can be recreated with four Orifice (IL) blocks with:

- **Orifice type** set to `Variable`.

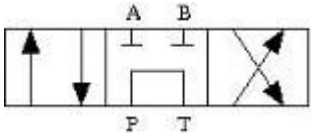
- **Control member position at closed orifice** set to `0` (all valves are closed in the neutral position).
- In Orifice (IL) P-A and Orifice (IL) B-T, **Opening orientation** set to `Positive control member displacement opens orifice`.
- In Orifice (IL) B-P and Orifice (IL) A-T, **Opening orientation** set to `Negative control member displacement opens orifice`.
- For equally proportional flow, set a uniform **Control member travel between closed and open orifice** value for all orifices. For more control over the amount of flow through an orifice, increase the control member travel value with respect to the other orifices.



A different four-way, three-position valve that starts with closed load ports and an open connection between the pump and tank is:

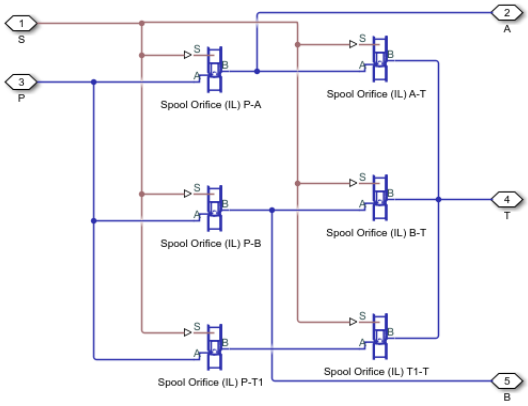


When the control member displaces in the positive direction, flow paths A-T and P-B open. When the control valve displaces in the negative direction, flow paths A-P and B-T open. In the neutral position, flow path P-T is open. This is represented schematically as:

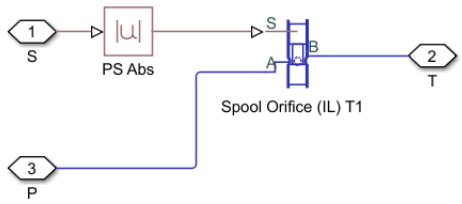


This valve can be built with a collection of spool orifice blocks. One way to model this valve is with two sets of orifices that correspond to an intermediate opening, T1, that represents an opening between **P** and **T** that is only open in the neutral position. One orifice is configured to be open with positive spool displacement, and the other is configured to be open with negative spool displacement.

In addition to the variable orifice block features, the spool orifice block allows you to set the orifice geometry (a series of round holes or one rectangular slot), record the orifice flow force, or connect the flow force physical signal to another block. In the isothermal liquid domain, this directional valve can be represented as:



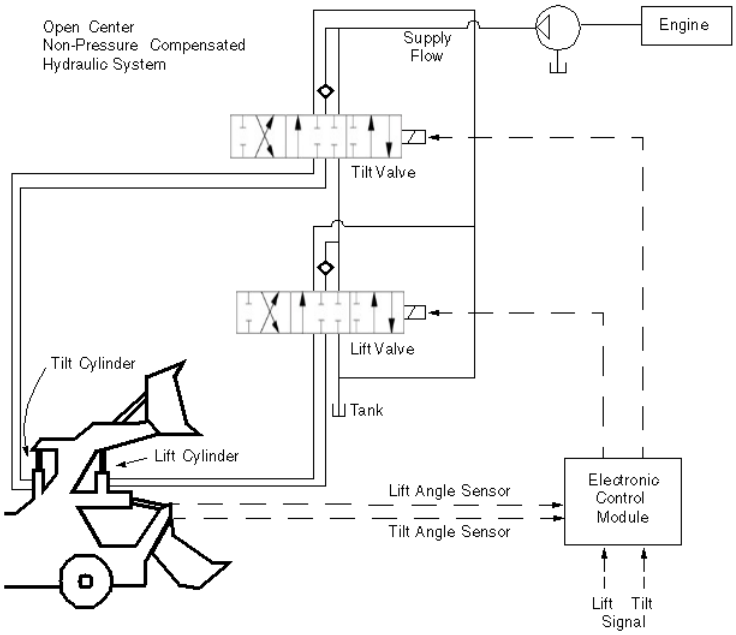
Another way to model an orifice that is always open in the neutral position is to connect one orifice **S** port to the position signal through a PS Abs block and set **Opening orientation** to `Negative spool displacement opens the orifice`:



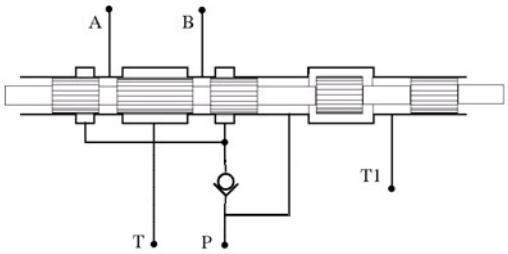
The Spool Orifice (IL) T1 block is configured with **Spool position at closed orifice** set to `0` m.

Custom Directional Valve for a Front Loader

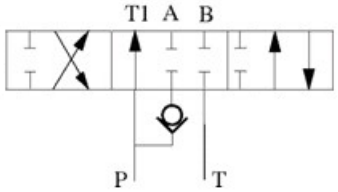
The front loader shown below handles lift and the tilt actuation with two six-way three-position directional valves. If both valves are in the neutral position, the pump port connects to the tank port. The pump is disconnected from the tank if either of the two valves moves from the neutral position.



The valve looks like this:



and is represented schematically as:

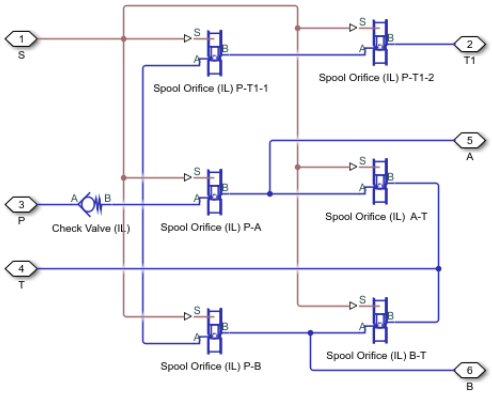


The valve can be modeled as a combination of spool orifice blocks and a check valve block. Orifices P-A, P-B, B-T and A-T are closed when the valve is in the neutral position. Orifices P-T1-1 and P-T1-2 are open at the beginning of the simulation, which allows pump delivery at low pressure to the tank.

In the isothermal liquid domain, in the Spool Orifice (IL) blocks:

- Set **Orifice geometry** to `Round holes`.
- Set **Diameter of round holes** to `0.006`.

- Set **Number of round holes** to 6.
- In Orifices P-A, P-B, A-T, and B-T, set **Spool position at closed orifice** to `-0.006`.
- In Orifices P-B and A-T, set **Opening orientation** to `Positive spool displacement opens the orifice`.
- In Orifices P-A and B-T, set **Opening orientation** to `Negative spool displacement opens the orifice`.
- In Orifices P1-T1-1 and P1-T1-2, set **Spool position at closed orifice** to `0.008`.



See Also

Multiposition Valve Actuator | 4-Way 3-Position Directional Valve (IL) | 4-Way 2-Position Directional Valve (IL) | Orifice (IL) | Spool Orifice (IL)

Related Examples

- “Modeling Valves in Simscape Fluids” on page 2-31
- “Front-Loader Actuation System” on page 5-42
- “4-Way Directional Valve with Mechanical Effects” on page 5-372
- “4-Way 3-Position Valve Parameterization” on page 5-26

Modeling Hydraulic Directional Valves

In this section...
“Classifying Directional Valves” on page 2-42
“4-Way Directional Valve Schematic” on page 2-42
“Building a Custom Directional Valve” on page 2-47

Classifying Directional Valves

Directional valves connect different flow paths based on the displacement of a central control member. They can be proportional, such as a 4-way directional valve, or control flow switching, such as a shuttle valve. Proportional directional valves are configured based on the flow paths that open and close when the control member displaces, and are classified by:

- The number of external paths or connecting ports.
- The number of positions the valve can assume.
- Being continuous or discrete.

The amount of open orifice at a given time is determined by:

$$h = x_0 + x_{sp}\varepsilon,$$

where:

- h is the current open orifice distance.
- x_0 is the initial orifice offset. This value is positive for underlapped (initially open) valves and negative for overlapped (initially closed) valves.
- x_{sp} is the spool displacement from its original position.
- ε is the orifice orientation. It is +1 is the orifice opens when the spool displaces in the positive direction and -1 if the orifice closes when the spool displaces in the positive direction.

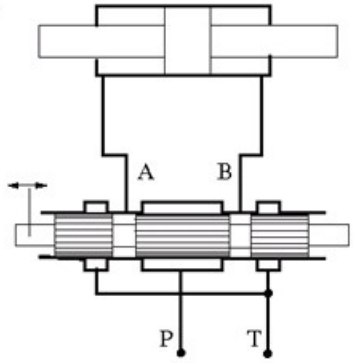
The directional valves in the Simscape Fluids Hydraulics (Isothermal) library comprise two-way, three-way, four-way, and six-way directional valves with a single, continuous spool control member. Block ports are named to correspond with common connections:

- P — Pump or Pressure port
- T — Tank or Return port
- A, B — Load ports
- X, Y — Pilot ports

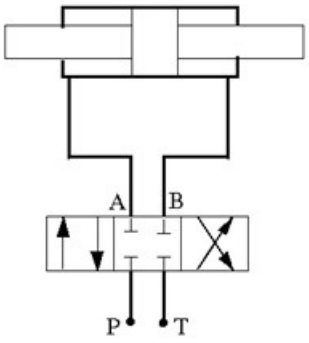
4-Way Directional Valve Schematic

A proportional flow directional valve is characterized by its initial open flow path, the flow paths that open or close with positive or negative control member displacement, and the orifice opening offset. The relationship between the control member stroke and the opening identifies whether an orifice is overlapped, underlapped, or neutral.

One configuration of a 4-way, 3-position directional valve connected to a double-acting cylinder can be connected with all four ports closed in the neutral position. The valve looks like this:



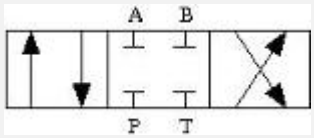
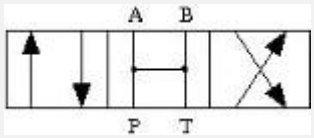
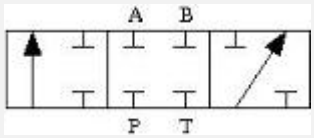
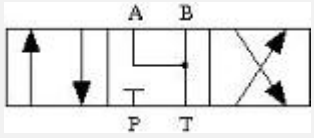
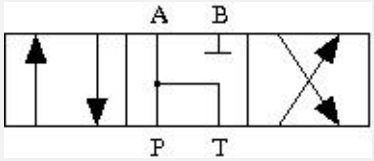
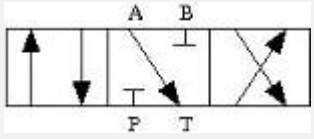
When the control member displaces in the positive direction, to the right in the valve diagram, the flow paths P-A and B-T open. When the control member displaces in the negative direction, to the left in the valve diagram, flow paths B-P and A-T open. This configuration for positive and negative displacement is schematically is represented as:

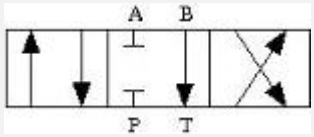
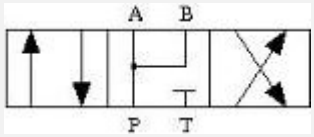
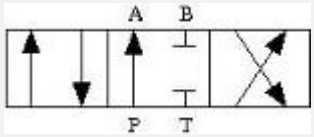
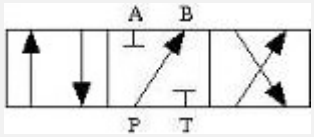
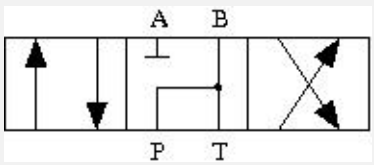


where the box on the right represents the positive spool displacement connections, and the box on the left represents the negative spool displacement connections.

The 4-Way Directional Valve block models this and ten additional configurations, specified in the following table.

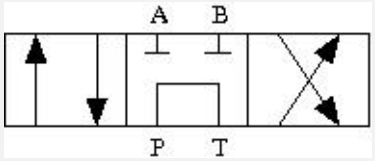
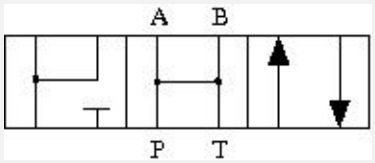
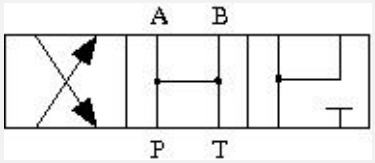
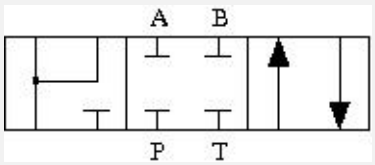
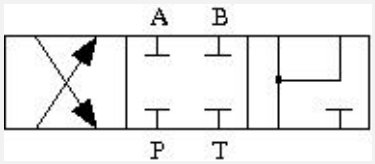
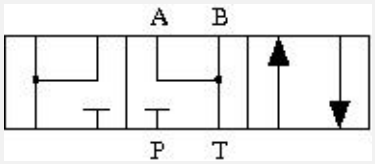
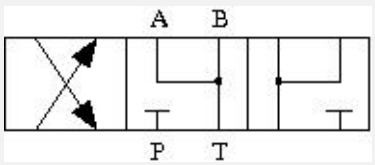
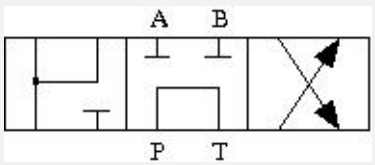
The 4-Way Directional Valve Configurations

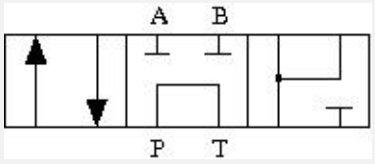
No	Configuration	Initial Openings
1		All four orifices are overlapped in neutral position: • **Orifice P-A initial opening** < 0 • **Orifice P-B initial opening** < 0 • **Orifice A-T initial opening** < 0 • **Orifice B-T initial opening** < 0
2		All four orifices are open (underlapped) in neutral position: • **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		Orifices `P-A` and `P-B` are overlapped. Orifices `A-T` and `B-T` are overlapped for more than valve stroke: • **Orifice P-A initial opening** < 0 • **Orifice P-B initial opening** < 0 • **Orifice A-T initial opening** < – *`valve_stroke`* • **Orifice B-T initial opening** < – *`valve_stroke`*
4		Orifices `P-A` and `P-B` are overlapped, while orifices `A-T` and `B-T` are open: • **Orifice P-A initial opening** < 0 • **Orifice P-B initial opening** < 0 • **Orifice A-T initial opening** > 0 • **Orifice B-T initial opening** > 0
5		Orifices `P-A` and `A-T` are open in neutral position, while orifices `P-B` and `B-T` are overlapped: • **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		Orifice `A-T` is initially open, while all three remaining orifices are overlapped: • **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
7		Orifice B-T is initially open, while all three remaining orifices are overlapped: • **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		Orifices P-A and P-B are open, while orifices A-T and B-T are overlapped: • **Orifice P-A initial opening** > 0 • **Orifice P-B initial opening** > 0 • **Orifice A-T initial opening** < 0 • **Orifice B-T initial opening** < 0
9		Orifice P-A is initially open, while all three remaining orifices are overlapped: • **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		Orifice P-B is initially open, while all three remaining orifices are overlapped: • **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		Orifices P-B and B-T are open, while orifices P-A and A-T are overlapped: • **Orifice P-A initial opening** < 0 • **Orifice P-B initial opening** > 0 • **Orifice A-T initial opening** < 0 • **Orifice B-T initial opening** > 0

Additionally, the Hydraulics (Isothermal) library has nine additional 4-way directional valve blocks, the 4-Way Directional Valves A - K, shown in the following table.

4-Way Directional Valves A - K Configurations

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

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

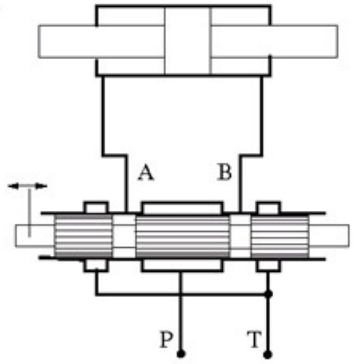
Building a Custom Directional Valve

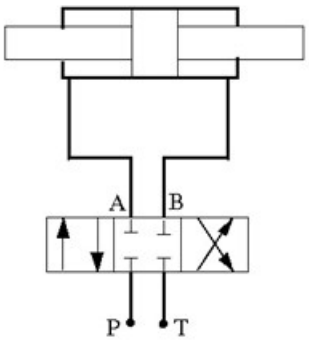
Directional valves can be modeled as a collection of orifices in series and parallel. Available building blocks in the Hydraulics (Isothermal) library for building directional valves are:

- Annular Orifice
- Orifice with Variable Area Round Holes
- Orifice with Variable Area Slot
- Variable Orifice
- Ball Valve
- Needle Valve
- Poppet Valve

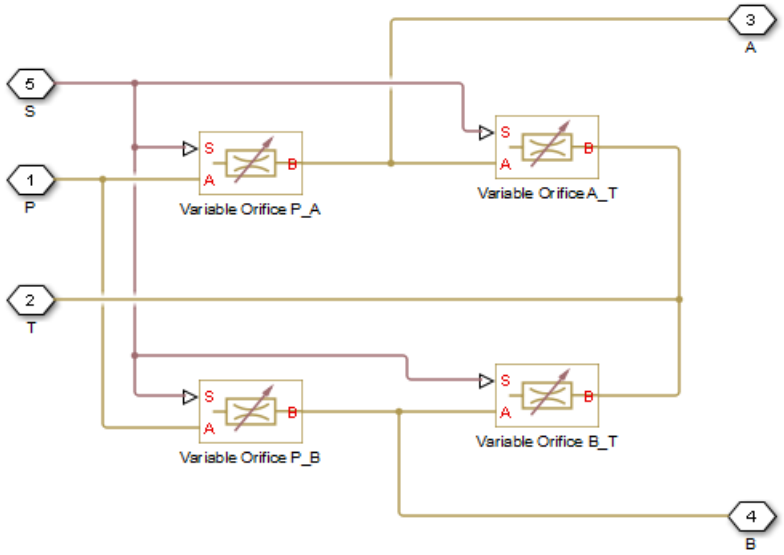
Fundamental Directional Valve Composite Structures

The four-way, three-position valve introduced above:

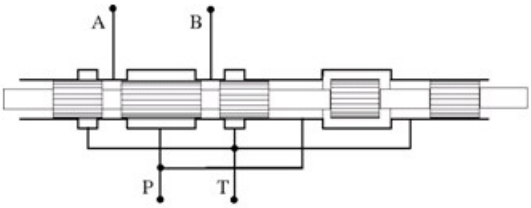




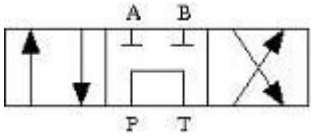
can be modeled as a collection of Variable Orifice blocks:



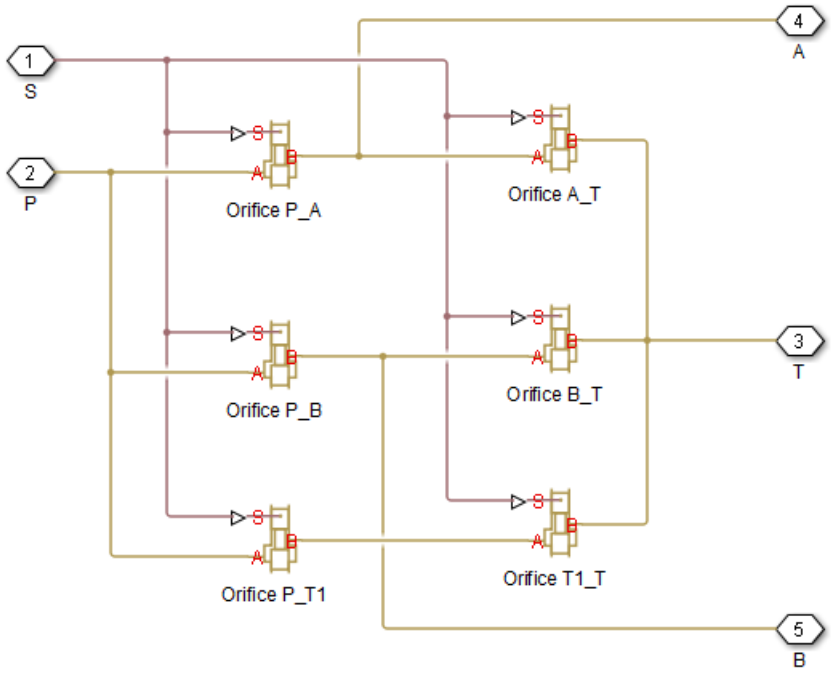
A different valve that starts with closed load ports and an open connection between the pump and tank is:



When the control member displaces in the positive direction, the flow paths P-A and B-T open. When the control valve displaces in the negative direction, flow paths B-P and A-T open. In the neutral position, flow path P-T is open. This is represented schematically as:

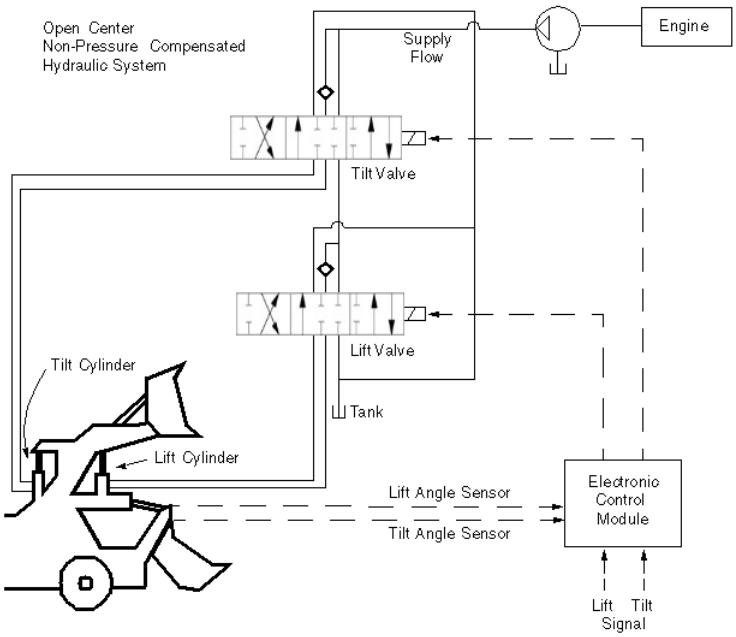


It can be built with a collection of Orifice with Variable Area Round Holes blocks. T1 corresponds to an intermediate point between the pump and tank.

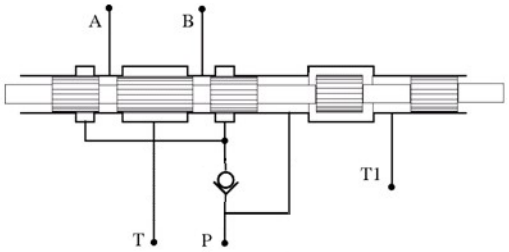


Custom Directional Valve for a Front Loader

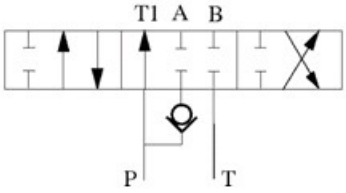
The front loader shown below handles lift and the tilt actuation with two 6-way 3-position directional valves. If both valves are in the neutral position, the pump port connects to the tank port. The pump is disconnected from the tank if either of the two valves moves from the neutral position.



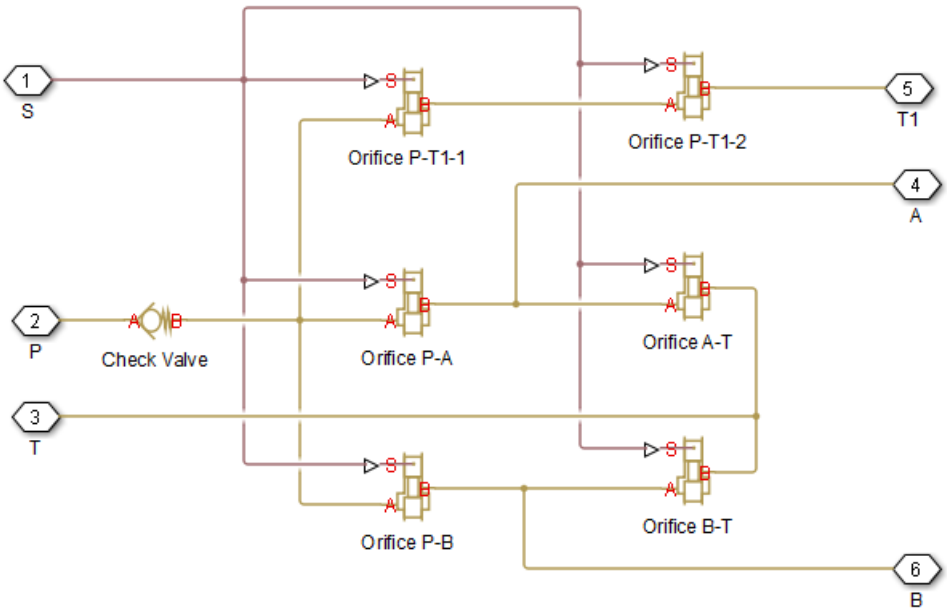
The valve looks like this:



and is represented schematically as:



It can be modeled as a combination of Orifice with Variable Area Round Holes blocks and a Check Valve block.



All of the orifices, except `P_T1_1` and `P_T1_2`, are overlapped when the valve is in the neutral position. Orifices `P_T1_1` and `P_T1_2` begin the simulation with an opening offset (underlapping) that allows pump delivery at low pressure to the tank.

See Also

4-Way Directional Valve | 3-Way Directional Valve | 2-Way Directional Valve | Shuttle Valve | Check Valve

Related Examples

- “Power-Assisted Steering Mechanism” on page 5-270
- “Hydrostatic Transmission with Shuttle Valve” on page 5-236

Modeling Low-Pressure Fluid Transportation Systems

In this section...
“How Fluid Transportation Systems Differ from Power and Control Systems” on page 2-52 “Available Blocks and How to Use Them” on page 2-53 “Low-Pressure Fluid Transportation System” on page 2-55

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

$$-\frac{\dot{W}_s}{\dot{m}g} = \frac{V_2^2 - V_1^2}{2g} + \frac{p_2 - p_1}{\rho g} + z_2 - z_1 + h_L \tag{2-1}$$

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” 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” for a typical passive component is presented in the form

$$\frac{\rho}{2}V_1^2 + p_1 + \rho g z_1 = \frac{\rho}{2}V_2^2 + p_2 + \rho g z_2 + p_L \tag{2-2}$$

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" takes the form

$$p_1 + \rho g z_1 = p_2 + \rho g z_2 + p_L \tag{2-3}$$

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 2-53) have been developed with respect to static pressures, with the following equations

$$\begin{aligned} p &= p_L = f(q) \\ q &= f(p_1, p_2) \end{aligned} \tag{2-4}$$

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

$$\begin{aligned} p &= p_L = f(q, z_1, z_2) \\ q &= f(p_1, p_2, z_1, z_2) \end{aligned} \tag{2-5}$$

Equations in the form "Equation 2-5" 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 2-53 .
- 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.

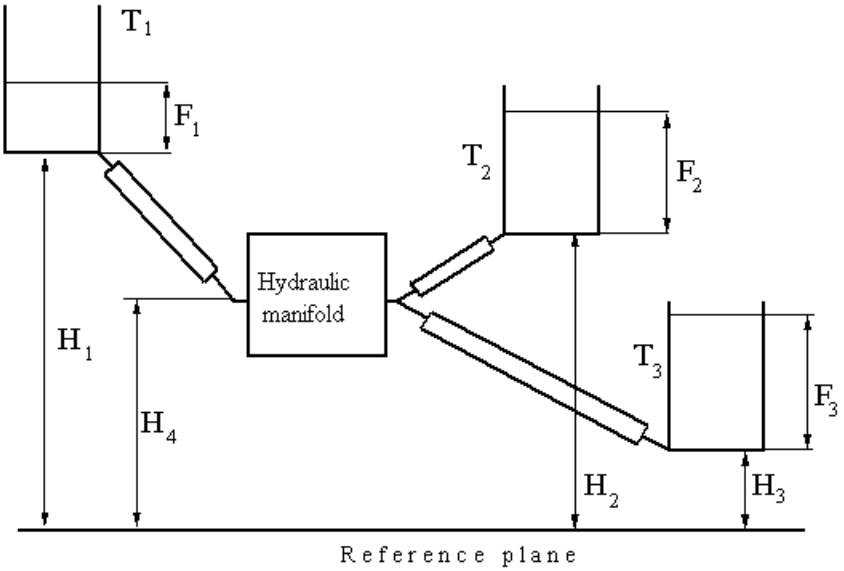
Use these low-pressure pipe blocks to connect all Hydraulic nodes in your model with difference in elevation, because these are the only blocks that provide information about the vertical location of the ports. Nodes connected with any other blocks, such as valves, orifices, actuators, and so on, will 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 H1, H2, and H3, 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 H4.



The models of tanks account for the fluid level heights F1, F2, and F3, respectively, and represent pressure at their bottoms as

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

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” on page 5-328 and the “Three Constant Head Tanks” on page 5-331 examples.

Numerical Smoothing

Zero-Crossings

Two types:

Bounded Variables

Bounded

See Also

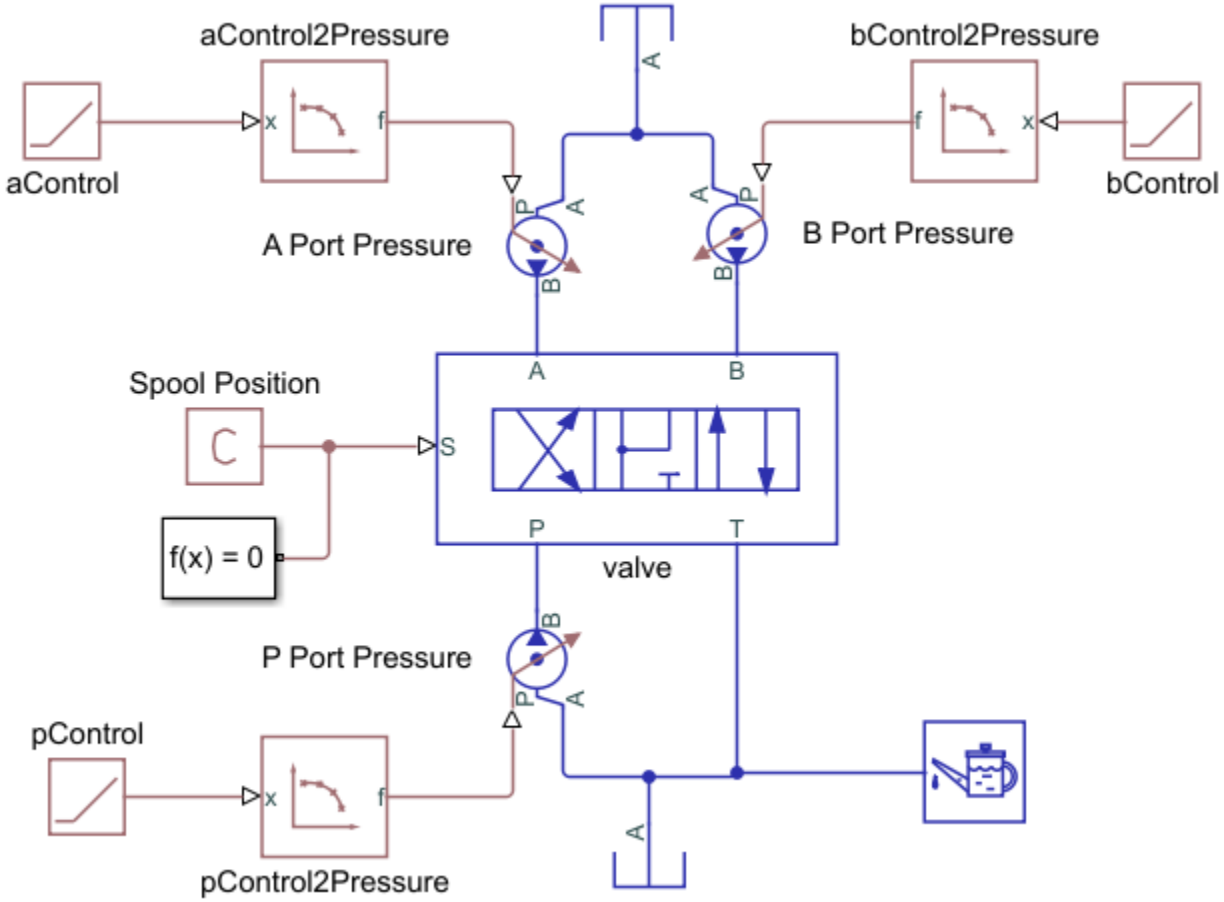
3

Verification and Validation

Validating Flow Response With the Simulation Data Inspector

This example shows the workflow for a unit test assessment of model suitability based on the mass flow rate through a valve by using the Simulation Data Inspector (Data Inspector). Performing a successful validation ensures that you are using the most appropriate equations to model a specific application or phenomena. Testing a model's response in a *test harness* helps to assess the response of a model to a controlled range of inputs. You can find other test harness examples in “Test Harnesses”.

This tutorial uses the example `sscfluids_4_way_3_position_valve`, which is composed of a Simulink model and a live script that runs the harness for multiple spool positions:



The `sscfluids_4_way_3_position_valve` example uses a script to run and process the model and plot the simulation data. This tutorial shows an alternative way to validate the model by using the Simulation Data Inspector. Open the model by entering `sscfluids_4_way_3_position_valve` at the command line. Open the live script by clicking the Open live script link at the bottom of the model.

Build a Test Harness

To collect simulation data to validate the four-way valve, construct a unit test that interrogates the valve at a range of pressures. The `sscfluids_4_way_3_position_valve` is an example of a four-way valve in a test harness:

- Use a Ramp block and a PS-Lookup Table (1D) block to set the testing range of the valve. The ramp block specifies the simulation time and the look-up table specifies the validating data range. In `sscfluids_4_way_3_position_valve`:
  - Mass flow rate is the chosen variable to validate against specification sheet data. The test harness is configured to steadily increase pressure from 0 to 5.8 bar.
  - The specification sheet data is for maximum open area, which is when the spool positions are fully extended in the positive and negative positions. The simulation runs two tests: one for the flow paths open at the maximum positive position, and one for the flow paths open at the maximum negative spool position.
- Store your validating data in a Simulation Data Inspector-friendly format. The validating data is specified as volumetric flow rate for pressure differential over the four valve flow paths. However, the data need to be converted to a timeseries in order to compare it in the Data Inspector with the simulation results:

  ```
  %For 11 data points in the look-up table during a ramp from 0 to 100s:
  flowRateData_PB = timeseries(0.0166*[0.58 9.45 20.6 29.9 41.7 55.9 69.3 82.2 94 107 120], lins
  flowRateData_PB.Name = 'Mass Flow Rate PB';

  save validating_data.mat flowRateData_PB
  ```

  The specification sheet data is in lpm. To match the simulation default units, the data set is multiplied by `0.0166` to convert the data to kg/s.

  You can also import timeseries data from an Excel sheet. For more information on formatting data for the Simulation Data Inspector, see `Simulink.sdi.createRun` and “Microsoft Excel Import, Export, and Logging Format”.
- In `sscfluids_4_way_3_position_valve`, the valve performance is modeled by default with tabulated data. Change this to a linear model in the live script at line 26 with the following command:

  ```
  % Setting parameters from Orifice parameterization dropdown
  set_param('sscfluids_4_way_3_position_valve/valve','valve_spec',...
      'fluids.isothermal_liquid.valves_orifices.directional_control_valves.enum.directional_valv
  ```

  This is equivalent to setting the valve **Orifice parameterization** to `Linear - Area vs. spool travel`.
- The test harness is constructed as a unit within a system. To maintain flow through the valve, pumps are connected between the look-up tables and ports.

Collect Simulation Data

The interrogation range is set by the limits of the validating data.

Once the ramp, valve, and 1-D look-up table data and parameters have been set, log the valve data so it can be used in the Simulation Data Inspector. To log the data:

1. In Modeling, select **Model Settings** > **Model Settings**, select Simscape.
2. In the Simscape pane, check **Record data in Simulation Data Inspector**.
3. Check **Open viewer after simulation**.
4. Un-check **Limit data points**.

When the data are recorded in the Simulation Data Inspector, they are saved there under the block name and port. For example, in a test harness saved as sscfluids_4_way_3_position_valve.slx, the mass flow rate in the valve block between ports **P** and **B** is saved as a signal as `simlog_sscfluids_4_way_3_position_valve.valve.orifice_PB.mdot_A`.

When **Record data in Simulation Data Inspector** is left unchecked, the results are saved as a `Series` object under the `simlog` variable. For example, the time series for the mass flow rate through the P-B orifice of the valve block is saved as a series as `simlog_sscfluids_4_way_3_position_valve.valve.orifice_PB.mdot_A.series.values`. For more information, see `simscape.logging.Series`.

After you enable logging, run the live script.

Assess Model Results

1. Import your validating data.
   - Click **Import**. Select **Import From** and navigate to the pressure timeseries data Mass Flow Rate PB. The import pane displays timeseries data currently in your workspace.
   - In the **To** section, ensure that **New run** is selected and that the time series data Mass Flow Rate PB is checked. Click **Import**.
2. In the left-hand pane, click **Compare**.
3. Set **Baseline** to **Signals** > **Mass Flow Rate PB**.
4. Set **Compare to** to **Signals** > **simlog_sscfluids_4_way_3_position_valve.valve.orifice_PB.mdot_A (Run 1: 4_way_3_position_valve)**. This run contains the signal data from when the P-B orifice is in open in the positive and neutral positions.
5. The tolerance of your comparison is the precision to which you can validate your model. Depending on the size of a signal, the Simulation Data Inspector uses either the relative or the absolute tolerance for validation:
   - *Global absolute tolerance* signifies the overall absolute range of acceptable variation between data and a simulation point. This tolerance is applied when the signal is small.
   - *Global relative tolerance* signifies the maximum acceptable difference between data and a simulation point. This tolerance is applied when the signal is large.
   - *Global time tolerance* signifies the range in time one value can be compared to another.

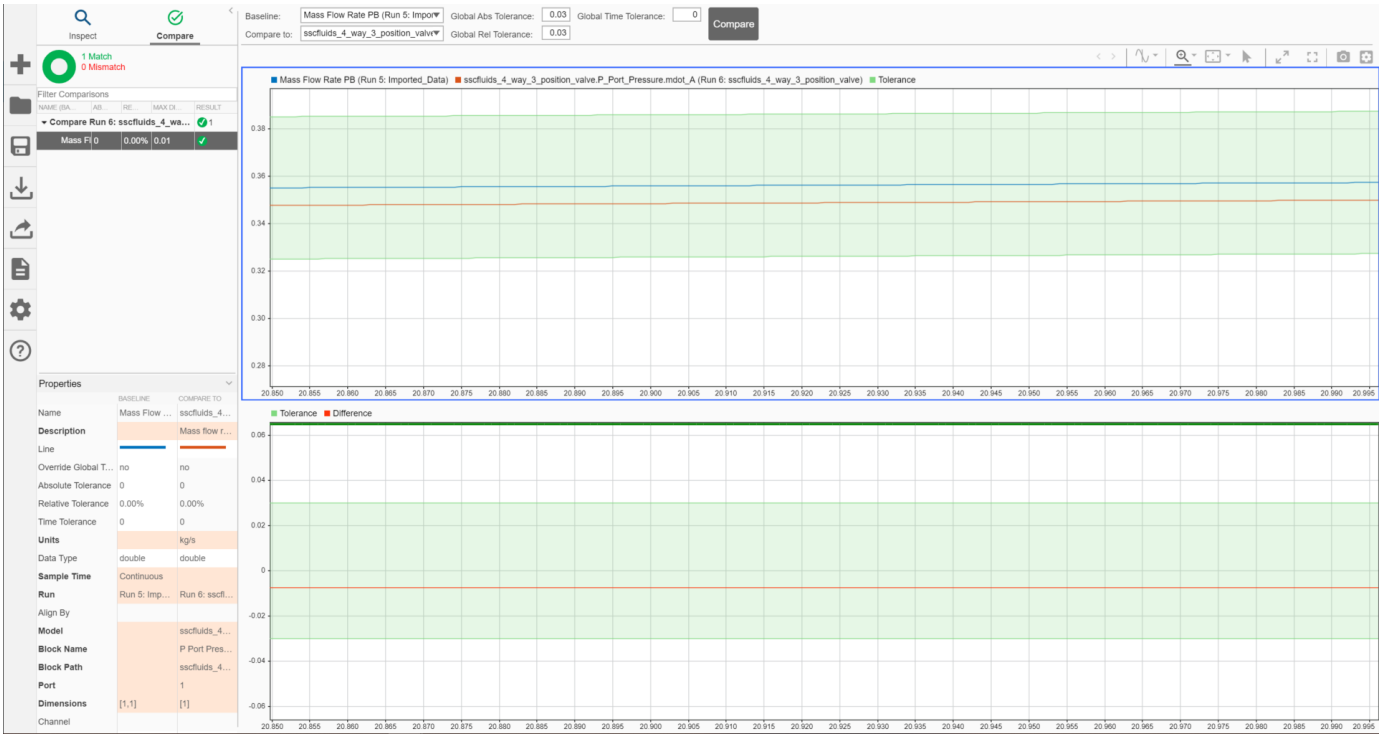
   The absolute and relative tolerances should be equal to or larger than the solver tolerance. In this example, set the absolute and relative tolerances to `0.03` and set the time tolerance to `0`. A nonzero time tolerance value is useful when you expect an impulse response during simulation.
6. Click **Compare**. The simulation and data appear with the tolerance bounding lines:

With global and absolute tolerances of 0.03, the linear model is out of tolerance with the data. When the simulation results are fully within the green band limits, the valve is validated for the tested range of the simulation. The shape of the error suggests that it isn't the most suitable model for the application.

The default parameterization of `sscfluids_4_way_3_position_valve` is the same tabulated data as the specification sheet. To see a model in the Simulation Data Inspector that is fully in tolerance, change the parameterization at line 26 in the live script back to:

```
% Setting parameters from Orifice parameterization dropdown
set_param('sscfluids_4_way_3_position_valve/valve','valve_spec',...
    'fluids.isothermal_liquid.valves_orifices.directional_control_valves.enum.directional_valve_s
```

Run the script again and set **Compare to** to the new run. The error trends shows that this is the correct model for the application, and the results are within tolerance:



However, the slight error offset suggests that some tuning would improve the model accuracy.

See Also

More About

-
- Simulation Data Inspector
- “Creating Custom Components”

How To

Upgrading Simscape Fluids Models Containing Hydraulics (Isothermal) Blocks

The Isothermal Liquid library, which was introduced in R2020a, models mass-flow-rate-based components. This library introduces updates to existing Hydraulics (Isothermal) library blocks, combines similar Hydraulics (Isothermal) library blocks into single blocks, and better aligns with the Thermal Liquid library, which is mass-flow-rate-based.

All blocks in the Isothermal Liquid library account for density as a function of pressure, and pipe and actuator blocks account for compressibility by default. You should expect some differences between a model that uses Hydraulics (Isothermal) library blocks and a converted model when compressibility effects play a significant role due to the treatment of conservation equations in the volumetric-flow-rate-based and mass-flow-rate-based libraries. An example of this is provided in “Conversion When Fluid Dynamic Compressibility is Modeled” on page 4-10.

Multiple models can be converted at the same time. For information on the conversion tool, see `hydraulicToIsothermalLiquid`. For information on upgrading models with blocks from the Simscape Foundation Library, see “Upgrading Hydraulic Models To Use Isothermal Liquid Blocks”.

Special Pre-Conversion Considerations

Physical Signal Blocks

During the conversion process, some blocks in the Hydraulics (Isothermal) library are converted into subsystems that contain physical signal blocks. Physical Signal blocks inserted during conversion do not have the option to assign units of `1`, meaning that the block inherits the units of the incoming signal, as is the case for legacy Physical Signal blocks. If there are physical signal blocks providing input to newly-inserted Physical Signal blocks in your model, a compilation error may occur. To avoid an error, you can change the units from `1` to an appropriate unit in existing physical signal blocks. To propagate units through physical signal blocks, you can check and upgrade your model before conversion with the Upgrade Advisor. See “Upgrading Models with Legacy Physical Signal Blocks” for more information.

Derived Parameters

A derived parameter is a new Isothermal liquid block parameter that is constructed from a Hydraulics (Isothermal) block setting, such as area computed from a user-defined diameter. Parameter conversion may depend on the active Hydraulics (Isothermal) block setting during conversion. If the Isothermal Liquid block setting is changed after conversion, a derived parameter may be inaccurate. This is typically true when converting orifice and valve blocks into their Isothermal Liquid equivalents.

If you change a parameter after the block has been converted, you can check that it has been appropriately updated by running the conversion tool on the Hydraulics (Isothermal) block or model with the new setting and comparing the performance of the two blocks.

For example, in the 2-Way, 3-Way, and 4-Way Directional Valve blocks, switching the setting for **Model parameterization** after conversion may result in different derivations of the spool position at maximum orifice area parameter.

When you change the **Model parameterization** setting to or from `By maximum area and opening` after converting a block, the expression for spool position is not updated. This is due to the

fact that, during conversion, maximum opening is translated into a spool position at maximum opening. In the `By maximum area and opening` parameterization, it is adjusted by the maximum opening value, and in the `Area vs. opening table` parameterization, it is adjusted by the final element of the valve opening vector.

To correct this, manually adjust the expression for spool position at maximum opening if the parameterization changes after conversion, or change the setting in the Isothermal (Hydraulics) library block and run the conversion tool on the block.

Element Order When Using Tabulated Area or Volumetric Flow Rate Parameterizations

In Hydraulics (Isothermal) blocks where parameterization by area-opening or pressure-flow rate is an option,vector elements can be in any order. However, the vector elements in Isothermal Liquid blocks must be entered in a specific order.

When an Isothermal Liquid block is parameterized by opening, such as when the Orifice (IL) block **Orifice parameterization** is set to `Tabulated data - Area vs. control member position`, the **Orifice area vector** elements must be increasing. This parameterization corresponds to the Hydraulics (Isothermal) block setting `By area vs. opening table`.

When an Isothermal Liquid block is parameterized by pressure, such as when the Orifice (IL) block **Orifice parameterization** is set to `Tabulated data - Volumetric flow rate vs. control member position and pressure drop`, the **Volumetric flow rate** table must have increasing rows and increasing or decreasing columns. This parameterization corresponds to the Hydraulics (Isothermal) block setting `By pressure-flow characteristic`.

If the vector elements are not ordered according to the Isothermal Liquid block requirements, you will receive a warning message like the following:

In the Isothermal Liquid library, Orifice area vector must contain monotonically ascending or descending values. Adjustment of Control member position vector and Orifice area vector may be required.

Parameter Comments

If a parameter in your Hydraulics (Isothermal) blocks contain comments, the conversion tool may remove % symbols and any comment content. This happens when a parameter is used in a derived parameter. Comments can be manually added again to a converted block parameter.

Pump and Motor Blocks

Density and kinematic viscosity, which are specified as parameters in the Hydraulics (Isothermal) Fixed-Displacement Motor, Fixed-Displacement Pump, Variable-Displacement Motor, Variable-Displacement Pump, Centrifugal Pump, and Variable-Displacement Pressure-Compensated Pump blocks, are removed as parameters in the associated Isothermal Liquid library blocks. These blocks instead use the network fluid properties. You may need to adjust parameters in the network fluid properties block or adjust the Isothermal Liquid block **Volumetric efficiency at nominal conditions** in order to match the functionality of your Hydraulics (Isothermal) block.

Custom Blocks

The conversion tool will not convert custom blocks with Hydraulics (Isothermal) library ports. These blocks should be connected to converted sections of the model with an Interface (H-IL) block or manually updated.

Upgrading Your Models

The examples below show how to run the conversion tool, respond to warning messages, and compare and check converted models with the Simulation Data Inspector.

Automatic Conversion

This example uses the Diesel Engine In-Line Injection System model, which you can open by executing `sh_diesel_injection` on the MATLAB command line.

Ensure that logging is enabled by clicking on the signal line connected to a Scope and clicking **Log Signals** on the **Simulation** tab.

1. Save the model as `sh_diesel_injection_hydro.slx` to a location where you have write permissions.
2. Set the MATLAB Current Folder to this location. The converted model and the conversion report will also be saved here.
3. Run `sh_diesel_injection_hydro.slx`.
4. On the **Simulation** tab, click **Data Inspector**.
5. To see one of the logged flow rates, expand **Mux:1** and check **Mux:1(1)**. This shows the outlet flow rate in the Injector 1 subsystem.
6. At the command prompt, enter `hydraulicToIsothermalLiquid('sh_diesel_injection_hydro')`.
7. The conversion report, `sh_diesel_injection_hydro_converted.html`, opens when the model conversion is complete. There are three types of warning messages:

   - Predefined fluid has been reparameterized. Behavior change not expected at most temperatures.

     This message indicates that the fluid in the Fuel Properties block in `sh_diesel_injection_hydro` has been redefined for the Isothermal Liquid library, but no specific parameter requires user input.
   - Critical Reynolds number set to 150. Behavior change not expected.

     The parameter that indicates flow regime is updated to the **Critical Reynolds number**. The parameter has a default value of 150. You will not need to adjust anything in the converted model.
   - Original block had Specific heat ratio of 1.4 Set Air polytopic index to this value in an Isothermal Liquid Properties (IL) block or Isothermal Liquid Predefined Properties (IL) block.

     In the Fuel Properties block of the converted model, in the **Entrained Air** tab, set **Air polytropic index** to `1.4`.
8. Run `sh_diesel_injection_hydro_converted`.
9. Open the Simulation Data Inspector and select **Compare** at the top of the left-hand pane. Set the **Baseline** data set to `sh_diesel_injection_hydro`. Set **Compare to** to `sh_diesel_injection_hydro_converted`.
10. Set **Global Abs Tolerance** to `1e-5`. Set **Global Time Tolerance** to `1e-4`.
11. Click **Compare**. The four green check marks next to the four Mux signals show that the results of the models agree within the specified tolerances.

Conversion With Some Manual Adjustments

This example uses a modified dual counterbalance valve model, which you can open by entering `sh_HtoIL_dual_counterbalance_start` at the MATLAB command line.

Data logging for the pump angular velocity and cylinder position is already enabled in the model.

1. Save the model as `sh_cyl_dual_counterbalance_hydro_start` to a location where you have write permissions.
2. Set the MATLAB Current Folder to this location. The converted model and the conversion report will be saved here.
3. Run the model.
4. At the command line, enter `hydraulicToIsothermalLiquid('sh_HtoIL_dual_counterbalance_hydro_start')`.
5. Run the converted model.
6. You will receive the following error: `Invalid use of -. At least one of the operands must be scalar or the operands must be the same size. The units of the operands must be commensurate`. Click the link to the left of this error to see the block in question.

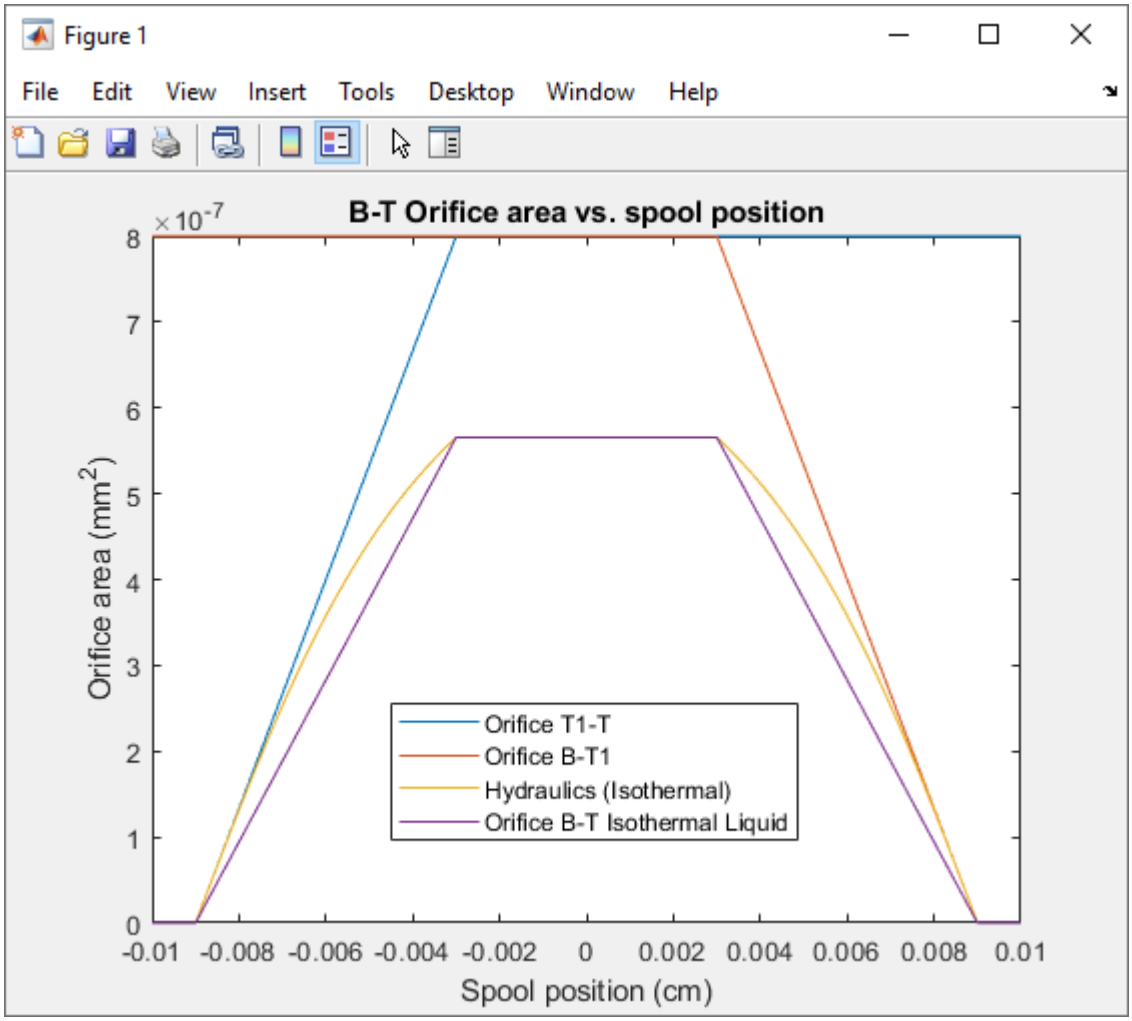
   Go to the Pipe A level of the model to see the PS Subtract inputs.

   Notice that the PS Constant blocks attached to the PS Subtract block have different units: the signal from **el_B** has units of `1`, while the signal from **el_A** has units of `m`. Change the units of B Elevation to `m` and click **OK**.
7. Run the converted model.
8. In the **Simulation** tab, click **Data Inspector** to compare the pump angular velocity and cylinder position between the two models with the Simulation Data Inspector.

   Select **Compare** at the top of the left-hand pane. Set the **Baseline** data set to `sh_HtoIL_dual_counterbalance_hydro_start`. Set **Compare to** to `sh_HtoIL_dual_counterbalance_hydro_start_converted`.
9. Set **Global Abs Tolerance** to `1e-5`. Set **Global Time Tolerance** to `1e-4`.
10. Click **Compare**. Notice that the signals do not agree.
11. Check the conversion report, `sh_HtoIL_dual_counterbalance_hydro_start_converted.html`. The report includes additional warnings from “Automatic Conversion” on page 4-4:

    - B-T Orifice reparameterized.

      The directional valves in the Isothermal Liquid library that open in the neutral spool position are reparameterized from two orifices in series to one orifice between ports. This results in a modification to the orifice area when approaching the orifice maximum area, as shown below. No modification is required for this model.

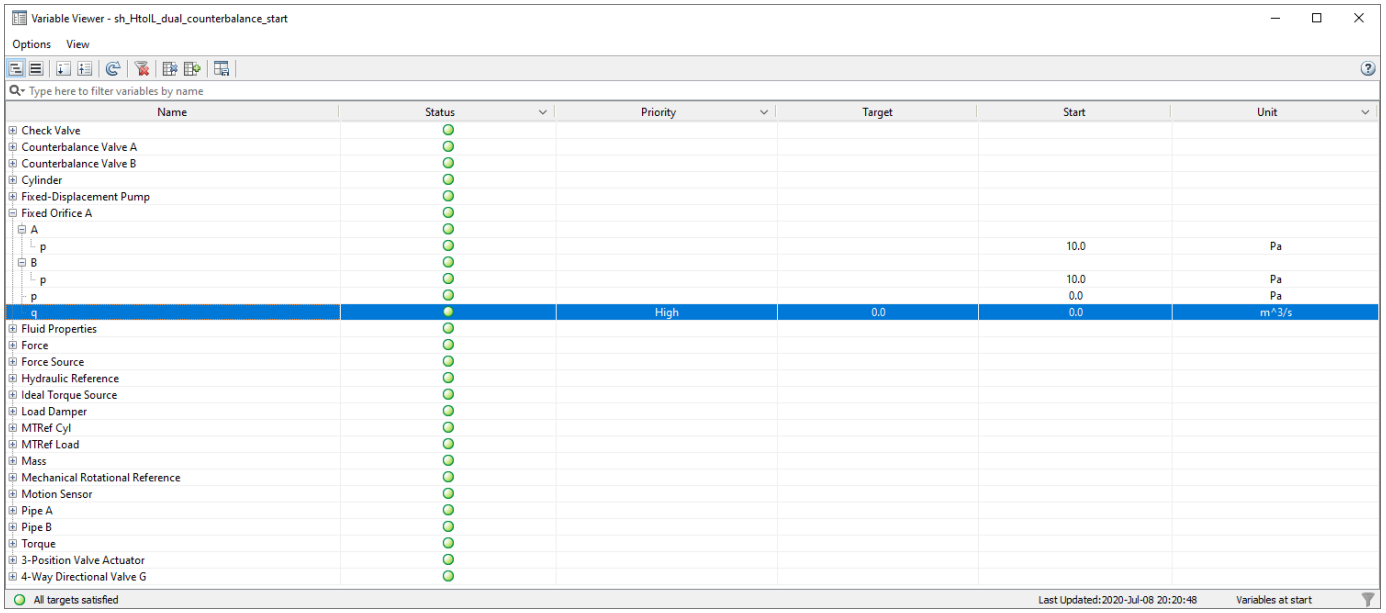


Note that this is true for converted orifices in the 4-Way 3-Position Directional Valve (IL) that open in the neutral spool position. The P-B Orifice, which opens only in the positive and negative spool positions, is not influenced by this modification.

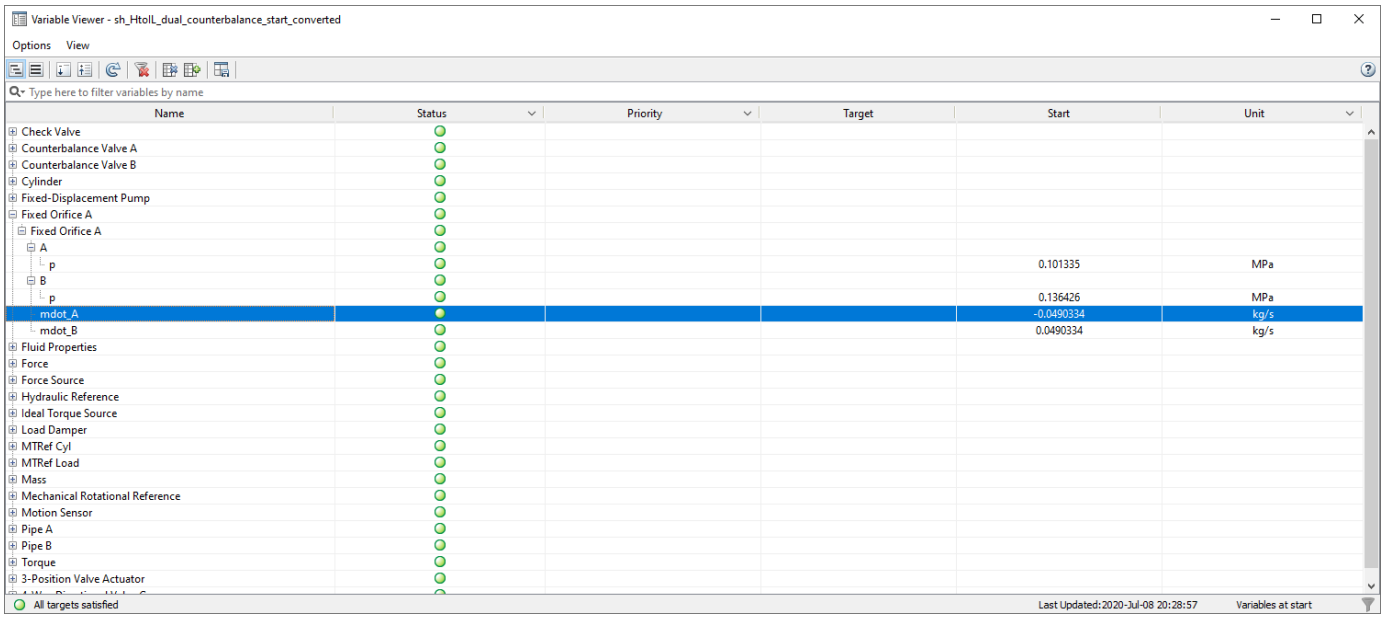
- Beginning value of Flow rate removed. Adjustment of model initial conditions may be required.

  This message indicates that setting the priority and target value of the related block is not an option in the converted model. You can ensure the converted model performs as the original model by setting the priority and target of a variable in an adjacent block.

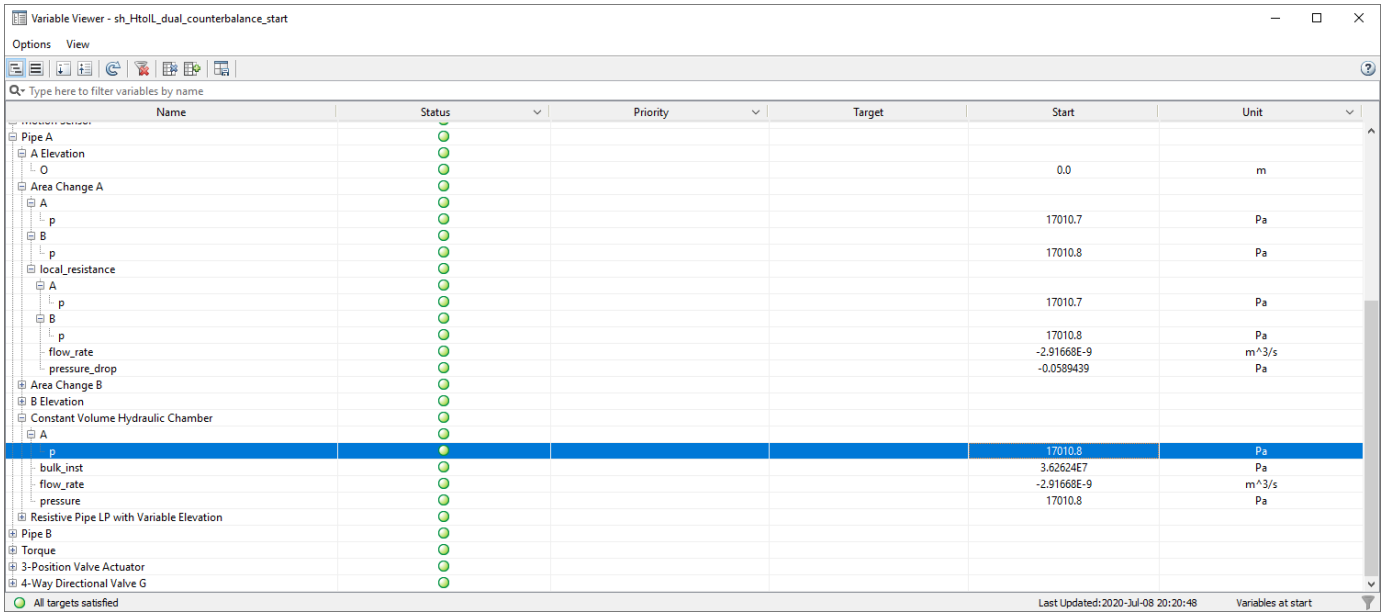
  - In `sh_HtoIL_dual_counterbalance_hydro_start`, check the initial conditions of Fixed Orifice A. In the block dialog, in the **Variables** tab, the flow rate is set to `0 m^3/s` with **Priority** set to `High`.



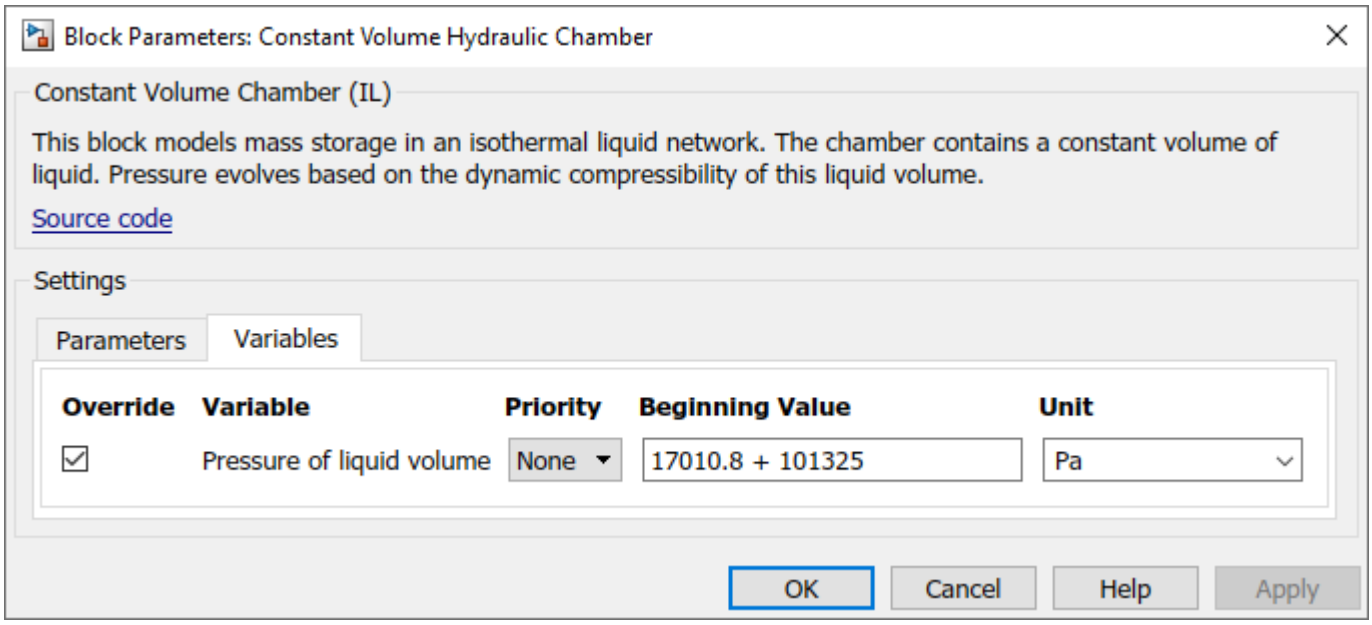
- In `sh_HtoIL_dual_counterbalance_hydro_start_converted`, check the initial conditions of Fixed Orifice A. In the Variable Viewer, the mass flow rate is initialized to a nonzero value.



- The initial conditions can be set in the Constant Volume Hydraulic Chamber block, which is located in the Pipe A subsystem, in the block dialog **Variables** tab. Setting the target and priority of this variable to match the original model should resolve the different starting flow rates in the Fixed Orifice A.
- In `sh_HtoIL_dual_counterbalance_hydro_start`, open the Variable Viewer. The Constant Volume Hydraulic Chamber initialized value is `17010.8 Pa`.



- Set this target in the converted model with high priority. In the Pipe A subsystem of `sh_HtoIL_dual_counterbalance_hydro_start_converted`, in the Constant Volume Hydraulic Chamber dialog on the **Variables** tab, set Pressure of liquid volume **Priority** to `High` and **Beginning value** to `17010.8 + 101325` Pa.



- Pressure losses due to kinetic energy change added. Adjustment of Expansion correction factor and Contraction correction factor may be required.

  If the area change is large, meaning that the pressure change due to the second term and power adjustment are within your model's tolerance, no adjustment is necessary. If the contraction is small, then you can tune the model by adjusting the smaller or both of the port areas. In the Hydraulics (Isothermal) block, the pressure loss is modeled as:

$$\Delta p \approx \frac{K_{PC}\dot{m}^2}{4\rho A_{small}^2}\left(1-\frac{A_{small}}{A_{large}}\right)^{0.75};$$

in the Isothermal Liquid block, the pressure loss is modeled as:

$$\Delta p \approx \frac{K_{contraction}\dot{m}^2}{4\bar{\rho}A_{small}^2}\left(1-\frac{A_{small}}{A_{large}}\right)+\frac{\dot{m}^2}{2\bar{\rho}A_{small}^2}\left(1-\frac{A_{small}}{A_{large}}\right).$$

In this model, no adjustment is required.

- Contraction loss coefficient reformulated. Adjustment of Contraction correction factor may be required.

  The same conditions and corrective actions apply to the **Contraction correction factor** as they do for the **Expansion correction factor**. In this model, no adjustment is required.

- Only elements greater than or equal to 0 retained in Reynolds number vector. Expansion loss coefficient values mapped to these Reynolds numbers. Adjustment of Reynolds number vector, Contraction loss coefficient vector, and Expansion loss coefficient vector may be required.

  The **Reynolds number vector** in the original Area Change B block has both negative and positive Reynolds numbers, `[-6000 -4000 -1000 -200 -50 -30 -20 -10 -1 1 20 40 100 500 2000 5000]`, but the converted Area Change B takes only positive Reynolds numbers. To match the original model behavior, you will need to extend the data set to *Re = 6000* in the base workspace. The commands in the following message will resolve model discrepancies due to both warnings.

- Interpolation method changed to Linear. Additional elements in the Reynolds number vector, Contraction loss coefficient vector, and Expansion loss coefficient vector may be required.

  You can add additional elements to the Reynolds number and loss coefficient vectors in the command line with the `interp1` function.

  In the hydraulics model, the workspace variable for the **Reynolds number vector** parameter is defined as `Re_vec` and the **Loss coefficient vector** parameter as `loss_coeff_vec`. In the MATLAB command window, enter:

  ```
  Re_vec_smooth = -6000:100:6000;
  loss_coeff_vec_smooth = interp1(Re_vec, loss_coeff_vec, Re_vec_new, 'makima', 'extrap');
  ```

  In the Area Change B block dialog, replace `Re_vec` with `Re_vec_smooth` and `loss_coeff_vec` with `loss_coeff_vec_smooth`:

  - Set the **Reynolds number vector** parameter to `Re_vec_smooth(Re_vec_smooth>0)`.
  - Set the **Contraction loss coefficient vector** parameter to `[ interp1( -fliplr(Re_vec_smooth(Re_vec_smooth<0)), fliplr(loss_coeff_vec_smooth(Re_vec_smooth<0)), Re_vec_smooth(Re_vec_smooth>0), 'linear', loss_coeff_vec_smooth(1))]`.
  - Set the **Expansion loss coefficient vector** parameter to `loss_coeff_vec_smooth(Re_vec_smooth>0)`.

  The data set is linearly extrapolated to *Re = 6000* and applies a smooth interpolation method the next time the simulation runs.

- Nominal fluid density and kinematic viscosity removed. Pump uses network fluid properties. Adjustment of Volumetric efficiency at nominal conditions may be required.

In the Hydraulics (Isothermal) library, you can define density and viscosity specifically for the pump or motor block. In the Isothermal Liquid blocks, these parameters are removed and the network properties are used. To modify the functionality of the block due to this change, adjust the **Volumetric efficiency at nominal conditions** parameter in the Fixed-Displacement Pump (IL) block:

- In the Fixed-Displacement Pump (IL) dialog, set the **Volumetric efficiency at nominal conditions** parameter to `0.957`, which is determined using the following formula:

$$\eta_{IL,equiv} = 1 - \frac{\nu_{H,nom}\rho_{H,nom}}{\nu_{IL}\rho_{IL}}(1 - \eta_H),$$

where:

- $\nu_{H,nom}$ is the **Nominal kinematic viscosity** in the Fixed-Displacement Pump block.
- $\rho_{H,nom}$ is the **Nominal fluid density** in the Fixed-Displacement Pump block.
- η_H is the **Volumetric efficiency at nominal conditions** in the Fixed-Displacement Pump block.
- ν_{IL} is the **Kinematic viscosity at atmospheric pressure** in the Isothermal Liquid Properties (IL) block.
- ρ_{IL} is the **Density at atmospheric pressure (no entrained air)** in the Isothermal Liquid Properties (IL) block.

12 Save and run `sh_HtoIL_dual_counterbalance_hydro_converted`.

13 Compare this model with `sh_HtoIL_dual_counterbalance_end`, which includes all post-conversion updates. Run `sh_HtoIL_dual_counterbalance_end` at the command line and inspect the model signals with the Simulation Data Inspector.

Conversion When Fluid Dynamic Compressibility is Modeled

This example uses the orifice model `ssc_HtoIL_orifice`, which you can open by executing on the MATLAB command line.

1 Save the model as `ssc_HtoIL_hydro.mdl` to a location where you have write permissions.

2 Set the MATLAB Current Folder to this location. The converted model and the conversion report will be saved here.

3 Run the model.

4 Enter `hydraulicToIsothermalLiquid('ssc_HtoIL_hydro')`.

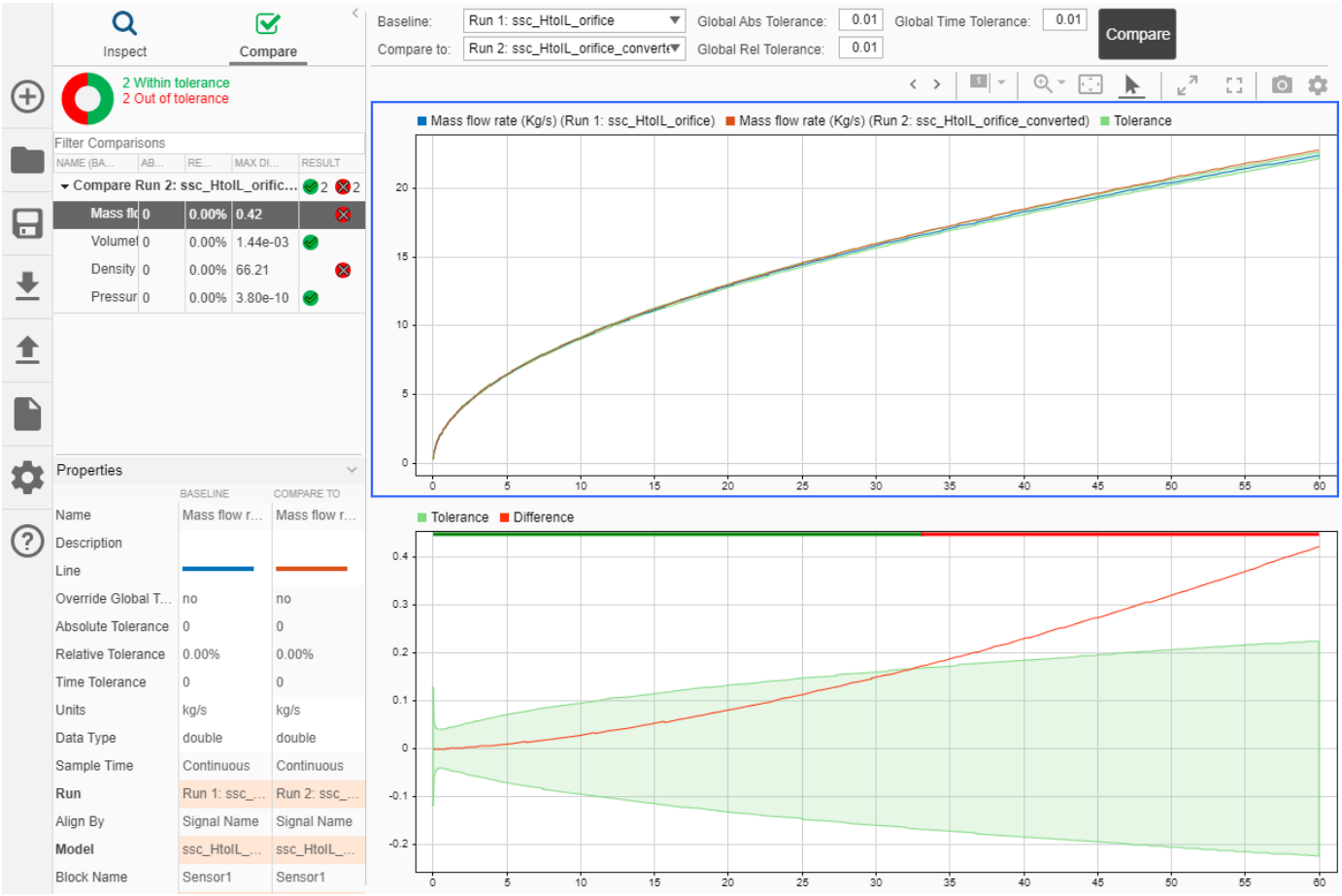
5 Run the converted model.

6 Open the Simulation Data Inspector and select **Compare** at the top of the left-hand pane. Set the **Baseline** data set to `ssc_HtoIL_hydro`. Set **Compare to** to `ssc_HtoIL_hydro_converted`.

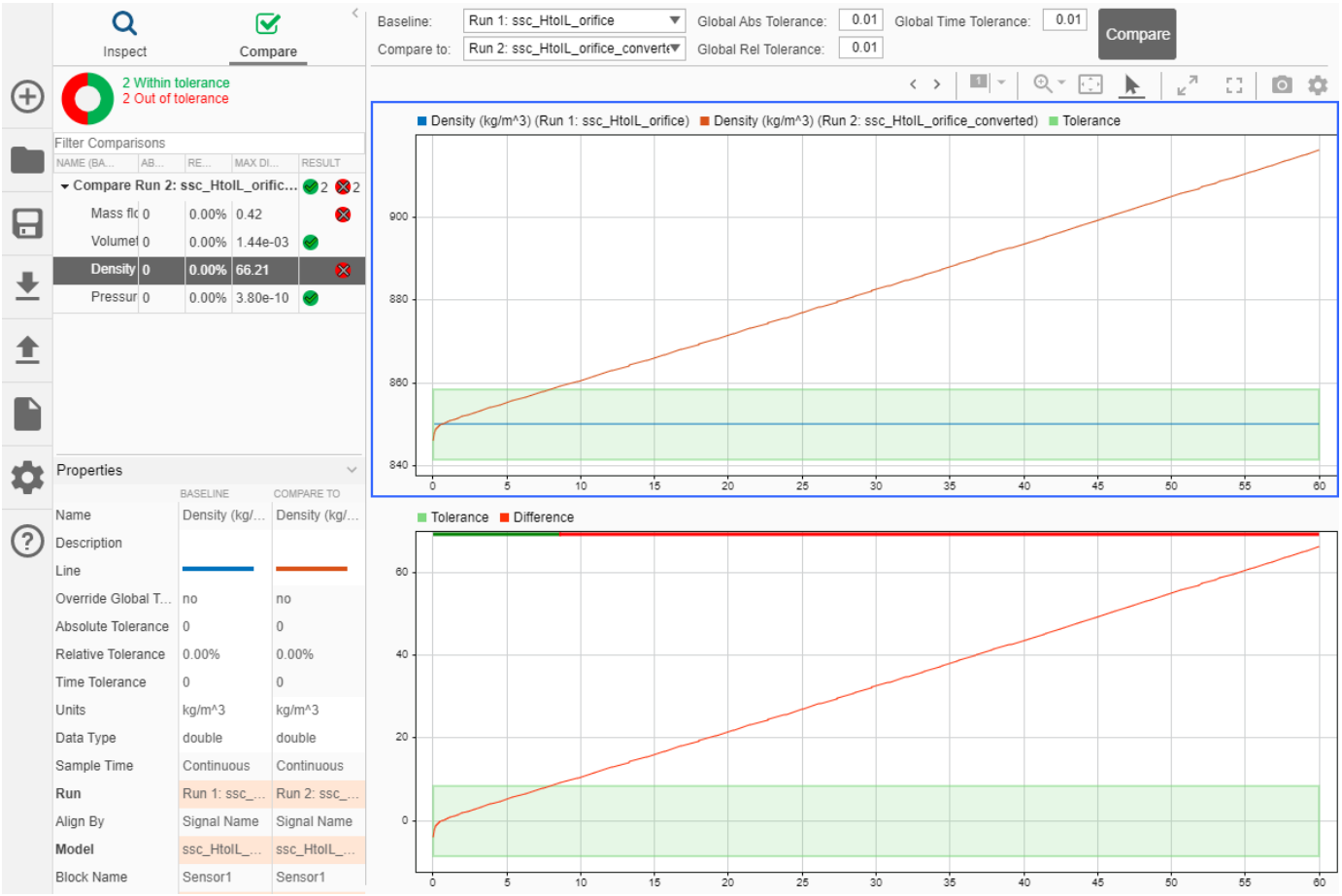
7 Set the **Global Abs Tolerance**, **Global Rel Tolerance**, and **Global Time Tolerance** parameters to `0.01`.

8 Click **Compare**.

As the applied pressure increases, the mass flow rates in the Hydraulics (Isothermal) and Isothermal Liquid models diverge.



This is due to the fact that density is constant in the Hydraulics (Isothermal) block and varies as a function of pressure in the Isothermal Liquid block.



Depending on your application, your converted model may require tuning to account for density changes due to changes in pressure or temperature.

Fixing Post-Conversion Warning Messages

While converting blocks between libraries, you may encounter warnings or errors that will require manual adjustments to your model. Warnings are generated when parameters do not map one-to-one, and only when the model behavior may be affected. Below is a selection of messages you may receive during conversion, and suggested actions for fixing the model.

Due to the different structures of the two isothermal domains, some Isothermal Liquid library blocks are parameterized differently from the equivalent Hydraulics (Isothermal) library blocks. When a parameter has been added, removed, or modified, you receive a notification with the new parameter value or the new means of parameterization. Some properties may be recalculated based on a shift from gauge to absolute pressure, or changes to a specified value, such as reservoir pressure at a specified fluid density. If this is the case, you may receive a message indicating that another parameter may require adjustment. Use the “Variable Viewer” to ensure your model behaves as expected.

In some cases, you can no longer set the priority for some variable initial conditions. If the desired conditions are not met, adjust the initial conditions of other blocks in your model so that they match the initial values in the original model when initial conditions were prioritized for the original block.

For example, if you would like to maintain a specific pressure differential over an orifice that is connected to a valve, adjust the valve mass flow rate conditions assigned to the valve during initialization to achieve the desired pressure differential. Use this method when messages indicate that a beginning or initial value has been removed.

Message	Reason	Suggested Actions
Original block had Specific heat ratio of 1.4. Set Air polytropic index to this value in an Isothermal Liquid Properties (IL) block or Isothermal Liquid Predefined Properties (IL) block.	The specific heat ratio influences fluid compressibility. For isentropic compression, the polytropic index equals the specific heat ratio.	Assign the indicated specific heat ratio in the fluid properties block connected to your network.
Hard-stop model has been reparameterized and uses default parameter values.	The **Hard stop model** used in Hydraulics (Isothermal) library blocks is set to `Stiffness and damping applied smoothly through transition region, damped rebound` in the Isothermal Library blocks. Because this does not correspond directly to the former model, the block hard-stop parameters are set to default values.	Adjust the **Hard stop model**, **Hard stop stiffness coefficient**, **Hard stop damping coefficient**, and **Transition region** parameters in the converted block as needed.
20 degC used to evaluate Density, Isothermal bulk modulus, and Kinematic viscosity.	The Hydraulic Fluid block is converted to an Isothermal Liquid Properties (IL) block. Parameters in the Isothermal Liquid Properties (IL) block are evaluated at 20°C if the original system temperature cannot be determined.	Adjust the **Density**, **Isothermal bulk modulus**, and the **Kinematic viscosity** parameters if your network operates at a different temperature.
The block now models pressure loss due to kinetic energy change. Correction factors have been reformulated to minimize difference in numerical results. Further adjustment of Expansion correction factor and Contraction correction factor may be required.	The Gradual Area Change and Sudden Area Change blocks calculate the hydraulic loss coefficient in terms of area change. The Area Change (IL) block calculates the loss coefficient in terms of area change and mass flow rate.	Further adjustment of the **Expansion correction factor** and **Contraction correction factor** may be required. Refer to the Area Change (IL) and Sudden Area Change block pages to compare changes in the equations.

Message	Reason	Suggested Actions
Power in the contraction loss coefficient has been reformulated from 0.75 to 1. Contraction correction factor has been reformulated to minimize difference in numerical results. Further adjustment of Contraction correction factor may be required.	The power of the equation for calculating the loss coefficient for a sudden area contraction is updated from 0.75 to 1. Compare the equations for K_{SC} in Sudden Area Change and $K_{Contraction}$ in Area Change (IL).	Further adjustment of the **Contraction correction factor** may be required. Refer to the Area Change (IL) and Sudden Area Change block pages to compare changes in the equations.
Warning for minimum fluid level converted to Warning for liquid level below inlet height.	You receive a warning when the fluid level falls below the tank inlet height instead of a minimum fluid level.	Adjust the warning setting or the **Inlet height** to generate a warning at a different fluid level.
Critical Reynolds number set to 150.	The block uses the **Critical Reynolds number** instead of the **Laminar pressure ratio** to identify the transition between laminar and turbulent flow regimes. The default **Critical Reynolds number** is 150.	If the flow through the block is in the fully turbulent or fully laminar regime, this change will not influence performance. If the block experiences transitional flow during simulation, ensure that the **Critical Reynolds number** parameter reflects the correct point of flow transition.
Interpolation or Extrapolation method changed to Linear.	Interpolation and extrapolation methods are no longer user-defined parameters. Interpolation and extrapolation are linear.	If you would like to preserve the 'nearest' method for interpolation or extrapolation, manually enter a vector element next to the 'nearest' element. If you would like to preserve the 'smooth' interpolation method, add additional smoothed elements to the vectors.
Only elements greater than or equal to 0 retained in Reynolds number vector. Expansion loss coefficient values mapped to these Reynolds numbers.	The loss coefficient parameterization due to an area change is updated from table look-up to separate vectors for contraction or expansion, which are applied based on the direction of flow.	Ensure that the **Reynolds number vector** elements associated with the loss coefficient vectors are positive, nonzero, and correspond to the desired data limits. Add additional elements to the parameters to capture losses at a specific Reynolds number.

Message	Reason	Suggested Actions
Transition slot angle and Transition slot maximum area removed due to reparameterization of block region smoothing. Significant behavior change not expected.	The **Transition slot angle** and **Transition slot maximum area** parameters are user-controlled smoothing factors. The Isothermal Liquid library block internally applies a third-order smoothing function.	No action required.
If only non-negative values are provided for the Pressure drop vector, then the block internally extends the table to contain negative Pressure differential and Volumetric flow rate values.	When the block is parameterized by tabulated data, the vector elements are mirrored for negative pressure differential (pressure gain) if no negative elements are provided.	Extend the pressure drop vector and associated volumetric flow rate table manually if you would like to specify the relationship in this region.
Opening time constant is applied to control pressure instead of valve area.	In the Isothermal Liquid library, valves and orifices that have the option to model dynamics apply dynamic modelling to the valve pressure. In the Hydraulics (Isothermal) library, dynamic modelling is applied to the valve area.	Adjust the **Opening time constant** parameter to match the desired opening response.
Valve opening adjustment coefficient for smoothing removed.	The Pressure Reducing 3-Way Valve incorporates smoothing at the extremes of valve opening and closing for numerical robustness. The Pressure-Reducing 3-Way Valve (IL) does not apply smoothing to opening or closing.	You can match the effect of smoothing by adjusting the **Opening time constant** parameter or adjusting the values in the opening area vectors when parameterizing by look-up table.
Updated actuator response when Initial position is Extended. Updated actuator response when Initial position is non-neutral.	The Hydraulics (Isothermal) library block maintains its initial position until the position signal turns off, which triggers the piston to return to neutral. In the Isothermal Liquid library block, the initial position begins to return to neutral at the beginning of the simulation and responds dynamically to the position signal.	Adjust the model initial conditions to match the behavior of the Multiposition Valve Actuator block. Use the Variable Viewer to ensure that the model initial conditions are correct.

Message	Reason	Suggested Actions
The area at port B is now calculated as the sum of the area at ports X and Y minus the area at port A. Formerly, it was the difference between the areas at port X and port A.	The parameterization of the Hydraulic 4-Port Cartridge Valve Actuator block differs from the Cartridge Valve Actuator (IL) block.	Adjust the **Port A poppet area**, **Port A poppet to port X pilot area ratio**, and/or the **Port Y pilot area** parameters according to the force balance on Cartridge Valve Actuator (IL) and Hydraulic 4-Port Cartridge Valve Actuator if any difference in the preload (poppet) force is observed.
Converted subsystem assumes input and output signals have units of 1.	There is no equivalent Isothermal Liquid library block to the Proportional and Servo-Valve Actuator block. The block is converted to a subsystem of physical signal blocks that maintain the original block functionality.	Convert the subsystem block input and output signal units to 1 if the input and output signals of the Proportional and Servo-Valve Actuator block specify any other units.
New parameters Minimum volumetric efficiency and Minimum mechanical efficiency set to 1e-3. Smaller parameter values may be required to avoid unintended efficiency saturations.	New parameters in the Variable-Displacement Motor (IL) block are set to the block defaults.	You many need to adjust the block defaults to meet your model requirements.
New parameters Pressure drop threshold for motor-pump transition set to 10 rad/s, Angular velocity threshold for motor-pump transition set to 10 rad/s, and Displacement threshold for motor-pump transition set to 0.1 cm^3/rev. Parameter adjustment may be required to match original Power threshold of <5> W.	New parameters in the Variable-Displacement Motor (IL) block are set to the block defaults.	You many need to adjust the block defaults to meet your model requirements.

Block Conversions

Below is a list of Hydraulics (Isothermal) library blocks and their associated Isothermal Liquid library block equivalents. To match the Hydraulics (Isothermal) block configuration, some converted models may include a subsystem with additional Simscape Fluids or Simscape blocks. Some blocks in the Hydraulics (Isothermal) library do not have an equivalent Isothermal Liquid library block. In this case, the functionality is reconstructed from a collection of Simscape and Simscape Fluids blocks.

For more information on conversion between Hydraulics (Isothermal) and Isothermal Liquid blocks in the Simscape Foundation Library, see “Upgrading Your Models”.

Accumulators: Block Substitution

Hydraulics (Isothermal) Block	Isothermal Liquid Block
Gas-Charged Accumulator	Gas-Charged Accumulator (IL)
Spring-Loaded Accumulator	Spring-Loaded Accumulator (IL) Hard-stop damping is not modeled in the isothermal liquid block.

Hydraulic Cylinders: Block Substitution

Hydraulics (Isothermal) Block	Isothermal Liquid Block
Centrifugal Force in Rotating Cylinder	Rotating Cylinder Force (IL) The **Fluid density** parameter is replaced with port **X**, which senses the network density.
Cylinder Cushion	Cylinder Cushion (IL) The new block is parameterized in terms of plunger diameter, area, and length. The Hydraulics (Isothermal) block is parameterized in terms of tabulated area and displacement. The plunger length, or the orifice control member travel, is calculated from the Hydraulics (Isothermal) block as: opening(end) - opening(1).
Cylinder Friction	Cylinder Friction (IL) The Cylinder Friction (IL) block does not assign beginning values to variables.
Single-Acting Rotary Actuator	Single-Acting Rotary (IL) The block is converted to a subsystem. If leakage is modeled in the Hydraulics (Isothermal) block, port **A** connects to port **A** of a Laminar Leakage (IL) block, which is connected to a Reservoir (IL) block at atmospheric pressure.
Double-Acting Rotary Actuator	Double-Acting Rotary Actuator (IL) The block is converted to a subsystem. If leakage is modeled in the Hydraulics (Isothermal) block, ports **A** and **B** connect in parallel to ports **A** and **B**, respectively, of a Laminar Leakage (IL) block.
Single-Acting Hydraulic Cylinder	Single-Acting Actuator (IL)
Single-Acting Hydraulic Cylinder (Simple)	Single-Acting Actuator (IL) The **Hard stop model** is set to `Stiffness and damping applied smoothly through transition region, damped rebound`, with default hard stop parameters.
Double-Acting Hydraulic Cylinder	Double-Acting Actuator (IL)
Double-Acting Hydraulic Cylinder (Simple)	Double-Acting Actuator (IL) The **Hard stop model** is set to `Stiffness and damping applied smoothly through transition region, damped rebound`, with default hard stop parameters.

Utilities: Block Substitution

Hydraulics (Isothermal) Block	Isothermal Liquid Block
Hydraulic Fluid	If **Hydraulic fluid** is set to `Water`, `Water-glycol 60/40`, `Diesel fuel`, or `Jet fuel`, the Hydraulic Fluid block is converted to an Isothermal Liquid Predefined Properties (IL) block. For all other fluids, the Hydraulic Fluid block is converted to an Isothermal Liquid Properties (IL) block. If the conversion tool cannot assess the fluid temperature definition, the properties are defined for 20°C.
Reservoir	Reservoir (IL), Simscape Foundation Library.

Hydraulic Resistances: Block Substitution

Hydraulics (Isothermal) Block	Isothermal Liquid Block
Elbow	Elbow (IL)
Gradual Area Change	Area Change (IL) The block is converted into a subsystem to maintain the original port orientation.
Local Resistance	Local Resistance (IL)
Pipe Bend	Pipe Bend (IL) The **Initial liquid pressure** parameter is converted from gauge to absolute pressure. The **Critical Reynolds number** parameter is converted into two internally fixed threshold Reynolds numbers. 2000 indicates a fully laminar flow and 4000 indicates a fully turbulent flow.
Sudden Area Change	Area Change (IL)
T-Junction	This block is converted to a subsystem with Local Resistance (IL) blocks.

Low Pressure Blocks: Block Substitution

Hydraulics (Isothermal) Block	Isothermal Liquid Block
Hydraulic Pipe LP Resistive Pipe LP Resistive Pipe LP with Variable Elevation Segmented Pipe LP	Pipe (IL)
Hydraulic Pipe LP with Variable Elevation	Pipe (IL) Port **EL** is exposed for elevation change as a physical signal.
Partially Filled Vertical Pipe LP	Partially Filled Pipe (IL) The Partially Filled Pipe (IL) block receives a liquid level instead of a liquid volume at its physical signal port. The block divides the original fluid volume by a constant tank cross-section within a subsystem. You will likely need to modify the assumed tank cross-section.
Constant-Head Tank	This block is converted to a subsystem of Reservoir (IL), Orifice (IL), PS Integrator, and Flow Rate Sensor blocks.
Tank	Tank (IL)

Orifices: Block Substitution

Hydraulics (Isothermal) Block	Isothermal Liquid Block
Annular Orifice	Annular Leakage (IL)
Fixed Orifice Fixed Orifice With Fluid Inertia Fixed Orifice Empirical Variable Orifice	Orifice (IL)
Journal Bearing Pressure-Fed	The block is converted to a subsystem with two Annular Leakage (IL) blocks.
Orifice With Variable Area Round Holes Orifice With Variable Area Slot	Spool Orifice (IL)
Variable Orifice Between Round Holes	Variable Overlapping Orifice (IL)

Pipes: Block Substitution

Hydraulics (Isothermal) Block	Isothermal Liquid Block
Rotating Pipe	Rotating Channel (IL)
Hydraulic Pipeline	Pipe (IL)
Segmented Pipeline	Pipe (IL)

Pumps and Motors: Block Substitution

Hydraulics (Isothermal) Block	Isothermal Liquid Block
Angle Sensor	The block is converted to a subsystem containing an Ideal Rotational Motion Sensor.
Centrifugal Pump	Centrifugal Pump (IL)
Fixed-Displacement Motor	Fixed-Displacement Motor (IL) The Fixed-Displacement Motor (IL) block uses network fluid properties. The **Volumetric efficiency at nominal conditions** parameter may require adjustment to match the Hydraulics (Isothermal) block functionality.
Fixed-Displacement Pump	Fixed-Displacement Pump (IL) The Fixed-Displacement Pump (IL) block uses network fluid properties. The **Volumetric efficiency at nominal conditions** parameter may require adjustment to match the Hydraulics (Isothermal) block functionality.
Jet Pump	Jet Pump (IL)
Porting Plate Variable Orifice	Valve Plate Orifice (IL)
Swash Plate	Swash Plate
Variable-Displacement Motor	Variable Displacement Motor (IL) The Variable Displacement Motor (IL) block uses network fluid properties. The **Volumetric efficiency at nominal conditions** may require adjustment to match the Hydraulics (Isothermal) block functionality.
Variable-Displacement Pressure-Compensated Pump	Pressure-Compensated Pump (IL) The block is converted to a subsystem. The Pressure-Compensated Pump (IL) block uses network fluid properties. The **Volumetric efficiency at nominal conditions** may require adjustment to match the Hydraulics (Isothermal) block functionality.
Variable-Displacement Pump	Variable Displacement Pump (IL) The Variable Displacement Pump (IL) block uses network fluid properties. The **Volumetric efficiency at nominal conditions** may require adjustment to match the Hydraulics (Isothermal) block functionality.

Directional Valves: Block Substitution

Hydraulics (Isothermal) Block	Isothermal Liquid Block
2-Way Directional Valve	2-Way Directional Valve (IL) The 2-Way Directional Valve (IL) block is based on the spool position at the maximum orifice area and the maximum spool travel distance, not the initial spool position. Changing the valve parameterization after conversion can lead to inconsistencies. See “Derived Parameters” on page 4-2.
3-Way Directional Valve	3-Way Directional Valve (IL) The 3-Way Directional Valve (IL) block is based on the spool position at the maximum orifice area and the maximum spool travel distance, not the initial spool position. Changing the valve parameterization after conversion can lead to inconsistencies. See “Derived Parameters” on page 4-2.
4-Way Ideal Valve	4-Way 3-Position Directional Valve (IL) The 4-Way 3-Position Directional Valve (IL) block is based on the spool position at the maximum orifice area and the maximum spool travel distance, not the initial spool position. Changing the valve parameterization after conversion can lead to inconsistencies. See “Derived Parameters” on page 4-2.
4-Way Directional Valve	4-Way 3-Position Directional Valve (IL) The 4-Way 3-Position Directional Valve (IL) block is based on the spool position at the maximum orifice area and the maximum spool travel distance, not the initial spool position. Changing the valve parameterization after conversion can lead to inconsistencies. See “Derived Parameters” on page 4-2.
4-Way Directional Valves A-K	4-Way 3-Position Directional Valve (IL) The 4-Way 3-Position Directional Valve (IL) block is based on the spool position at the maximum orifice area and the maximum spool travel distance, not the initial spool position. Changing the valve parameterization after conversion can lead to inconsistencies. See “Derived Parameters” on page 4-2.
Cartridge Valve Insert Cartridge Valve Insert With Conical Seat	Cartridge Valve Insert (IL)
Check Valve	Check Valve (IL) When dynamics are enabled, the valve area is impacted in the Hydraulics (Isothermal) library block and the valve pressure in the Isothermal Liquid library block.
Hydraulically Operated Remote control Valve	Pressure Compensator Valve (IL)

Hydraulics (Isothermal) Block	Isothermal Liquid Block
Pilot-Operated Check Valve	Pilot-Operated Check Valve (IL)
Shuttle Valve	Shuttle Valve (IL)

Flow Control Valves: Block Substitution

Hydraulics (Isothermal) Block	Isothermal Liquid Block
Ball Valve	Poppet Valve (IL)
Counterbalance Valve	Counterbalance Valve (IL)
Flow Divider	The block is converted to a subsystem.
Flow Divider-Combiner	The block is converted to a subsystem.
Gate Valve	Variable Overlapping Orifice (IL)
Needle Valve	Needle Valve (IL)
Poppet Valve	Poppet Valve (IL)
Pressure-Compensated 3-Way Flow Control Valve	Pressure-Compensated 3-Way Flow Control Valve (IL)
Pressure-Compensated Flow Control Valve	Pressure-Compensated Flow Control Valve (IL)

Pressure Control Valves: Block Substitution

Hydraulics (Isothermal) Block	Isothermal Liquid Block
Pressure Compensator	Pressure Compensator Valve (IL) When dynamics are enabled, the valve area is impacted in the Hydraulics (Isothermal) library block and the valve pressure in the Isothermal Liquid library block.
Pressure Reducing Valve	Pressure-Reducing Valve (IL) When dynamics are enabled, the valve area is impacted in the Hydraulics (Isothermal) library block and the valve pressure in the Isothermal Liquid library block.
Pressure Relief Valve	Pressure Relief Valve (IL) When dynamics are enabled, the valve area is impacted in the Hydraulics (Isothermal) library block and the valve pressure in the Isothermal Liquid library block.
Pressure Reducing 3-way Valve	Pressure-Reducing 3-Way Valve (IL) When dynamics are enabled, the valve area is impacted in the Hydraulics (Isothermal) library block and the valve pressure in the Isothermal Liquid library block.

Valve Actuators: Block Substitution

Hydraulics (Isothermal) Block	Isothermal Liquid Block
2-Position Valve Actuator 3-Position Valve Actuator	Multiposition Valve Actuator
Double-Acting Servo Cylinder	Double-Acting Servo Valve Actuator (IL) The **Hard stop model** is set to `Stiffness and damping applied smoothly through transition region, damped rebound`, with default hard stop parameters.
Hydraulic 4-Port Cartridge Valve Actuator	Cartridge Valve Actuator (IL) The block is converted to a subsystem. The area at port **B** is now calculated as the sum of the area at ports **X** and **Y** minus the area at port **A**. Formerly, it was the difference between the areas at port **X** and port **A**.
Hydraulic Cartridge Valve Actuator	Cartridge Valve Actuator (IL)
Hydraulic Single-Acting Valve Actuator Hydraulic Double-Acting Valve Actuator	Pilot Valve Actuator (IL)
Proportional and Servo-Valve Actuator	The block is converted into a susbsytem of physical signal blocks.
Valve Actuator	Proportional Valve Actuator

Valve Forces: Block Substitution

Hydraulics (Isothermal) Block	Isothermal Liquid Block
Spool Orifice Hydraulic Force	Spool Orifice Flow Force (IL)
Valve Hydraulic Force	The block is converted to a subsystem.

Fluid Network Interfaces Library: Block Substitution

Interface (TL-IL)	Interface (TL-IL) The additional setting `Thermal Liquid (TL) - Isothermal Liquid (IL)` is added and selected for the **Fluids domain interface** parameter. Note that in R2020a and R2020b, this block is named Interface.
Double-Acting Actuator (H-G)	Double-Acting Actuator (G-IL) This block is converted to a subsystem.

SimHydraulics Legacy Library: Block Substitution

Variable-Displacement Hydraulic Machine	Variable-Displacement Motor (IL) This block is converted to a subsystem. The Variable-Displacement Motor (IL) **Leakage and friction paramerization** is set to `Input signal - volumetric and mechanical losses`.
Variable-Displacement Hydraulic Machine (External Efficiencies)	Variable-Displacement Motor (IL) This block is converted to a subsystem. The Variable-Displacement Motor (IL) **Leakage and friction paramerization** is set to `Input signal - volumetric and mechanical efficiencies`.

See Also

`hydraulicToIsothermalLiquid` | Interface (TL-IL) | Interface (H-IL)

More About

- “Upgrading Hydraulic Models To Use Isothermal Liquid Blocks”
- “Modeling Isothermal Liquid Systems”

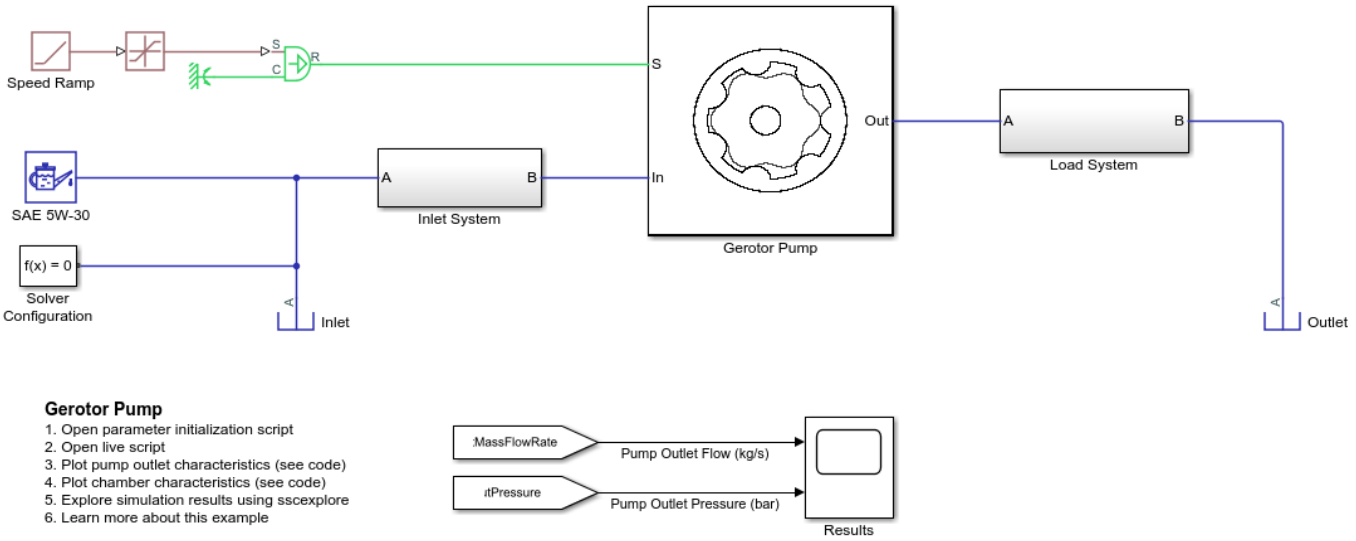
Simscape Fluids Examples

Gerotor Pump

This example shows how to model, parameterize, and test a gerotor pump.

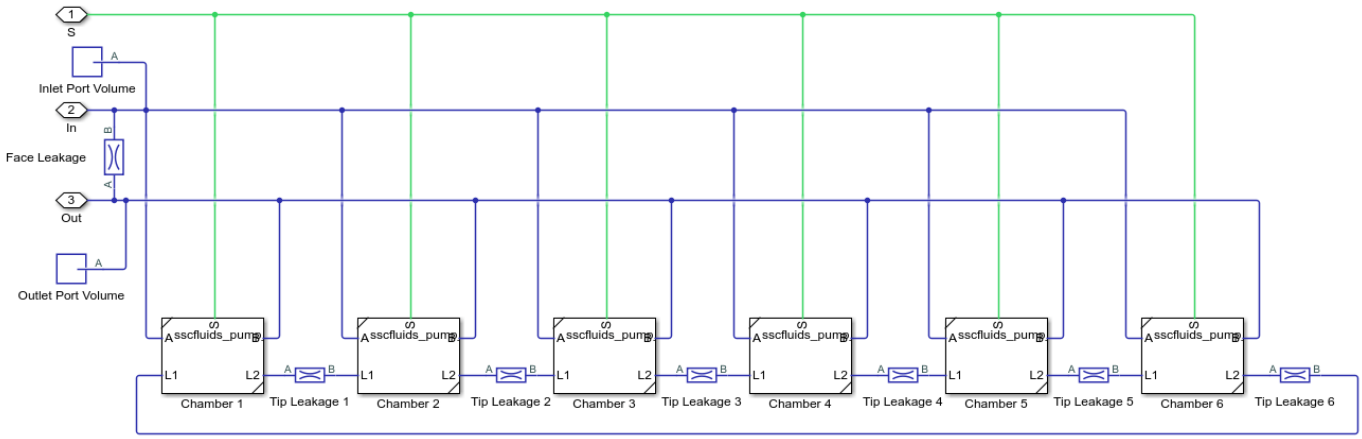
Model

The following figure shows a model of the gerotor pump with a lubrication system. The flow circuit consists of an inlet system upstream of the pump, the gerotor pump and a load system downstream of the pump.



Gerotor Pump Subsystem

This subsystem shows how the gerotor pump is modeled. The gerotor pump comprises of an outer gear (driven gear) and an inner gear (driving gear) mounted on a rotating shaft, modeled as an angular velocity source. For the pump modeled here, the inner gear teeth profile is based on a trochoid curve geometry, and the outer gear teeth profile is a conjugate of the inner gear teeth profile. Gerotor pump is a positive displacement type pump, as the volumetric flow is effectively produced by the continual suction and delivery actions performed by the cyclical variation of the volume between the teeth of the mating gear set. This liquid volume between the mating gears can be divided into a number of sub-volumes or chambers which are connected with the adjacent ones through a gap due to teeth (or tip) clearance leading to tip leakage between chambers. The inlet and outlet port volumes are connected through a leakage flow called face leakage resulting from the clearance between the gear faces and housing inner faces. The volumes of the chambers change as the pump rotates and they are connected to the inlet port volume and outlet port volume through port area profiles. The number of variable volume liquid chambers equal the number of outer gear teeth.

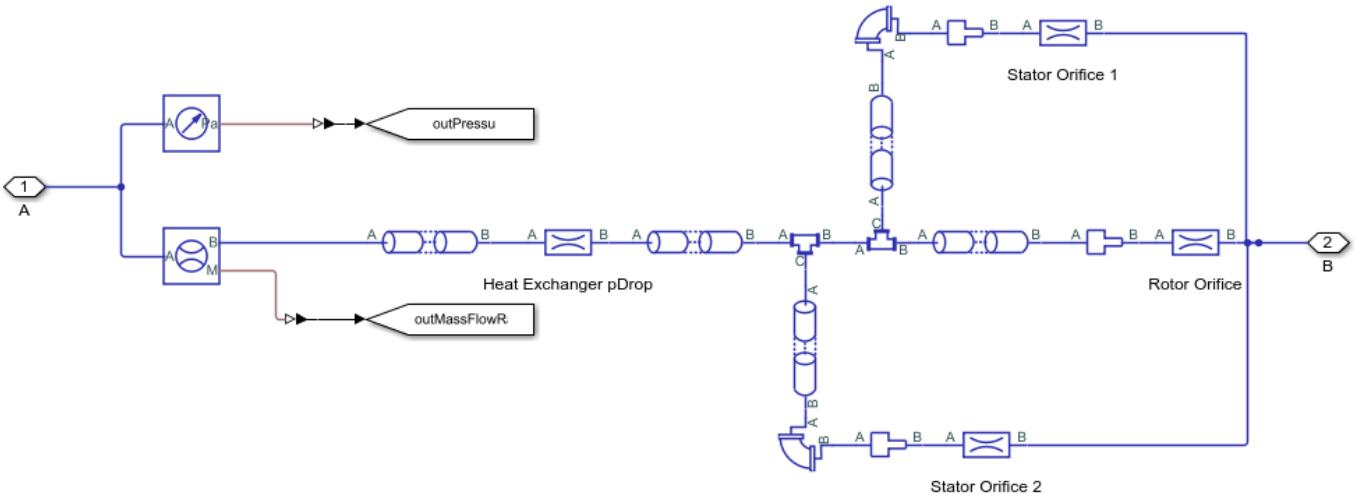


Inlet System Subsystem

This subsystem shows a representation of a inlet system or suction system upstream of a gerotor pump. The inlet system is modeled using a combination of pipes, bends, sudden area changes and an orifice representing a pressure drop across a filter. These components contribute to pressure drop upstream of the pump.

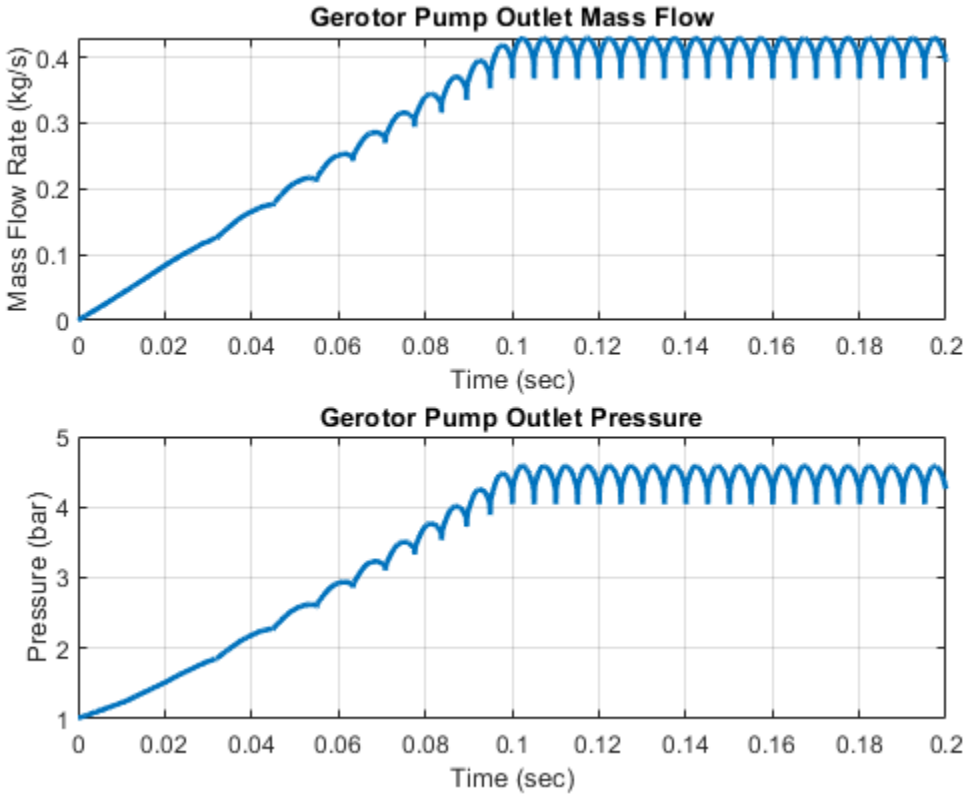
Load System Subsystem

This subsystem shows a representation of a lubrication and cooling system in an automotive where a motor is lubricated and cooled by the delivery flow of a gerotor pump. The load system is modeled using a combination of pipes, bends, sudden area changes, and orifices representing the pressure drop across a heat exchanger and orifices at entry to stator and rotor. All components contribute to load system pressure drop. The thermal effects are not considered in this model as it is purely isothermal.



Simulation Results from Simscape™ Logging

This model generates a plot of the mass flow rate and pressure at the outlet of the gerotor pump versus time.

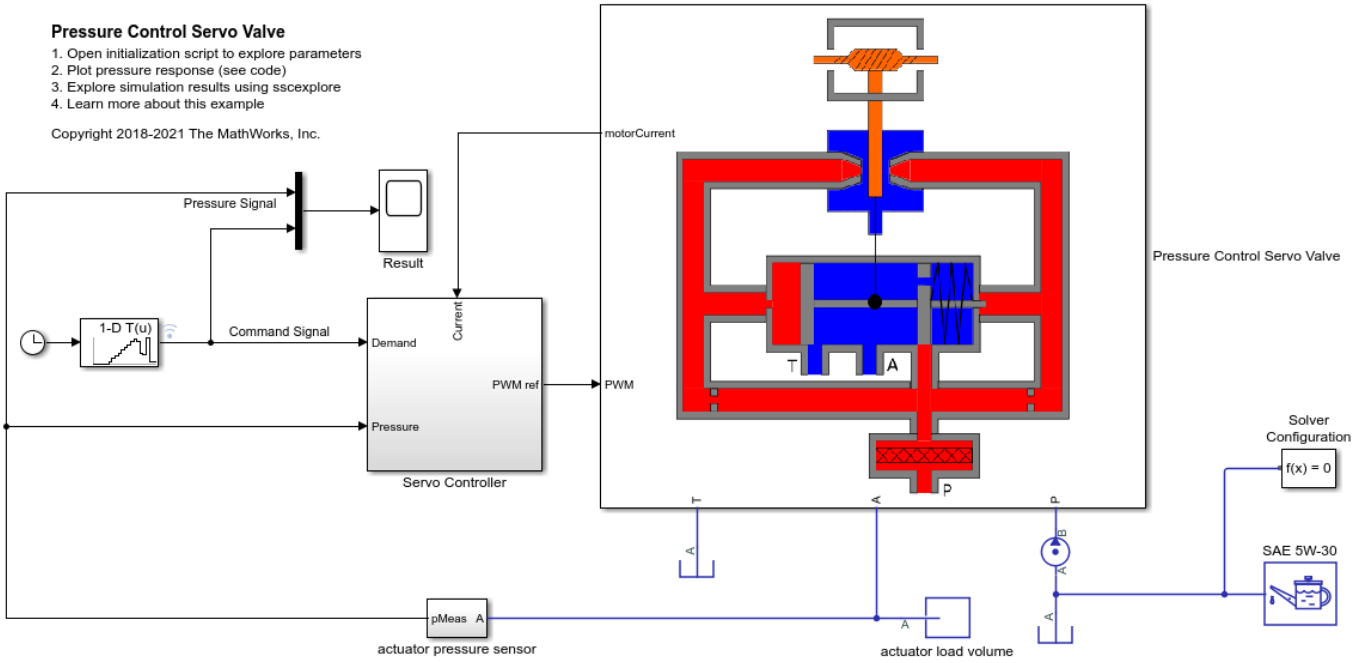


Pressure Control Servo Valve

This example shows how to model, parameterize, and test a pressure control servo valve with a closed loop control. This example also generates a comparison plot between the commanded and the achieved pressure in the actuator with respect to the time. This valve provides a precise and fast control of the pressure in the actuator with a very small electrical signal to the torque motor. These valves are used in the aerospace industry and the automotive industry for the safety critical applications.

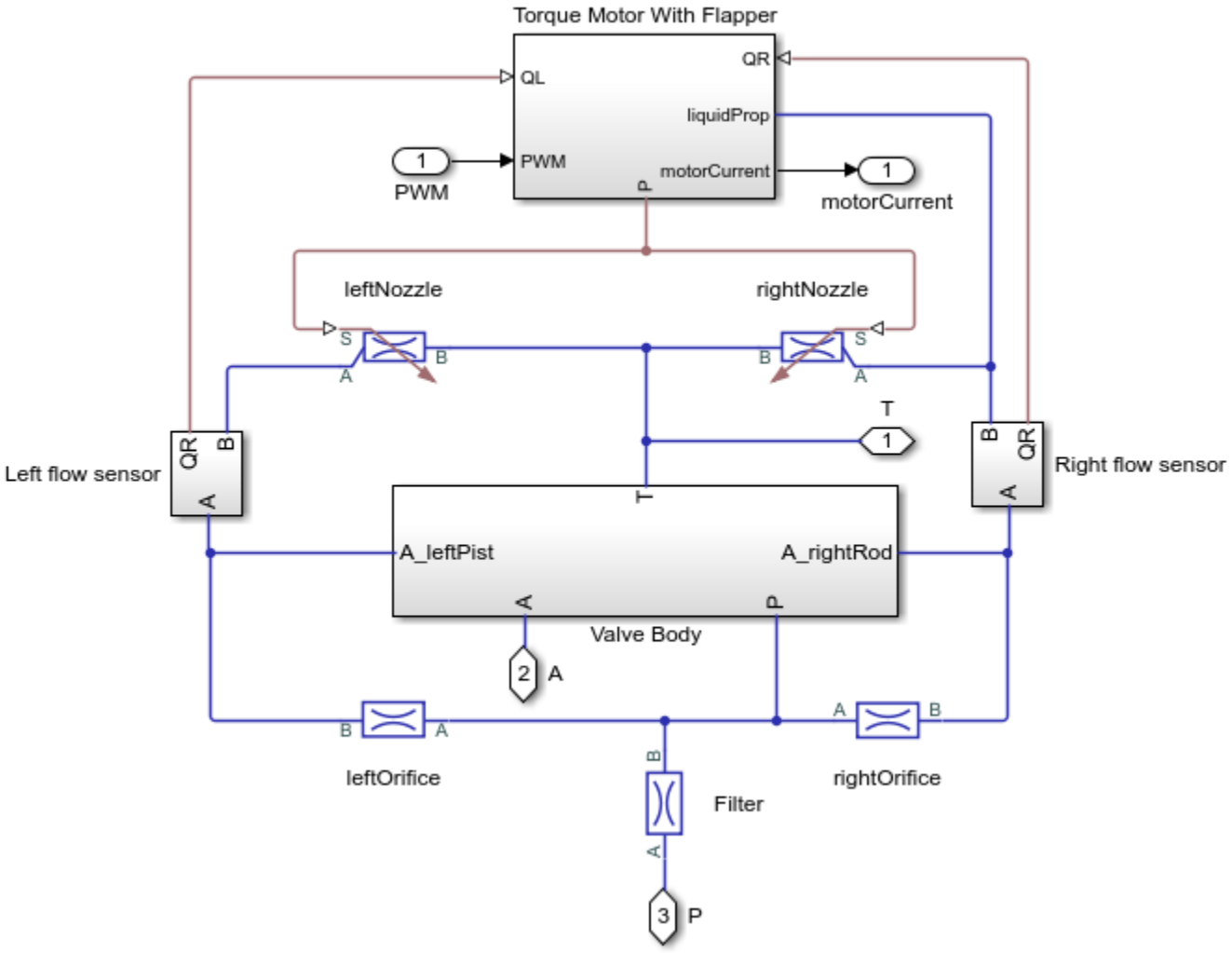
Model

The following figure shows the model of a pressure control servo valve. Here P represents the pressure source port, T represents the reservoir, and A represents the actuator port.



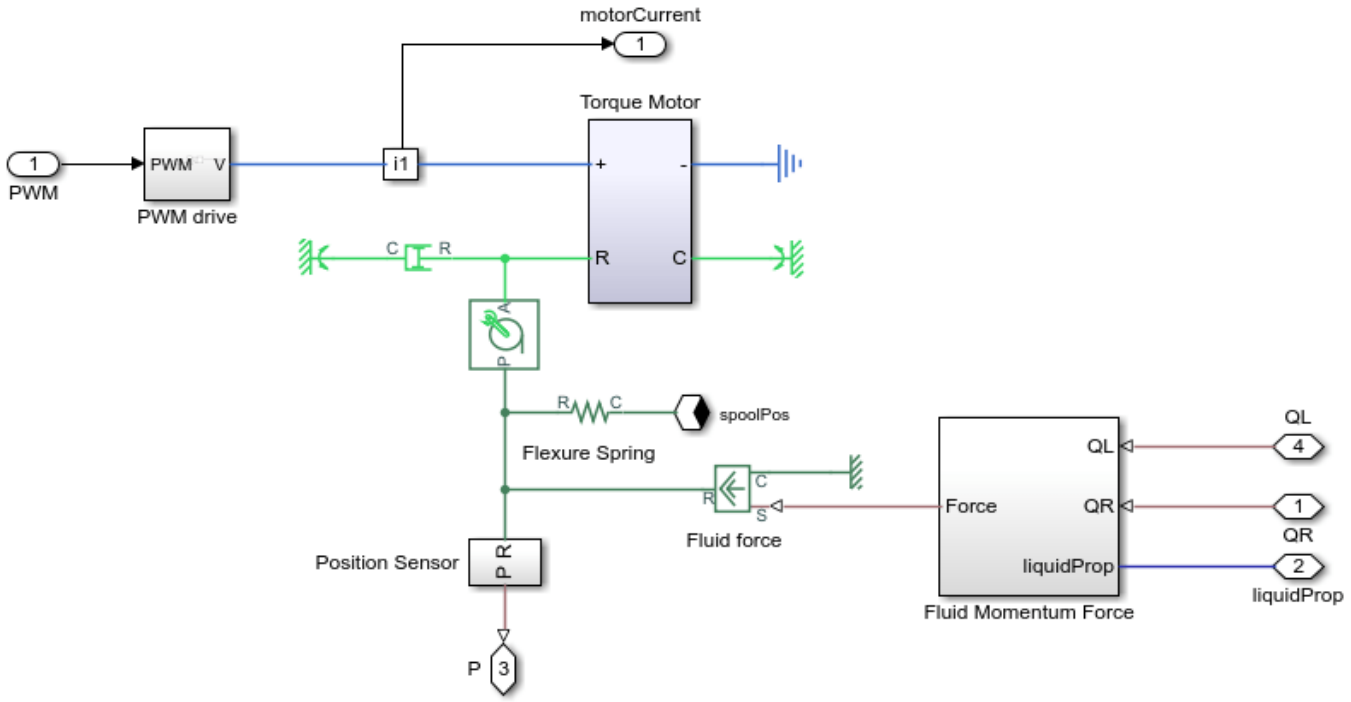
Pressure Control Servo Valve Subsystem

This subsystem shows how the pressure control servo valve is modeled. This subsystem includes modeling of the flapper nozzle amplifier circuit , the torque motor, and the valve body. The flapper nozzle amplifier circuit includes modeling of the left and the right side nozzle around the flapper, the filter, and the left and right side fixed orifices. The flapper nozzle amplifier circuit controls the pressure on the spool's left surface and the spool's right side extended rod.



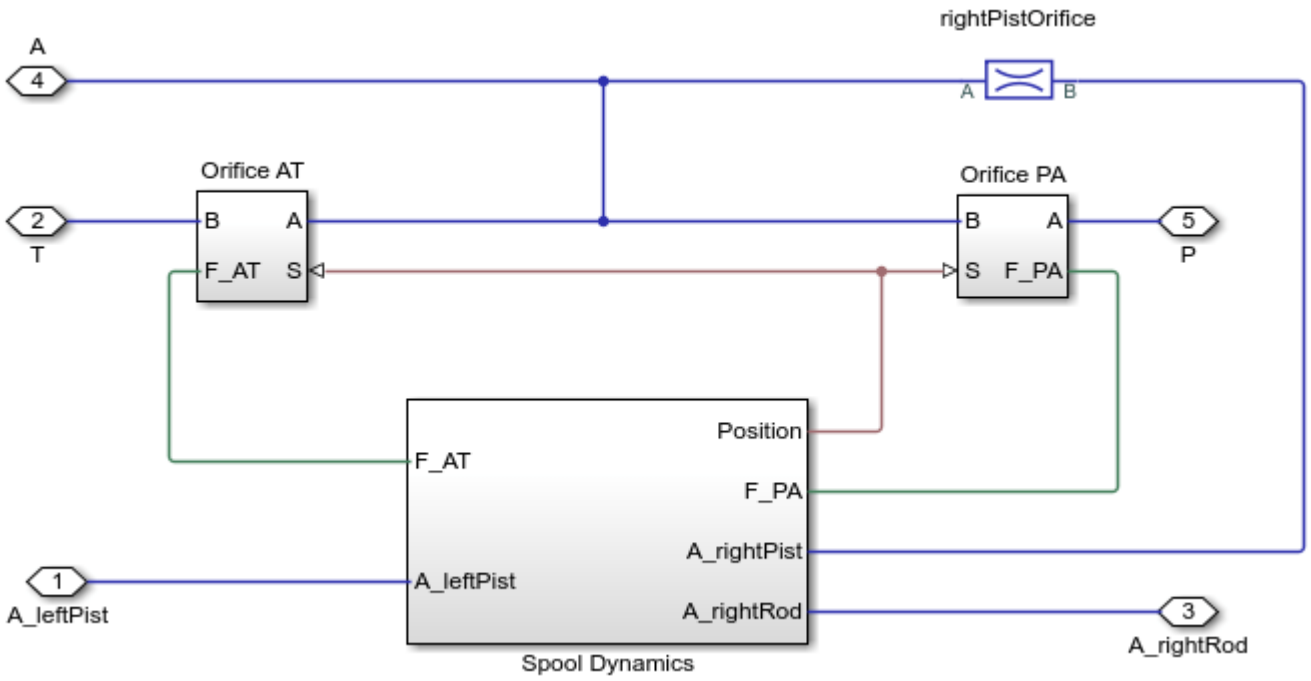
Torque Motor With Flapper Subsystem

This subsystem shows how the torque motor with the flapper is modeled. The flapper transfers the load torque to the torque motor. The load torque is generated due to the forces on the flapper from the flexure tube compression and the nozzle jets.



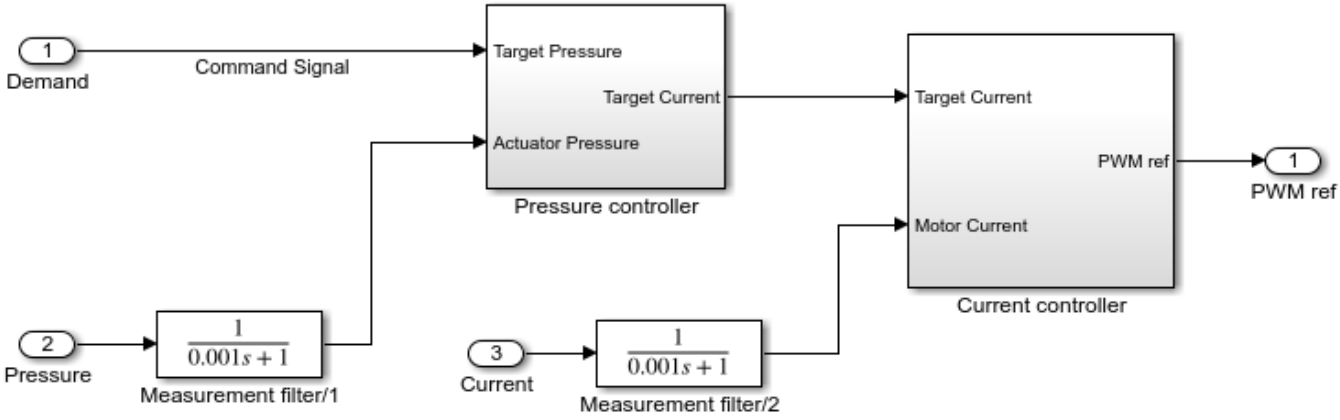
Valve Body Subsystem

This subsystem shows how the valve body is modeled. This subsystem includes modeling of the spool dynamics, the orifices opened and closed between the P to A and A to T ports due to the spool movement, and the drilled orifice in the spool's right surface. The spool dynamics includes modeling of the forces on the spool due to pressure, flexure tube compression, change in the direction of the fluid flow, friction, damping, spool spring compression, and the hard stop.



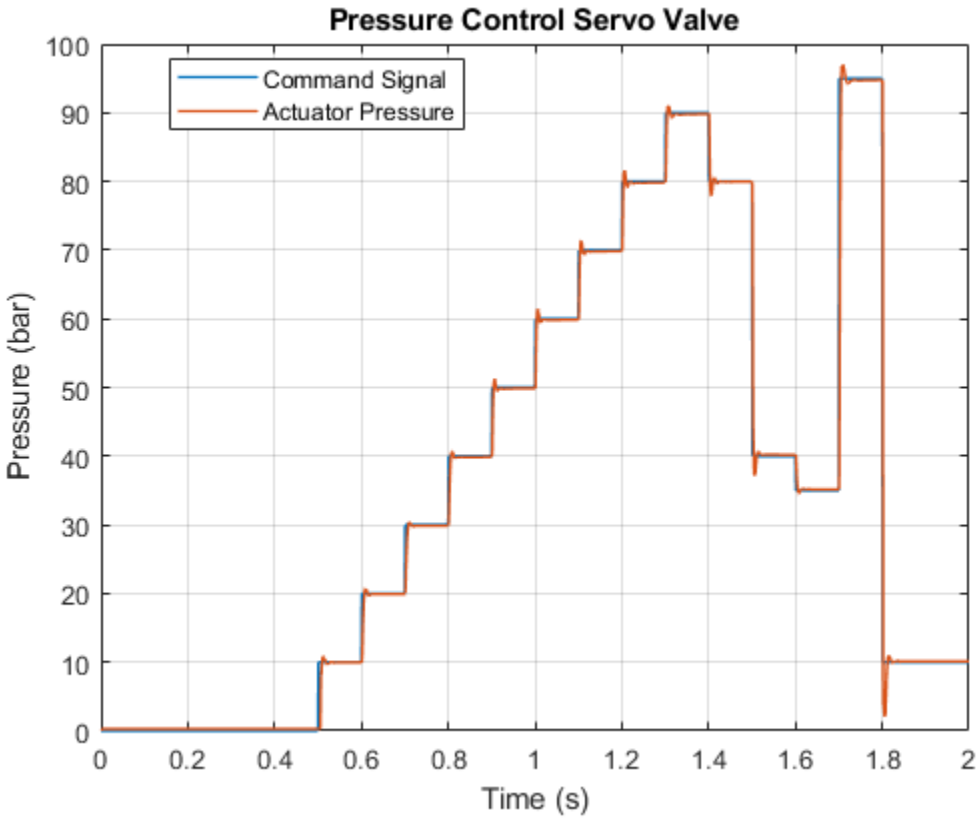
Servo Controller Subsystem

This subsystem shows how the servo controller is modeled. There are two control loops. The inner loop controls the motor current and the outer loop controls the pressure in the actuator.



Simulation Results from Simscape™ Logging

This model generates a comparison plot between the commanded and the achieved pressure in the actutator with respect to the time.

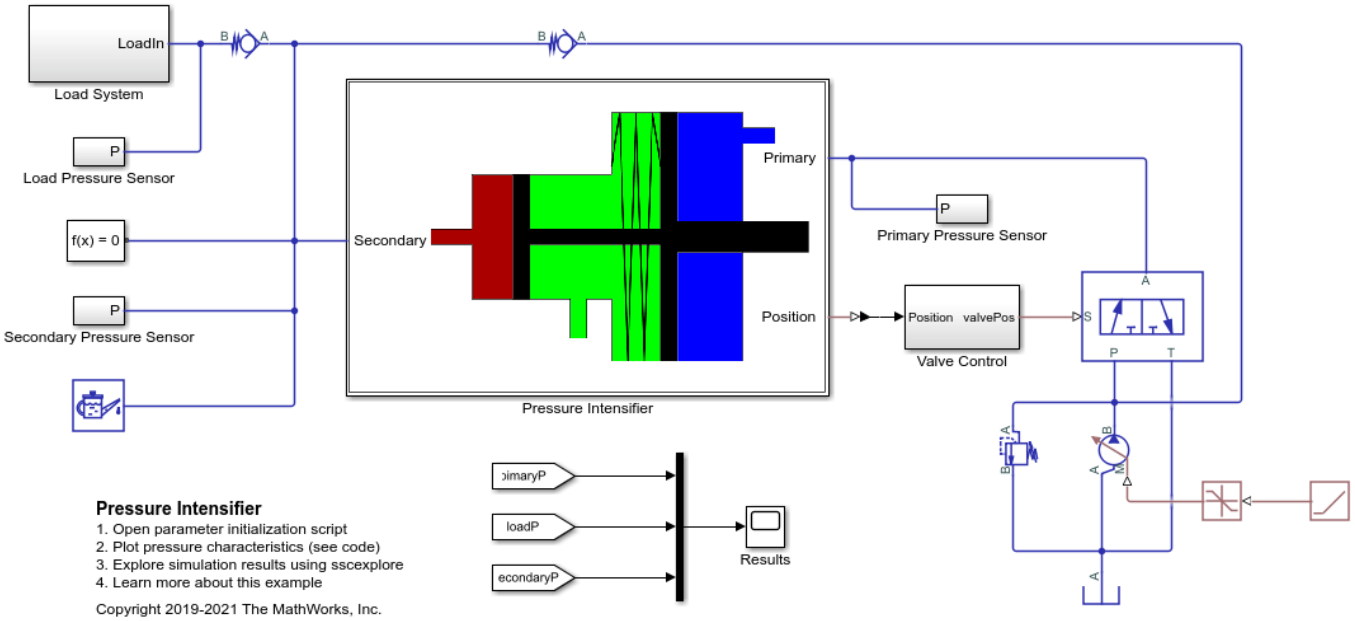

Pressure Control Servo Valve
Command Signal
Actuator Pressure
Pressure (bar)
Time (s)
0
10
20
30
40
50
60
70
80
90
100
0
0.2
0.4
0.6
0.8
1
1.2
1.4
1.6
1.8
2

Pressure Intensifier

This example shows how to model, parameterize, and test a pressure intensifier.

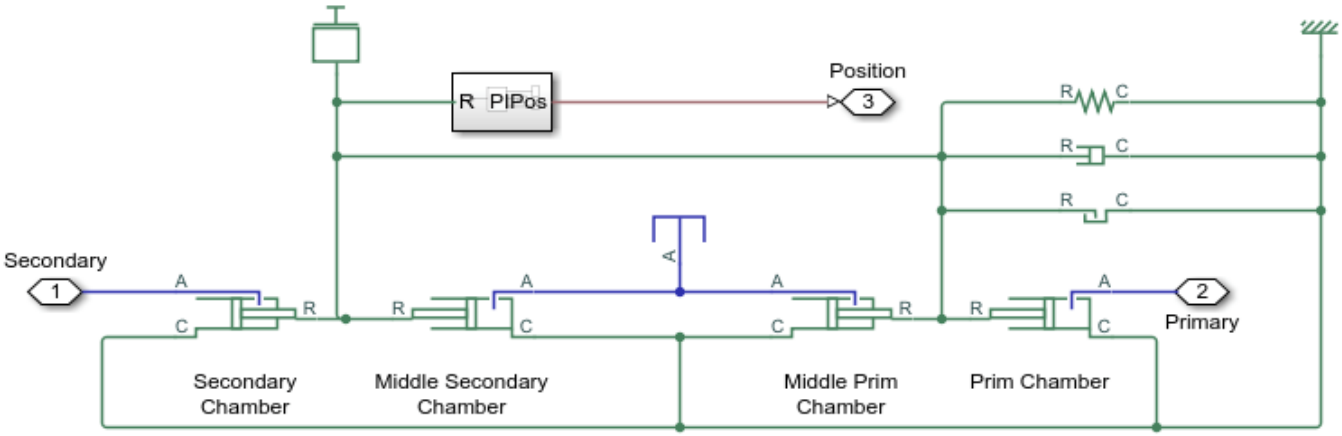
Model

The following figure shows a model of the pressure intensifier with a test harness. A fixed displacement pump with a relief valve is used as a power source to the system. The 3-way 2-position valve connects the primary side of the presure intensifier with the power source or the reservoir as per command signal from the valve control. The secondary side of the presure intensifier is connected with the load system via a check valve.



Pressure Intensifier Subsystem

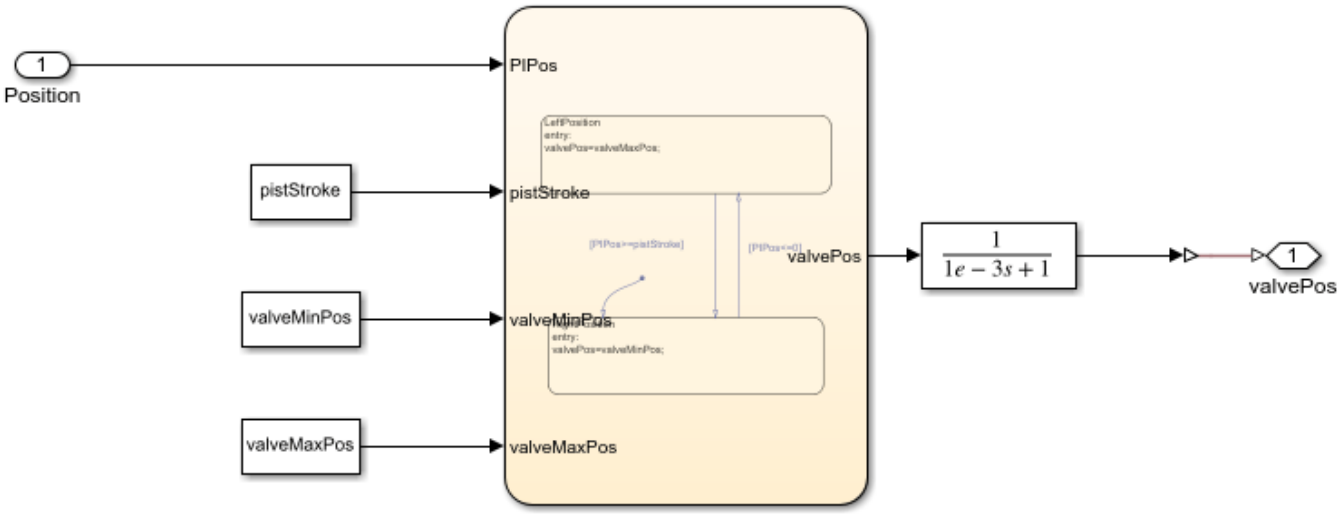
This subsystem shows how the pressure intensifier is modelled. The pressure intensifier boosts the pressure of the primary side to the pressure of the secondary side by the area ratio between the primary side and the secondary side pistons.



Valve Control Subsystem

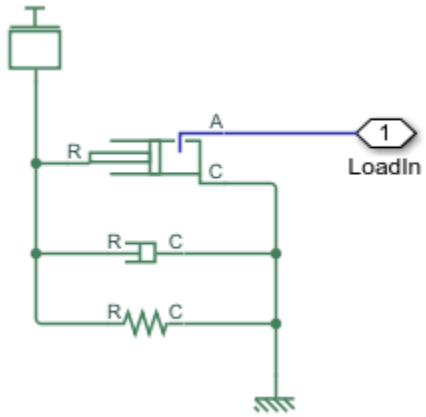
This subsystem shows an abstraction of a control system for the solenoid operated 3-way 2-position directional control valve. The control system senses the position of the pressure intensifier spool and signals the directional control valve to move to the A-T open or the P-A open position.

When the directional control valve is signaled to the A-T position, the primary side's fluid goes to the tank and the secondary side is connected to the pump. And, when the directional valve is signaled to the P-A position, the primary side is connected to the pump and the secondary side starts filling the load.



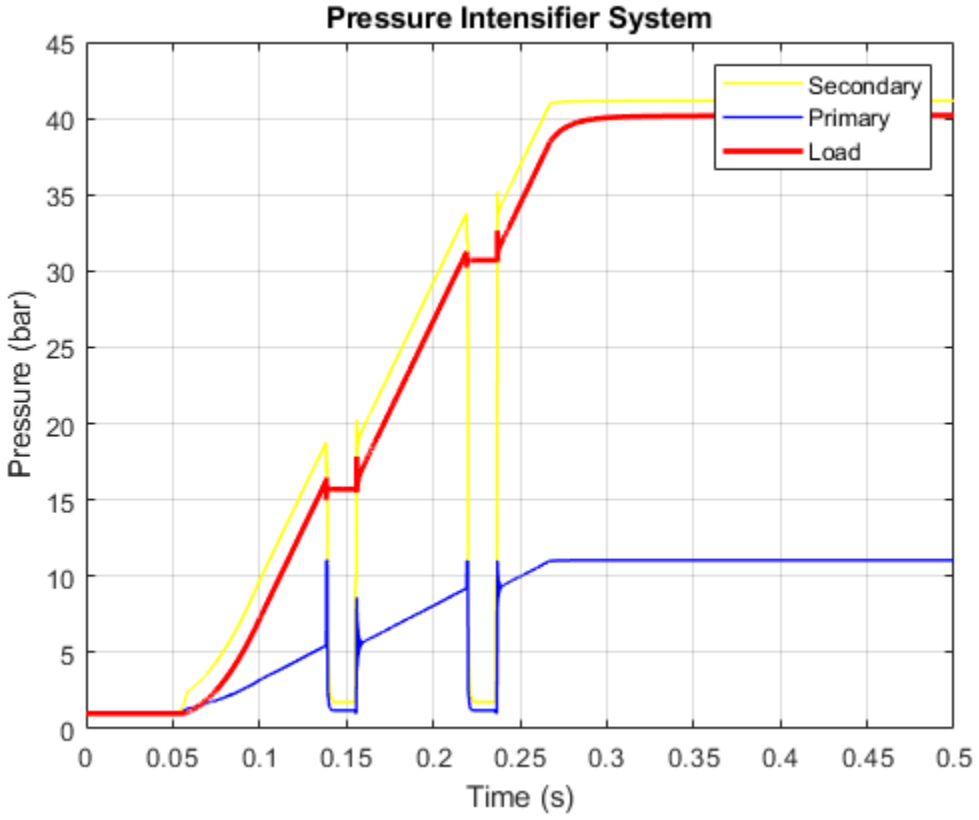
Load System Subsystem

This subsystem shows how the load system is modelled.



Simulation Results from Simscape™ Logging

This model generates a plot between the time and the generated pressures at the primary side, the secondary side, and at the load.

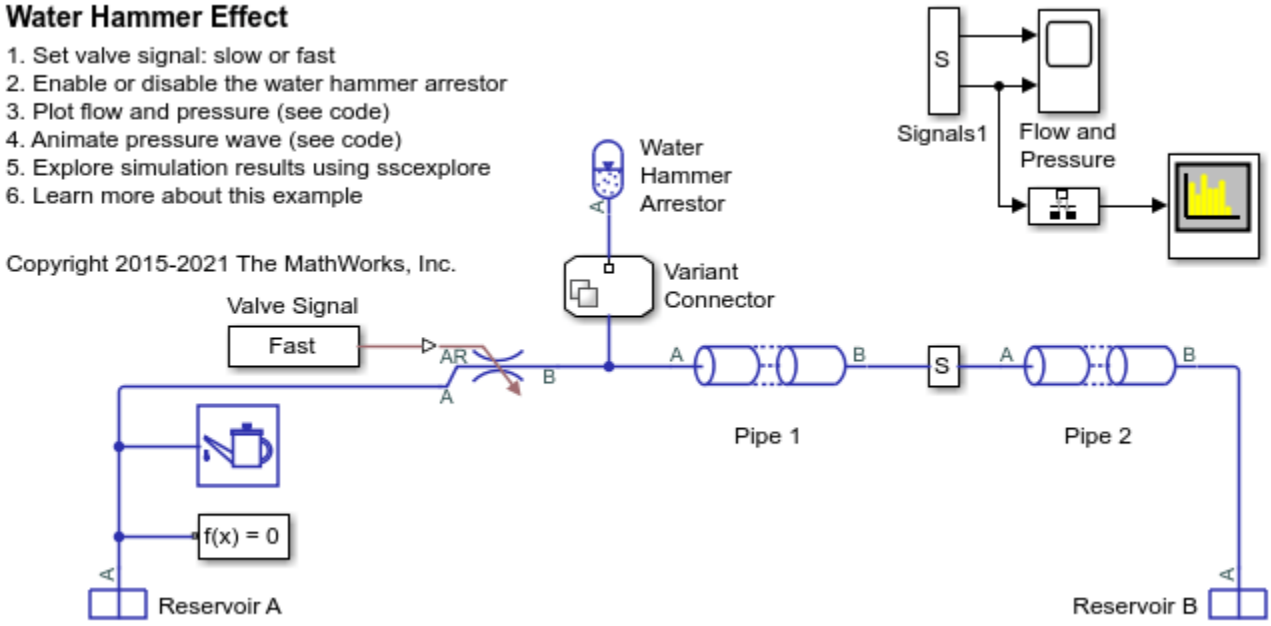


Water Hammer Effect

This demo shows how the Isothermal Liquid library can be used to model water hammer in a long pipe. After opening a valve to slowly establish steady flow in the pipe, the valve is quickly shut. If the Valve is shut quickly enough, it triggers a water hammer effect. A water hammer arrestor suppresses the pressure spikes.

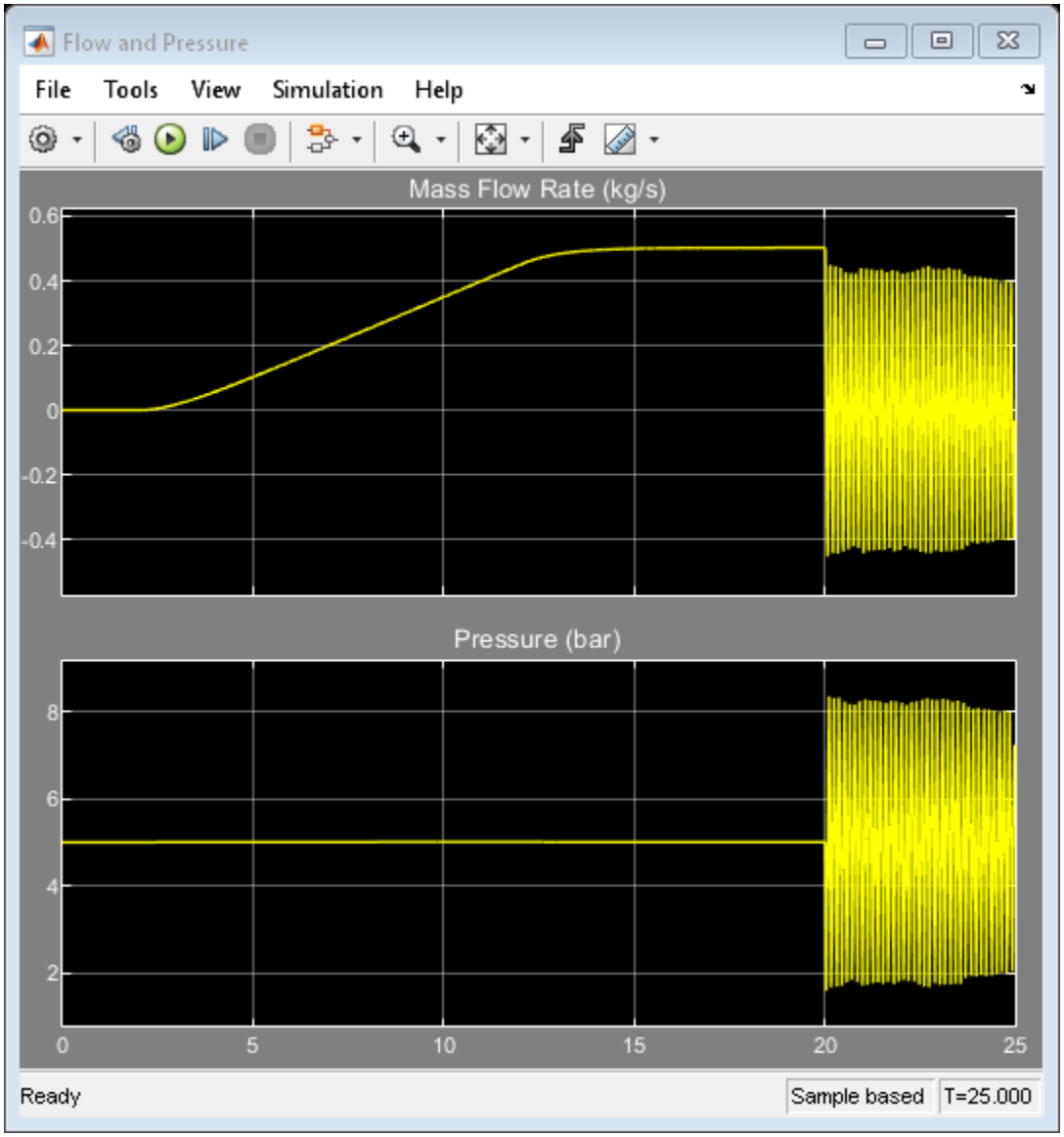
The valve is modeled using a Variable Local Restriction (IL) block, and the pipe is divided into 20 segments using two Pipe (IL) blocks. Breaking the pipe into more segments increases fidelity at the expense of simulation performance. The water hammer effect requires modeling both dynamic compressibility and inertia in the pipes. The water hammer arrestor is modeled using a Gas-Charged Accumulator (IL) block. A Variant Connector block enables and disables the water hammer arrestor.

Model

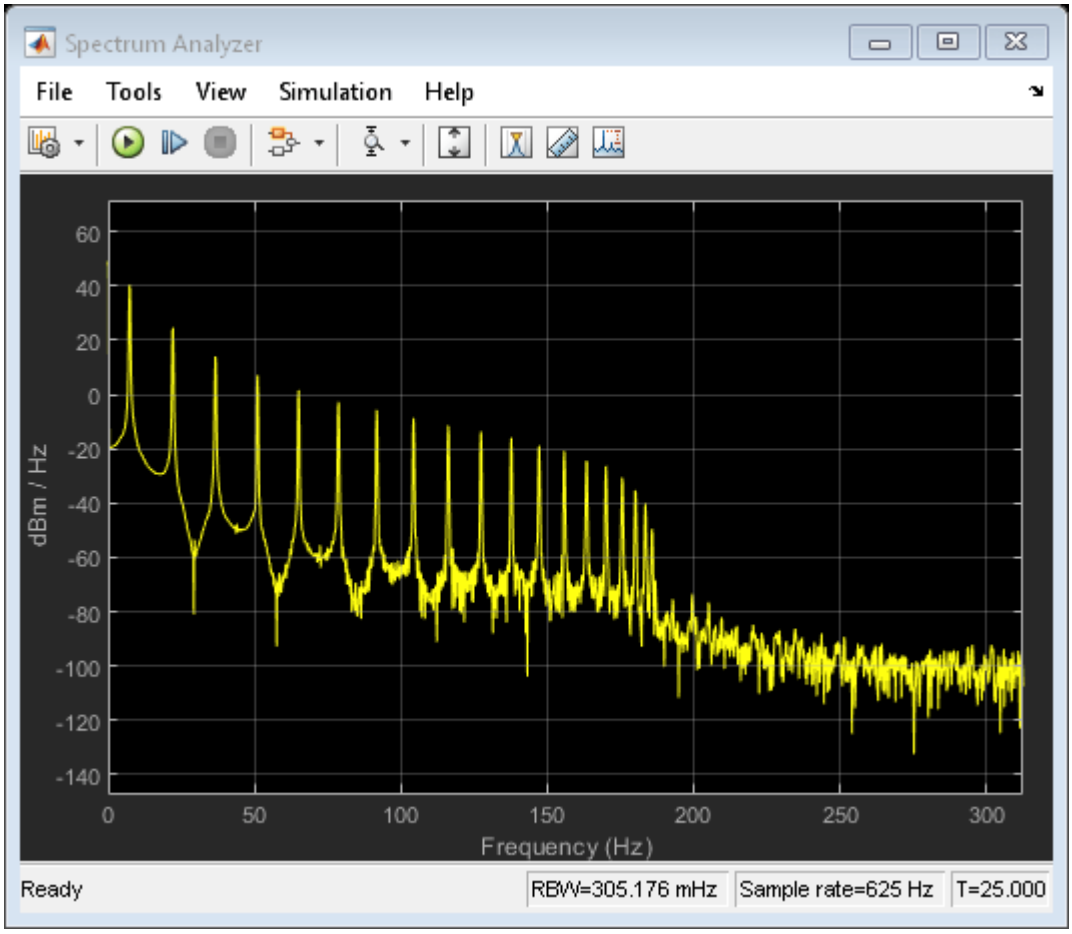


Simulation Results from Scopes

These figures show the pressure shock wave and resulting oscillations due to a sudden valve closure when the water hammer arrestor is disabled.

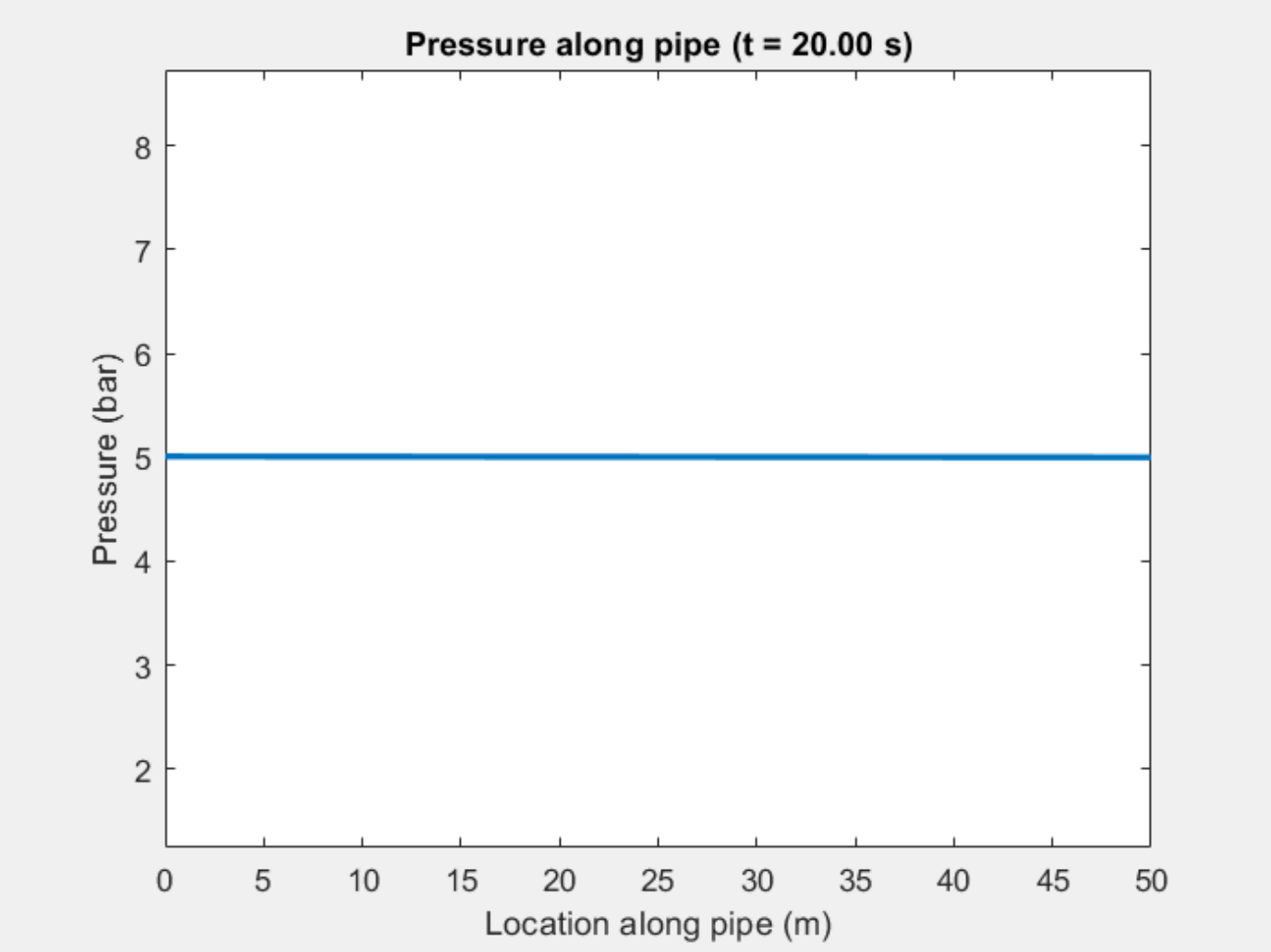

Flow and Pressure
File
Tools
View
Simulation
Help
Mass Flow Rate (kg/s)
0.6
0.4
0.2
0
-0.2
-0.4
Pressure (bar)
8
6
4
2
0
5
10
15
20
25
Ready
Sample based
T=25.000

Power Spectral Density



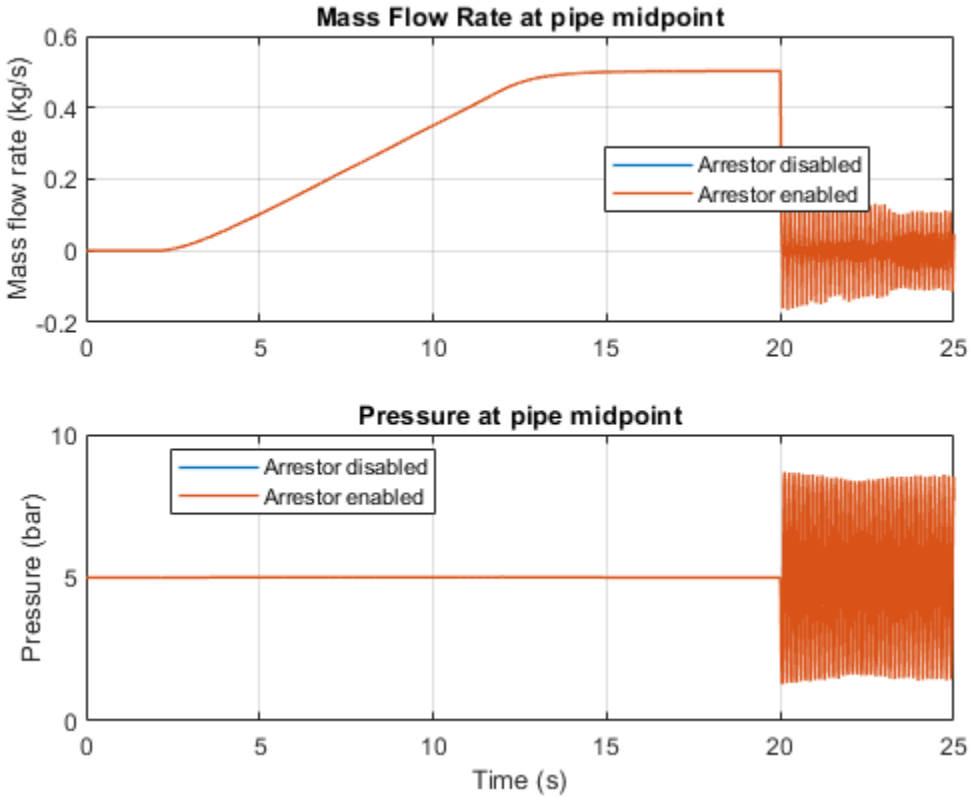
Animation of Simscape Logging Results

This animation shows the pressure shock wave and resulting oscillations along the pipeline due to a sudden valve closure when the water hammer arrestor is disabled.



Simulation Results from Simscape Logging

This figure compares the pressure and flow rate at the pipe midpoint with the water hammer arrestor disabled and enabled.

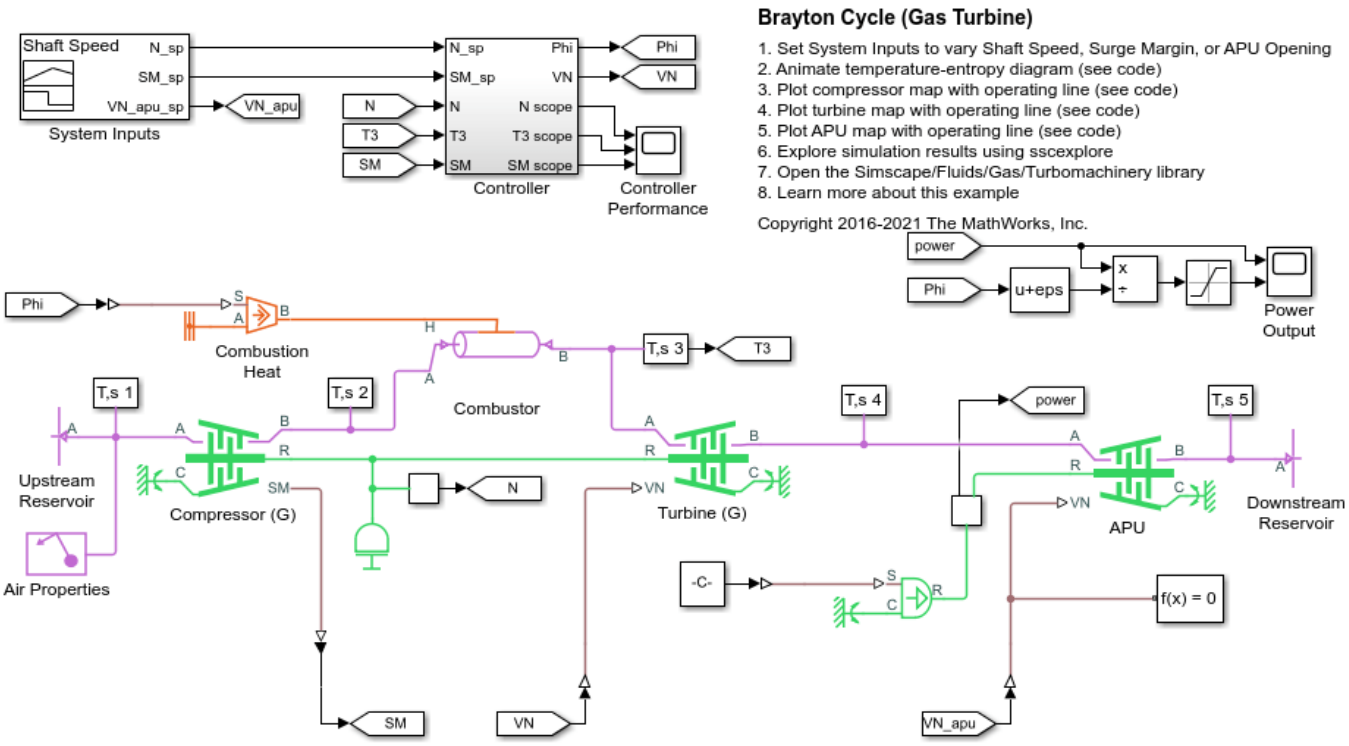

Mass Flow Rate at pipe midpoint
Mass flow rate (kg/s)
0.6
0.4
0.2
0
-0.2
0
5
10
15
20
25
Arrestor disabled
Arrestor enabled
Pressure at pipe midpoint
Pressure (bar)
10
5
0
Time (s)

Brayton Cycle (Gas Turbine)

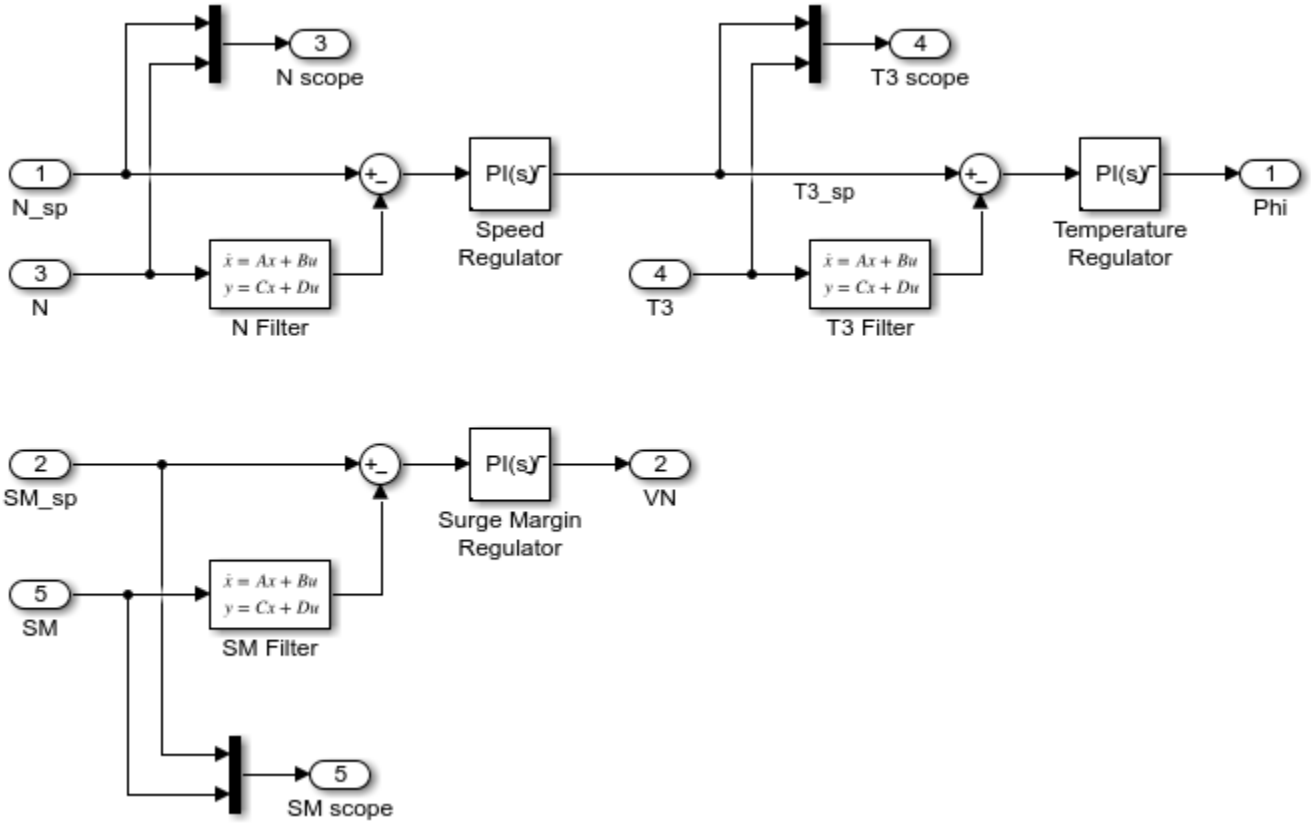
This example models a gas turbine auxiliary power unit (APU) based on the Brayton Cycle. The Compressor and Turbine blocks are custom components based on the Simscape™ Foundation Gas Library. The power input to the system is represented by heat injection into the combustor; actual combustion chemistry is not modeled. A single shaft connects the compressor and the turbine so that the power from the turbine drives the compressor. The APU is a free turbine that further expands the exhaust stream to produce output power.

Three PID controllers regulate the shaft speed, the turbine inlet temperature, and the compressor surge margin. System inputs are defined for three scenarios: varying shaft speed, varying surge margin, and varying APU vane opening. Running the first scenario produces the typical operating line on the compressor map. Running the second and third scenarios show where the maximum power output and maximum global efficiency occurs.

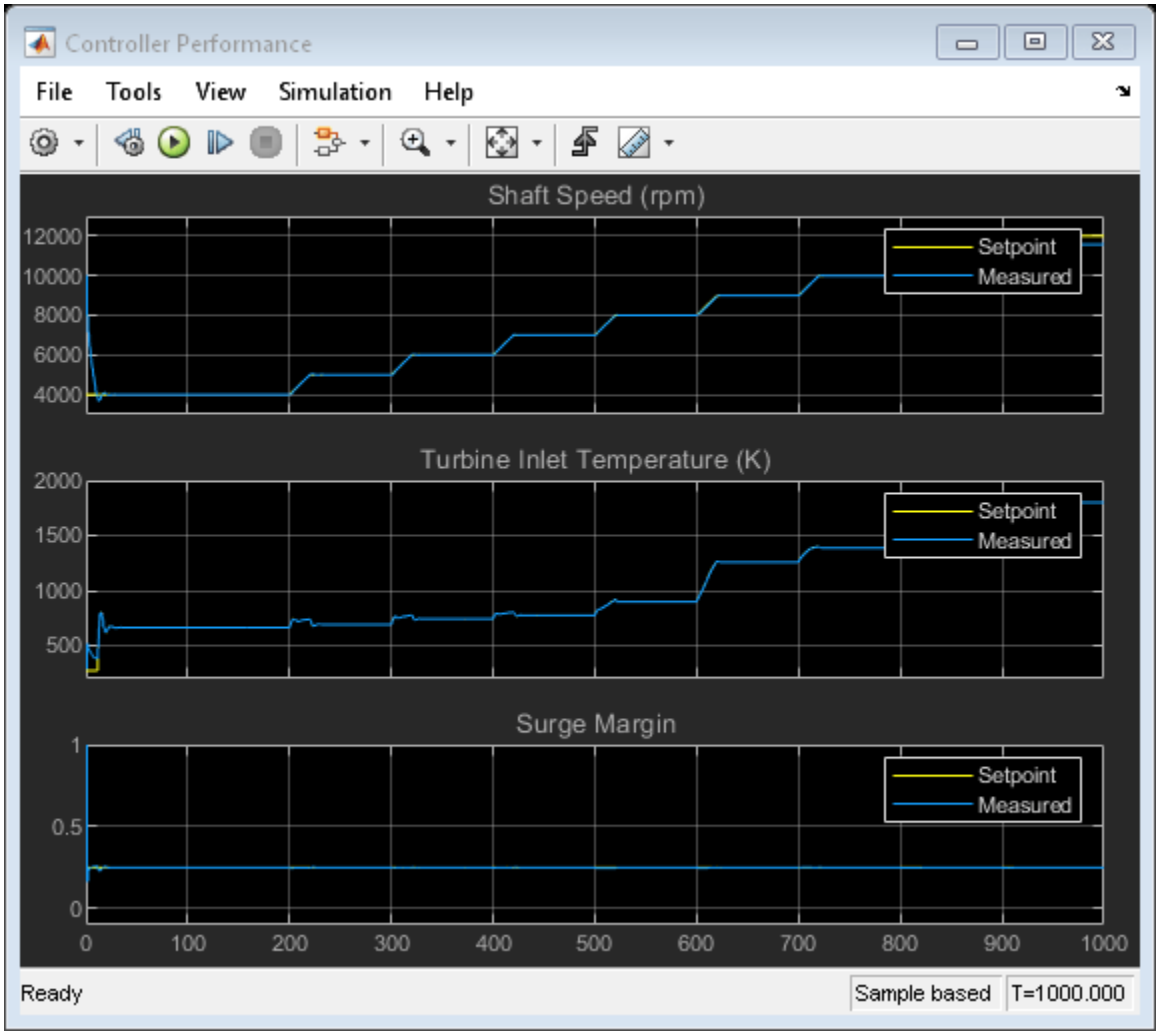
Model

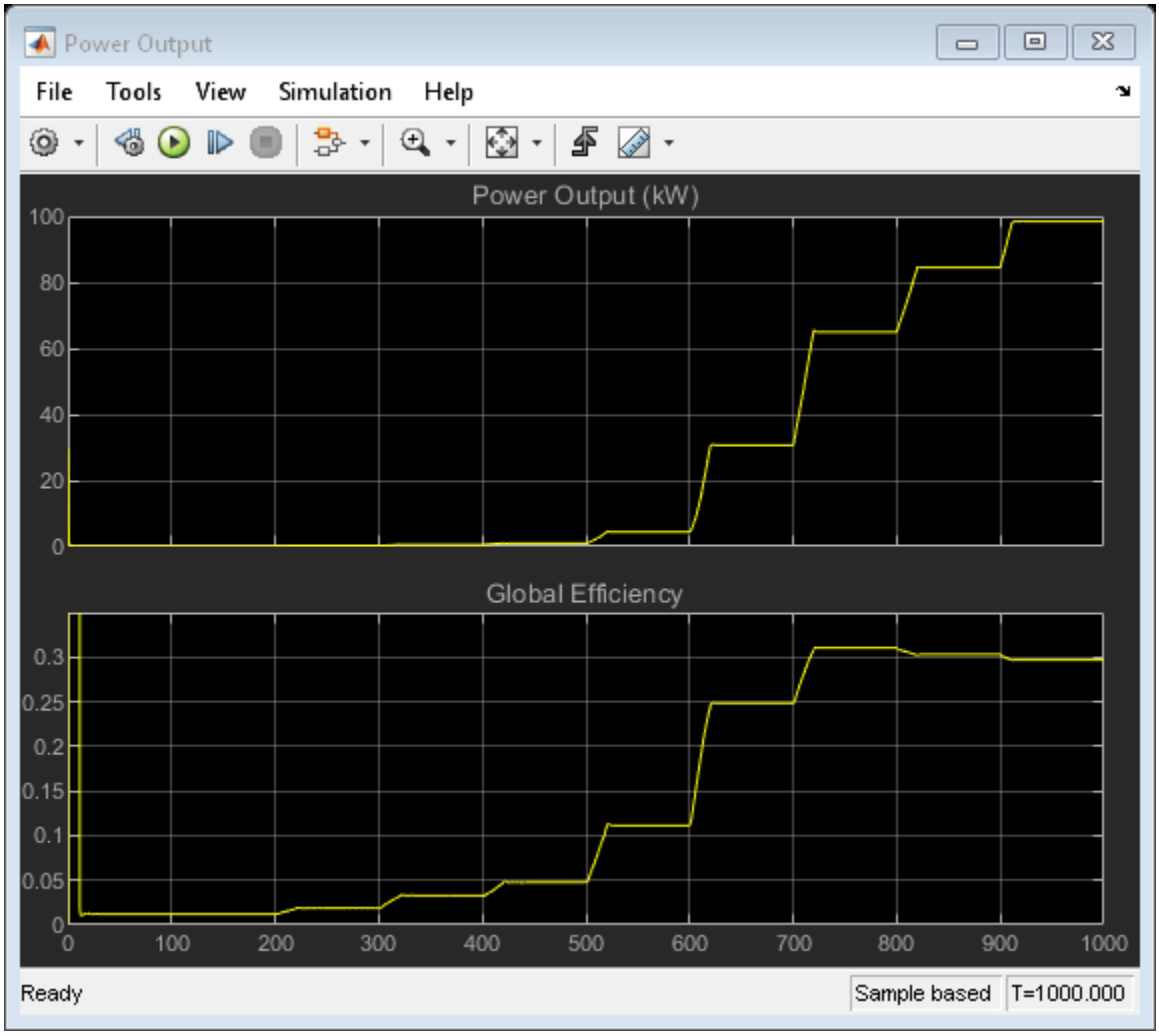


Controller Subsystem



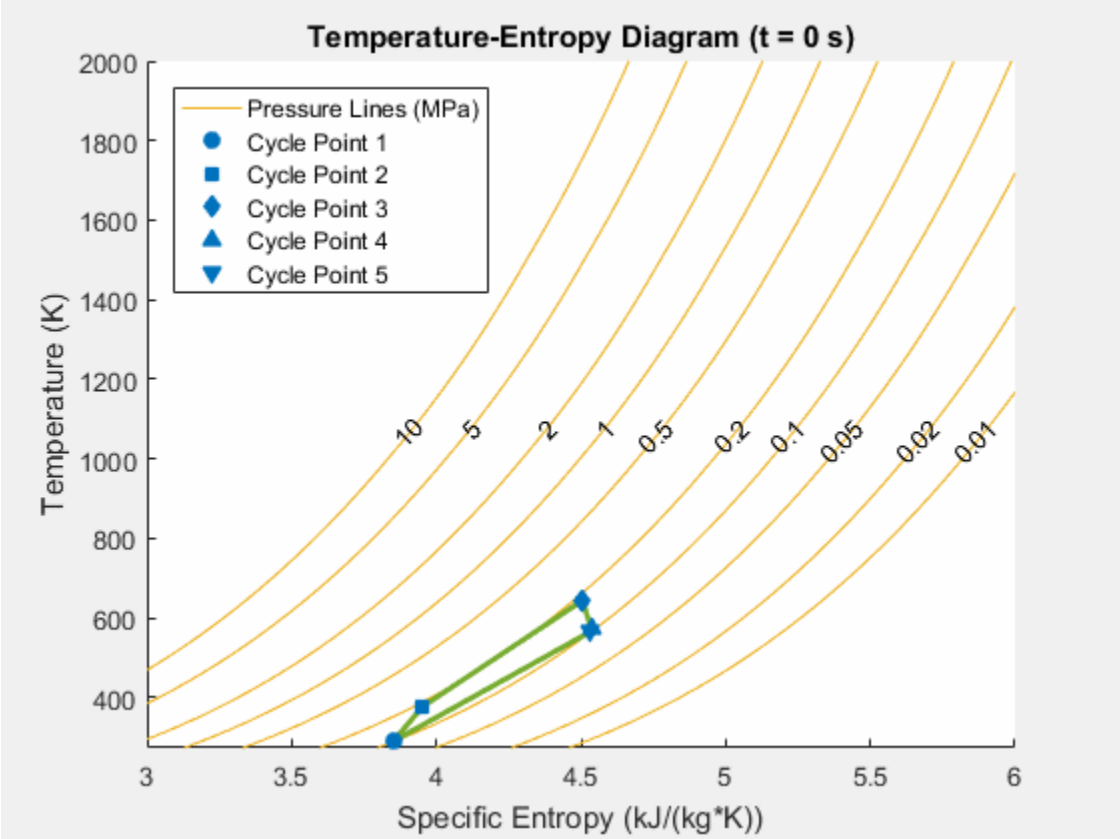
Simulation Results from Scopes



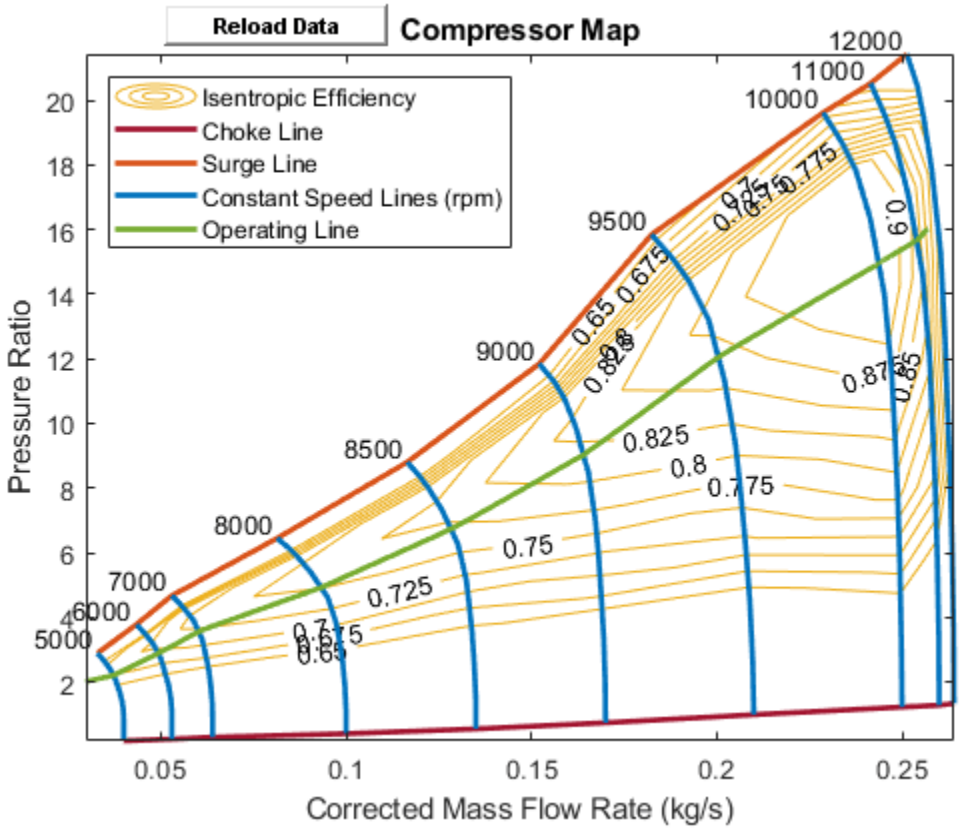


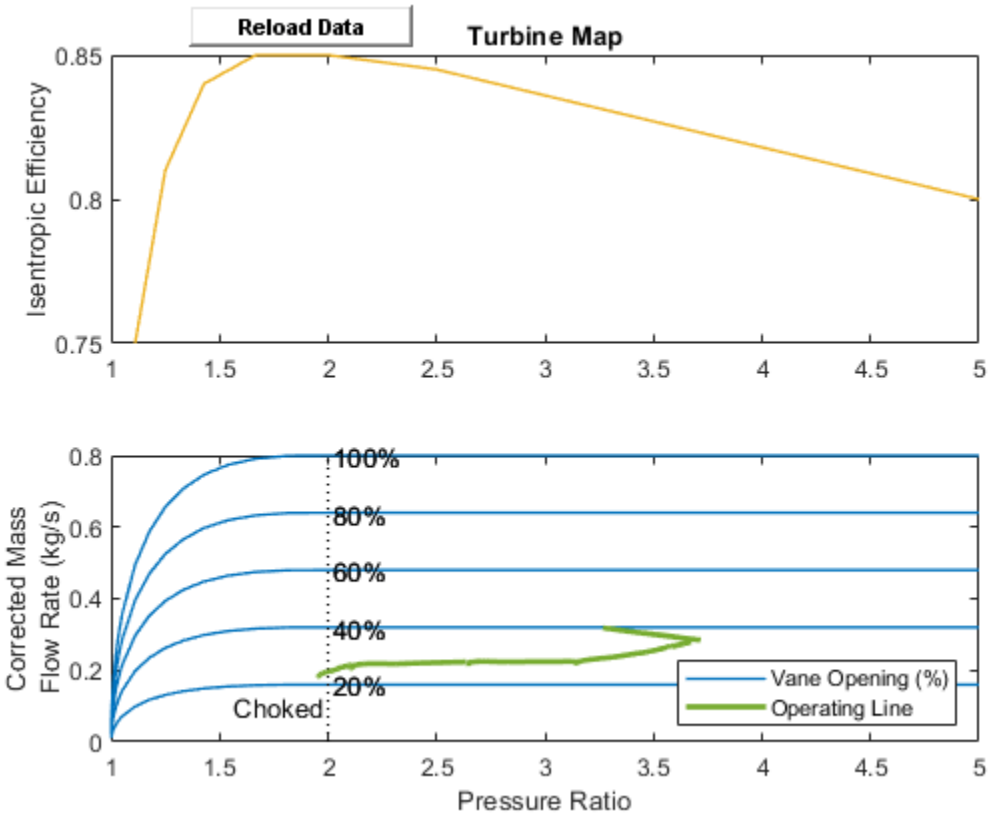
Animation of Simscape Logging Results

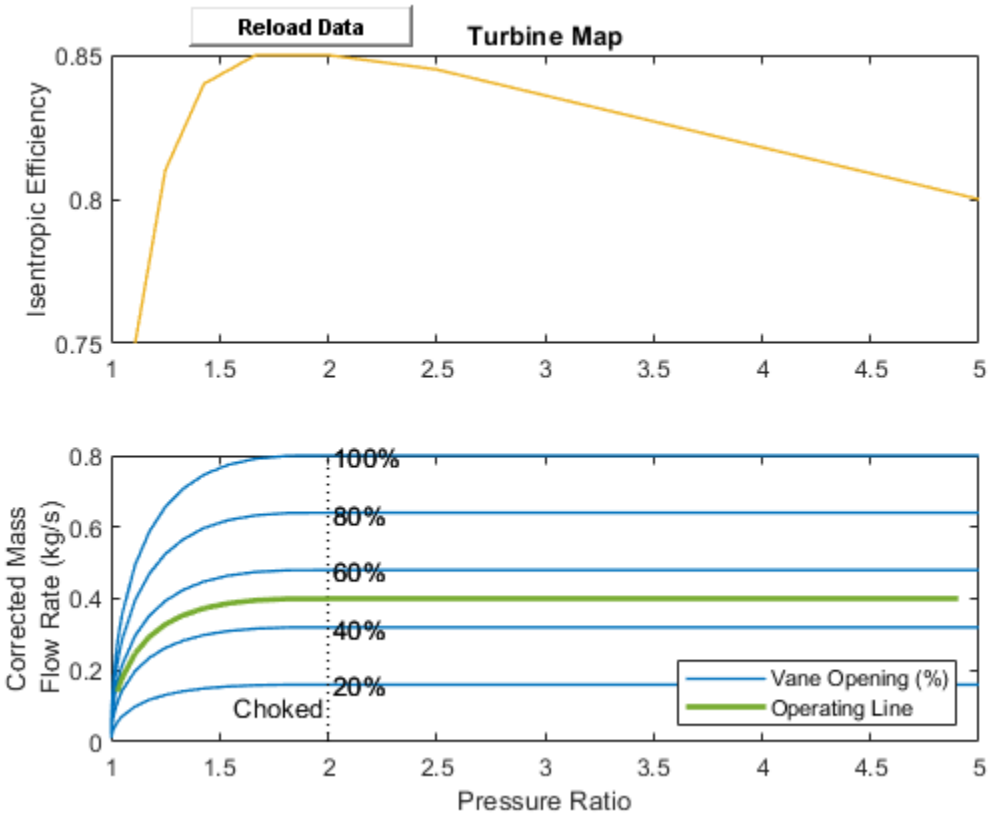
This figure shows an animation of the Brayton Cycle on a temperature-entropy diagram over time. The five cycle points on the figure correspond to the sensor measurements labeled "T,s 1" to "T,s 5", respectively.


Temperature-Entropy Diagram (t = 0 s)
Pressure Lines (MPa)
Cycle Point 1
Cycle Point 2
Cycle Point 3
Cycle Point 4
Cycle Point 5
10
5
2
1
0.5
0.2
0.1
0.05
0.02
0.01
Temperature (K)
2000
1800
1600
1400
1200
1000
800
600
400
3
3.5
4
4.5
5
5.5
6
Specific Entropy (kJ/(kg*K))

Simulation Results from Simscape Logging




Reload Data
Turbine Map
Isentropic Efficiency
0.85
0.8
0.75
1
1.5
2
2.5
3
3.5
4
4.5
5
Corrected Mass Flow Rate (kg/s)
0.8
0.6
0.4
0.2
0
100%
80%
60%
40%
20%
Choked
Vane Opening (%)
Operating Line
Pressure Ratio

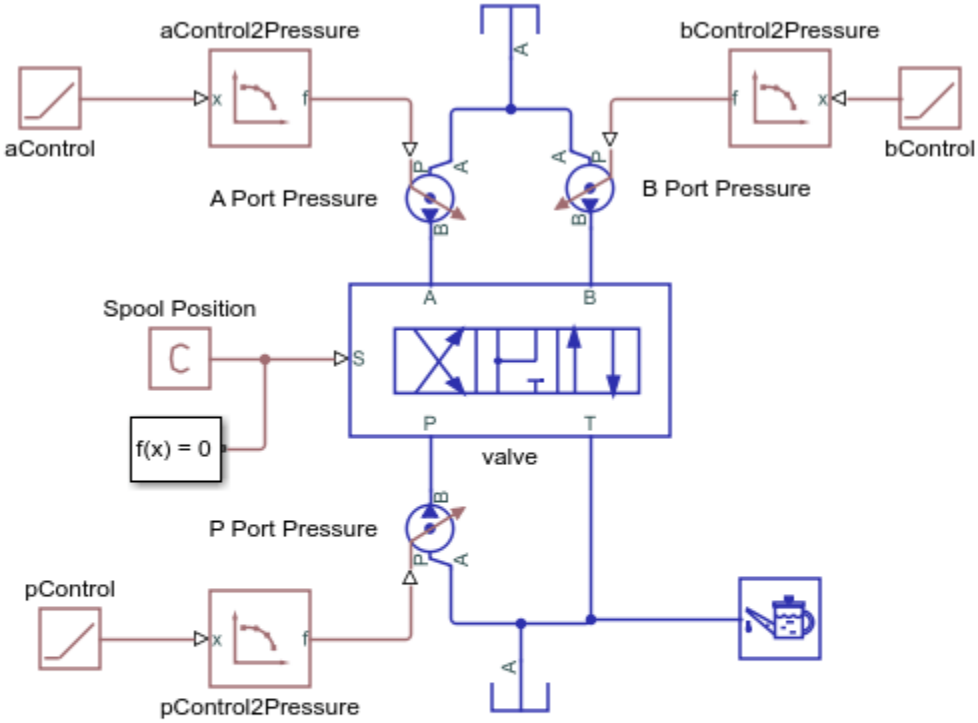

Reload Data
Turbine Map
Isentropic Efficiency
0.85
0.8
0.75
1
1.5
2
2.5
3
3.5
4
4.5
5
Corrected Mass Flow Rate (kg/s)
0.8
0.6
0.4
0.2
0
100%
80%
60%
40%
20%
Choked
Vane Opening (%)
Operating Line
Pressure Ratio

4-Way 3-Position Valve Parameterization

This example shows how to parameterize and test a 4-way 3-position valve with a test harness. A plot script is provided with the example for comparing output flow between the block and data to verify the test harness. A live script is also provided with this example to explain the parameterization and the test harness workflow in detail.

Model

The following figure shows a 4-way 3-position valve test harness. The Pressure Source (IL) blocks; with Ramp blocks are used to create pressure boundaries for the directional control valve. The spool displacement is provided using a Constant block.



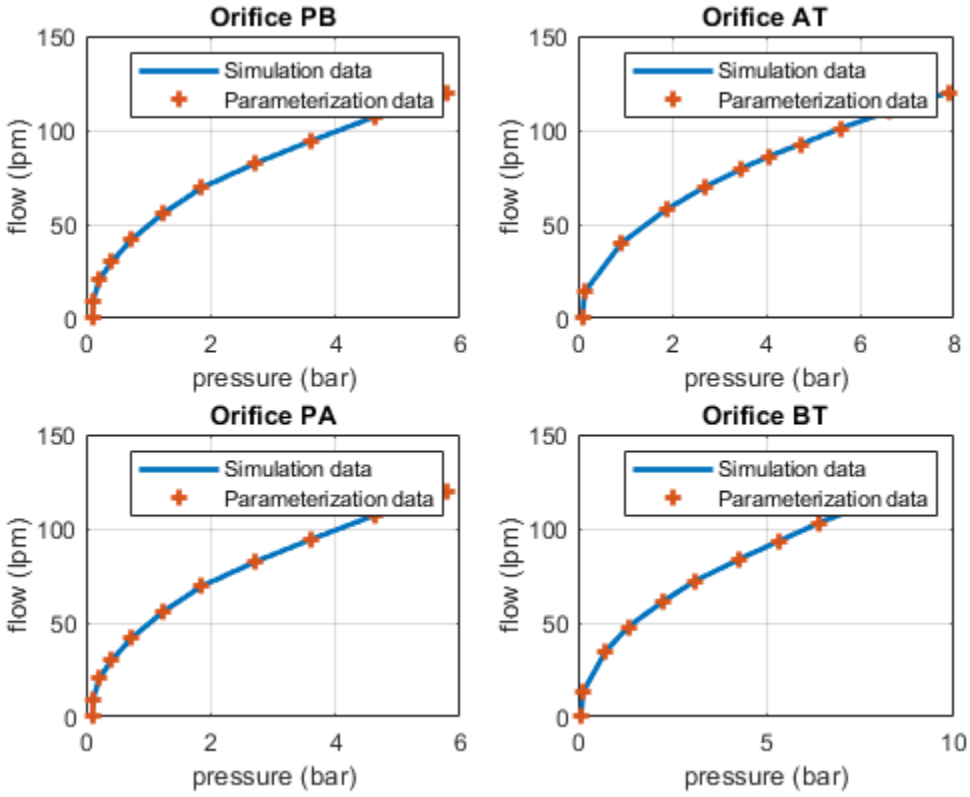
4-Way 3-Position Valve Parameterization

1. Open parameter input script
2. Open live script
3. Compare datasheet and simulation results (see code)
4. Explore simulation results using sscexplore
5. Learn more about this example

Copyright 2019-2021 The MathWorks, Inc.

Simulation results from simscape™ logging

The model test harness is parameterized for the positive and the negative spool positions and the simulated output is compared with the parameterization setting.

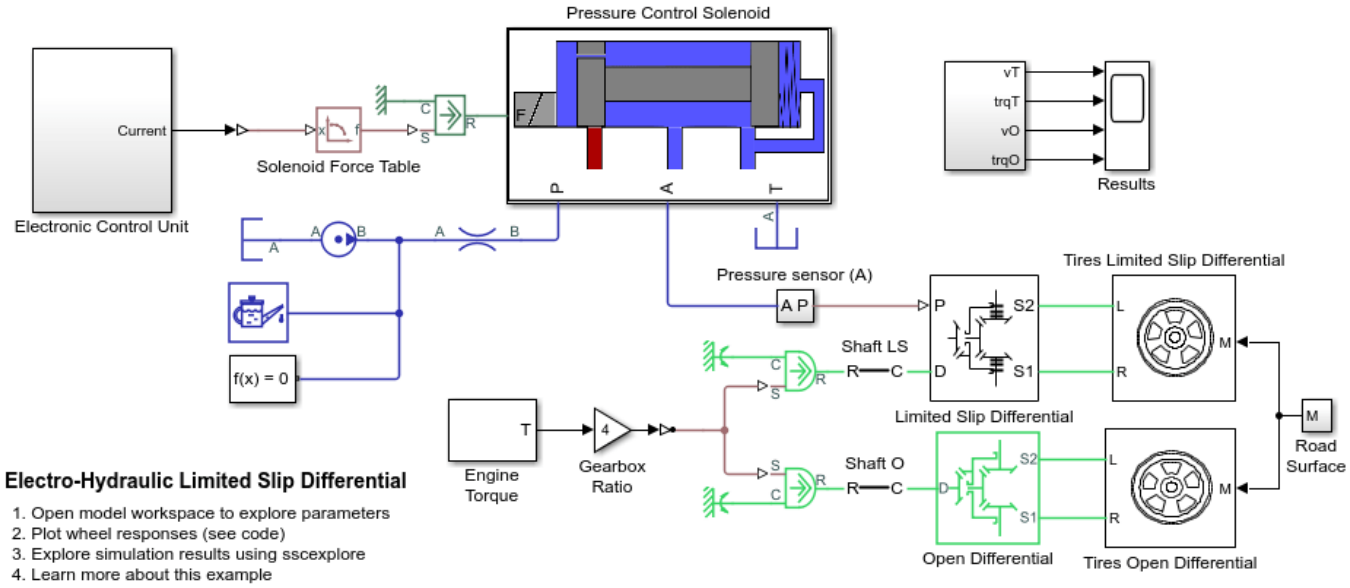

Orifice PB
Simulation data
Parameterization data
flow (lpm)
pressure (bar)
Orifice AT
Simulation data
Parameterization data
flow (lpm)
pressure (bar)
Orifice PA
Simulation data
Parameterization data
flow (lpm)
pressure (bar)
Orifice BT
Simulation data
Parameterization data
flow (lpm)
pressure (bar)

Electro-Hydraulic Limited Slip Differential

This example shows a simple way of modeling an electro-hydraulic limited slip differential system. The model shows the velocity and the torque profile responses achieved for the left and the right wheels.

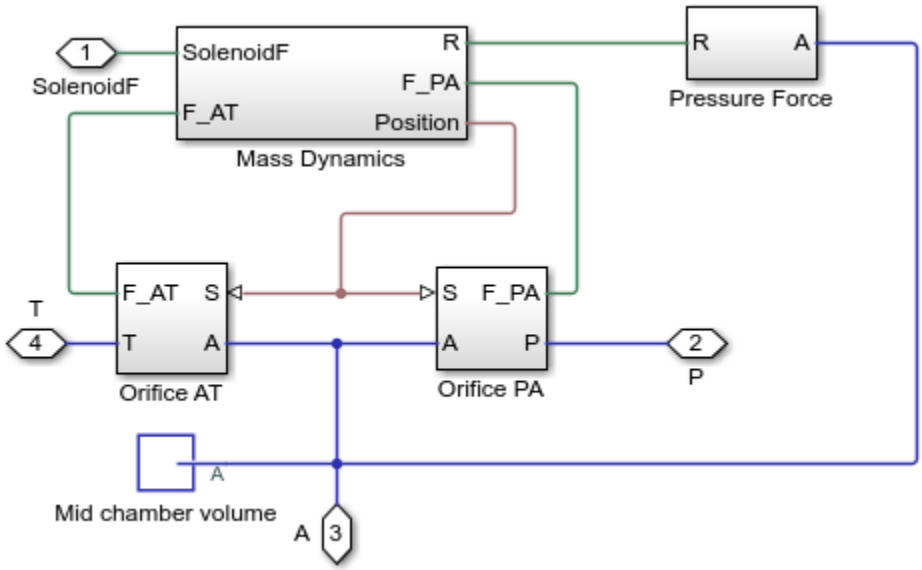
Model

The following figure shows the model of an electro-hydraulic limited slip differential system with a test harness. The electronic control unit sends a command to the pressure control solenoid valve based on the relative slip between the right and the left wheels. The pressure control solenoid valve applies a pressure on the actuating mechanism which in turn applies the limited slip control.



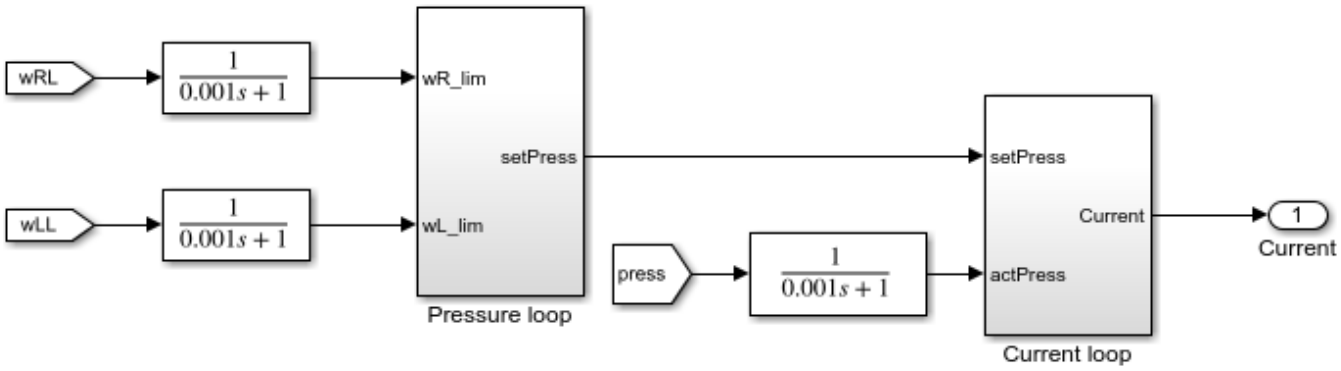
Pressure Control Solenoid Subsystem

This subsystem shows the modeling of the pressure control solenoid valve.



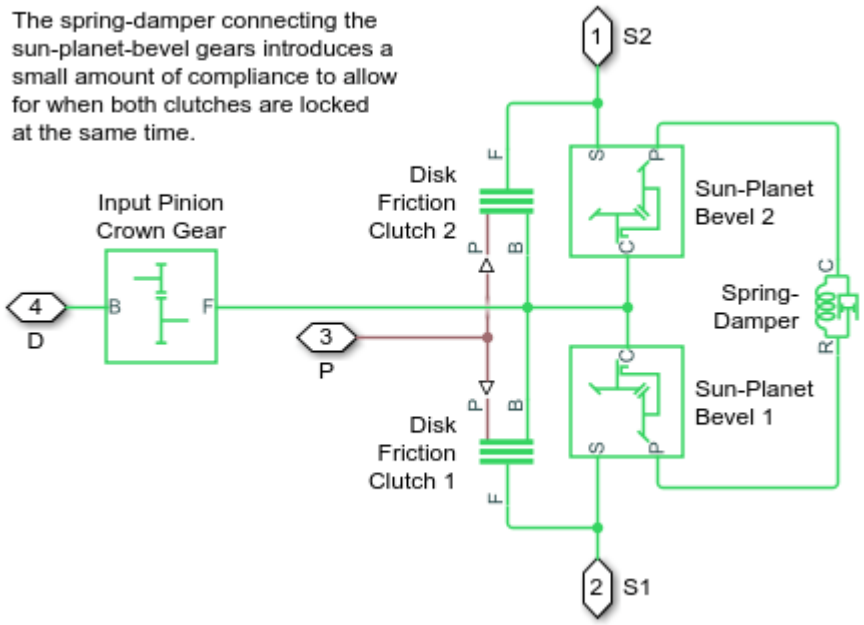
Electronic Control Unit Subsystem

This subsystem shows the modeling of the control of the pressure control solenoid.



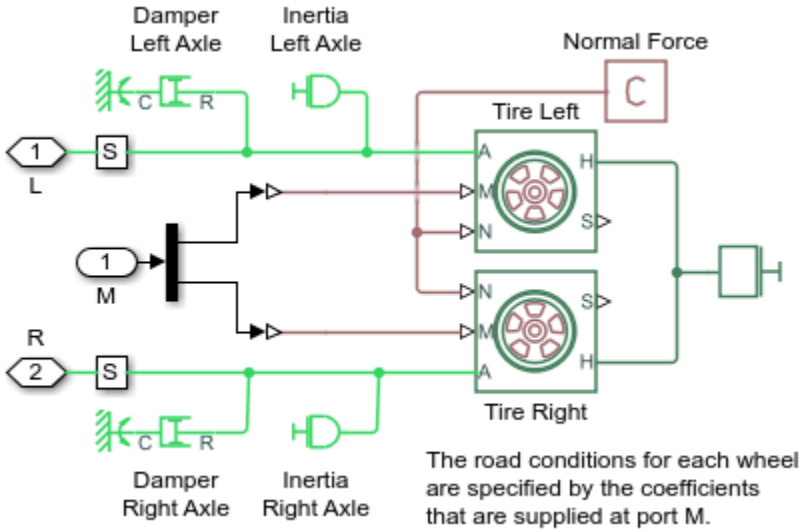
Limited Slip Differential Subsystem

This subsystem shows the modeling of the limited slip differential.



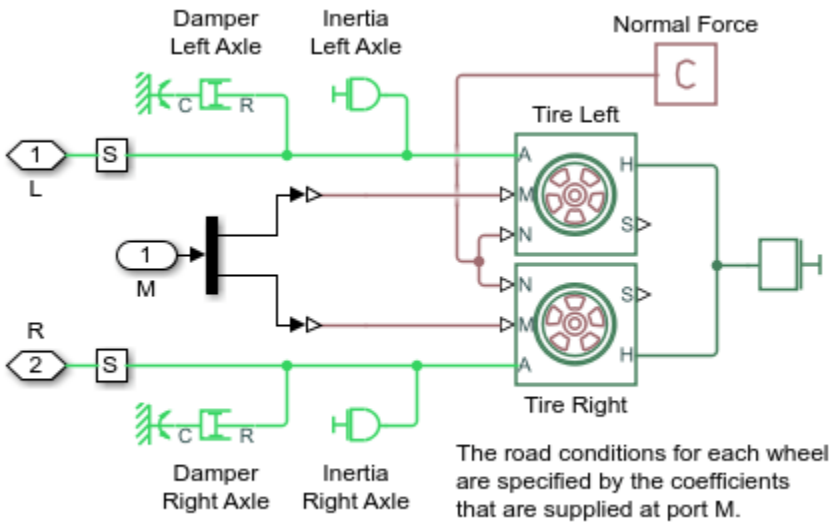
Tires Limited Slip Differential Subsystem

This subsystem shows the modeling of the tires for limited slip differential.



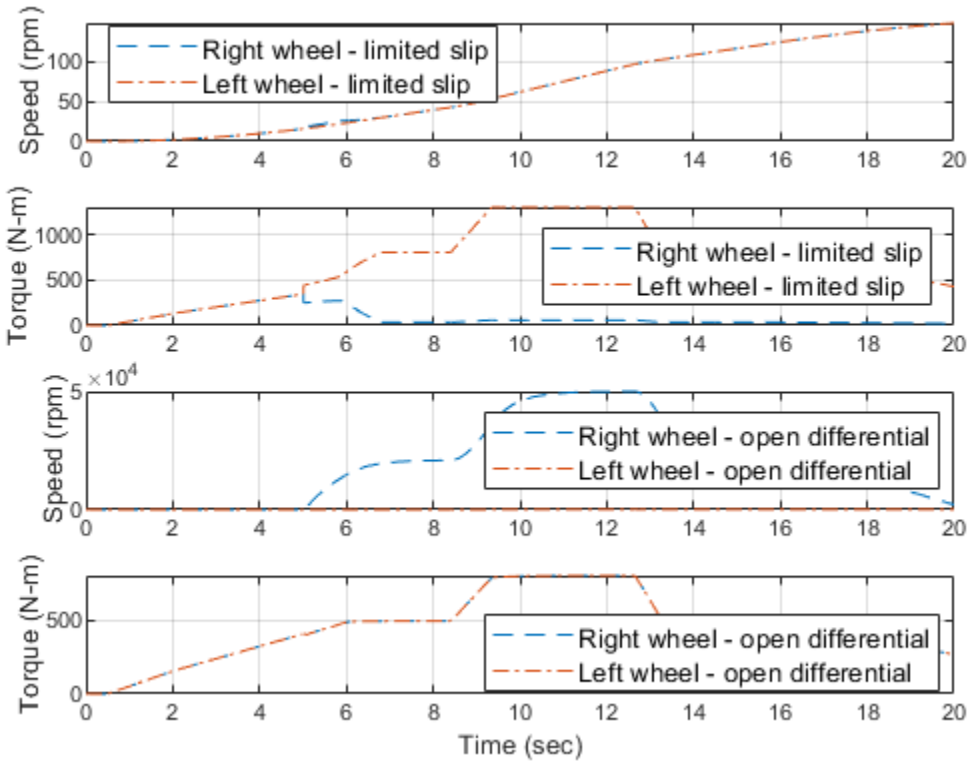
Tires Open Differential Subsystem

This subsystem shows the modeling of the tires for open differential.



Simulation results from Simscape™ logging

Based on the given inputs, the model generates a plot of the velocity and the torque profile achieved for the left and the right wheels.

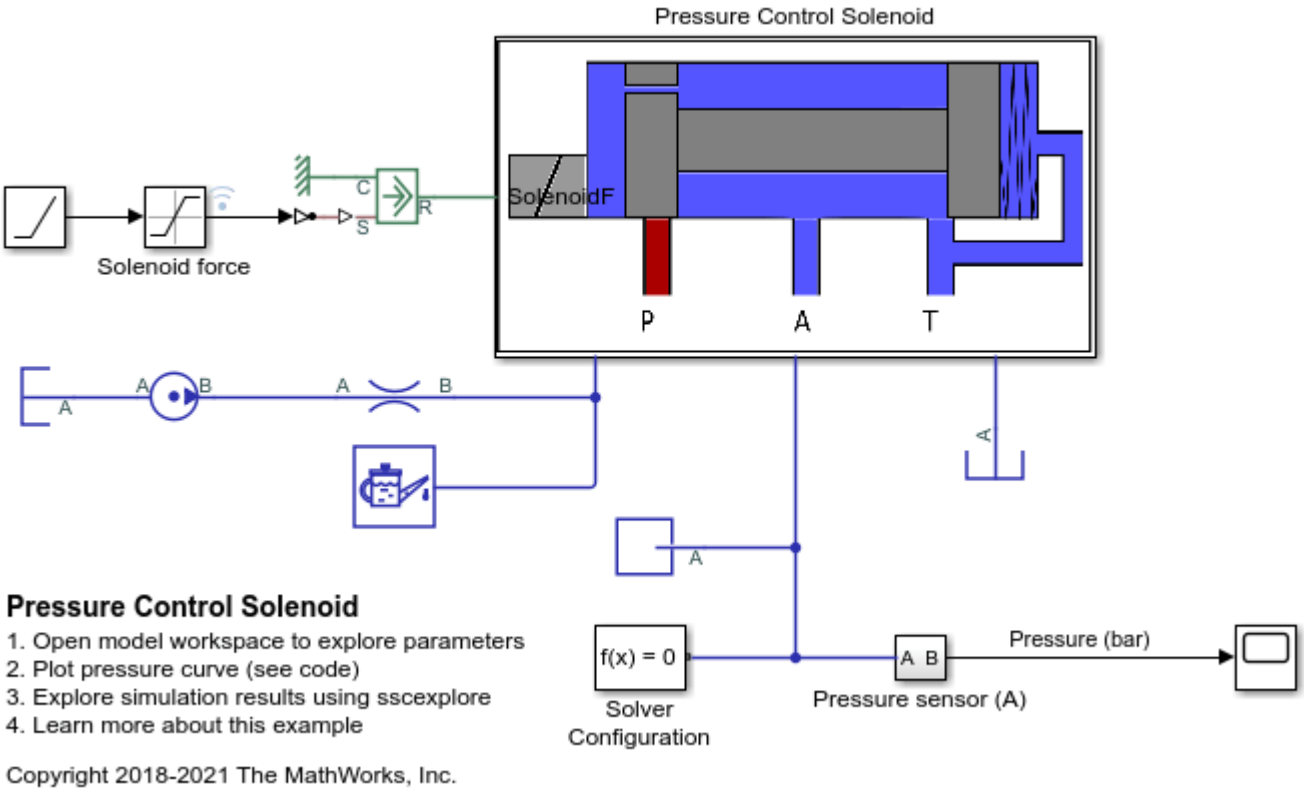


Pressure Control Solenoid

This example shows how to model, parameterize, and test a pressure control solenoid valve. This example also generates a plot of the relationship between applied solenoid force and the resulting actuator port pressure.

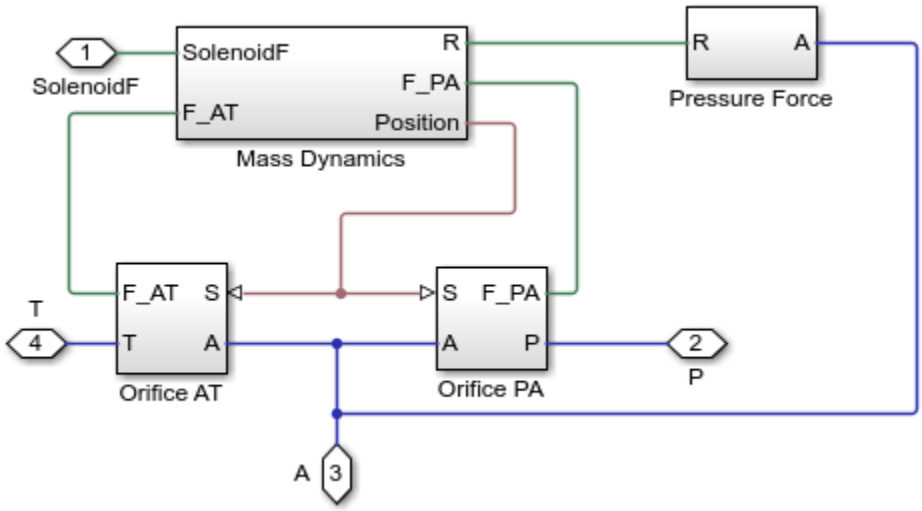
Model

The following figure shows the model of a pressure control solenoid valve. Here P represents the pressure source port,T represents the reservoir, and A represents the actuator port.



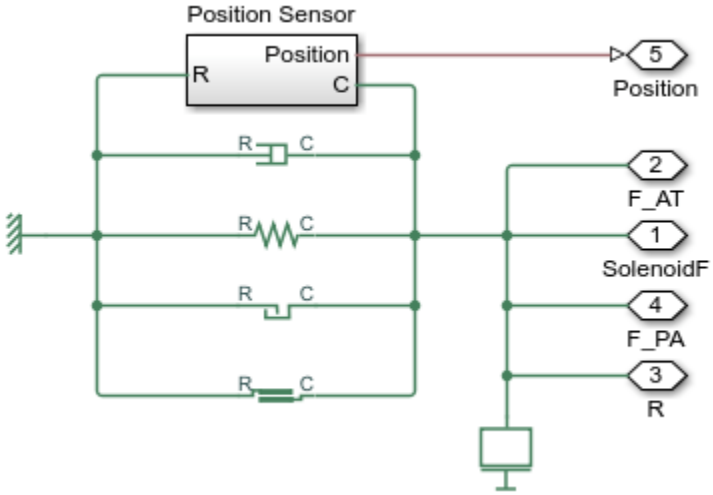
Pressure Control Solenoid Subsystem

This subsystem shows how the pressure control solenoid valve is modelled.



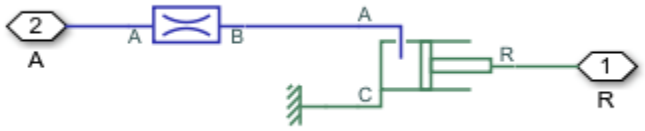
Mass Dynamics Subsystem

This subsystem shows how the spool mass dynamics are modelled, including the forces on the spool due to spool inertia; the forces on the spool due to the spring, damper, and hard stop; friction between the spool and the valve body; the change in fluid momentum at the valve entry and exit; and the pressure acting on the spool surface.



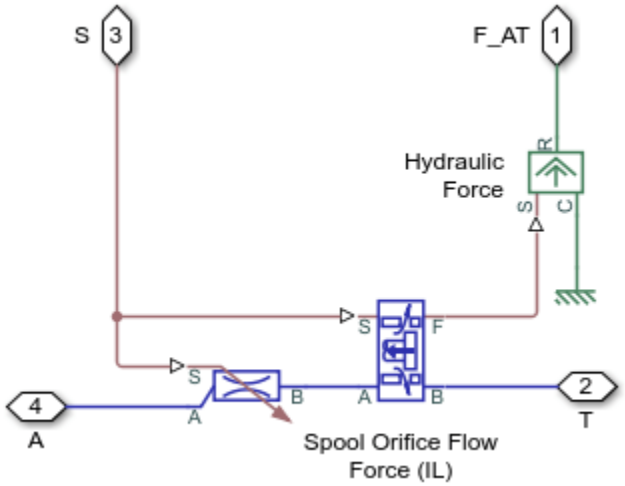
Pressure Force Subsystem

This subsystem shows how the fluid pressure at port A is modelled as a force on the spool left face.



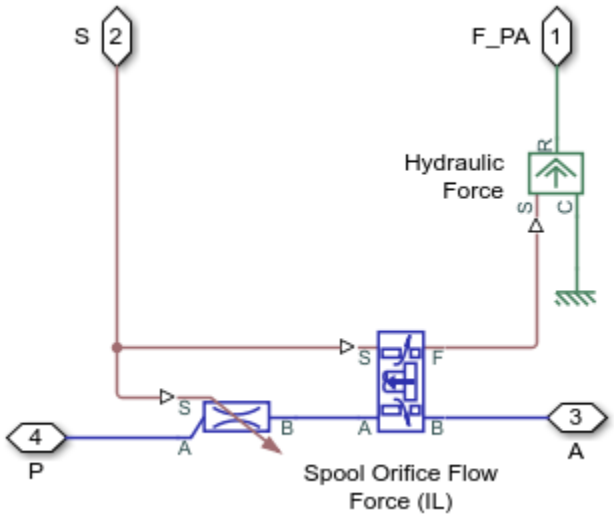
Orifice AT Subsystem

This subsystem shows how the orifice between ports A and T is modelled.



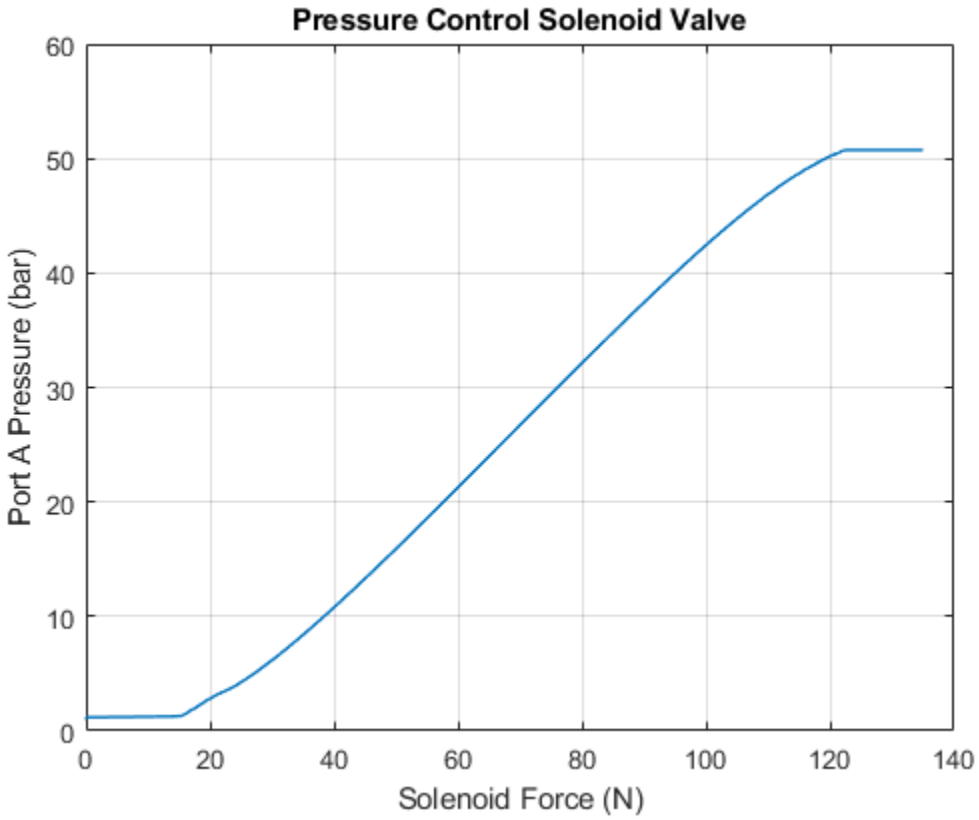
Orifice PA Subsystem

This subsystem shows how the orifice between ports A and P is modelled.



Simulation Results from Simscape™ Logging

This model generates a plot between the applied solenoid force and the generated actuator port pressure.

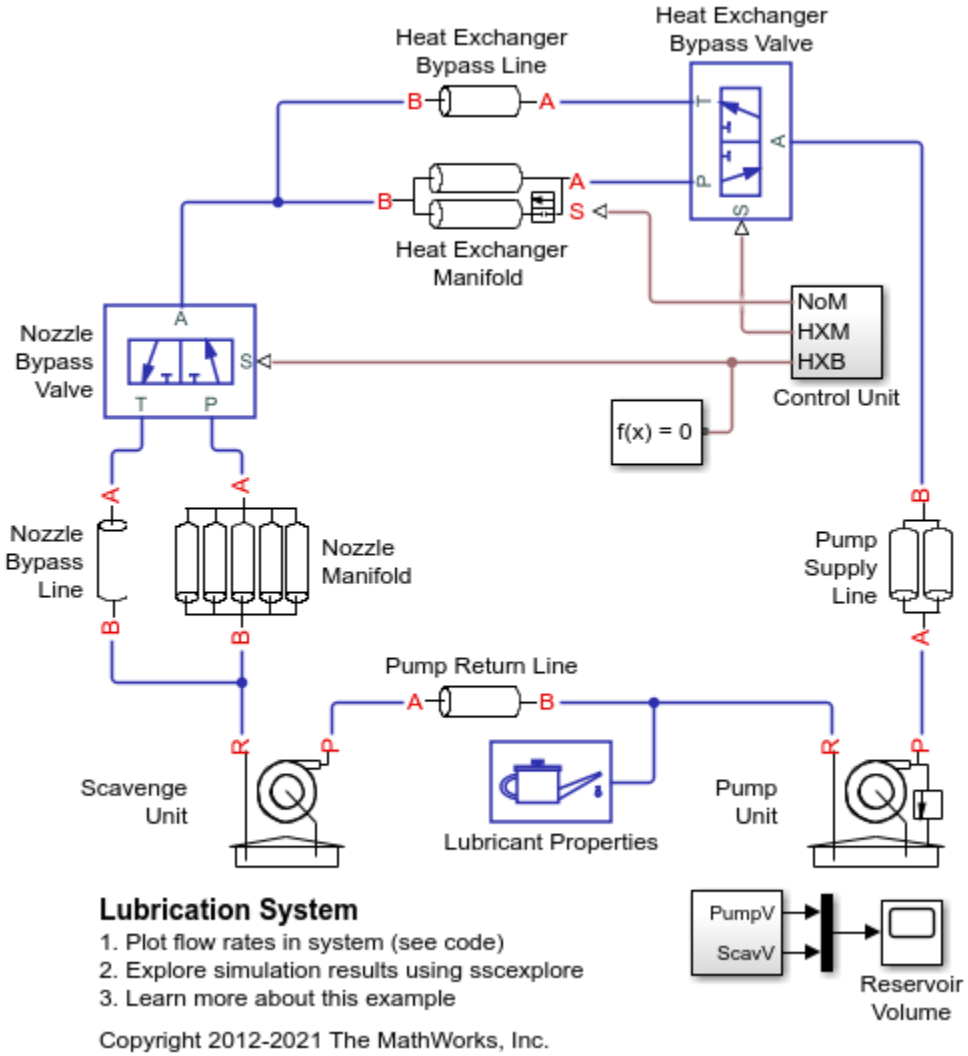


Lubrication System

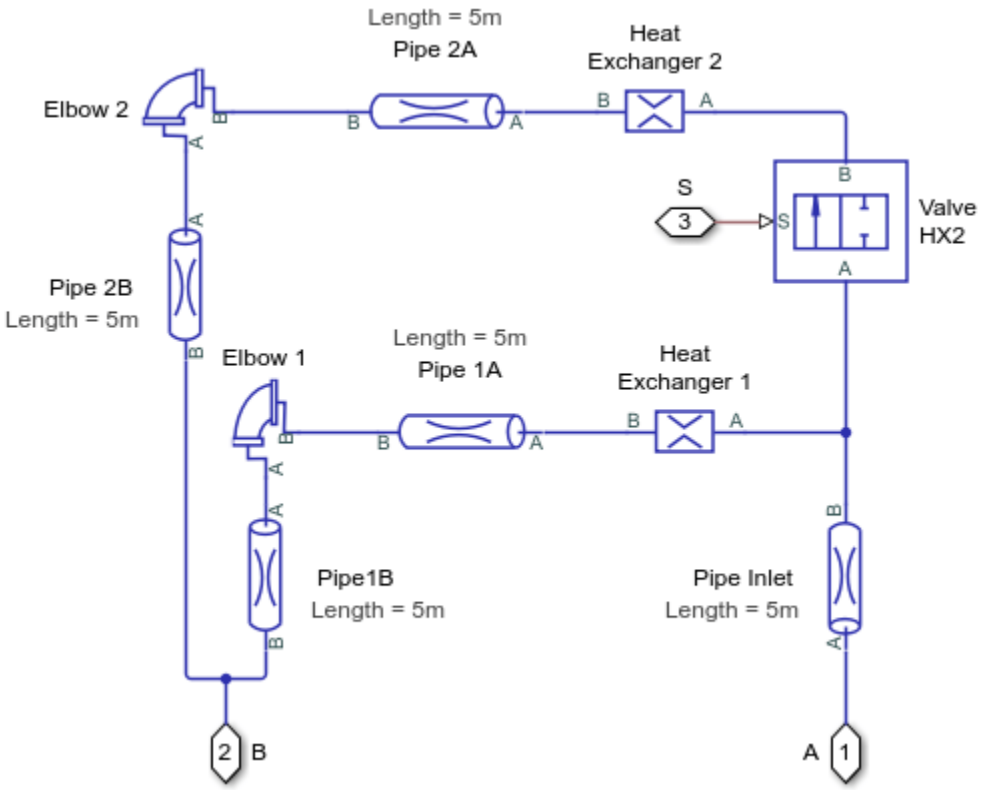
This example shows a simplified version of a lubrication system fed with the centrifugal pump. The system consists of five major units: Pump Unit, Scavenge Unit, Heat Exchanger Manifold, Nozzle Manifold, and Control Unit. Both the pump and the scavenge unit are built around the centrifugal pump. The Scavenge Unit collects fluid discharged by nozzles and pumps it back into the reservoir of the Pump Unit. The Control Unit generates commands to bypass either the heat exchanger, represented as a local resistance, or the nozzles block. In a real system, these commands are generated by temperature sensors installed in lubrication cavities.

The cycle simulated in the example allows you to observe pressures and flow rates at various points of the system. The example also shows the usage of local resistance blocks such as the Elbow, T-junction, and Sudden Area Change.

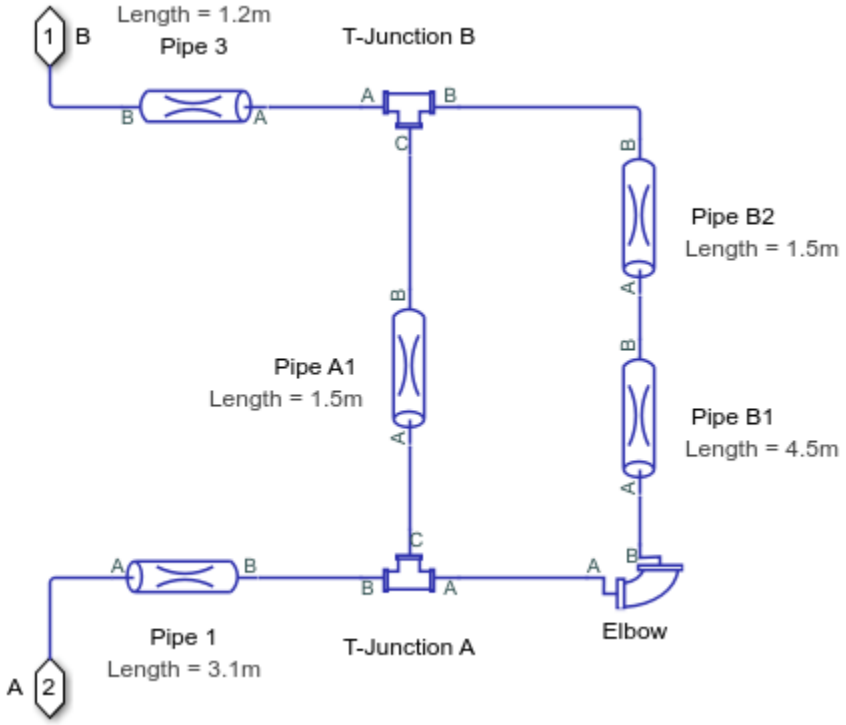
Model



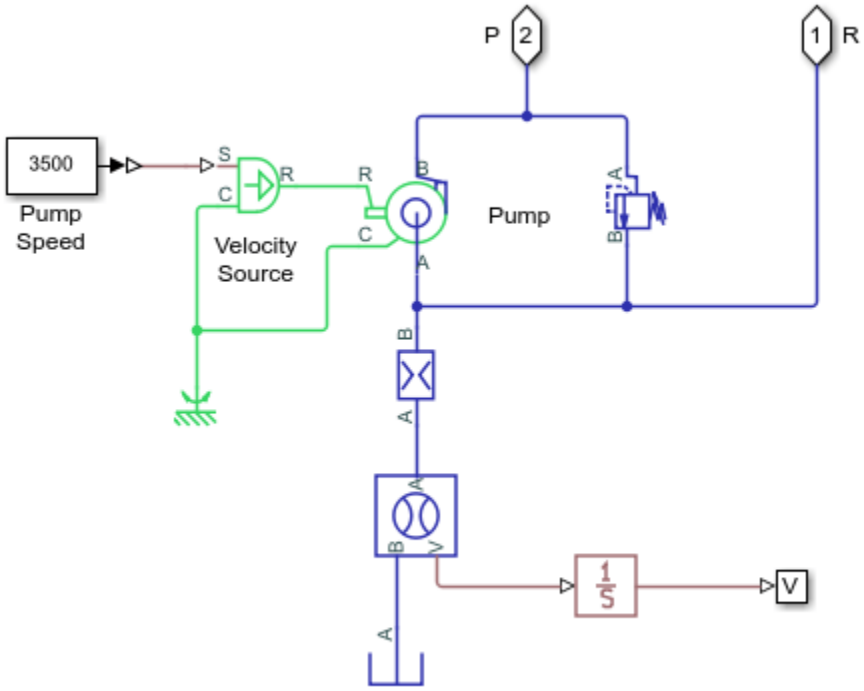
Heat Exchanger Manifold Subsystem



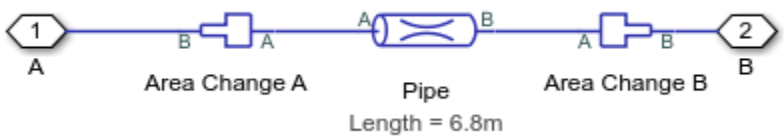
Pump Supply Line Subsystem



Pump Unit Subsystem

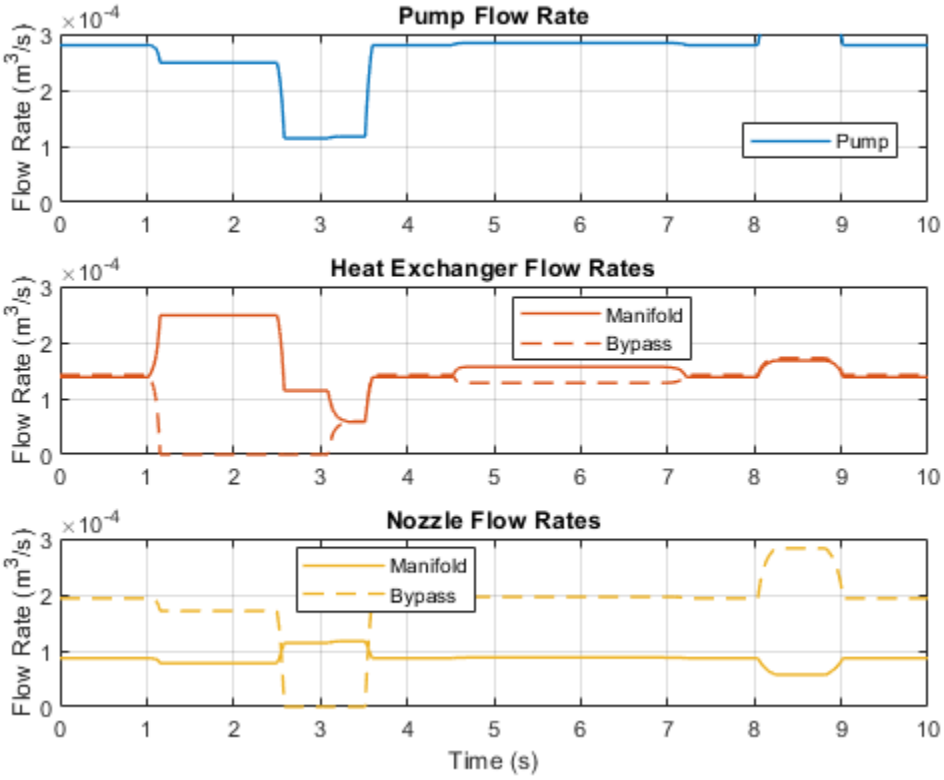


Pump Return Line Subsystem



Simulation Results from Simscape Logging

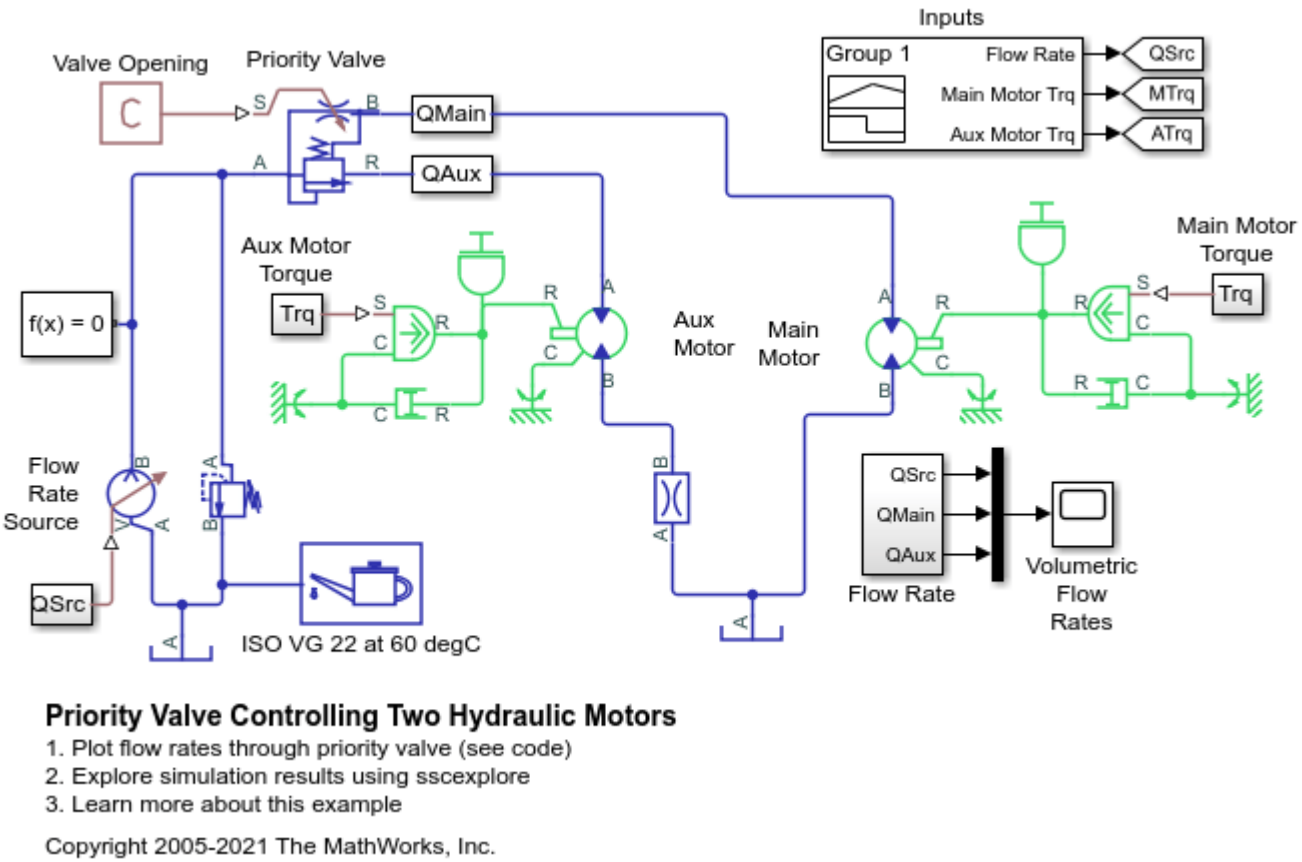
The plots below show the flow rates through different parts of the lubrication system. As the control member within the different valves are adjusted, flow rates change within the system.



Priority Valve Controlling Two Hydraulic Motors

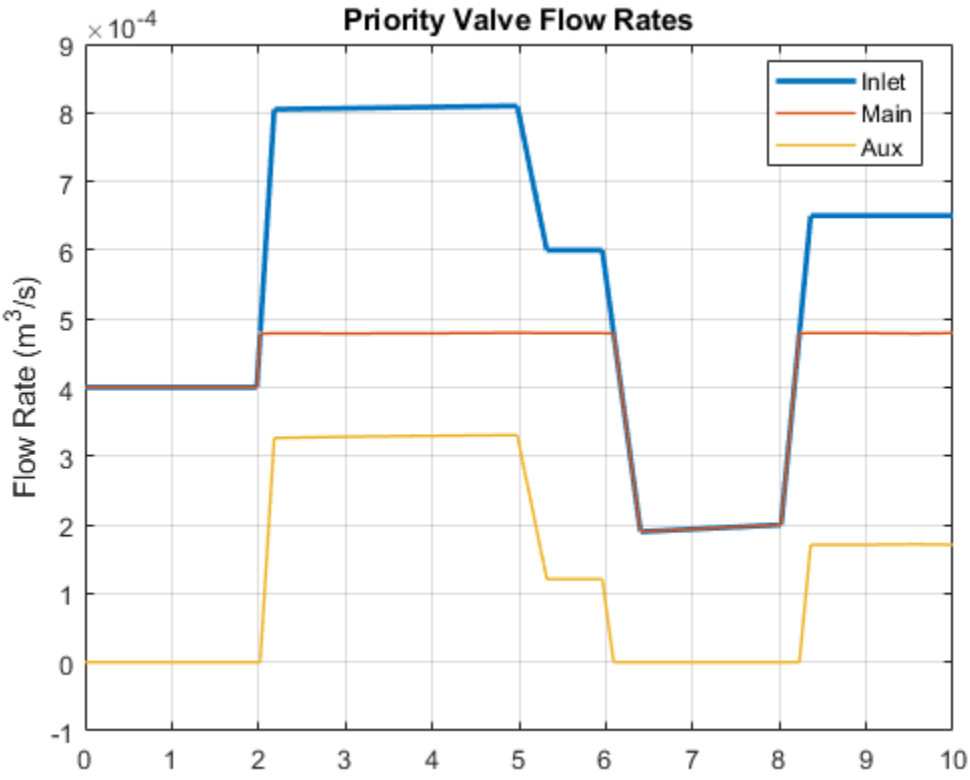
This example shows a pressure-compensated 3-way flow control valve. This valve maintains constant flow rate through the main hydraulic motor, which is connected to the pressure-compensated outlet of the flow control valve. It acts as a priority valve, diverting the excess flow to the auxiliary hydraulic motor if the main hydraulic motor receives enough fluid to maintain a preset angular velocity. The auxiliary motor is shut off completely if there is insufficient flow to power the main hydraulic motor.

Model



Simulation Results from Simscape Logging

The plot below shows the flow rates through the priority valve. As the flow rate at the inlet varies, the main motor is driven at the desired speed and excess flow is directed to the auxiliary hydraulic motor. When the inlet flow drops below the threshold to maintain the speed for the main hydraulic motor, the main motor slows down as well.


Priority Valve Flow Rates
×10^-4
Flow Rate (m^3/s)
Inlet
Main
Aux
9
8
7
6
5
4
3
2
1
0
-1
0
1
2
3
4
5
6
7
8
9
10

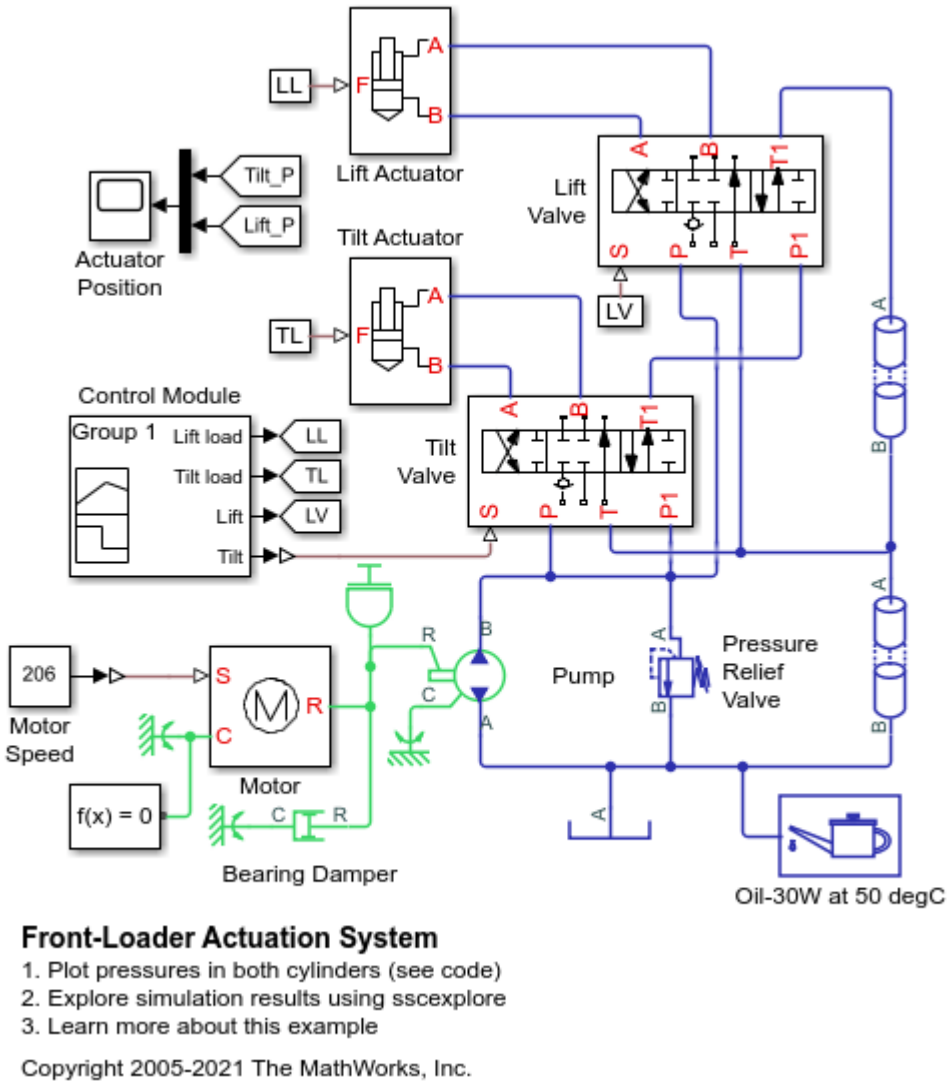
Front-Loader Actuation System

This example shows a simplified version of an actuation system consisting of the lift and tilt cylinders. Each cylinder is controlled by an open center, 6-way, 3-position directional valve. The valves are connected in series through their unloading branch such that the system pump is unloaded when both command levers are in neutral position. If either tilt or lift command is applied, the unloading path is closed.

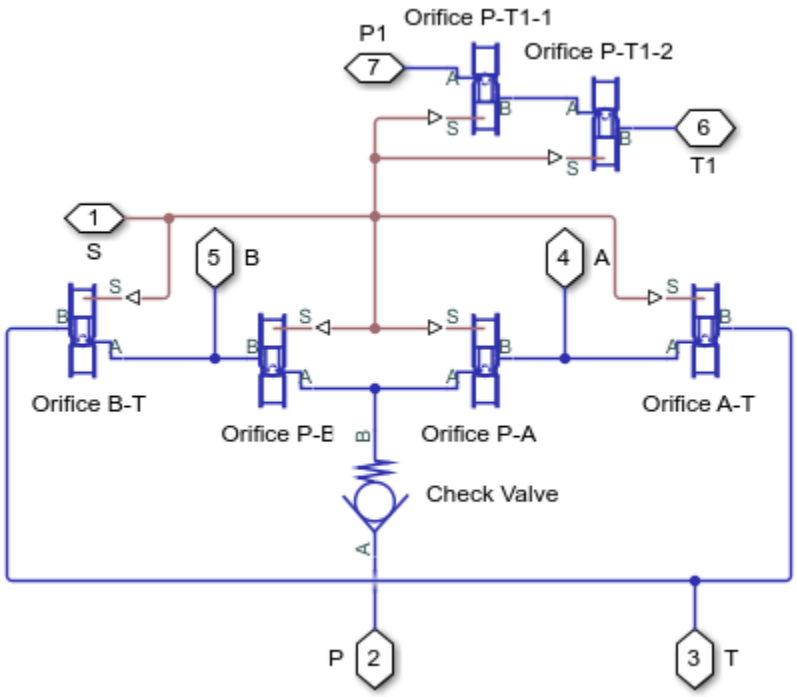
The pump is driven by a diesel engine simulated with a model that accounts for velocity change caused by the load on the output shaft. The directional valve is a custom model composed of Orifice with Variable Area Round Holes blocks and a Check Valve block.

A typical cycle of a front-loader actuation system is simulated with all the motion and load commands generated as a function of time.

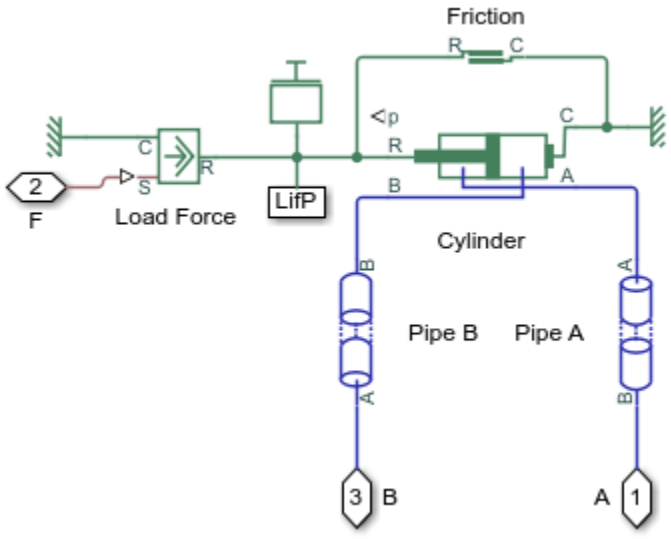
Model



Lift Valve Subsystem

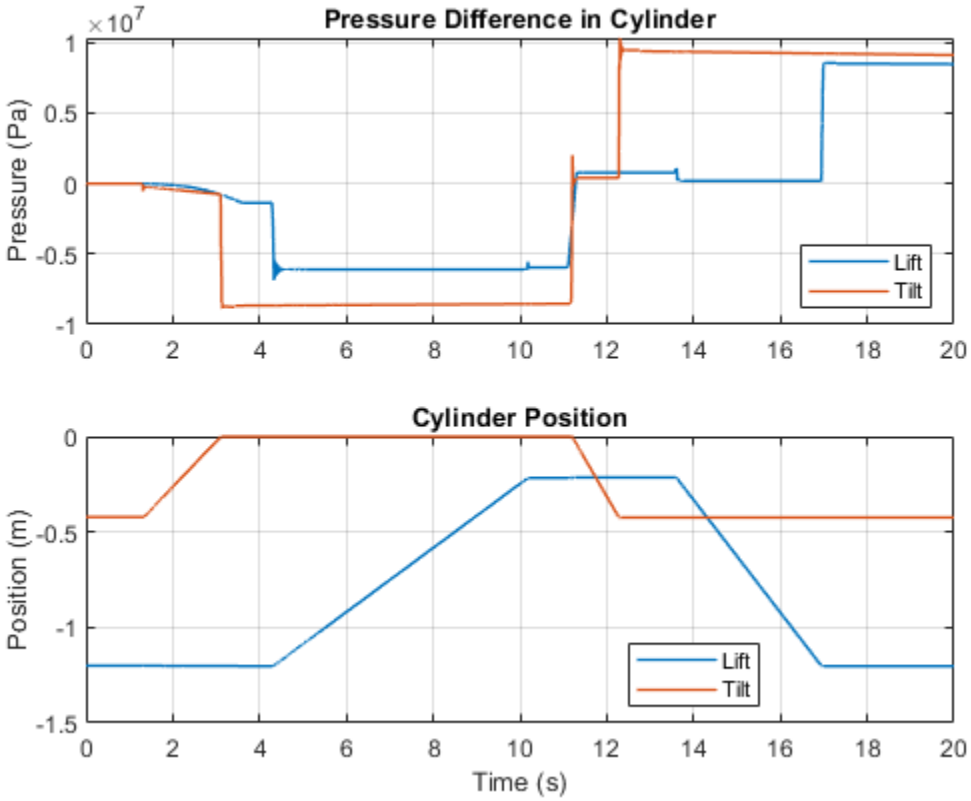


Lift Actuator Subsystem



Simulation Results from Simscape Logging

The plots below show the pressure difference in the lift and tilt cylinders and the position of the in each cylinder.


Pressure Difference in Cylinder
×10^7
Pressure (Pa)
1
0.5
0
-0.5
-1
0
2
4
6
8
10
12
14
16
18
20
Lift
Tilt
Cylinder Position
Position (m)
0
-0.5
-1
-1.5
Time (s)

Drill-Ream Actuator

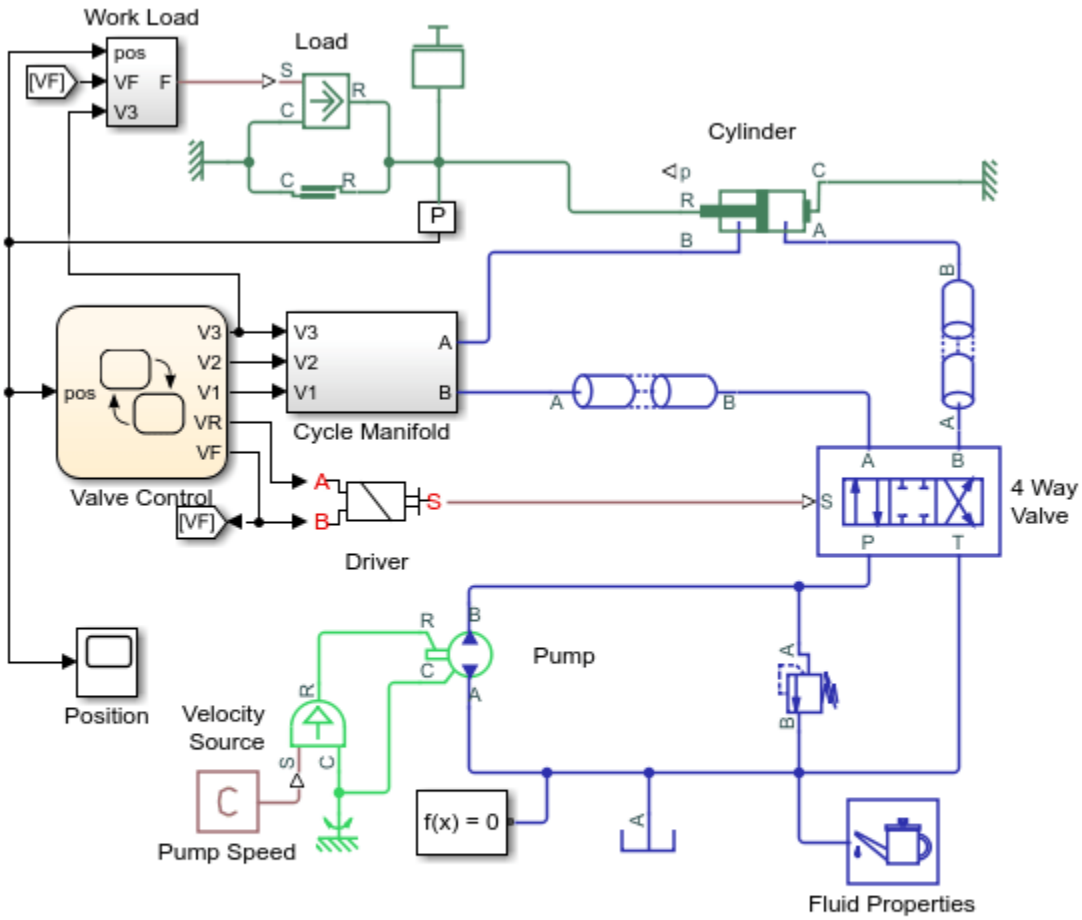
This example shows an actuator that drives a machine tool working unit performing a sequence of three technological operations: coarse drilling, fine drilling, and reaming. The actuator speed is controlled by one of three pressure-compensated flow control valves metering out return flow from the cylinder. The selection of an appropriate flow control is performed by directional valves that are activated by a control unit.

From the Valve Control, signals VF and VR are the signals that activate forward and return motions, respectively, by shifting the main directional valve. Signal V1 activates normally- closed valve Valve1 Retract. When opened, Valve1 Retract bypasses all flow controls and sets the fast traverse speed. V2 controls valve Valve2 Ream. When deenergized, this valve directs return flow through Flow Control Reaming, which is set to reaming speed. Signal V3 directs flow either through Flow Control Fine Drilling or Flow Control Coarse Drilling.

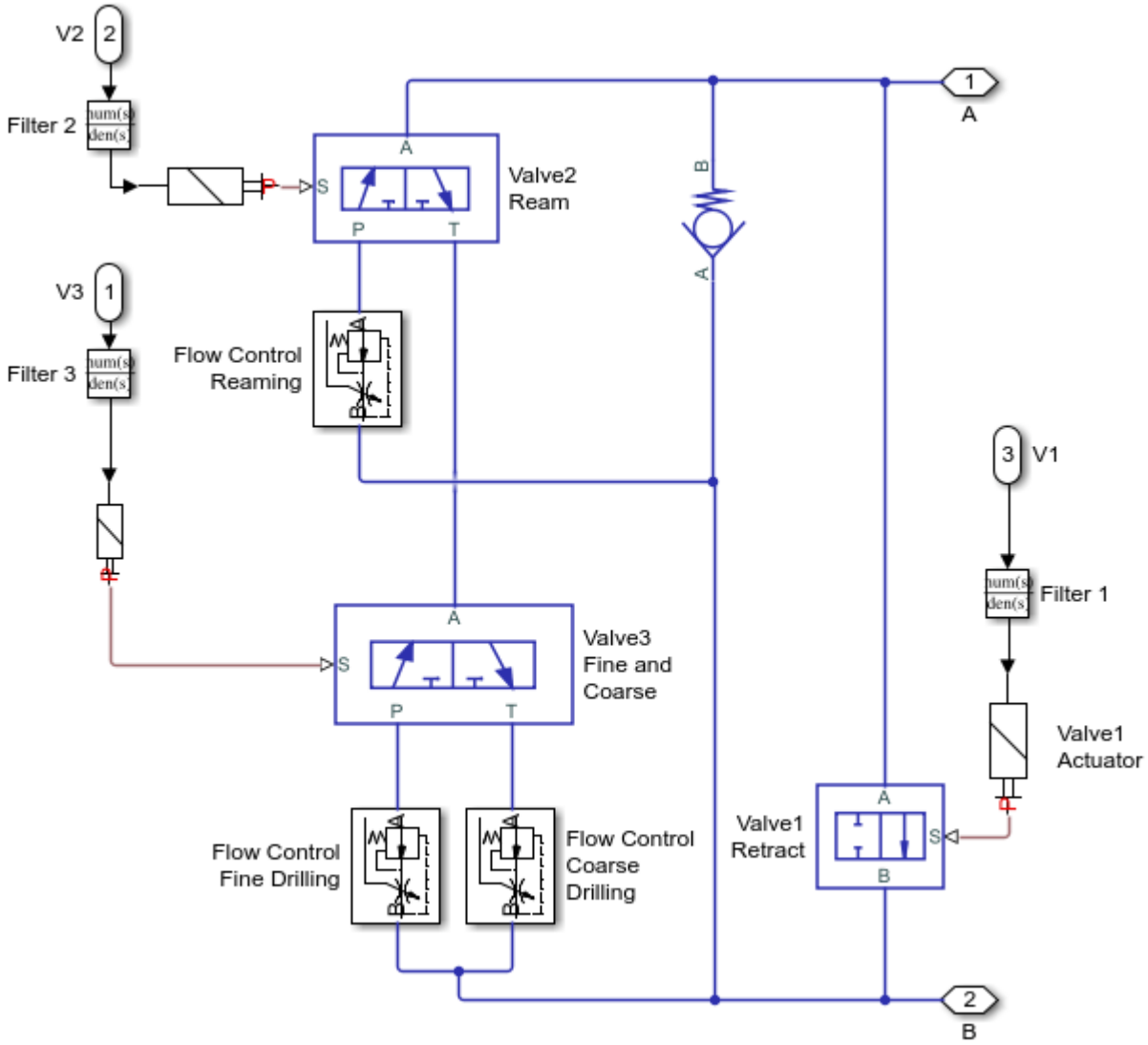
The load is simulated as a position-dependent force using a table lookup block.

Flow control valves are arranged as a subsystem comprised of a fixed orifice and a pressure compensator.

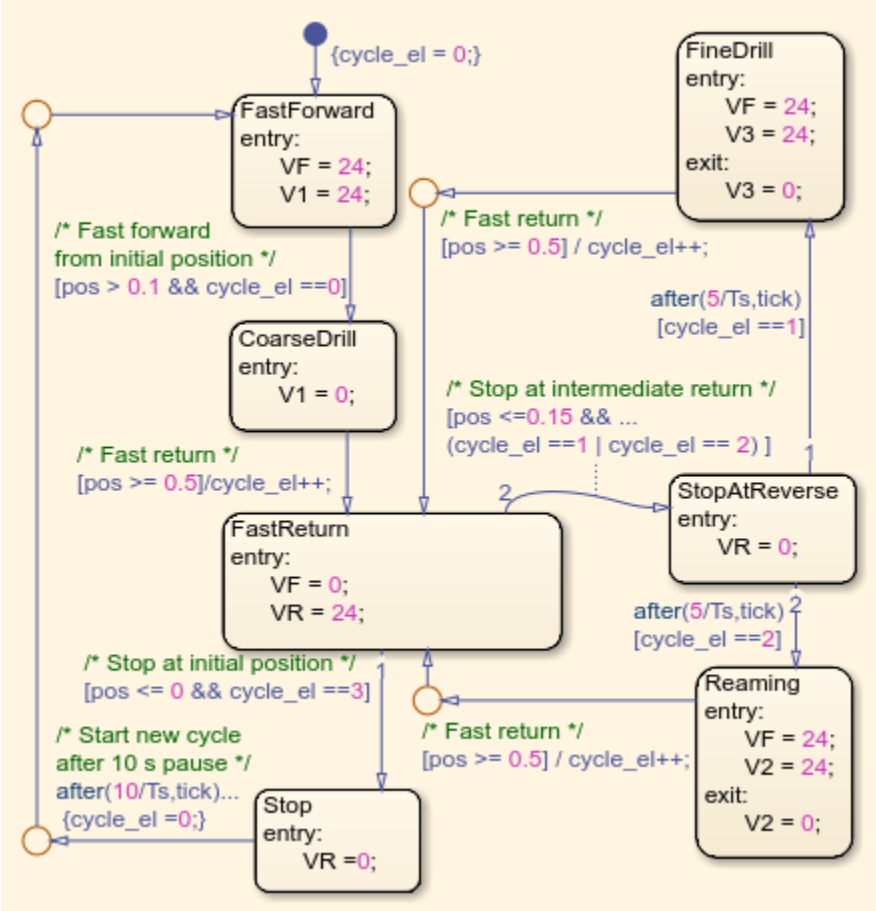
Model


Work Load
pos
VF
V3
F
[VF]
Load
S
R
C
Cylinder
p
P
V3
V2
V1
VR
VF
pos
Valve Control
Cycle Manifold
A
B
4 Way
Valve
S
P
T
Driver
Pump
Position
Velocity
Source
Pump Speed
f(x) = 0
Fluid Properties
Drill-Ream Actuator
1. Plot pressures in cylinder (see code)
2. Plot valve states (see code)
3. Explore simulation results using sscexplore
4. Learn more about this example
Copyright 2005-2021 The MathWorks, Inc.

Cycle Manifold Subsystem

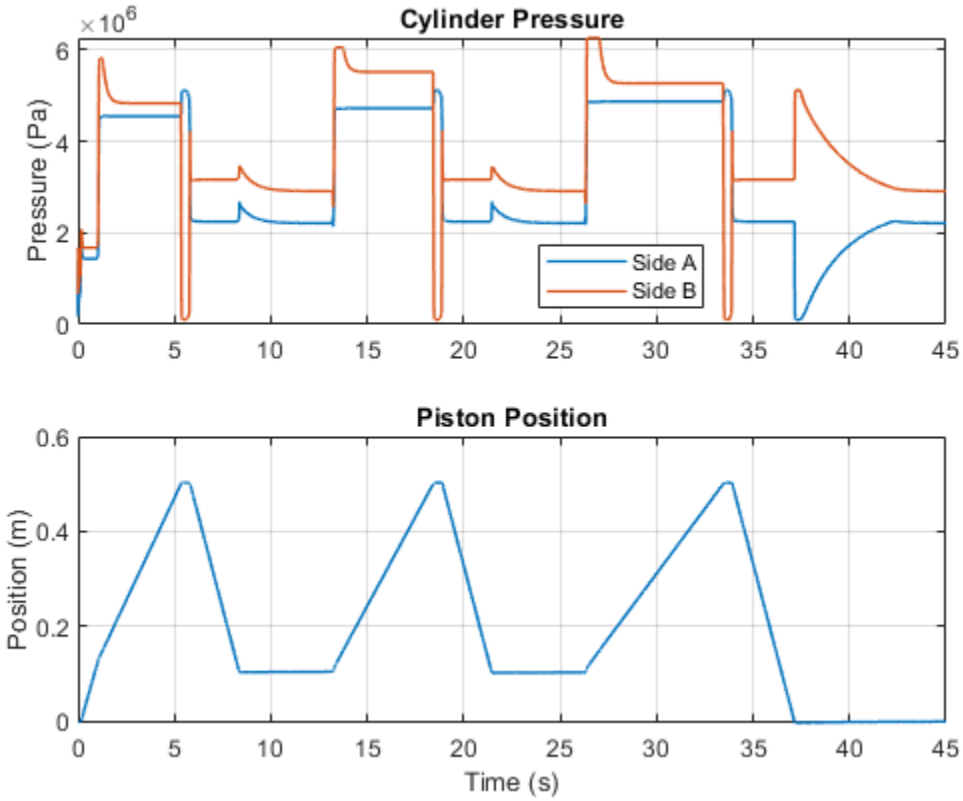


Valve Control Subsystem

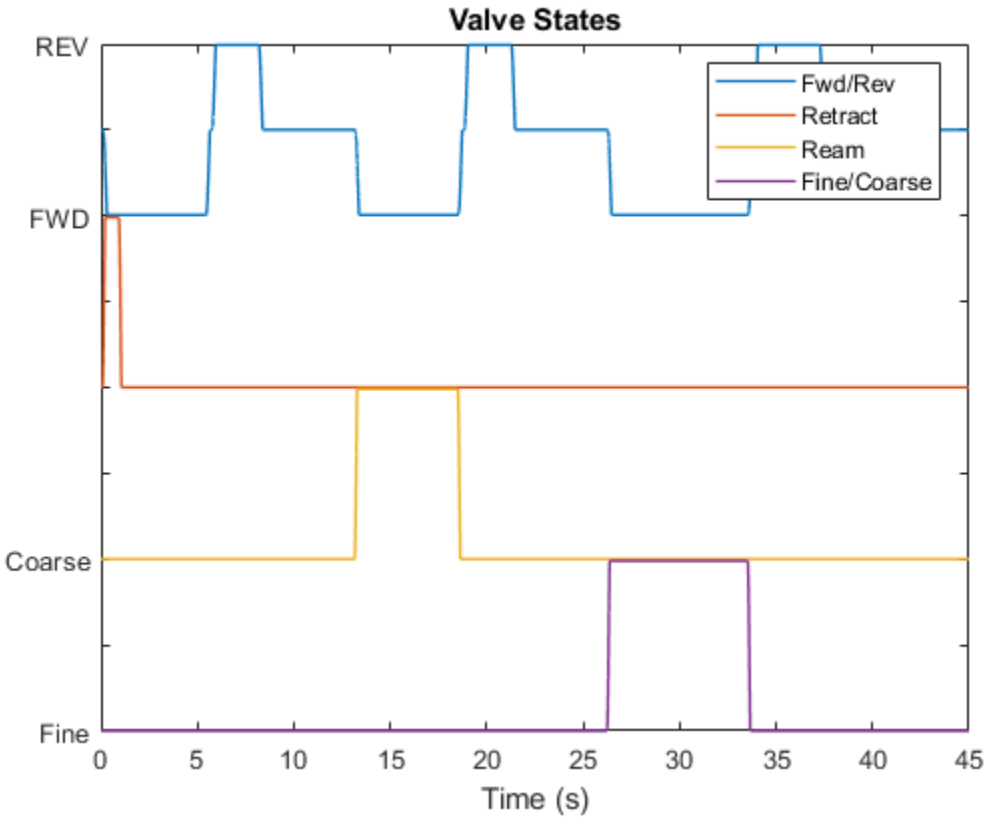


Simulation Results from Simscape Logging

The plots below show the pressures in the cylinder and the position of the piston. Three operating speeds are shown in the plots.



The plots below show the states of the different valves. Forward/Reverse has 3 states (Forward/Locked/Reverse), while the others have two states. Retract and Ream have states of on/off, and Fine/Coarse valve selects between two different flow restrictions.


Valve States
REV
FWD
Coarse
Fine
0
5
10
15
20
25
30
35
40
45
Time (s)
Fwd/Rev
Retract
Ream
Fine/Coarse

Electric Vehicle Thermal Management

This example models the thermal management system of a battery electric vehicle. The system consists of two coolant loops, a refrigeration loop, and a cabin HVAC loop. The thermal load are the batteries, powertrain, and cabin.

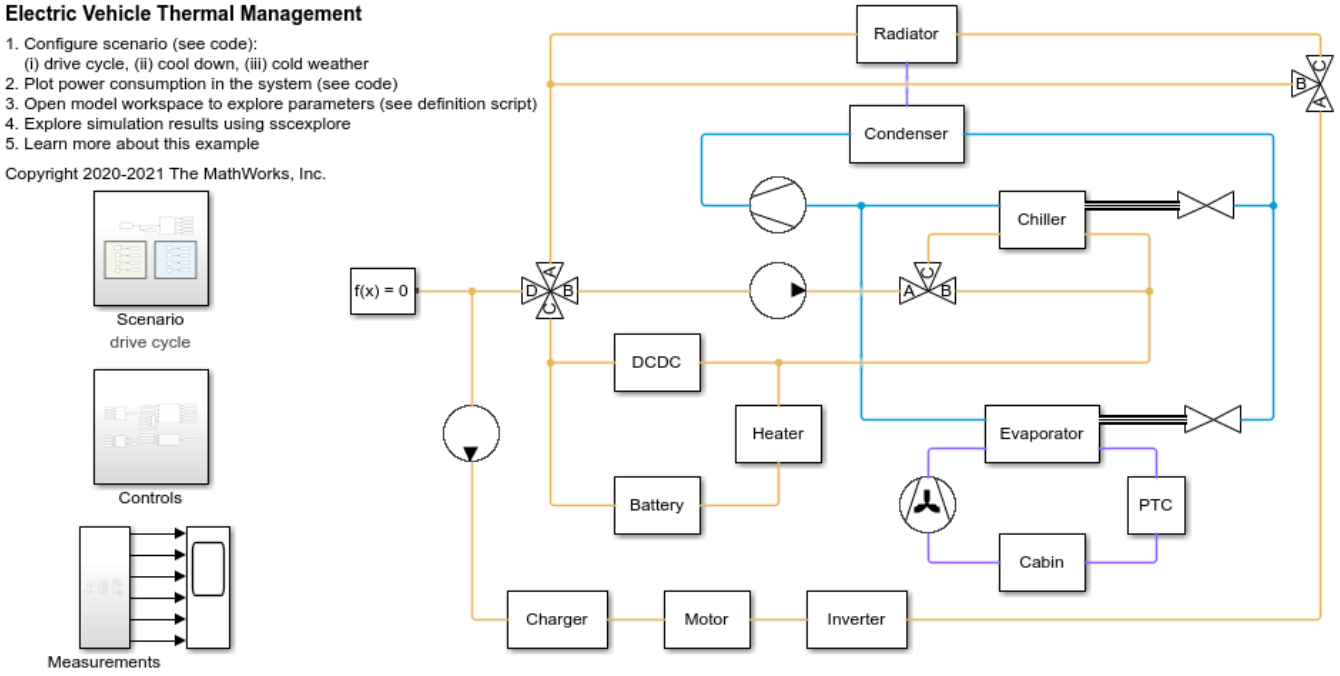
The two coolant loops can be joined together in serial mode or kept separate in parallel mode using the 4-way valve. In cold weather, the coolant loops are in serial mode so that heat from the motor warms the batteries. If necessary, a heater can provide additional heat. In warm weather, the coolant loops remain in serial mode and both the batteries and the powertrain are cooled by the radiator. In hot weather, the coolant loop switches to parallel mode and separates. One loop cools the powertrain using the radiator. The other cools the batteries using the chiller in the refrigeration loop.

The refrigeration loop consists of a compressor, a condenser, a liquid receiver, two expansion valves, a chiller, and an evaporator. The chiller is used to cool the coolant in hot weather when the radiator alone is insufficient. The evaporator is used to cool the vehicle cabin when air conditioning is turned on. The compressor is controlled such that the condenser can dissipate the heat absorbed by either or both the chiller and the evaporator.

The HVAC loop consists of a blower, an evaporator, a PTC heater, and the vehicle cabin. The PTC heater provides heating in cold weather; the evaporator provides air conditioning in hot weather. The blower is controlled to maintain the specified cabin temperature setpoint.

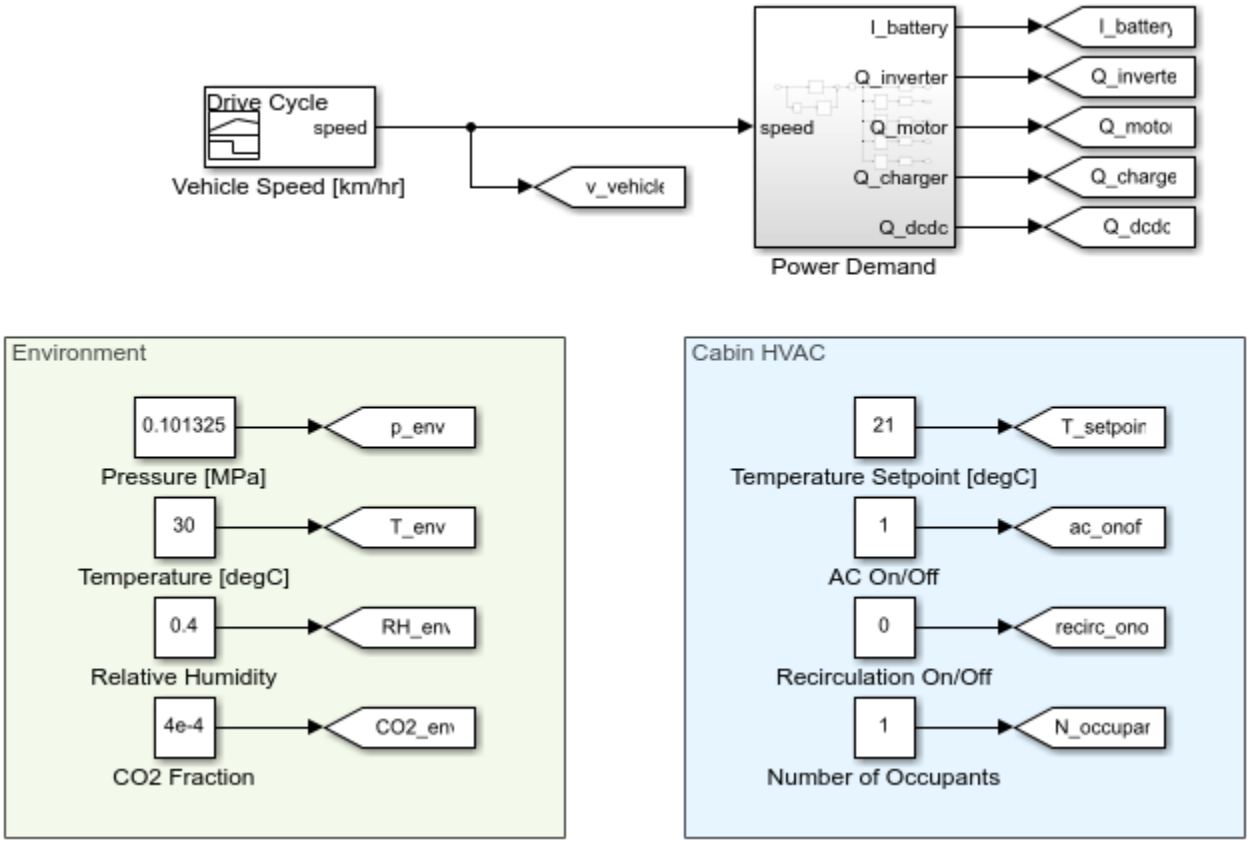
This model has three scenarios set up. The drive cycle scenario simulates driving conditions in 30 degC weather with air conditioning on. The vehicle speed is based on the NEDC followed by 30 min of high speed to push the battery heat load. The cool down scenario simulates a stationary vehicle in 40 degC weather with air conditioning on. Finally, the cold weather scenario simulates driving conditions in -10 degC weather, which requires the battery heater and PTC heater to warm up the batteries and cabin, respectively.

Model



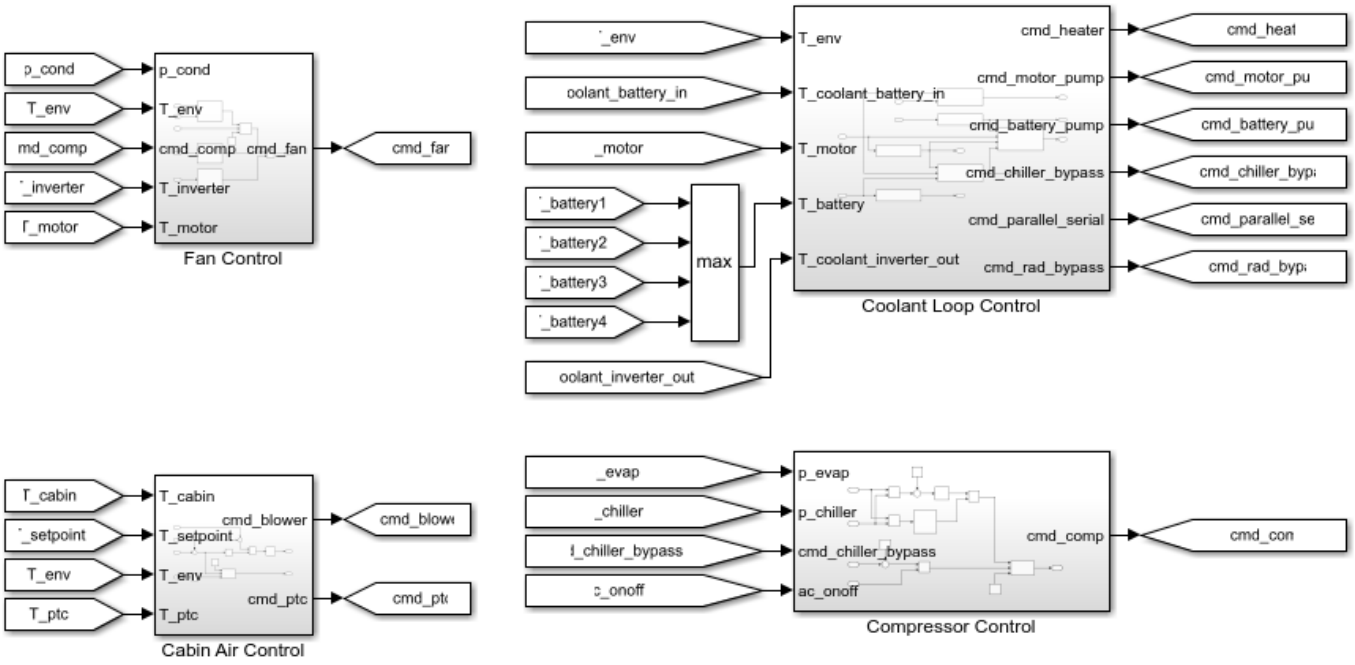
Scenario Subsystem

This subsystem sets up the environment conditions and inputs to the system for the selected scenario. The battery current demand and powertrain heat load are a function of the vehicle speed based on tabulated data.



Controls Subsystem

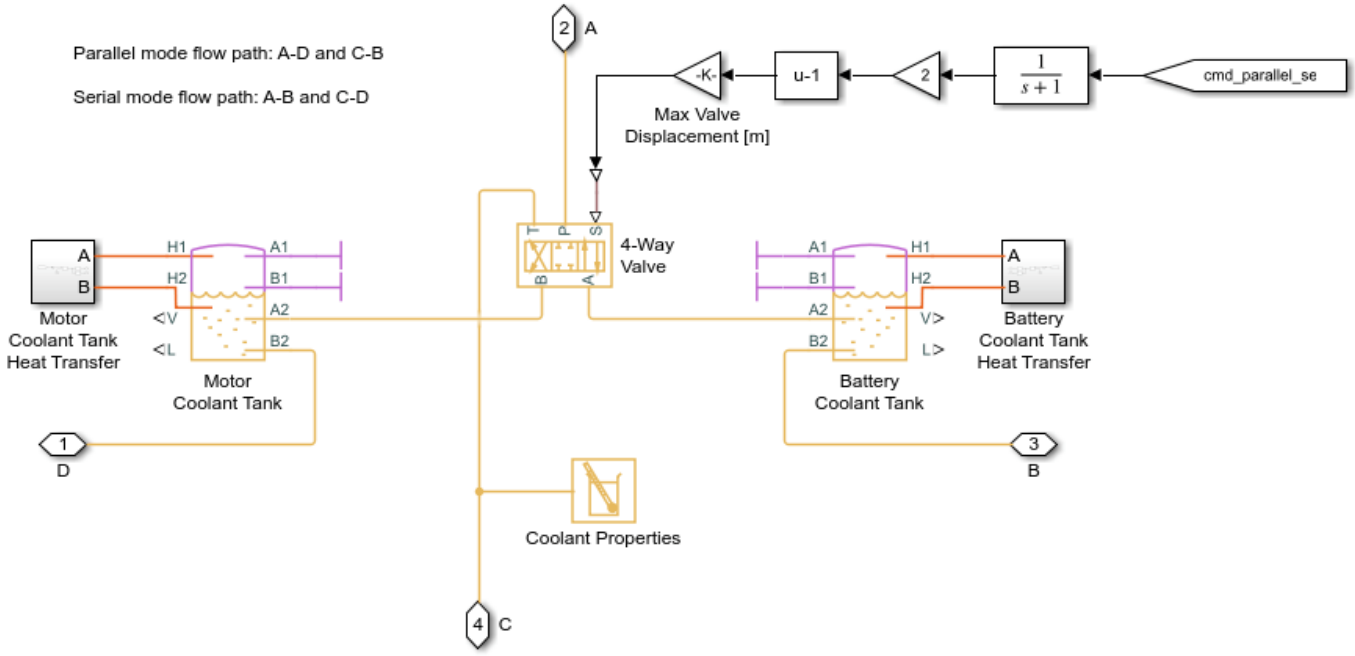
This subsystem consists of all of the controllers for the pumps, compressor, fan, blower, and valves in the thermal management system.



Parallel-Serial Mode Valve Subsystem

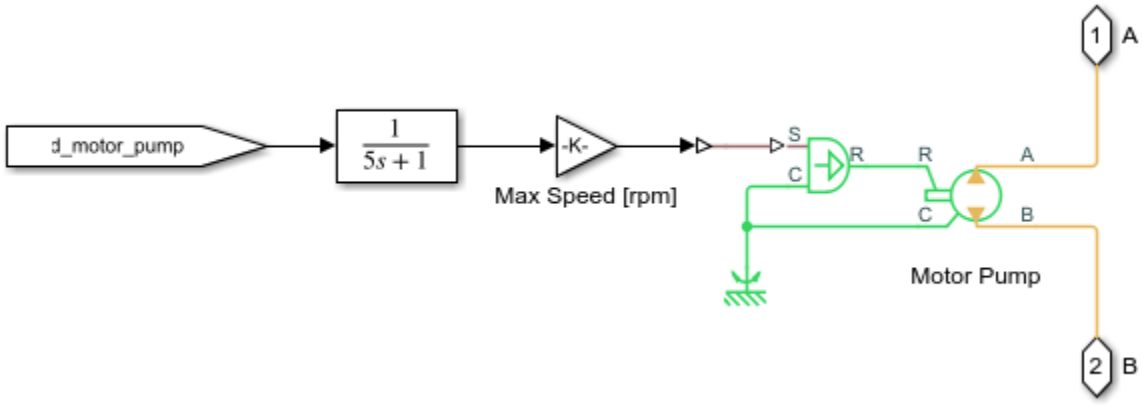
The 4-way valve in this subsystem controls whether the coolant loop operates in parallel or serial mode. When ports A and D are connected and ports C and B are connected, it is in parallel mode. The two coolant loops are separated with their own coolant tanks and pumps.

When ports A and B are connected and ports C and D are connected, it is in serial mode. The two coolant loops are merged and the two pumps are synchronized to provide the same flow rate.



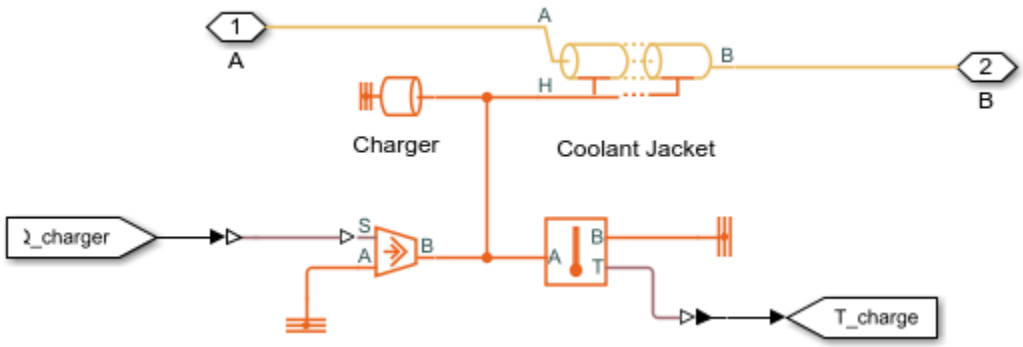
Motor Pump Subsystem

This pump drives the coolant loop that cools the charger, motor, and inverter.



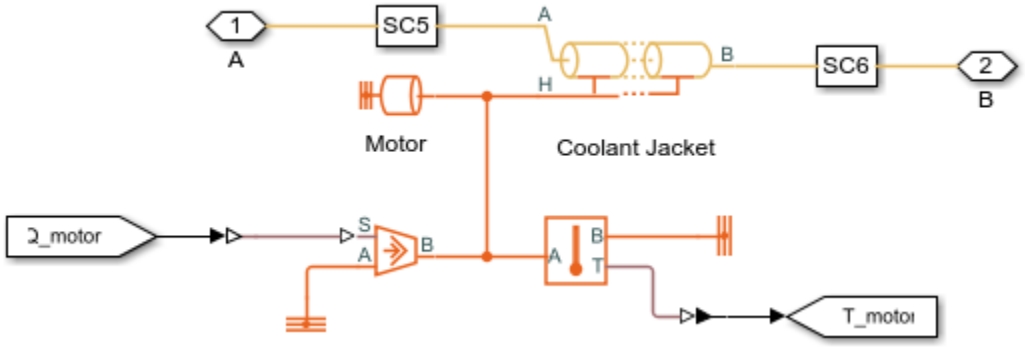
Charger Subsystem

This subsystem models a coolant jacket around the charger, which is represented by a heat flow rate source and a thermal mass.



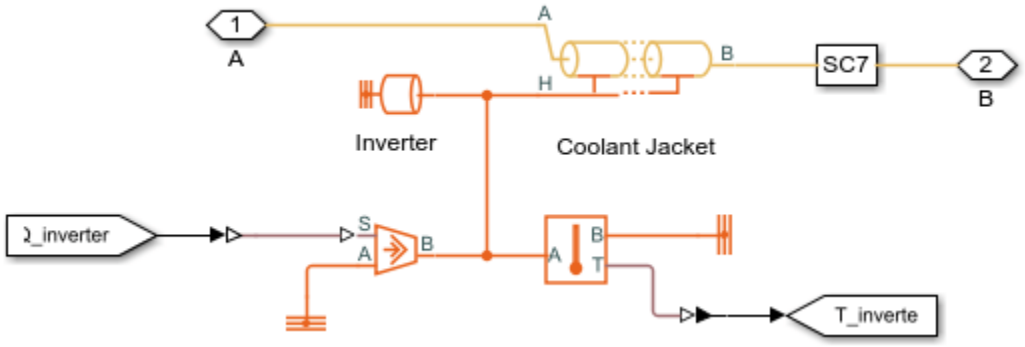
Motor Subsystem

This subsystem models a coolant jacket around the motor, which is represented by a heat flow rate source and a thermal mass.



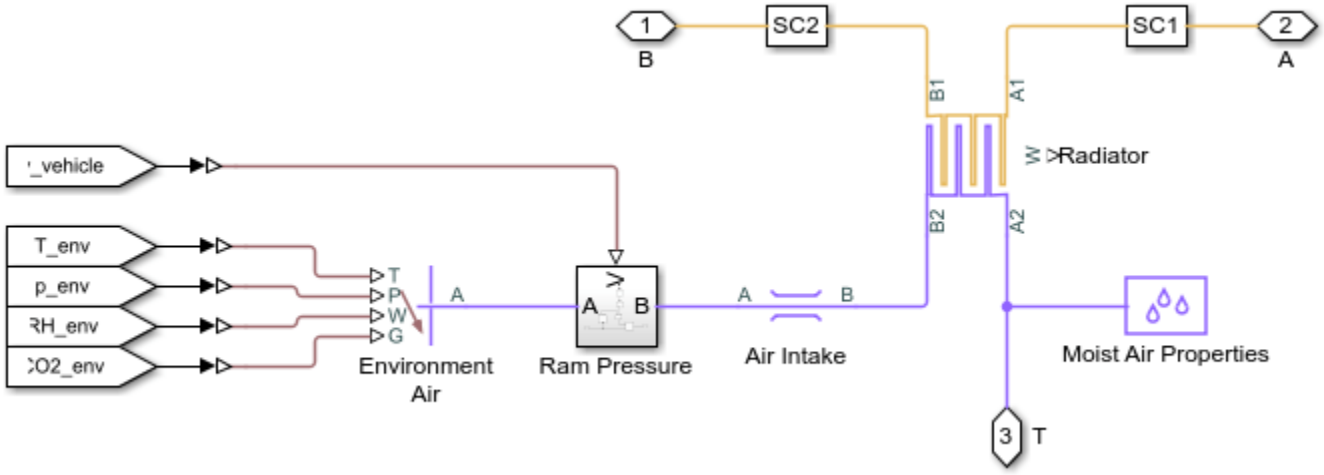
Inverter Subsystem

This subsystem models a coolant jacket around the inverter, which is represented by a heat flow rate source and a thermal mass.



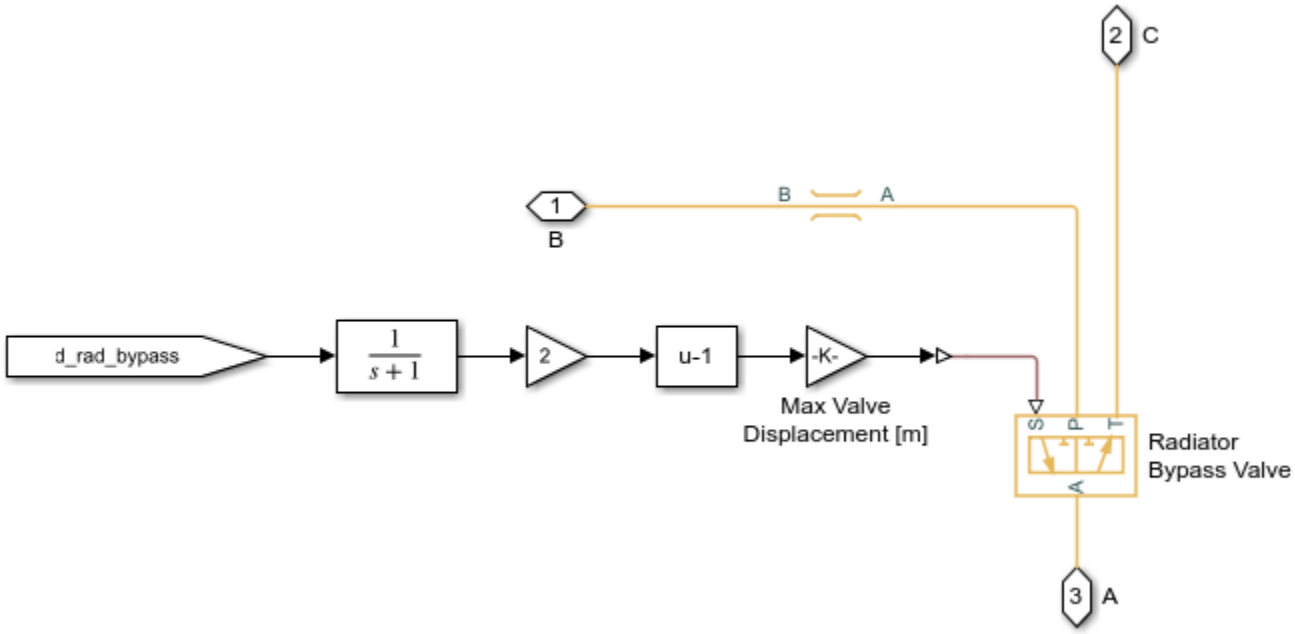
Radiator Subsystem

The radiator is a rectangular tube-and-fin type heat exchanger that dissipates coolant heat to the air. The air flow is driven by the vehicle speed and the fan located behind the condenser.



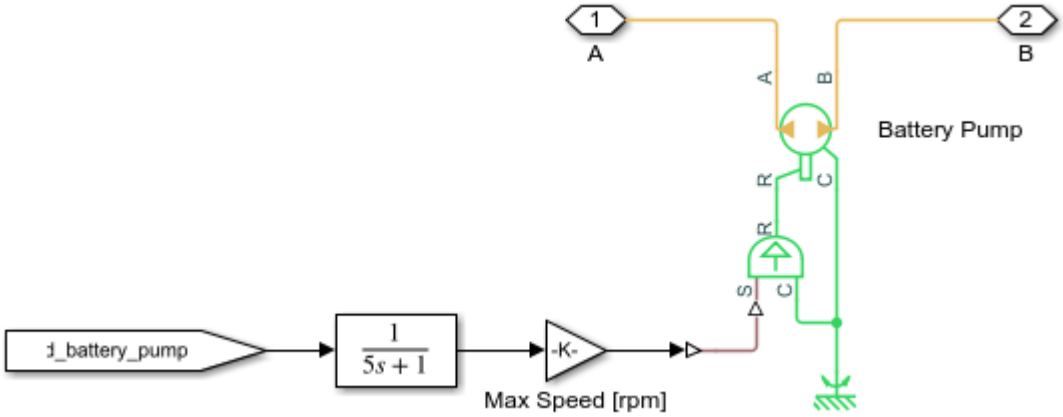
Radiator Bypass Valve Subsystem

In cold weather, the radiator is bypassed so that heat from the powertrain can be used to warm up the batteries. This is controlled by the the 3-way valve that either sends coolant to the radiator or bypasses the radiator.



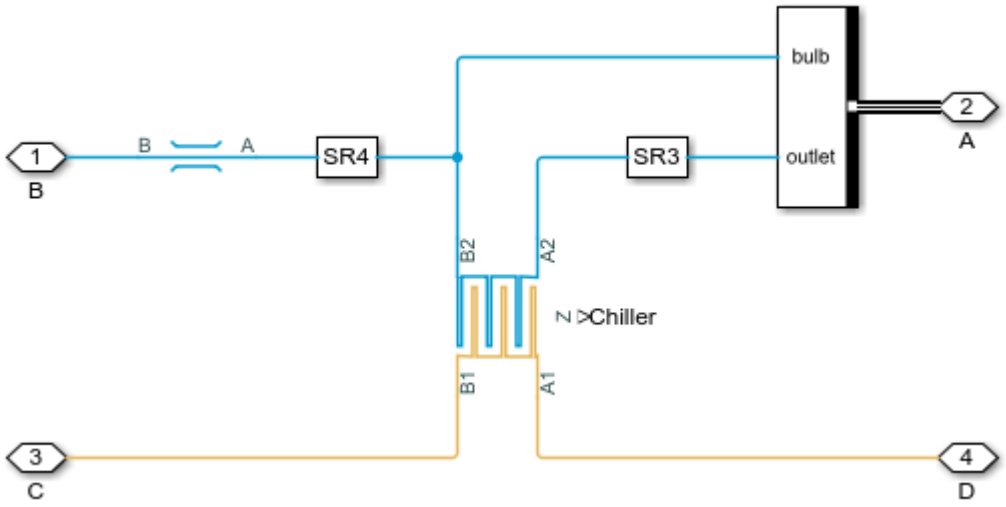
Battery Pump Subsystem

This pump drives the coolant loop that cools the batteries and the DC-DC converter.



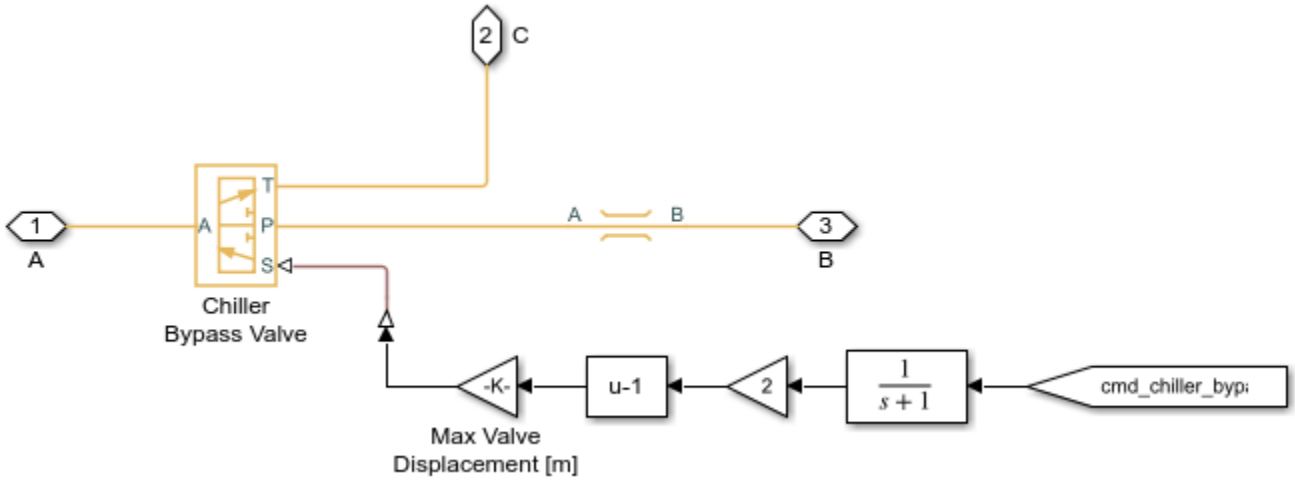
Chiller Subsystem

The chiller is assumed to be a shell-and-tube type heat exchanger that lets the refrigerant absorb heat from the coolant.



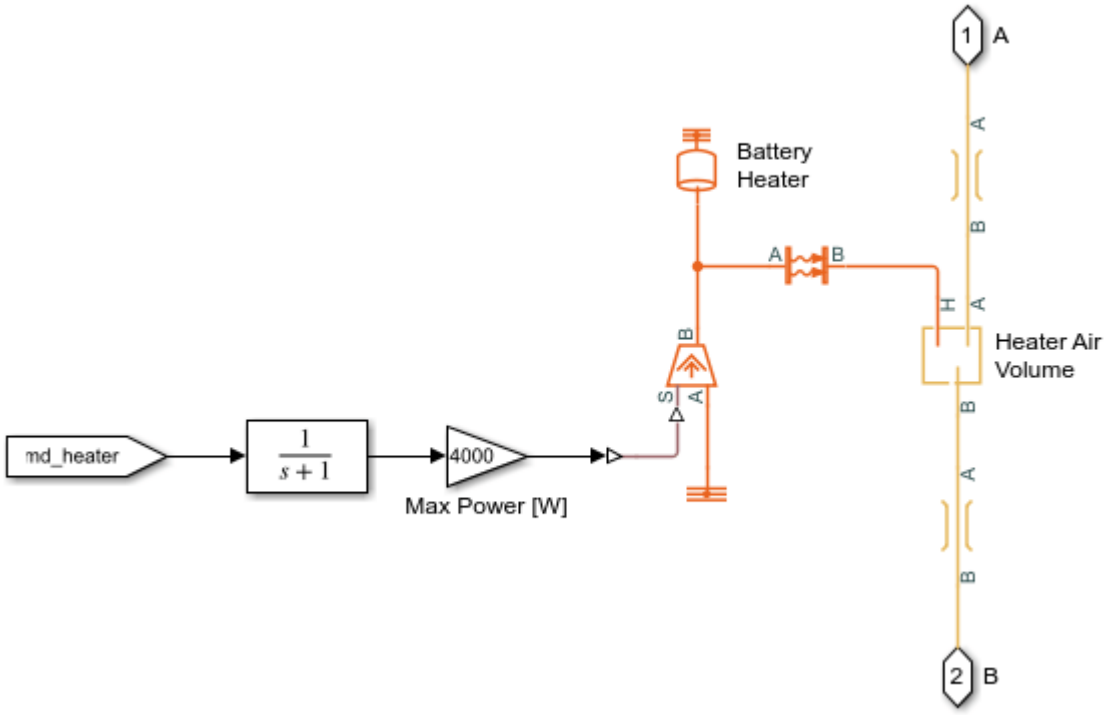
Chiller Bypass Valve Subsystem

The chiller operates in an on-off manner depending on the battery temperature. This is controlled by the the 3-way valve that either sends coolant to the chiller or bypasses the chiller.



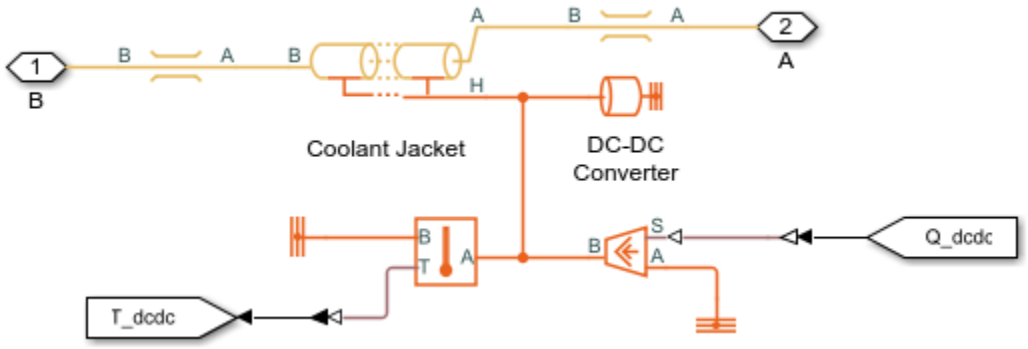
Heater Subsystem

The battery heater is modeled as a heat flow rate source and a thermal mass. It is turned on in cold weather to bring the battery temperature above 5 degC.



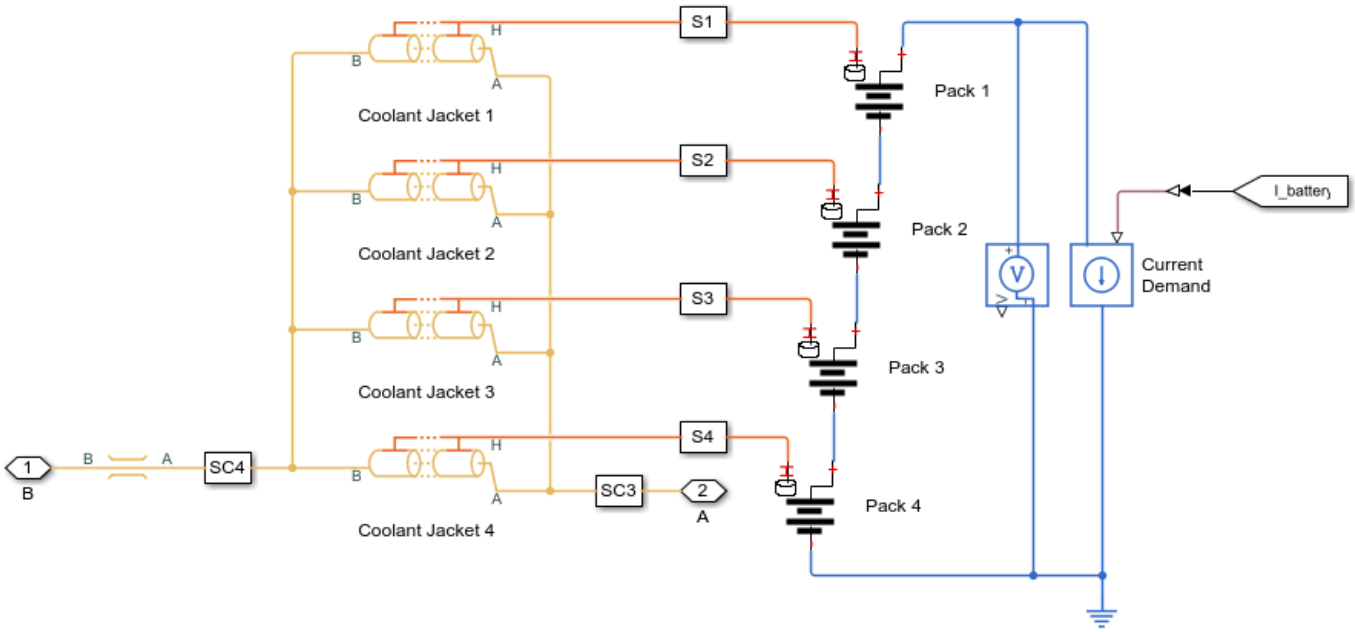
DCDC Subsystem

This subsystem models a coolant jacket around the DC-DC converter, which is represented by a heat flow rate source and a thermal mass.



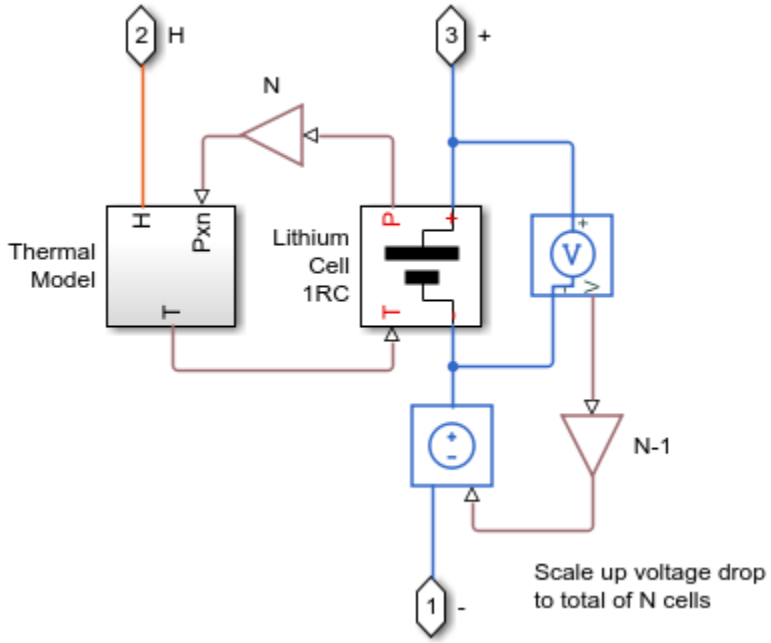
Battery Subsystem

The batteries are modeled as four separate packs surrounded by a coolant jacket. The battery packs generate voltage and heat based on the current demand. The coolant is assumed to flow in narrow channels around the battery packs.



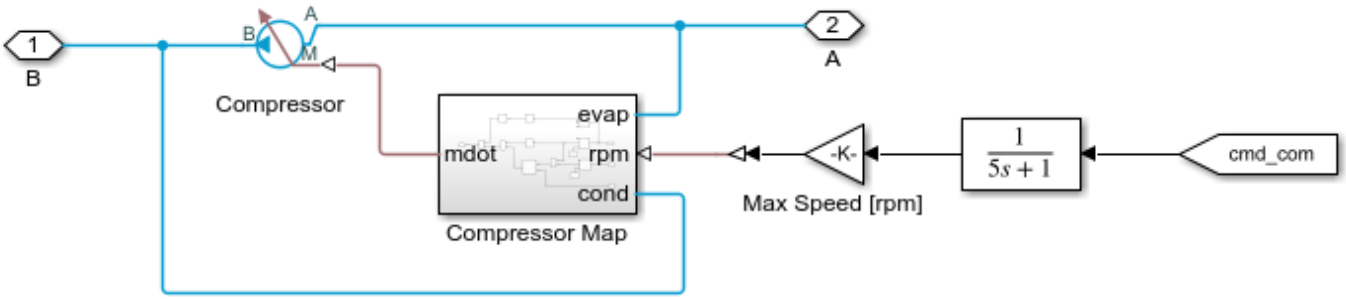
Pack 1 Subsystem

Each battery pack is modeled as a stack of lithium-ion cells coupled with a thermal model. Heat is generated based on the power losses in the cells.



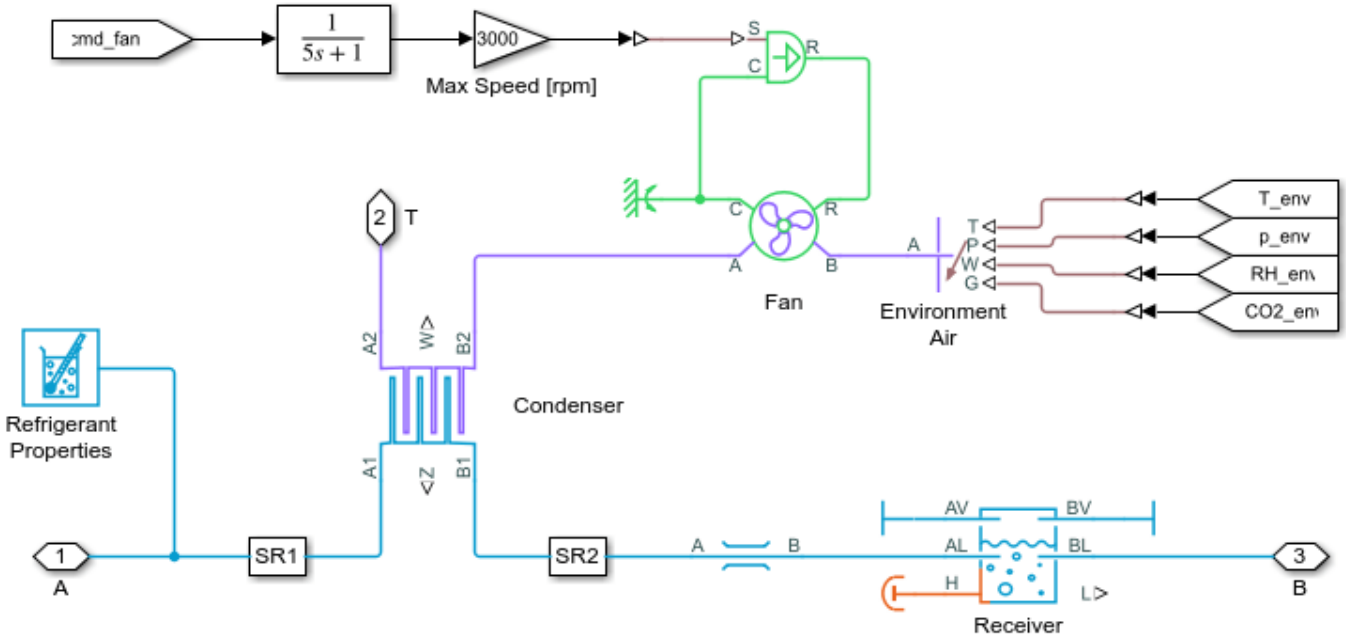
Compressor Subsystem

The compressor drives the flow in the refrigerant loop. It is controlled to maintain a pressure of 0.3 MPa in the chiller and the evaporator, which corresponds to a saturation temperature of around 1 degC.



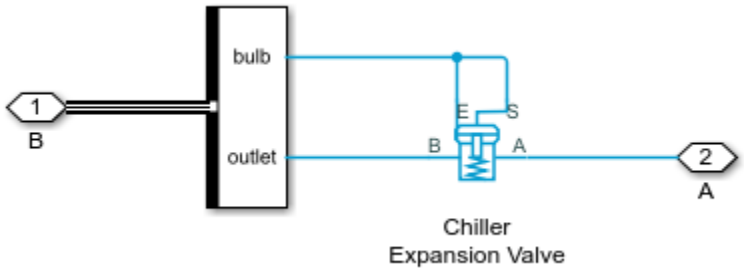
Condenser Subsystem

The condenser is a rectangular tube-and-fin type heat exchanger that dissipates refrigerant heat to the air. The air flow is driven by the vehicle speed and the fan. The liquid receiver provides storage for the refrigerant and permits only subcooled liquid to flow into the expansion valves.



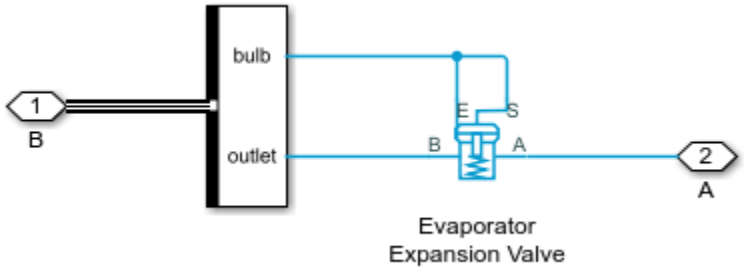
Chiller Expansion Valve Subsystem

This expansion valve meters refrigerant flow to the chiller to maintain a nominal superheat.



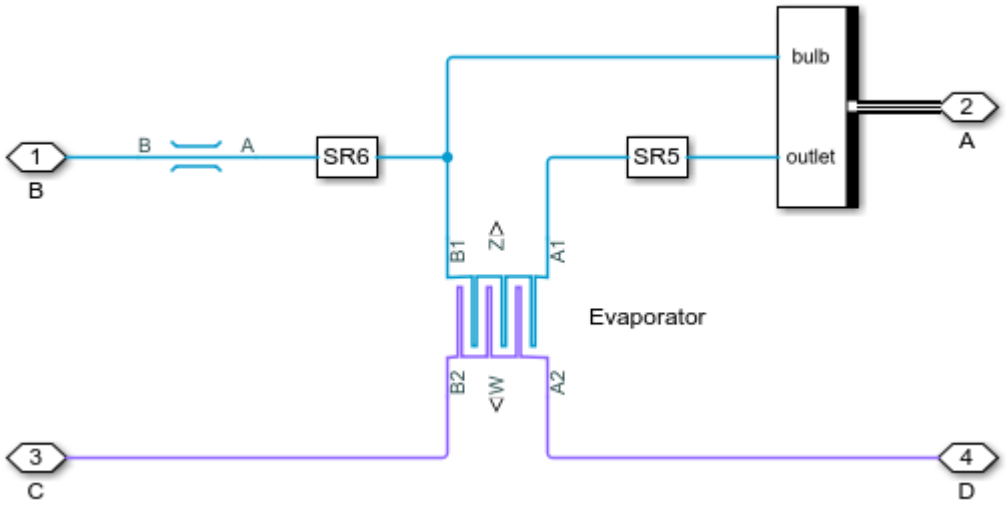
Evaporator Expansion Valve Subsystem

This expansion valve meters refrigerant flow to the evaporator to maintain a nominal superheat.



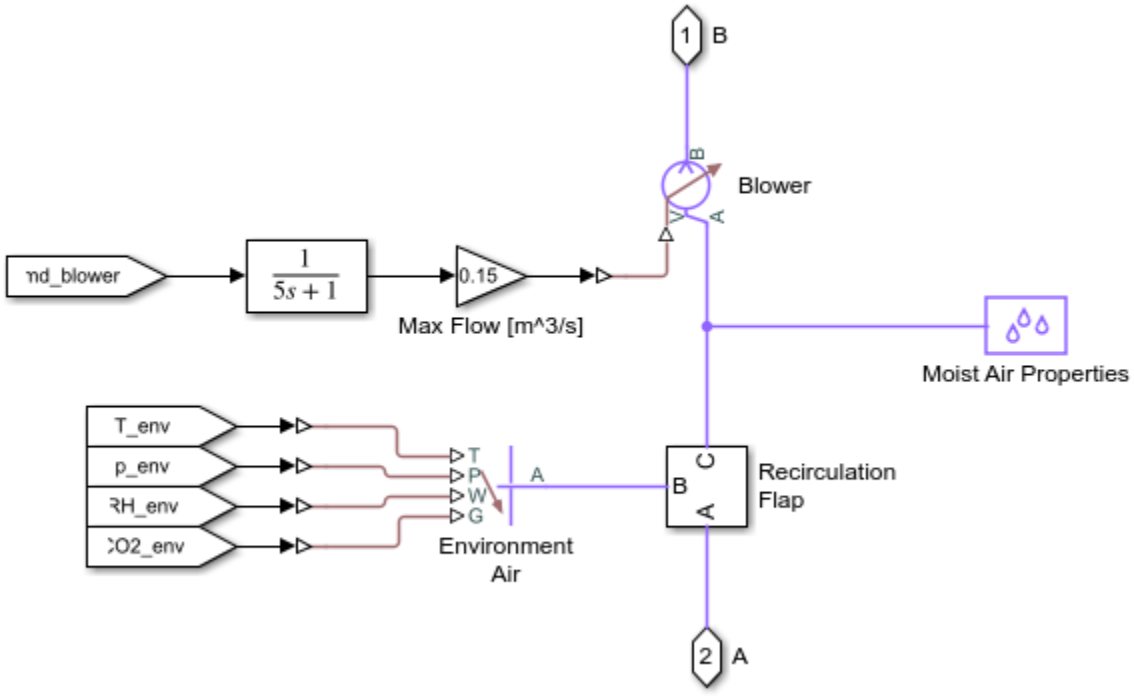
Evaporator Subsystem

The evaporator is a rectangular tube-and-fin type heat exchanger that lets the refrigerant absorb heat from the air. It also dehumidifies the air when the air is humid.



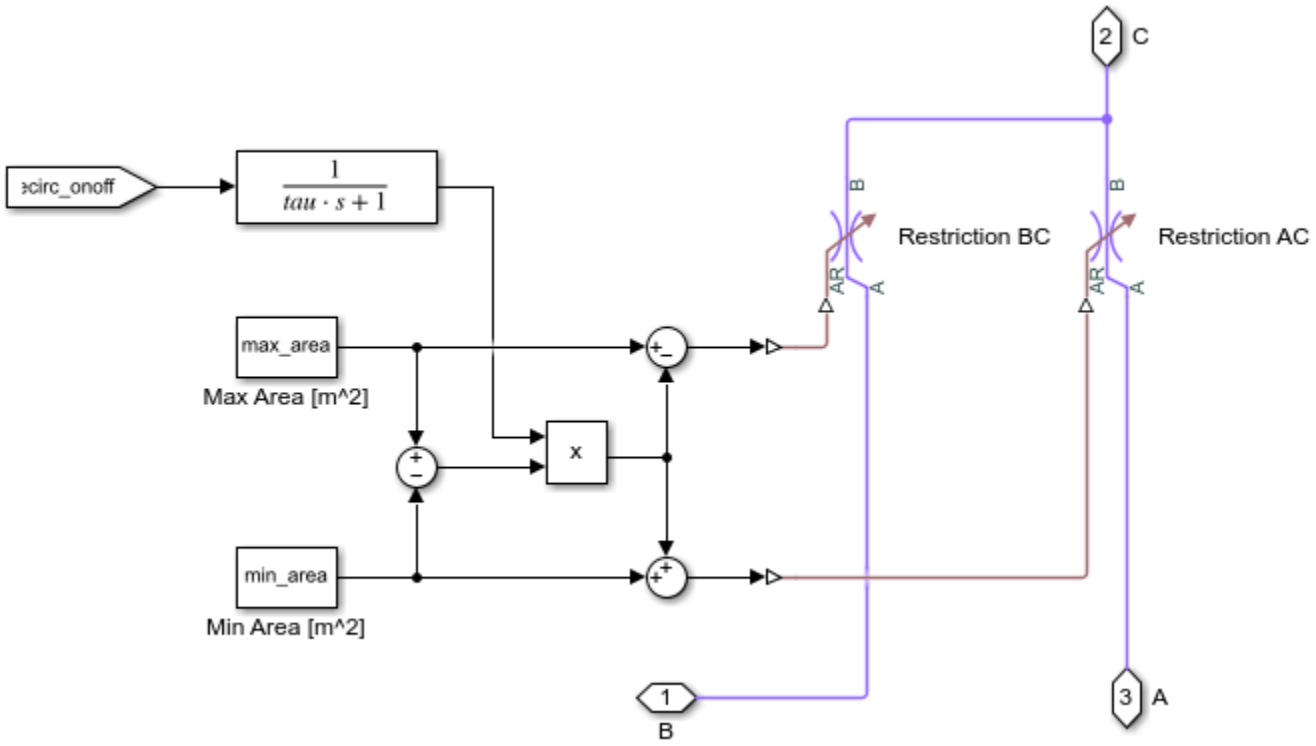
Blower Subsystem

The blower drives the air flow in the HVAC loop. It is controlled to maintain the cabin temperature setpoint. The source of air can come from the environment or from recirculated cabin air.



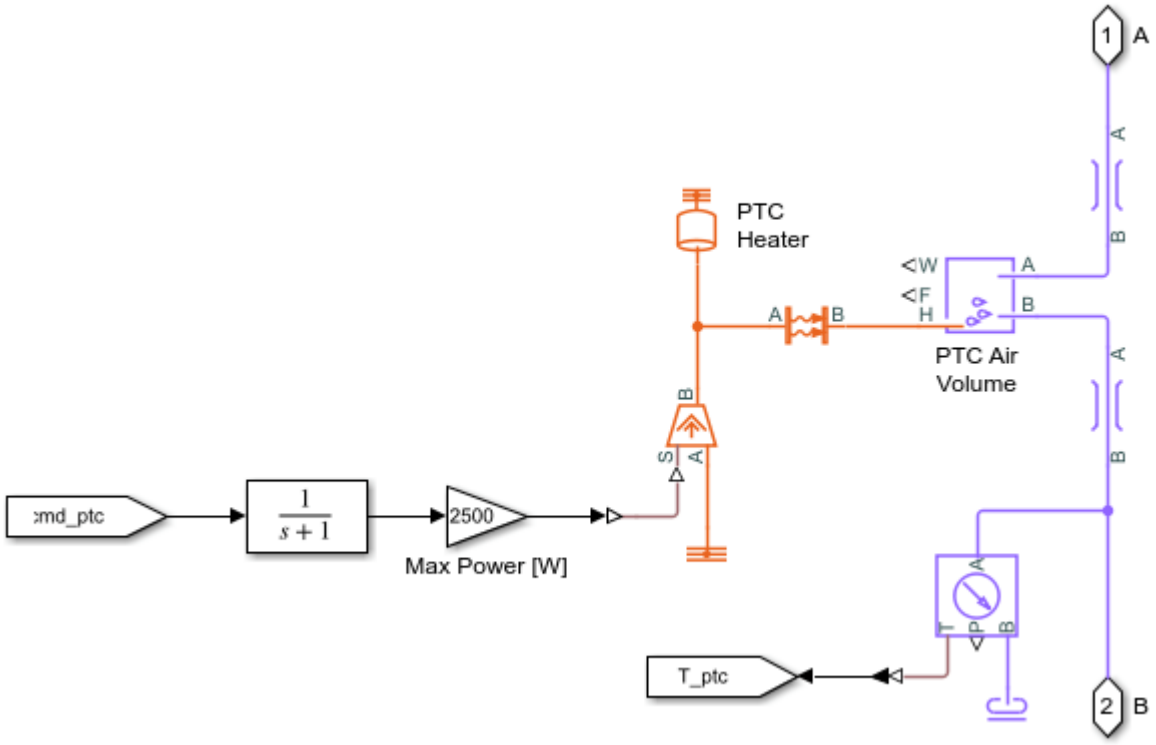
Recirculation Flap Subsystem

The recirculation flap is modeled as two restrictions operating in the opposite manner to let either environment air or cabin air to the blower.



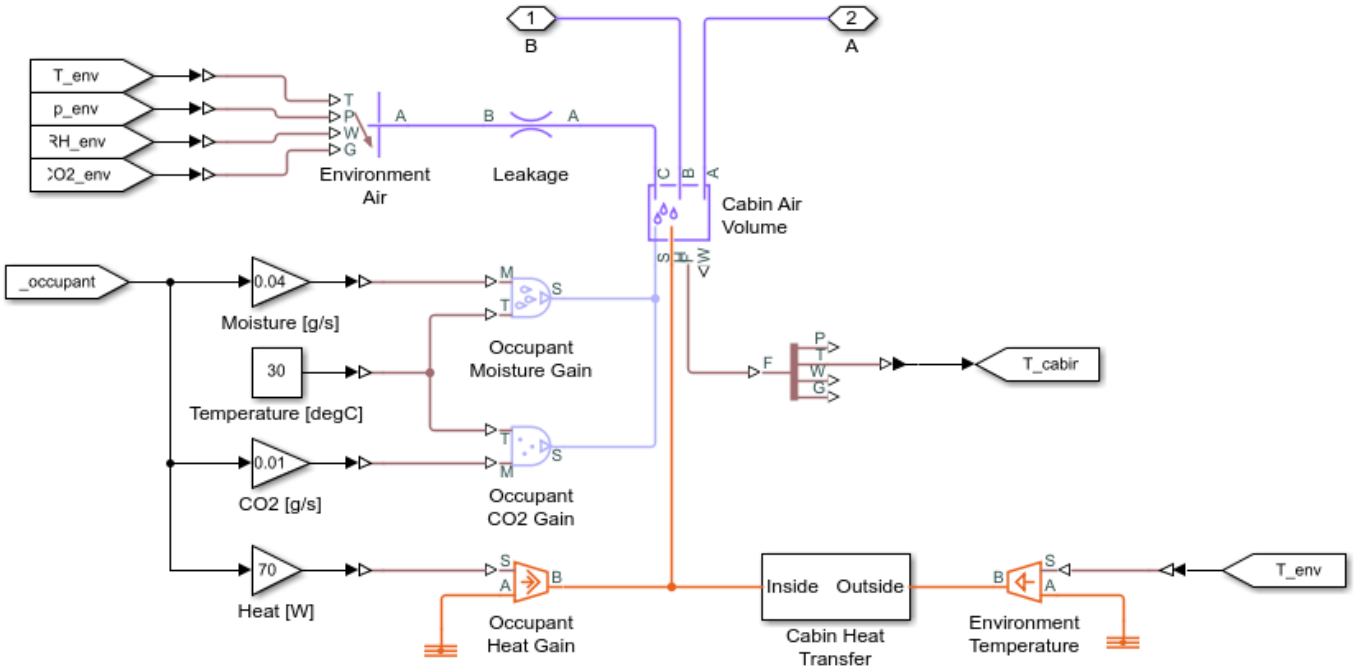
PTC Subsystem

The PTC heater is modeled as a heat flow rate source and a thermal mass. It is turned on in cold weather to provide heating to the vehicle cabin.



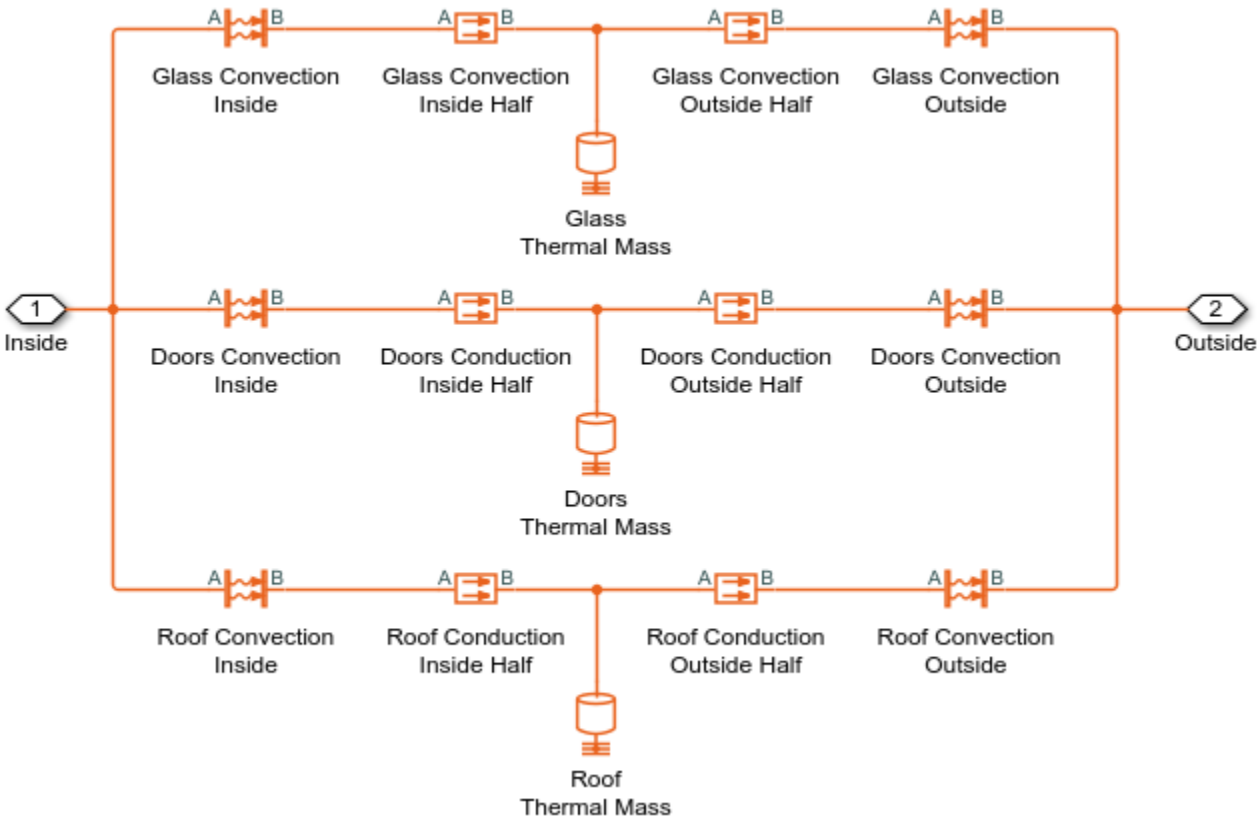
Cabin Subsystem

The vehicle cabin is modeled as a large volume of moist air. Each occupant in the vehicle is a source of heat, moisture, and CO2.



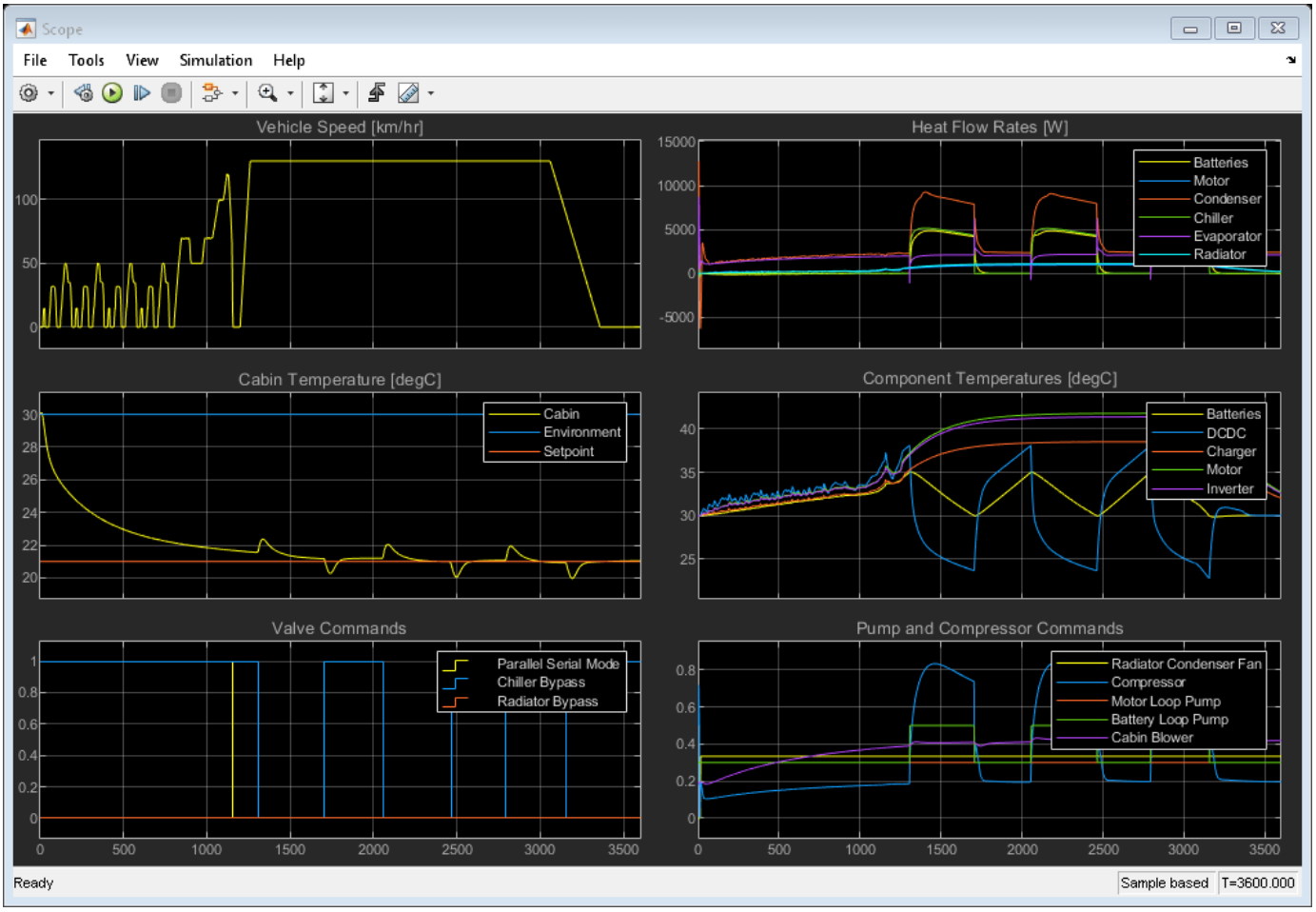
Cabin Heat Transfer Subsystem

This subsystem models the thermal resistances between the cabin interior and the external environment.



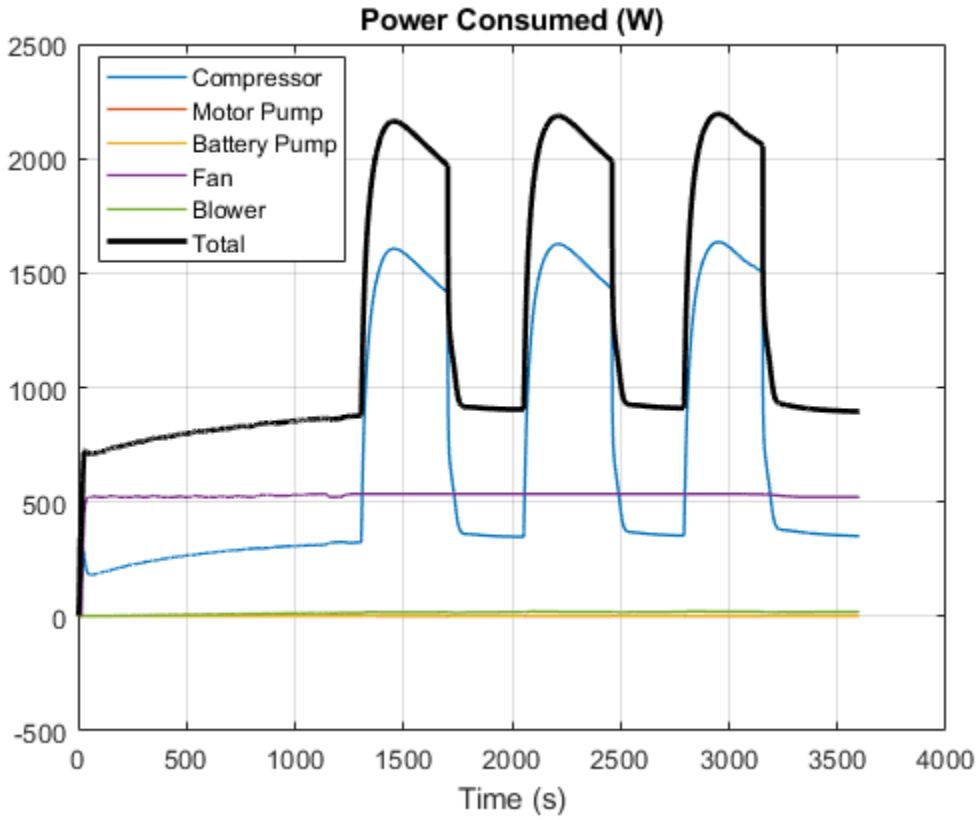
Simulation Results from Scopes

The following scope shows the vehicle speed, heat dissipation, cabin temperature, component temperatures, and control commands for the drive cycle scenario. At the beginning, the coolant loop is in serial mode. After about 1100 s, it switches to parallel mode and the chiller is used to keep the batteries below 35 degC.



Simulation Results from Simscape Logging

This plot shows the power consumed by the thermal management system to cool the vehicle components and cabin. The largest power consumption occurs in the refrigerant compressor when the chiller is used to cool the batteries.


Power Consumed (W)
Compressor
Motor Pump
Battery Pump
Fan
Blower
Total
2500
2000
1500
1000
500
0
-500
0
500
1000
1500
2000
2500
3000
3500
4000
Time (s)

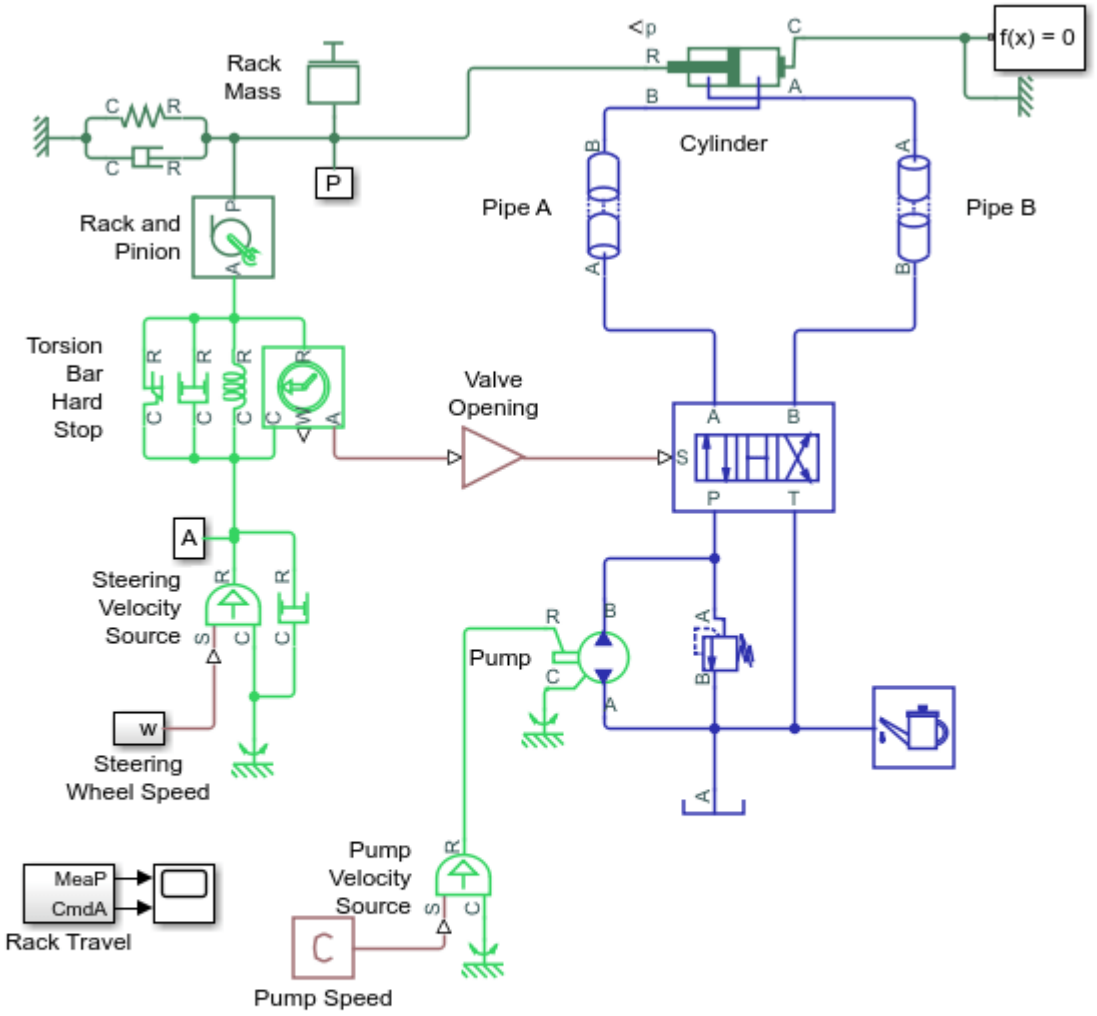
Power-Assisted Steering Mechanism

This example shows a simplified version of a power-assisted steering mechanism. The hydraulic actuation system includes a double-acting hydraulic cylinder, 4-way valve, fixed-displacement pump, and a pressure-relief valve. The steering rack acts against a load modeled by a spring and damper.

The valve opens based on the amount of twist of a torsion bar installed between the steering wheel and the pinion of a rack-and-pinion mechanism. The rotation of the steering wheel causes the torsion bar to twist relative to the pinion. The deformation of the bar is transformed into opening of the 4-way valve, which connects the chambers of the cylinder to pressure or return lines depending upon the direction of rotation. The PS Gain block Valve Opening converts torsion bar twist to the amount of opening for the 4-way valve. Setting the Gain parameter of this block to 0 disables the power steering.

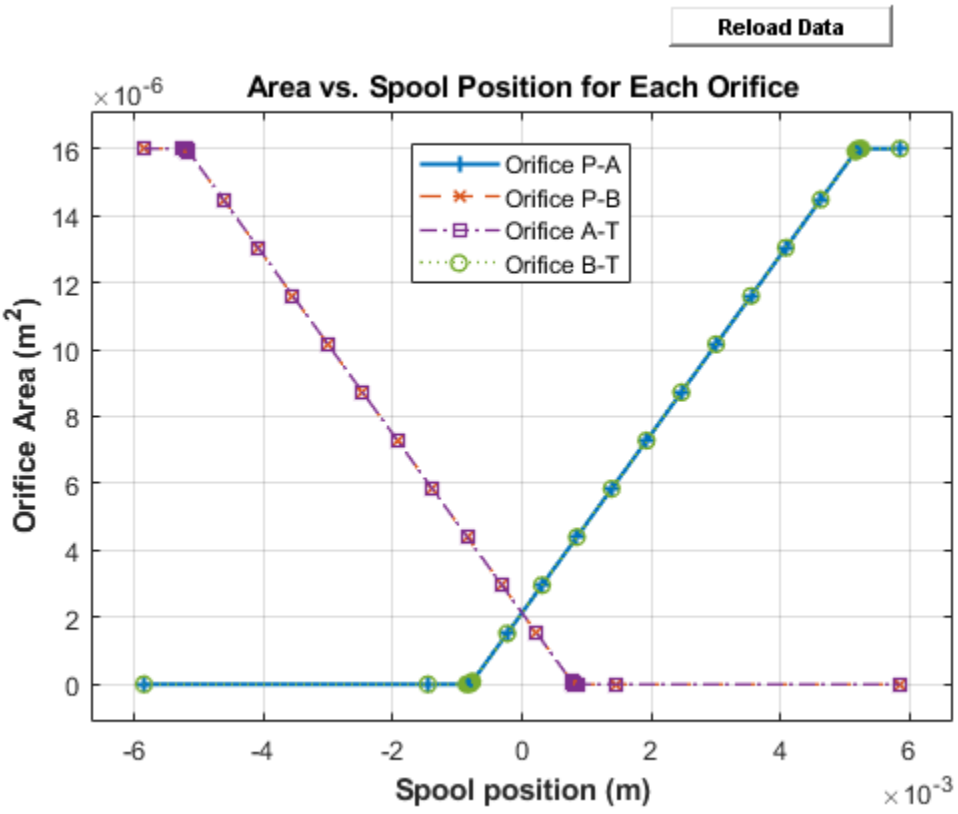
If the torsion bar deformation exceeds 9 degrees the wheel is connected directly to the pinion through the hard stops installed in parallel with the torsion bar. The cylinder moves the steering rack which twists the torsion rod in the opposite direction until the valve is in the neutral position.

Model

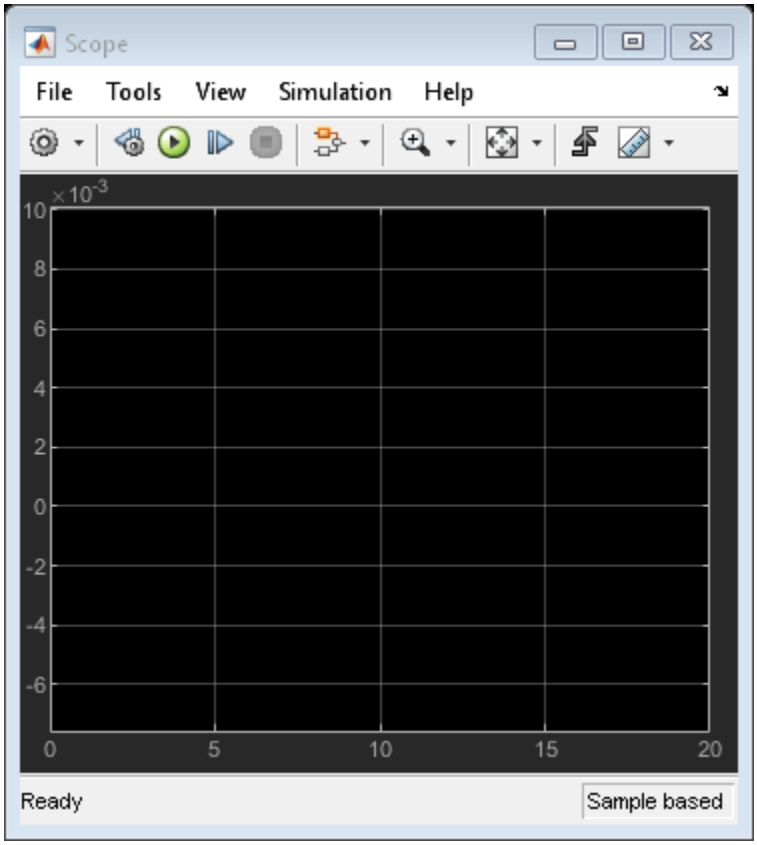


4-Way Valve Opening Characteristics

This plot is generated by right-clicking on the 4-Way Directional Valve (IL) and selecting Fluids > Plot Valve Characteristics.

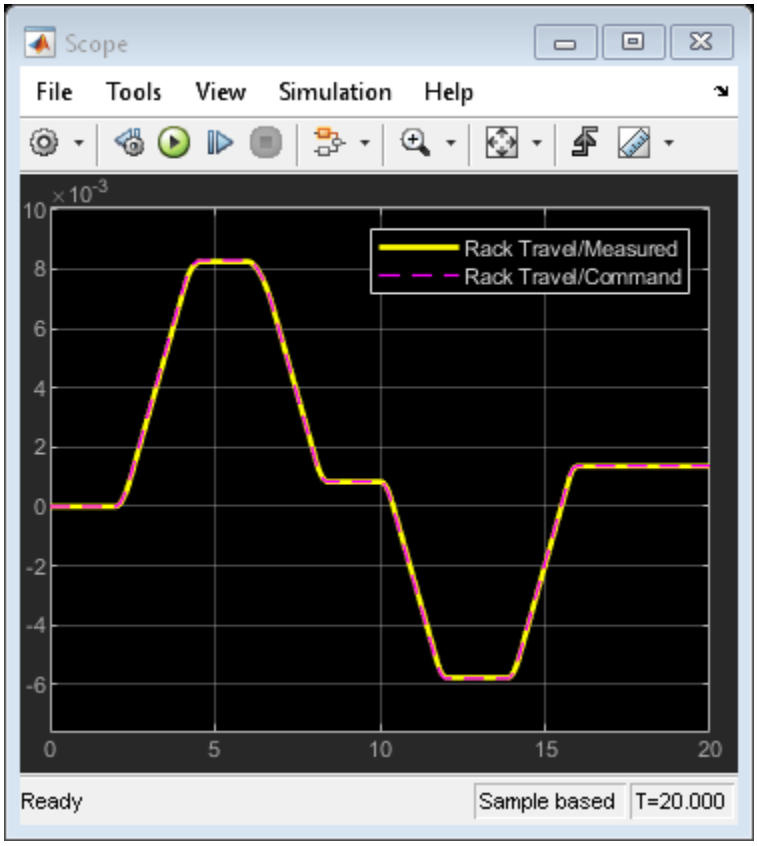

Reload Data
Area vs. Spool Position for Each Orifice
×10^-6
Orifice Area (m^2)
16
14
12
10
8
6
4
2
0
Orifice P-A
Orifice P-B
Orifice A-T
Orifice B-T
-6
-4
-2
0
2
4
6
Spool position (m)
×10^-3

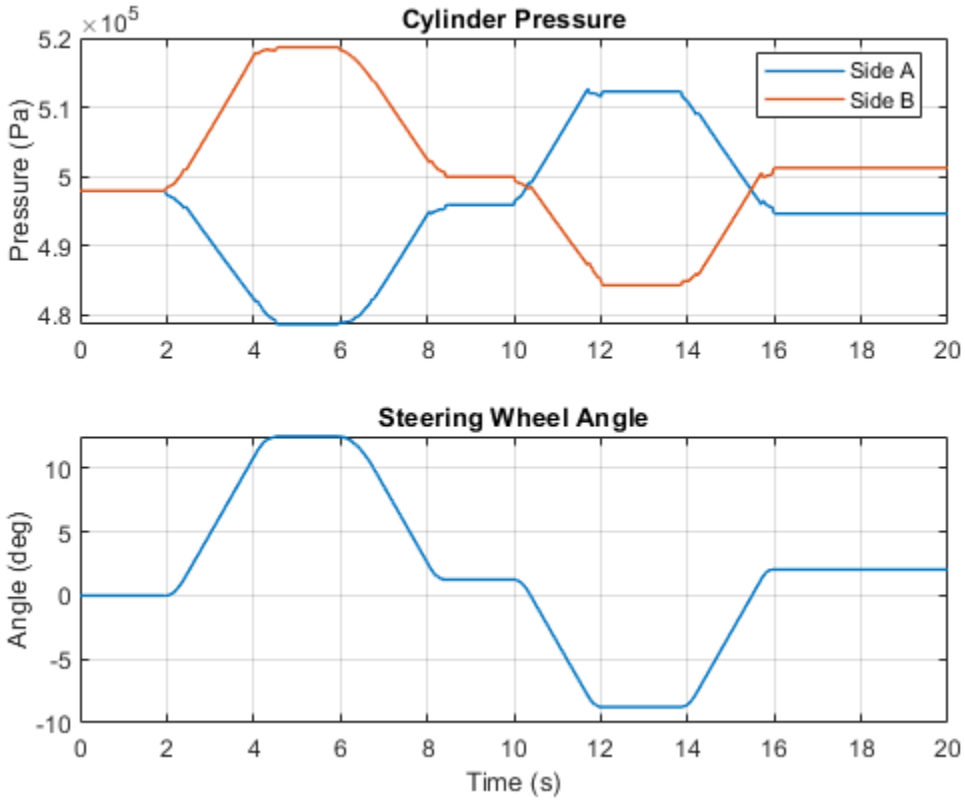
Simulation Results from Scopes



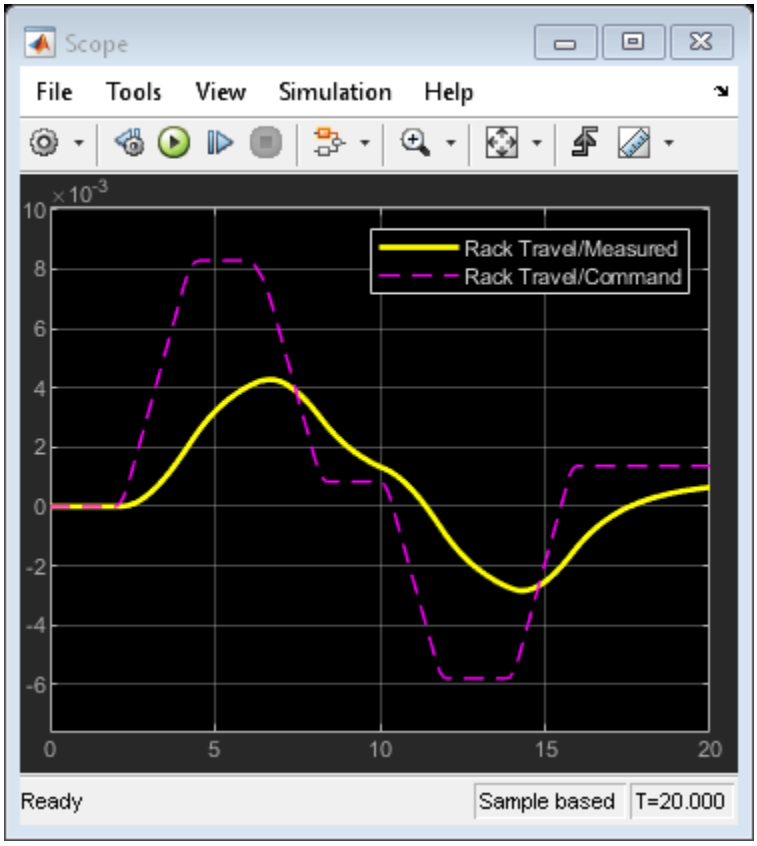
Simulation Results from Simscape Logging

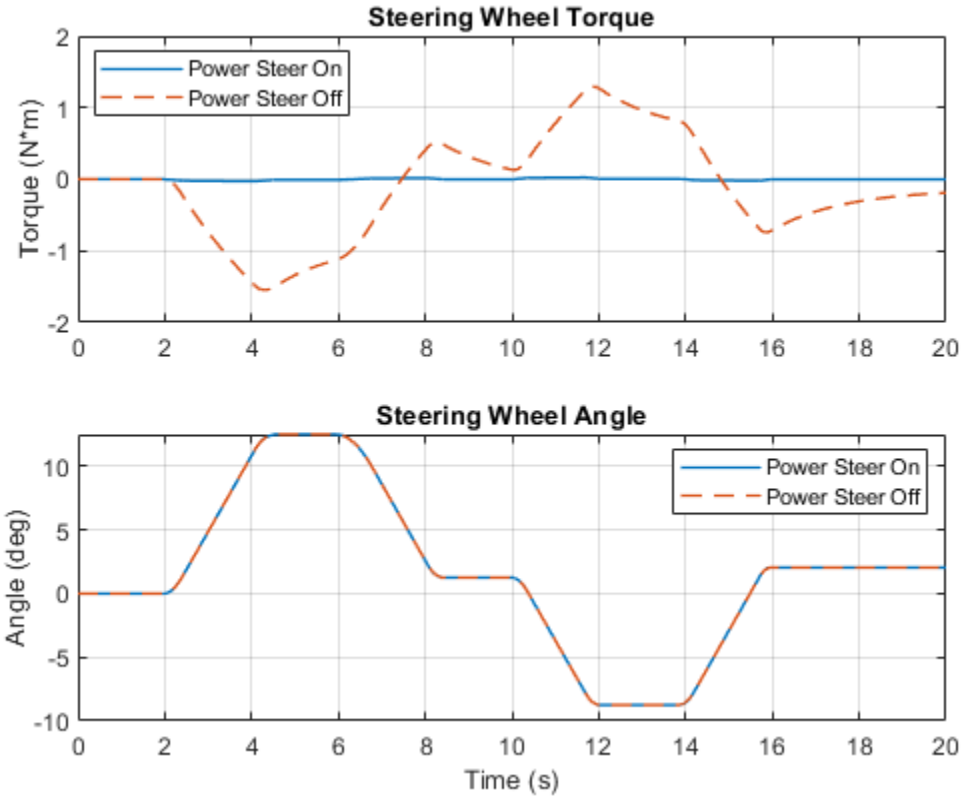
The plots below show the amount of assistance the power steering system provides to the driver as the wheel is turned. The greater the difference in the cylinder chamber pressures, the more assistance that is provided.


Scope
File
Tools
View
Simulation
Help
×10^-3
10
8
6
4
2
0
-2
-4
-6
0
5
10
15
20
Rack Travel/Measured
Rack Travel/Command
Ready
Sample based
T=20.000



The plots below compare the amount of effort required when the power steering is turned on and turned off. For the same steering sequence, the amount of torque required when the power steering is turned off is much higher. The power steering is turned off by setting the gain in Valve Opening to 0.


Scope
File
Tools
View
Simulation
Help
×10^-3
10
8
6
4
2
0
-2
-4
-6
0
5
10
15
20
Rack Travel/Measured
Rack Travel/Command
Ready
Sample based
T=20.000

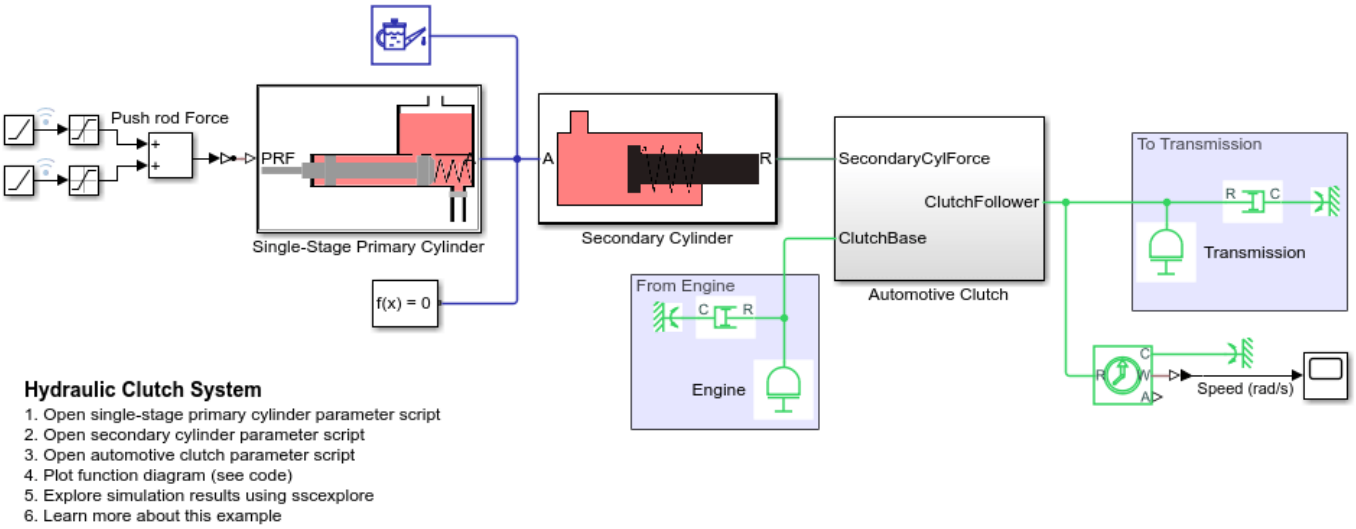

Steering Wheel Torque
Power Steer On
Power Steer Off
Torque (N*m)
2
1
0
-1
-2
0
2
4
6
8
10
12
14
16
18
20
Steering Wheel Angle
Power Steer On
Power Steer Off
Angle (deg)
10
5
0
-5
-10
Time (s)

Hydraulic Clutch System

This example shows how to model, parameterize, and test a hydraulic clutch system. The model is used to generate the plot of the engine and the transmission system speeds during a declutching and clutch re-engaging scenario.

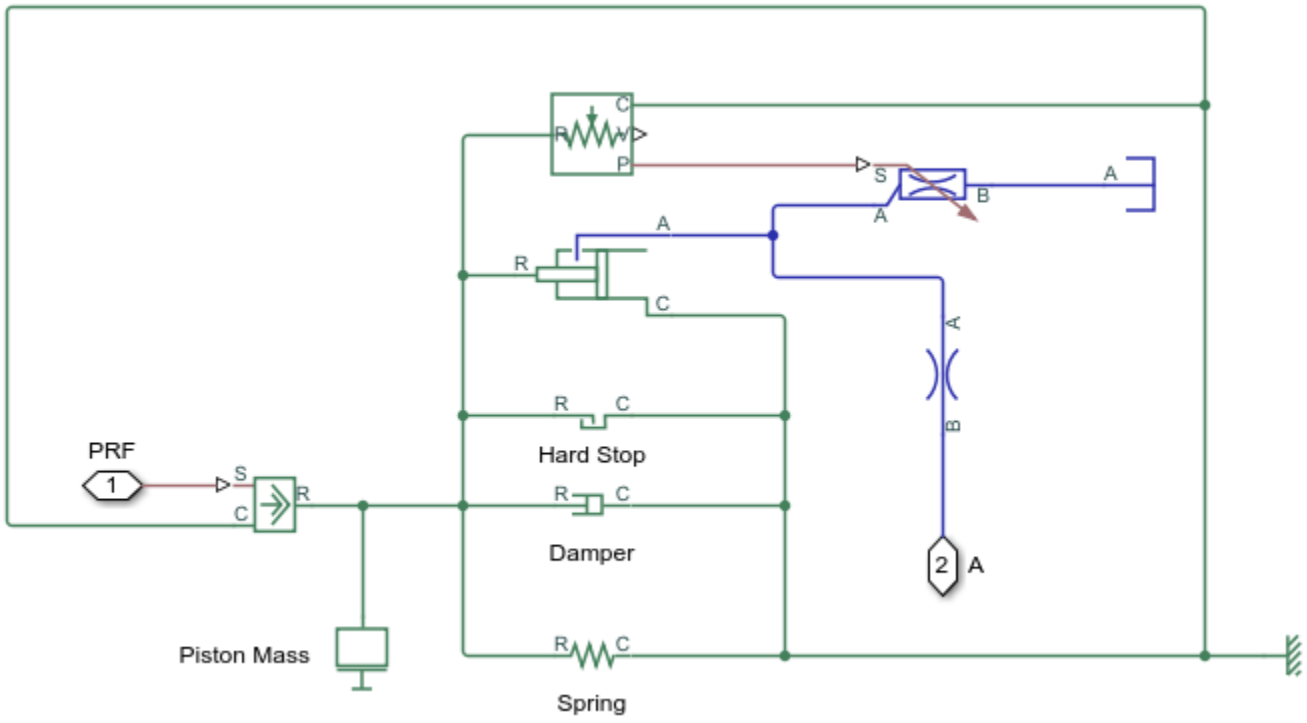
Model

The following figure shows the model of a hydraulic clutch system in a test harness. The output of the model is a plot of speeds versus time of the engine and the transmission systems during the declutching and clutch re-engaging scenario.



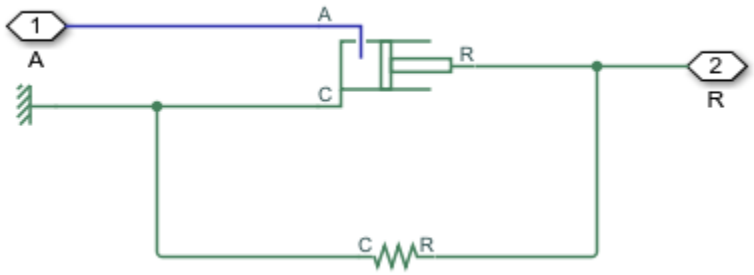
Single-Stage Primary Cylinder Subsystem

The following figure shows the modeling part of the single-stage primary cylinder.



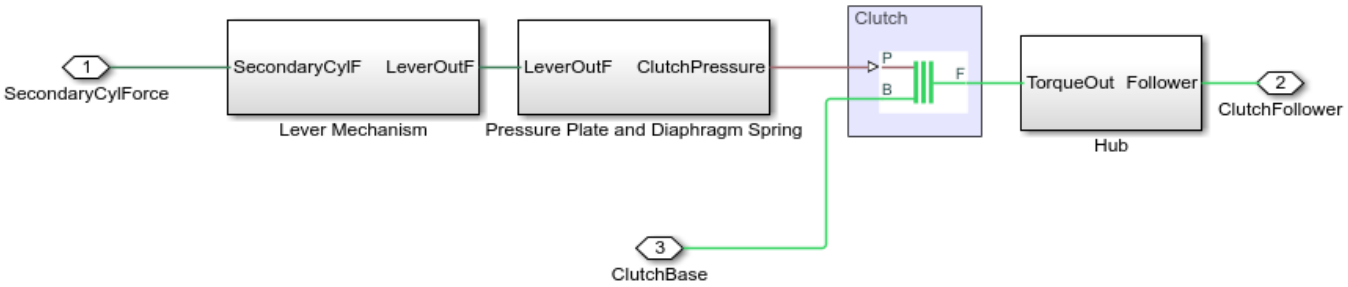
Secondary Cylinder Subsystem

The following figure shows the modeling part of the secondary cylinder.



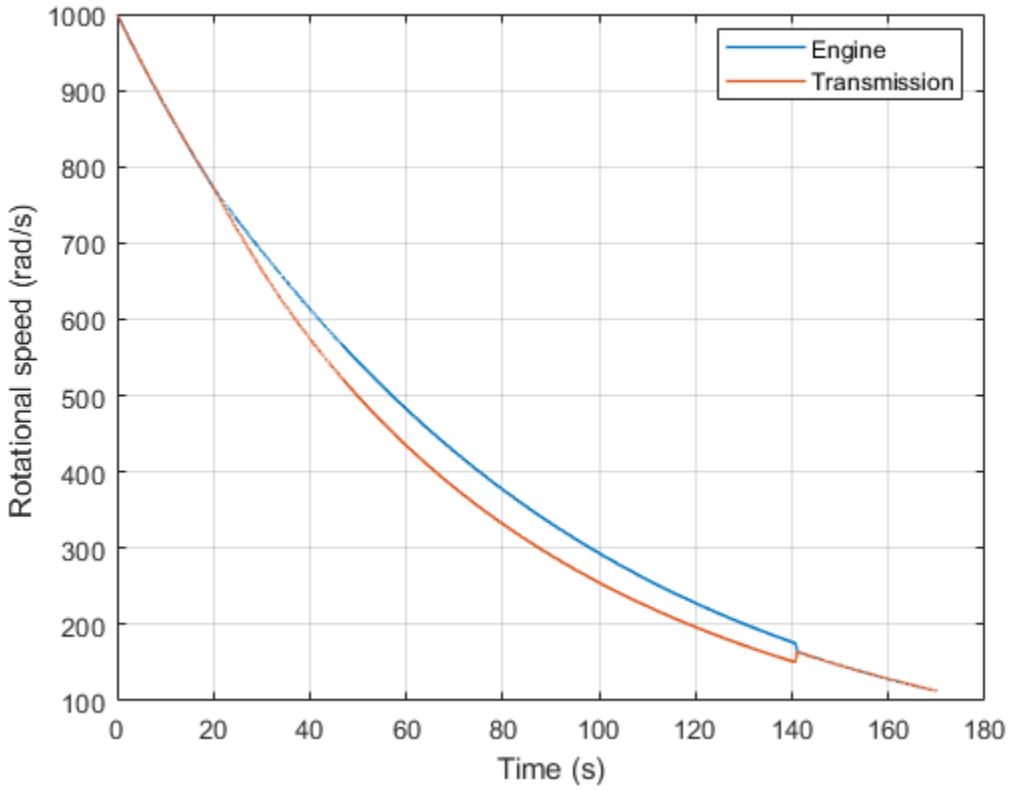
Automotive Clutch Subsystem

This subsystem shows how the clutch is modeled. The clutch consists of a lever mechanism, a diaphragm spring, a pressure plate, a set of clutch plates, and a hub. The lever transfers force to the diaphragm spring when a force is applied on the other end of the lever by the secondary cylinder. The applied force on the diaphragm spring causes the pressure plate to remove the pre-applied force on the clutch plate. The declutching takes place when the force on the clutch plate is removed through the removal of the force on the pressure plate by the diaphragm spring's action. No torque is transferred from the engine to the transmission after the declutching event.



Simulation Results from Simscape™ Logging

This model generates a plot of speeds versus time of the engine and the transmission systems during the declutching and clutch re-engaging scenario.

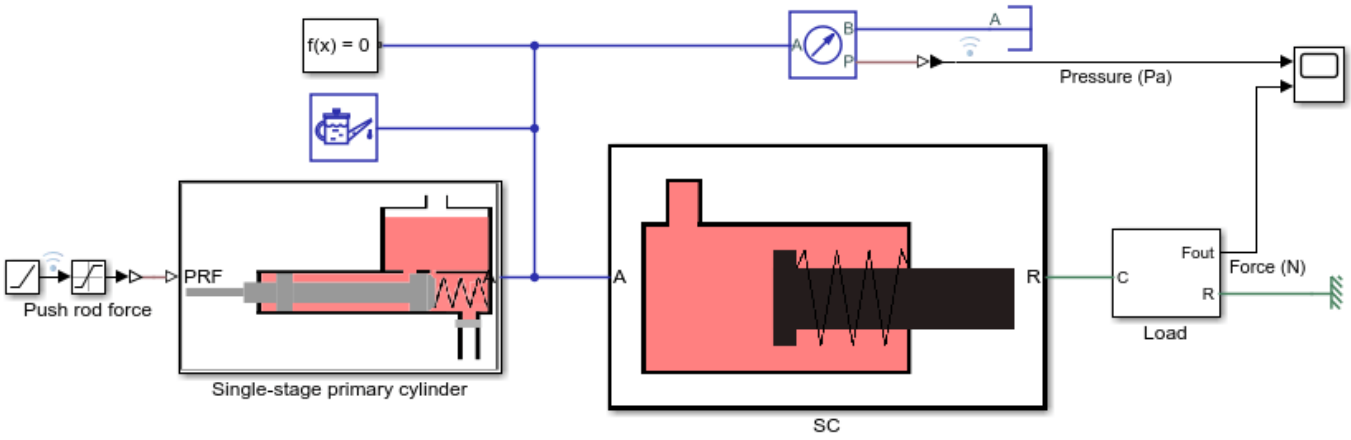


Secondary Cylinder

This example shows how to model, parameterize and, test a secondary cylinder. Given the numerical data extracted from the datasheet of the single-stage primary cylinder, the unknown parameters of the secondary cylinder are calculated. The model is simulated to generate the plot between the applied force to the single-stage primary cylinder and the pressure developed inside the secondary cylinder.

Model

The following figure shows the model of a secondary cylinder with the test harness. The output of the model is the plot between the applied force to the single-stage primary cylinder and the pressure developed inside the secondary cylinder.



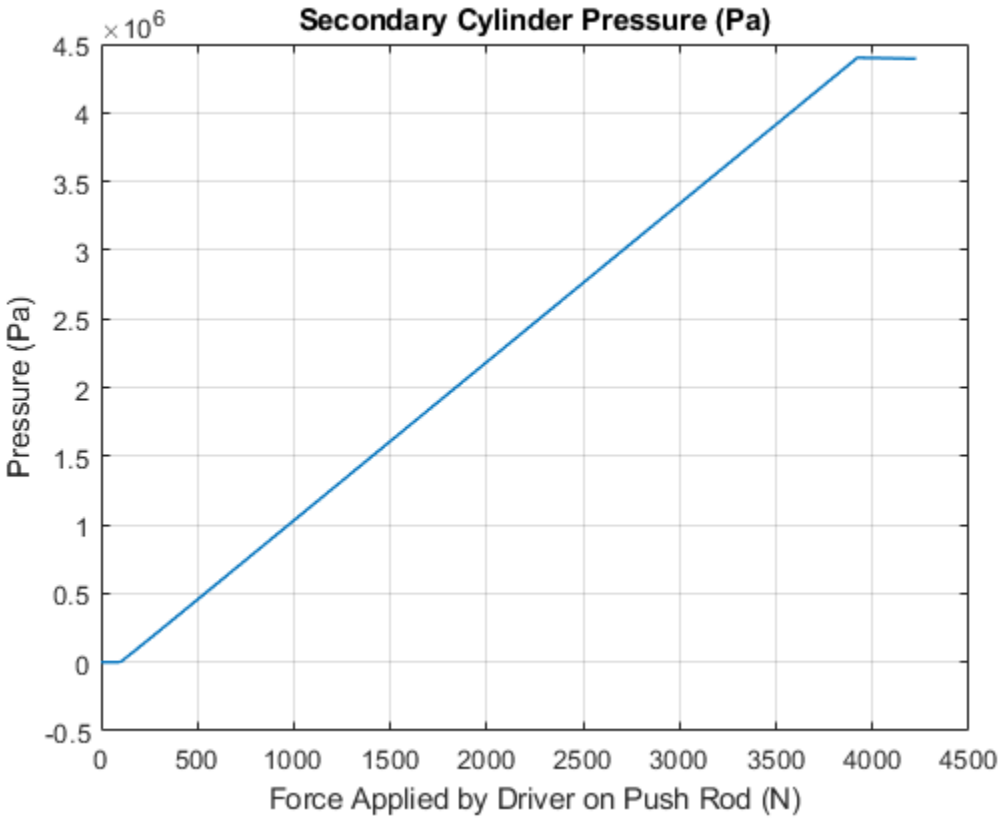
Secondary Cylinder

1. Open input script
2. Plot function diagram (see code)
3. Explore simulation results using sscexplore
4. Learn more about this example

Copyright 2019-2021 The MathWorks, Inc.

Simulation results from Simscape™ logging

This model generates a plot between the applied force to the single-stage primary cylinder and the pressure developed inside the secondary cylinder.

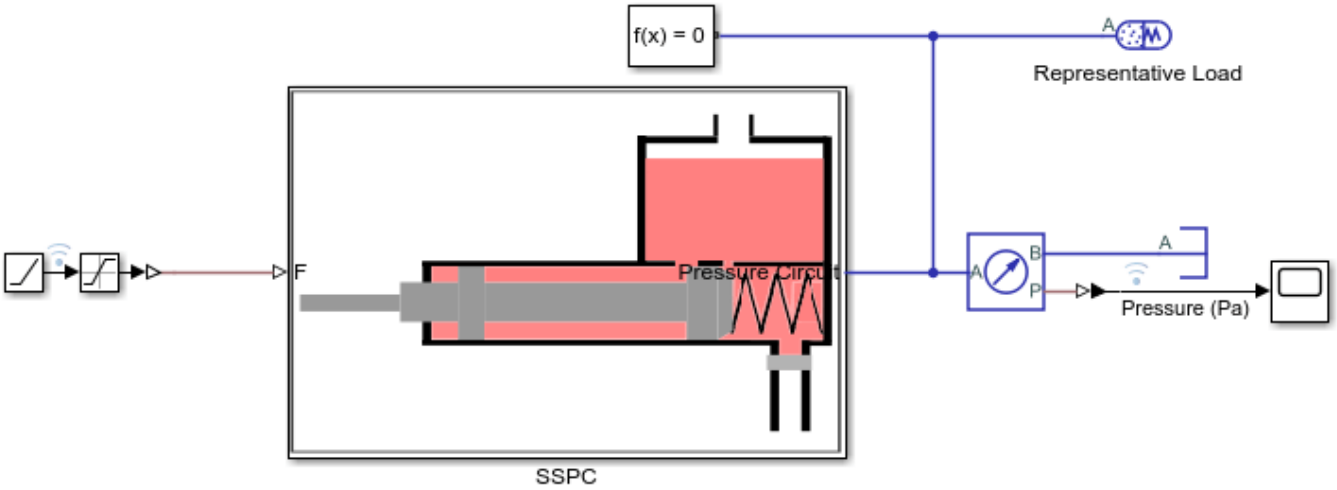

Secondary Cylinder Pressure (Pa)
×10^6
4.5
4
3.5
3
2.5
2
1.5
1
0.5
0
-0.5
Pressure (Pa)
0
500
1000
1500
2000
2500
3000
3500
4000
4500
Force Applied by Driver on Push Rod (N)

Single-Stage Primary Cylinder

This example shows how to model, parameterize and test a single-stage primary cylinder starting from manufacturer datasheet information. Given the numerical data extracted from the datasheet, the unknown parameters are calculated. The model is then simulated and the resulting push rod force versus pressure relationship curve is compared with curve provided on the manufacturer datasheet.

Model

The following figure shows the model of a single-stage primary cylinder with the test harness. The output of the model is the plot of push rod force versus pressure relationship curve.



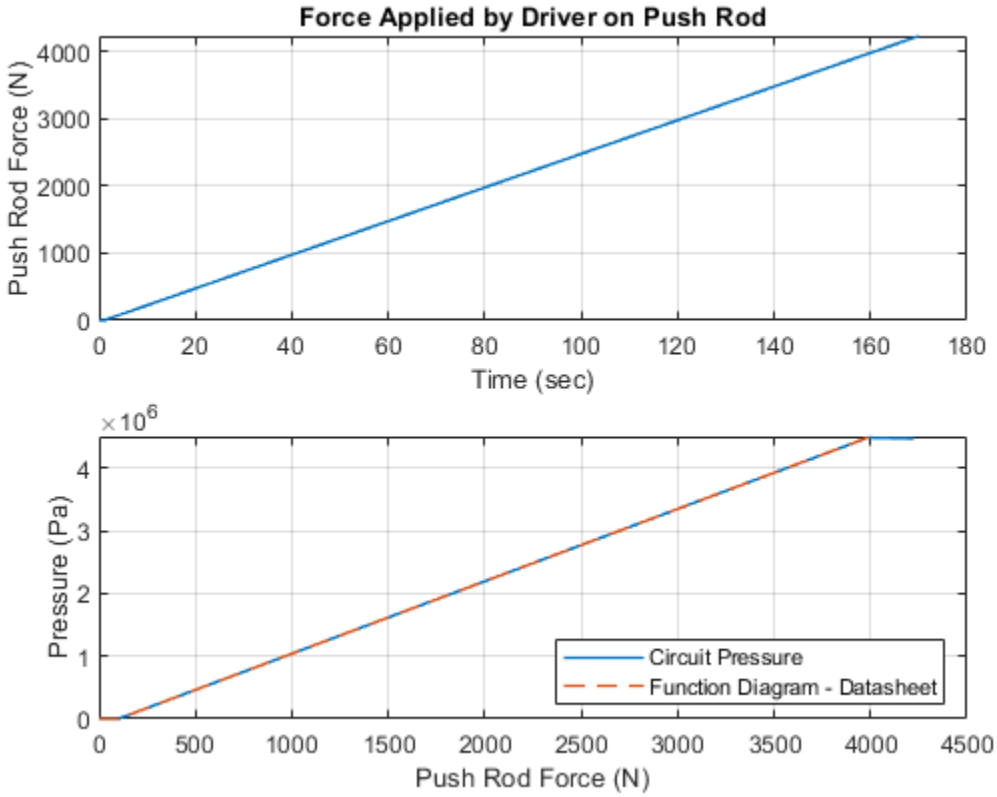
Single-Stage Primary Cylinder

1. Open live script
2. Plot function diagram (see code)
3. Explore simulation results using sscexplore
4. Learn more about this example

Copyright 2019-2021 The MathWorks, Inc.

Simulation results from Simscape™ logging

This script generates a plot of the push rod force versus pressure relationship in single-stage primary cylinder.

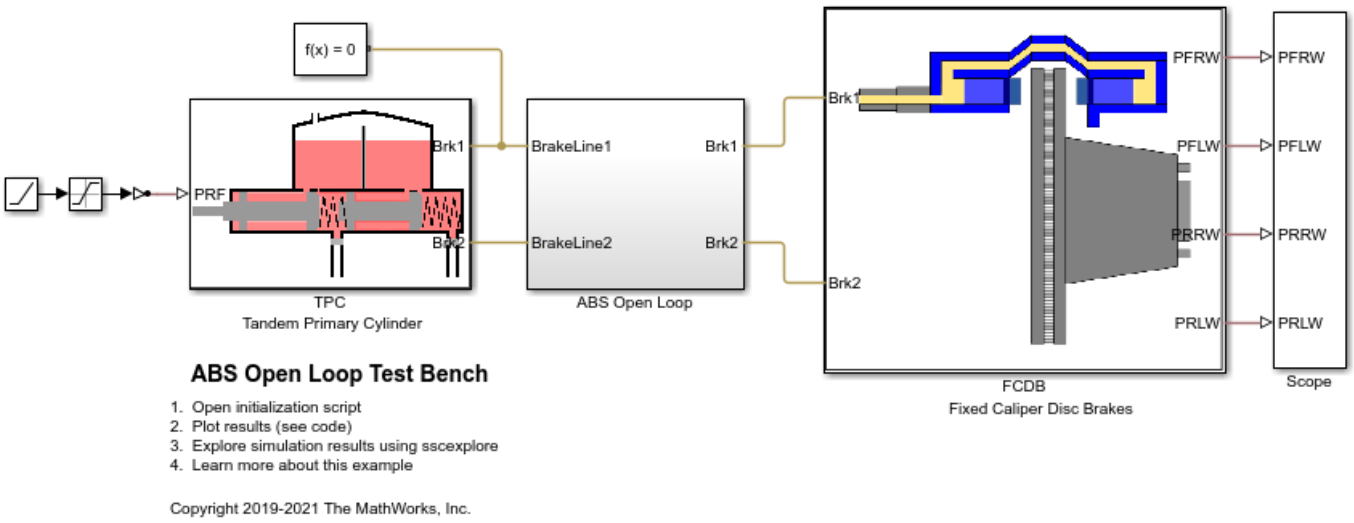

Force Applied by Driver on Push Rod
Push Rod Force (N)
Time (sec)
×10^6
Pressure (Pa)
Push Rod Force (N)
Circuit Pressure
Function Diagram - Datasheet

ABS open loop test bench

This example shows a simple way of modeling of the anti-lock braking system (ABS) and using it with the manual braking components. The model simulates for the open loop ABS and shows the response of the pressures achieved in the caliper disc brakes. This model can be utilized in sizing of the release valve, apply valve and accumulator for the hydraulic control unit of the ABS.

Model

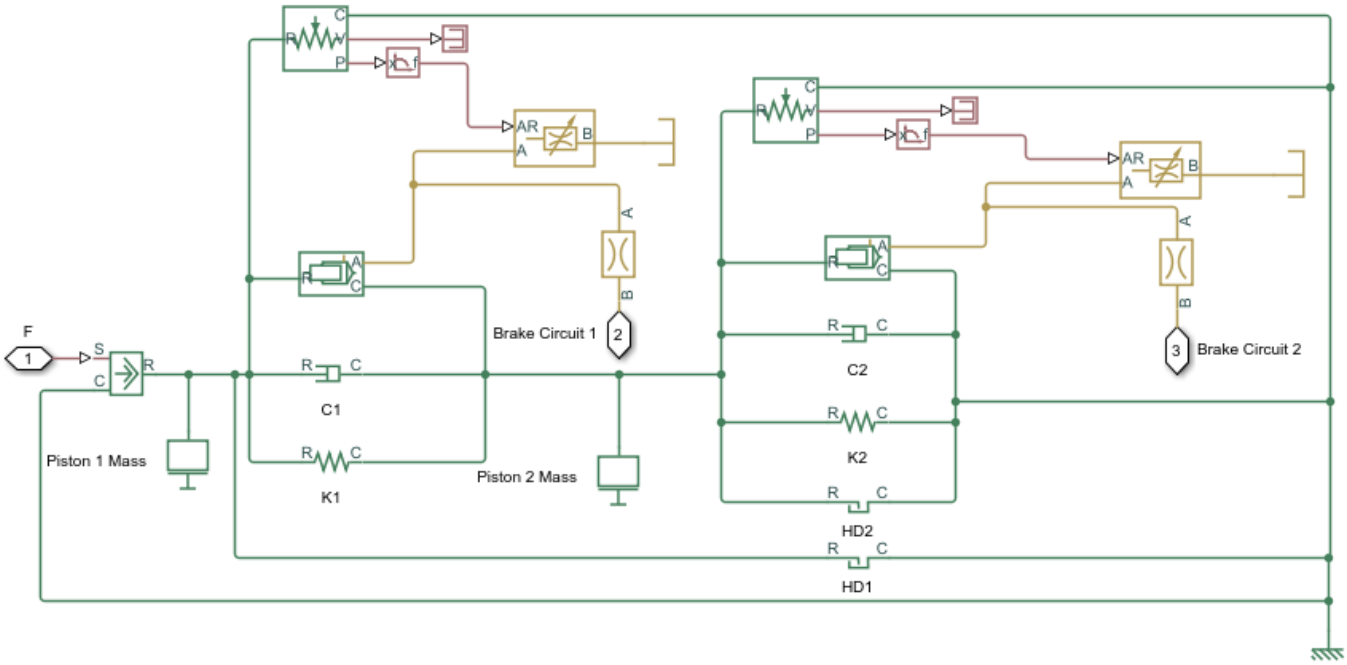
The following figure shows the model of an ABS system in open loop with the test harness. The output of the model is the plot of the pressures in the caliper disc brakes.



Tandem primary cylinder subsystem

This subsystem shows the modeling of the tandem primary cylinder.

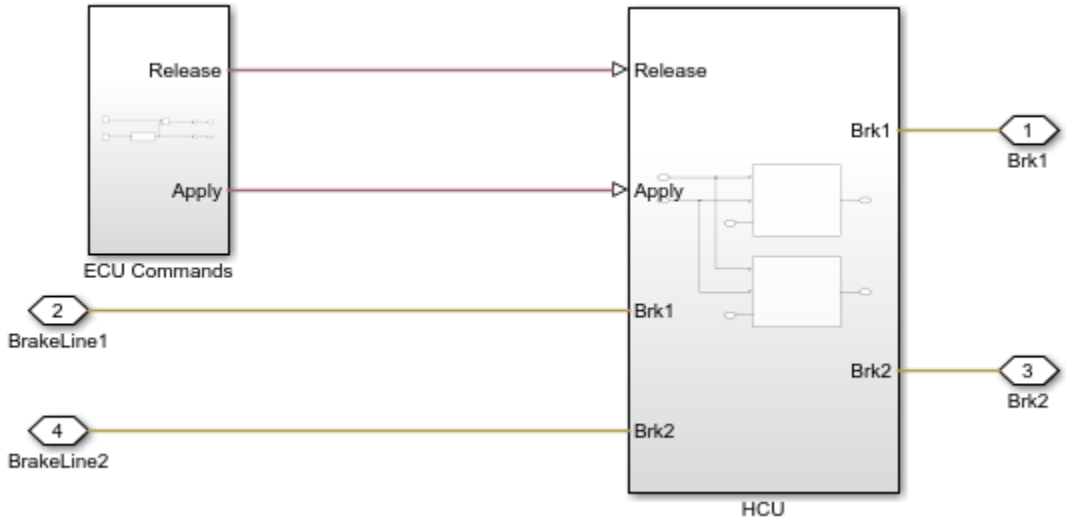
Open Subsystem



ABS open loop subsystem

This subsystem shows the modeling of the ABS. It contains the electronic control unit (ECU) and the hydraulic control unit (HCU) subsystems.

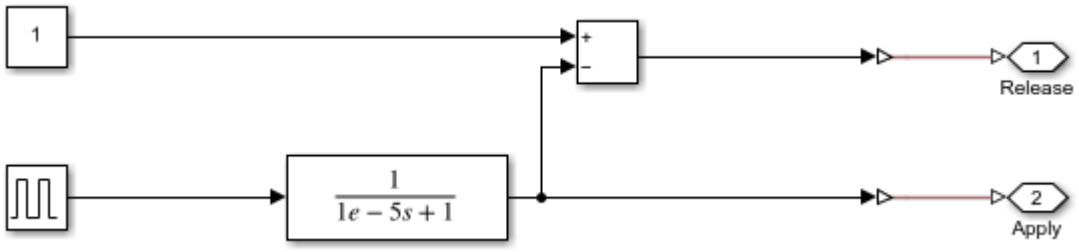
Open Subsystem



ECU commands subsystem

The ECU provides the control signals to the release and the apply valves.

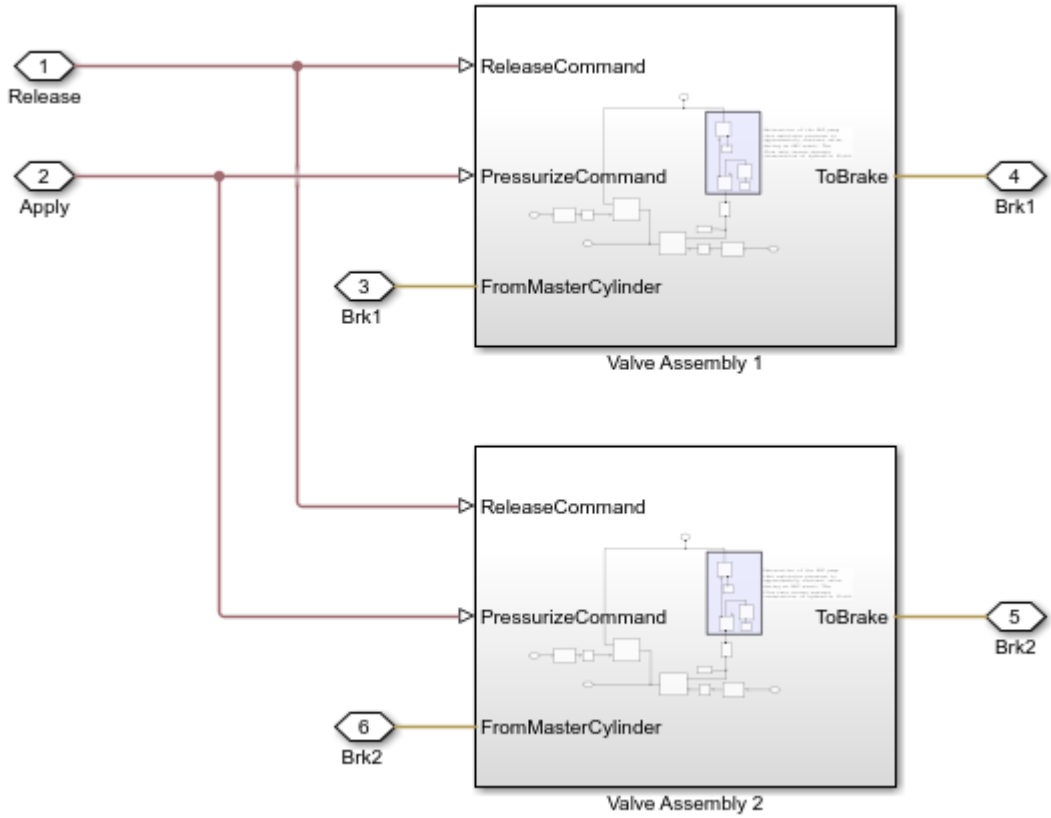
Open Subsystem



HCU subsystem

The HCU subsystem shows, how to model the release valve, apply valve, pump and accumulator in the simplest possible way.

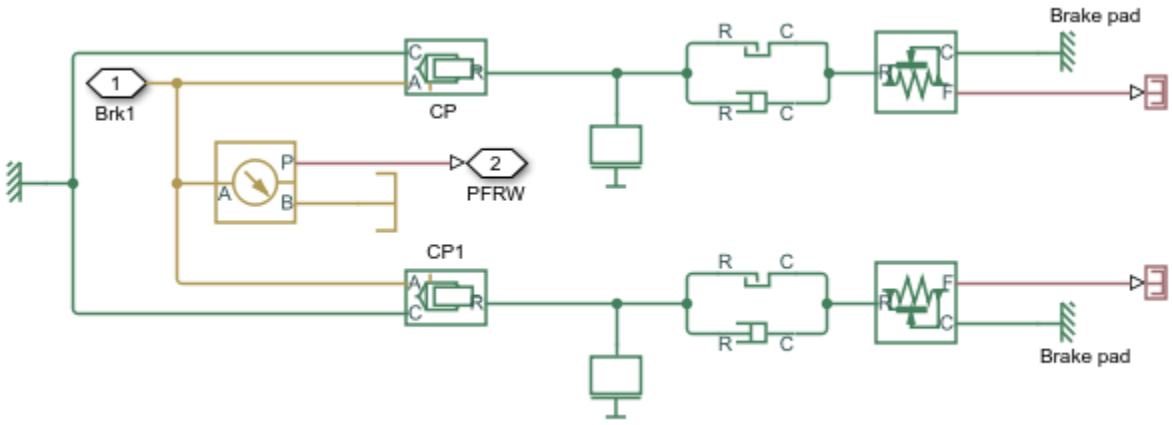
Open Subsystem



FRW (front right wheel) subsystem

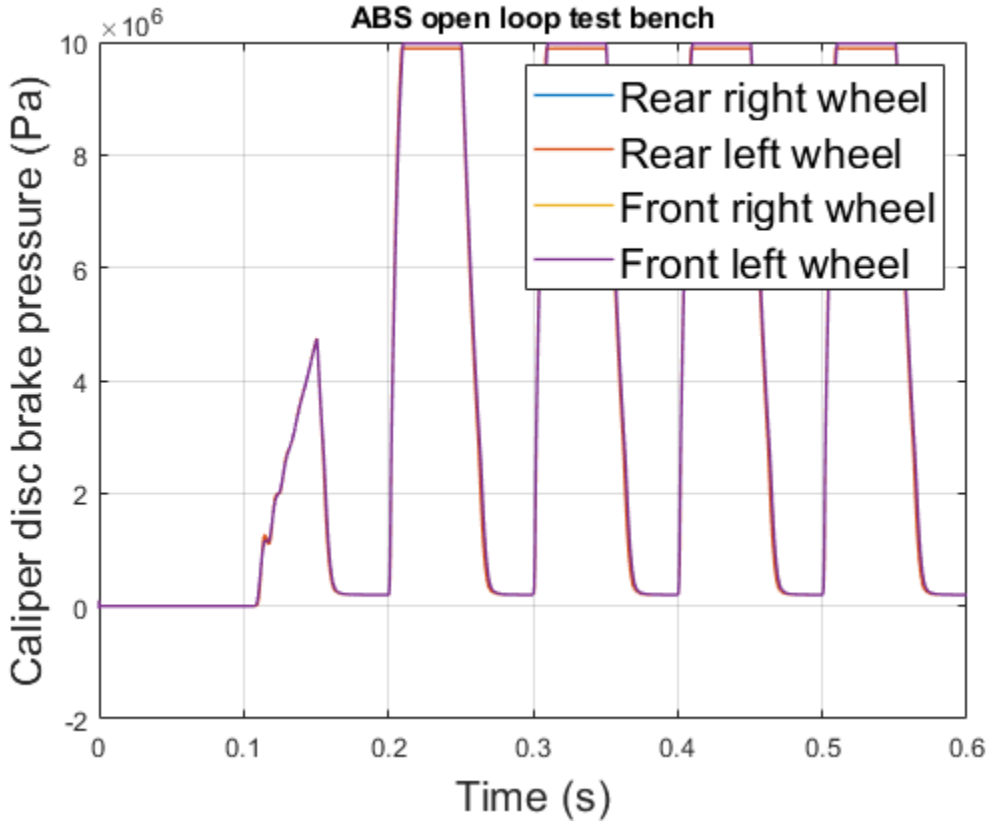
This subsystem shows the modeling of the fixed caliper disc brake's front right wheel subsystem. The friction plates are not modeled in this subsystem as the vehicle mass body is not attached to the brakes. Other wheels brakes are also modeled in the same way.

Open Subsystem



Simulation results from simscape logging

The model has two inputs. The first input is a force applied to the tandem primary cylinder's push rod. The second input is the ECU commands applied to the release and apply valve in the ABS open loop subsystem. Based on the given inputs, the model generates a plot of the pressures in the caliper cylinders of the brakes.

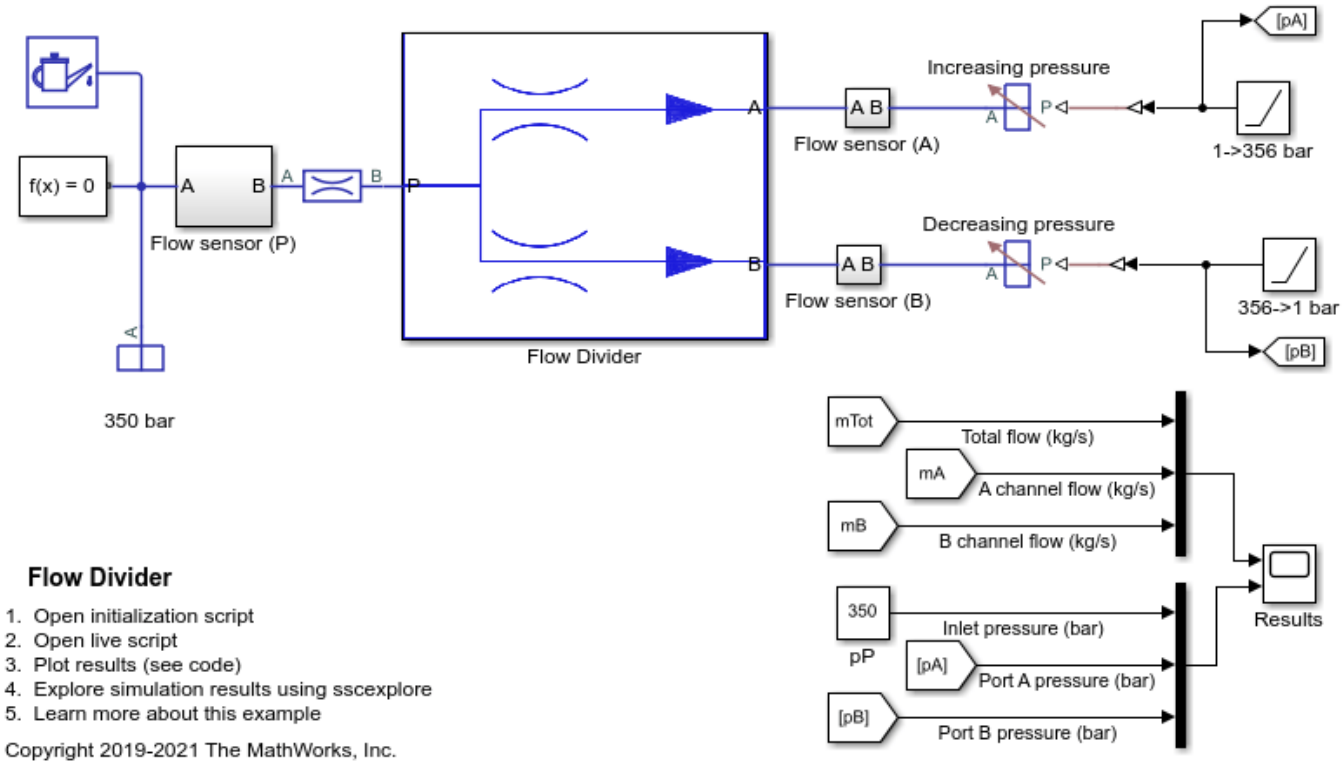


Flow Divider

This example shows a simple way of modeling of the flow divider and using it with the load harness. The flow divider helps in dividing the input flow in desired percentage when the loads connected to the two lines are unequal. Flow divider allows flow only in forward direction. Therefore when it is used with the loads where unloading is done through reverse flows, then it should be used in conjunction with the direction control valves.

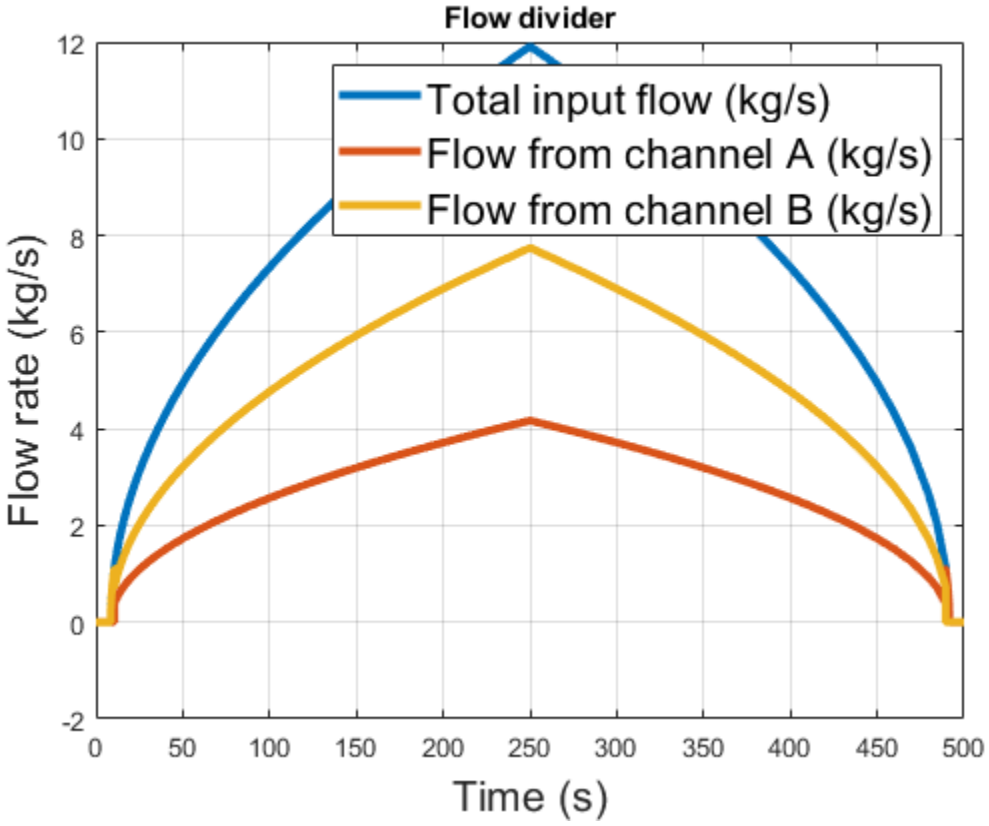
Model

The following figure shows the model of a flow divider with the test harness. The output of the model is the plot of flow through the channels A and B for a given input flow.



Simulation Results from Simscape Logging

This script generates a plot of flow through the channels A and B for a given input flow.

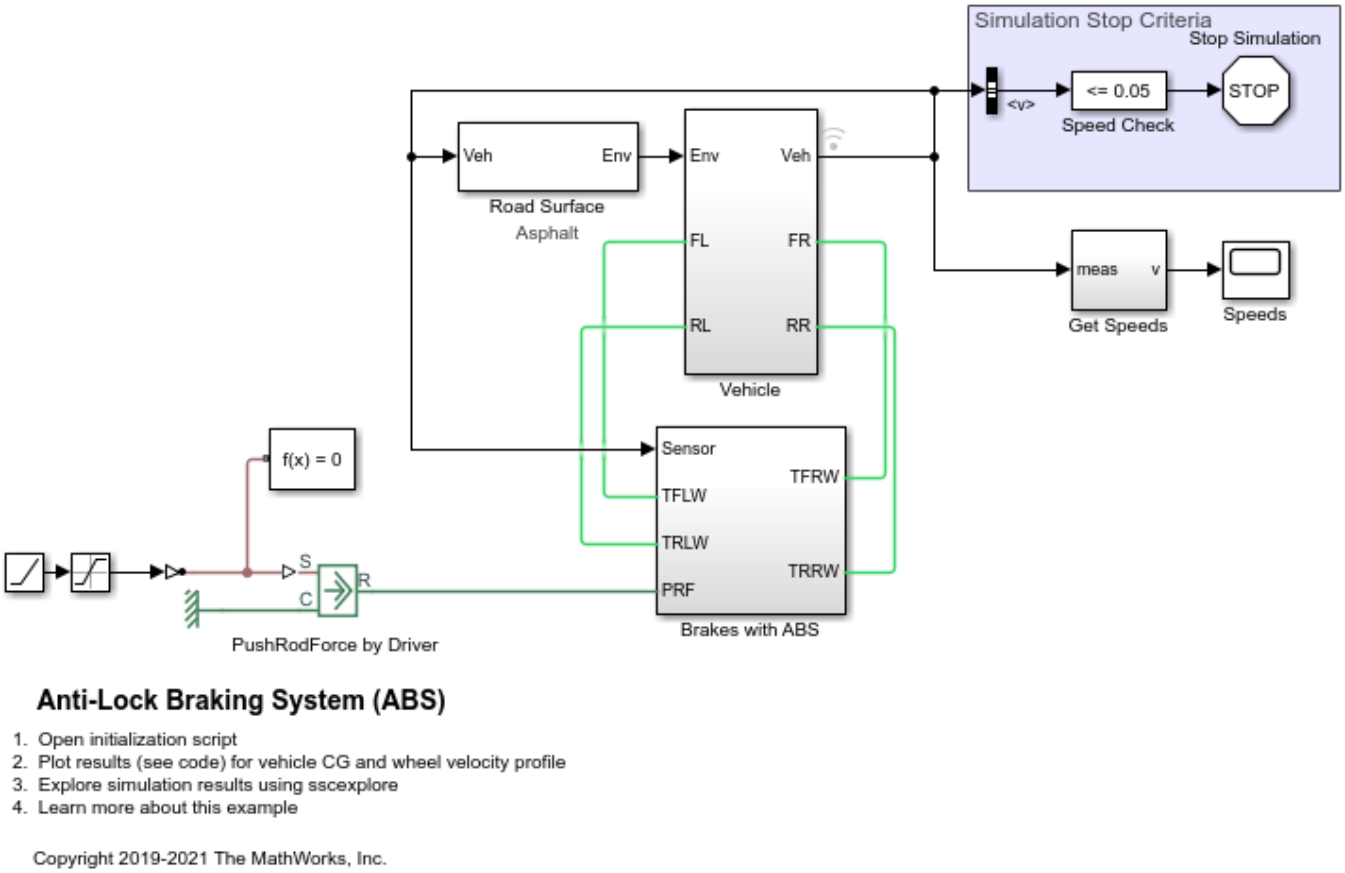

Flow divider
Total input flow (kg/s)
Flow from channel A (kg/s)
Flow from channel B (kg/s)
Flow rate (kg/s)
Time (s)
12
10
8
6
4
2
0
-2
0
50
100
150
200
250
300
350
400
450
500

Anti-Lock Braking System (ABS)

This example shows a simple way of modeling an ABS braking system. The model shows the velocity profile responses achieved for the vehicle CG and the wheels.

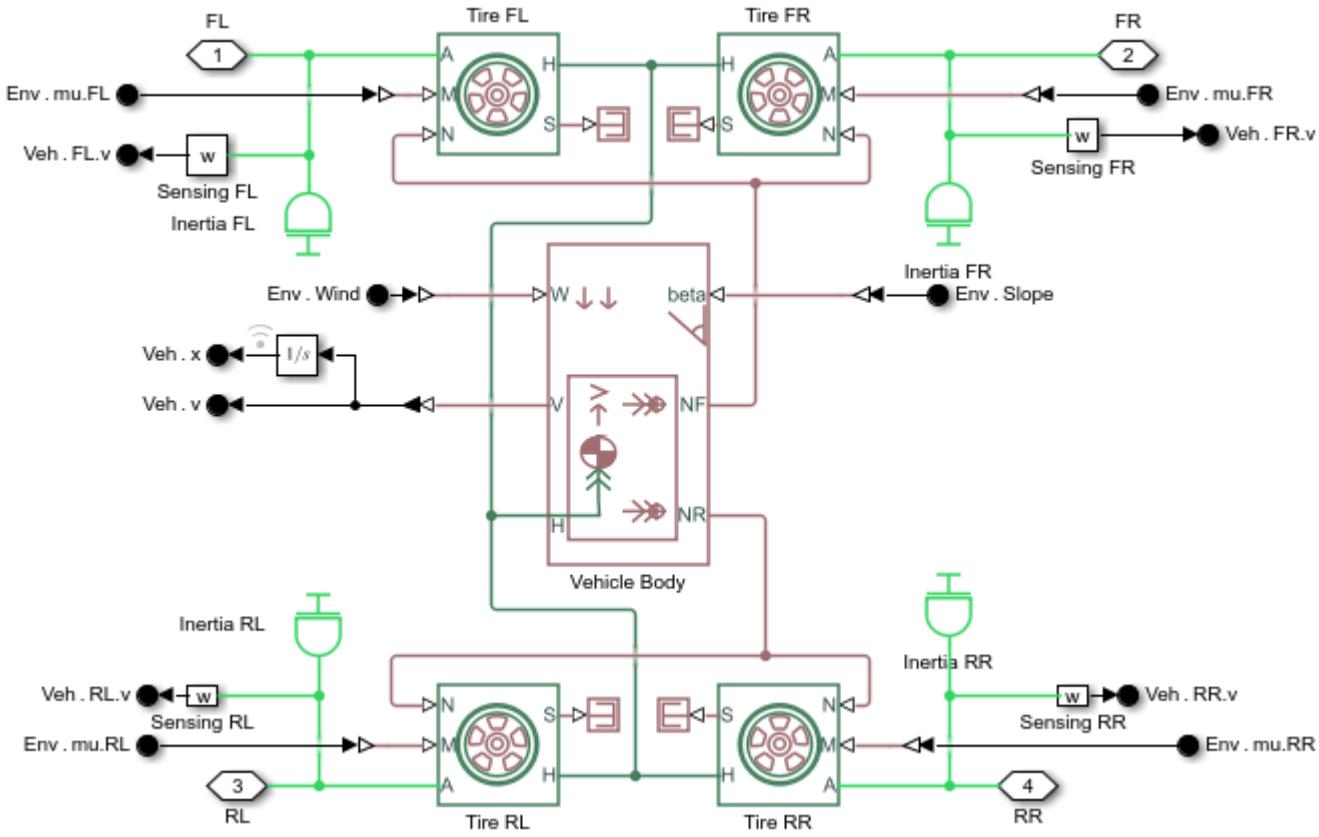
Model

The following figure shows the model of an ABS system with a test harness. The output of the model is the plot of the velocity profile achieved for the vehicle CG and the wheels.



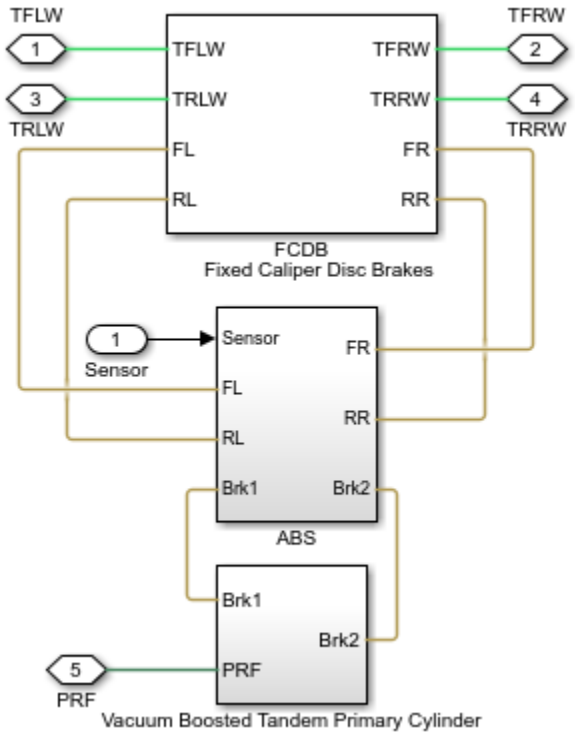
Vehicle subsystem

This subsystem shows the modeling of the vehicle body with the four wheels.



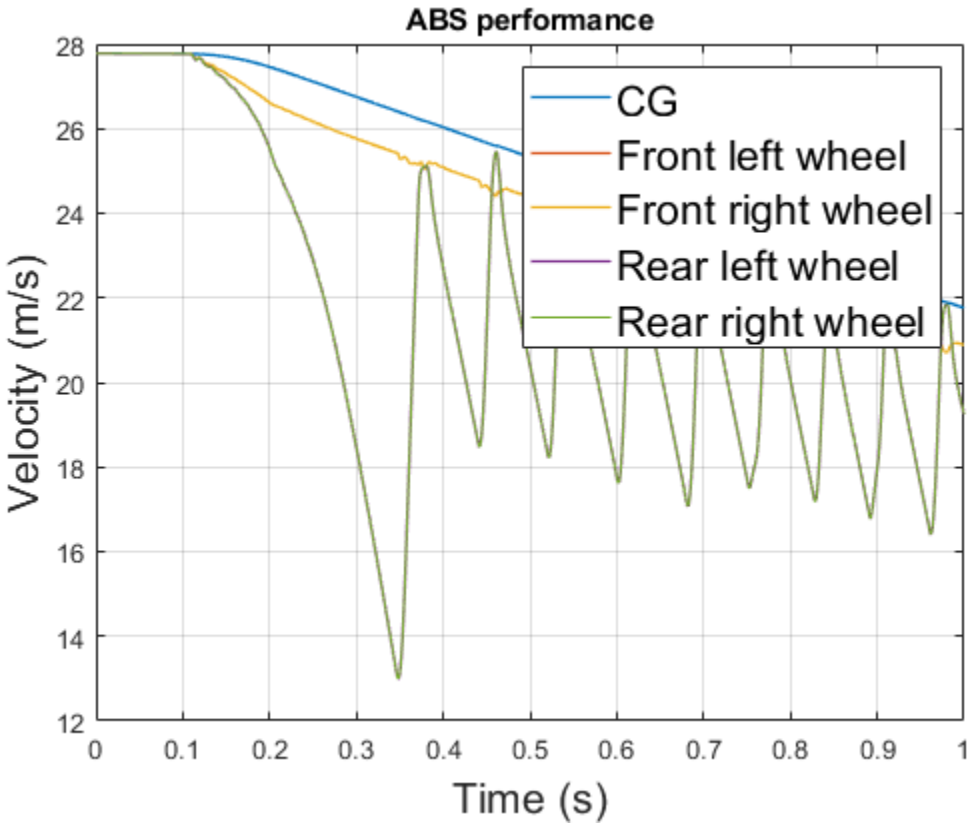
Brakes with ABS

This subsystem shows the modeling of the brakes with the ABS.



Simulation results from Simscape™ logging

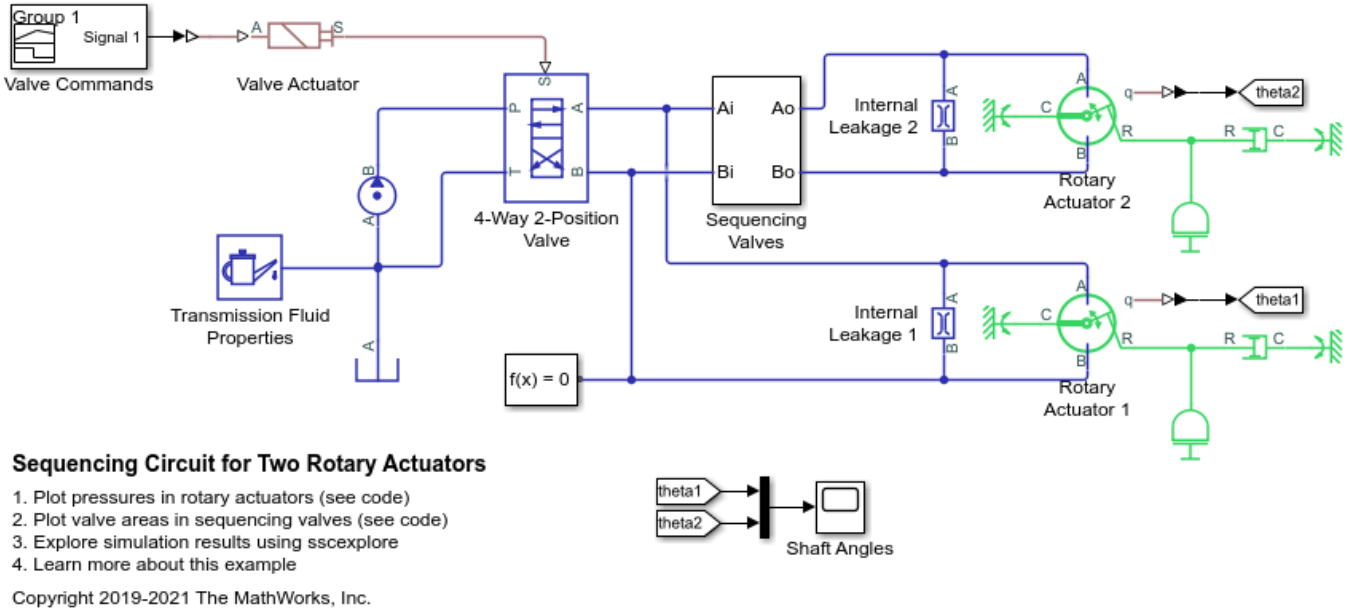
Based on the given inputs, the model generates a plot of the velocity profile achieved for the vehicle CG and the wheels.


ABS performance
CG
Front left wheel
Front right wheel
Rear left wheel
Rear right wheel
Velocity (m/s)
Time (s)
28
26
24
22
20
18
16
14
12
0
0.1
0.2
0.3
0.4
0.5
0.6
0.7
0.8
0.9
1

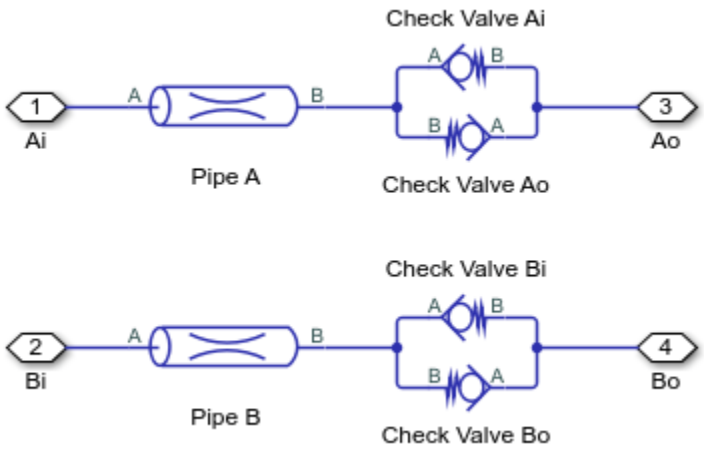
Sequencing Circuit for Two Rotary Actuators

This example shows a sequencing circuit that is based on four check valves installed in the pressure and return lines of the second rotary actuator. The cracking pressure of the meter-in check valves is set high enough to prevent flow into rotary actuator 2 while rotary actuator 1 is rotating, but lower than the pressure that develops once rotary actuator 1 reaches its hard stop. As a result, rotary actuator 2 starts moving only after rotary actuator 1 completes its stroke.

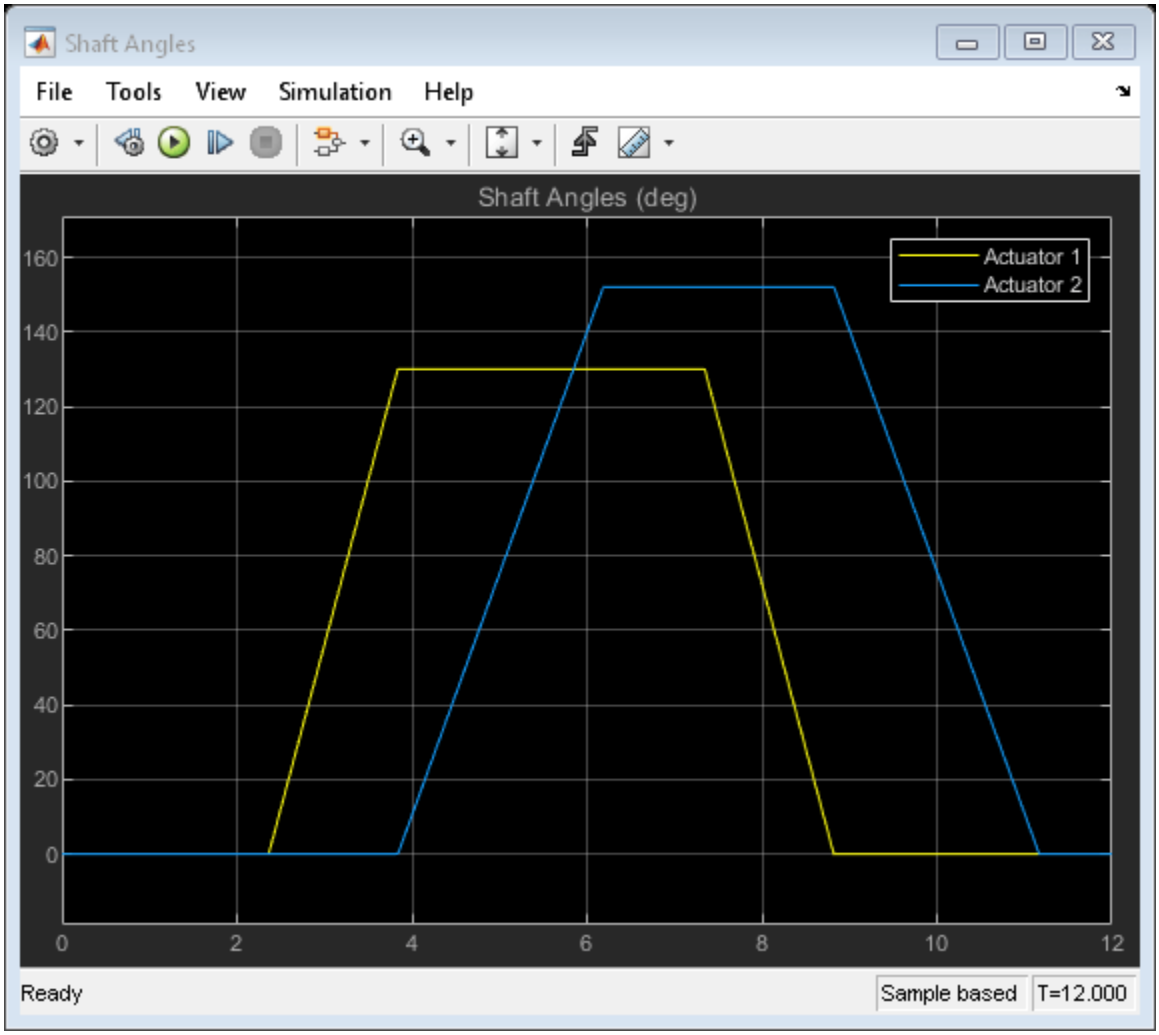
Model



Sequencing Valves Subsystem

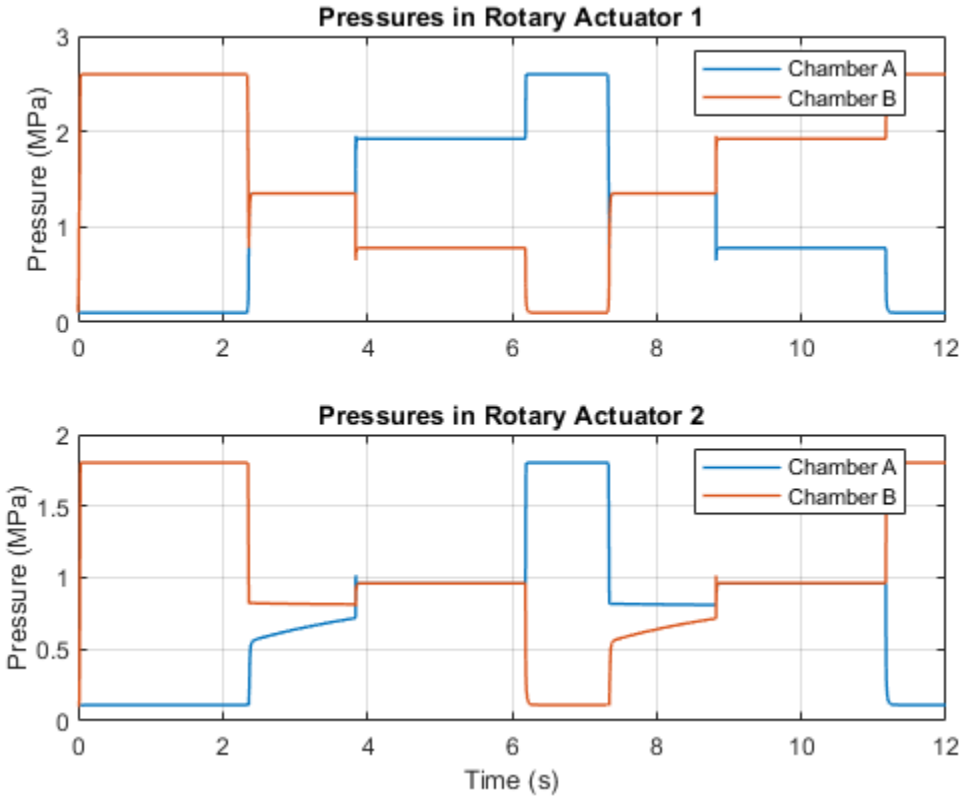


Simulation Results from Scopes

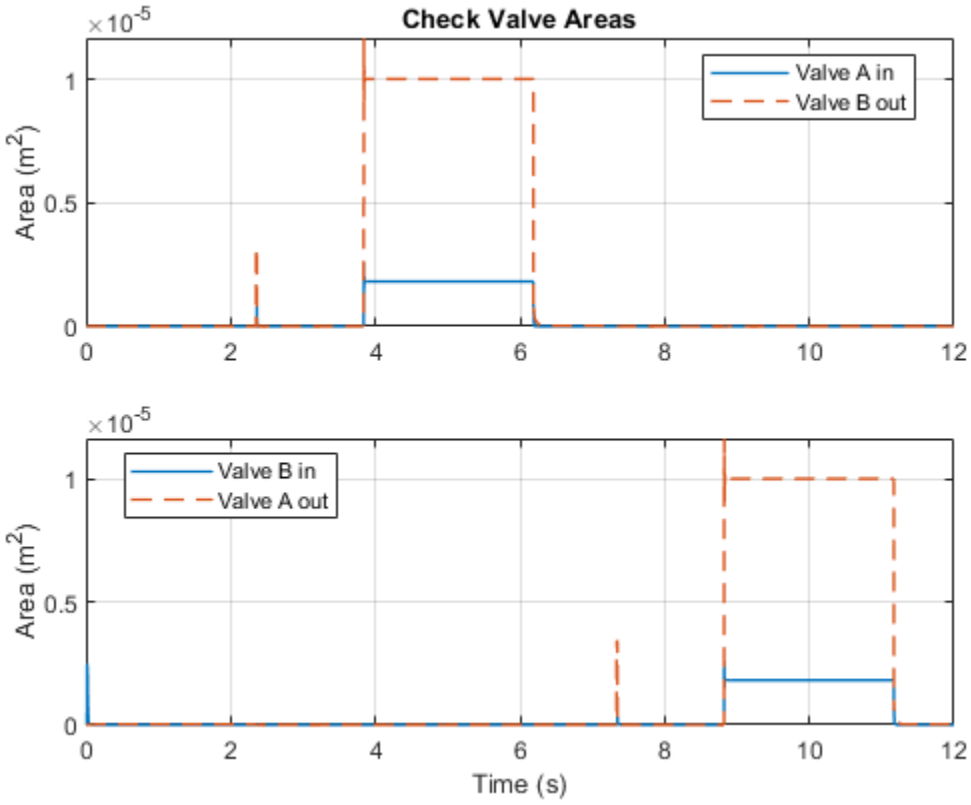


Simulation Results from Simscape Logging

This plot shows the sequence of pressures in the rotary actuator chambers. At the start of the simulation, both actuators are driven against their hard stops at 0 degree. At 2 seconds, the high pressure line is connected to chamber A of the actuators. The sequencing valves (check valves) prevent flow to rotary actuator 2 and maintain greater pressure in its chamber B over chamber A until rotary actuator 1 reaches its hard stop at 3.8 seconds. At that point, rotary actuator 1 is driven against its hard stop at maximum stroke and pressure builds up sufficiently to open the check valves. This lets flow into chamber A of rotary actuator 2 and causes it to rotate to its maximum stroke. At 7 seconds, the high pressure line switches and the opposite set of check valves repeat the sequence for the actuator rotation in the opposite direction.



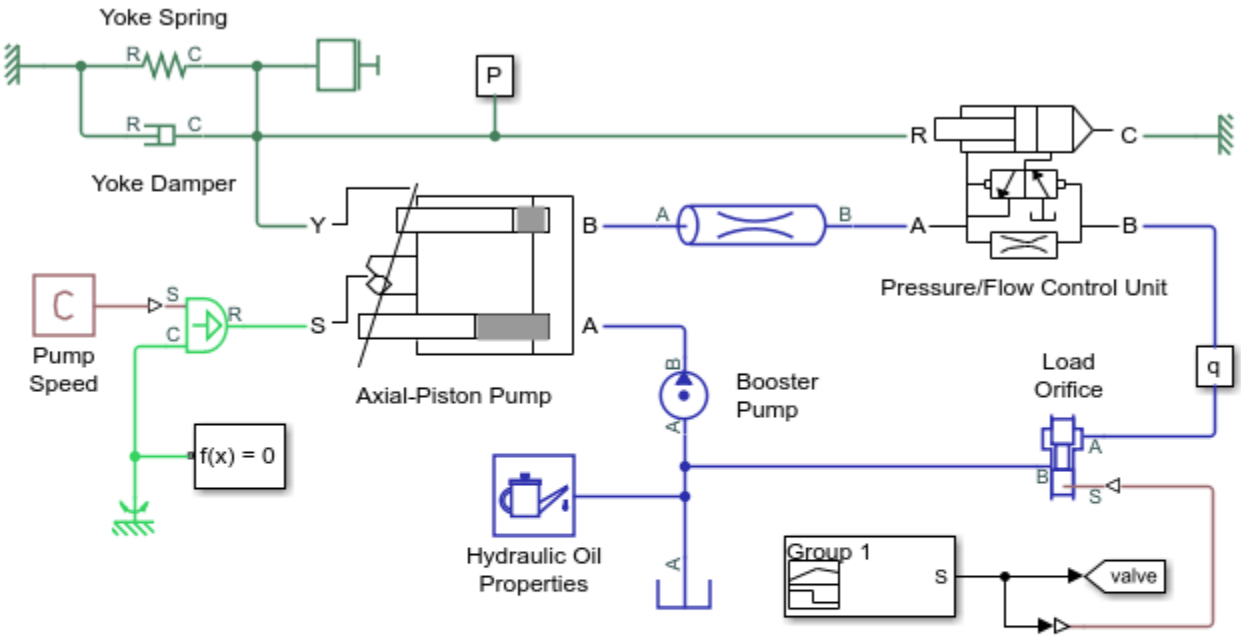
This plot shows the check valve areas that maintain the sequence of operations in the two rotary actuators. When rotary actuator 1 rotates, the meter-in check valve to chamber A of rotary actuator 2 remains closed until rotary actuator 1 completes its stroke. It then opens to enable flow into rotary actuator 2 so that it can rotate. The same sequence occurs when rotary actuator 1 rotates in the opposite direction.

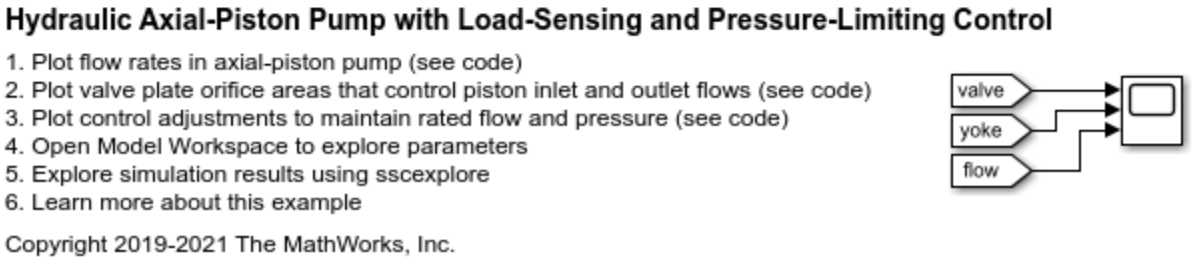

Check Valve Areas
×10^-5
Area (m^2)
Valve A in
Valve B out
Valve B in
Valve A out
Time (s)

Hydraulic Axial-Piston Pump with Load-Sensing and Pressure-Limiting Control

This example shows a test rig designed to investigate the interaction between an axial-piston pump and a typical control unit simultaneously performing the load-sensing and pressure-limiting functions. To increase the fidelity of the simulation, this example uses a detailed model of the pump that accounts for the interaction between the pistons, swash plate, and valve plate.

Model





Test Rig Description

The pump is modeled by the Axial-Piston Pump subsystem. The prime mover rotating the pump is represented by an angular velocity source. The pump's intake is connected to the output of a low pressure booster pump of 0.5 MPa. The pump output passes through a pipe, a control unit, and a variable orifice that acts as the load. To test the control unit's response to a variable load, the load orifice area is changed during simulation via the Signal Builder block.

The control unit is modeled by the Pressure/Flow Control Unit subsystem. The load-sensing function of the pump control uses an orifice installed on the output flow path of the pump. The control unit

keeps the pressure differential across this orifice constant regardless of the pump loading by adjusting the yoke displacement, which affects the tilt angle of the pump swash plate. This helps maintain the desired the flow rate through the orifice as well as prevent the pump pressure from exceeding the preset value.

The test rig basic parameters are:

Pump maximum displacement	7.8877e-6 m^3/rad
Cylinder block pitch radius	0.04 m
Piston diameter	0.015 m
Number of pistons	5
Maximum piston stroke	0.06 m
Swash plate maximum angle	35 deg (0.6109 rad)
Arm length between the control actuator and the swash plate pivoting point	0.055 m
Swash plate control actuator stroke	0.04 m
Diameter of the orifice at the bottom of the piston chamber	0.007 m
Pump maximum rated speed	2500 RPM
Maximum pressure	27 MPa
Rated flow rate	1.1e-3 m^3/s

Axial-Piston Pump Subsystem

The pump under investigation has five pistons. Its schematic is shown below:

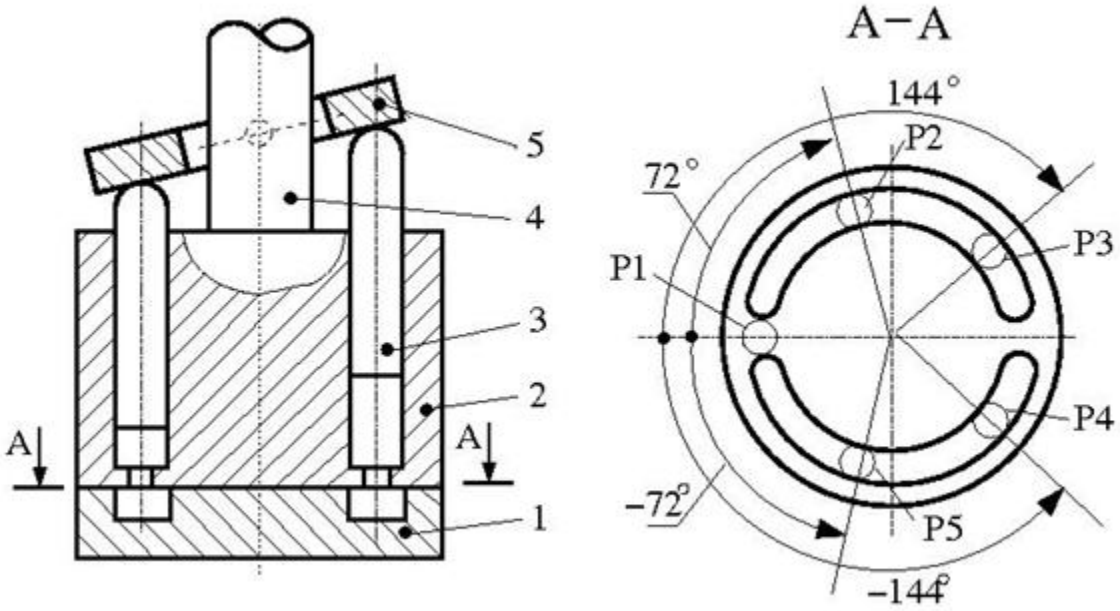


Figure 1: Axial-Piston Pump Schematic

where

1 - Valve plate (porting plate)

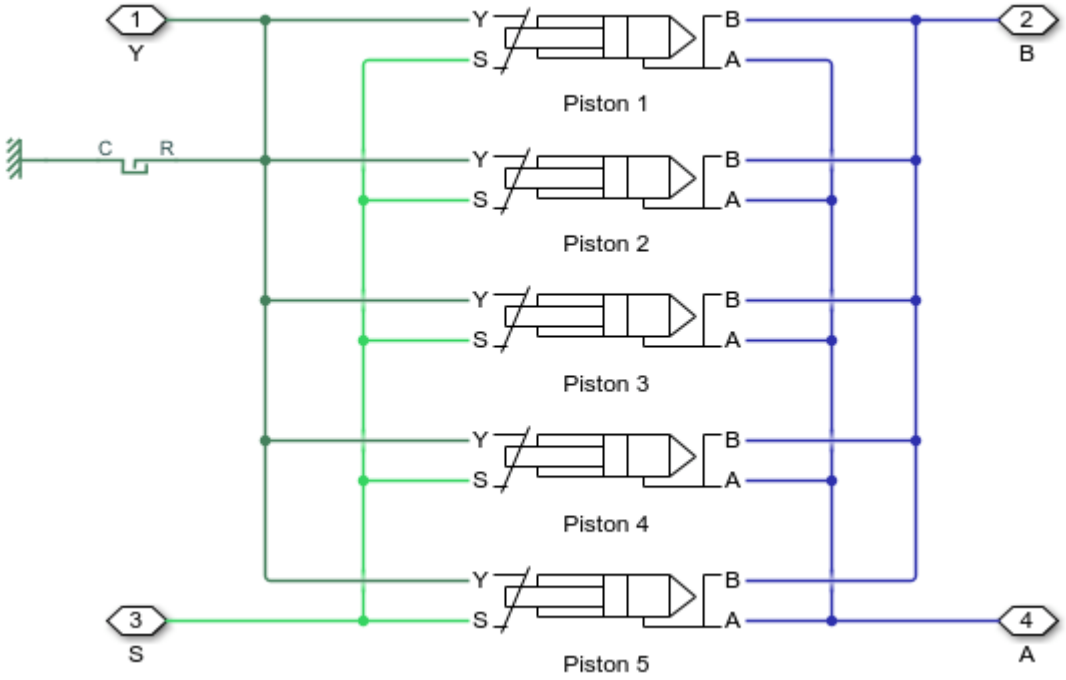
2 - Cylinder block (rotor)

3 - Piston

4 - Drive shaft

5 - Swash plate

The five pistons are modeled in the Axial-Piston Pump subsystem:

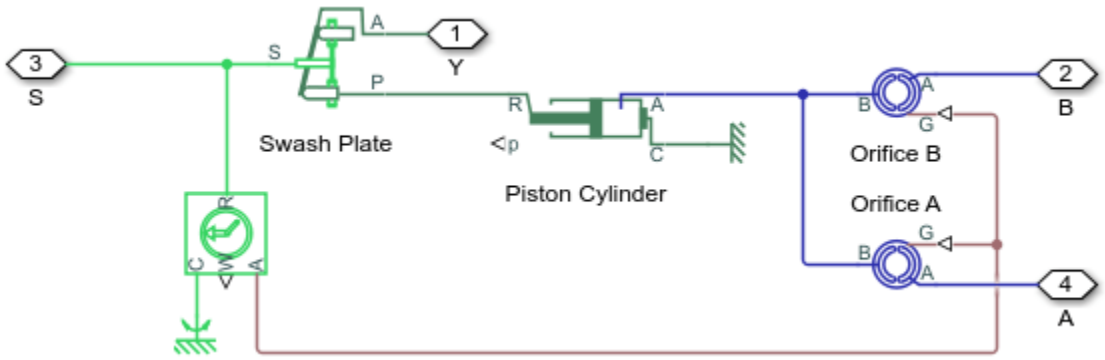


The pistons are identical and are modeled in their own identical subsystems. They are all connected to the following ports of the pump model:

- S - Drive shaft
- Y - Yoke connected with the inclined plate of the swash mechanism
- A - Pump inlet (intake) port
- B - Pump outlet (discharge) port

The yoke displacement is limited by a hard stop.

Piston Subsystem

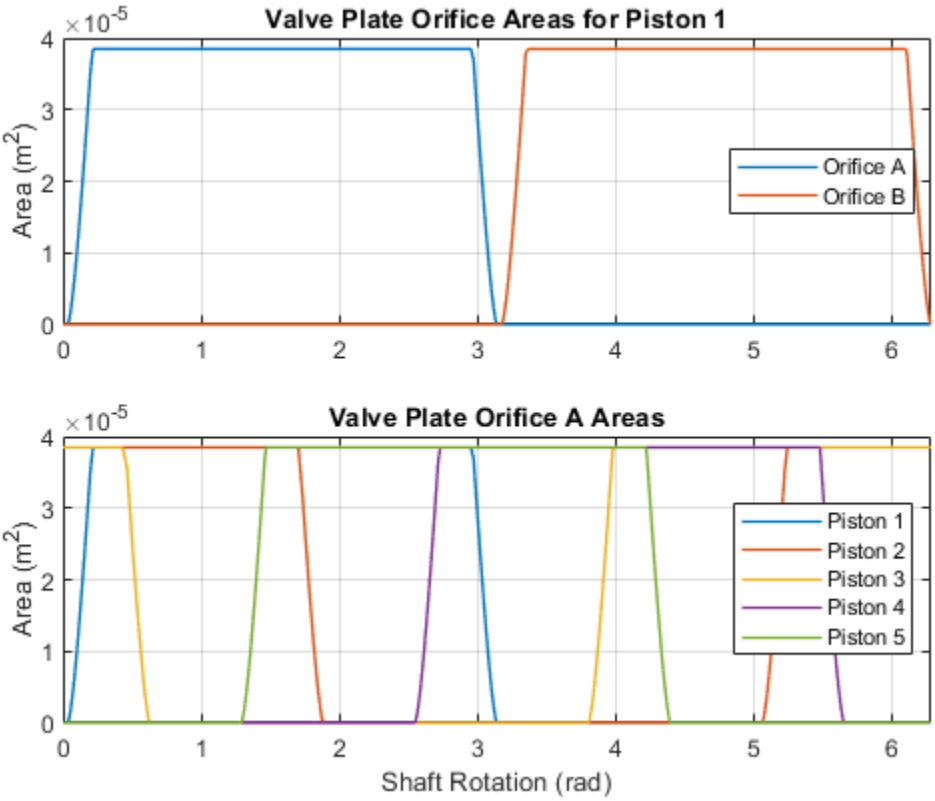


The piston model consists of a Single-Acting Actuator (IL) block connected mechanically to the drive shaft via the Swash Plate block. The piston is also hydraulically connected to ports A and B via two Valve Plate Orifice (IL) blocks. Ports A and B represent the pump intake and discharge ports, respectively.

The piston sucks in liquid during its upstroke and expels liquid during its downstroke. The two Valve Plate Orifice (IL) blocks models the interaction between the fixed orifice at the bottom of the piston chamber and the two crescent-shaped grooves on the valve plate so that the piston is connected to port A during its upstroke and port B during its downstroke.

The following plot shows the flow areas formed by the interaction between the piston and the two crescent-shaped grooves on the valve plate. As the cylinder block rotates, the piston chamber orifice slides over the groove to open the flow area and then slides off the groove to close the flow area. The two grooves are separated by a phase angle of 180 degrees so that the piston slides over the groove connected to the intake (A) during the upstroke and slides over the groove connected to the discharge (B) during the downstroke. This results in the alternately open flow areas as shown in the first subplot.

The pistons are spread evenly along the pitch circle of the cylinder block, as shown in Figure 1. This makes the angle between pistons 360/5 = 72 degrees. Therefore, the phase angles of the Valve Plate Orifice (IL) blocks of each piston is offset from the previous piston by 72 degrees. This results in the evenly distributed and overlapping open flow areas for the intake groove as shown in the second subplot.



The tilt of the swash plate is what produces the piston motion as the cylinder block rotates. A Swash Plate block is connected to the Single-Acting Actuator (IL) to actuate that piston. Similar to the Valve Plate Orifice (IL) blocks, the phase angle of the Swash Plate block of each piston is offset from the previous piston by 72 degrees.

A greater swash plate angle results in larger piston motion which produces greater flow volume. The piston stroke must be long enough to allow the piston to reciprocate even at the maximum swash plate angle:

$$x_{stroke} > 2r_p \tan \alpha_{max}$$

where $r_p = 0.04$ m is the pitch radius and $\alpha_{max} = 35$ deg is the maximum swash plate angle. This means the piston stroke must be greater than 0.056 m and is thus set to 0.06 m.

The piston initial positions must be equal to half of the stroke if the swash plate angle is zero. It is offset from half stroke depending on the phase angle of the piston if the initial swash plate angle is nonzero. The initial swash plate angle depends on the initial control actuator position. Therefore, the initial position of each piston is

$$x_{p0} = \frac{x_{stroke}}{2} - x_{a0}\frac{r_p}{r_a}\cos\gamma$$

where x_{a0} is the swash plate control actuator initial position, r_a is the swash plate control actuator arm, and γ is the phase angle of the piston in the cylinder block. This calculation is implemented in the Initialization tab of the Axial-Piston Pump subsystem mask.

Pressure/Flow Control Unit Subsystem

The purpose of the control unit is to implement two functions: load sensing and pressure limiting. Load sensing is implemented by maintaining a set pressure differential across a flow control orifice. The flow control orifice is installed in the output flow of the pump. The following figure shows the schematic of the control unit where the ports A and B are connected in series with the pump output.

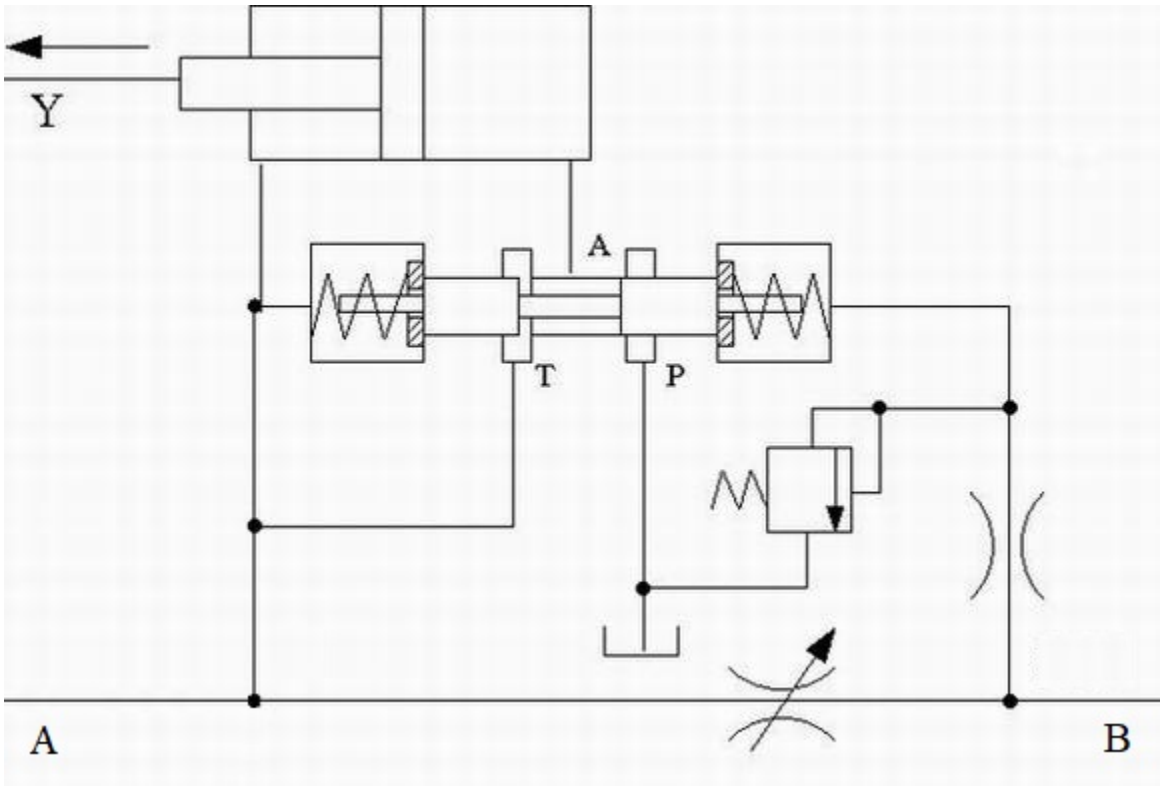
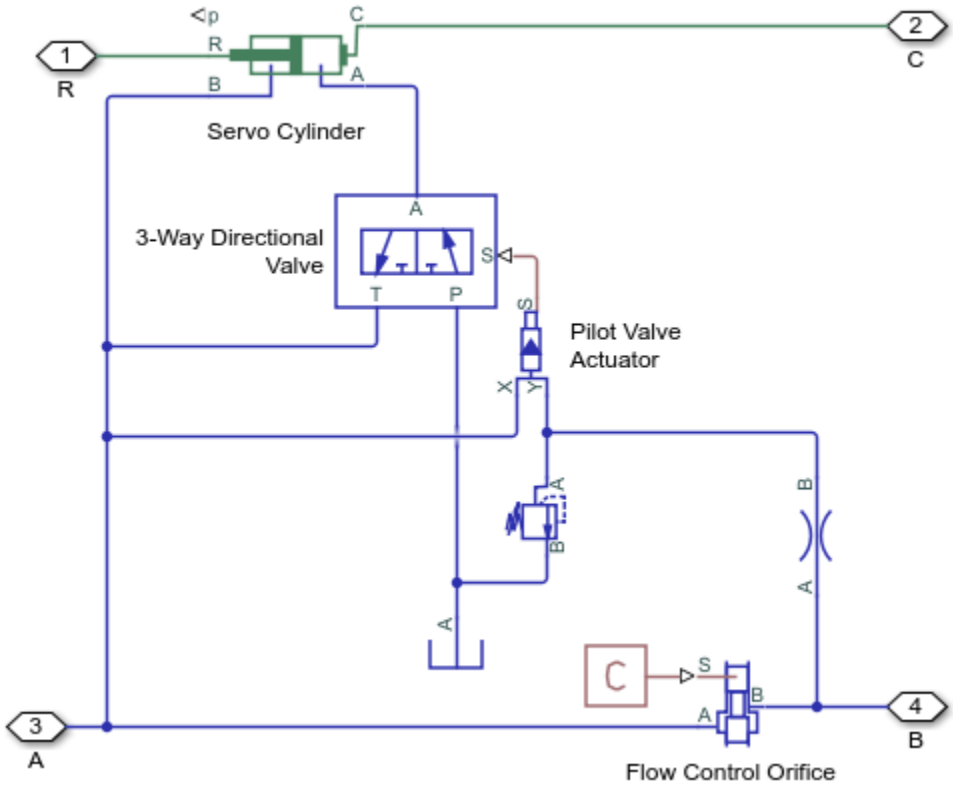


Figure 2: Pressure/Flow Control Unit Schematic

The pressure before and after the flow control orifice acts on the side faces of the 3-way directional valve and shifts the valve spool proportionally to the pressure difference and the setting of the two centering springs. The valve connections are selected such that an increase in pressure differential across the flow control orifice opens the directional valve path P-A and closes path A-T. The control actuator is arranged as a single-rod differential hydraulic cylinder with the rod connected to the yoke of the swash plate. The pump displacement is increased if the rod moves in the positive direction shown in the schematic. Due to the difference in piston areas of the two sides of the cylinder, the displacement is increased if both cylinder chambers are connected ot the pump and decreased if the chamber without the rod is connected to the tank. As a result, an increase in pressure differential across the flow control orifice causes the pump to decrease its displacement until it returns to the preset value.

The pressure-limiting function prevents the pump output pressure from exceeding the preset value. It is implemented with a pressure relief valve and a fixed orifice on the pressure line downstream of the flow control orifice. The pressure relief valve is set to the desired maximum value. When the pump pressure builds up to this value, the pressure relief valve opens to lower the pressure and causes the directional valve to increase path P-A decrease path A-T. This shifts the control actuator to decrease its displacement until pressure returns to the preset value.

The model of the control unit is shown below:



The control actuator is modeled with the Double-Acting Actuator (IL) block. The flow control orifice is modeled with the Spool Orifice (IL) block with the rectangular slot geometry option. The opening area of this orifice is chosen to give the desired rated flow rate of 1.1e-3 m^3/s for the load-sensing function.

The combination of the 3-Way Directional Valve (IL) block and the Pilot Valve Actuator (IL) block represents the directional valve in the schematic in Figure 2. Ports X and Y of the pilot valve senses the pressure acting on the two sides of the directional valve and adjusts the spool position of the 3-Way Directional Valve (IL) block. The centering springs in the Pilot Valve Actuator (IL) block returns the spool to the neutral position. Path A-T of the 3-Way Directional Valve (IL) block is set to be half open in the neutral position to force the pump to increase its displacement at the start of the operation. Other parameters such as spring stiffness, valve stroke, valve orifice areas, etc., are tuned in the model for accuracy and stability.

The set pressure differential in the pressure relief valve is set to 25 MPa. Since the load-sensing function maintains a roughly 2 MPa pressure difference across the flow control orifice, this causes the pressure-limiting function to maintain a maximum pressure of 27 MPa at port A of the control unit.

Test Rig Operation

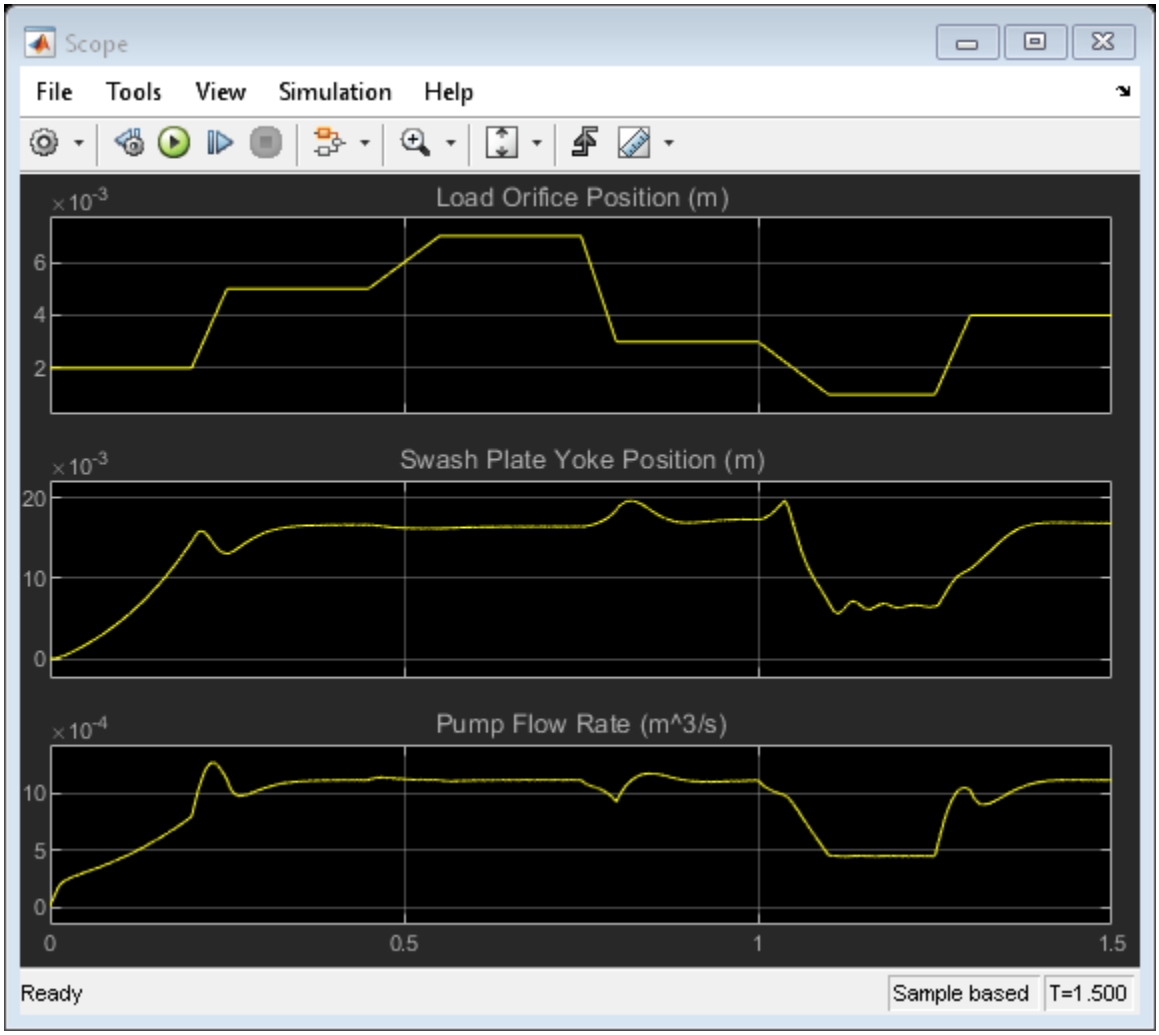
The load of the test rig is represented by a Spool Orifice (IL) block. The simulation consists of six different loading conditions achieved by changing the spool position of the load orifice to 2 mm, 5 mm, 7 mm, 3 mm, 1 mm, and 4 mm. The control actuator initially increases the pump displacement to

build up pressure. The process settles at around 0.35 s after the pressure difference across the flow control orifice becomes close to 2 MPa and the rated flow rate of 1.1e-3 m^3/s is achieved.

During the next three loading conditions, the control unit maintains the same pump delivery despite the varying load orifice opening.

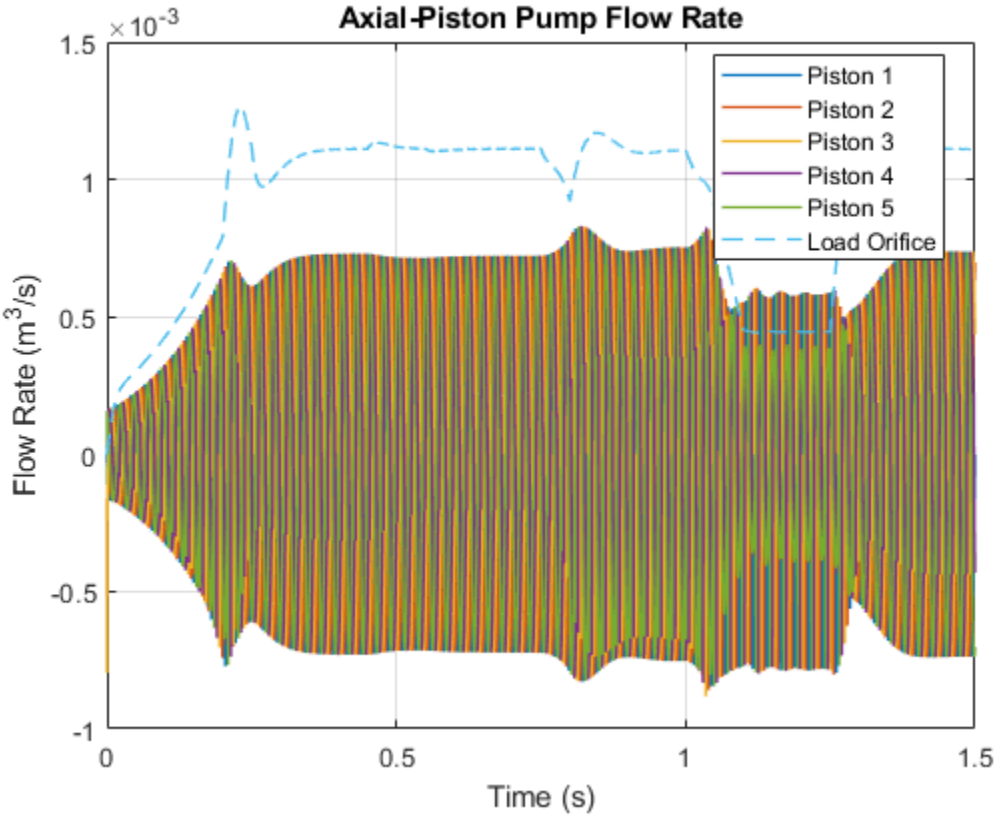
At 1 s, the load orifice is almost closed, causing pump pressure to rise. The pressure-limiting function becomes dominant as the pressure reaches 27 MPa. The control unit returns to the load-sensing mode after the pressure falls below 27 MPa.

Simulation Results from Scopes

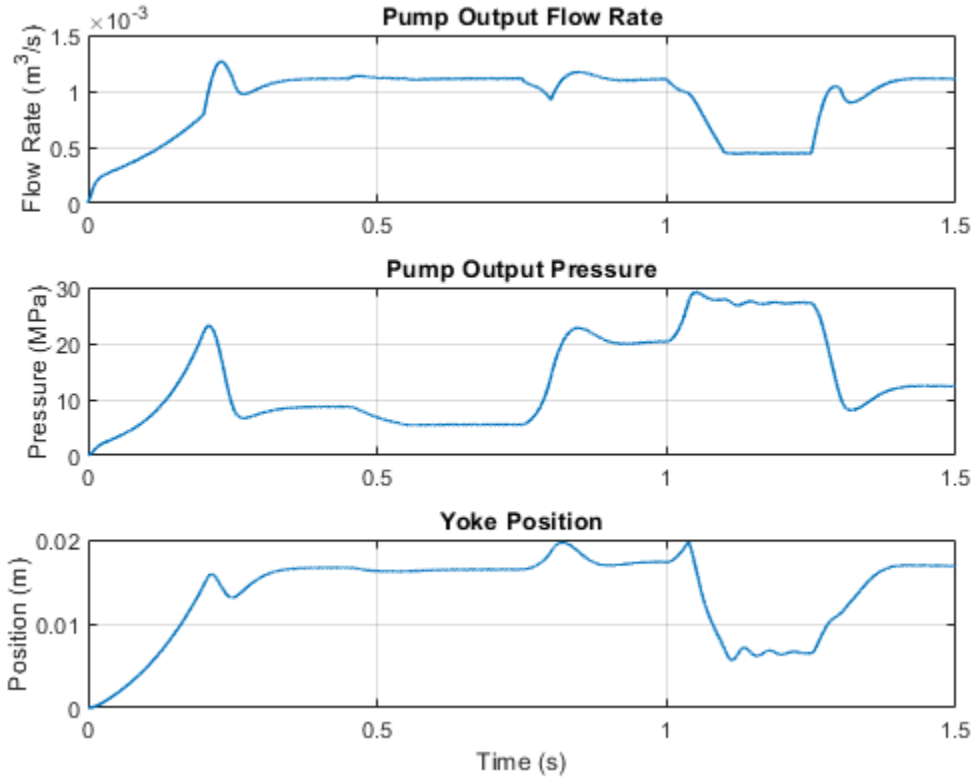


Simulation Results from Simscape Logging

This plot shows the flow rate within the axial-piston pump. The cyclic nature of the piston motion can be seen in the flow rate in each piston. The negative portion of the piston flow rate cycle corresponds to liquid being drawn in through valve plate A and the positive portion of the piston flow rate cycle corresponds to liquid being expelled through valve plate B. The combined effect of the five pistons results in the overall behavior of pump which produces a positive flow rate of about 1.1e-3 m^3/s through the load orifice.



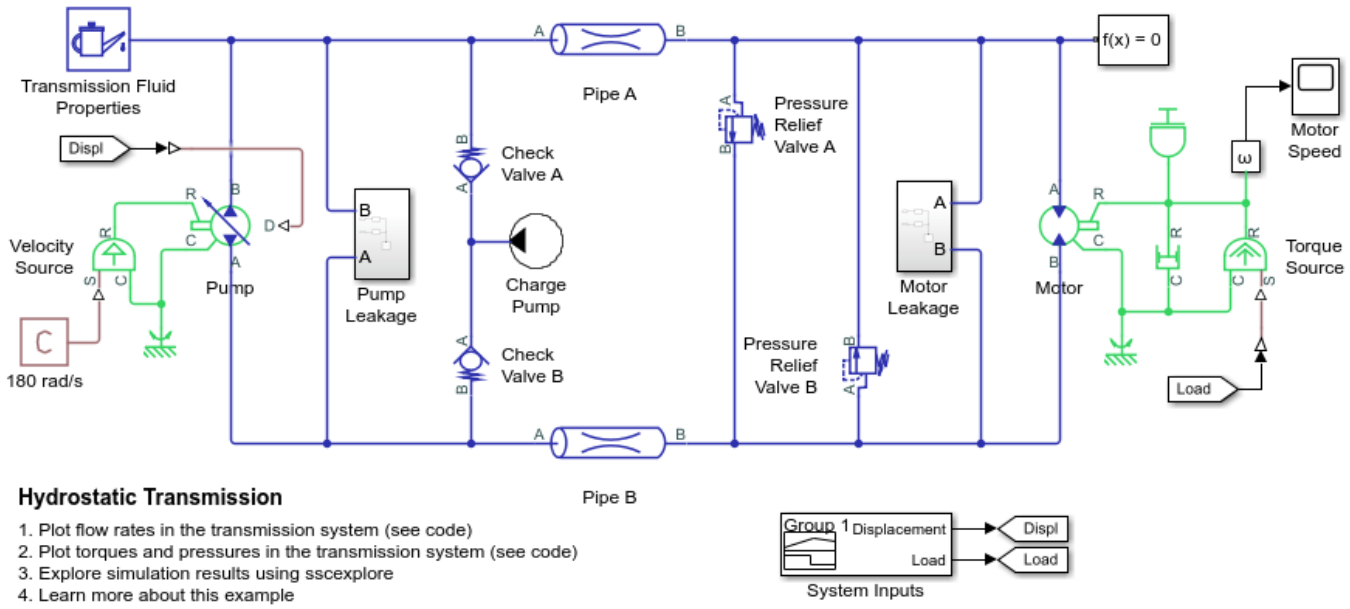
This plot shows the load-sensing and pressure-limiting control. The pump maintains its rated flow rate of 1.1e-3 m^3/s even as the load varies. However, as the pump output pressure rises to its maximum rated pressure of 27 MPa, the pressure-limiting control decreases the yoke position and the flow rate drops below its rated flow rate.


Pump Output Flow Rate
×10^-3
Flow Rate (m^3/s)
1.5
1
0.5
0
0
0.5
1
1.5
Pump Output Pressure
Pressure (MPa)
30
20
10
0
0
0.5
1
1.5
Yoke Position
Position (m)
0.02
0.01
0
0
0.5
1
1.5
Time (s)

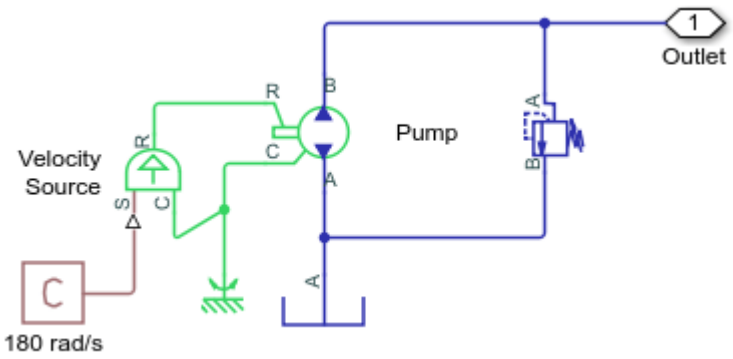
Hydrostatic Transmission

This example shows a hydraulic transmission system built of a variable-displacement pump and a fixed-displacement motor. The pipes between the pump and the motor are connected by two replenishing valves (check valves) and a charge pump on the pump side and two pressure relief valves on the motor side. The motor dirves a mechanical load consisting of an inertia, viscous friction, and time-varying torque. The system is tested with both positive and negative pump flow rate by varying the pump displacement.

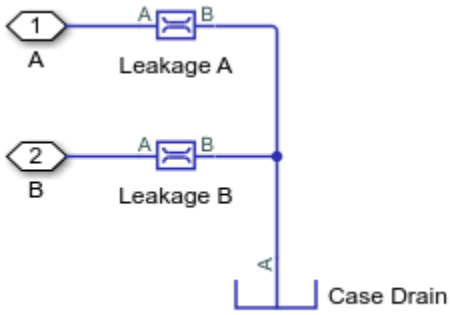
Model



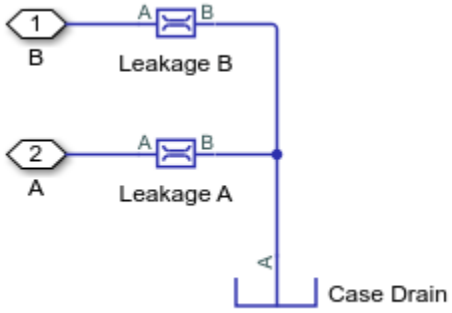
Charge Pump Subsystem



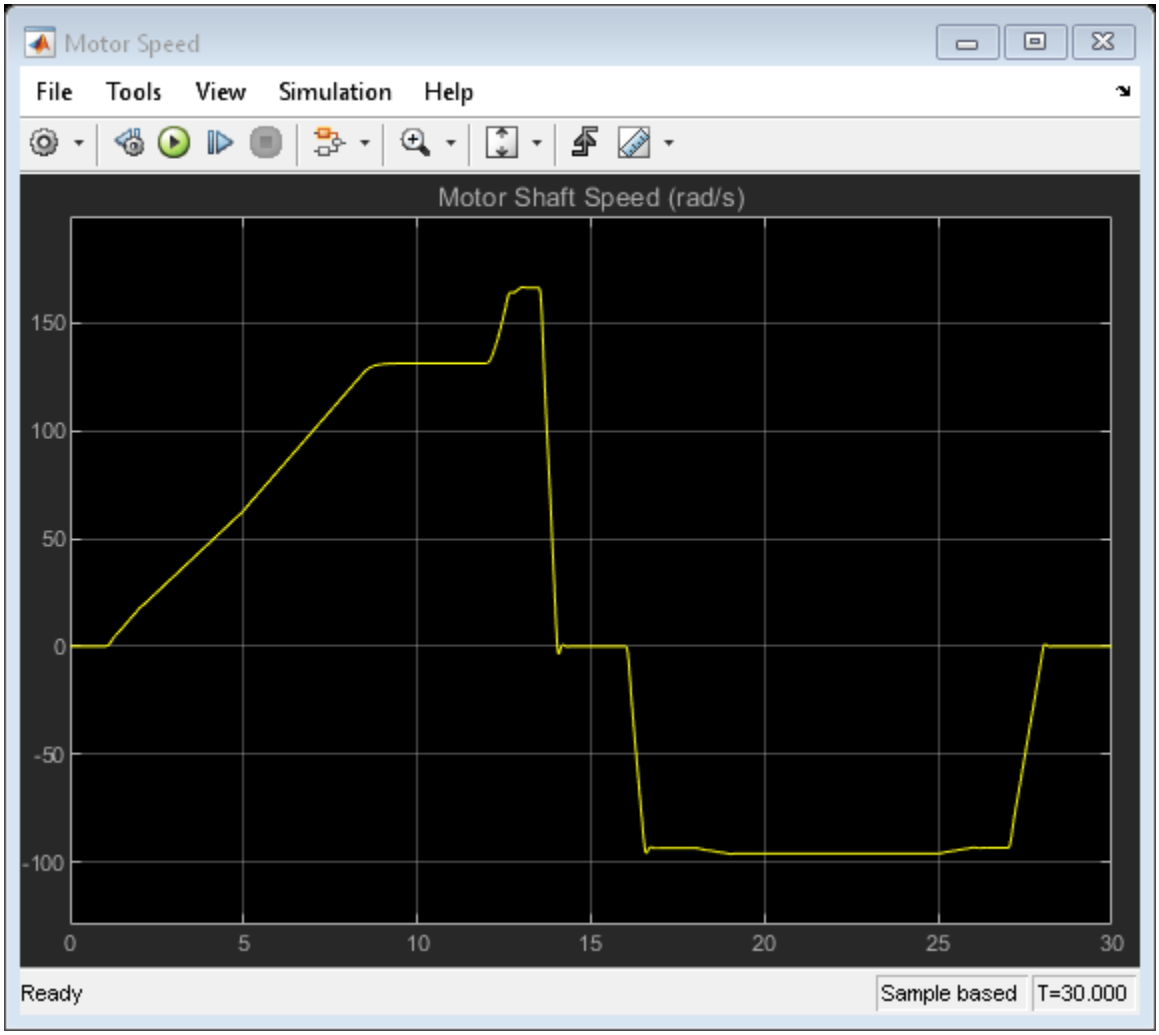
Motor Leakage Subsystem



Pump Leakage Subsystem

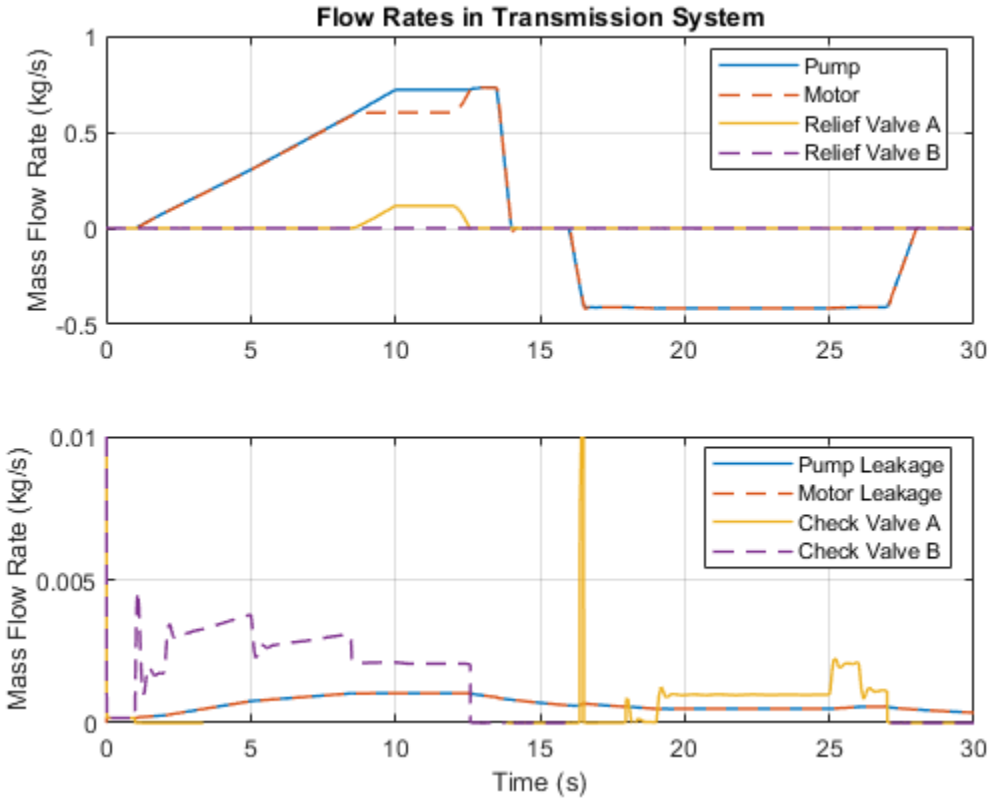


Simulation Results from Scopes

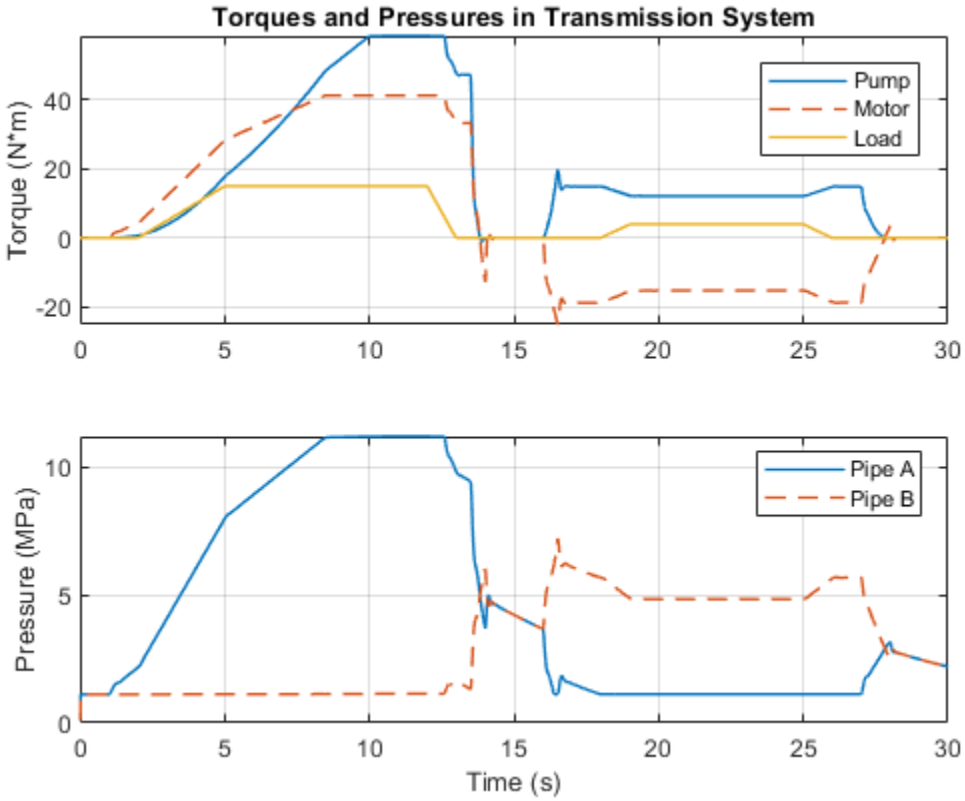


Simulation Results from Simscape Logging

This plot shows the mass flow rates through the pump, motor, and valves in the transmission system. Flow from the charge pump through the check valves compensate for external leakage through the pump and motor casing.



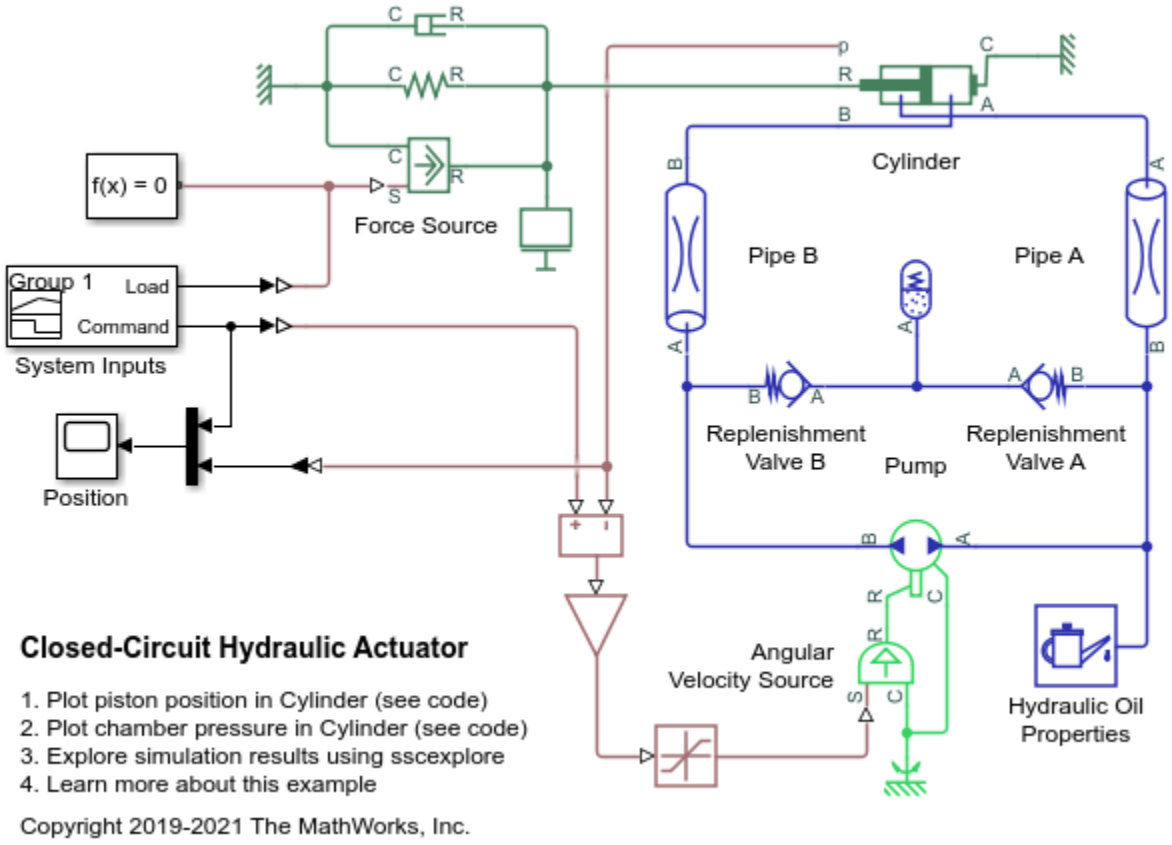
This plot shows the torque generated by the motor as well as the torque driving the pump and the mechanical load. It also shows the pressures in the two transmission pipes switch between high and low pressures as the pump is reversed.


Torques and Pressures in Transmission System
Torque (N*m)
40
20
0
-20
0
5
10
15
20
25
30
Pump
Motor
Load
Pressure (MPa)
10
5
0
Pipe A
Pipe B
Time (s)

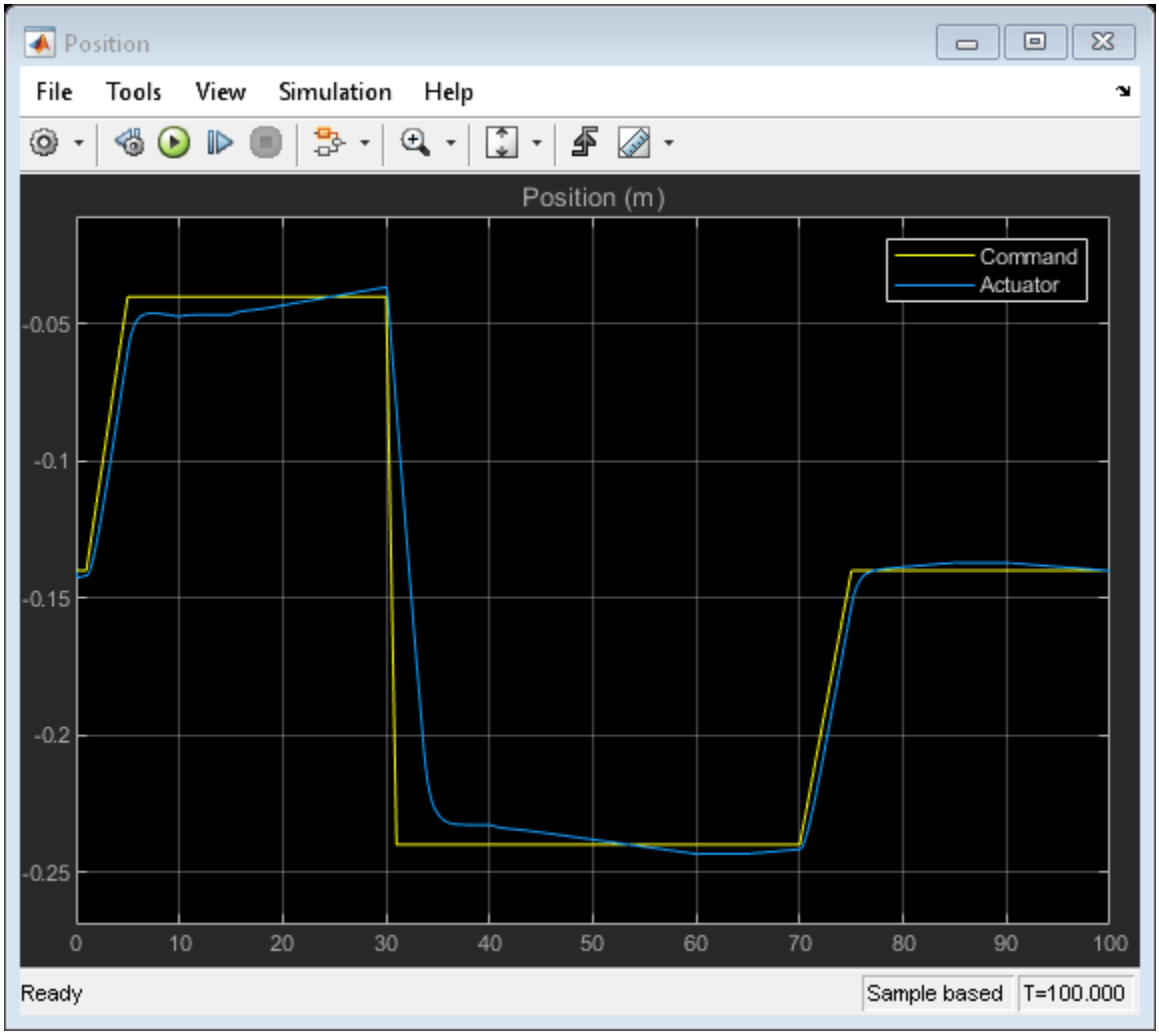
Closed-Circuit Hydraulic Actuator

This example shows a closed-circuit hydraulic actuator driven by a variable-speed pump. The actuator is arranged as a closed fluid system with two replenishment valves (check valves) and a spring-loaded accumulator serving as a replenishment reservoir. The pump speed is controlled by the difference between the commanded and measured piston position. The actuator acts against a spring, a damper, and a time-varying load.

Model

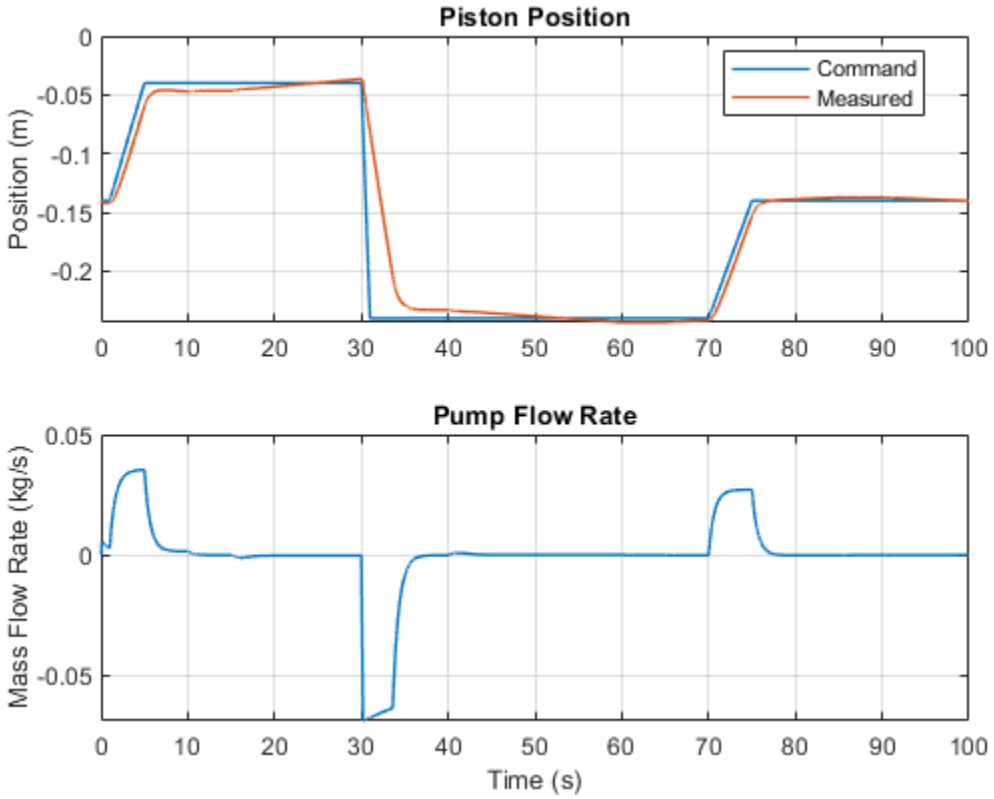


Simulation Results from Scopes

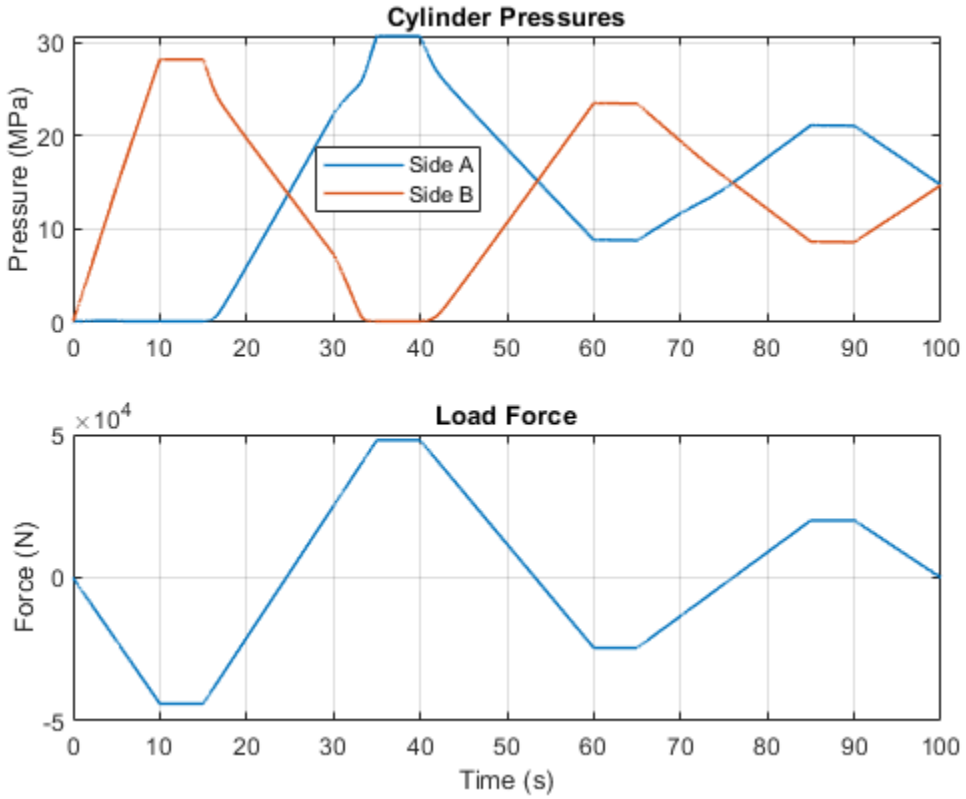


Simulation Results from Simscape Logging

This plot shows the position of the piston and the flow rate through the pump.



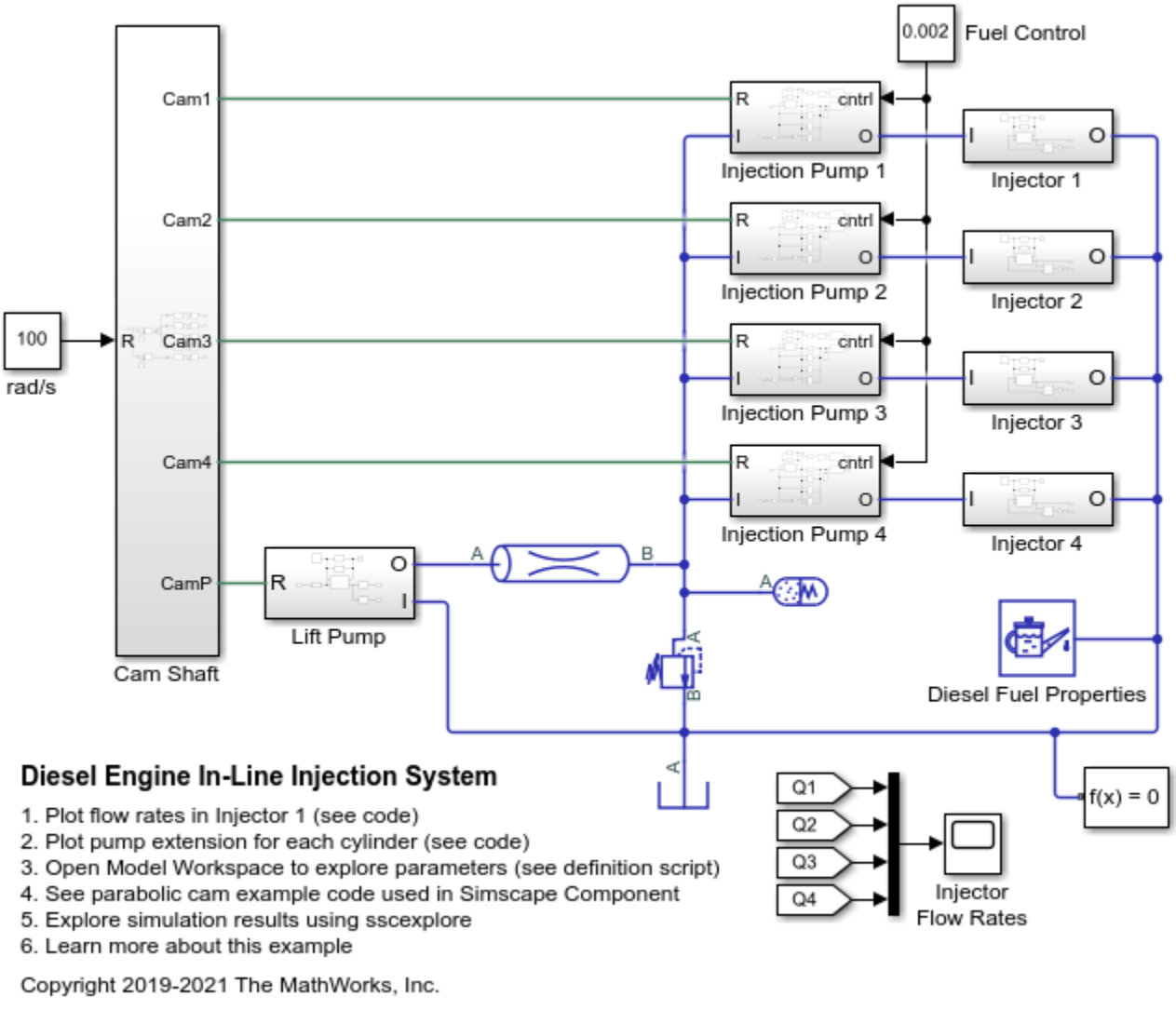
This plot shows the pressures inside the cylinder chambers and the time-varying load on the piston.


Cylinder Pressures
Pressure (MPa)
Side A
Side B
Load Force
×10^4
Force (N)
Time (s)

Diesel Engine In-Line Injection System

This example shows an in-line multi-element diesel injection system. It consists of a cam shaft, a lift pump, four in-line injection pumps, and four injectors.

Model



Injection System Description

The diesel injection system simulated by this model is shown in the schematic below.

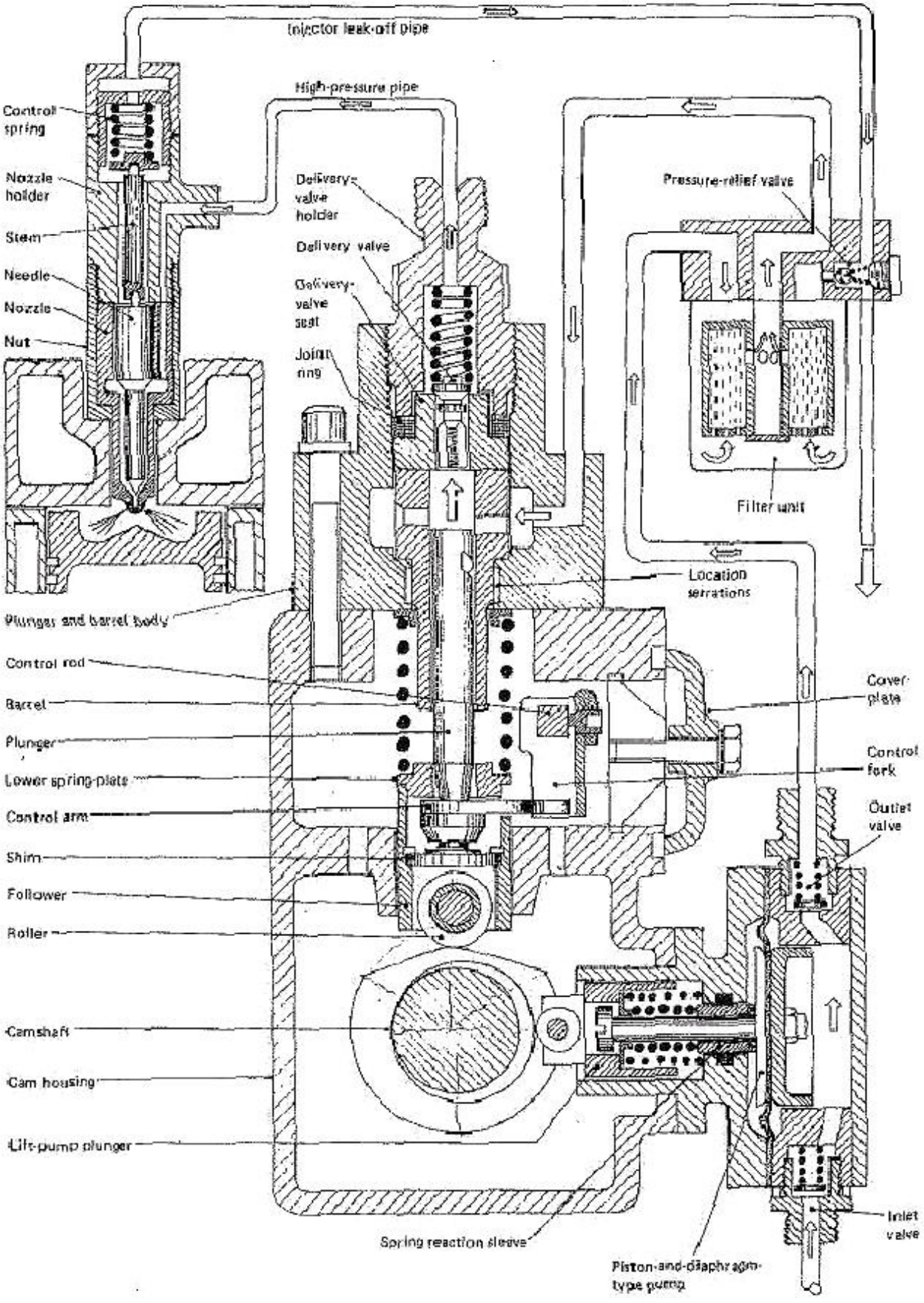

Injector leak-off pipe
High-pressure pipe
Control spring
Nozzle holder
Stem
Needle
Nozzle
Nut
Delivery-valve holder
Delivery valve
Delivery-valve seat
Joint ring
Pressure-relief valve
Filter unit
Location serrations
Plunger and barrel body
Control rod
Barrel
Plunger
Lower spring-plate
Control arm
Shim
Follower
Roller
Camshaft
Cam housing
Lift-pump plunger
Cover plate
Control fork
Outlet valve
Inlet valve
Spring reaction sleeve
Piston-and-diaphragm-type pump

Figure 1: Injection System Schematic Diagram

The system structure is reproduced from H. Heisler, *Vehicle and Engine Technology* (second edition), 1999, and is categorized as an in-line multi-element injection system. It consists of the following basic units:

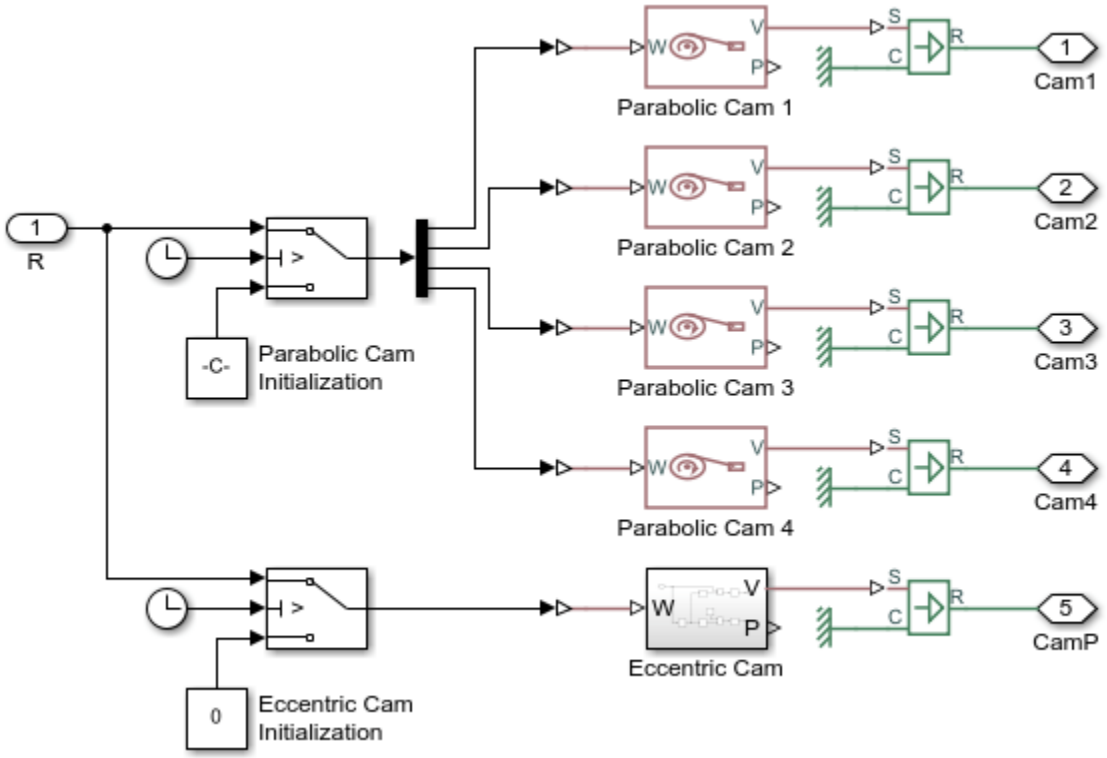
- Cam shaft
- Lift pump
- In-line injection pump with four pumping elements, one element per engine cylinder
- Injectors

The cam shaft carries five cams. The first one is the eccentric cam to actuate the lift pump. The remaining four drive the plungers of the injection pump. The cams are installed in such a way that pumping elements deliver fuel in the firing order and at the correct instant in the engine's cycle of operation. The lift pump supplies fuel to the intake of injection pump elements. Each element of the injection pump consists of a cam-driven plunger, a delivery valve, and the governor assembly. The purpose of the governor is to control the volume of the fuel delivered by the plunger to the engine cylinder.

The objective of the simulation is to investigate the entire system operation. The objective dictates the extent of idealization of each component in the system. All of the system units will be described in more detail in the following sections.

Note: The model of the system does not represent any particular injection system. All workspace parameters have been assigned based on practical considerations and do not represent any particular manufacturer parameters.

Cam Shaft Subsystem



The model of a cam shaft is built of five cam models. There are four parabolic profile cams and one eccentric cam. This motion profile is computed as a function of the shaft angle, obtained by integrating the shaft angular velocity. The cam that drives the plunger of the injection pump has a parabolic profile under which the follower moves back and forth at constant acceleration. It is described by the following set of equations:

$$x = 2s\left(\frac{\theta - \theta_E}{\alpha}\right)^2, \quad \text{if} \quad \theta_E \le \theta < \theta_E + \frac{\alpha}{2}$$

$$x = s\left[1 - 2\left(1 - \frac{\theta - \theta_E}{\alpha}\right)^2\right], \quad \text{if} \quad \theta_E + \frac{\alpha}{2} \le \theta < \theta_E + \alpha$$

$$x = s, \quad \text{if} \quad \theta_E + \alpha \le \theta < \theta_R$$

$$x = s\left[1 - 2\left(\frac{\theta - \theta_R}{\beta}\right)^2\right], \quad \text{if} \quad \theta_R \le \theta < \theta_R + \frac{\beta}{2}$$

$$x = 2s\left(1 - \frac{\theta - \theta_R}{\beta}\right)^2, \quad \text{if} \quad \theta_R + \frac{\beta}{2} \le \theta < \theta_R + \beta$$

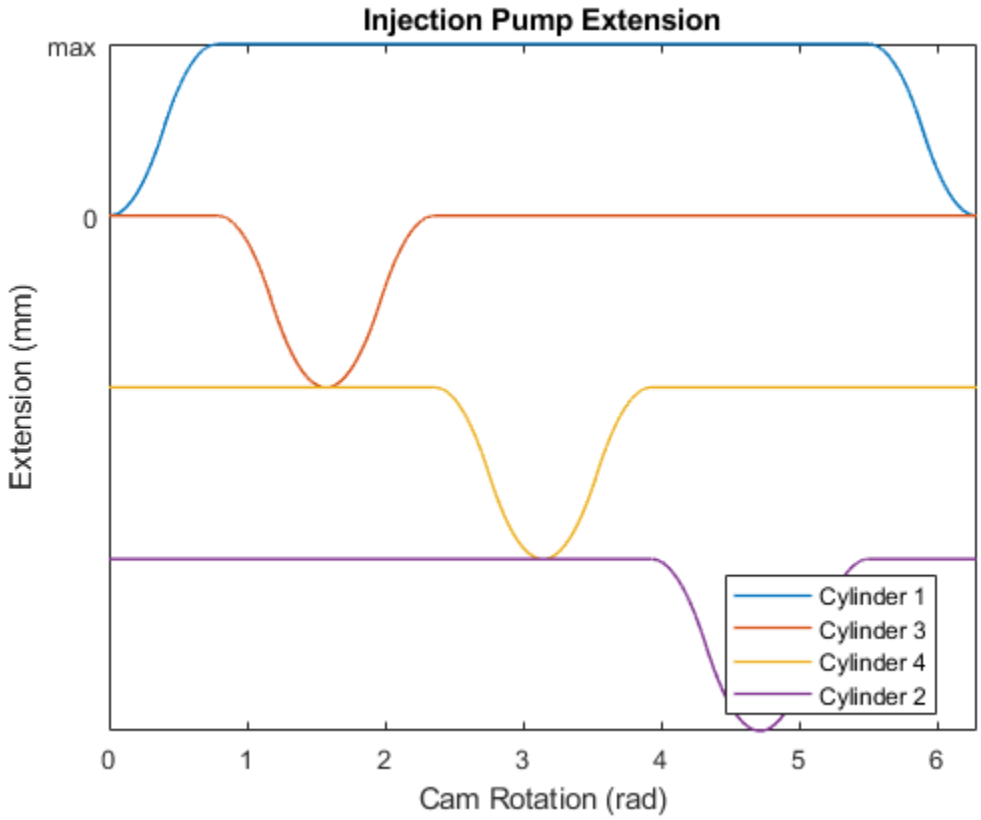
where

- x is the follower position
- s is the follower stroke
- θ is the cam angle (with phase angle accounted)
- θ_E is the start of extension angle
- α is the extension angle
- θ_R is the start of retraction angle
- β is the retraction angle

As a result, at the *start of extension angle* the follower starts moving up and reaches its top position after the shaft turns an additional *extension angle*. The follower starts the return stroke at the *start of retraction angle* and it takes *retraction angle* to complete this motion. The difference between the end of extension and the start of retraction sets the *dwell angle* at fully extended position. The profile is implemented in the Simscape language file `ParabolicCam.ssc` using the Simscape Component block.

Each of the four parabolic cams are offset from each other by a phase angle. The corresponding injection pump plunger positions are thus also offset from each other. To achieve this, for a brief period of `initialization_time` after the start of simulation, an appropriate initialization shaft speed is sent to each parabolic cam to rotate it to the correct phase angle. After the `initialization_time`, the main portion of the fuel injection system simulation begins.

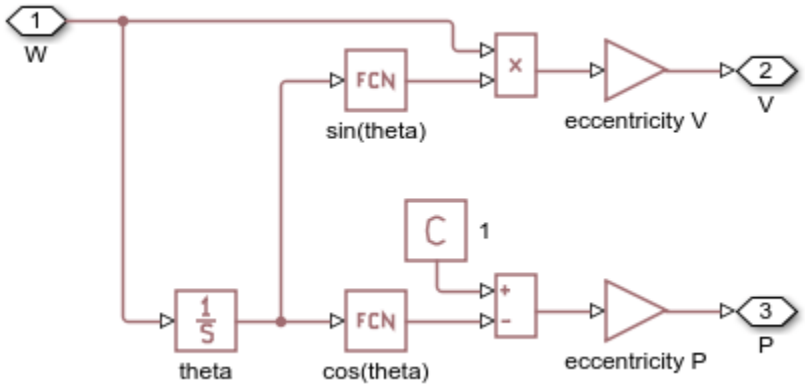
The following plot shows the extension of each injection pump as the cam shaft rotates. When the extension is zero, all of the fuel has been expelled from the pump. The firing order of the engine cylinders is 1-3-4-2.



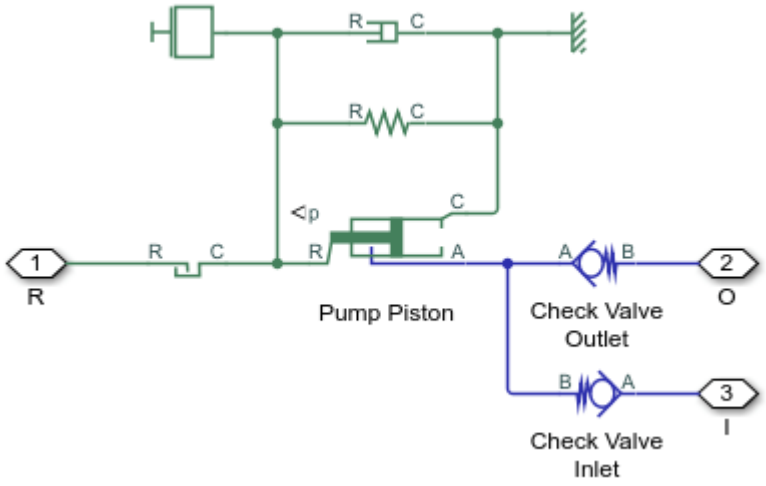
The profile of the eccentric cam is computed with the formula

$$x = \varepsilon\,(1 - \cos\theta)$$

where ε is the eccentricity. The equation is implemented using physical signal blocks in the Simscape Foundation Library.

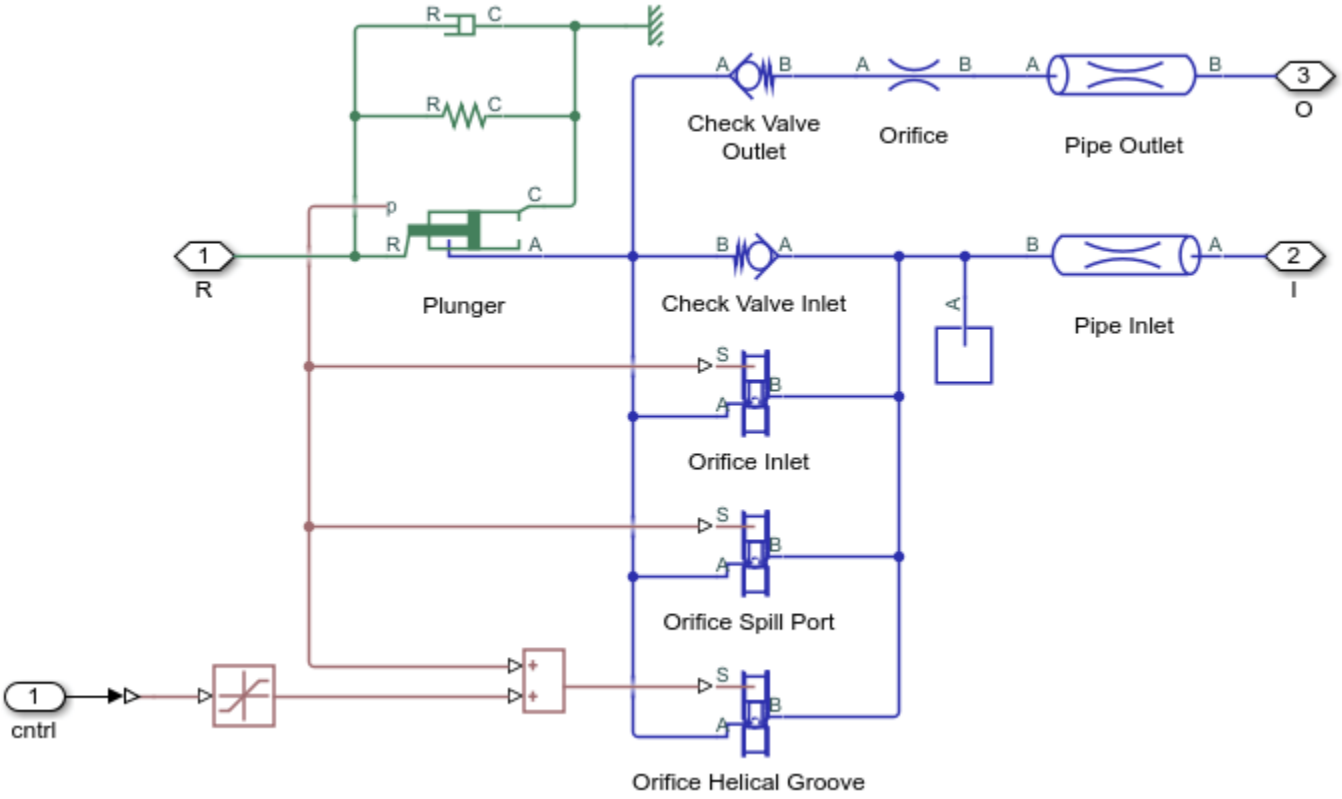


Lift Pump Subsystem



The model of the lift pump, which is a piston-and-diaphragm type pump, is built of a Single-Acting Actuator (IL) block and two Check Valve (IL) blocks. The check valves model inlet and outlet valves installed on both sides of the lift pump (see Figure 1). The contact between the pump rod roller and the cam is represented with the Translational Hard Stop block. The Translational Spring block models two springs in the pump that maintain permanent contact between the roller and the cam.

Injection Pump Subsystem



The in-line injection pump is a four-element pumping unit. Each element supplies fuel to injector which expells fuel to its engine cylinder. All four elements are identical by design. The Injection Pump subsystem represents the pump plunger and the pump control mechanism, while the Injector subsystem models the injector installed directly on the engine cylinder (see Figure 1).

The pump plunger oscillates inside the pump barrel driven by the cam (see Figure 1). The plunger is modeled as a Single-Acting Actuator (IL) block. When the plunger moves down, the plunger chamber is filled with fuel under pressure developed by the lift pump. The fuel enters through two orifices, the inlet port and the spill port (see Figure 2 below).

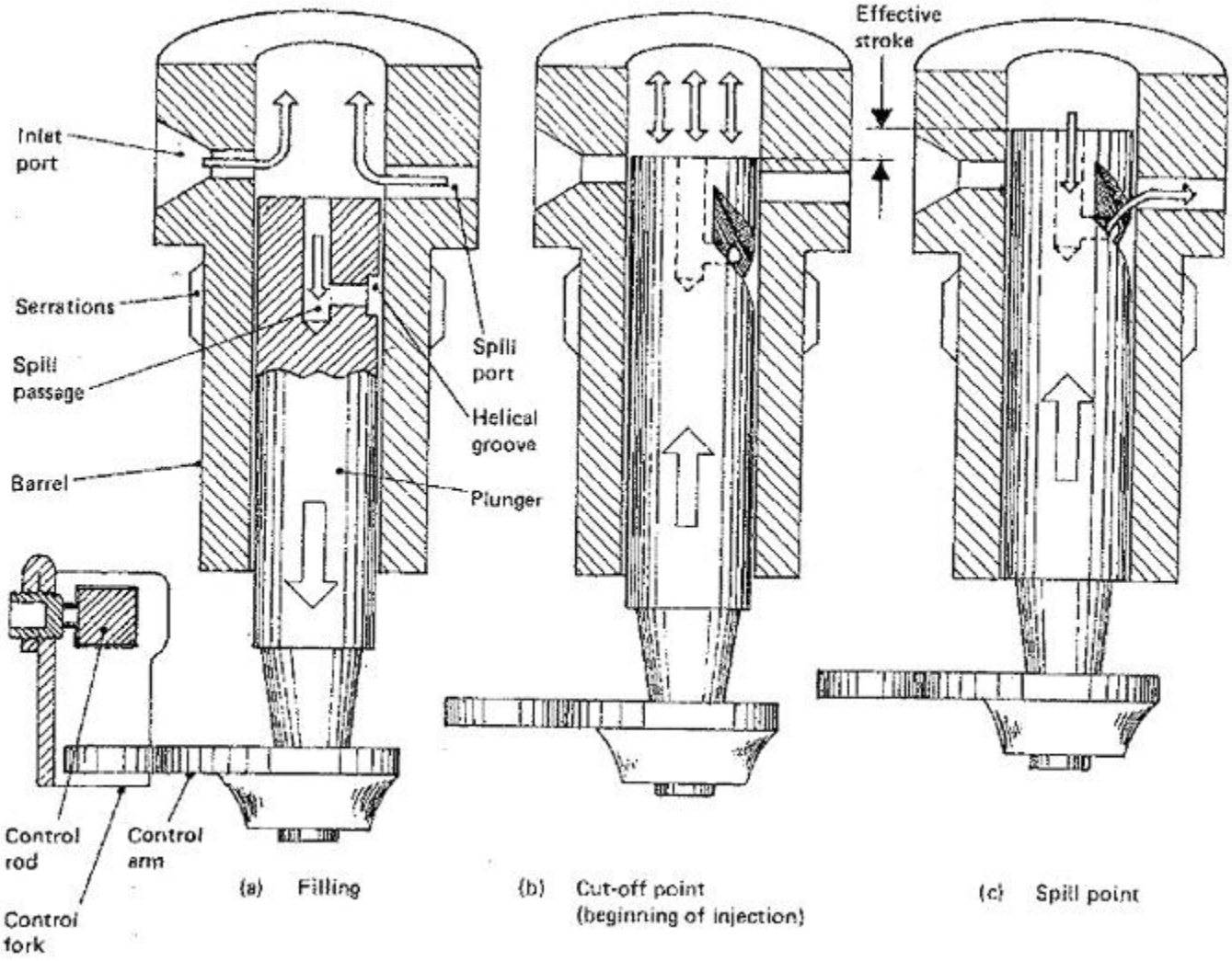


Figure 2: Plunger Interaction with Control Orifices in the Barrel

The model of the plunger control mechanism is based on the following assumptions:

1. There are three variable orifices modeled by Spool Orifice (IL) blocks in the control circuit: inlet port, spill port, and orifice formed by the helical groove and the spill port. The openings of the inlet and the spill orifices depend on the plunger motion, while the opening of the helical groove orifice is a function of the plunger motion and the plunger rotation. For the sake of simplicity, the displacement generated by the plunger rotation is represented as a source of linear motion which is combined with the plunger displacement.

2. The figure below shows all the dimensions necessary to parameterize the Spool Orifice (IL) blocks:

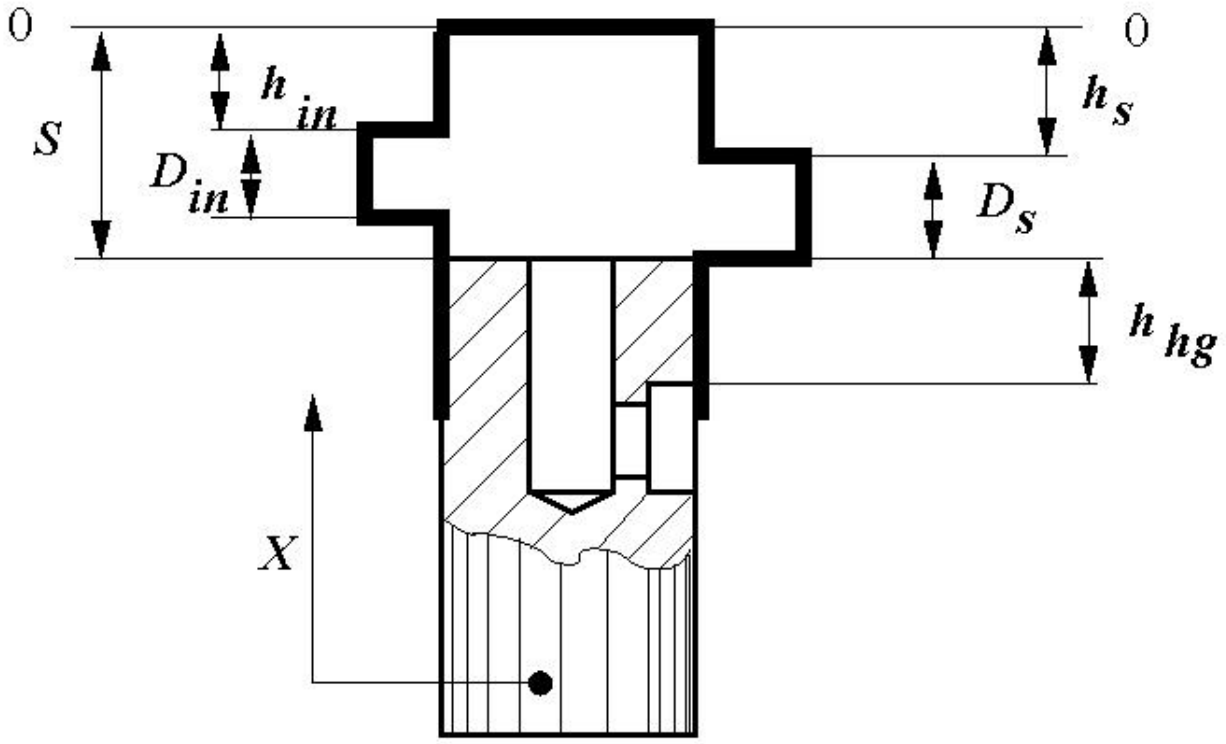


Figure 3: Schematic of injection plunger dimensions

where the schematic notations and corresponding workspace parameter names are

- D_{in} is the inlet port orifice diameter (`inlet_orifice_diameter`)
- D_s is the spill port orifice diameter (`spill_orifice_diameter`)
- S is the plunger stroke (`plunger_stroke`)
- h_{in} is the distance between the inlet orifice and the top plunger position (`-plunger_stroke + safety_gap + inlet_orifice_diameter + inlet_offset`)
- h_s is the distance between the spill port orifice and the top plunger position (`-plunger_stroke + safety_gap + spill_orifice_diameter`)
- h_{hg} is the distance between the spill port orifice and the upper edge of the helical groove

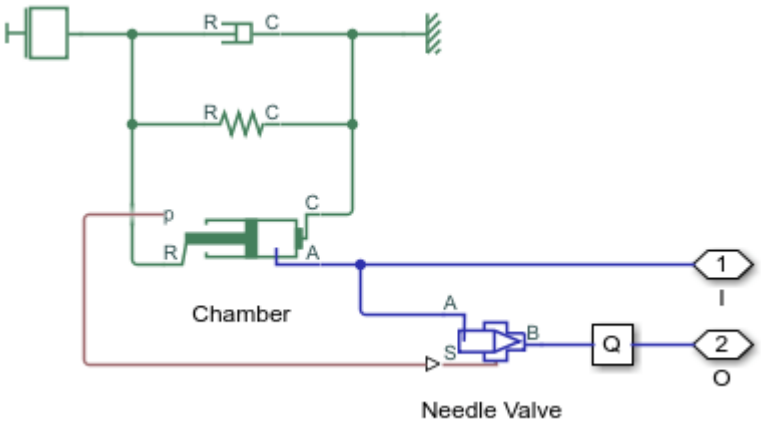
The inlet orifice is generally located higher than the spill orifice. This is represented by the workspace parameter `inlet_offset`.

3. In parameterizing the orifices, the plunger top position is taken as the origin and the motion in the upward direction is considered as the motion in positive direction. Therefore, the inlet and spill port orifices must be set to open with negative spool displacement, while the helical groove orifice must be set to open with positive spool displacement since it opens when the plunger moves upward.

4. The plunger effective stroke is `plunger_stroke - safety_gap - inlet_orifice_diameter - inlet_offset`. The distance between the spill port orifice and the upper edge of the helical

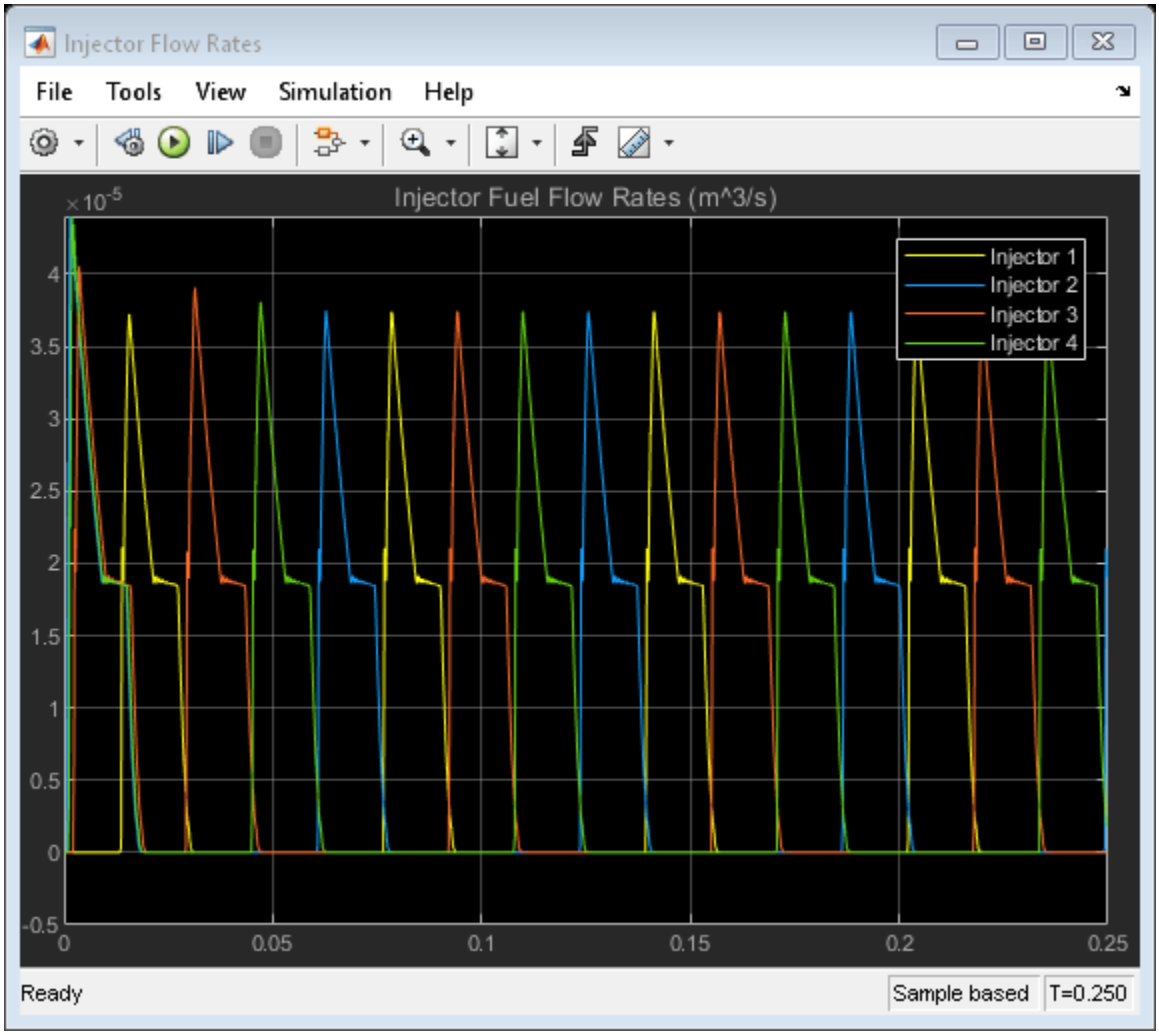
groove, h_{hg}, can be adjusted by rotating the plunger. This adjustment is modeled by shifting the plunger position signal to the helical groove orifice by a control signal. The larger the control signal, the sooner the helical groove orifice is opened, thus decreasing the volume of fuel delivered to the cylinder. The maximum value of the control signal is equal to the effective stroke. At this value, the helical groove orifice remains open all the time.

Injector Subsystem



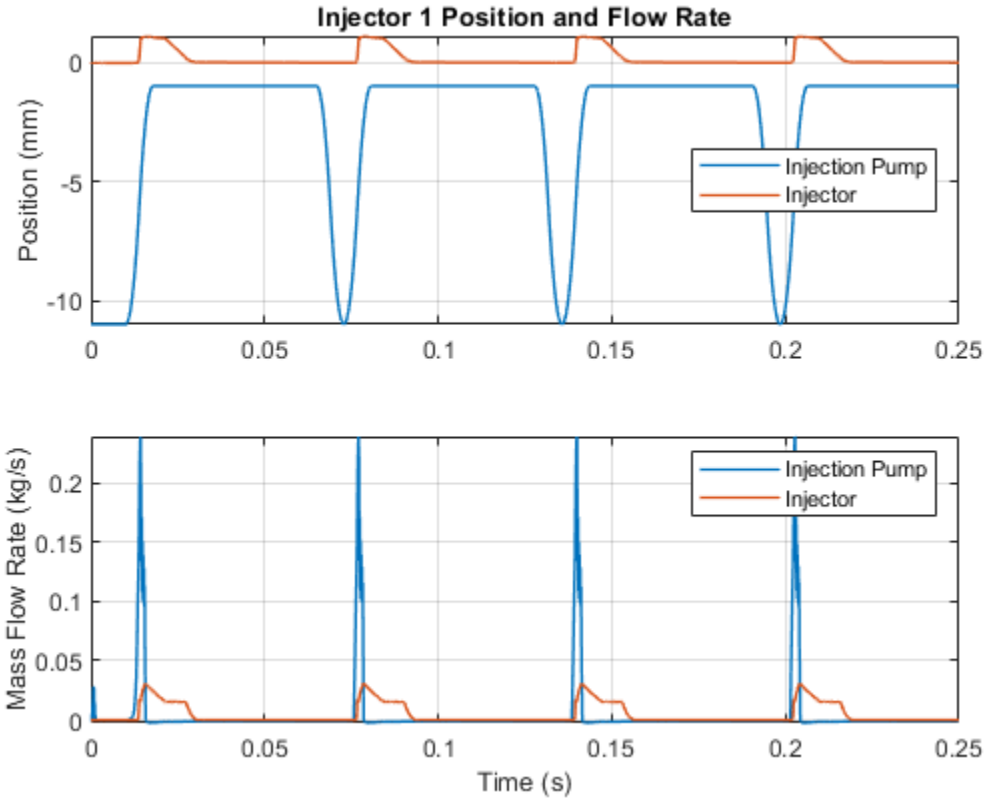
The model of the injector is based on the Single-Acting Actuator (IL) block and the Needle Valve (IL) block. The needle valve is normally closed by the force developed by the preloaded spring. As the force developed in the actuator chamber overcomes the spring force, the needle valve opens and allows fuel to be injected into the engine cylinder.

Simulation Results from Scopes



Simulation Results from Simscape Logging

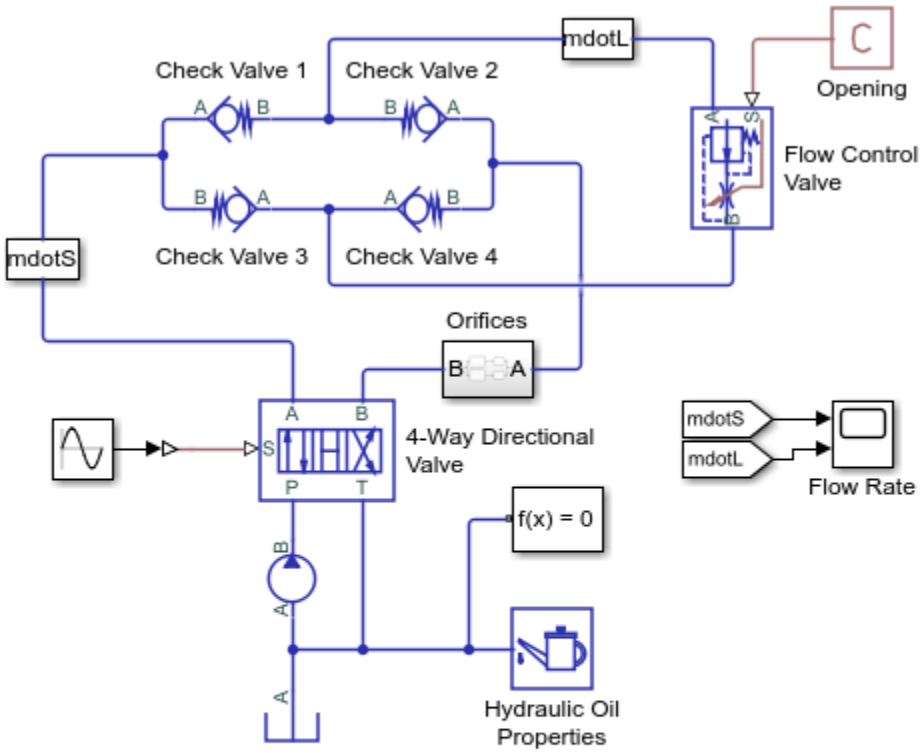
This plot shows the positions and outlet flow rates of Injection Pump 1 and Injector 1. The effect of the cam profile is shown in the displacement of Injection Pump 1. During the second half of the cam stroke, fuel exits the injection pump and passes into the injector. The fuel exits the injector via a needle valve. The injector is modeled as an actuator chamber with a preloaded spring which acts to store the fuel from the injection pump temporarily and push it out of the injector more smoothly.


Injector 1 Position and Flow Rate
Position (mm)
0
-5
-10
0
0.05
0.1
0.15
0.2
0.25
Injection Pump
Injector
Mass Flow Rate (kg/s)
0.2
0.15
0.1
0.05
0
Time (s)

Hydraulic Flow Rectifier Circuit

This example shows a flow rectifier circuit with four check valves and a flow control valve. It is used to allow a single flow control valve to control fluid flow in both directions. Similar to a Graetz circuit implemented with diodes, the check valves are arranged in such a way that flow always passes through the flow control valve in the same direction. In the Orifices subsystem, there are two more check valves that are used to select the orifice that the flow passes through depending on the flow direction.

Model

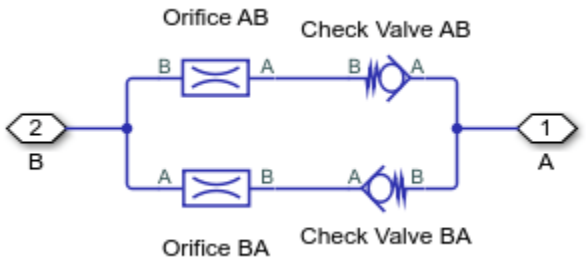


Hydraulic Flow Rectifier Circuit

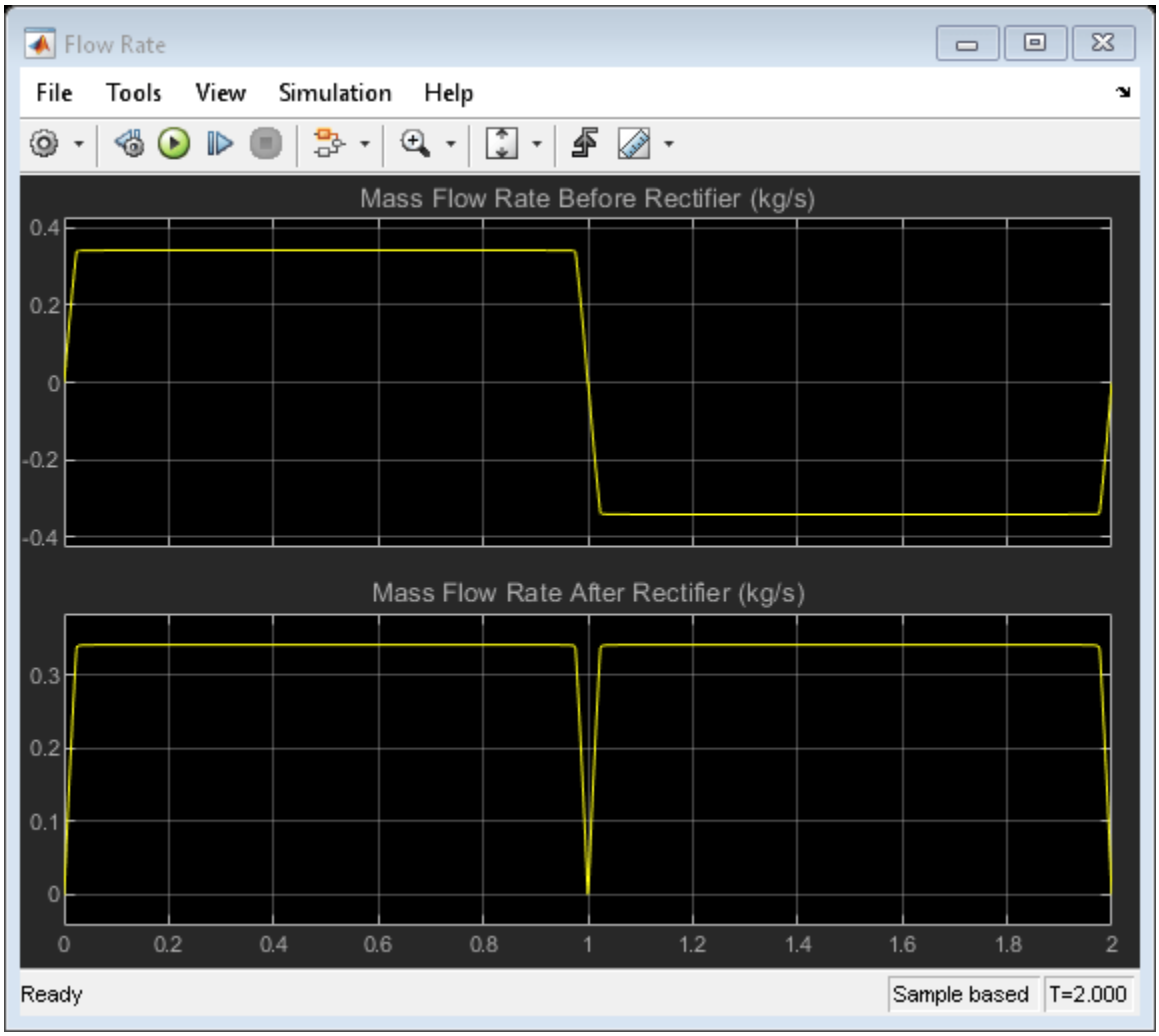
1. Plot flow rates in system (see code)
2. Open electrical model of a bridge rectifier (see example documentation)
3. Explore simulation results using sscexplore
4. Learn more about this example

Copyright 2019-2021 The MathWorks, Inc.

Orifices Subsystem



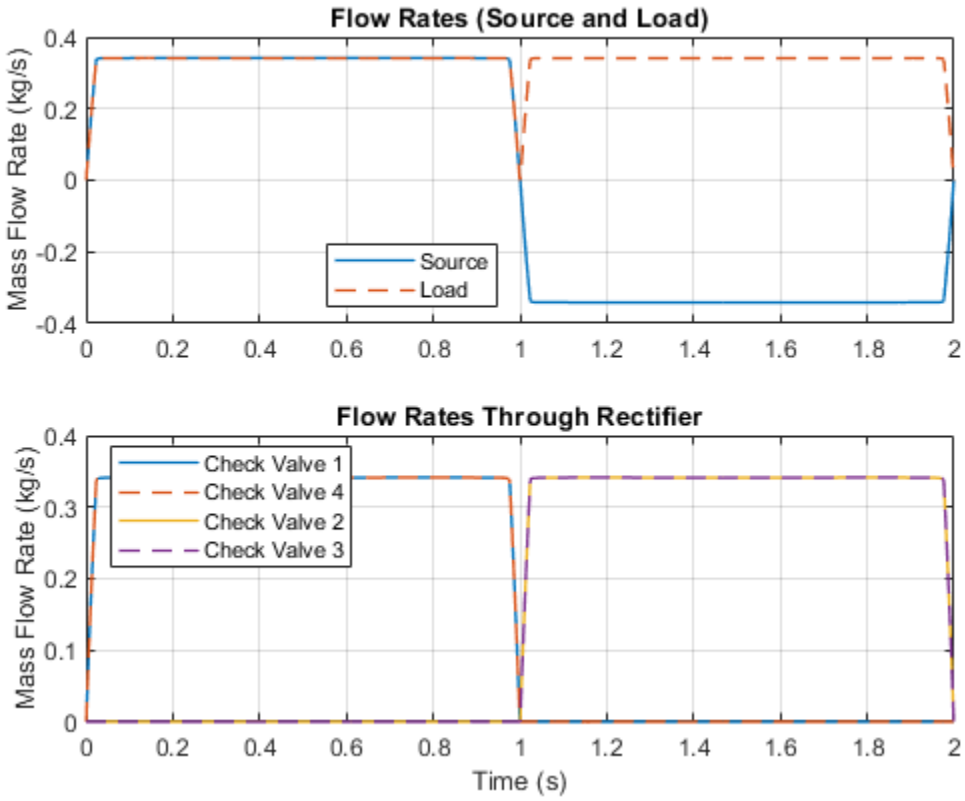
Simulation Results from Scopes



Simulation Results from Simscape Logging

This plot shows the mass flow rates through various components in the system. The flow rate from the source changes direction due to the 4-way directional valve. However, the flow rate to the load remains in the same direction due to the rectifier. When the flow rate from the source is positive, flow

passes through check valves 1 and 4 and is blocked by check valves 2 and 3. When the flow rate from the source is negative, flow passes through check valves 2 and 3 and is blocked by check valves 1 and 4.



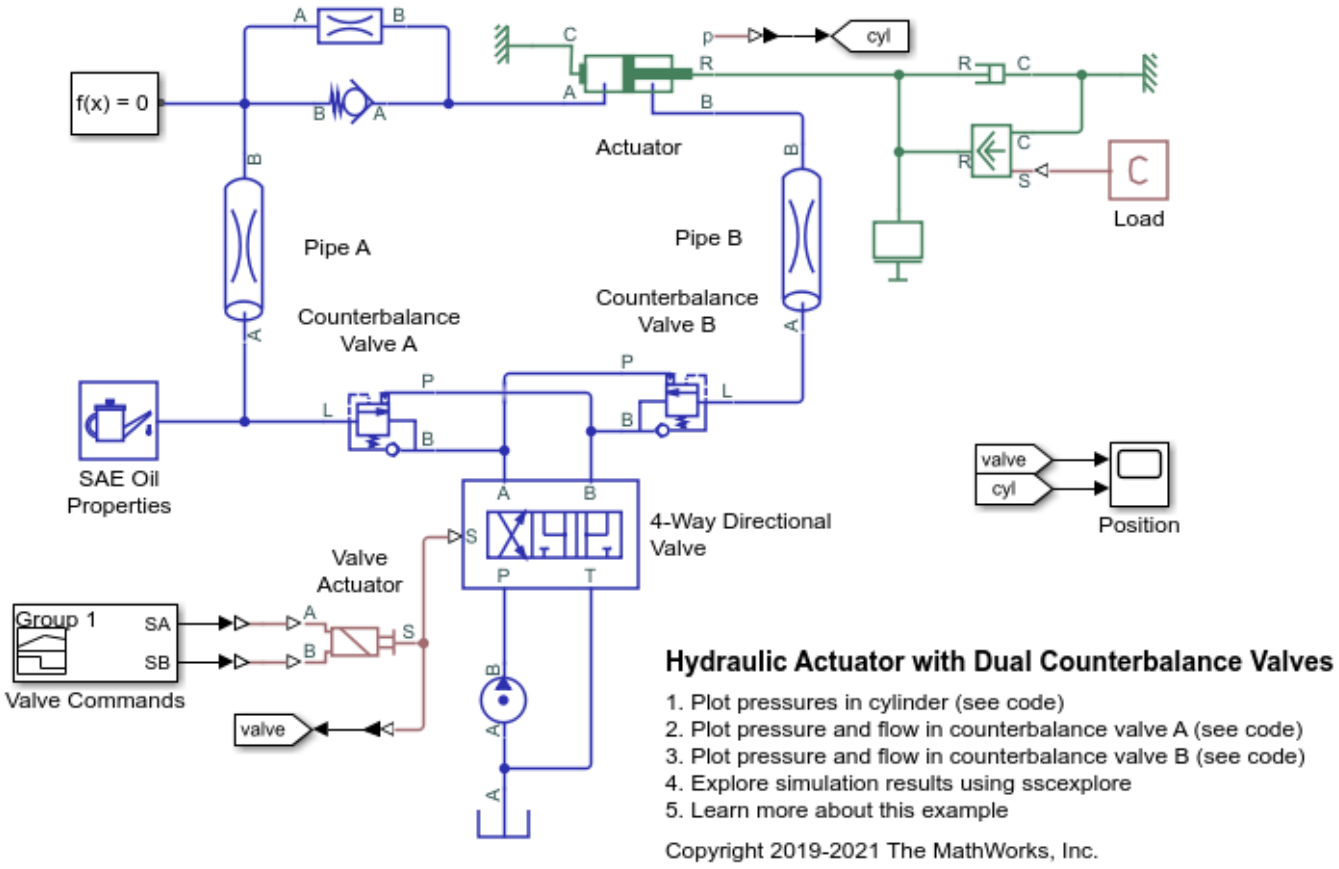
Hydraulic Actuator with Dual Counterbalance Valves

This example shows an actuator controlled by a 4-way directional valve and loaded with an overriding load, requiring the use of counterbalance valves to prevent the load from creeping when the directional valve is in the neutral position. In the neutral position, the directional valve connects ports A and B to the reservoir while blocking the pressure port P. The counterbalance valves block flow from returning to the reservoir, thus holding the actuator in place.

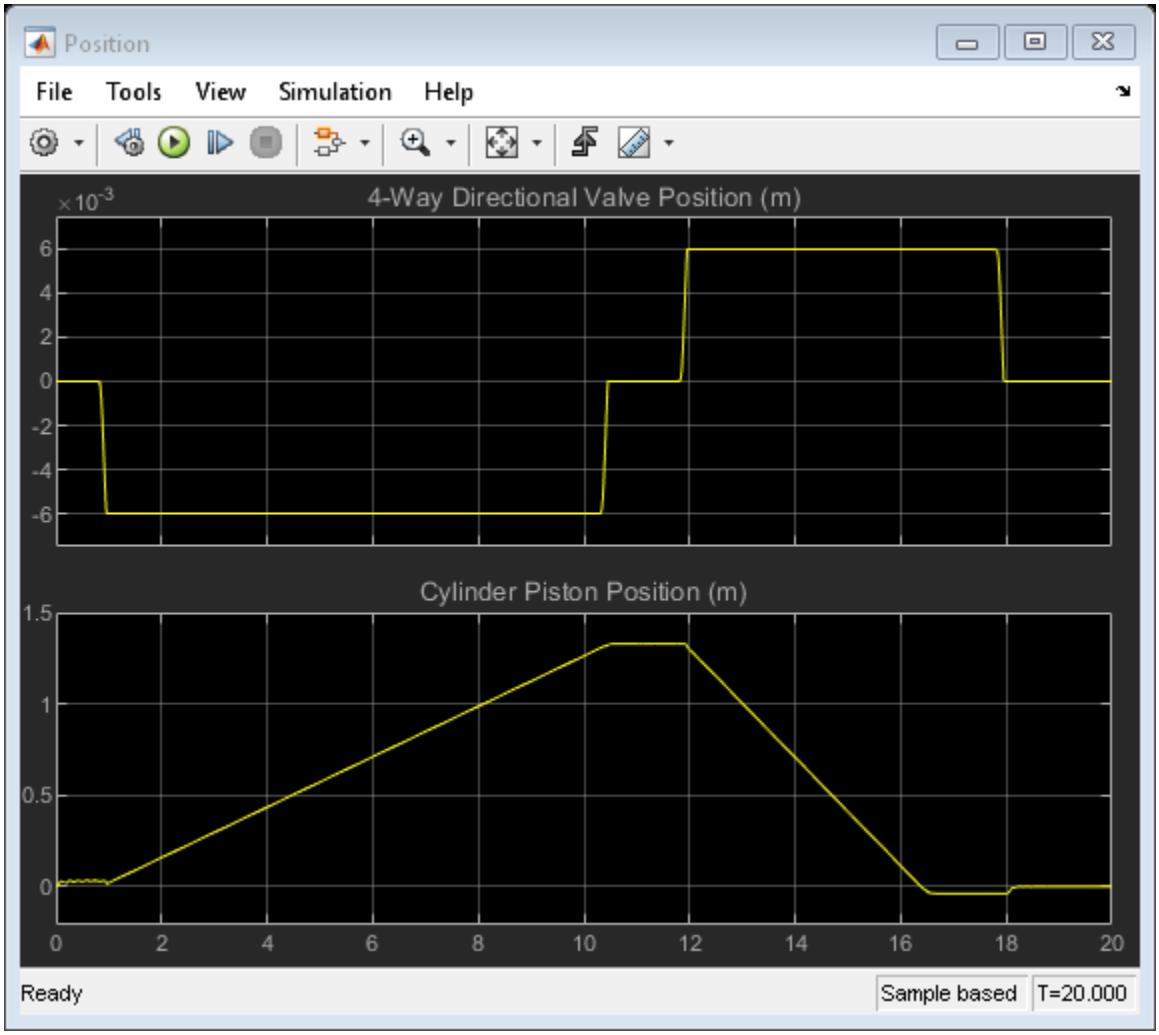
The valve actuator represents the electromagnets that control the directional valve position. When electromagnet B is energized, the directional valve is in the negative position and connects ports A and B to the pressure port P. This causes the actuator to extend due to the difference in the piston areas. Flow passes through the check valve in counterbalance valve A to enter cylinder chamber A, while pressure in the pilot line opens the orifice in counterbalance valve B to permit return flow from cylinder chamber B.

When electromagnet A is energized, the directional valve is in the positive position and connects port B to the pressure port P and port A to the reservoir. This causes the actuator to retract. Flow passes through the check valve in counterbalance valve B to enter cylinder chamber B, while pressure in the pilot line opens the orifice in counterbalance valve A to permit return flow from cylinder chamber A.

Model

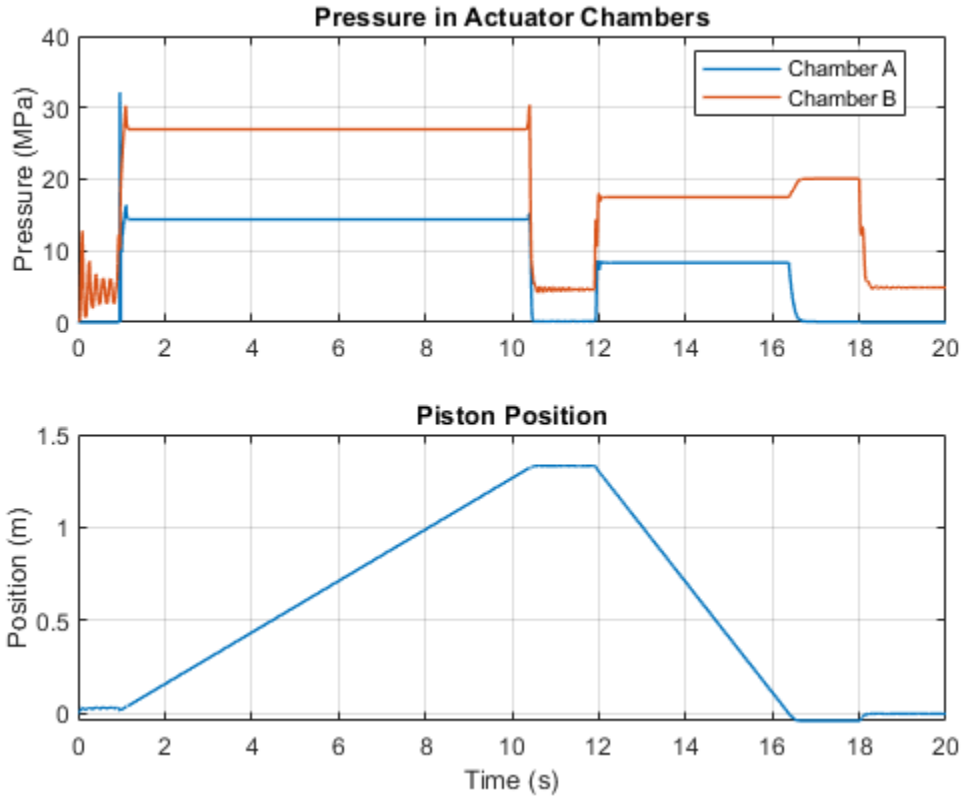


Simulation Results from Scopes

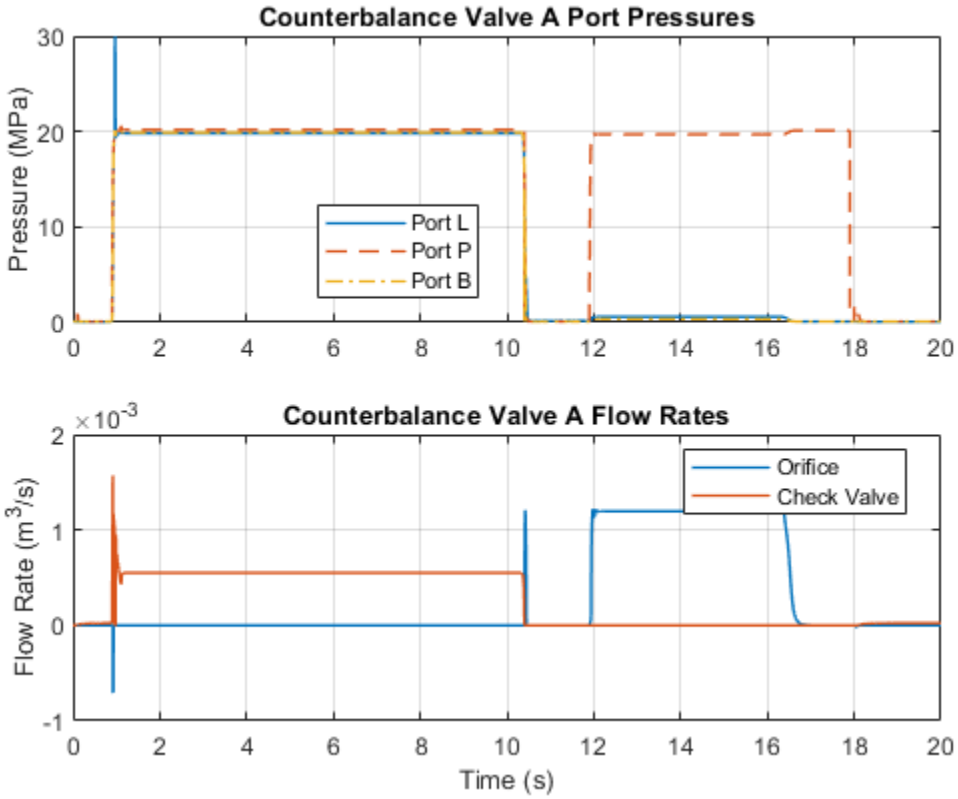


Simulation Results from Simscape Logging

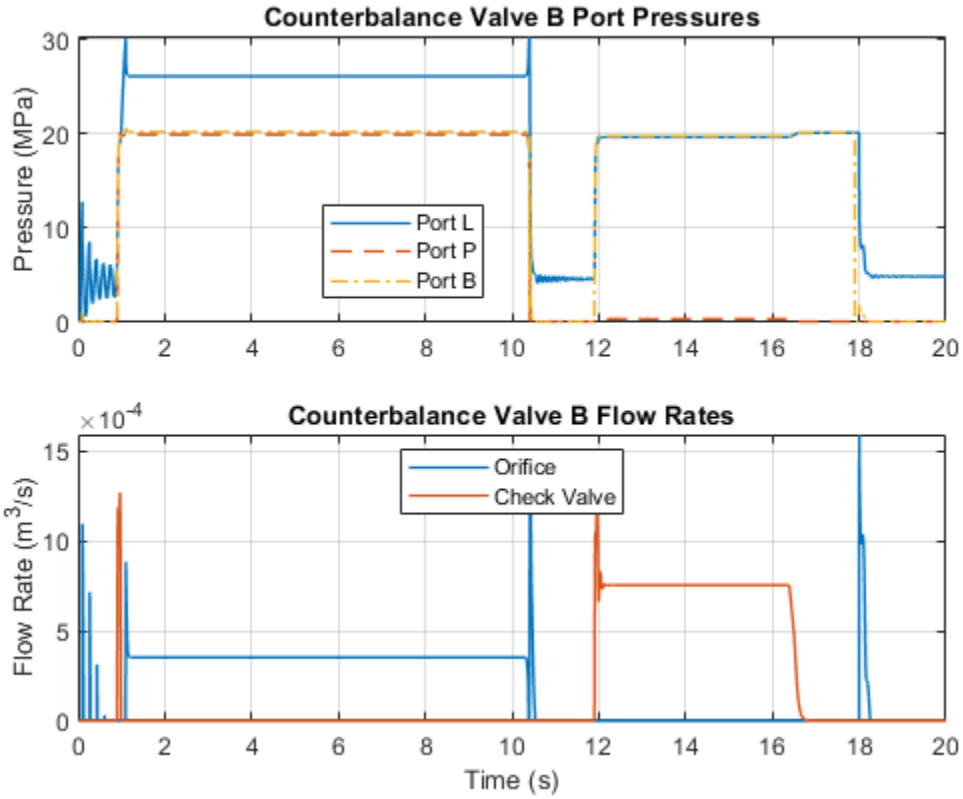
This plot shows the pressures in the two cylinder chambers and the position of the piston.



This plot shows the port pressures and flow rates for counterbalance valve A. In the first part of the simulation, flow passes through the check valve inside the counterbalance valve, allowing liquid to enter the cylinder chamber A to extend the piston. In the second part of the simulation, flow returning from the cylinder chamber A is controlled by the orifice inside the counterbalance valve. Pressures at port P and L must be sufficiently large to open the orifice and permit return flow.



This plot shows the port pressures and flow rates for counterbalance valve A. In the first part of the simulation, flow returning from the cylinder chamber B is controlled by the orifice inside the counterbalance valve. Pressures at port P and L must be sufficiently large to open the orifice and permit return flow. In the second part of the simulation, flow passes through the check valve inside the counterbalance valve, allowing liquid to enter the cylinder chamber B to retract the piston.


Counterbalance Valve B Port Pressures
Pressure (MPa)
Port L
Port P
Port B
Counterbalance Valve B Flow Rates
×10^-4
Flow Rate (m^3/s)
Orifice
Check Valve
Time (s)

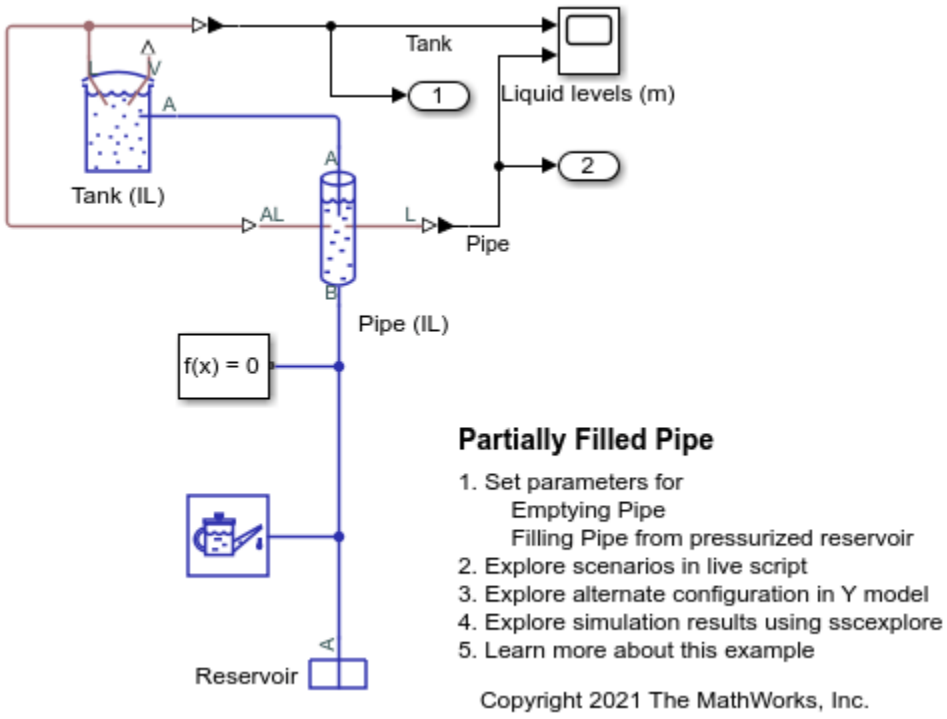
Partially Filled Pipe

This example shows the Partially Filled Pipe (IL) block used to model the emptying and filling of a tank in multiple configurations. You can use the live script provided with this example to understand the effect of different configurations.

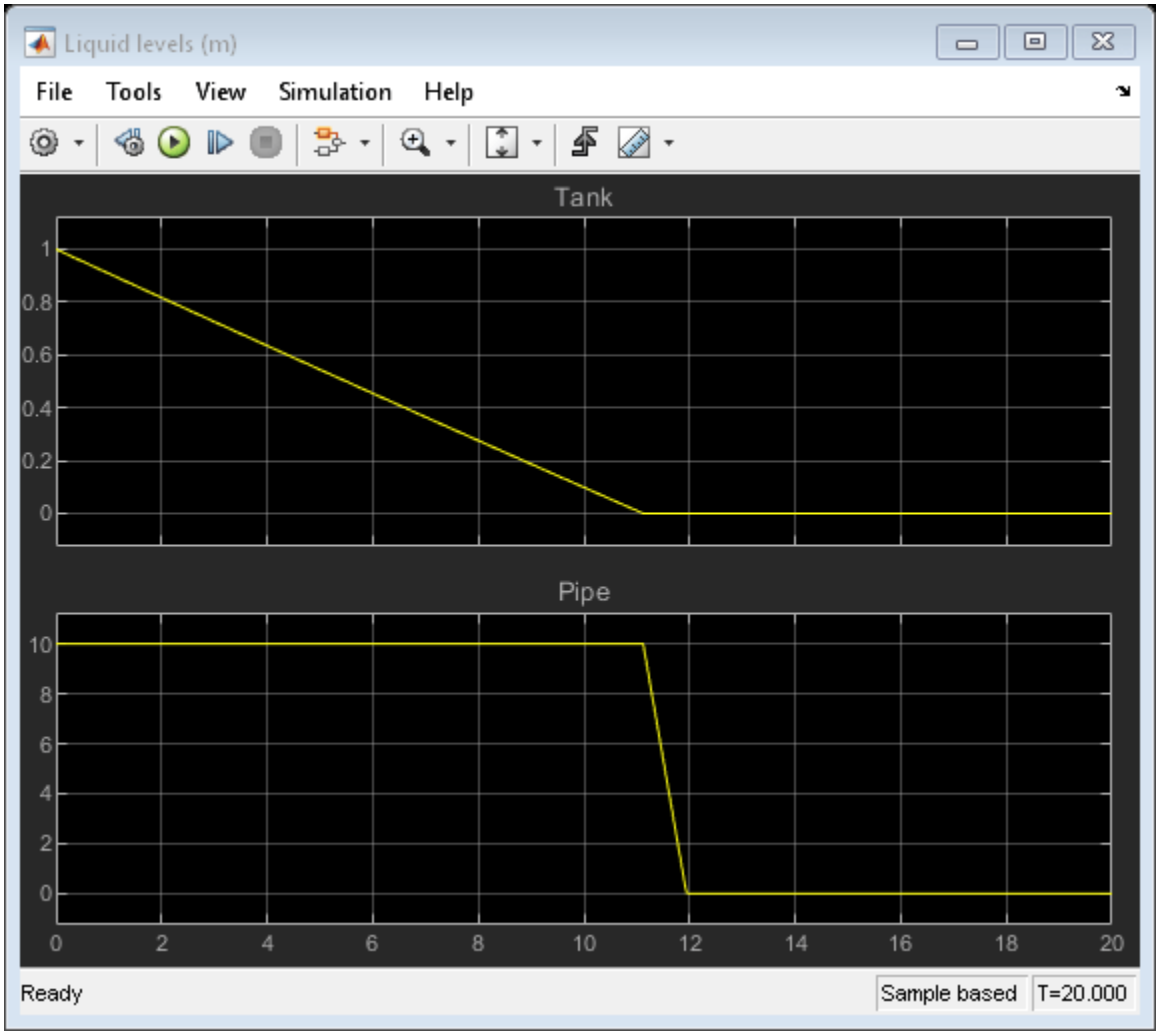
The block represents an arbitrarily full pipe in an isothermal liquid network. The block requires that Port A is higher than Port B. The physical signal input AL indicates the relative liquid level of the block that is connected to port A. When AL <= 0, the block assumes there is no liquid is entering port A.

Basic Model

This model demonstrates the basic function of the partially filled pipe using one Partially Filled Pipe (IL) block.

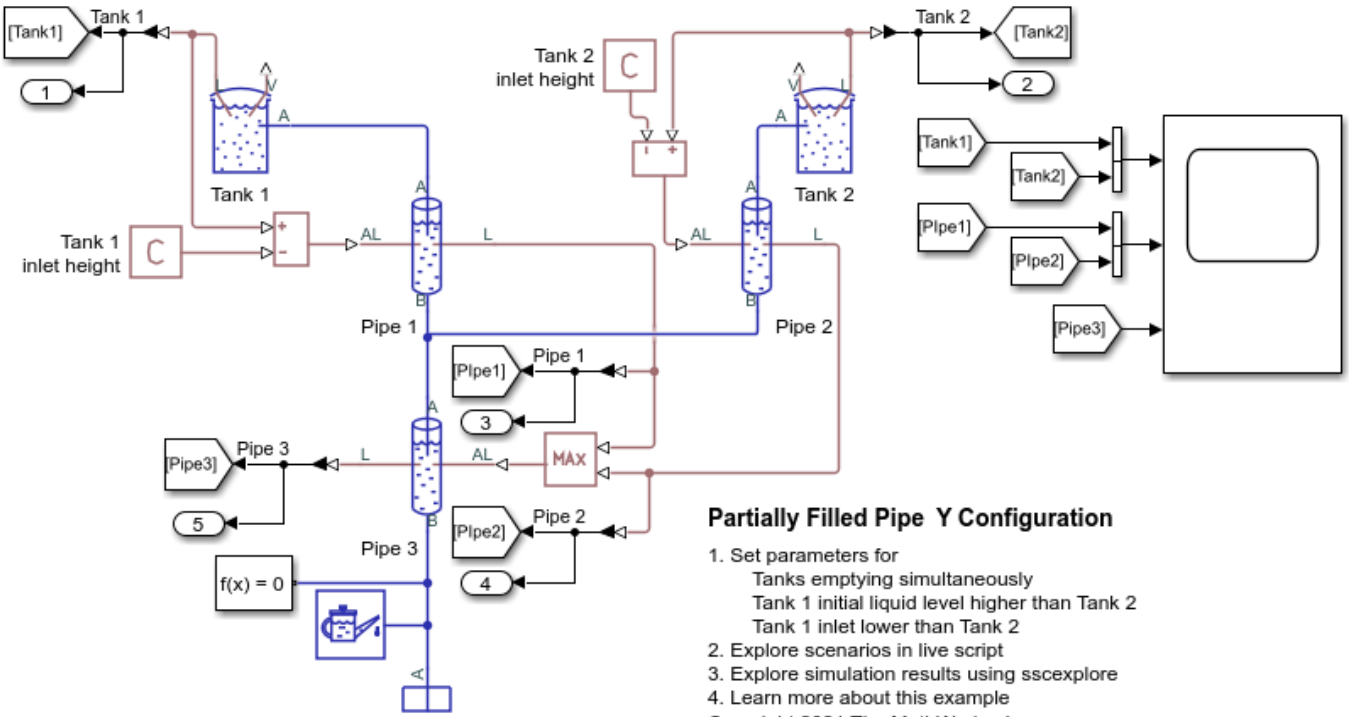


Simulation Results from Scopes

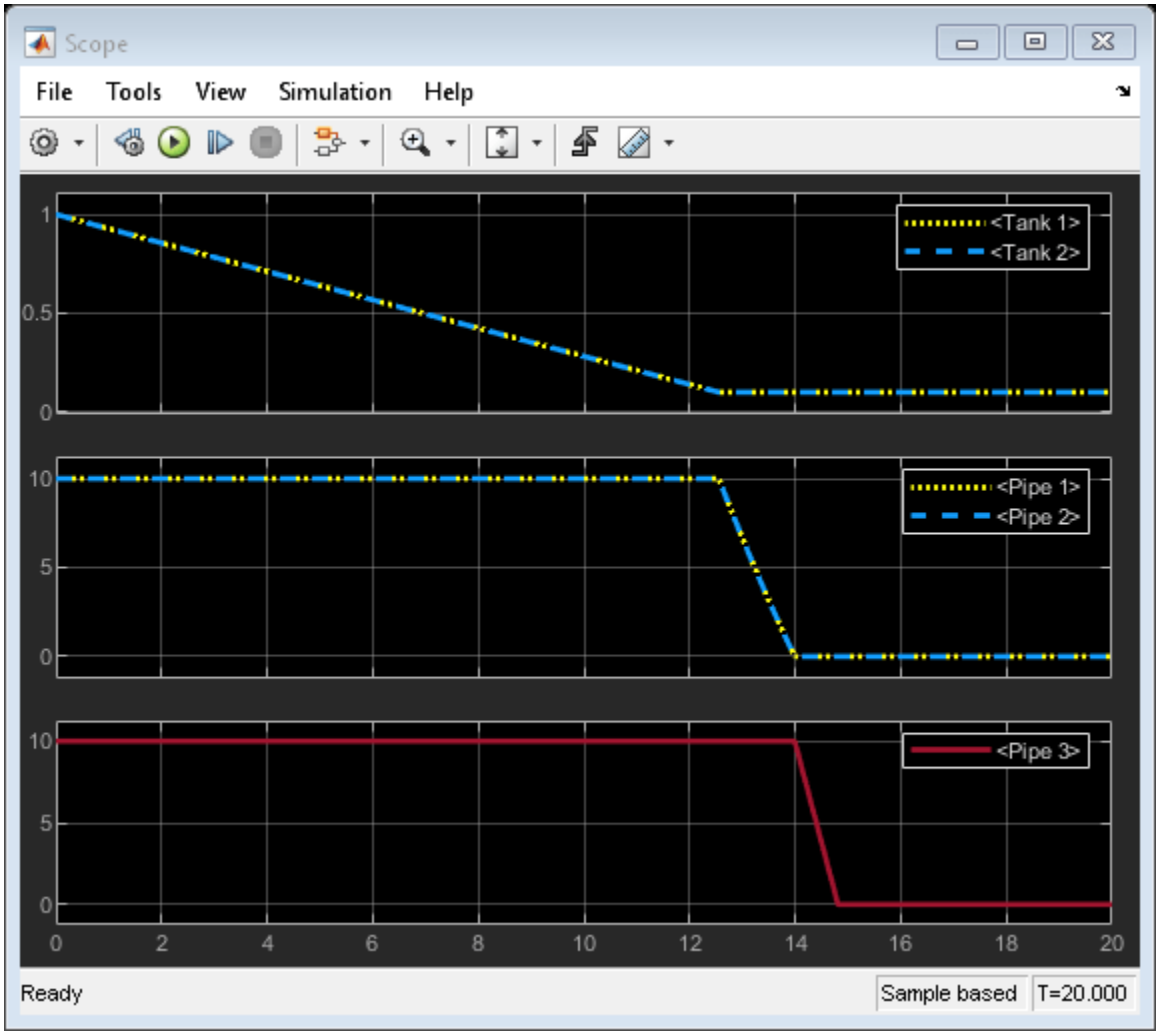


Y configuration Model

This model demonstrates a more complex configuration which uses 3 Partially Filled Pipe (IL) blocks arranged in a Y configuration.


[Tank1]
Tank 1
1
Tank 1
Tank 1
inlet height
C
AL
L
Pipe 1
Tank 2
inlet height
C
Tank 2
Tank 2
[Tank2]
2
[Tank1]
[Tank2]
[Pipe1]
[Pipe2]
[Pipe3]
Pipe 2
[Pipe1]
Pipe 1
3
[Pipe3]
Pipe 3
5
MAX
[Pipe2]
Pipe 2
4
Pipe 3
f(x) = 0
Partially Filled Pipe Y Configuration
1. Set parameters for
Tanks emptying simultaneously
Tank 1 initial liquid level higher than Tank 2
Tank 1 inlet lower than Tank 2
2. Explore scenarios in live script
3. Explore simulation results using sscexplore
4. Learn more about this example
Copyright 2021 The MathWorks, Inc.

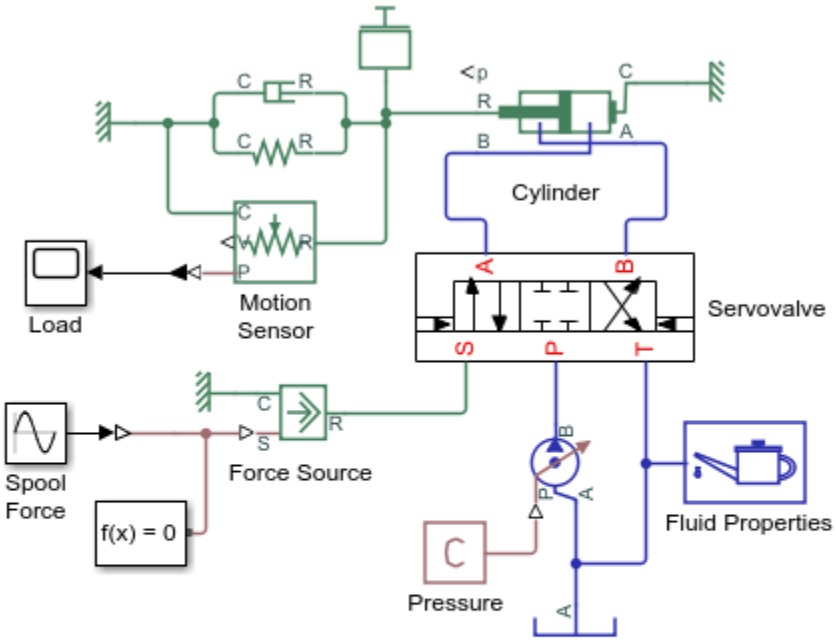
Simulation Results from Scopes



Servo-Valve Controlling Hydraulic Actuator

This example shows a model of a two-stage servo-valve with a 4-way, 3-position spool valve in the power stage and a flapper-nozzle amplifier in the pilot stage. The flapper is connected to the armature of a torque motor, which in the example is represented with an ideal translational force source. The servo-valve shown in the example is equipped with the spring feedback between the flapper and the spool of the main valve. To investigate the behavior of such a valve, axial hydraulic forces on all four spool orifices are accounted for in the model by using Spool Orifice Hydraulic Force blocks. The servo-valve controls a simple double-acting cylinder in an open-loop application.

Model

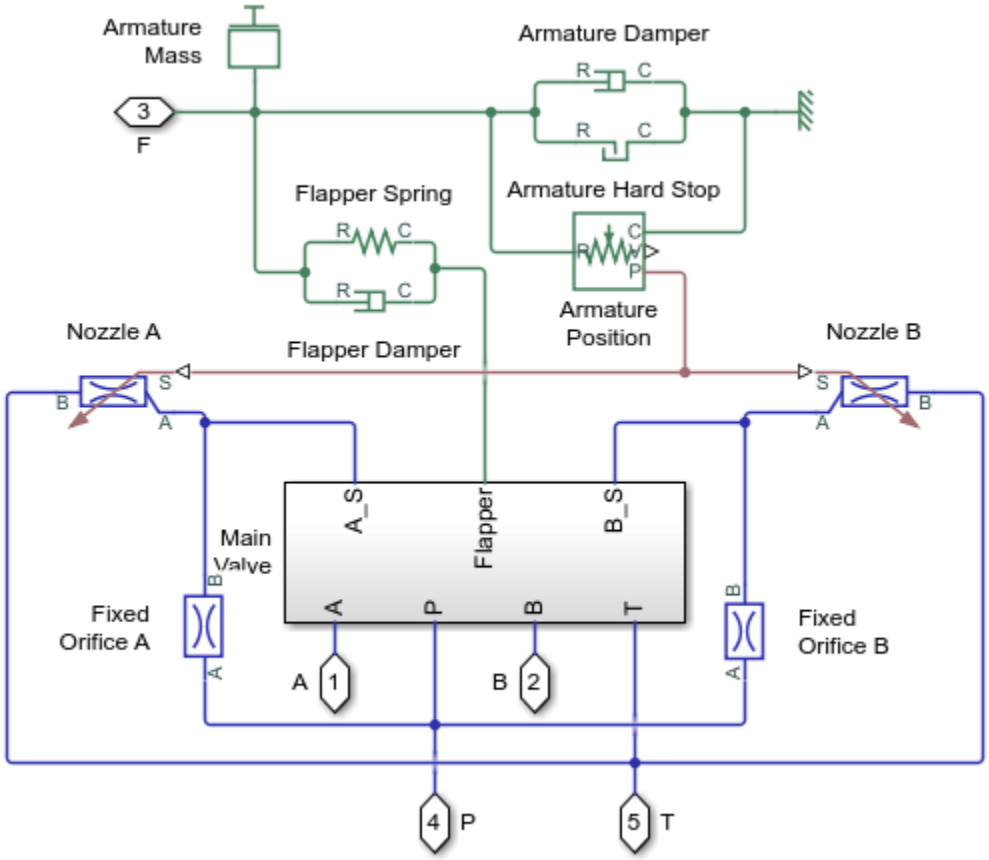


Servo-Valve Controlling Hydraulic Actuator

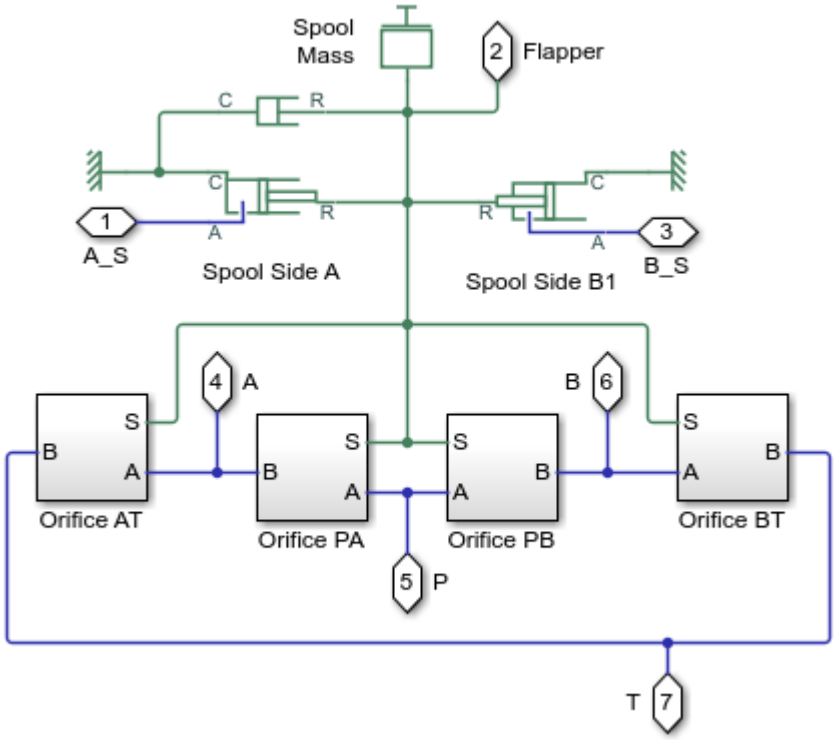
1. Plot pressures in cylinder (see code)
2. Plot flow rates in valve (see code)
3. Explore simulation results using sscexplore
4. Learn more about this example

Copyright 2005-2021 The MathWorks, Inc.

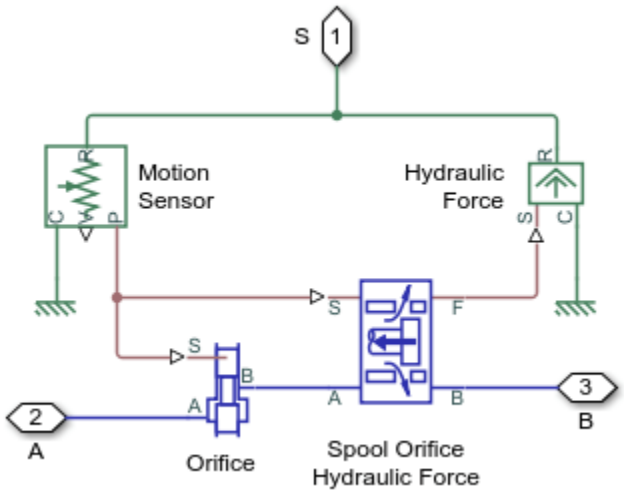
Servovalve Subsystem



Main Valve Subsystem

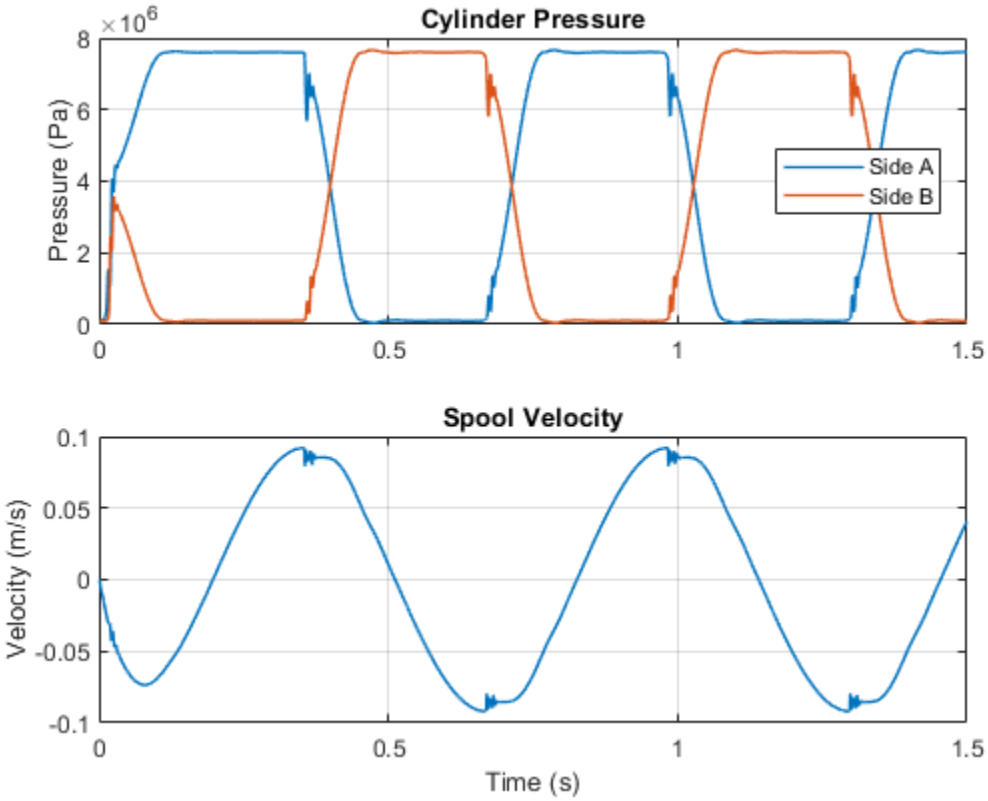


Orifice AT Subsystem

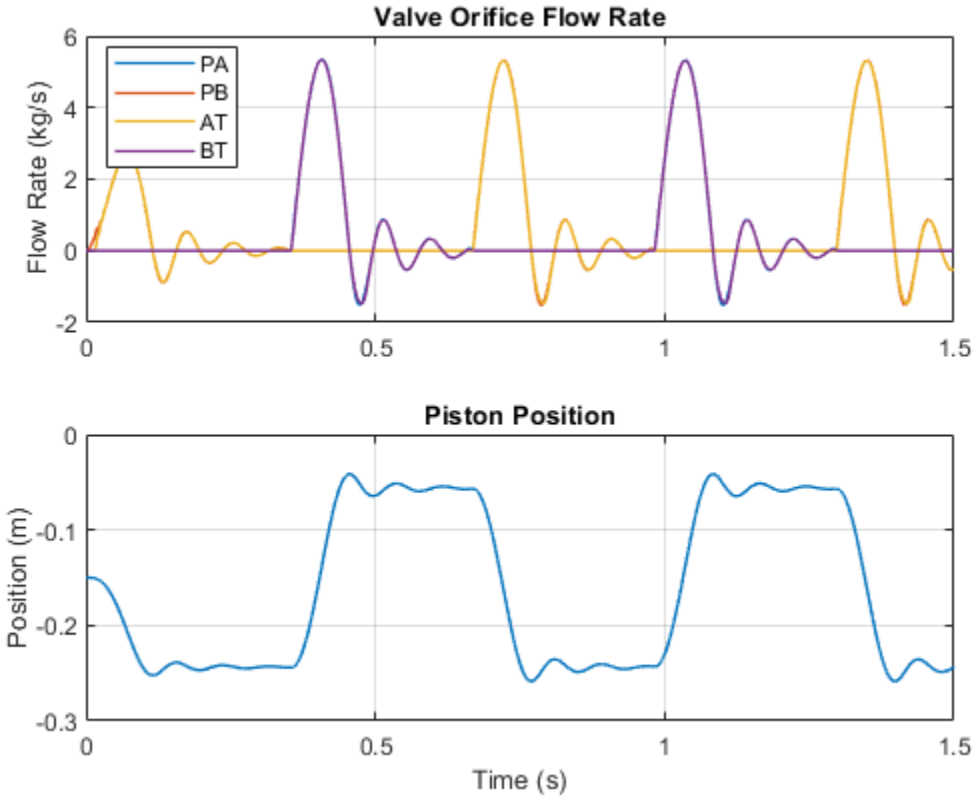


Simulation Results from Simscape Logging

The plots below show the pressures in the cylinder and the velocity of the spool in the pilot stage. The effect of the pilot stage spool oscillations are visible in the cylinder pressures.



The plots below show the flow rates in the valve and the position of the piston.

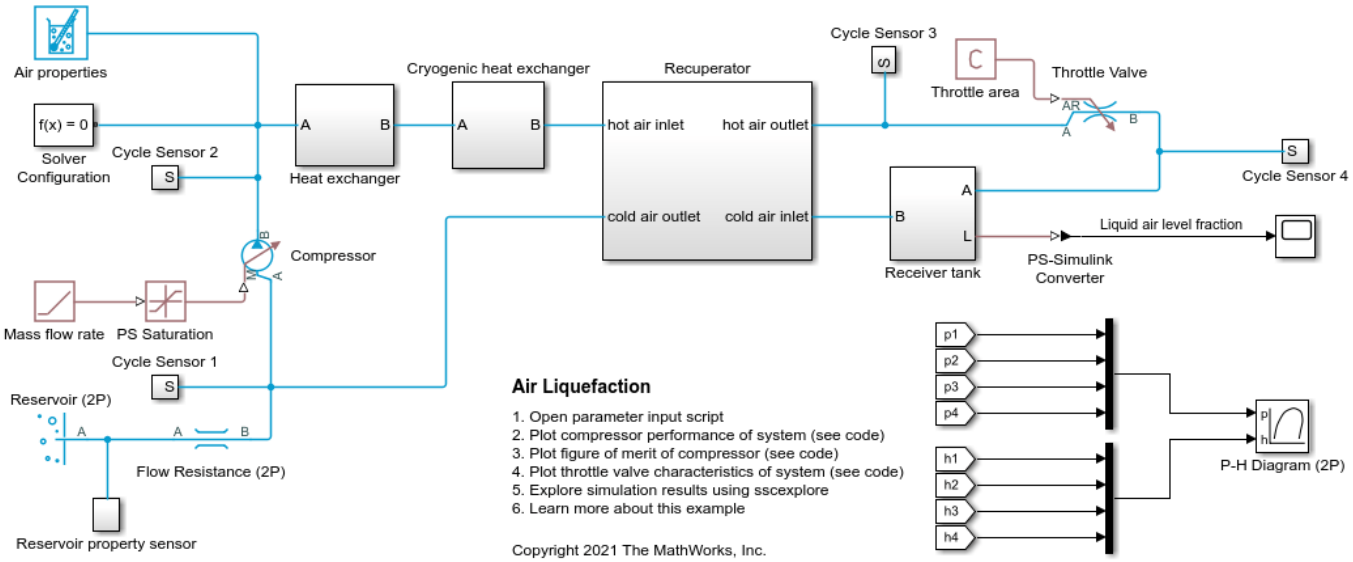

Valve Orifice Flow Rate
Flow Rate (kg/s)
PA
PB
AT
BT
6
4
2
0
-2
0
0.5
1
1.5
Piston Position
Position (m)
0
-0.1
-0.2
-0.3
0
0.5
1
1.5
Time (s)

Air Liquefaction

This example models a Linde-Hampson air liquefaction cycle using the two-phase fluid blocks. The compressor drives the vapor air through two heat exchangers, recuperator, expansion valve, and storage tank. The first heat exchanger contains water at atmospheric temperature, while the second heat exchanger contains refrigerent. The pressure drops as the cooled air passes through the expansion valve, which results in condensation. The air and entrained condensate then pass through a receiver tank where the condensate separates from the air. The liquid air is stored in a recevier tank while the vapor air is circulated back to the compressor.

Model

The figure shows the model of a air liquefaction cycle with its subcomponents.

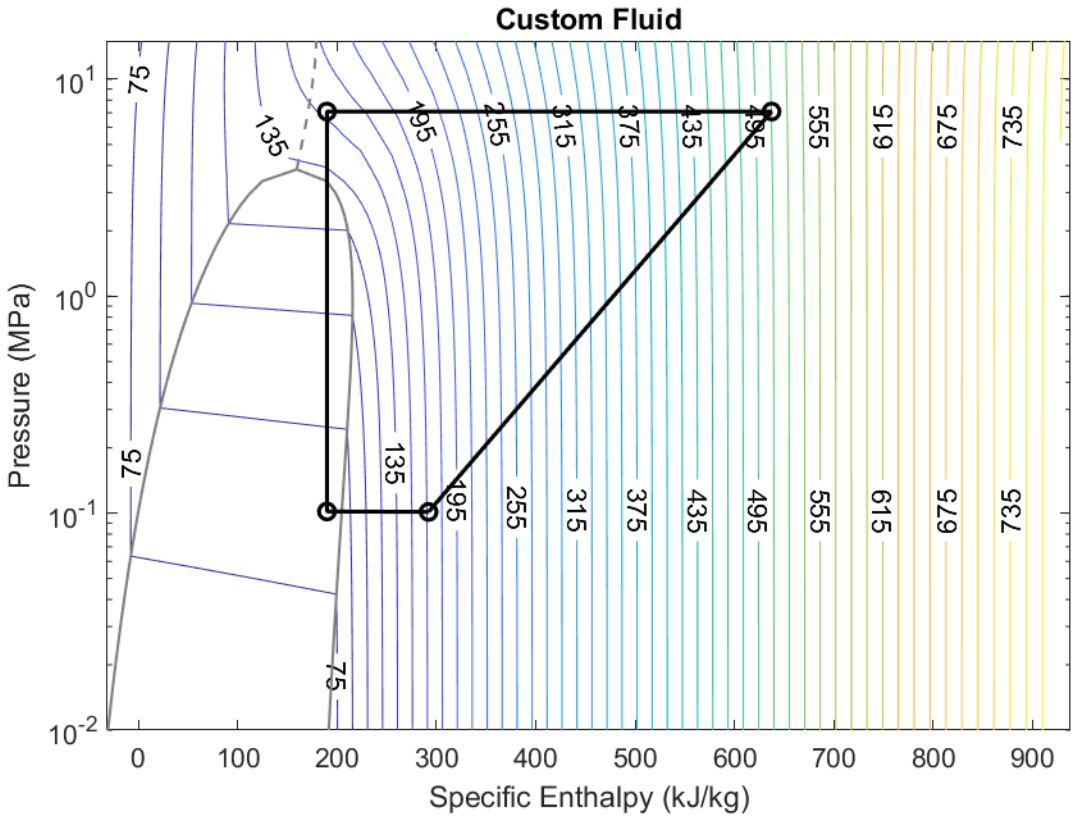


Fluid Properties

This example uses air as the working fluid, which is parameterized as a pseudo-pure fluid.

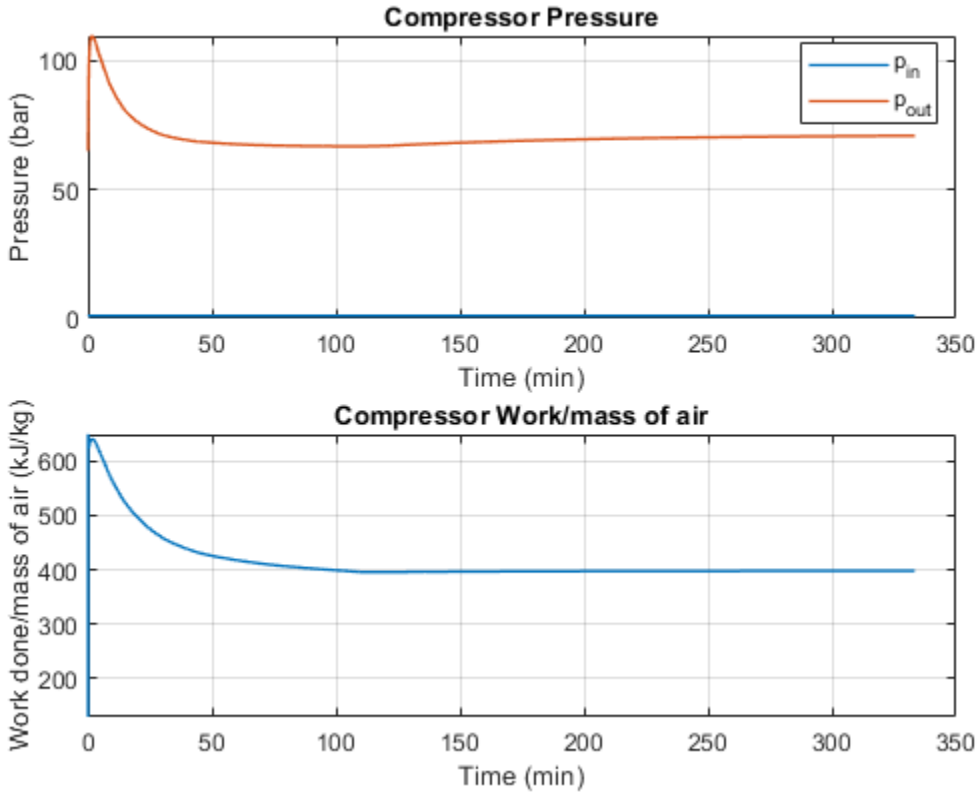
p-h Diagram of the Linde-Hampson Cycle

This figure shows the state of the fluid at the four points on the air liquefaction p-h diagram: at the compressor inlet, compressor outlet, expansion valve inlet, and expansion valve outlet. The black contour is the saturation dome of the air. Note that the lines joining the four states in the p-h diagram do not represent the fluid process.

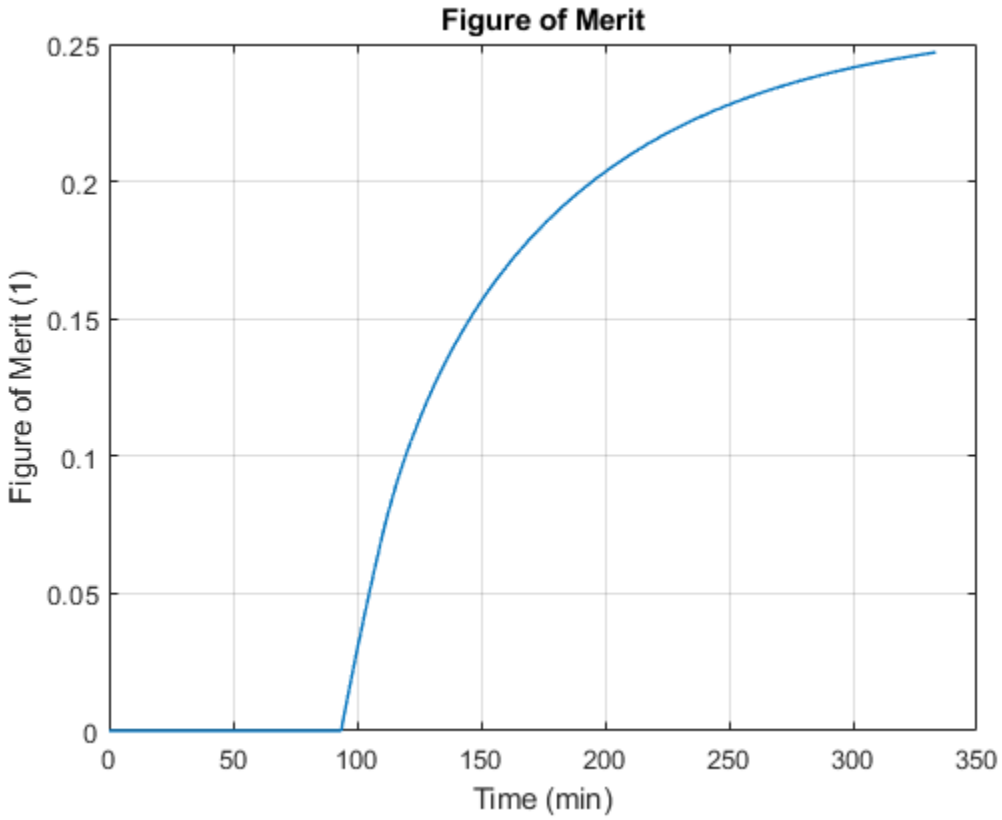


Simulation Results from Simscape™ Logging

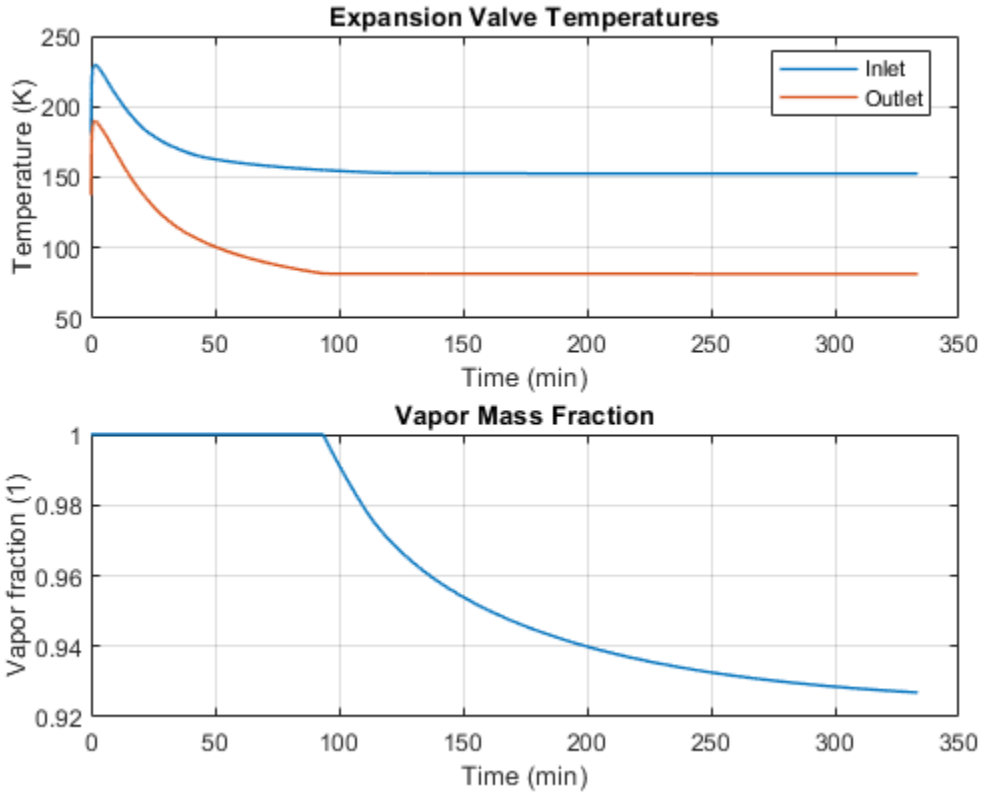
This figure shows the compressor parameters over time. The output pressure of the compressor increases to ~110 bar initially and settles around ~62 bar. The pressure output of the compressor can be varied by changing the area of the throttle valve.



This figure shows the figure of merit of the compressor over time. Figure of merit is the ratio of the ideal work done per unit mass of the air liquified to the actual work done per unit mass of the air liquified. It is a measurement of the efficiency of the compressor.



This figure shows the input and output temperatures for the throttle valve and vapor fraction of the fluid at the outlet of the throttle valve. Note that the temperature drop across the throttle valve is ~70K, and the vapor fraction is ~0.87. The liquid fraction of the air gets accumulated in the receiver tank while the gas fraction of the air goes to the inlet of the compressor.


Expansion Valve Temperatures
Temperature (K)
250
200
150
100
50
0
50
100
150
200
250
300
350
Time (min)
Inlet
Outlet
Vapor Mass Fraction
Vapor fraction (1)
1
0.98
0.96
0.94
0.92
0
50
100
150
200
250
300
350
Time (min)

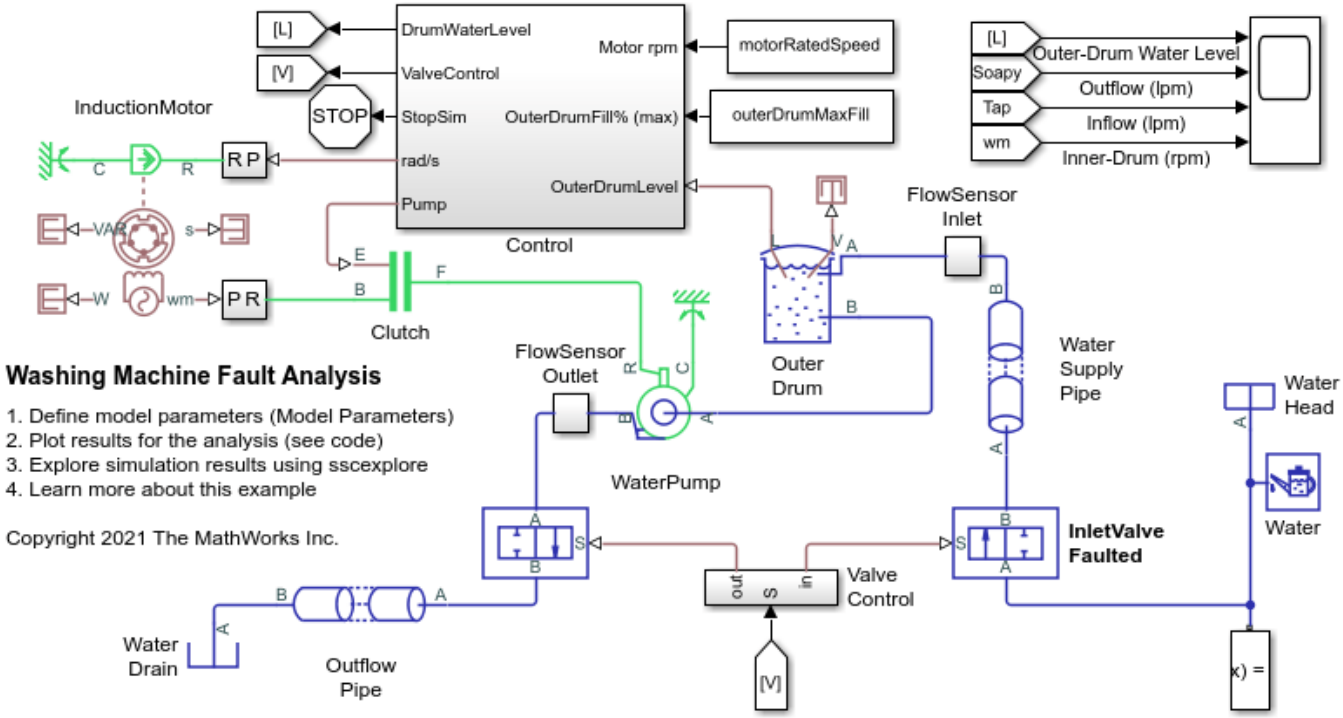
Washing Machine Fault Analysis

This example shows how to model a washing machine and introduce a fault in its operation. The machine water supply faults at a specific time and this example models the system response under this scenario. The electric motor switches off and the machine operation is aborted by discharging the partially filled outer drum.

Model Overview

```
open_system('sscv_washing_machine_fault_analysis.slx')
```

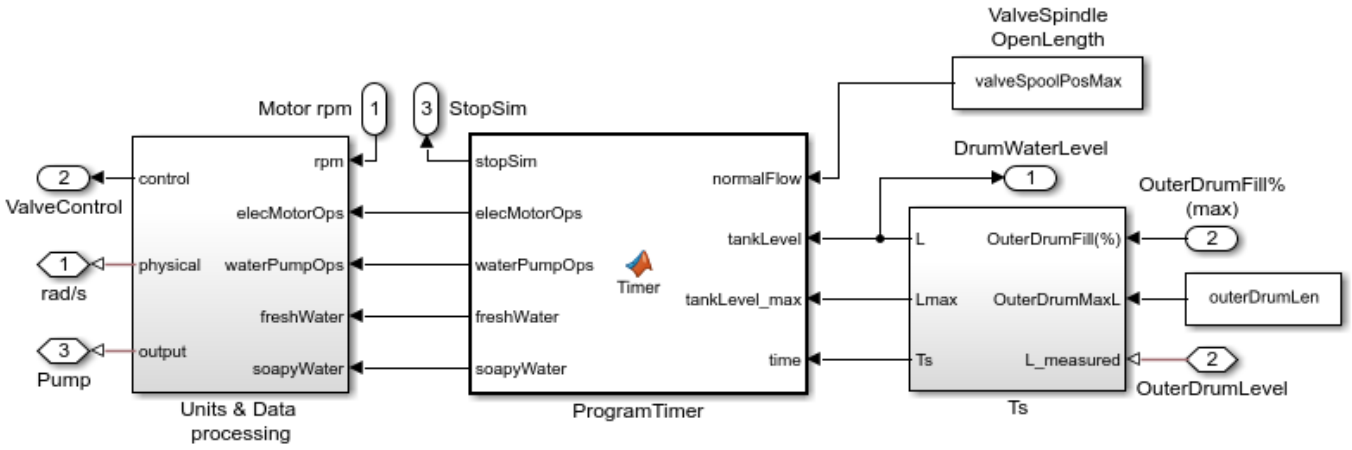
```
set_param(find_system('sscv_washing_machine_fault_analysis','FindAll', 'on','type','annotation',
```



Control Overview

To modify or edit the machine wash cycle, use the ProgramTimer subsystem. The ProgramTime subsystem comprises the response to a fault.

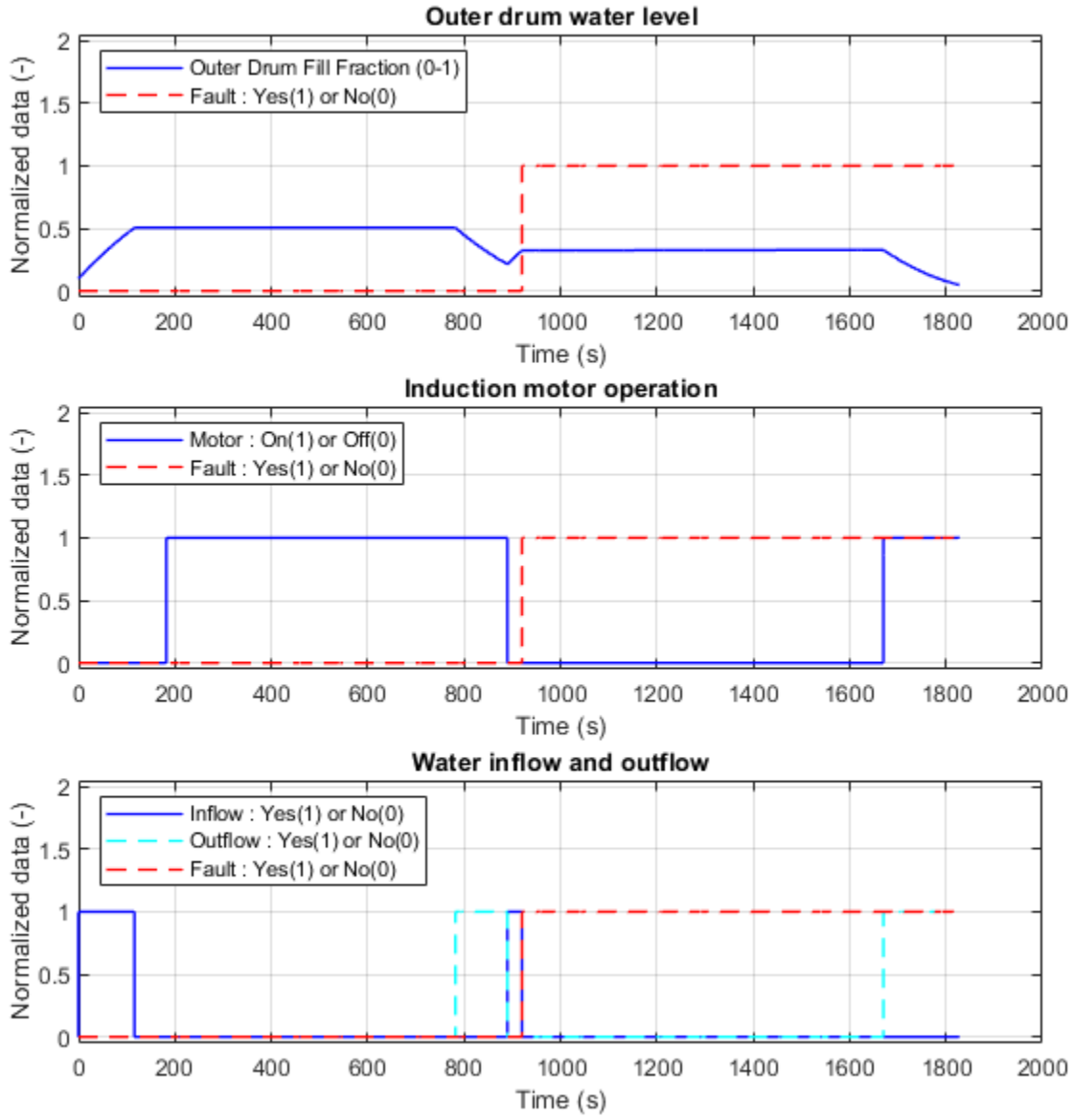
```
open_system('sscv_washing_machine_fault_analysis/Control','force')
```



Simulation Results from Simscape Logging

The plot below shows the water inlet and outlet valve operation (On/Off) and the pump operation with outer drum fill level. When the inlet valve faults, the pump switches off. After some time, the outlet valve opens to discharge the partially filled washing machine drum.

```
sscv_washing_machine_fault_analysis_plotResults;
```



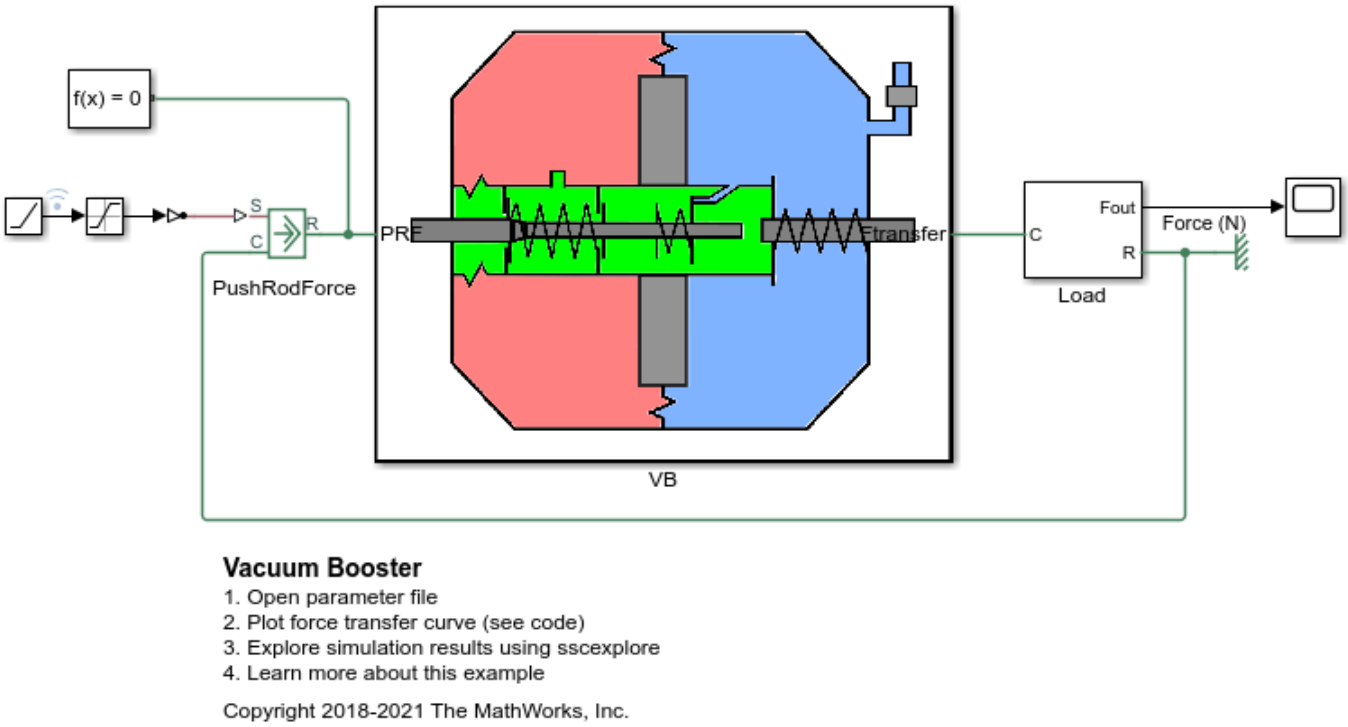

```
clear all
close all
bdclose all
```

Vacuum Booster

This example shows how to model, parameterize and test a vacuum booster. The model is used to generate the plot between applied and generated forces for the brake vacuum booster.

Model

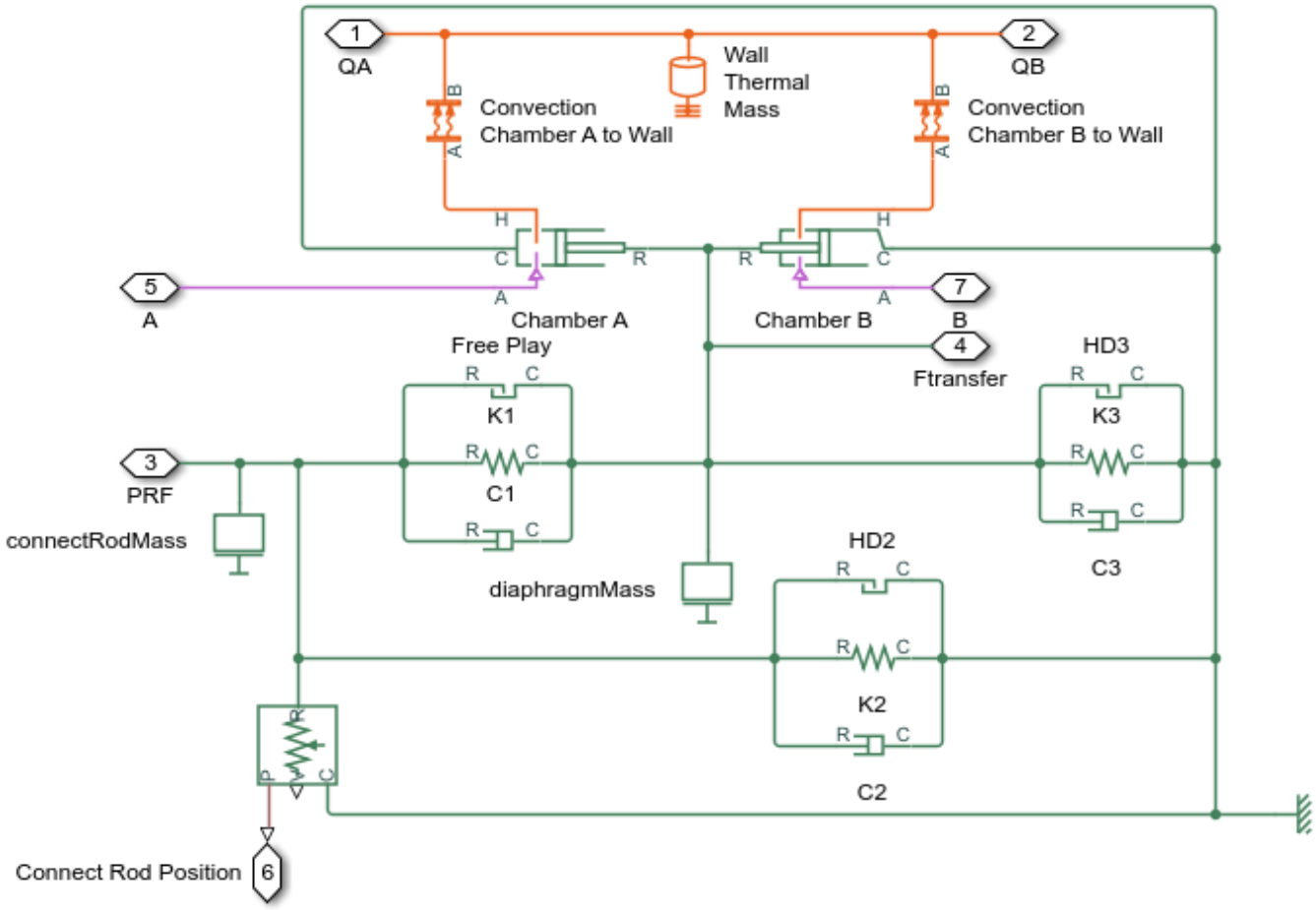
The following figure shows the model of a vacuum booster. The output of the model is the plot between applied and generated forces for the brake vacuum booster.



Diaphragm Chamber Subsystem

This subsystem shows the modeling of the diaphragm mass dynamics. The diaphragm has two sides. One side (chamber B) is permanently connected to the engine manifold, which experiences the vaccum pressure throughout the operation. However the other side (chamber A) is initially connected to the chamber B but after the rod moves a certain distance, based on design it gets disconnected from the chamber B and gets connected with the atmospheric pressure.

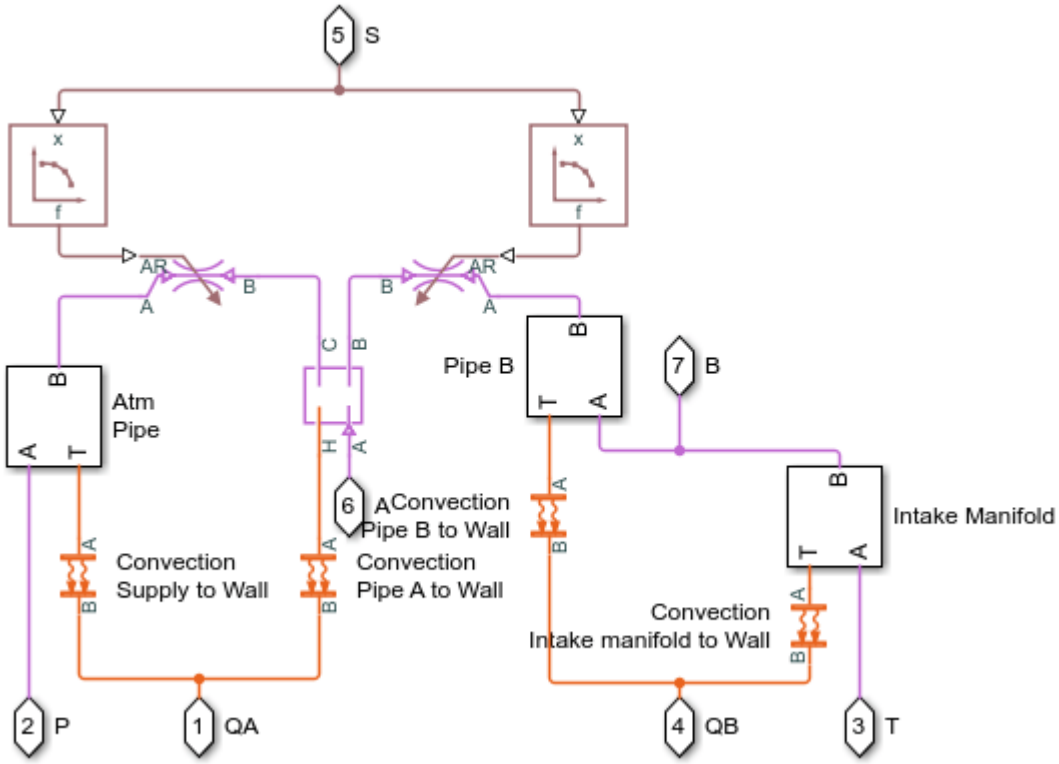
Open Subsystem



Manifold Subsystem

This subsystem shows the modeling of the gas dynamics.

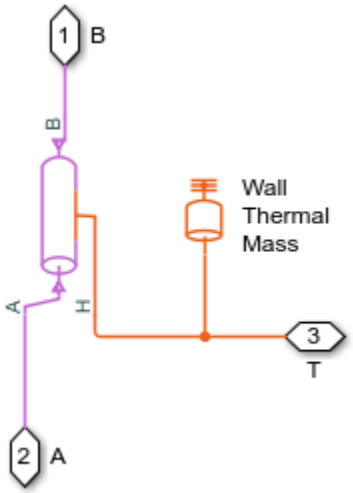
Open Subsystem



Pipe Subsystem

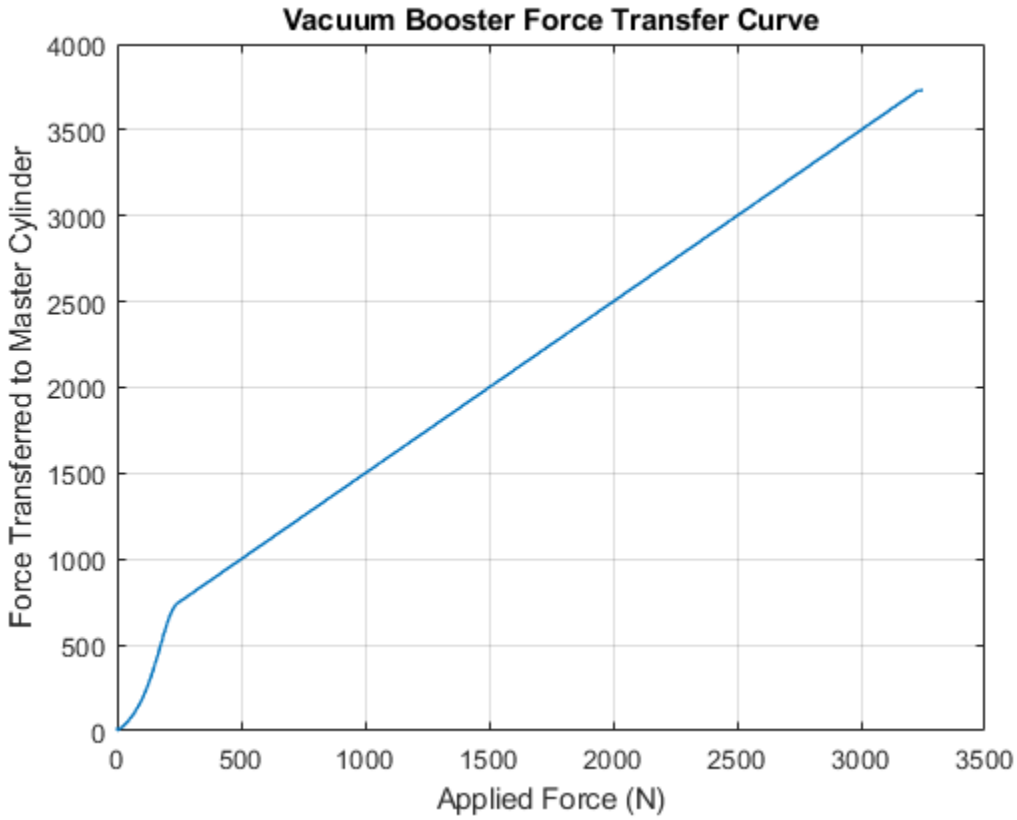
This subsystem shows the modeling of the various connection pipes in the vacuum booster.

Open Subsystem



Simulation Results from Simscape Logging

This script generates a plot between applied and generated forces for the brake vacuum booster.

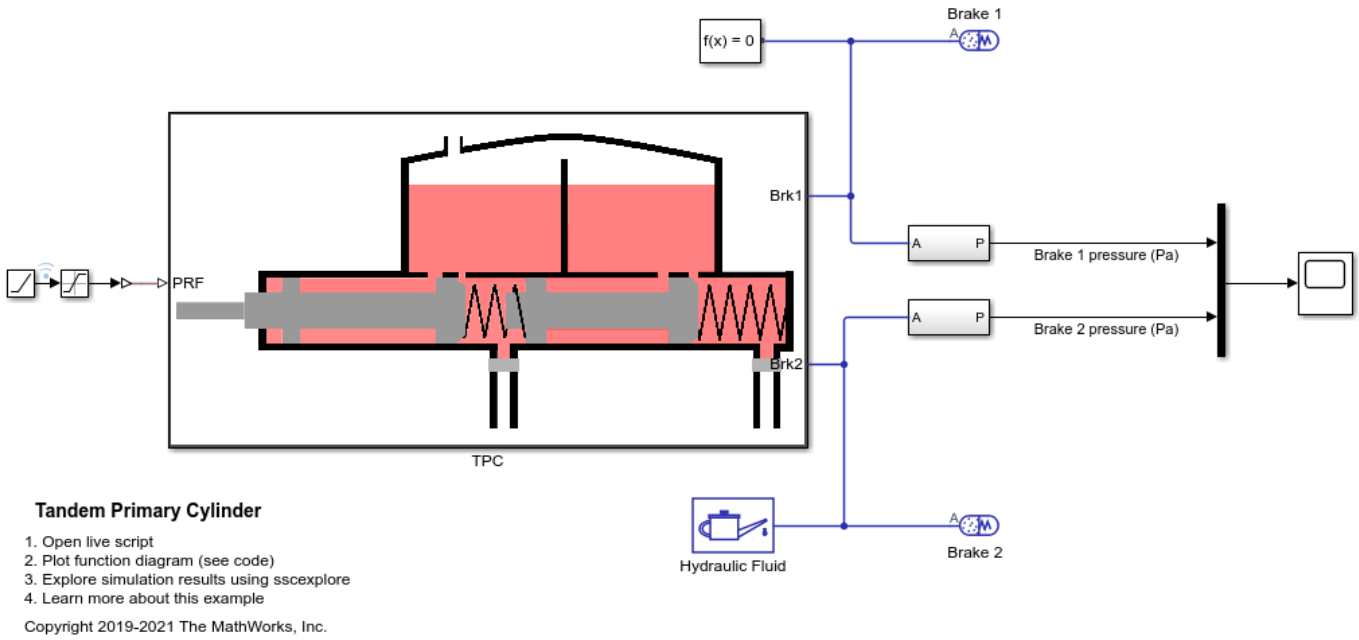

Vacuum Booster Force Transfer Curve
Force Transferred to Master Cylinder
Applied Force (N)
0
500
1000
1500
2000
2500
3000
3500
4000

Tandem Primary Cylinder

This example shows how to model, parameterize and test a tandem primary cylinder starting from manufacturer datasheet information. First, there is a brief discussion on the mathematical modeling of the system. Given the numerical data extracted from the datasheet, optimization is then used to determine remaining unknown parameters. The model is then simulated and the resulting push rod force - brake pressure relationship curve is compared with the curve provided on the manufacturer datasheet. Understanding the behavior of the tandem primary cylinder is an important prerequisite to selection of other braking system components.

Model

The following figure shows the tandem primary cylinder test harness. Spring-loaded accumulators are used for loading in the 1^{st} and the 2^{nd} circuits in place of disk or drum brakes. The input to the model is the push rod force coming from the lever mechanism or the brake booster. Output of the model is the pressures achieved in the brake circuits 1 and 2 respectively.



Manufacturer datasheet data

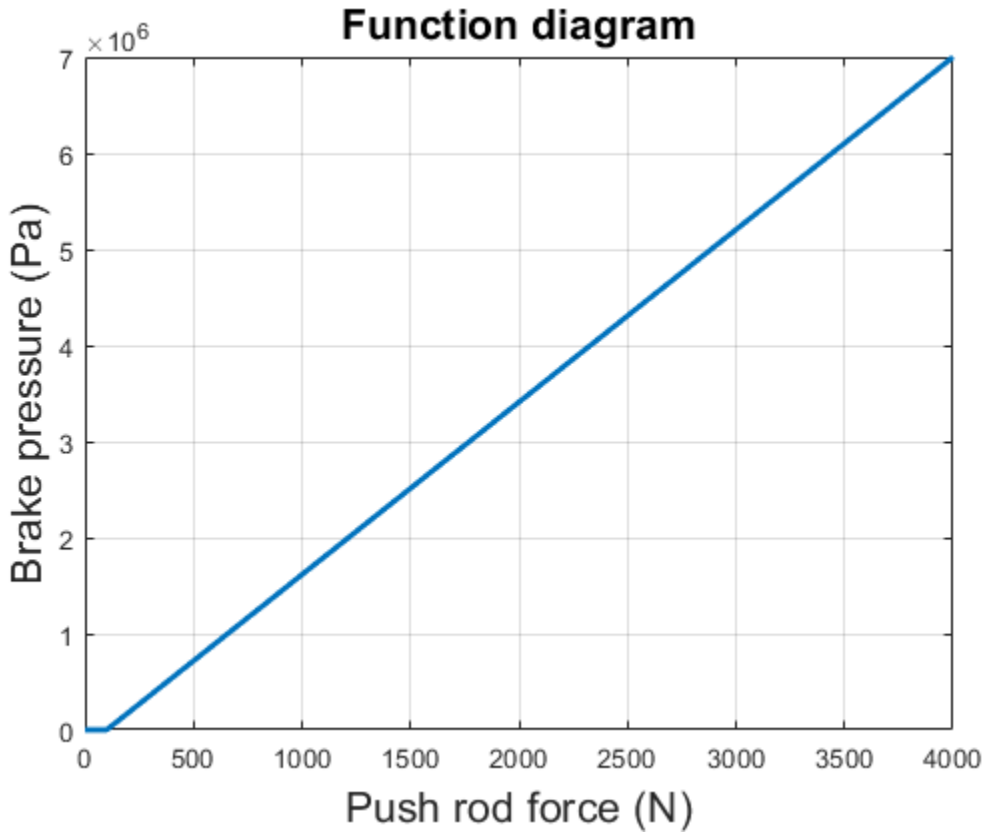
The supplier has provided the following data in the datasheet.

1. *dp* : first and second circuit piston diameter.
2. *stroke1* : first circuit piston stroke.
3. *stroke2* : second circuit piston stroke.
4. *totalStroke* : overall stroke.
5. *disp1* : first circuit displacement.
6. *disp2* : second circuit displacement.
7. *totalDisp* : net displacement.

8 *maxPress* : max pressure.

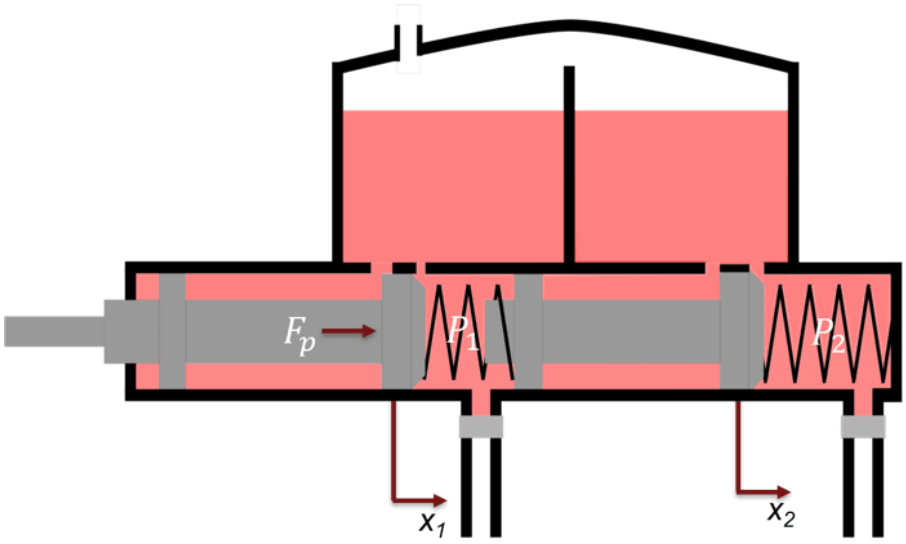
Pressure development characteristics

The supplier datasheet gives the following function diagram or the push rod force-brake pressure curve for the tandem primary cylinder.



Equations of motion (EoM) of system

The tandem primary cylinder is shown in the figure below. Mass 1 and mass 2 use the coordinate frame shown below. Only the mechanical EoM of the system are shown and the fluid dynamics part is not shown.



$$m_1\frac{d^2x_1}{dt^2} + c_1(\frac{dx_1}{dt} - \frac{dx_2}{dt}) + k_1(x_1 - x_2) = F_p - p_1A - Pre_{LL} + R_1, \text{ when } x_1 = 0$$

$$m_2\frac{d^2x_2}{dt^2} + c_1(\frac{dx_2}{dt} - \frac{dx_1}{dt}) + c_2\frac{dx_2}{dt} + k_1(x_2 - x_1) + k_2x_2 = p_1A - p_2A + Pre_{LL} - Pre_{LR} + R_2, \text{ when } x_2 = 0$$

$$m_1\frac{d^2x_1}{dt^2} + c_1(\frac{dx_1}{dt} - \frac{dx_2}{dt}) + k_1(x_1 - x_2) = F_p - p_1A - Pre_{LL}, \text{ when } x_1 > 0$$

$$m_2\frac{d^2x_2}{dt^2} + c_1(\frac{dx_2}{dt} - \frac{dx_1}{dt}) + c_2\frac{dx_2}{dt} + k_1(x_2 - x_1) + k_2x_2 = p_1A - p_2A + Pre_{LL} - Pre_{LR}, \text{ when } x_2 > 0$$

where,

x_1 and x_2 are positions of piston mass 1 and mass 2 from the left side assumed hard stop.

m_1 and m_2 are masses of piston 1 and 2 respectively.

c_1 and c_2 are damping coefficients of piston 1 and 2 respectively.

k_1 and k_2 are stiffness coefficients of left (Spring 1) and right (spring 2) respectively.

F_p is force applied on push rod of primary cylinder.

P_1 and P_2 are pressures in brake circuit 1 and 2 respectively.

R_1 and R_2 are reaction forces on mass 1 and 2 respectively when both masses are at the leftmost position.

Pre_{LL} and Pre_{LR} are preloads on left and right side springs respectively.

Parameter estimation method

Some parameters, which are important for functionality assessment, are not provided in the manufacturer's datasheet, therefore to simulate the model it is pertinent to estimate these parameters. The EoM are converted to steady state equations to estimate unknown parameters.

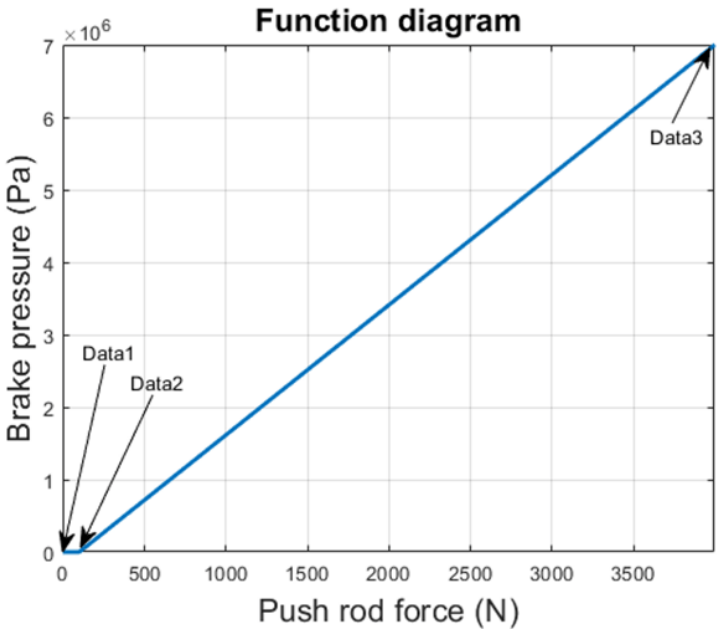
$$k_1(x_1 - x_2) = F_p - p_1 A - Pre_{LL} + R_1, \text{ when } x_1 = 0$$

$$k_1(x_2 - x_1) + k_2 x_2 = p_1 A - p_2 A + Pre_{LL} - Pre_{LR} + R_2, \text{ when } x_2 = 0$$

$$k_1(x_1 - x_2) = F_p - p_1 A - Pre_{LL}, \text{ when } x_1 > 0$$

$$k_1(x_2 - x_1) + k_2 x_2 = p_1 A - p_2 A + Pre_{LL} - Pre_{LR}, \text{ when } x_2 > 0$$

Data for the function diagram is explained with following figure.



	Datasheet Data			
Data Point	Force (N)	Pressure (bar)	Stroke 1 (m)	Stroke 2 (m)
1	F_{p1}	P_{11} & P_{21}	x_{11}	x_{21}
2	F_{p2}	P_{12} & P_{22}	x_{12}	x_{22}
3	F_{p3}	P_{13} & P_{23}	x_{13}	x_{23}

where,

F_{p1}, F_{p2} and F_{p3} are the push rod forces for points 1,2, and 3 respectively.

$P_{1*} = P_{2*}$,Here $*$ is $1, 2, 3$ for the brake circuit pressures at points 1,2, and 3 respectively.

x_{11} and $x_{21} = 0$ represent the leftmost positions for piston 1 and 2 respectively.

$x_{13} = TotalStroke$ and $x_{23} = Stroke2$ as per the manufacturer datasheet data.

x_{12} and x_{22} are piston positions at data point 2. This point needs to be iterated on during optimisation to estimate the unknown parameters.

The estimation scheme is as follows:

$$\begin{bmatrix} x_{11}-x_{21} & 0 & 1 & 0 & -1 & 0 \\ x_{21}-x_{11} & x_{21} & -1 & 1 & 0 & -1 \\ x_{12}-x_{22} & 0 & 1 & 0 & 0 & 0 \\ x_{22}-x_{12} & x_{22} & -1 & 1 & 0 & 0 \\ x_{13}-x_{23} & 0 & 1 & 0 & 0 & 0 \\ x_{23}-x_{13} & x_{23} & -1 & 1 & 0 & 0 \end{bmatrix} \begin{bmatrix} k_1 \\ k_2 \\ \mathrm{Pre}_{\mathrm{LL}} \\ \mathrm{Pre}_{\mathrm{LR}} \\ R_1 \\ R_2 \end{bmatrix} = \begin{bmatrix} F_{\mathrm{P1}} - P_{11}A \\ P_{11}A - P_{21}A \\ F_{\mathrm{P2}} - P_{12}A \\ P_{12}A - P_{22}A \\ F_{\mathrm{P3}} - P_{13}A \\ P_{13}A - P_{23}A \end{bmatrix}$$

The estimation should yield results with the following conditions met:

1. R_1 and R_2 should be greater than or equal to 0.
2. Pre_{LR} should be greater than Pre_{LL}.
3. k_1 and k_2 should be greater than 0.

Assumed parameters

Some parameters are assumed as they are neither given in the datasheet nor can they be estimated. The following is the list:

1. *c1* is damping coefficient for piston 1.
2. *c2* is damping coefficient for piston 2.
3. *mass1* is piston 1 mass.
4. *mass2* is piston 2 mass.
5. *v2D* is brake circuit 1 dead volume.
6. *v2M* is brake circuit 1 maximum volume.
7. *v4D* is brake circuit 2 dead volume.
8. *v4M* is brake circuit 2 maximum volume.
9. *a1Ori* is compensating orifice area.
10. *a2Ori* is brake circuit orifice area

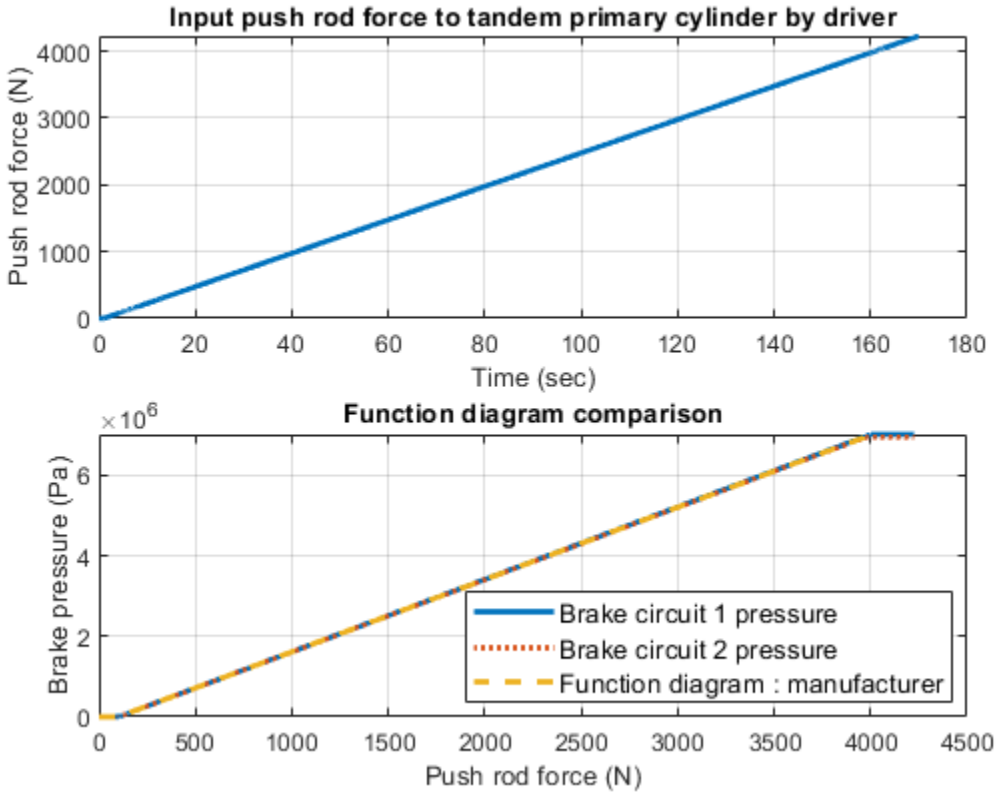
Load circuit

It is important to test the system with the correct load. In the test model, loading is done on the tandem primary cylinder using spring-based accumulators. Setting up correct parameters in the accumulators is important. From the datasheet it is known what should be the stroke values for brake circuits 1 and 2 respectively. Iterated parameters x_{12} and x_{22} are important for deciding the fluid chamber capacity of the accumulators. Assumed or estimated parameters for the load circuit accumulators are:

1. vol1 is fluid chamber capacity for brake circuit 1.
2. vol2 is fluid chamber capacity for brake circuit 2.
3. loadP1 is pressure at full capacity for brake circuit 1.
4. loadP2 is pressure at full capacity for brake circuit 2.

Simulation results from simscape logging

The model generates push rod force-brake pressure plots for selected manufacturer designs. The applied push rod force to the tandem primary cylinder is ramped at a 25 N/sec rate in the simulation .

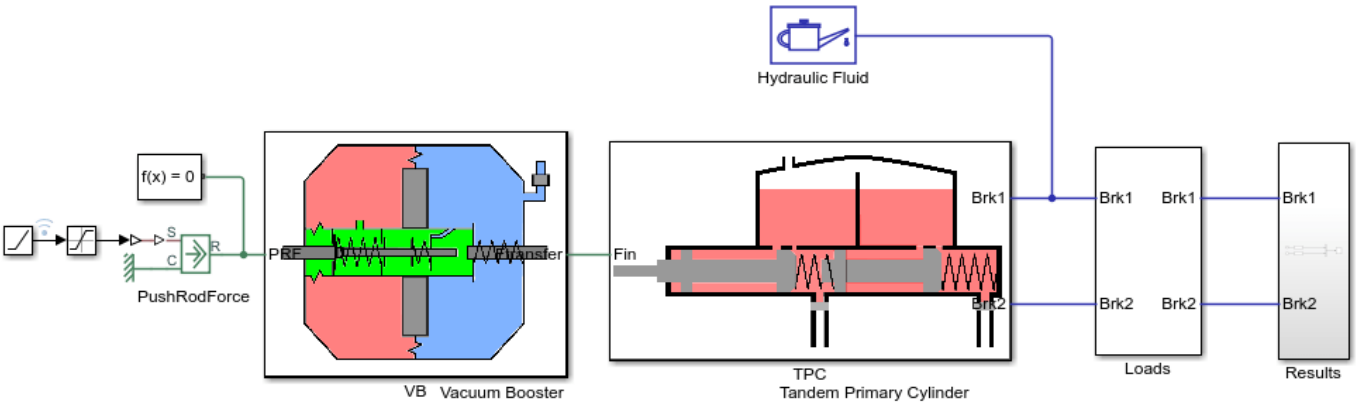


Vacuum Boosted Tandem Primary Cylinder

This example shows how to model, parameterize and test a vacuum boosted tandem primary cylinder. A model is used to generate the plot between the applied force and the generated pressures in the brake lines.

Model

The following figure shows the model of a vacuum boosted tandem primary cylinder. The model has push rod force as input and it generates the pressures in the brake lines as output.



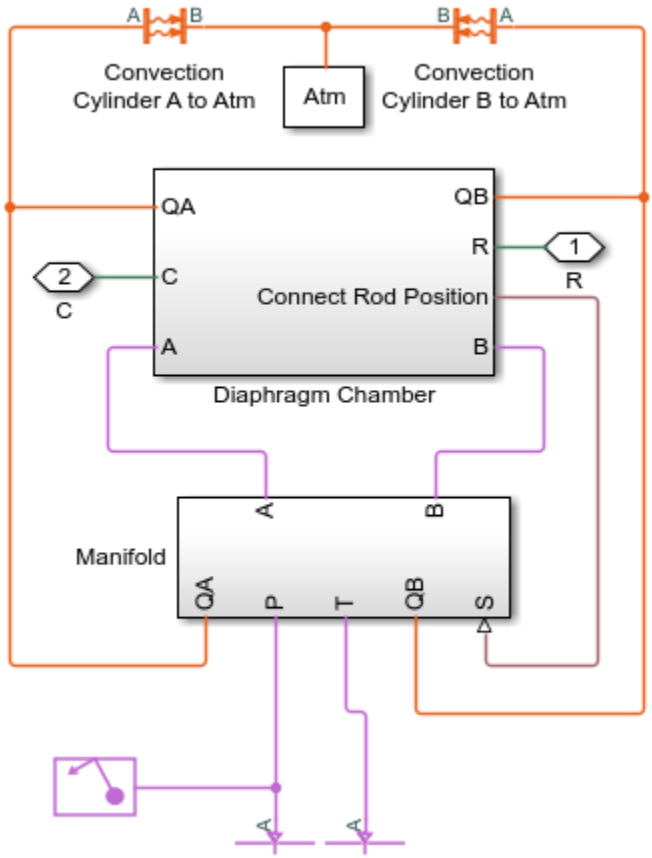
Vacuum Boosted Tandem Primary Cylinder
1. Open vacuum booster parameterization file
2. Open tandem primary cylinder parameterization file
3. Plot brake pressures (see code)
4. Explore simulation results using sscexplore
5. Learn more about this example

Copyright 2018-2021 The MathWorks, Inc.

Vacuum Booster Subsystem

This subsystem shows the modeling of the vacuum booster. A vacuum booster boosts the force applied by a driver to the tandem primary cylinder. The boosted force depends on the diameter of the diaphragm in the vacuum booster and the vacuum generated due to the air suction through the intake manifold of the engine.

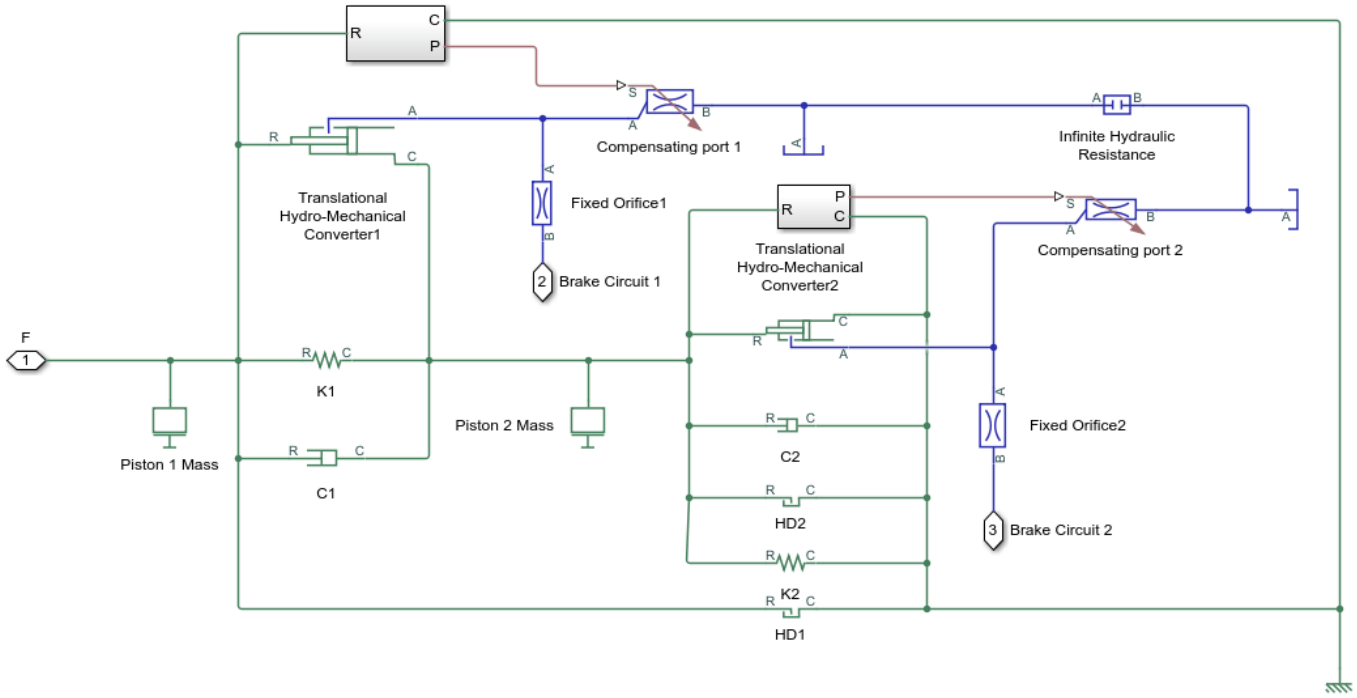
Open Subsystem



Tandem Primary Cylinder Subsystem

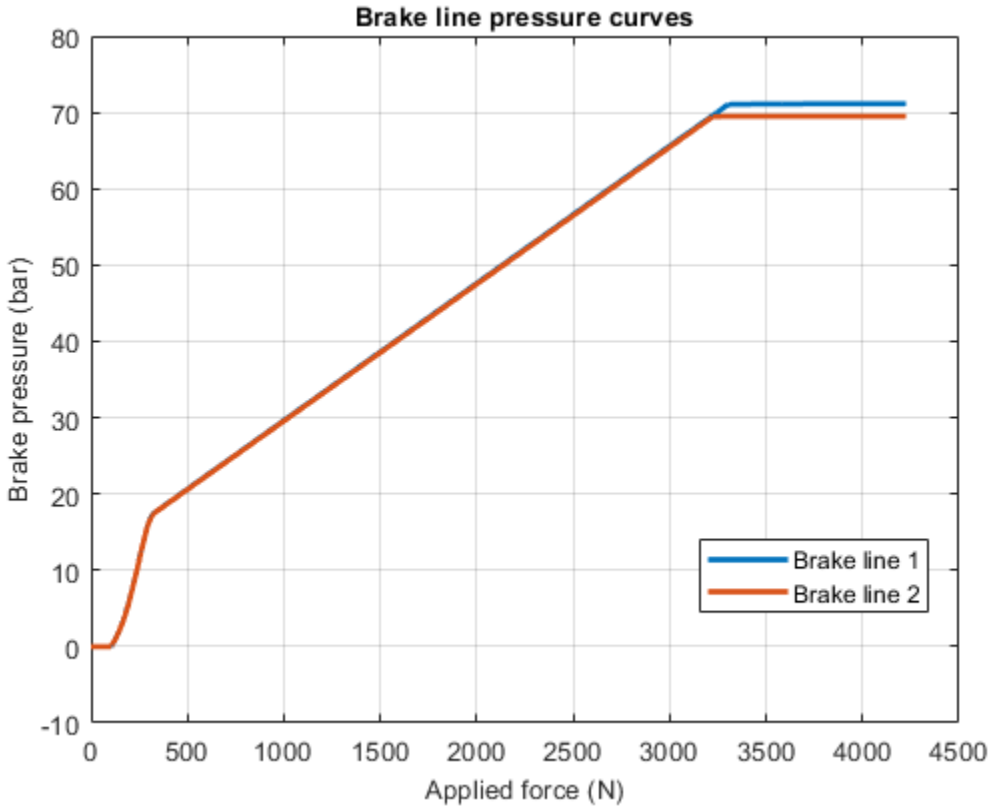
This subsystem shows the modeling of the tandem primary cylinder. A tandem primary cylinder pushes the hydraulic fluid to the caliper disk or the drum brakes through the hydraulic connections or pipes in an automotive vehicle. Pressure is generated in the hydraulic chambers of the brakes due to the fluid flow from the tandem primary cylinder.

Open Subsystem



Simulation Results from Simscape™ Logging

The model generates the relationship between the applied force and the generated pressures in the brake lines as shown in the plot below.

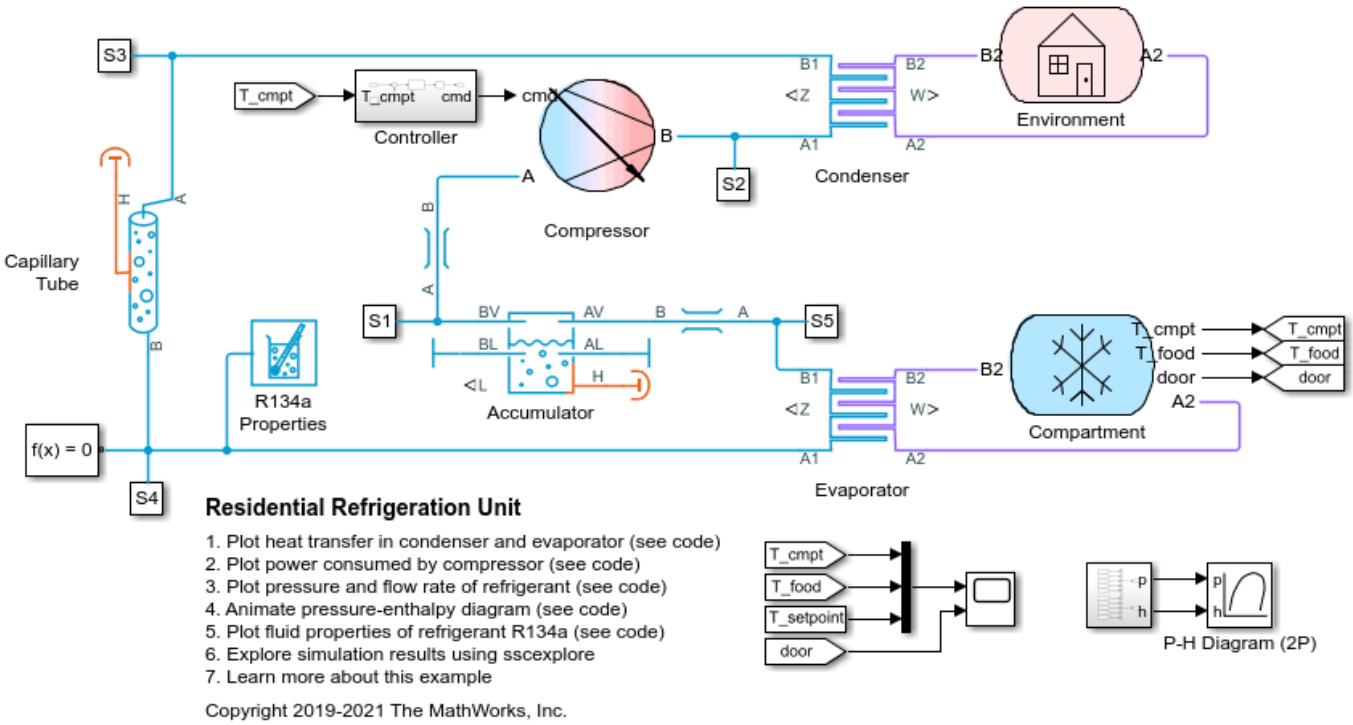

Brake line pressure curves
Brake pressure (bar)
Applied force (N)
Brake line 1
Brake line 2
80
70
60
50
40
30
20
10
0
-10
0
500
1000
1500
2000
2500
3000
3500
4000
4500

Residential Refrigeration Unit

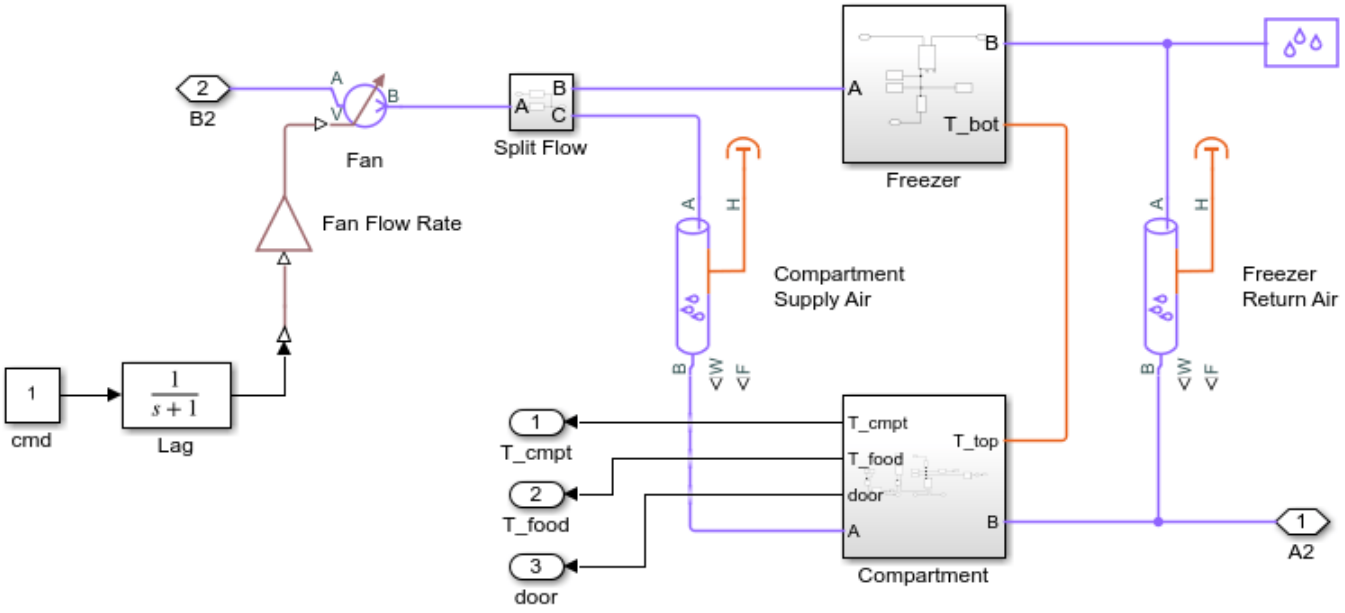
This example models a basic refrigeration system that transfers heat between the refrigerant two-phase fluid and the environment moist air mixture. The compressor drives the R134a refrigerant through a condenser, a capillary tube, and an evaporator. An accumulator ensures that only vapor returns to the compressor.

Two fans drive moist air flow over the condenser and the evaporator. The evaporator air flow is divided between the freezer compartment and the regular compartment. The controller turns the compressor on and off to keep the compartment air temperature at around 4 degC.

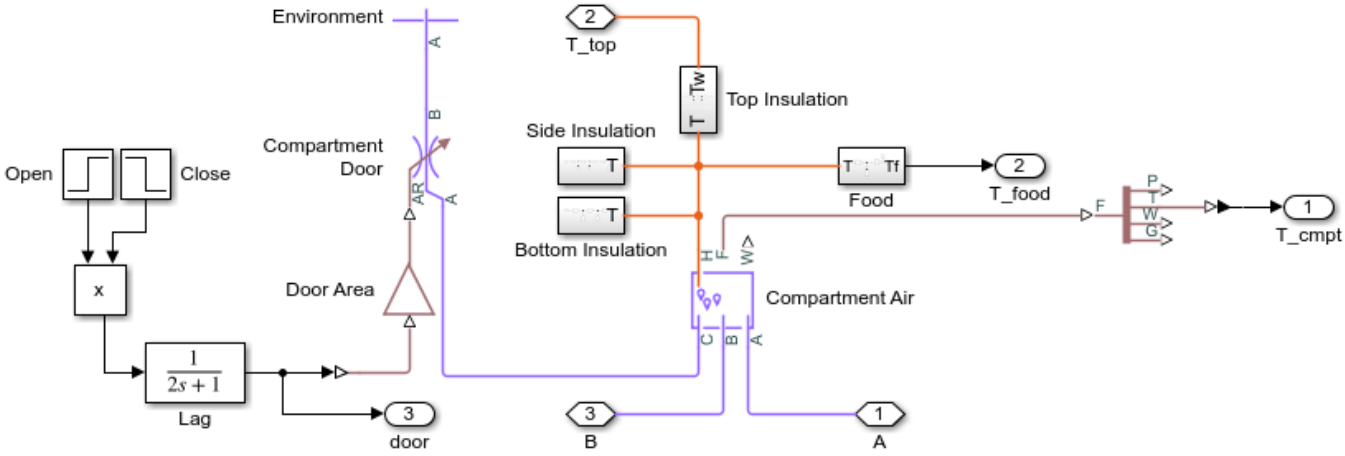
Model



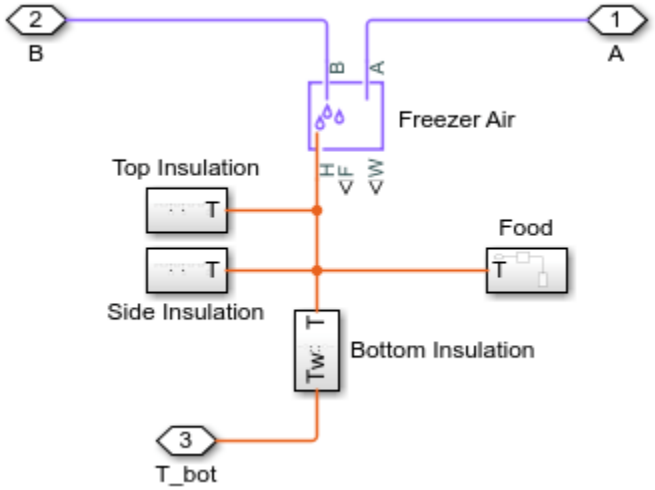
Compartment Subsystem



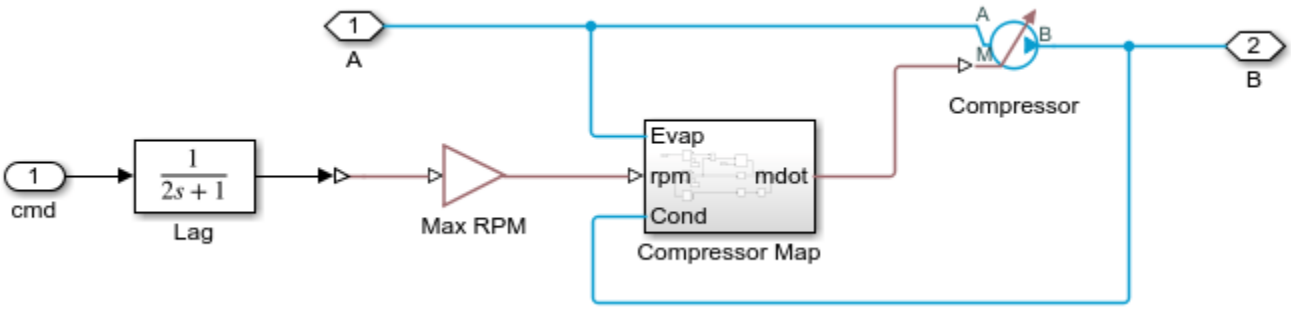
Compartment Subsystem



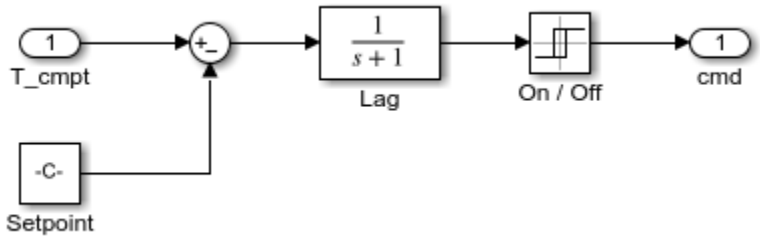
Freezer Subsystem



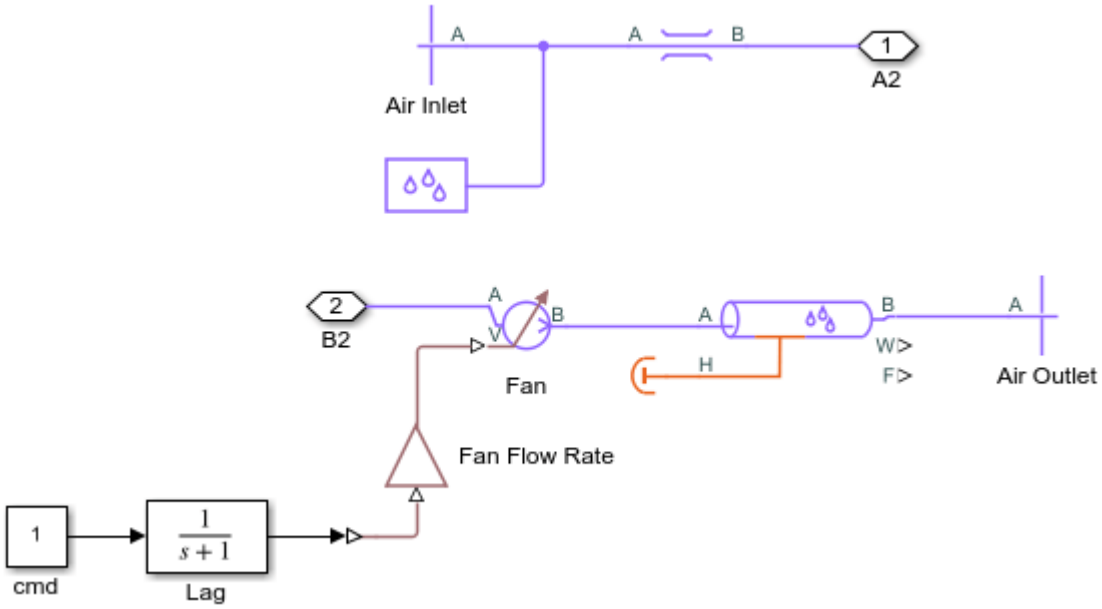
Compressor Subsystem



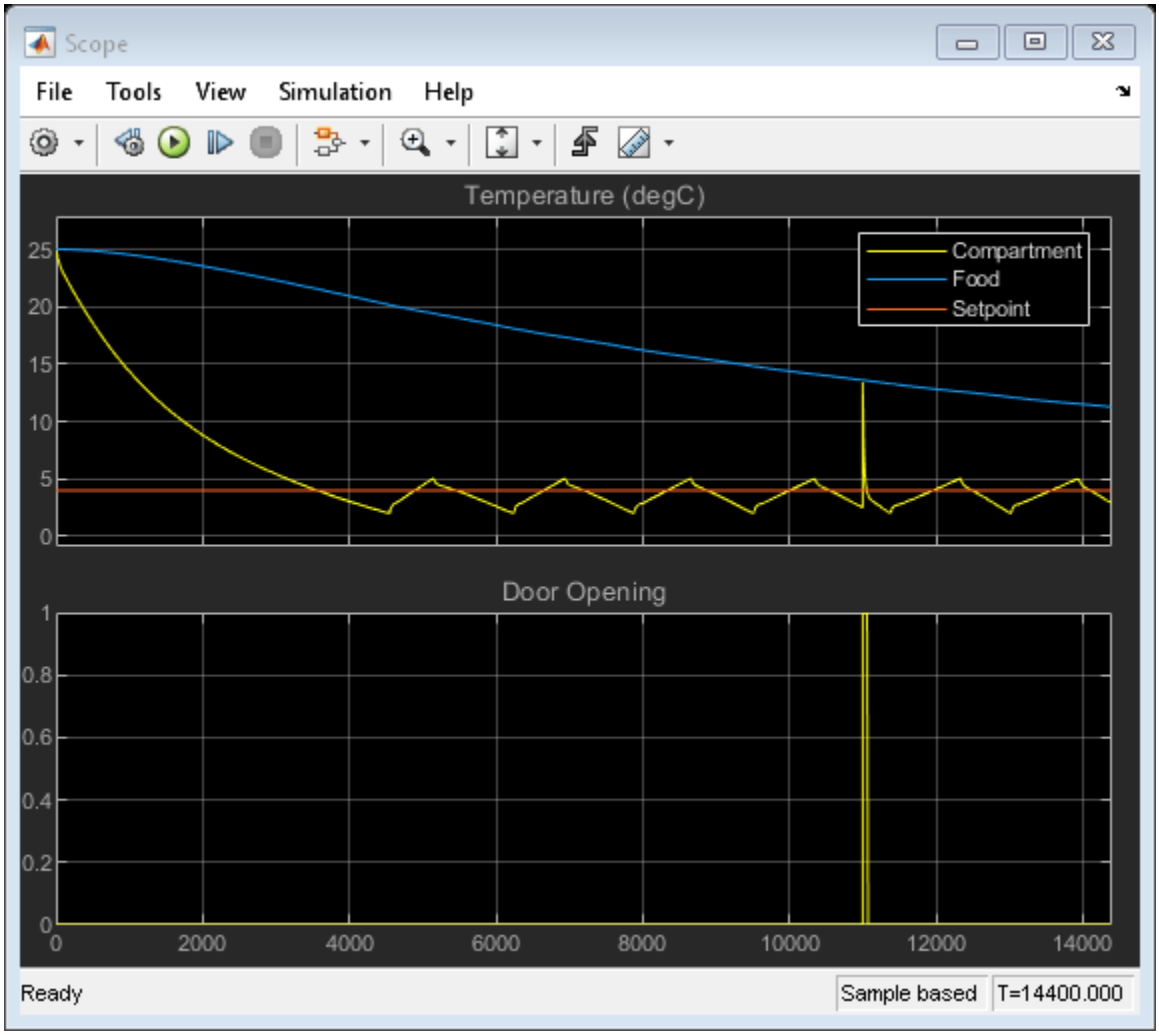
Controller Subsystem



Environment Subsystem

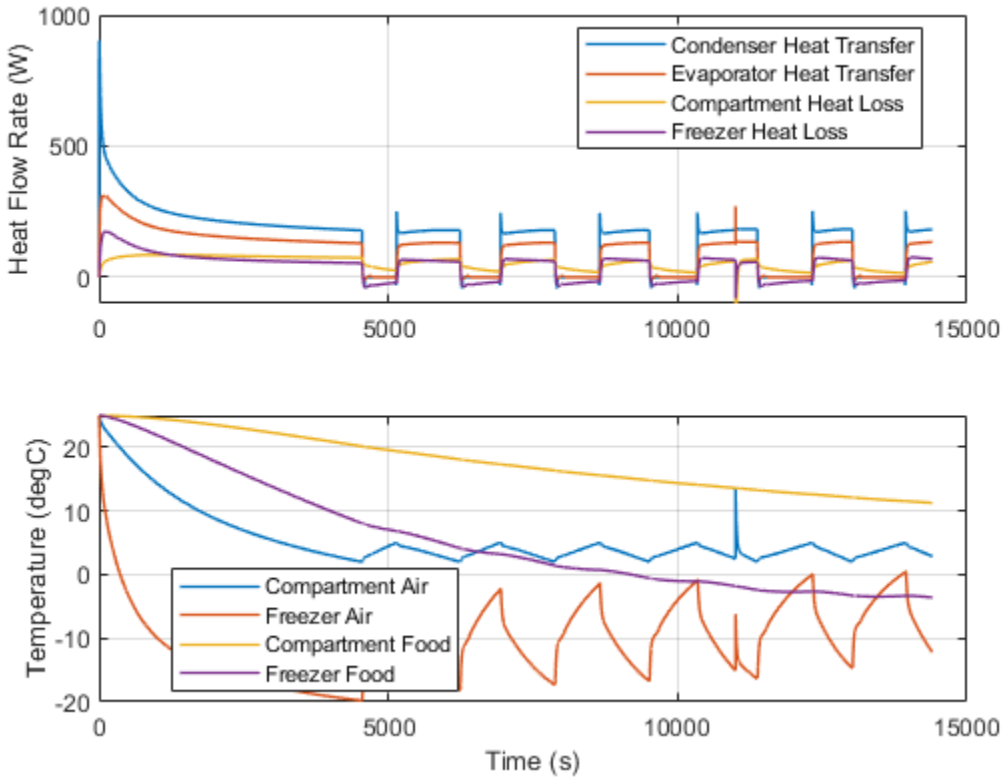


Simulation Results from Scopes

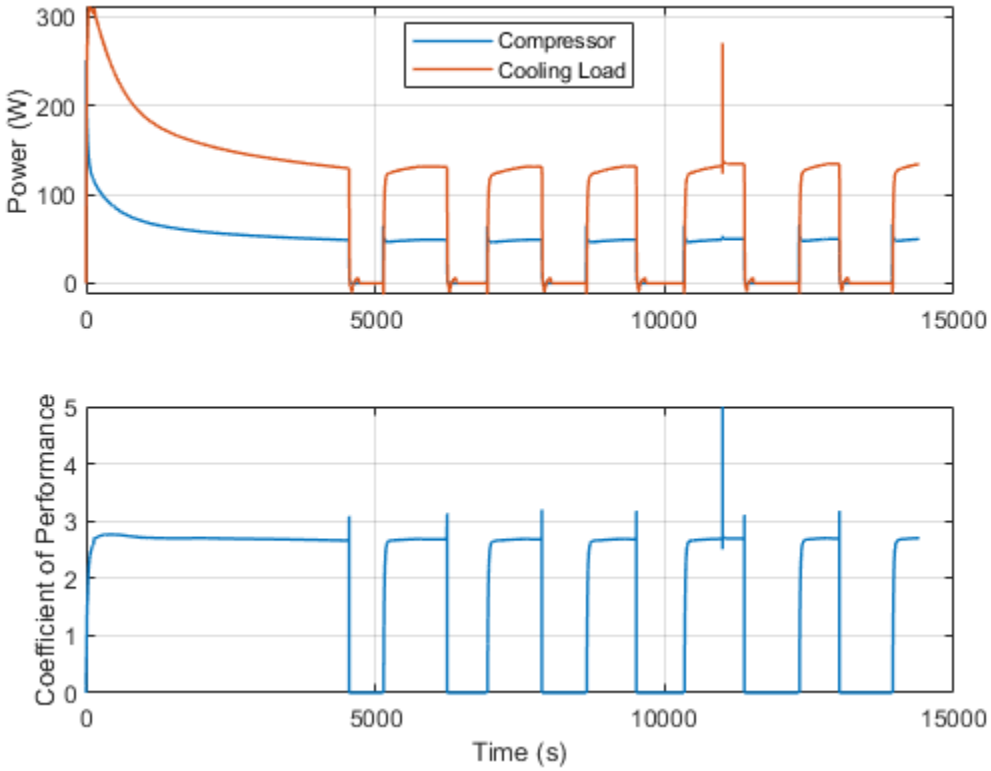


Simulation Results from Simscape Logging

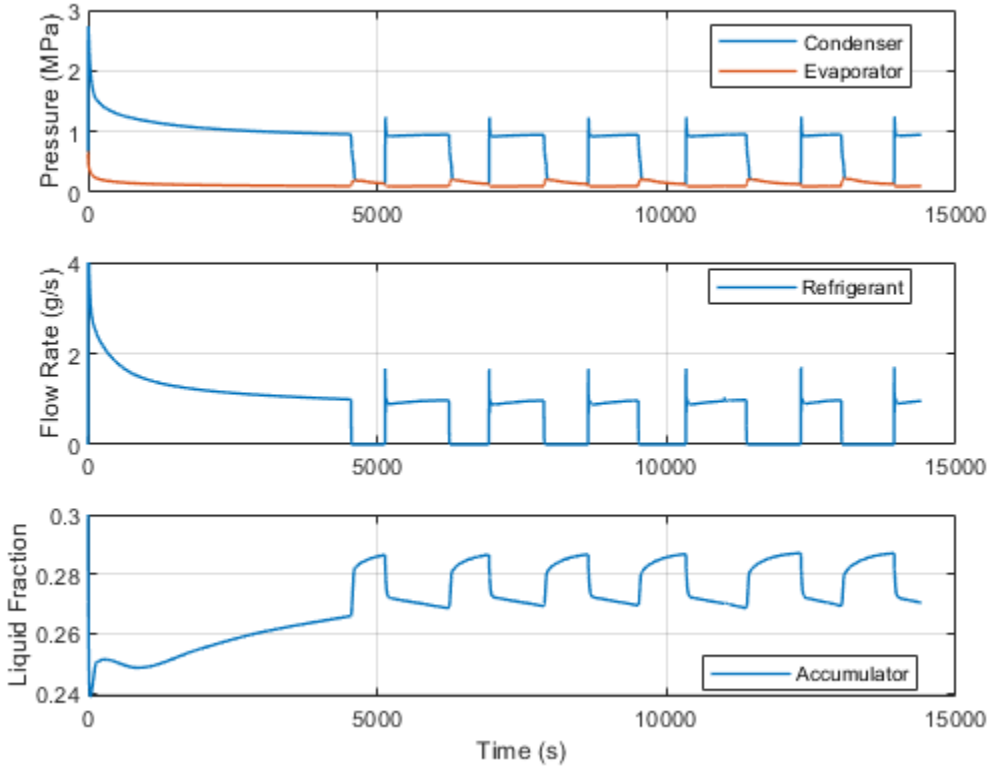
This plot shows the rate of heat transfer between refrigerant and moist air in the condenser and evaporator as well as the rate of heat loss through the insulation of the compartment and freezer. It also shows the temperature of cold air and food in the compartment and freezer. At 11000 s, the compartment door is opened for 60 s, resulting in a spike in compartment temperature.



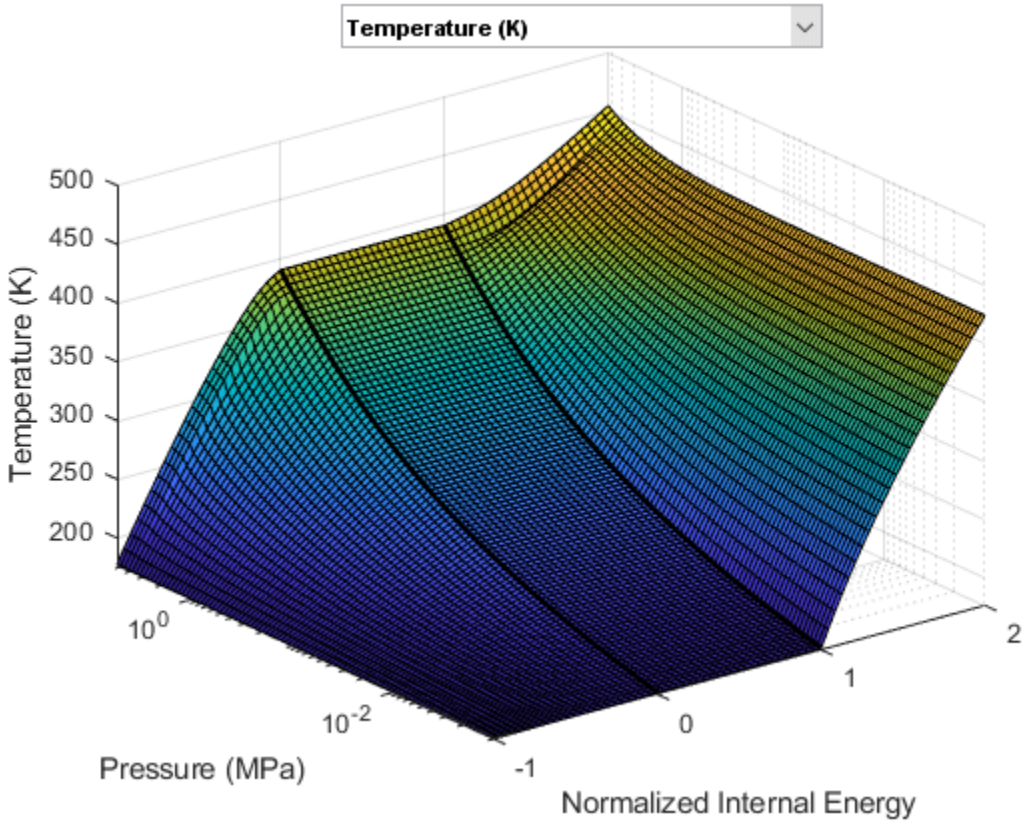
This plot shows the power consumed by the compressor and the cooling load of the refrigeration system, which is the rate of heat transfer in the evaporator. The coefficient of performance is the ratio of the cooling load and the power consumed.

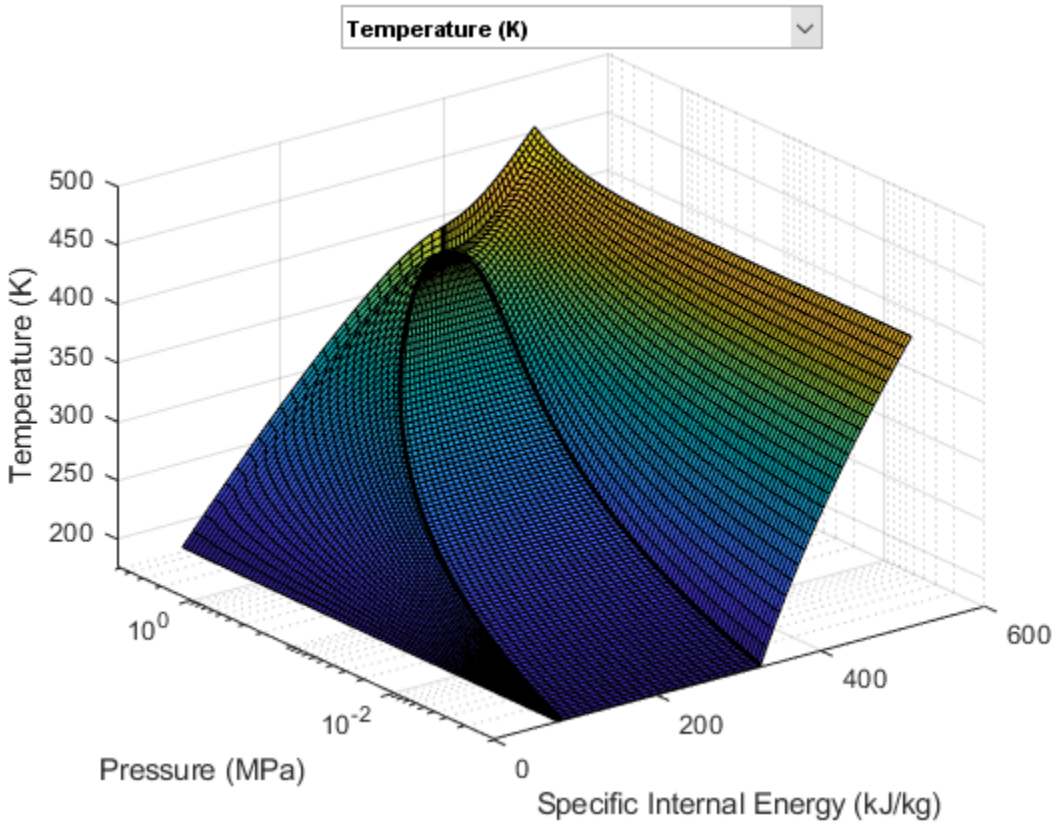


This plot shows refrigerant pressure and mass flow rate. The high pressure line is at around 1 MPa and the lower pressure line is at around 0.1 MPa. The nominal refrigerant flow rate is 1 g/s. The plot also shows the liquid volume fraction in the accumulator.


Pressure (MPa)
Condenser
Evaporator
Flow Rate (g/s)
Refrigerant
Liquid Fraction
Accumulator
Time (s)
0
5000
10000
15000

Fluid Properties



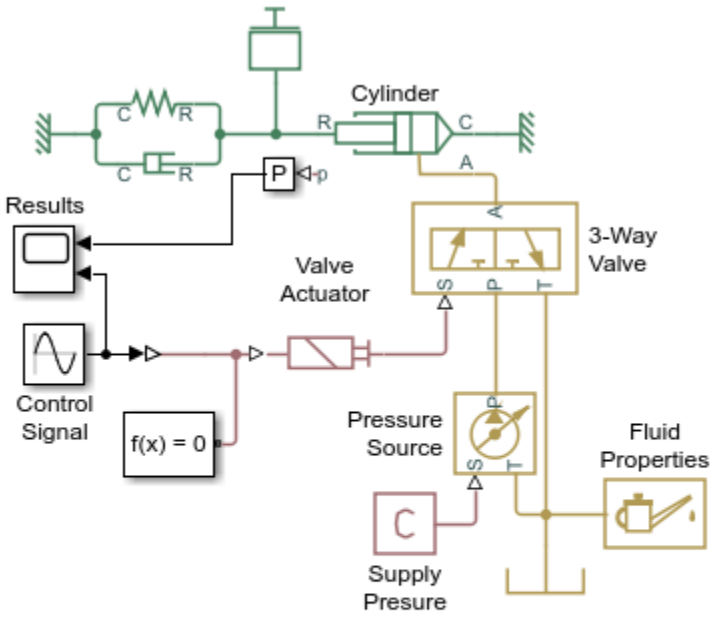

Temperature (K)
500
450
400
350
300
250
200
Temperature (K)
10^0
10^{-2}
Pressure (MPa)
0
200
400
600
Specific Internal Energy (kJ/kg)

Single-Acting Cylinder with 3-Way Valve

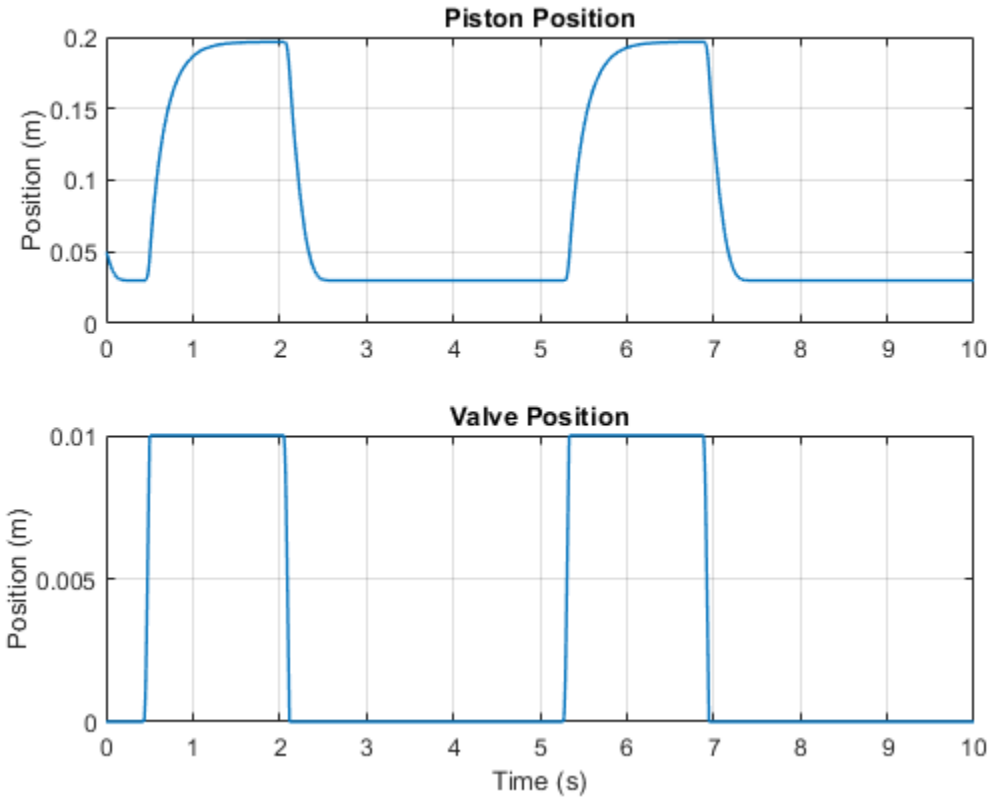
This example shows a single-acting hydraulic cylinder controlled by an 3-way directional valve. It drives a load consisting of a mass, viscous friction, and preloaded spring. The pump unit is assumed to be powerful enough to maintain constant pressure at the valve inlet.

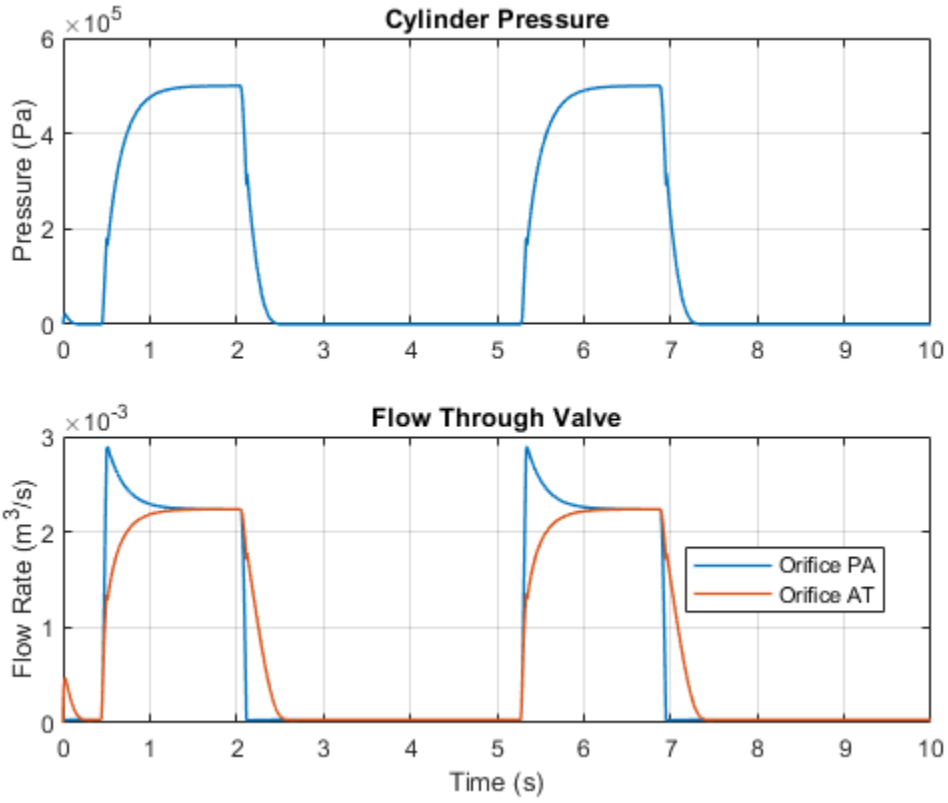
The valve is modeled by a 2-Position Valve Actuator block (Valve Actuator) and a 3-Way Directional Valve block (3-Way Valve). At the start of simulation, valve orifices A and T are connected and the cylinder chamber is connected to the tank. When the control signal for the valve exceeds 12, the valve spool moves, connecting valve ports P and A and closing the path between valve ports A and T. As a result, the cylinder chamber is connected to the pressure source and moves the load.

Model



Simulation Results from Simscape Logging

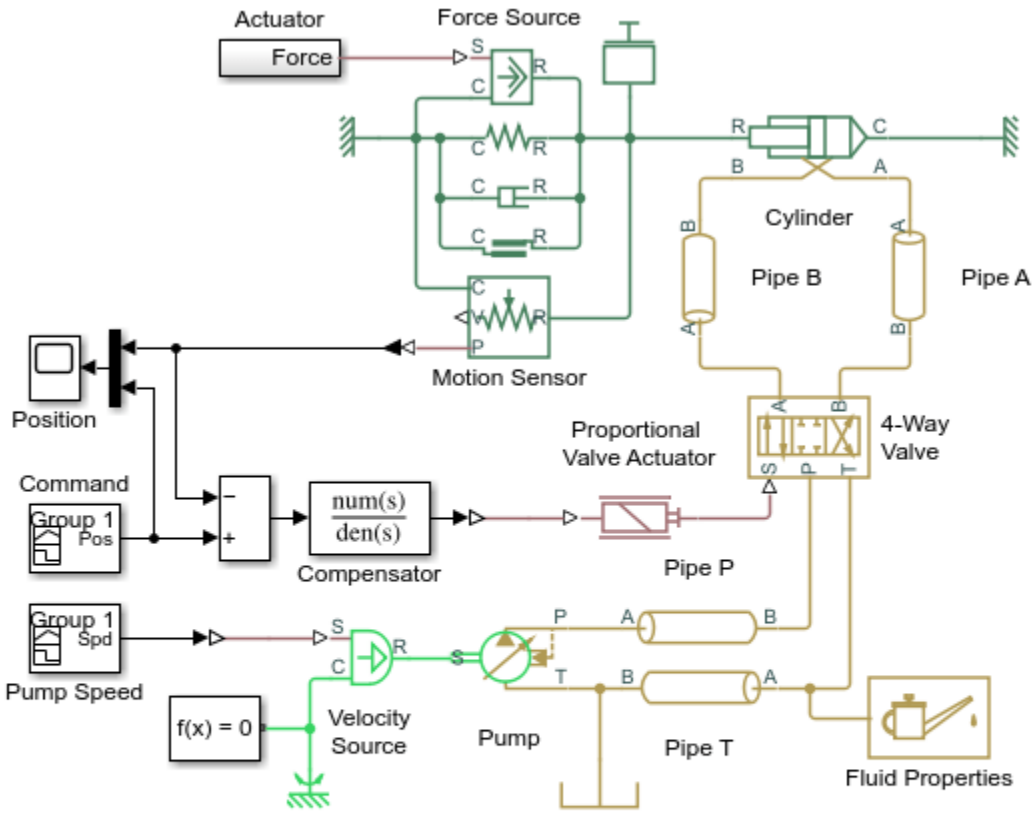



Cylinder Pressure
×10^5
Pressure (Pa)
0
2
4
6
0 1 2 3 4 5 6 7 8 9 10
Flow Through Valve
×10^-3
Flow Rate (m^3/s)
0
1
2
3
Orifice PA
Orifice AT
0 1 2 3 4 5 6 7 8 9 10
Time (s)

Closed-Loop Actuator with Variable-Displacement Pressure-Compensated Pump

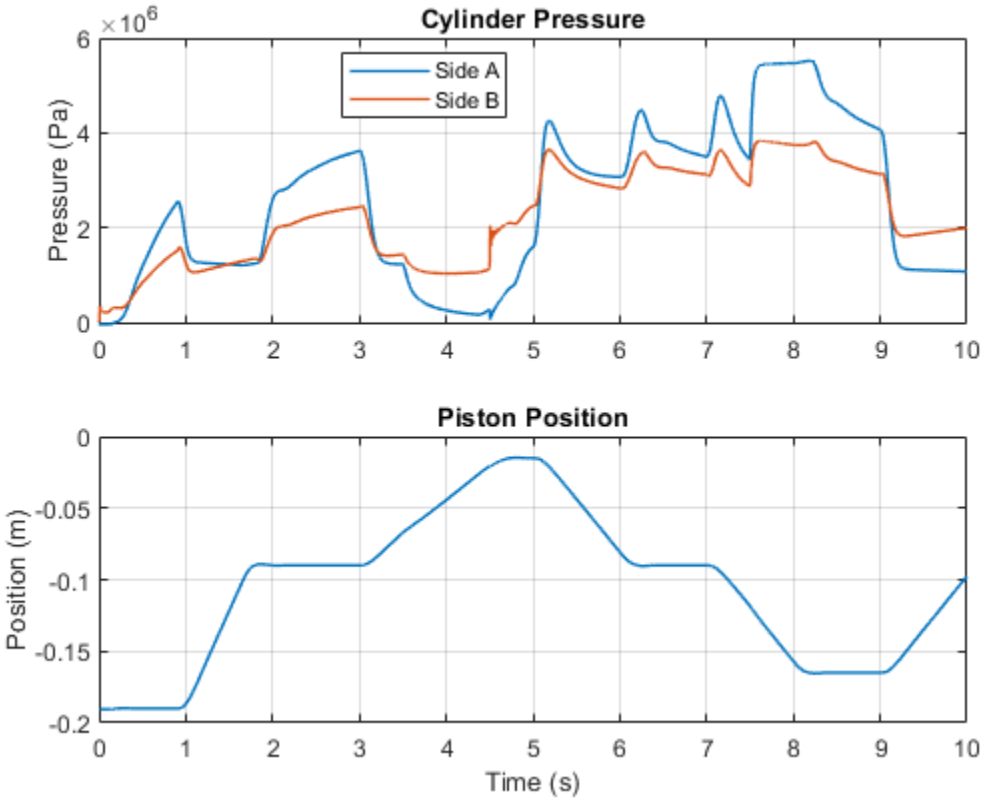
This example shows a closed-loop actuator that consists of a proportional 4-way directional valve driving a double-acting hydraulic cylinder. The cylinder drives a load consisting of a mass, viscous and Coulomb friction, constant force, and a spring. The actuator is powered by a variable-displacement, pressure-compensated pump, driven by a constant velocity motor. Pipelines between the valve, cylinder, pump, and the tank are modeled using the Hydraulic Pipeline blocks.

Model

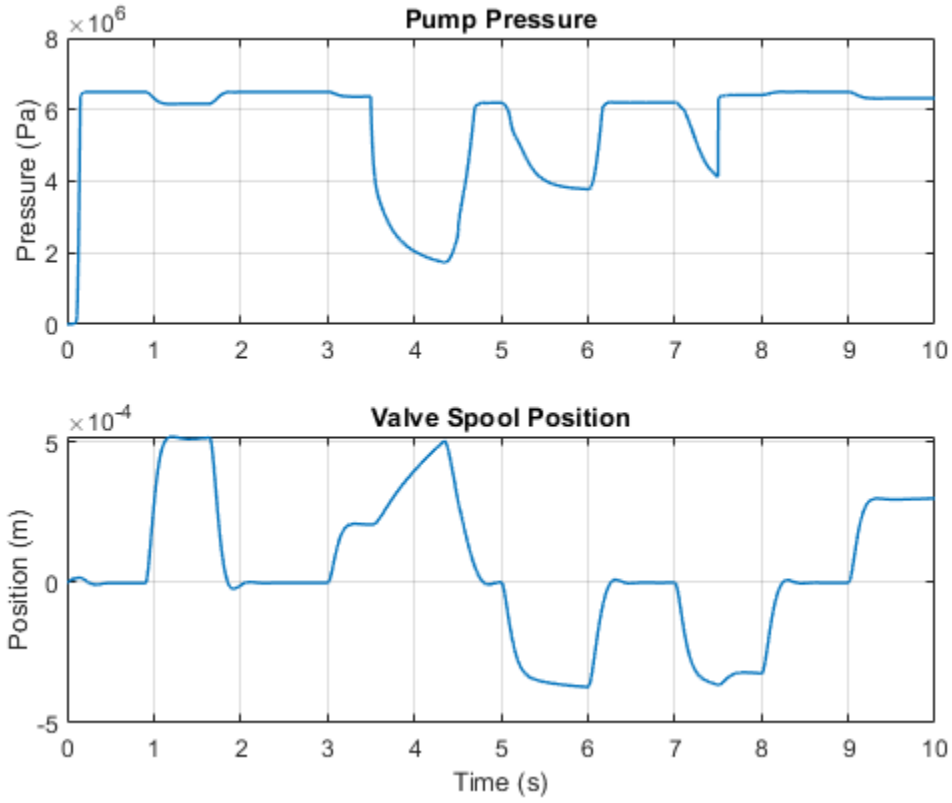


Simulation Results from Simscape Logging

The plots below show the pressures in the cylinder and the position of the piston.



The plots below show the pressure provided by the pump and the position of the spool in the 4-Way Valve. As the spool position changes the load on the pump increases, causing fluctuations in the pressure at the output of the pump.

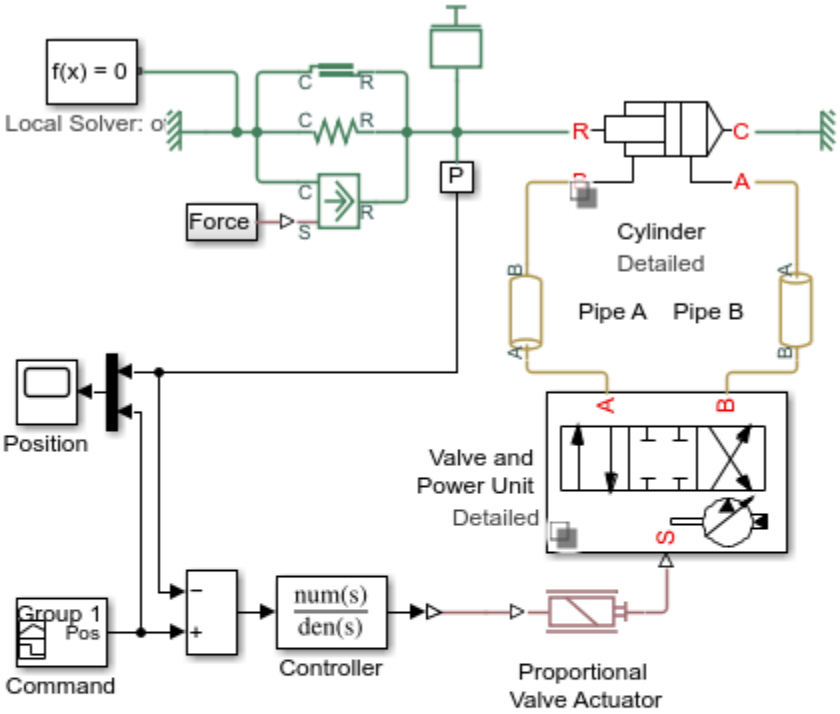

Pump Pressure
×10^6
Pressure (Pa)
8
6
4
2
0
0 1 2 3 4 5 6 7 8 9 10
Valve Spool Position
×10^-4
Position (m)
5
0
-5
0 1 2 3 4 5 6 7 8 9 10
Time (s)

Closed-Loop Hydraulic Actuator Model for Real-Time Simulation

The actuator consists of a proportional 4-way directional valve driving a double-acting hydraulic cylinder. The cylinder drives a load consisting of a mass, viscous and Coulomb friction, constant force, and a spring. The actuator is powered by a variable-displacement, pressure-compensated pump, driven by a constant velocity motor. Pipelines between the valve, cylinder, pump, and the tank are simulated with the Hydraulic Pipeline blocks.

Abstract variants of the hydraulic cylinder and of the valve and pump unit are also included in the model. The model can be configured to use these abstract variants, which neglect effects such as compressibility and use simpler equations that are numerically more efficient. The model can also be configured for simulation with a fixed-step solver, as would be used in hardware-in-the-loop testing.

Model

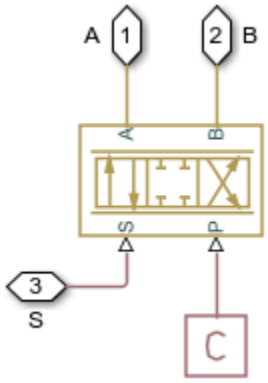


Closed-Loop Hydraulic Actuator Model for Real-Time Simulation

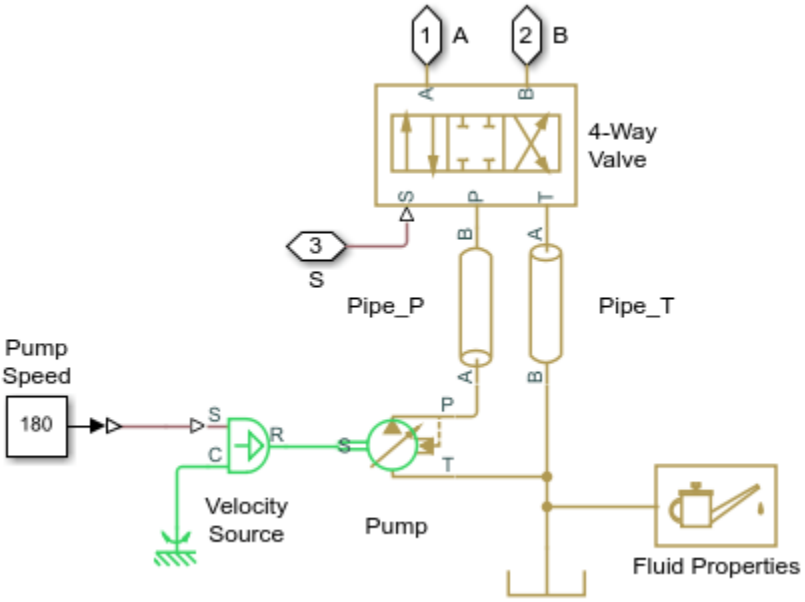
1. Plot pressure difference in cylinder chambers (see code)
2. Configure pump: Detailed, Abstract, (see code)
3. Configure cylinder: Detailed , Abstract (see code)
4. Configure solver: Variable-Step, Fixed Step (see code)
5. Compare results with abstract and detailed variants (see code)
6. Compare results with variable and fixed-step solvers (see code)
7. Explore simulation results using sscexplore
8. Learn more about this example

Copyright 2005-2021 The MathWorks, Inc.

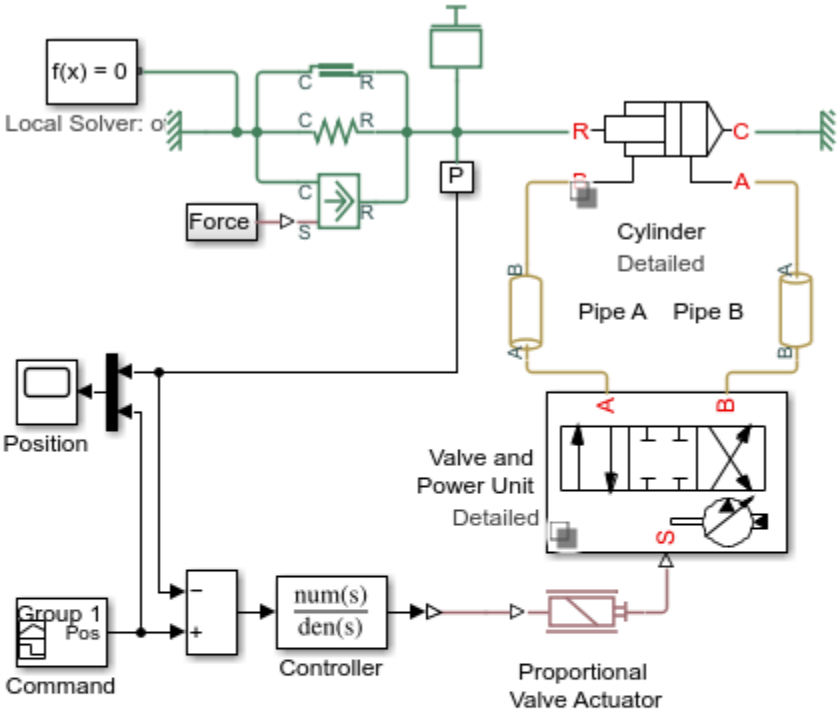
Valve and Power Unit, Abstract Variant



Valve and Power Unit, Detailed Variant



Simulation Results from Simscape Logging

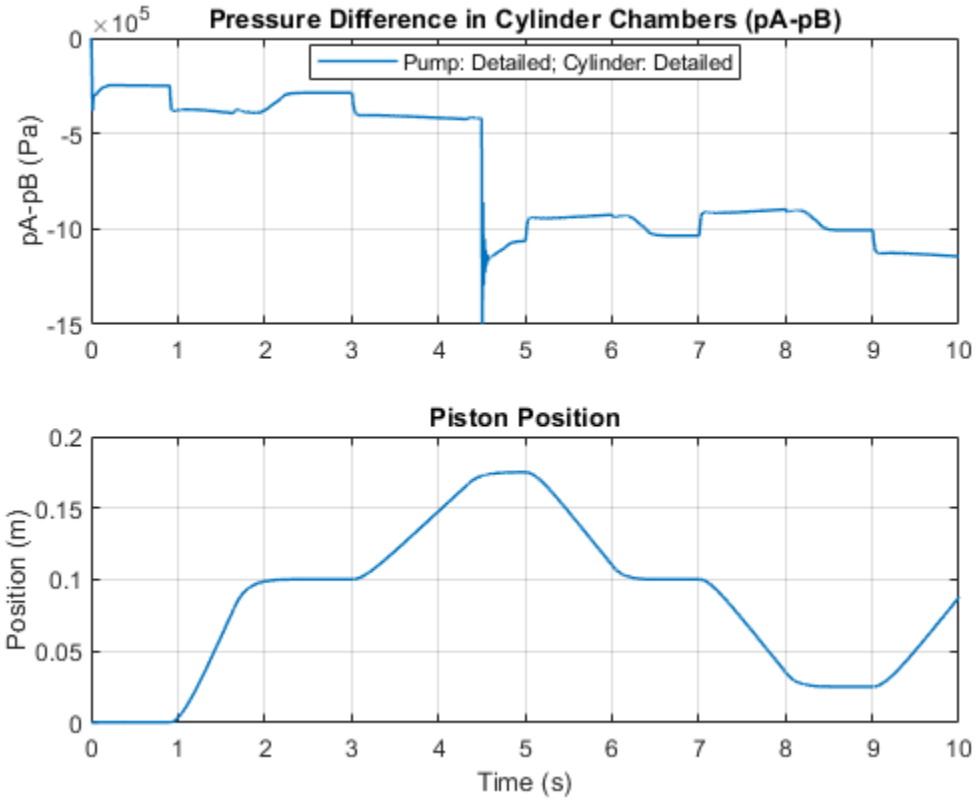


Closed-Loop Hydraulic Actuator Model for Real-Time Simulation

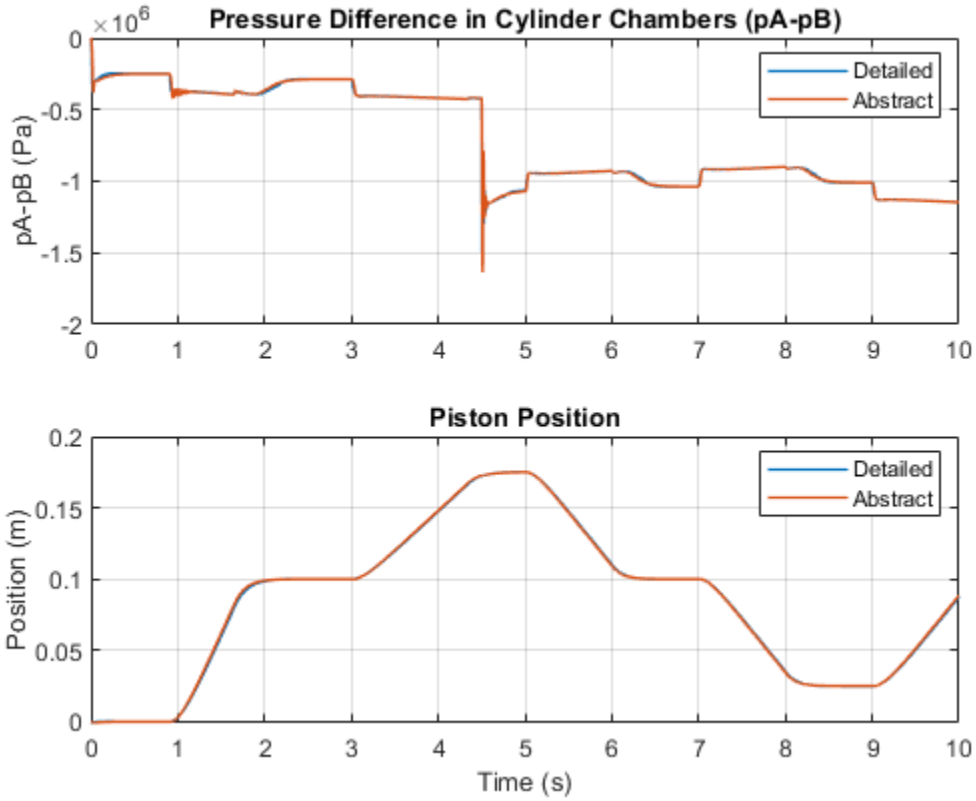
1. Plot pressure difference in cylinder chambers (see code)
2. Configure pump: Detailed, Abstract, (see code)
3. Configure cylinder: Detailed , Abstract (see code)
4. Configure solver: Variable-Step, Fixed Step (see code)
5. Compare results with abstract and detailed variants (see code)
6. Compare results with variable and fixed-step solvers (see code)
7. Explore simulation results using sscexplore
8. Learn more about this example

Copyright 2005-2021 The MathWorks, Inc.

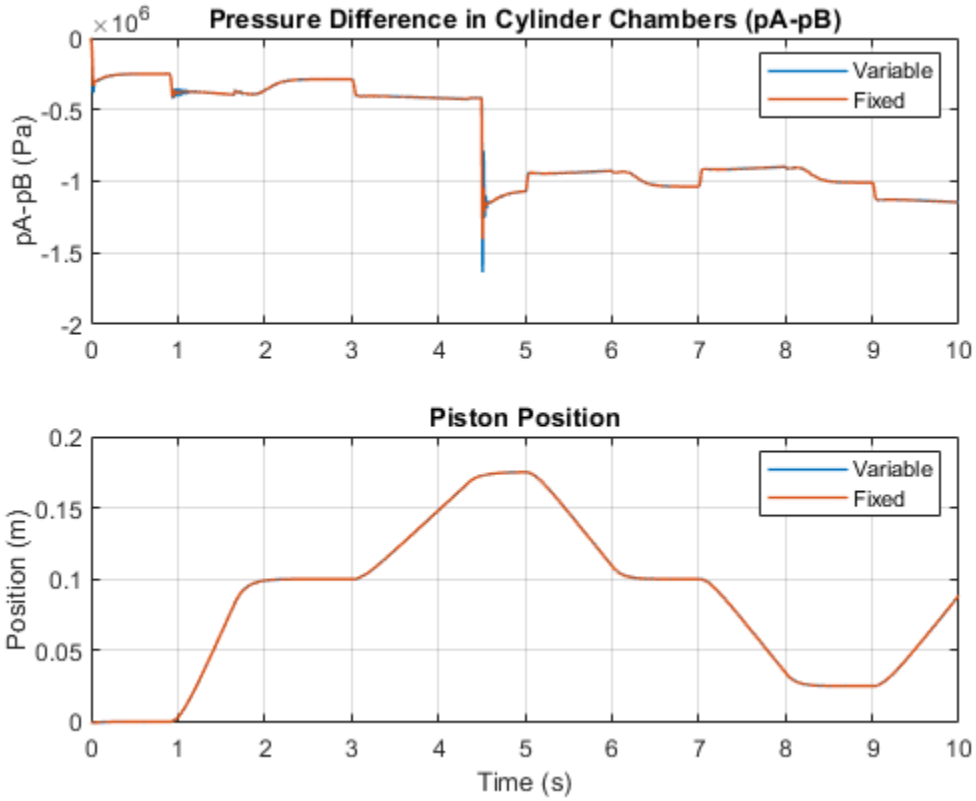
The plots below show the pressure difference between cylinder chambers A and B, as well as the position of the piston.



The plots below show the behavior of the abstract models and the detailed model. The results are very similar, indicating that the abstract models can be used without a significant loss of accuracy.



The plots below show the behavior during a simulation with a variable-step solver (Simscape™ local solver off) and a fixed-step solver (Simscape local solver on). The results are very similar, indicating that a fixed-step solver can be used without a significant loss of accuracy, and that the models can be used for hardware-in-the loop simulation.

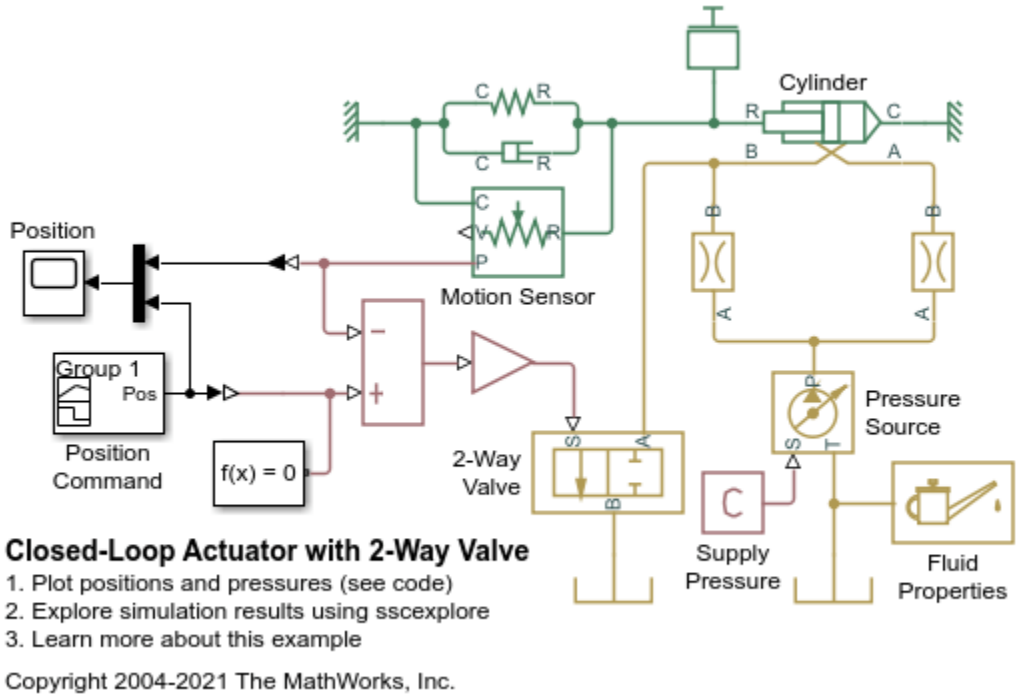

Pressure Difference in Cylinder Chambers (pA-pB)
×10^6
pA-pB (Pa)
Variable
Fixed
Piston Position
Position (m)
Time (s)

Closed-Loop Actuator with 2-Way Valve

This example shows the use of a 2-way valve in a closed-loop circuit together with a double-acting hydraulic cylinder, fixed orifice, mass, spring, damper, and control blocks.

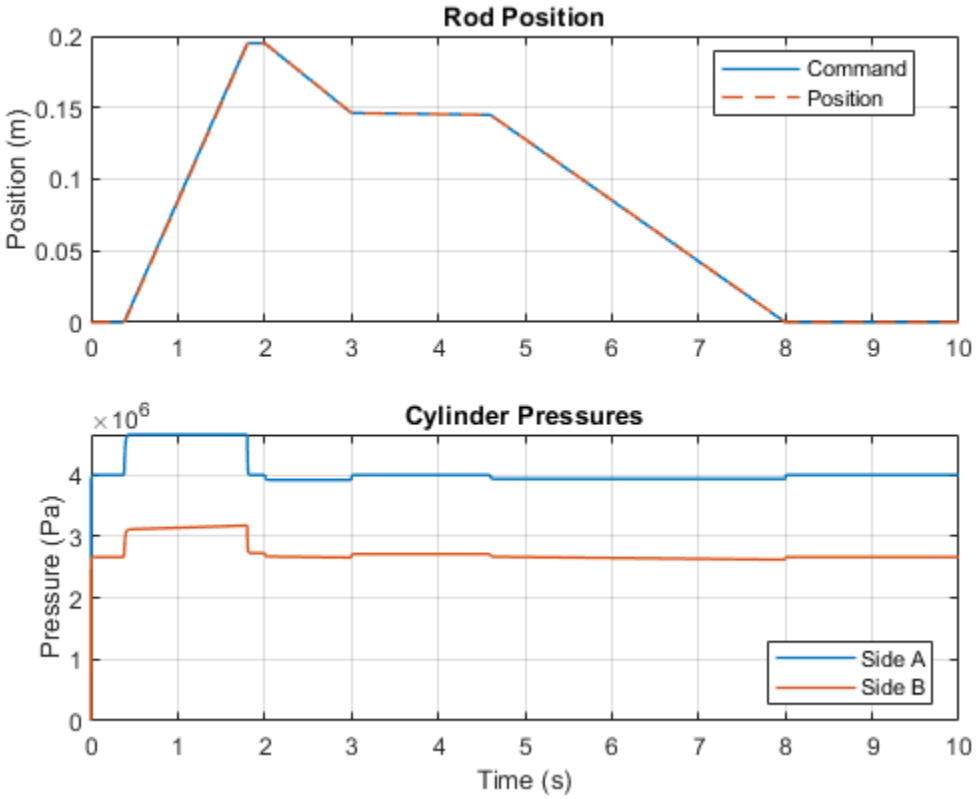
The valve is initially opened by 0.42 mm to assure that the cylinder force is close to zero at the start of simulation. The PS Gain block represents a lever through which a control signal is applied to the valve.

Model



Simulation Results from Simscape Logging

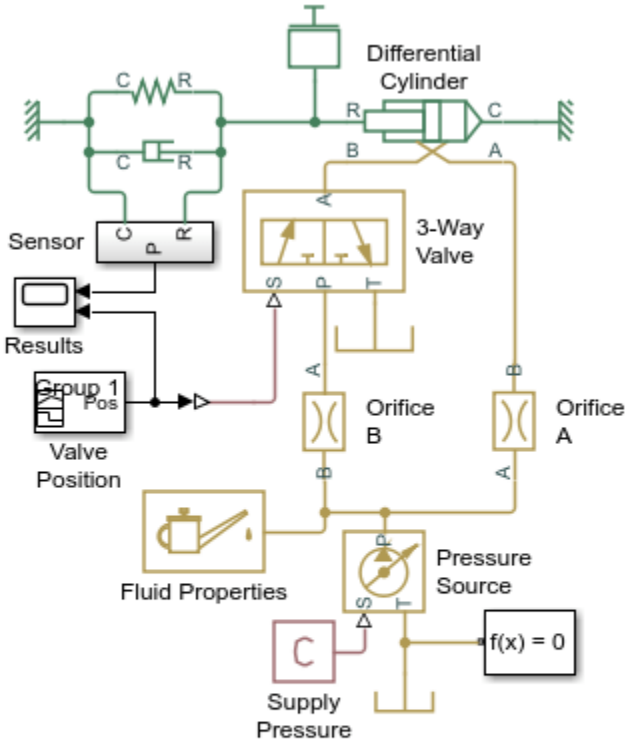
The plot below shows the measured and commanded position of the piston as well as the pressures in the hydraulic cylinder.


Rod Position
Position (m)
0.2
0.15
0.1
0.05
0
0 1 2 3 4 5 6 7 8 9 10
Command
Position
Cylinder Pressures
×10^6
Pressure (Pa)
4
3
2
1
0
0 1 2 3 4 5 6 7 8 9 10
Time (s)
Side A
Side B

Hydraulic Differential Cylinder with 3-Way Valve

This example shows the use of a 3-way valve to control the motion of a hydraulic differential cylinder. The valve connects cylinder port B either to tank or to cylinder chamber A through the fixed orifice. The cylinder extends when its ports are both connected to the pressure source due to difference in piston areas. Orifice B controls the extension speed. The cylinder retracts if chamber B is connected to tank.

Model



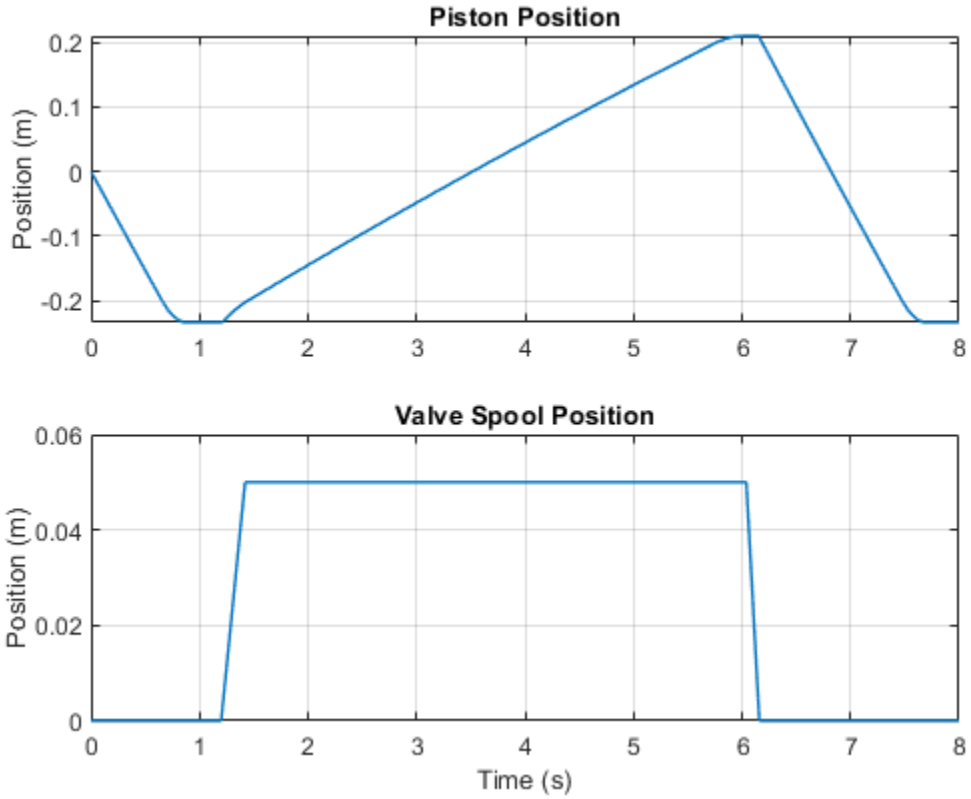
Hydraulic Differential Cylinder with 3-Way Valve

1. Plot positions (see code)
2. Plot pressures in cylinder (see code)
3. Explore simulation results using sscexplore
4. Learn more about this example

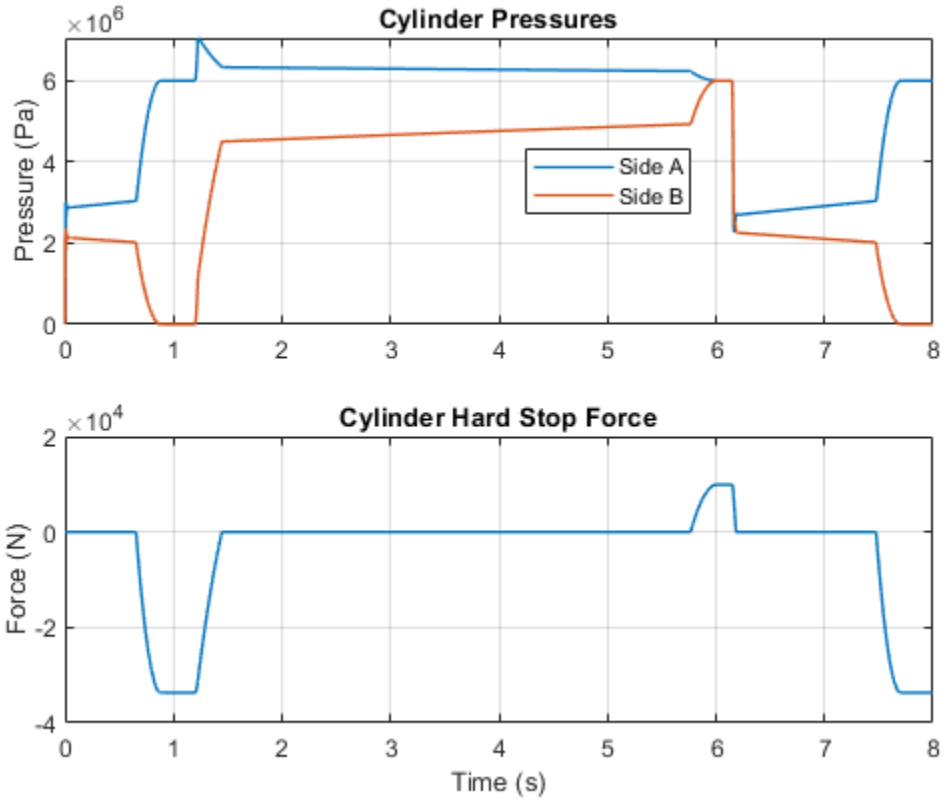
Copyright 2004-2021 The MathWorks, Inc.

Simulation Results from Simscape Logging

This plot shows the position of the piston (system output) and the position of the valve spool (system input).



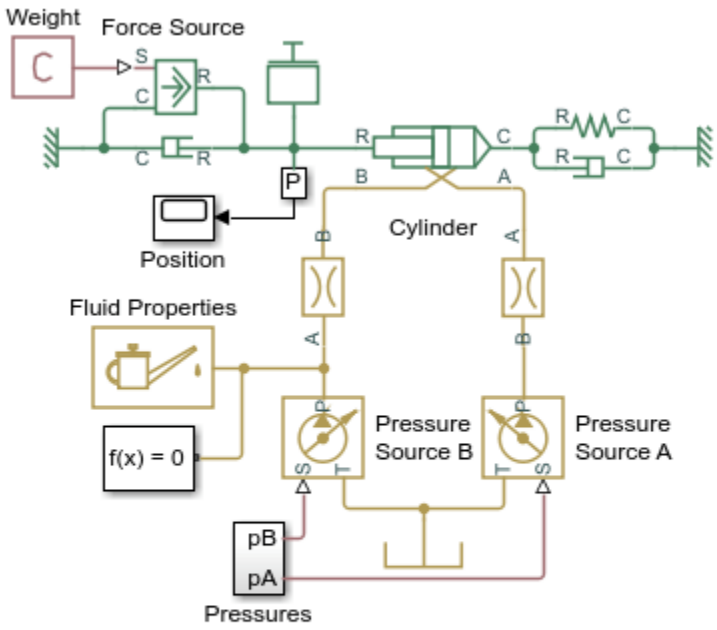
These plots show the pressure inside the cylinder, as well as the force between the piston and the hard stop inside the cylinder.


Cylinder Pressures
×10^6
Pressure (Pa)
Side A
Side B
Cylinder Hard Stop Force
×10^4
Force (N)
Time (s)

Hydraulic Cylinder with Flexible Mount

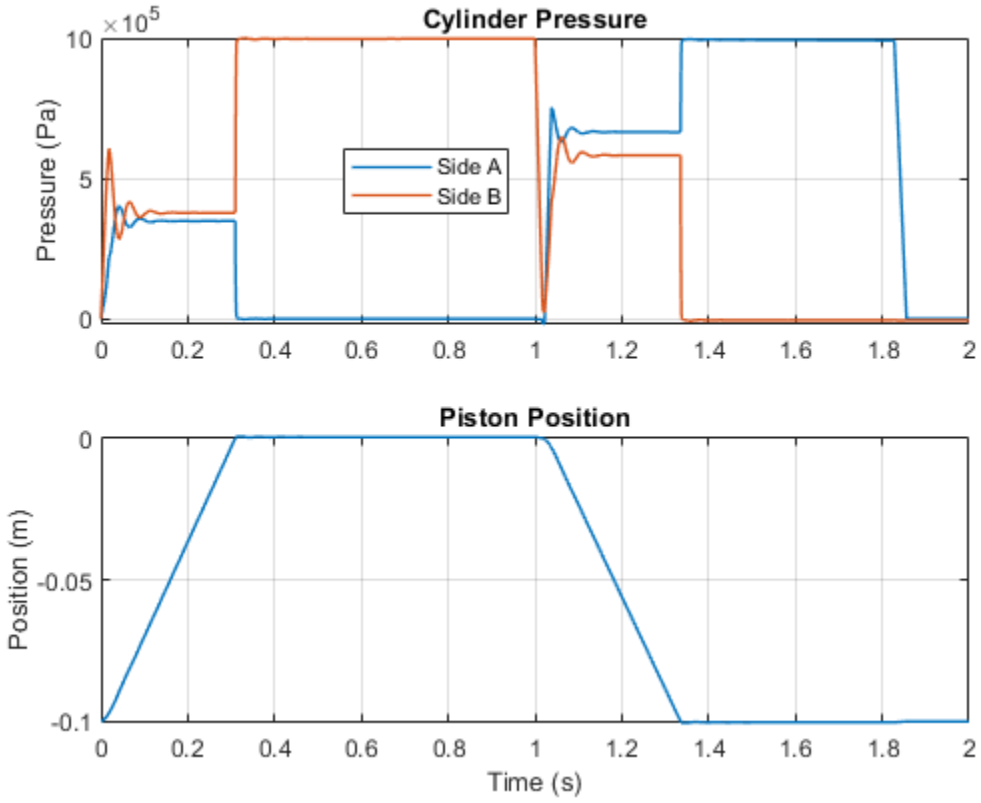
This example shows the use of a double-acting hydraulic cylinder. The clamping structure is simulated with the spring and damper installed between the cylinder case and the reference. The cylinder performs forward and return strokes caused by the change of pressures at ports A and B. The cylinder load consists of inertia, viscous friction, and constant opposing load of 400 N.

Model



Simulation Results from Simscape Logging

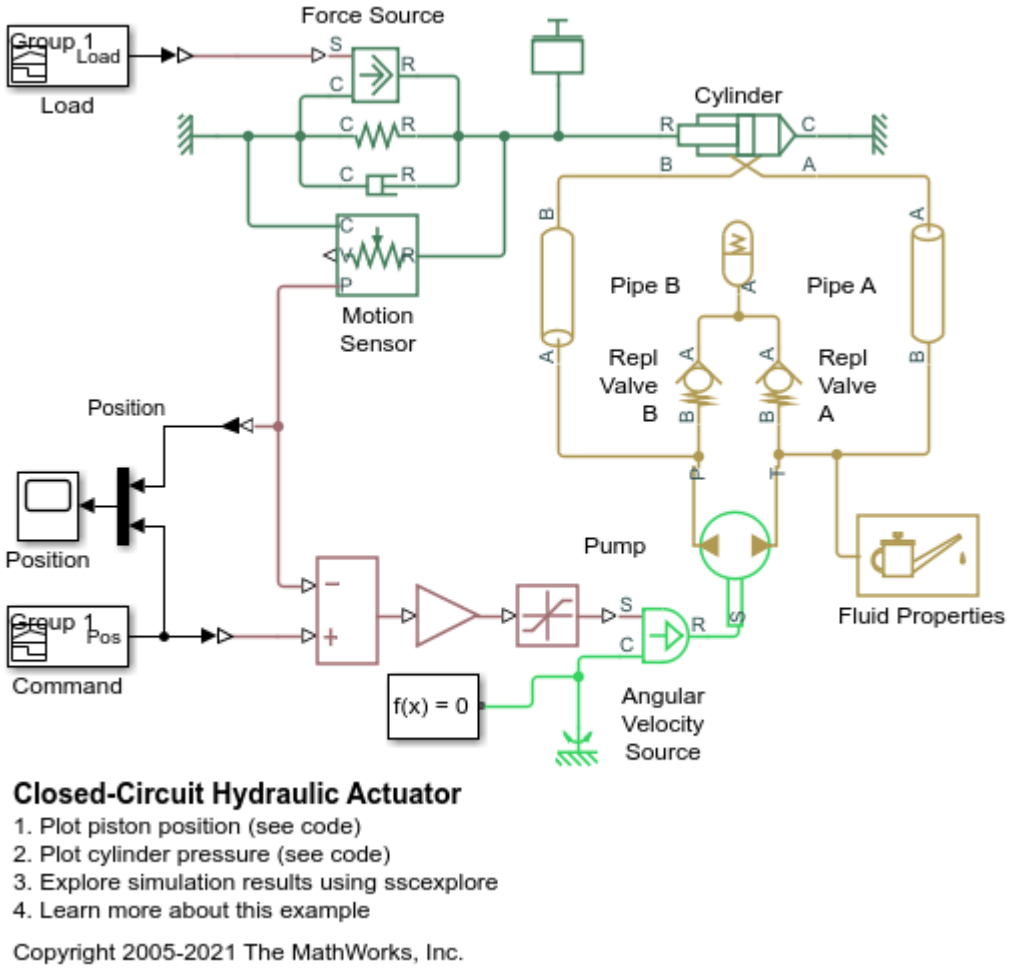
The plots below show the pressures in the cylinder and the position of the piston.


Cylinder Pressure
×10^5
10
5
0
Pressure (Pa)
Side A
Side B
0
0.2
0.4
0.6
0.8
1
1.2
1.4
1.6
1.8
2
Piston Position
0
-0.05
-0.1
Position (m)
Time (s)

Closed-Circuit Hydraulic Actuator

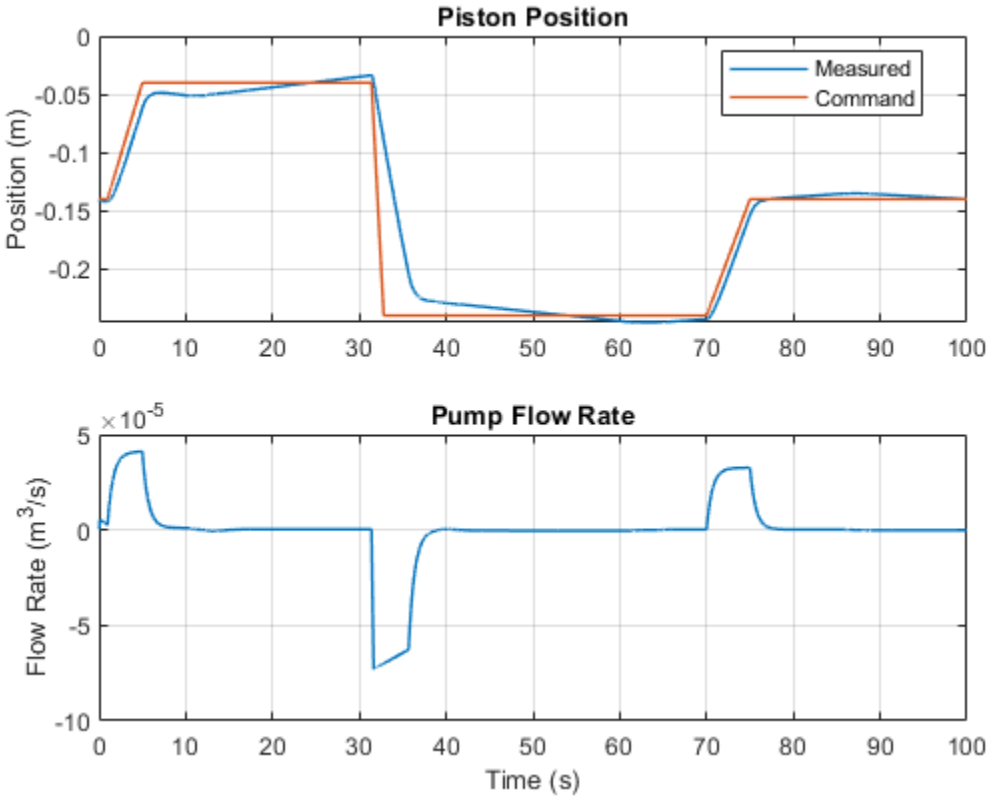
This example shows a closed-circuit hydraulic actuator driven by a variable-speed motor. The actuator is arranged as a closed system with two replenishment valves (check valves) and a spring accumulator serving as a replenishment reservoir. The motor velocity is controlled by the difference between the commanded and actual actuator position. The actuator acts against a spring, damper, and a time-varying load.

Model

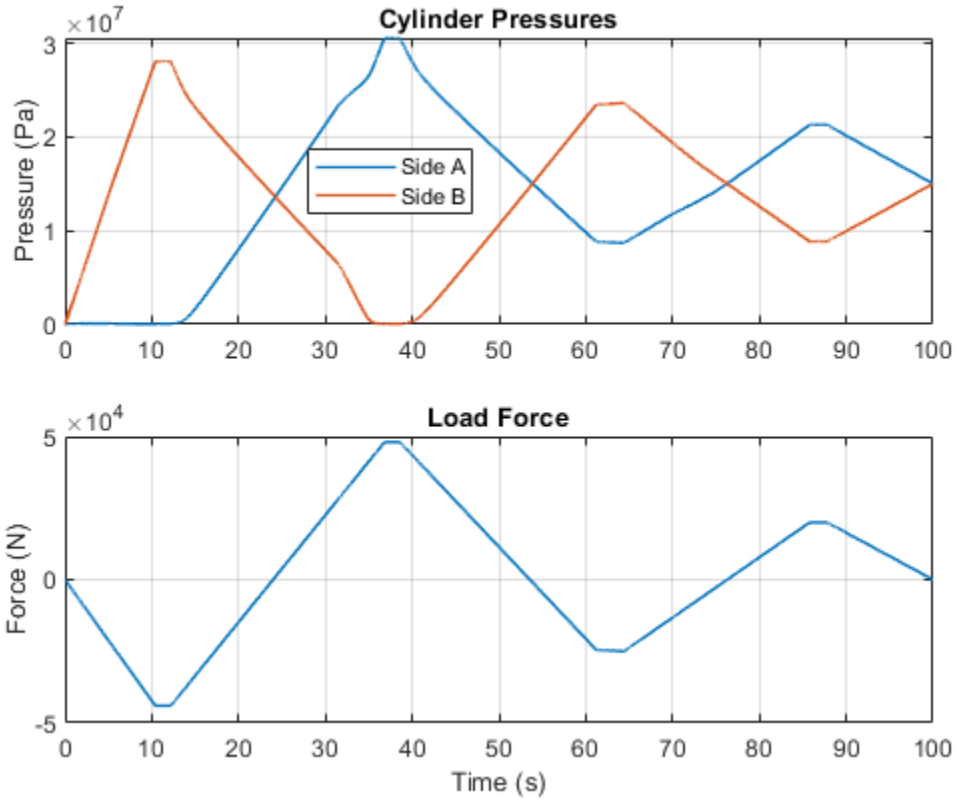


Simulation Results from Simscape Logging

This plot shows the position of the piston and the flow rate through the pump.



This plot shows the pressures inside the cylinder and the time-varying load.


Cylinder Pressures
×10^7
Pressure (Pa)
Side A
Side B
Load Force
×10^4
Force (N)
Time (s)

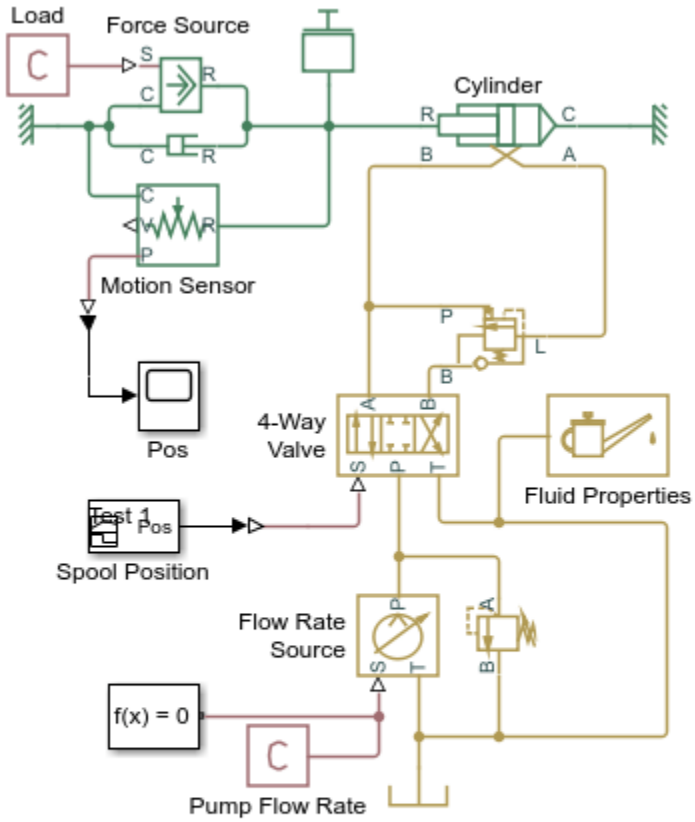
Hydraulic Actuator with Counterbalance Valve

The model consists of a double-acting hydraulic cylinder controlled by an open-center 4-way directional valve and a power unit built of a flow rate source and a pressure-relief valve. The cylinder is loaded with an active force in 10000 N trying to extend the rod. In other words, the cylinder experiences an overriding load while extending. To hold the rod in place while the directional valve is in the neutral position and prevent losing control of the cylinder extension, the counterbalance valve is used. The valve controls the return flow from port A. The valve pilot port P is connected to cylinder port B, thus making it impossible for the cylinder to move until pressure at port B builds up to certain level.

To hold the load, the counterbalance valve must be set to 1.3 * 10000/ 5e-3 = 26e5 Pa, where 1.3 is the safety coefficient, 10000 N is the load, 5e-5 m^2 is the effective piston area in the rod chamber. The cylinder starts moving as the pilot pressure builds up to approximately p = (p_s - p_L) / (k_p + k_c), where p_s is the valve setting pressure, p_L is the load pressure (p_L = L/A_r), L is the load, A_r is the rod chamber effective area (piston side B), k_p is the pilot ratio (k_p is set to 3 in the model), and k_c is the ratio between the cylinder blank chamber area to the rod chamber area ((piston area side B/piston area side A) is set to 2 in the model). As a result, the load starts moving at p = (26e5 - 100000/5e-3) / (3 + 2) = 1.2e5 Pa.

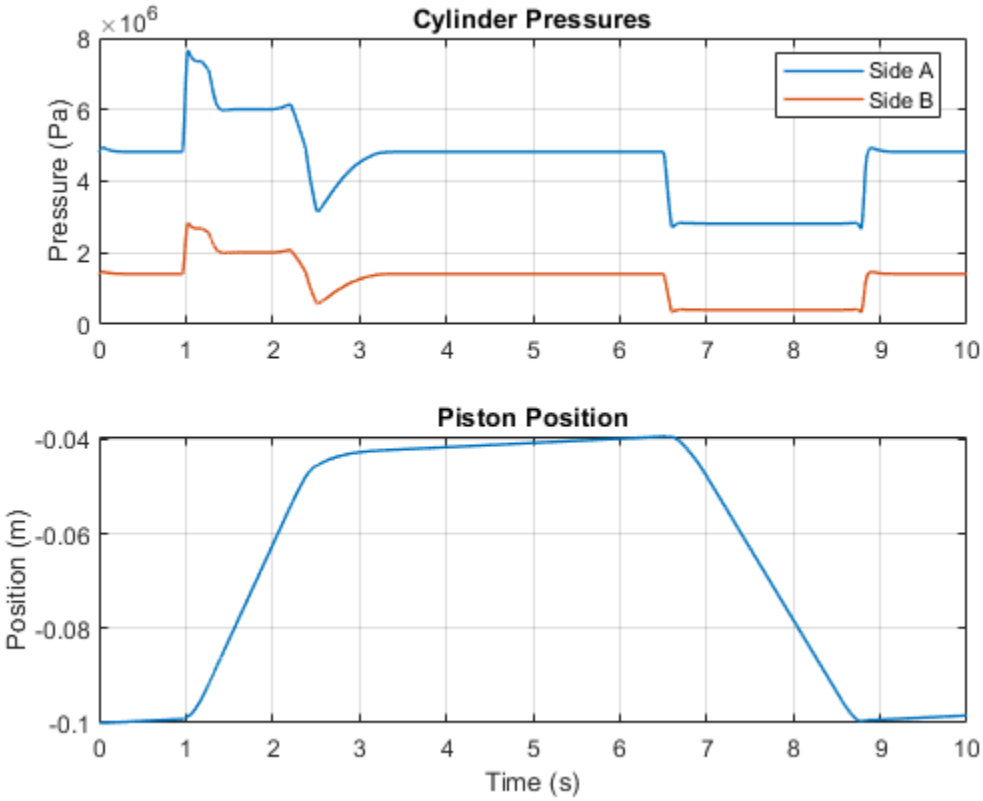
The system is simulated during a 10 second cycle, within which the rod extends by about 0.06 m and then returns to initial position. The counterbalance valve maintains enough pressure in the cylinder chamber to balance the load while the directional valve is in the neutral position.

Model

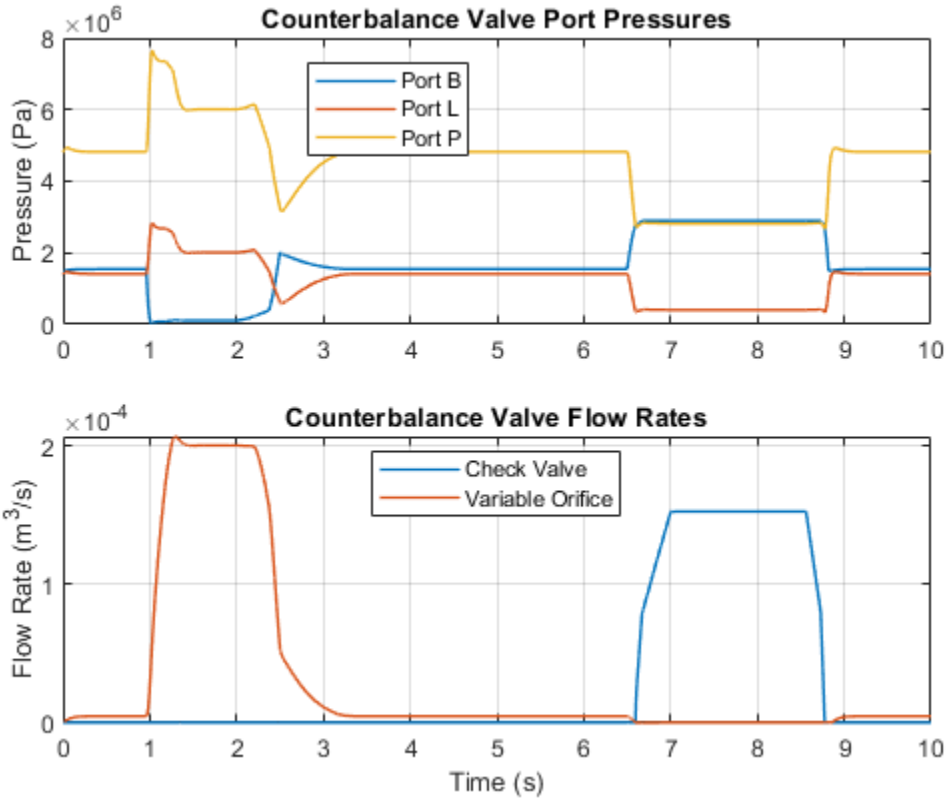


Simulation Results from Simscape Logging

The plots below show the pressures in the cylinder and the position of the piston.



The plots below show the port pressures and flow rates for the counterbalance valve. During the first part of the cycle, flow rate through the variable orifice is observed, indicating that the pressures at ports P and L create enough force to overcome the spring force and open the valve. In the second part of the cycle, flow is observed through the check valve in the counterbalance valve.


Counterbalance Valve Port Pressures
×10^6
Pressure (Pa)
Port B
Port L
Port P
Counterbalance Valve Flow Rates
×10^-4
Flow Rate (m^3/s)
Check Valve
Variable Orifice
Time (s)

Hydraulic Actuator with Dual Counterbalance Valves

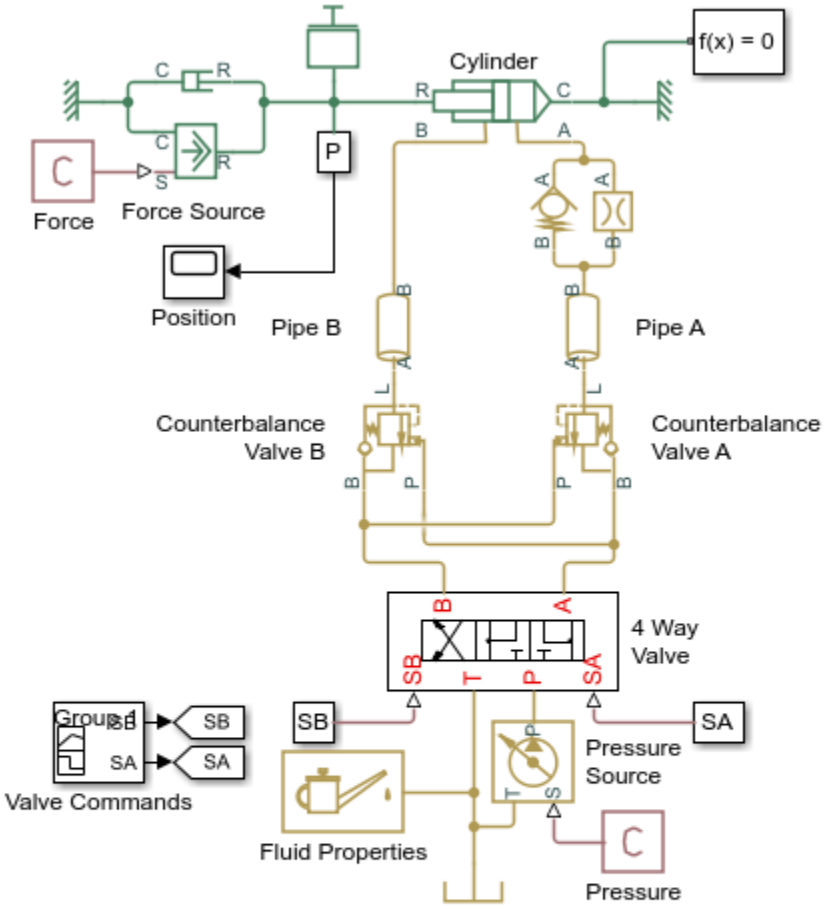
This example shows an actuator built of a double-acting hydraulic cylinder, directional valve, flow control, block of counterbalance valves, power unit, and a control unit.

The cylinder is loaded with an overriding load, which requires the use of counterbalance valves to prevent the load from creeping when the directional valve is in the neutral position. The actuator is equipped with a custom model of a 3-position, 4-way directional valve. The valve connects actuator ports A and B to tank in neutral position, while blocking the pressure port P. Ports A and B are connected to port P if electromagnet A is energized. This causes the cylinder to extend due to the difference in the cylinder effective areas. The piston retracts as electromagnet B is energized.

The counterbalance valves allows free flow from port B to port L through the check valve, but blocks the flow in the opposite direction unless pressure is applied to port P. When sufficient pressure is applied, a variable orifice opens.

The simulated actuator cycle starts with the piston rod at initial position. At 0.65 seconds electromagnet A is energized, causing the piston to extend until it reaches the end of the stroke. On its way forward, both bypass orifices are open and the cylinder differential connection is used to utilize the difference between piston areas. At 11.7 seconds electromagnet B is energized and the piston retracts to its initial position.

Model

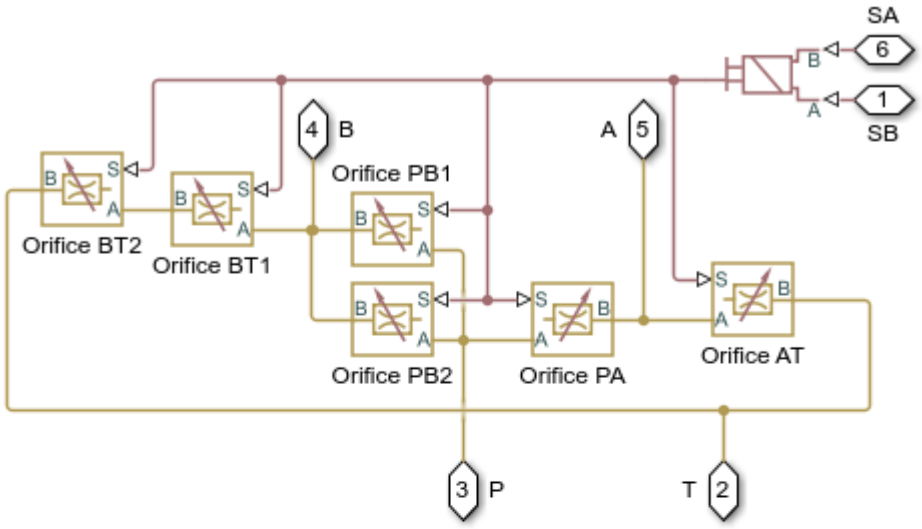


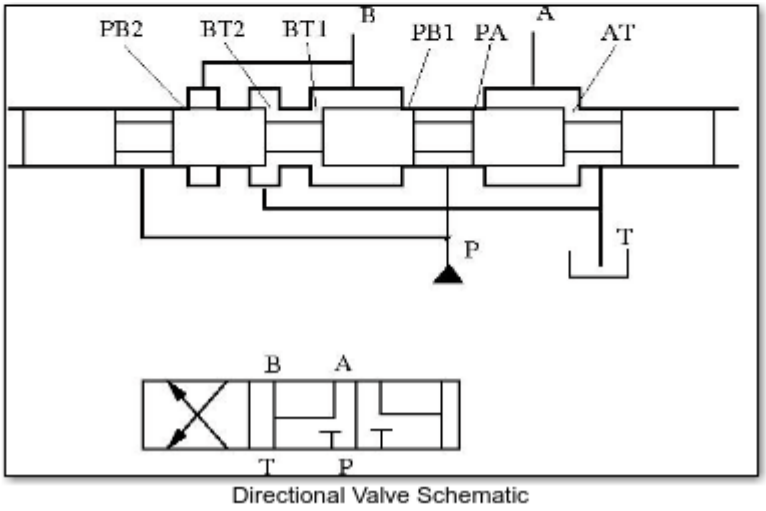
Hydraulic Actuator with Dual Counterbalance Valves

1. Plot pressures in cylinder (see code)
2. Plot pressures in counterbalance valve A (see code)
3. Plot pressures in counterbalance valve B (see code)
4. Explore simulation results using sscexplore
5. Learn more about this example

Copyright 2006-2021 The MathWorks, Inc.

4 Way Valve Subsystem

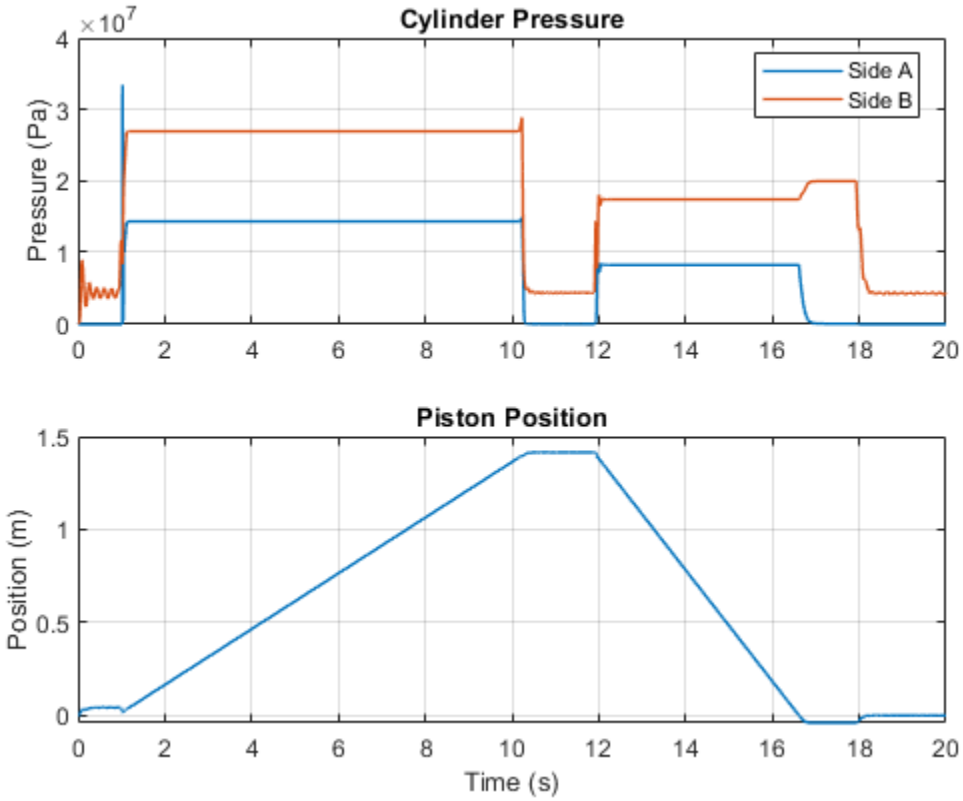




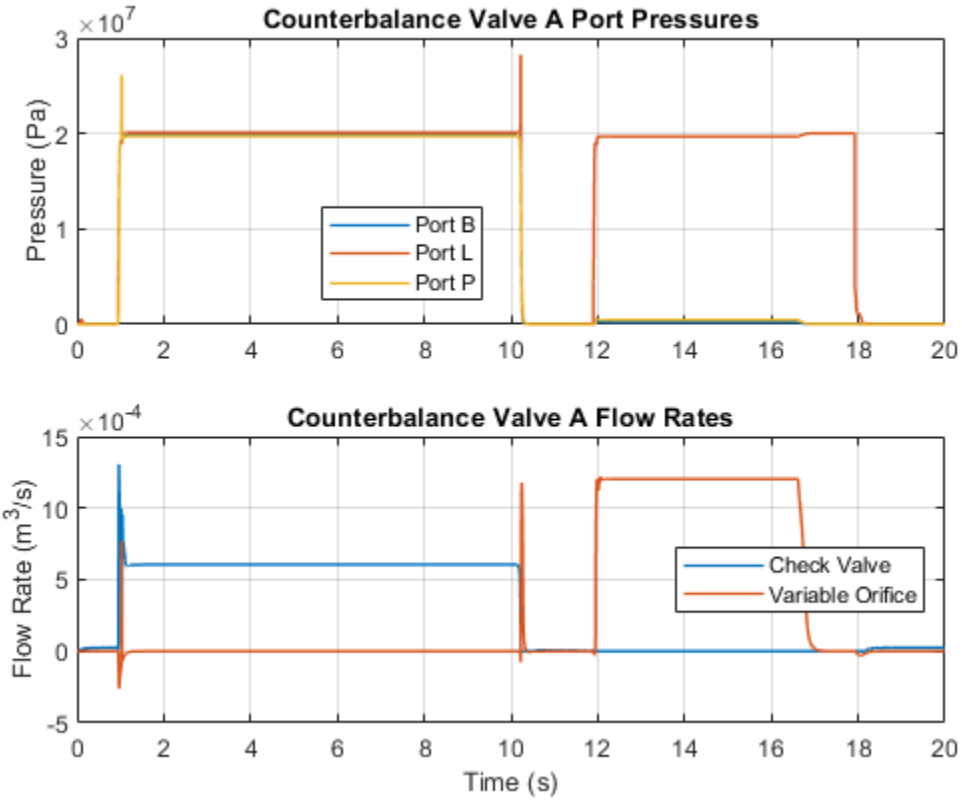
Directional Valve Schematic

Simulation Results from Simscape Logging

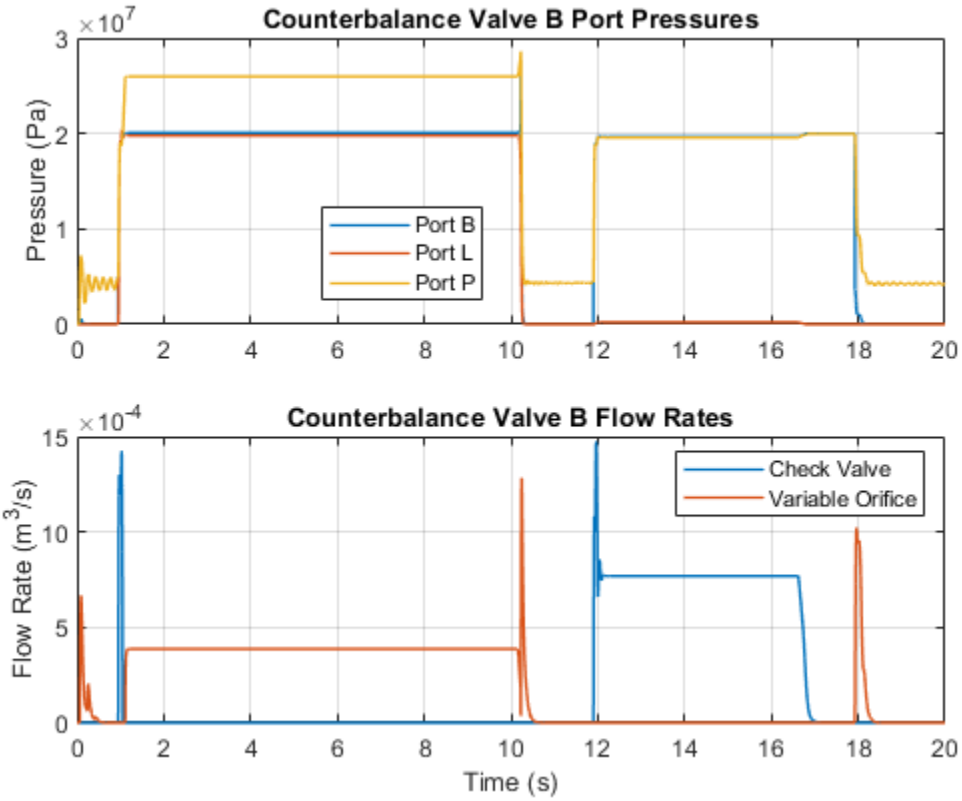
The plots below show the pressures in the cylinder and the position of the piston.



The plots below show the port pressures and flow rates for counterbalance valve A. In the first part of the cycle, flow is observed through the check valve in the counterbalance valve. During the second part of the cycle, flow rate through the variable orifice is observed, indicating that the ports at P and L create enough force to overcome the spring force and open the valve.



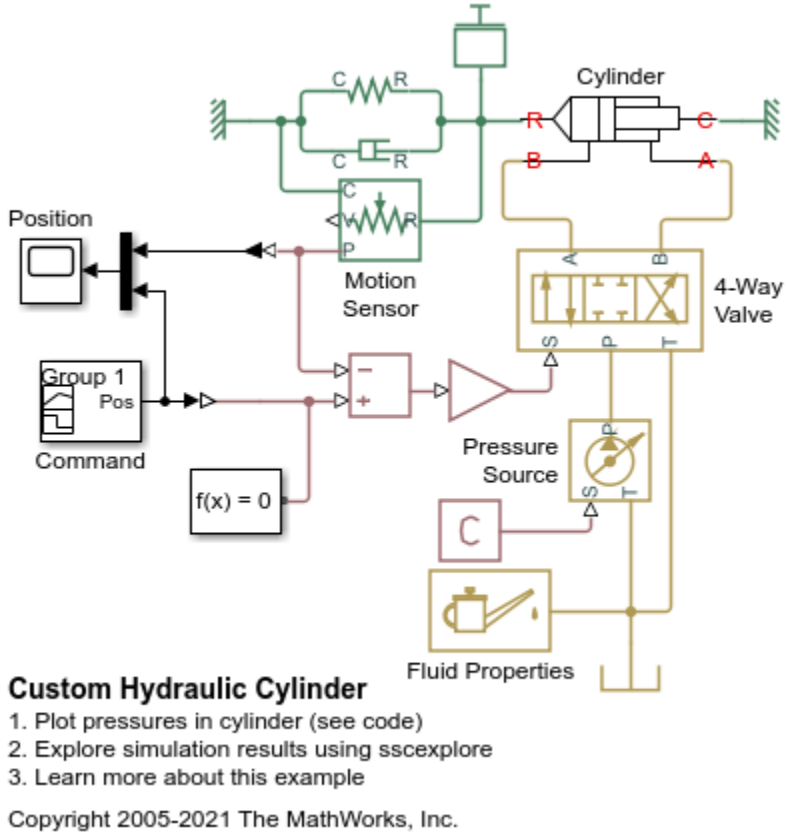
The plots below show the port pressures and flow rates for counterbalance valve B. During the first part of the cycle, flow rate through the variable orifice is observed, indicating that the ports at P and L create enough force to overcome the spring force and open the valve. In the second part of the cycle, flow is observed through the check valve in the counterbalance valve.


Counterbalance Valve B Port Pressures
×10^7
Pressure (Pa)
Port B
Port L
Port P
Counterbalance Valve B Flow Rates
×10^-4
Flow Rate (m^3/s)
Check Valve
Variable Orifice
Time (s)

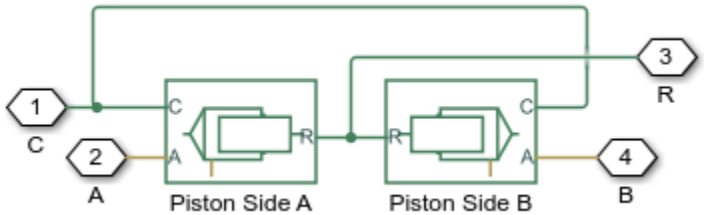
Custom Hydraulic Cylinder

This example shows a custom hydraulic cylinder model that omits the effect of fluid compressibility in the cylinder chambers. The double-acting cylinder connected to a 4-way valve to model a simple closed-loop actuator.

Model

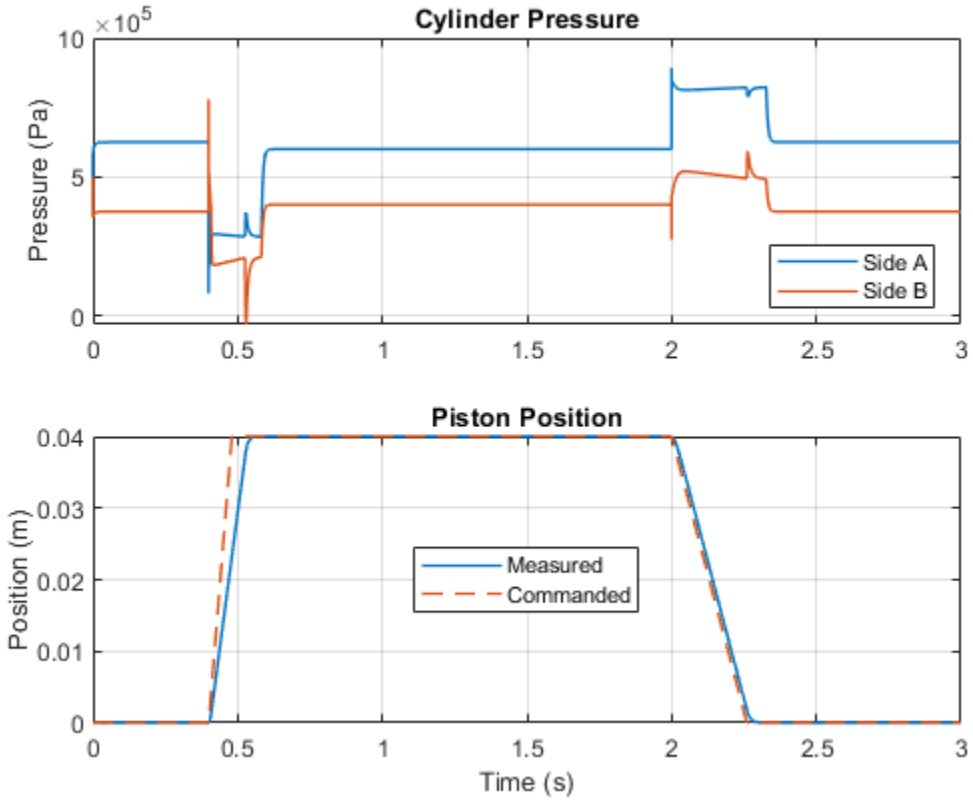


Cylinder Subsystem



Simulation Results from Simscape Logging

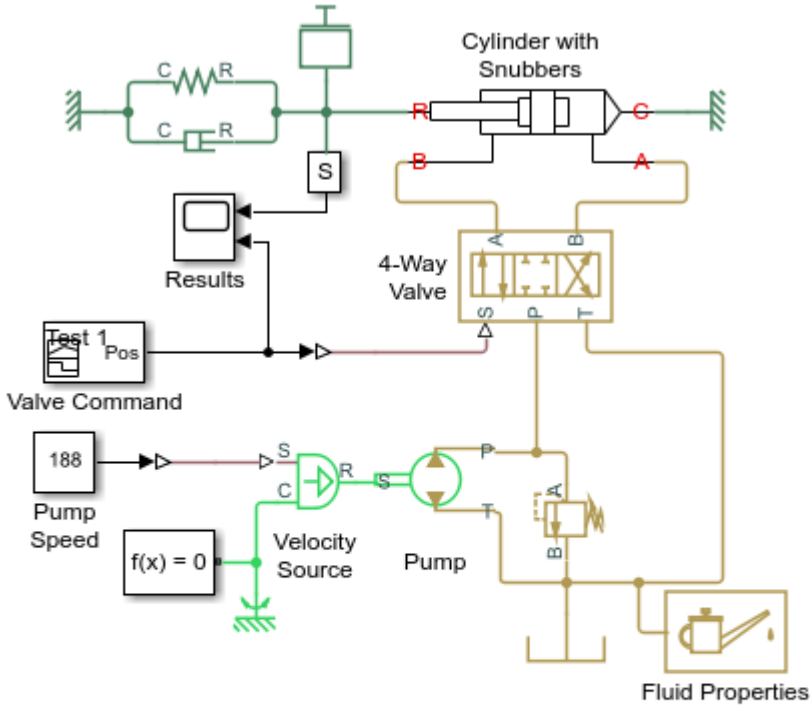
The plots below show the pressures in the cylinder and the position of the piston.


Cylinder Pressure
×10^5
10
5
0
Pressure (Pa)
0
0.5
1
1.5
2
2.5
3
Side A
Side B
Piston Position
0.04
0.03
0.02
0.01
0
Position (m)
Measured
Commanded
Time (s)

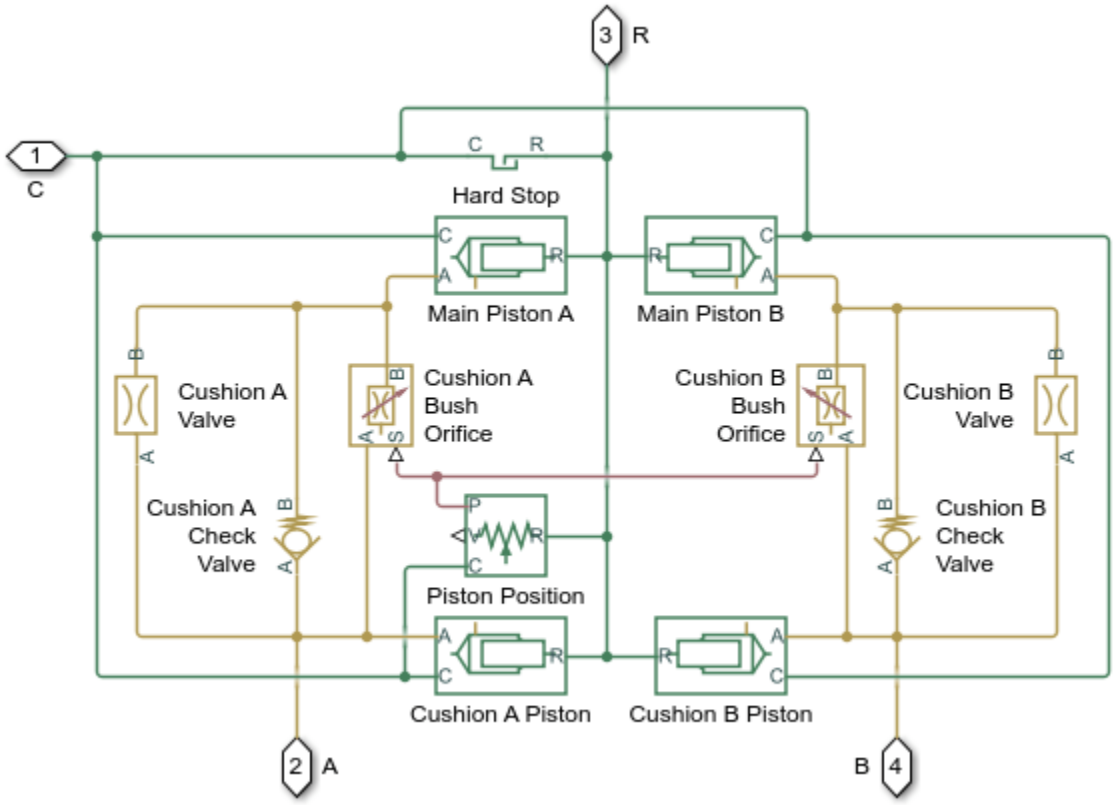
Hydraulic Cylinder with Two-Chamber Snubbers

This example shows a hydraulic cylinder equipped with a custom model of snubbers (cushions) on both sides of the cylinder. The snubbers are implemented as two-chamber cushions separated by the cushioning bush as the piston comes close to the end of the stroke. The cushioning provides hydraulic braking as the cylinder reaches the end of its stroke, absorbing some of the kinetic energy of the system before the piston reaches the end stop.

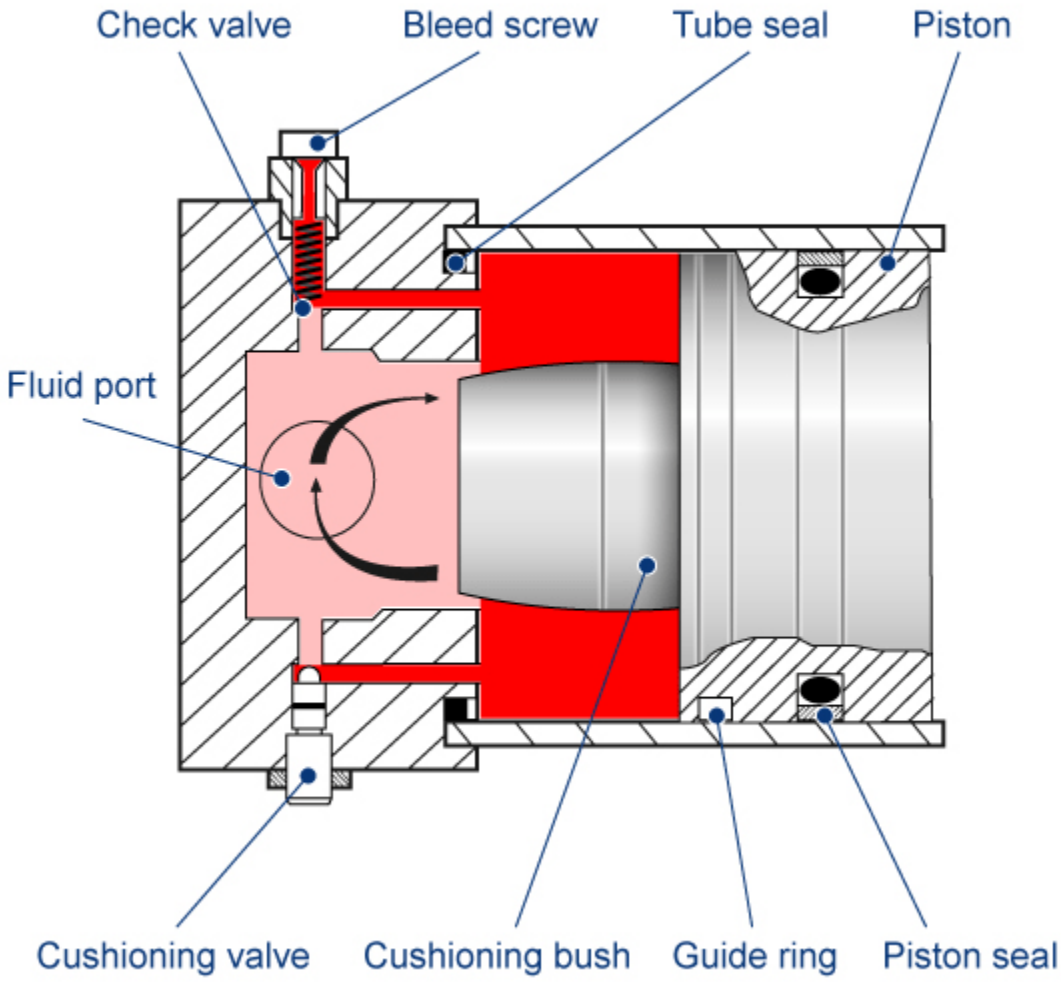
Model



Cylinder with Snubbers Subsystem

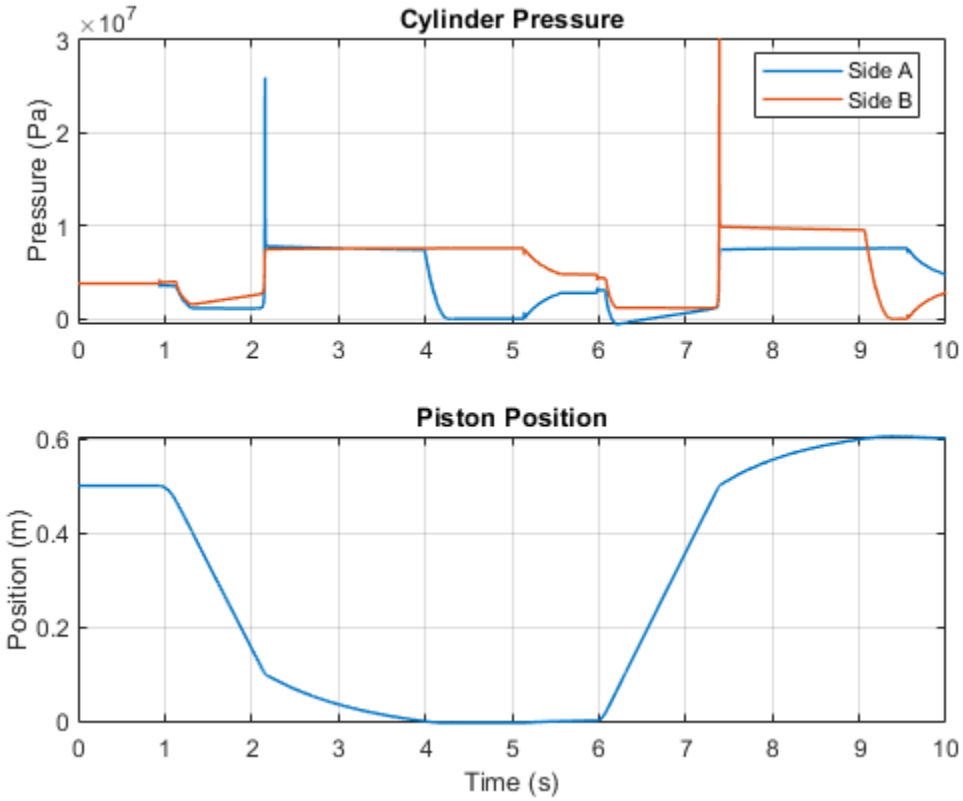


The following figure shows the schematic diagram of one end of the hydraulic cylinder with the snubber passages:

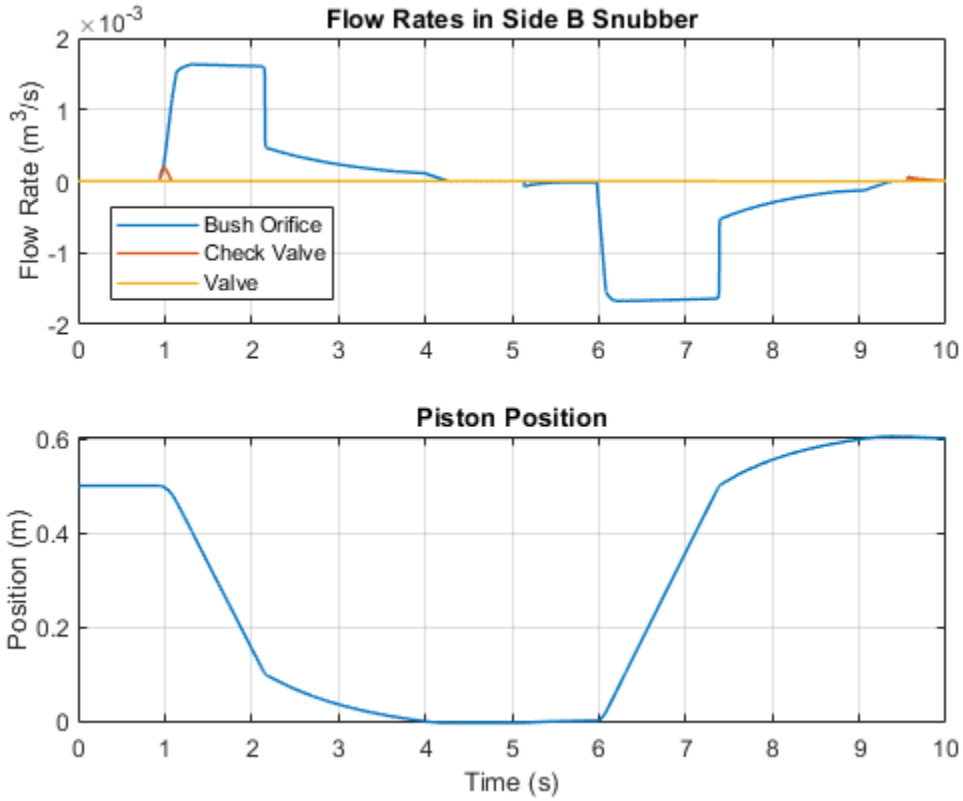


Simulation Results from Simscape Logging

The plots below show the pressures in the cylinder and the position of the piston.



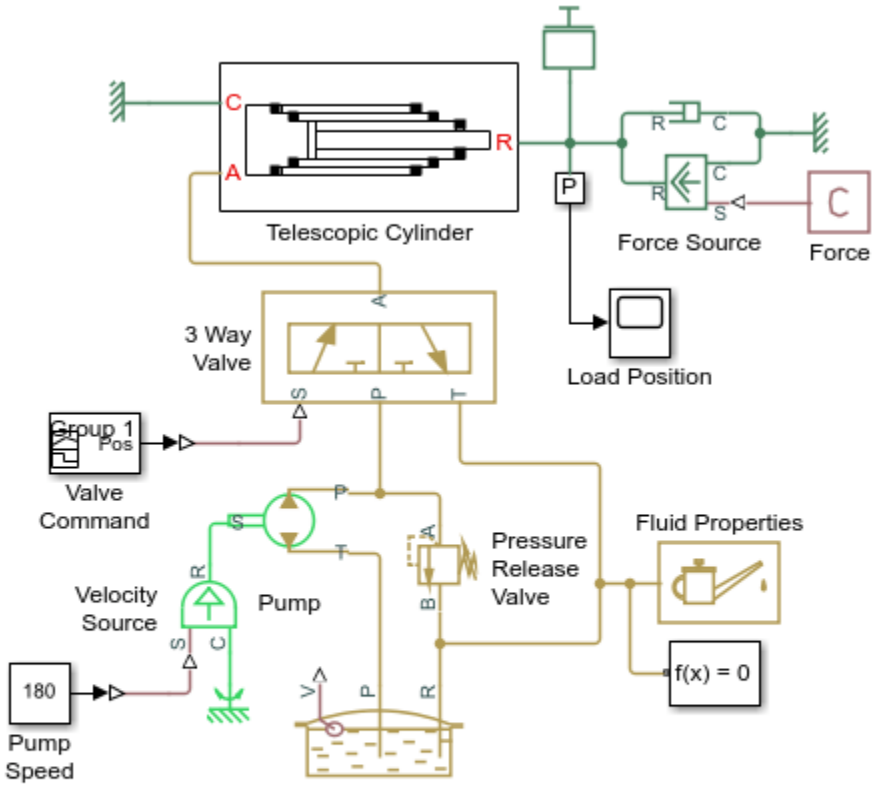
The plots below show the flow rate in the fluid passages that make up the snubber on side B of the cylinder. As the four-way valve connects the pump to side B, the check valve opens to permit the hydraulic fluid to flow out of the snubber and into the main chamber. The piston moves smoothly away from the hard stop, opening a larger passage between the fluid port and the chamber and the check valve closes. During the second half of the cycle, a very small amount of fluid flow passes through the cushion valve, allowing the piston to coast more smoothly into the hard stop.


Flow Rates in Side B Snubber
×10^-3
Flow Rate (m^3/s)
Bush Orifice
Check Valve
Valve
Piston Position
Position (m)
Time (s)

Hydraulic Actuator with Telescopic Cylinder

This example shows an actuator built around a telescopic hydraulic cylinder, which is equipped with three rods interacting with each other through hard stops. The effective areas of the rods are set to 20, 16, and 12 cm^2 respectively, which causes the first rod to move first, followed by the second, and then by the third rod. The actuator is controlled by a 2-position 3-way valve, which connects the actuator chamber to the tank at neutral position. As the control signal is applied to the valve, the actuator chamber is connected to the pump and the actuator starts extending. The rods are retracted by external load when the chamber is connected to the tank. The power unit relief valve is set to 50 bar, and maximum load to be handled by the actuator is 2500 N.

Model

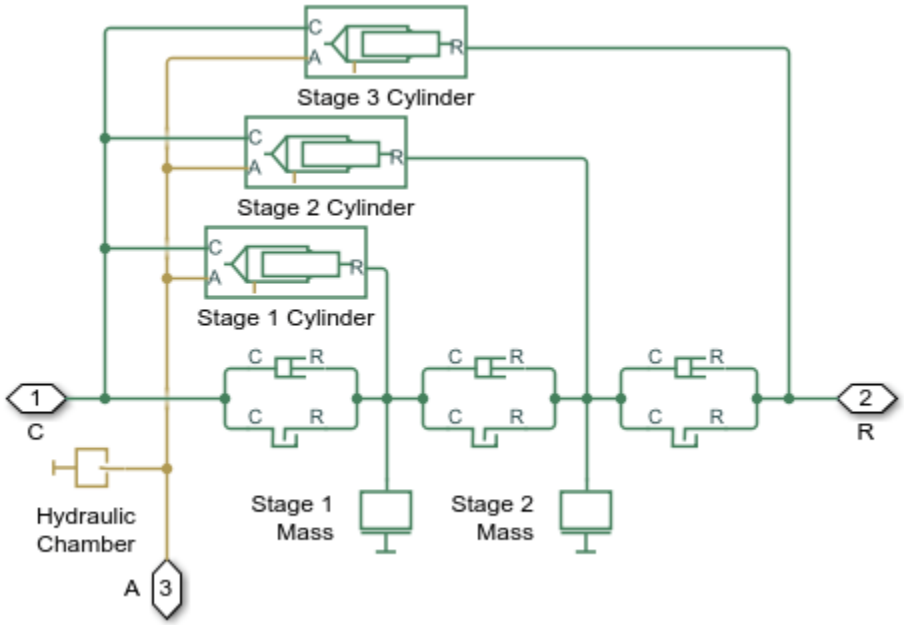


Hydraulic Actuator with Telescopic Cylinder

1. Plot pressure in telescopic cylinder (see code)
2. Explore simulation results using sscexplore
3. Learn more about this example

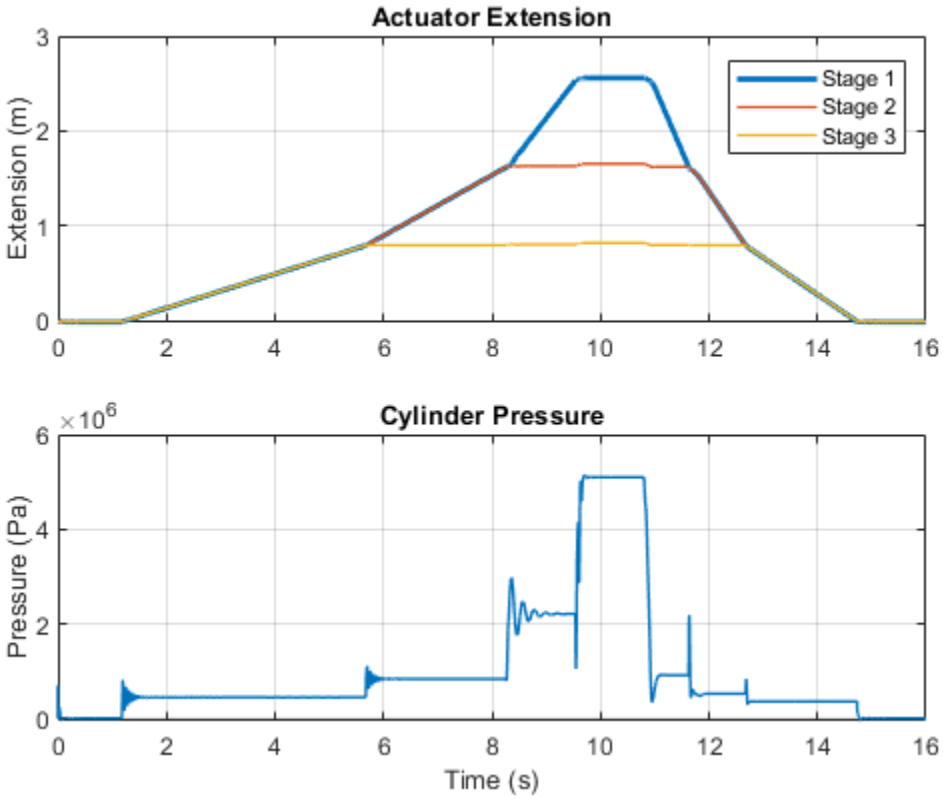
Copyright 2006-2021 The MathWorks, Inc.

Telescopic Cylinder Subsystem



Simulation Results from Simscape Logging

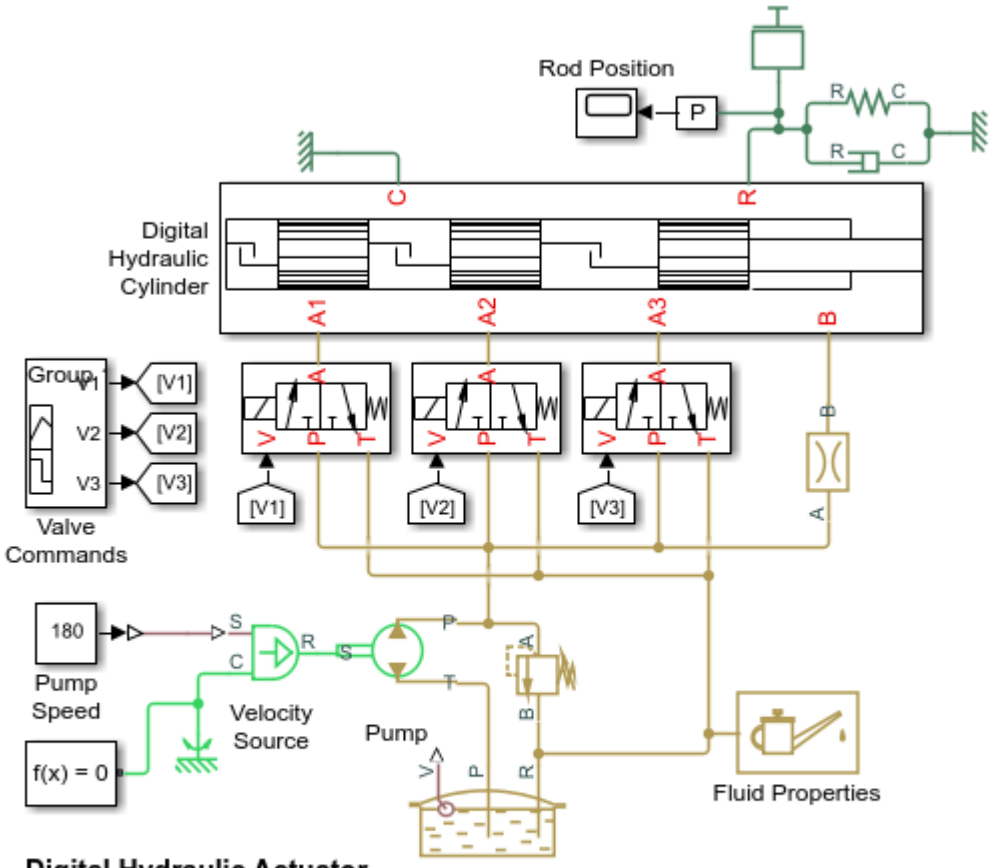
The plots below show the pressures in the cylinder and the extensions of each stage.


Actuator Extension
Extension (m)
Stage 1
Stage 2
Stage 3
0
1
2
3
0
2
4
6
8
10
12
14
16
Cylinder Pressure
×10^6
Pressure (Pa)
0
2
4
6
Time (s)

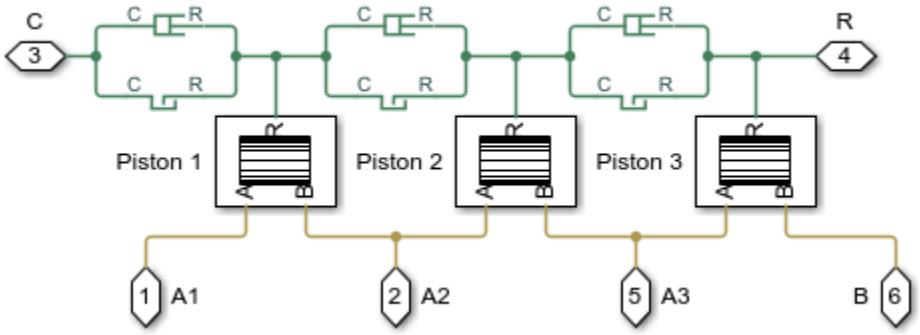
Digital Hydraulic Actuator

This example shows a digital hydraulic actuator that consists of 3 double-acting cylinders mounted in the same shell and interconnected through the hard stops. The gap between the leftmost piston and the shell is equal to 2^0*base_length, the gap between the first and the second pistons is set to 2^1*base_length, and the gap between the second and the third pistons is set to 2^2*base_length. The piston area in the rightmost chamber is half the size of the areas of the other chambers. The rightmost chamber is permanently connected to the pump while the other three chambers are connected either to pump or tank depending on the 2-position 3-way electro-hydraulic valves. As a result, 2^3 discrete positions can be reached within 2^3*base_length range.

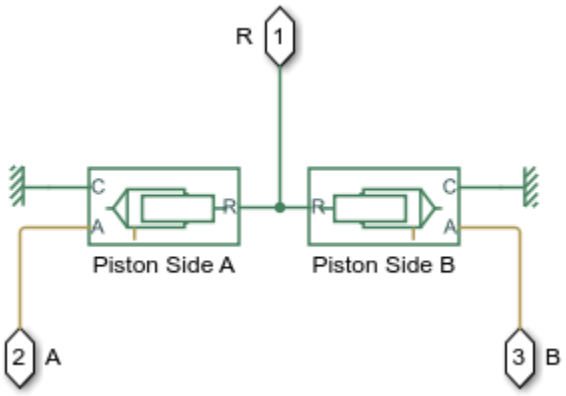
Model



Digital Hydraulic Cylinder Subsystem

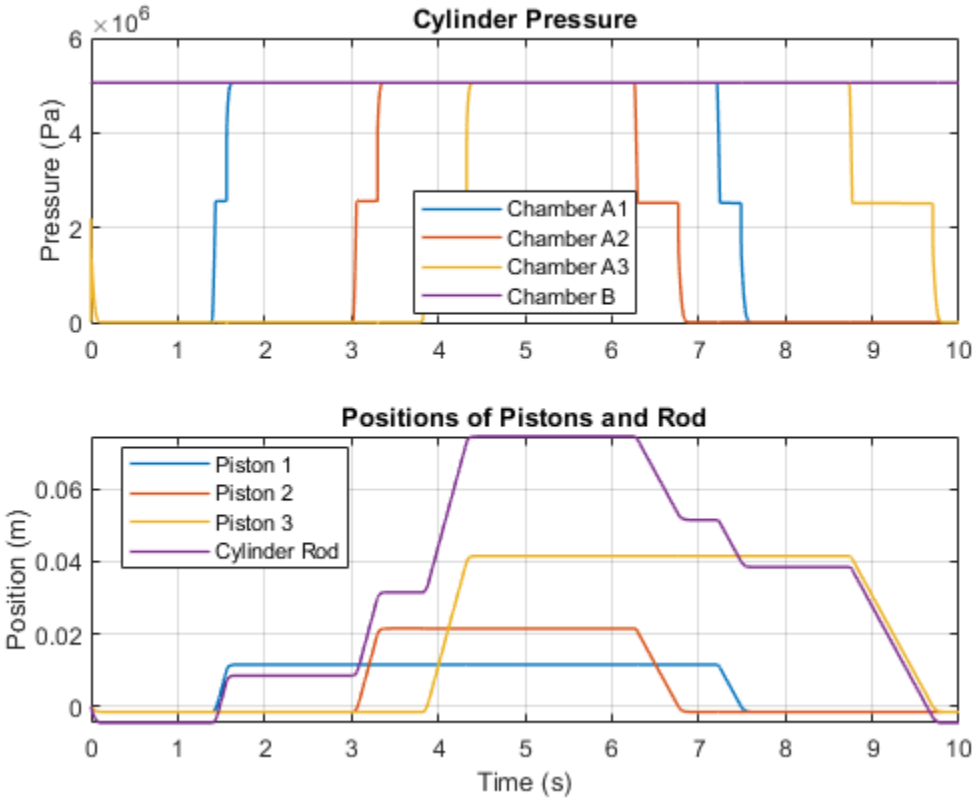


Piston 1 Subsystem



Simulation Results from Simscape Logging

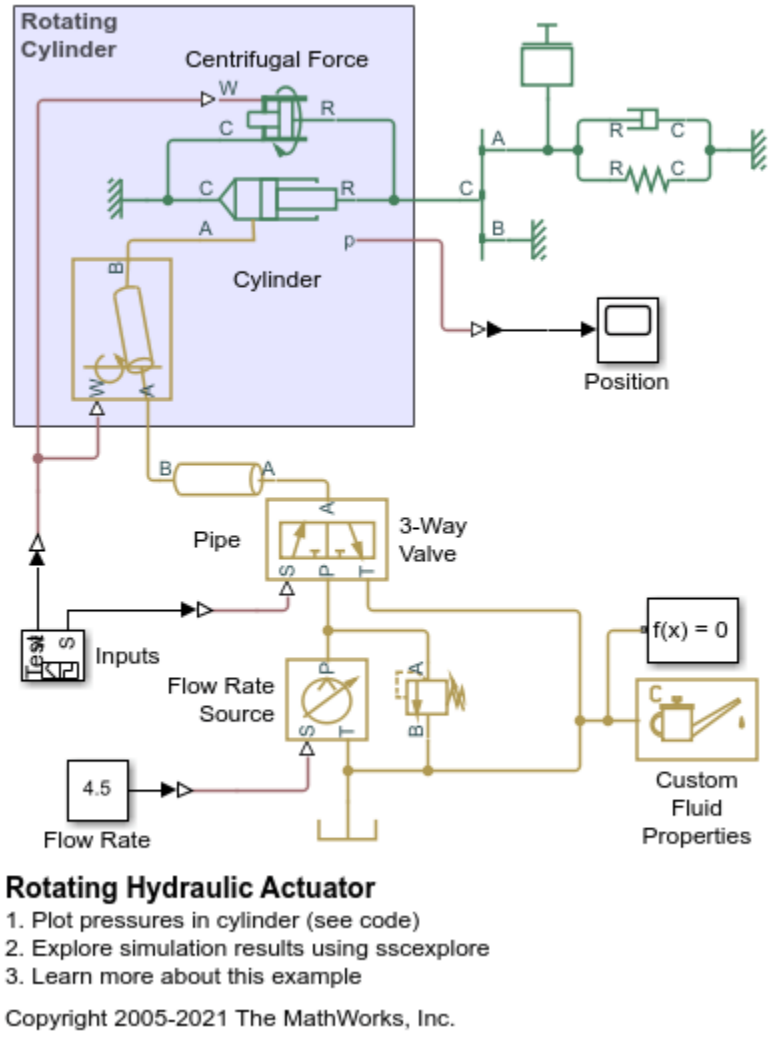
The plots below show the pressures in each cylinder chamber and the extension of each cylinder. Six of the eight possible positions are shown during this cycle.


Cylinder Pressure
×10^6
Pressure (Pa)
Chamber A1
Chamber A2
Chamber A3
Chamber B
Positions of Pistons and Rod
Position (m)
Piston 1
Piston 2
Piston 3
Cylinder Rod
Time (s)

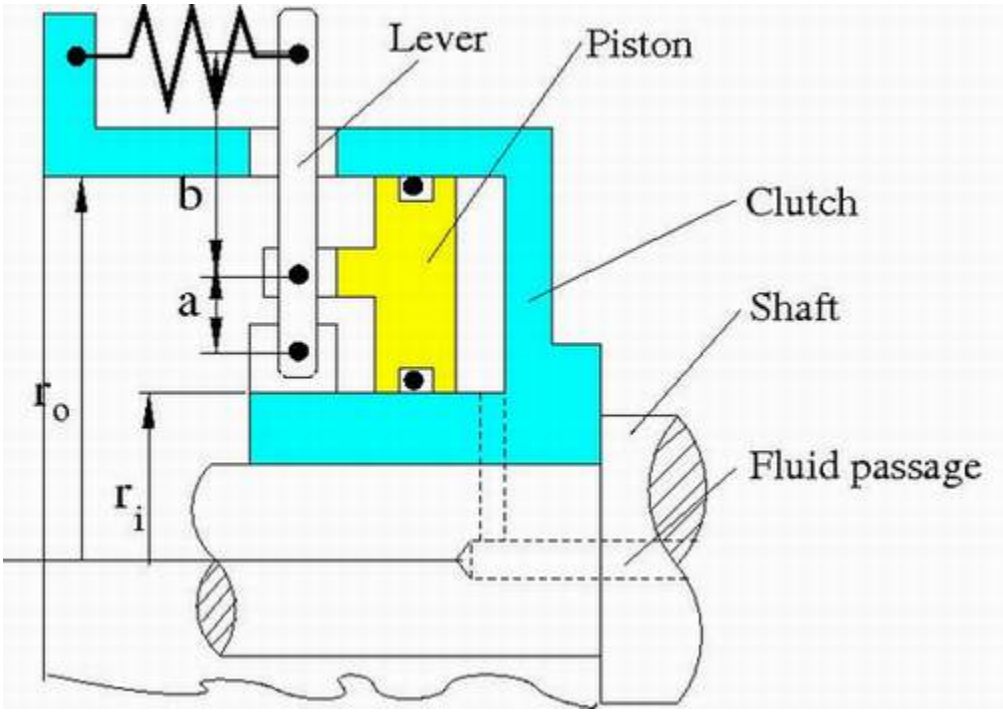
Rotating Hydraulic Actuator

This example shows a typical hydraulic cylinder actuator used to operate friction clutches, brakes and other devices installed on rotating shafts. The key element of the actuator is a piston that moves back and forth under pressure provided through the central drill in the shaft and through channels in the clutch. In the example, the actuator acts against a preloaded spring, which tends to push the piston against the clutch wall.

Model



The following figure shows the schematic diagram of the rotating hydraulic actuator:



r_i - Internal piston radius

r_o - External piston radius

a, b - Arm lengths of the lever

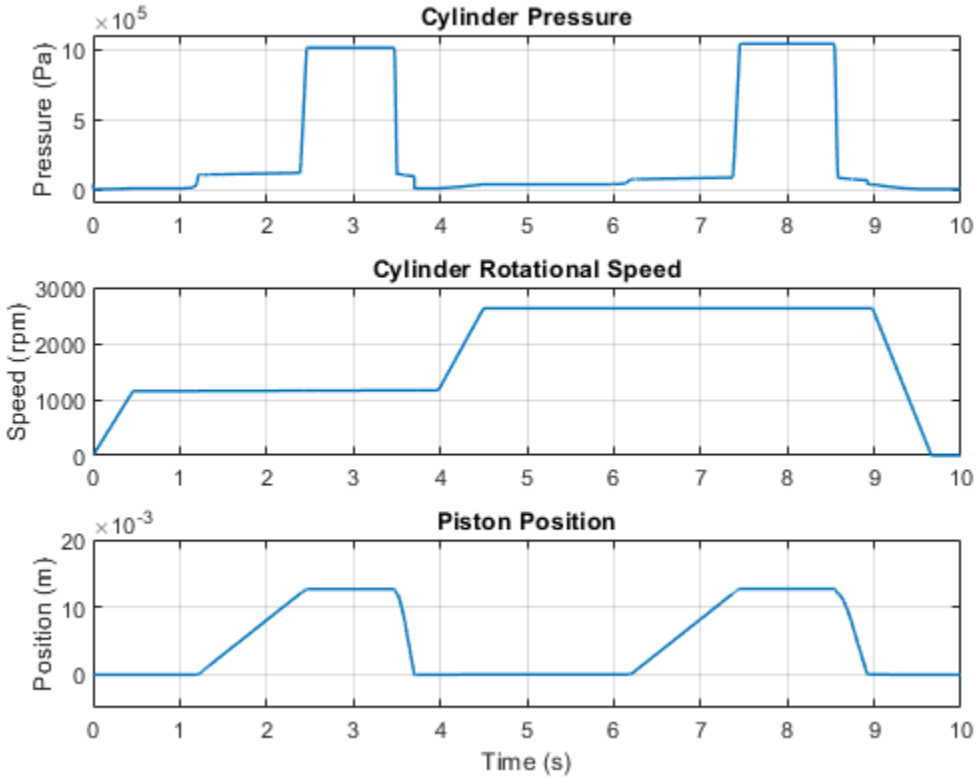
As pressure is applied to the fluid passage in the shaft, static pressure, together with pressure developed in rotating chamber, builds up until it overcomes the preload force and shifts the piston forward.

In the example, the hydraulic system is fed by a pump with delivery of 4.5 lpm. The relief valve is set to 20 bar. The motion control is performed with the 3-way directional valve, which initially connects the clutch chamber with the tank. As control signal is applied, the passage to the tank is closed and the chamber is connected with the pump.

Two actuator cycles are simulated in the example. The first stroke is performed with shaft rotating at angular velocity of 120 rad/s. The angular velocity during the second stroke is 275 rad/s.

Simulation Results from Simscape Logging

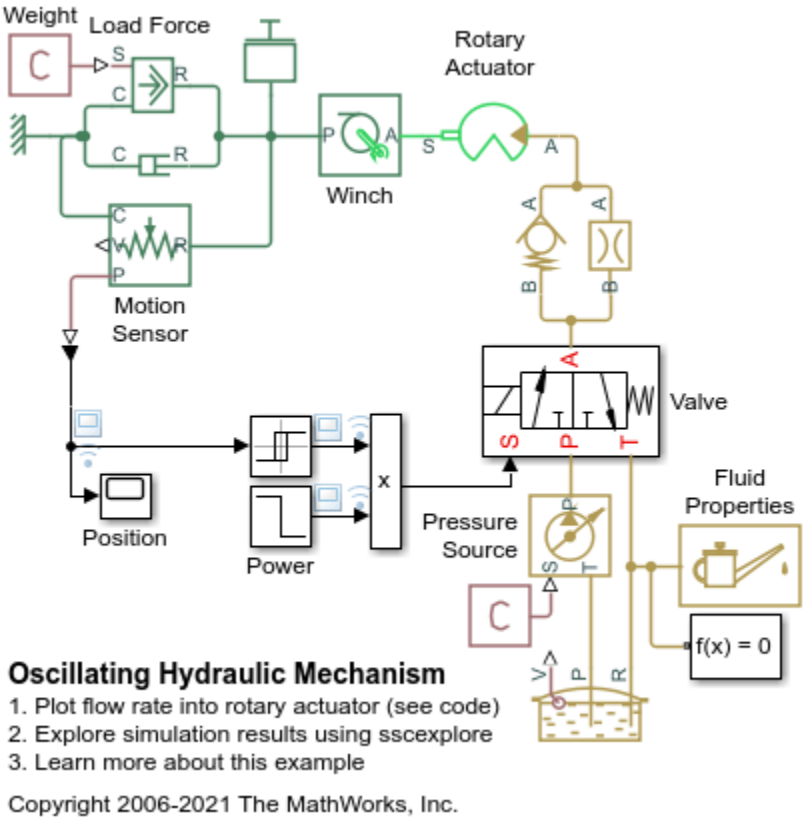
The plots below show the pressure, speed, and extension of the rotating hydraulic cylinder. A slightly higher pressure is seen in the second stroke due to the higher rotational speed of the cylinder.


Cylinder Pressure
×10^5
Pressure (Pa)
10
5
0
0 1 2 3 4 5 6 7 8 9 10
Cylinder Rotational Speed
Speed (rpm)
3000
2000
1000
0
0 1 2 3 4 5 6 7 8 9 10
Piston Position
×10^-3
Position (m)
20
10
0
0 1 2 3 4 5 6 7 8 9 10
Time (s)

Oscillating Hydraulic Mechanism

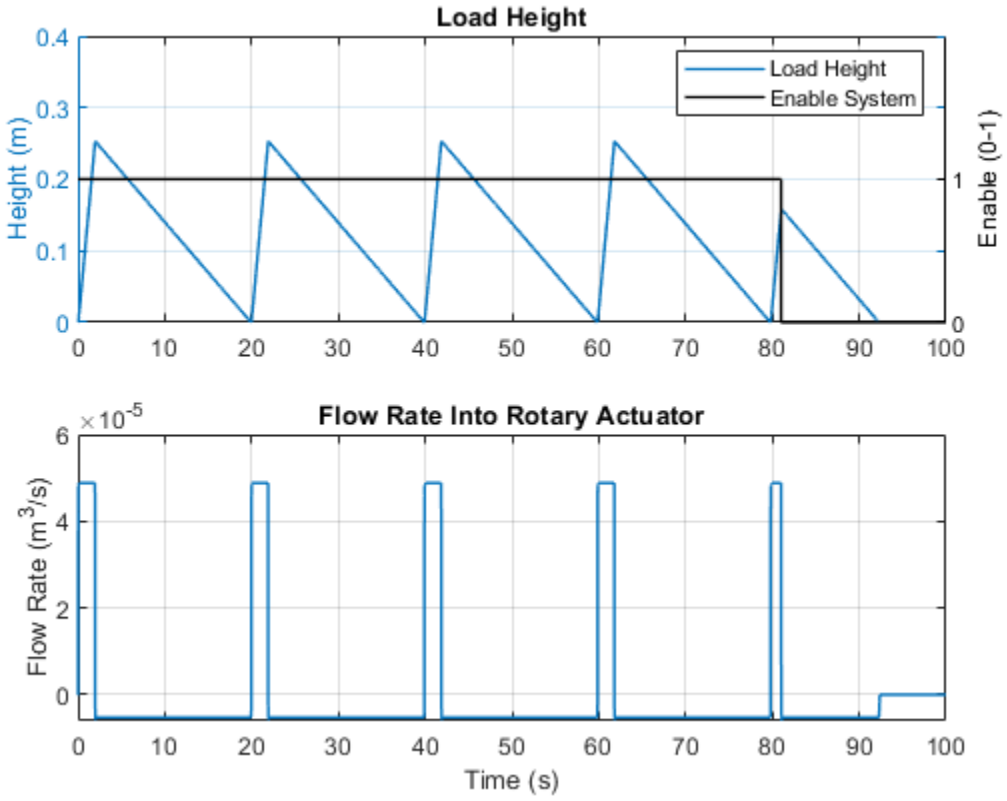
This example shows an oscillating hydraulic mechanism that consists of a single-acting hydraulic rotary actuator, winch, flow control valve, two-position electro-hydraulic valve, and power and control units. The mechanism maintains oscillating vertical motion of the weight by raising it to 25 cm and then allowing it to fall. To raise the weight, the electro-hydraulic valve is energized and connects the power unit to the hydraulic actuator inlet. As the weight reaches 25 cm the valve is deenergized, connecting the actuator chamber to tank.

Model



Simulation Results from Simscape Logging

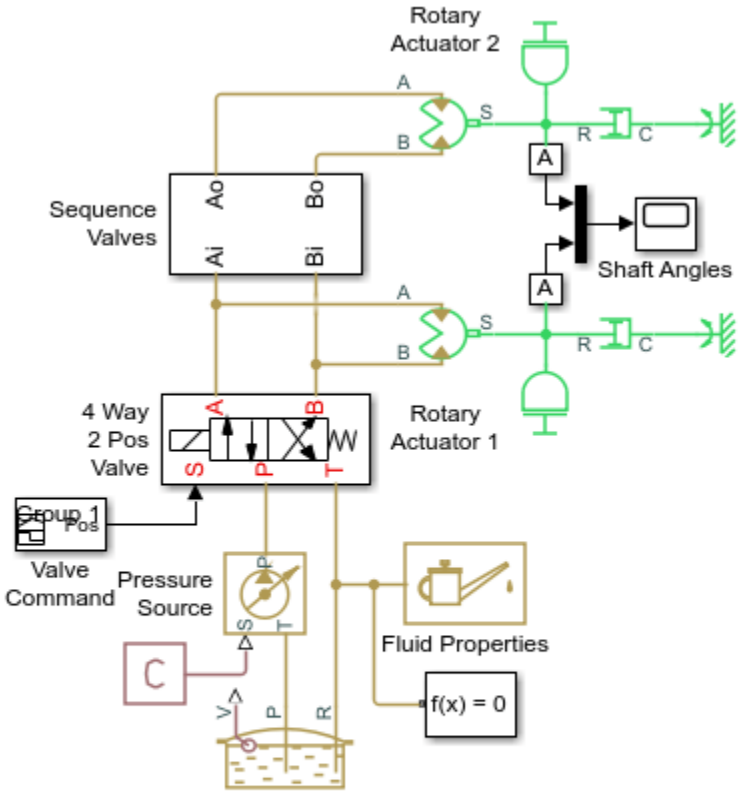
The plots below shows the behavior of the rotary actuator. The load rises much faster than it falls under the influence of gravity. Looking at the flow rate into and out of the rotary actuator, we see that fluid enters much faster than it exits, resulting in different speeds during the cycle. The last cycle is interrupted as the winch is being raised.


Load Height
Load Height
Enable System
Height (m)
Enable (0-1)
0.4
0.3
0.2
0.1
0
1
0
0
10
20
30
40
50
60
70
80
90
100
Flow Rate Into Rotary Actuator
×10^-5
Flow Rate (m^3/s)
6
4
2
0
Time (s)

Sequencing Circuit for Two Rotary Actuators

This example shows a sequence circuit that is based on four check valves installed in both pressure and return lines of the second rotary actuator. The cracking pressure of all the valves is set higher than any load pressure of Rotary Actuator 1, but lower than pressure that develops in its chambers at the end of the stroke when Rotary Actuator 1 reaches its hard stop. As a result, the Rotary Actuator 2 starts moving only after the Rotary Actuator 1 completes its stroke.

Model

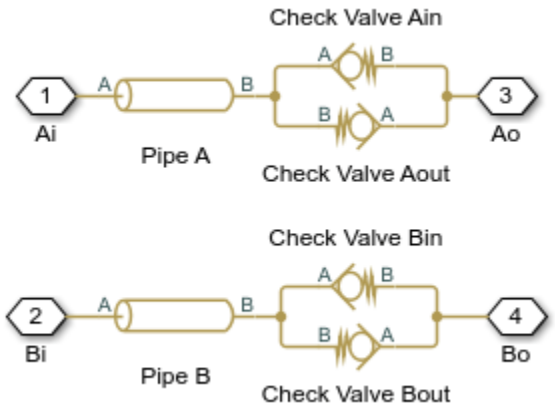


Sequencing Circuit for Two Rotary Actuators

1. Plot pressures in actuators (see code)
2. Explore simulation results using sscexplore
3. Learn more about this example

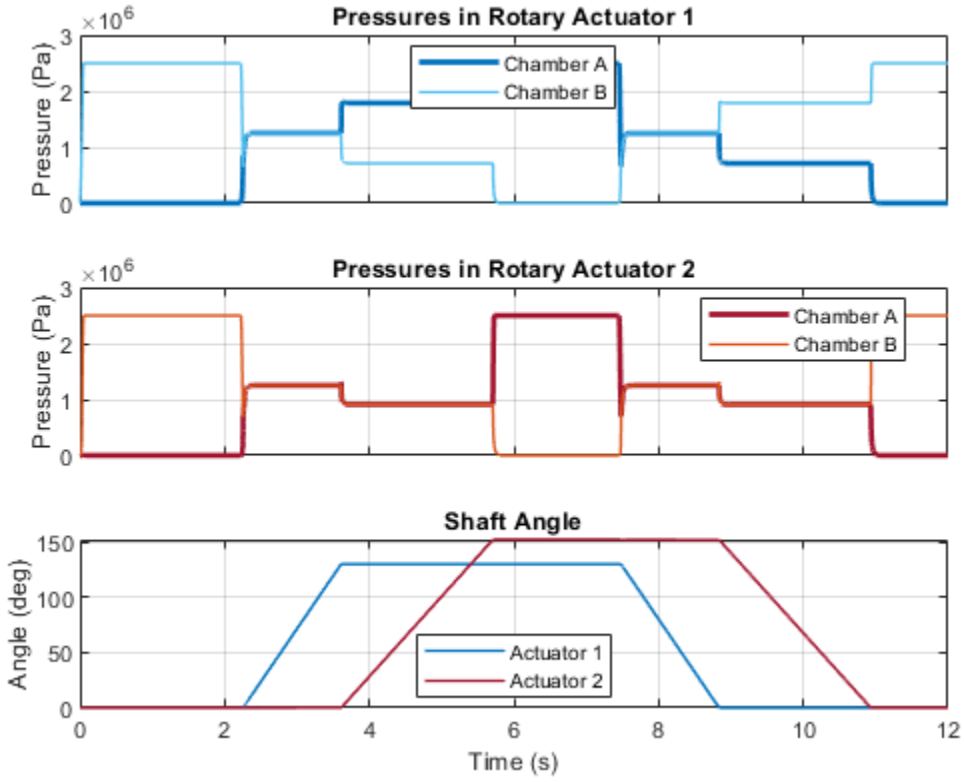
Copyright 2006-2021 The MathWorks, Inc.

Sequence Valves Subsystem



Simulation Results from Simscape Logging

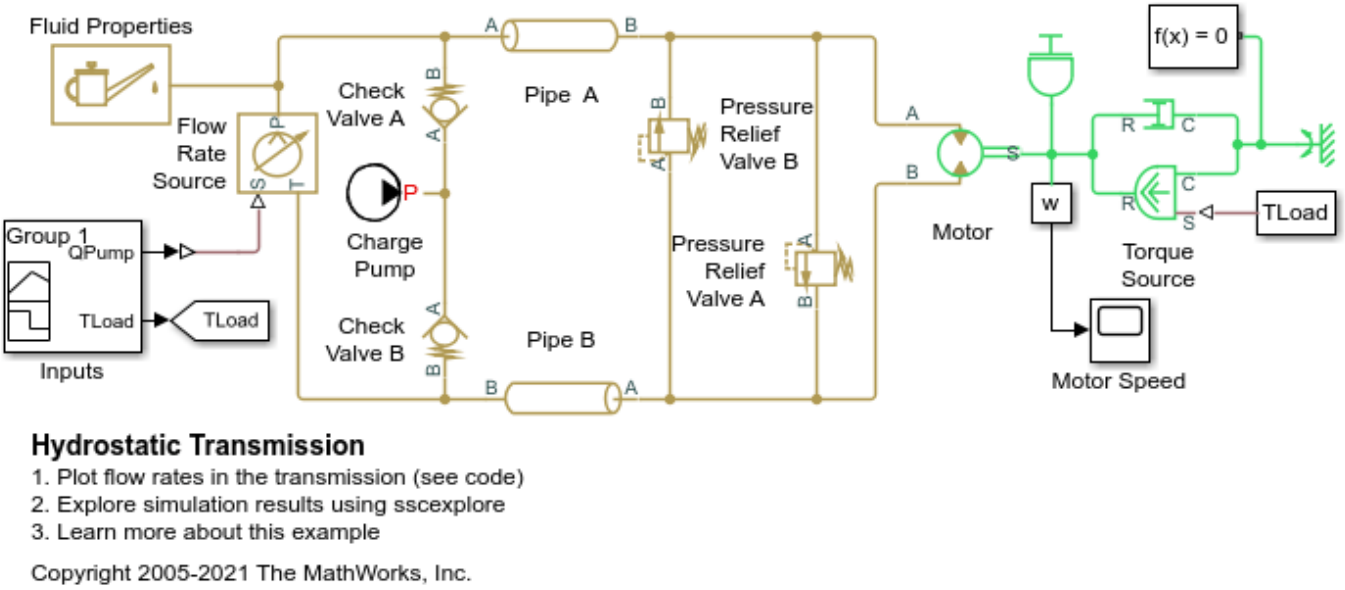
The plots below show the sequence of pressures in the rotary actuator chambers. At the start of the simulation, both actuators are driven against their hard stops at 0 degrees. At 2 seconds, the high pressure line is connected to the opposite chamber of the actuators. The sequencing valves (check valves) prevent flow to actuator 2 until actuator 1 reaches its hard stop. At that point, the pressure rises enough to overcome the check valves and actuator 2 rotates. The opposing set of check valves are similarly tuned to repeat this pattern in the opposite direction.


Pressures in Rotary Actuator 1
×10^6
Pressure (Pa)
Chamber A
Chamber B
Pressures in Rotary Actuator 2
×10^6
Pressure (Pa)
Chamber A
Chamber B
Shaft Angle
Angle (deg)
Actuator 1
Actuator 2
Time (s)

Hydrostatic Transmission

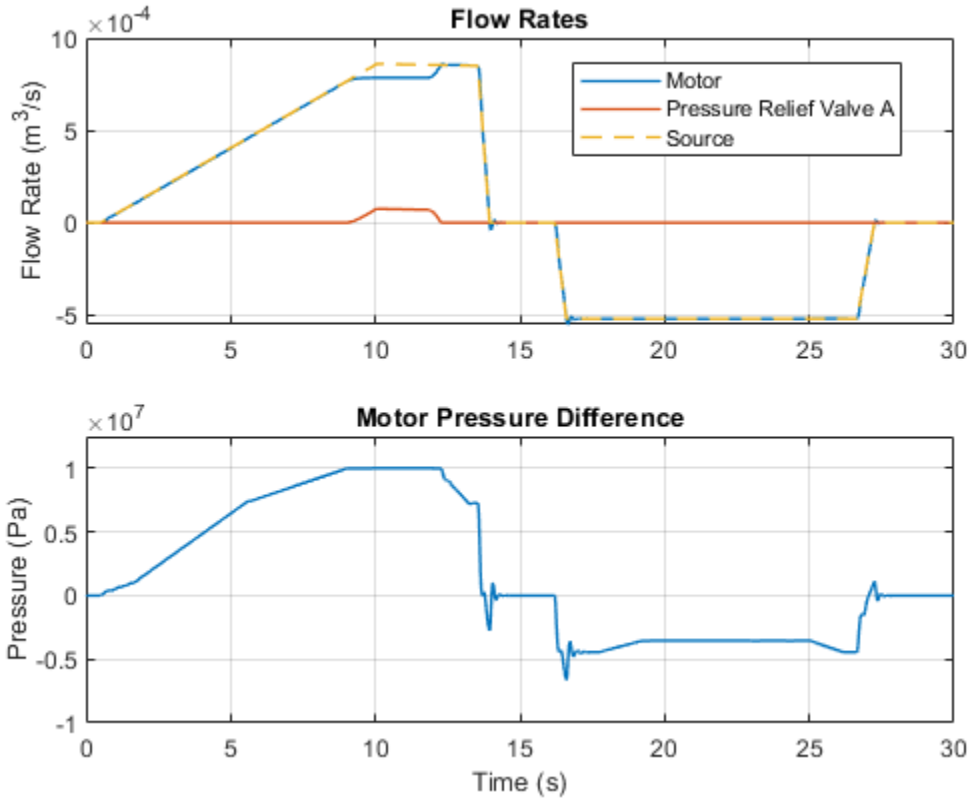
This example shows a transmission built of a variable-displacement pump, and a fixed-displacement hydraulic motor. Pipelines between the source and motor are connected by two replenishing valves (check valves) and a charge pump on the pump side and two pressure-relief valves on the motor side. The motor drives a mechanical load consisting of inertia, viscous friction, and time-varying torque. The system is tested with varying pump flow rate and motor torque load.

Model



Simulation Results from Simscape Logging

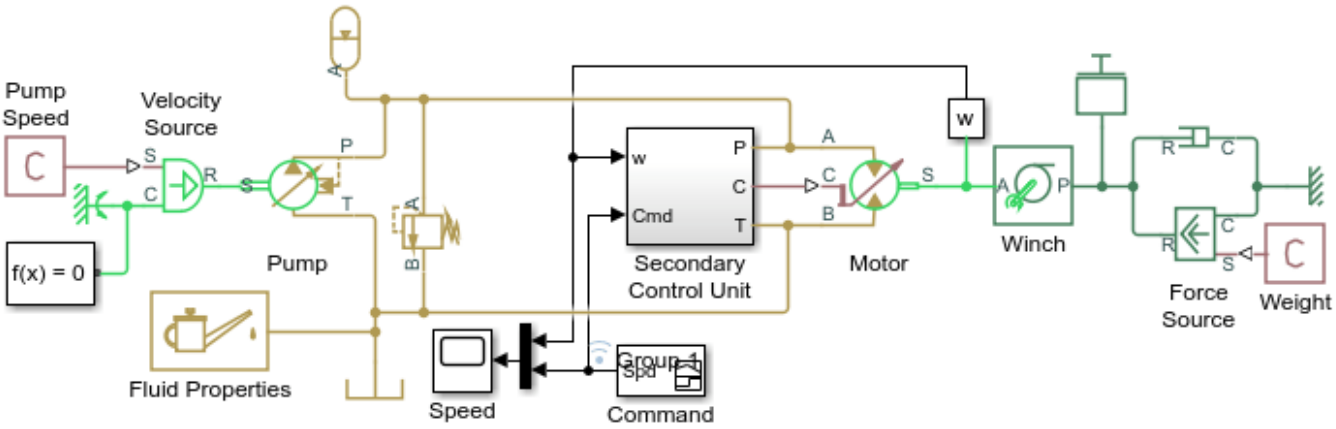
The plots below show the flow rate through the hydrostatic transmission and the pressure drop across the motor. As the motor reaches its nominal pressure, flow is diverted through a pressure relief valve.


Flow Rates
×10^-4
10
5
0
-5
Flow Rate (m^3/s)
Motor
Pressure Relief Valve A
Source
0
5
10
15
20
25
30
Motor Pressure Difference
×10^7
1
0.5
0
-0.5
-1
Pressure (Pa)
0
5
10
15
20
25
30
Time (s)

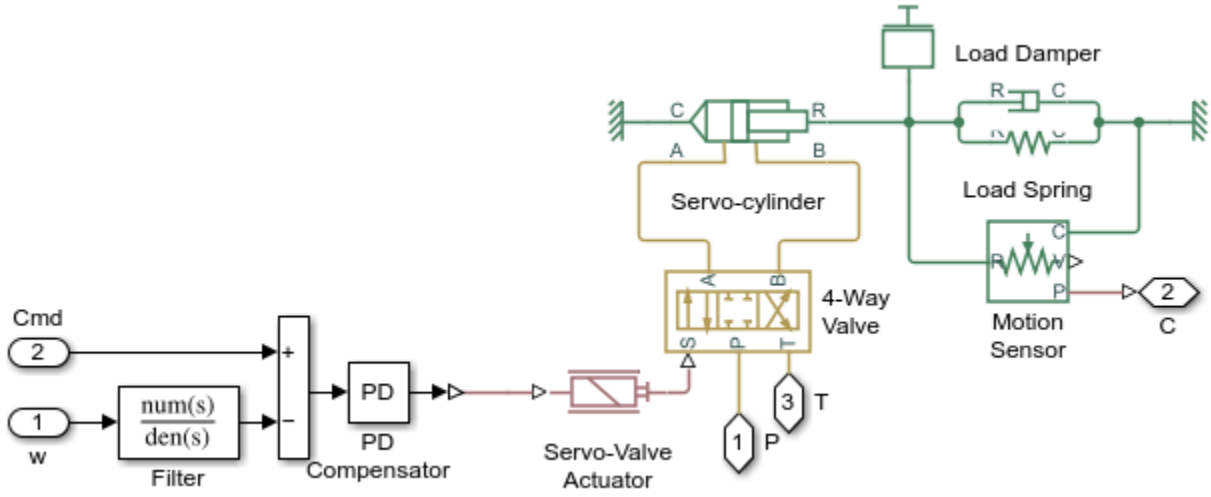
Hydrostatic Transmission with Secondary Control

This example shows the usage of secondary control in hydrostatic transmissions with a variable-displacement motor. It uses a pressure-compensated variable-displacement pump, a variable-displacement motor, and is controlled by a secondary control unit. The secondary control unit is a servo-cylinder controlled by a proportional 4-way valve. The servo-cylinder drives the control member of the variable-displacement motor, which is represented by a mass, spring, and damper. The hydrostatic transmission is used in a closed-loop control system with an angular velocity feedback. It drives a winch which lifts a mass and acts against a viscous damper. A dynamic compensator is included in the secondary control unit to improve system stability.

Model

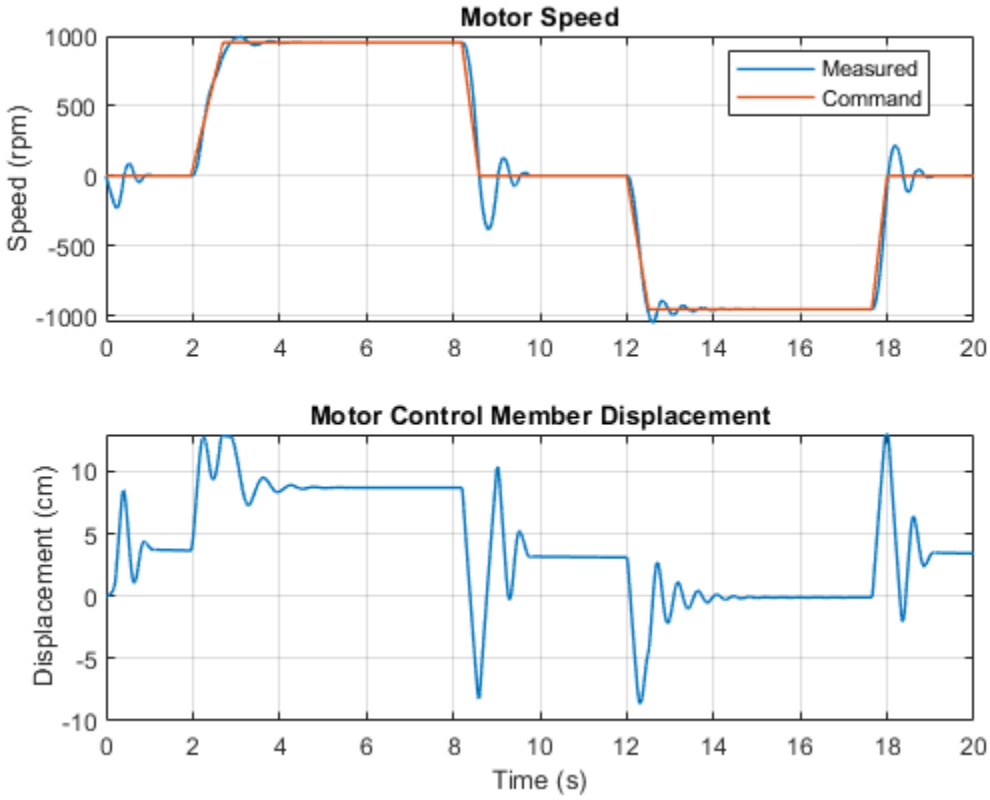


Secondary Control Unit Subsystem



Simulation Results from Simscape Logging

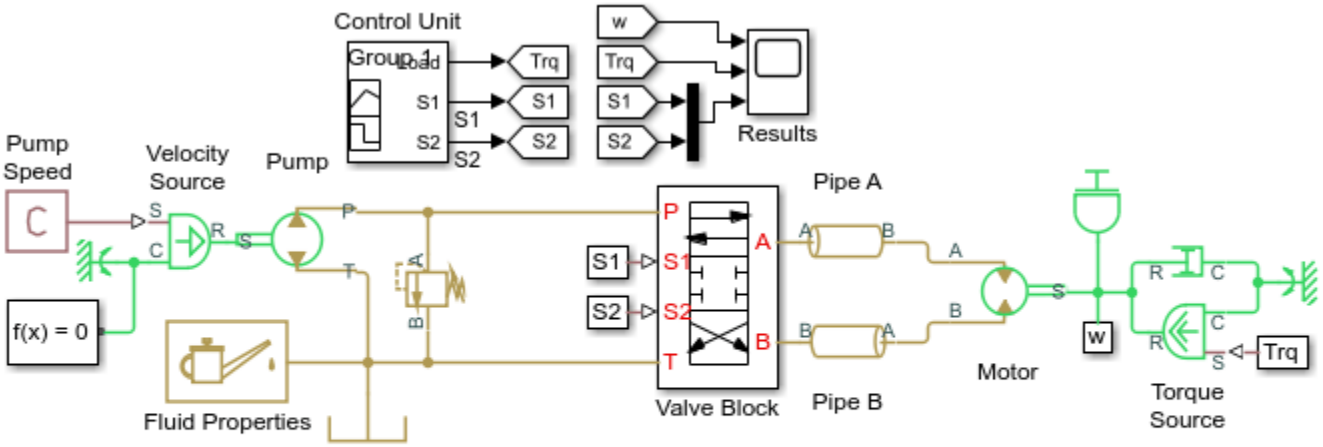
The plots below show the commanded and measured speed of the motor as well as the displacement of the control member which sets the displacement of the motor.



Hydrostatic Transmission with Shuttle Valve

This example shows a hydrostatic transmission with a shuttle valve in the control unit. The 4-way directional valve that controls the motor is actuated by a valve actuator whose input is controlled by two 2-position valves (Shuttle A Valve, Shuttle A1 Valve). The shuttle valve installed between the actuator and the 2-position valves allows the 4-way directional valve to be opened by opening either 2-position valve, and those valves are opened by signals S1 and S2. A fixed orifice is connected to the P port of the two-position valves to decrease flow consumption of the control circuit. Another fixed orifice introduces a small leak from the valve control chamber to avoid this chamber being isolated from the main flow. The motor is connected to pressure line at 1 second and 5 seconds by activating each of the two-position valves.

Model

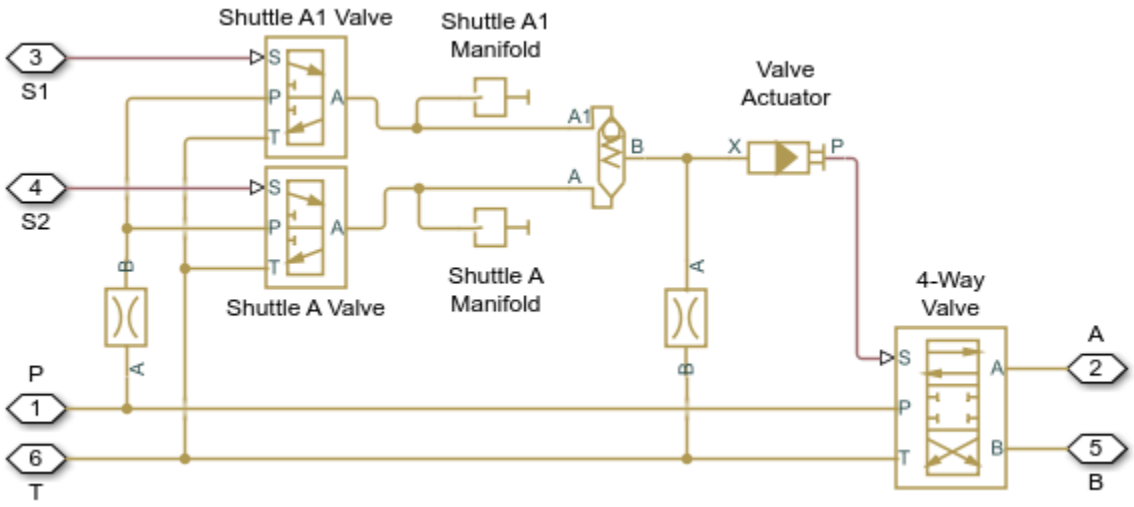


Hydrostatic Transmission with Shuttle Valve

1. Plot flow rate in motor (see code)
2. Explore simulation results using sscexplore
3. Learn more about this example

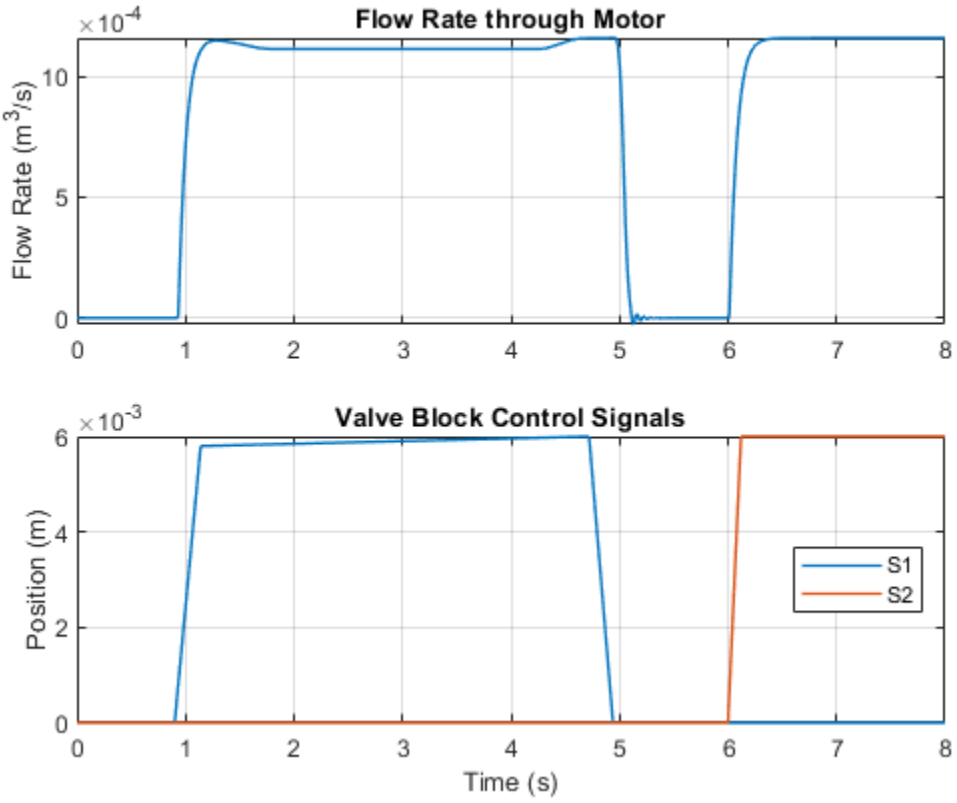
Copyright 2005-2021 The MathWorks, Inc.

Valve Block Subsystem



Simulation Results from Simscape Logging

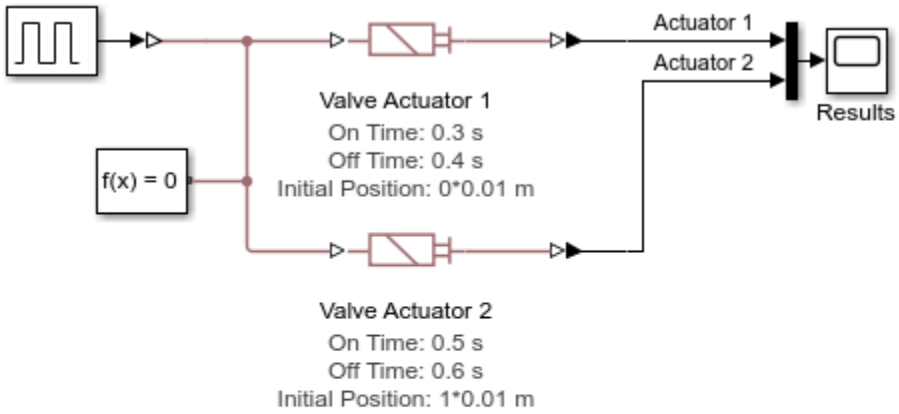
The plots below show the flow rate through the motor and the control signals to valves that connect the motor to the high pressure side of the pump.



2-Position Valve Actuator

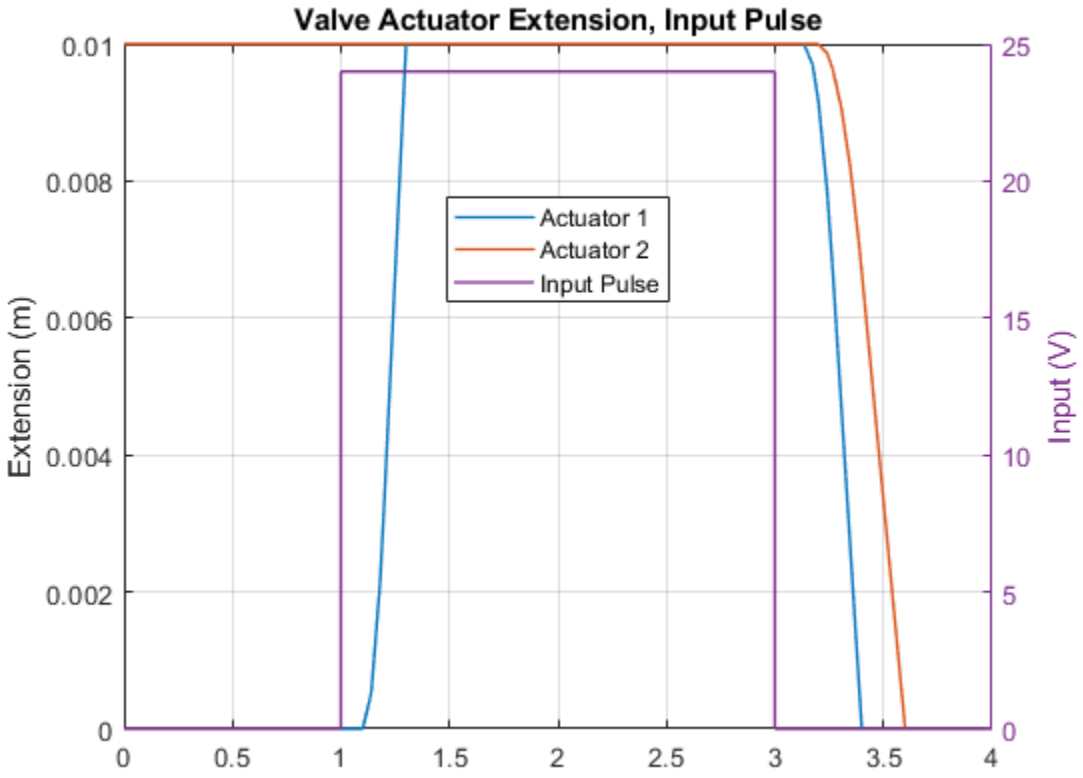
This example shows two valve actuators with different values for switching-on time and switching-off time. Valve Actuator 1 is set to start from the retracted position, while Valve Actuator 2 from the extended position. Both actuators are driven with the same pulse signal.

Model



Simulation Results from Simscape Logging

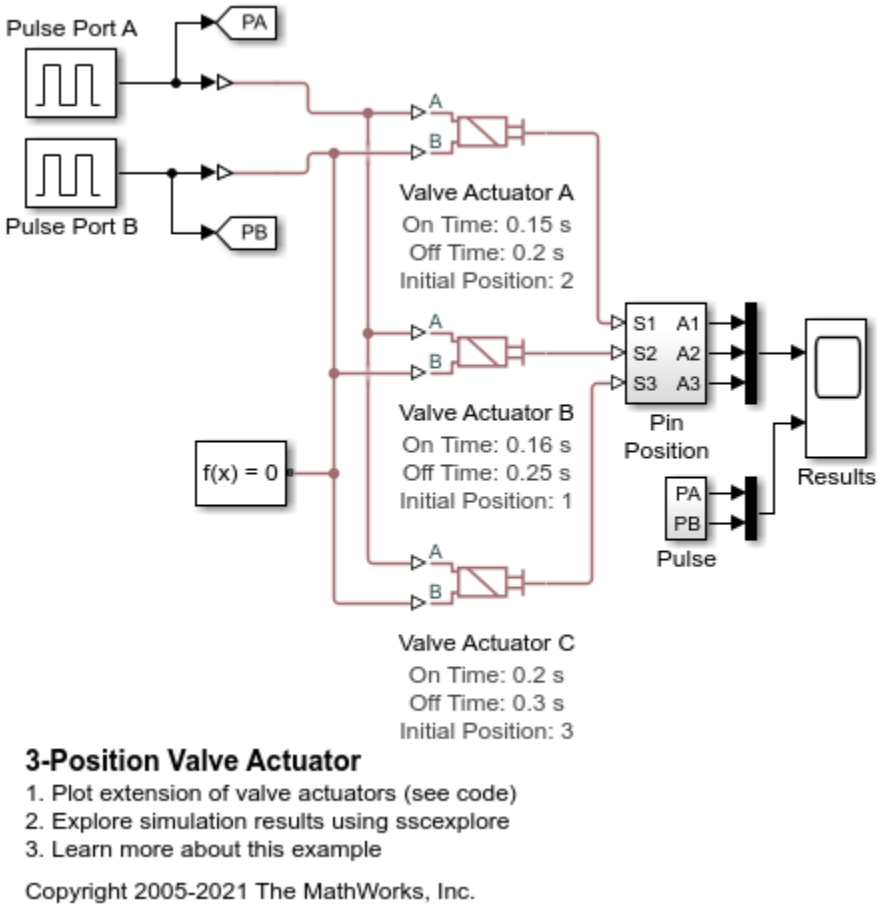
The plot below shows the behavior of two valve actuators with different values for switching-on time and switching-off time.


Valve Actuator Extension, Input Pulse
Extension (m)
Input (V)
Actuator 1
Actuator 2
Input Pulse
0.01
0.008
0.006
0.004
0.002
0
25
20
15
10
5
0
0
0.5
1
1.5
2
2.5
3
3.5
4

3-Position Valve Actuator

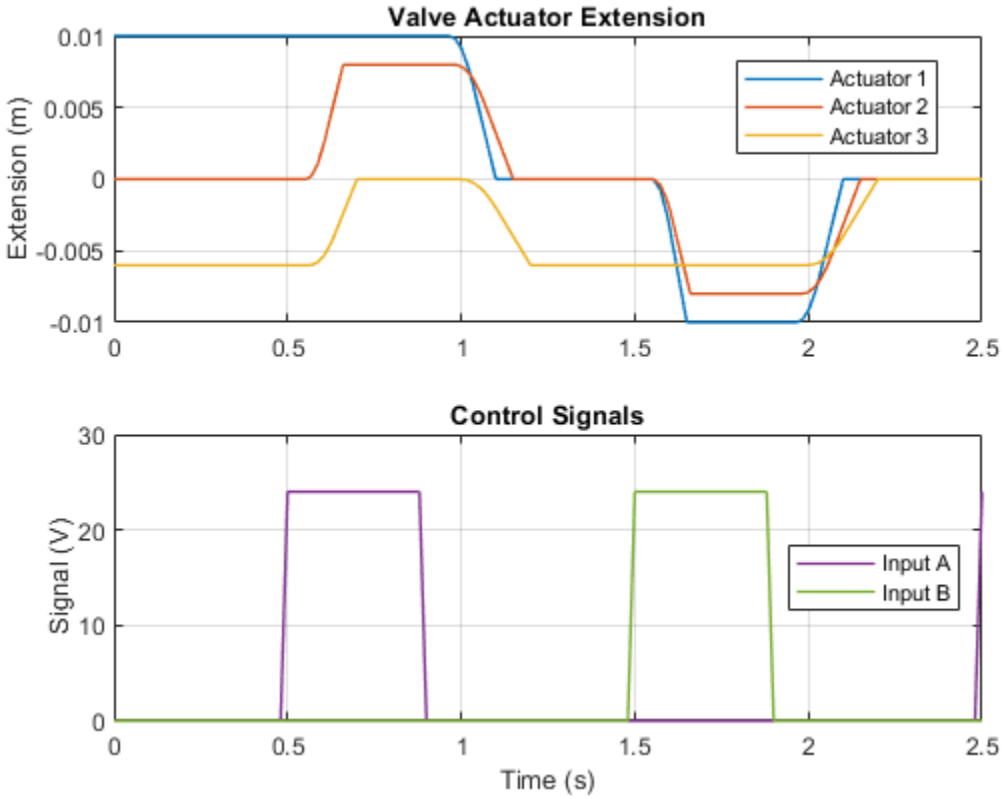
This example shows a use of a double-acting valve actuator. All three actuators are driven by the same pulse signals. A pulse is first applied to port A and after 1.5 s delay a pulse is applied to port B. Actuator A is set to start from the "Extended positive" position, while actuator C starts from the "Extended negative" position. As a result, actuators A and C move to the neutral position at the start of simulation and reach this position before the first pulse is applied. The actuators strokes, switch-on, and switch-off times are set to different values to illustrate the effects of these parameters.

Model



Simulation Results from Simscape Logging

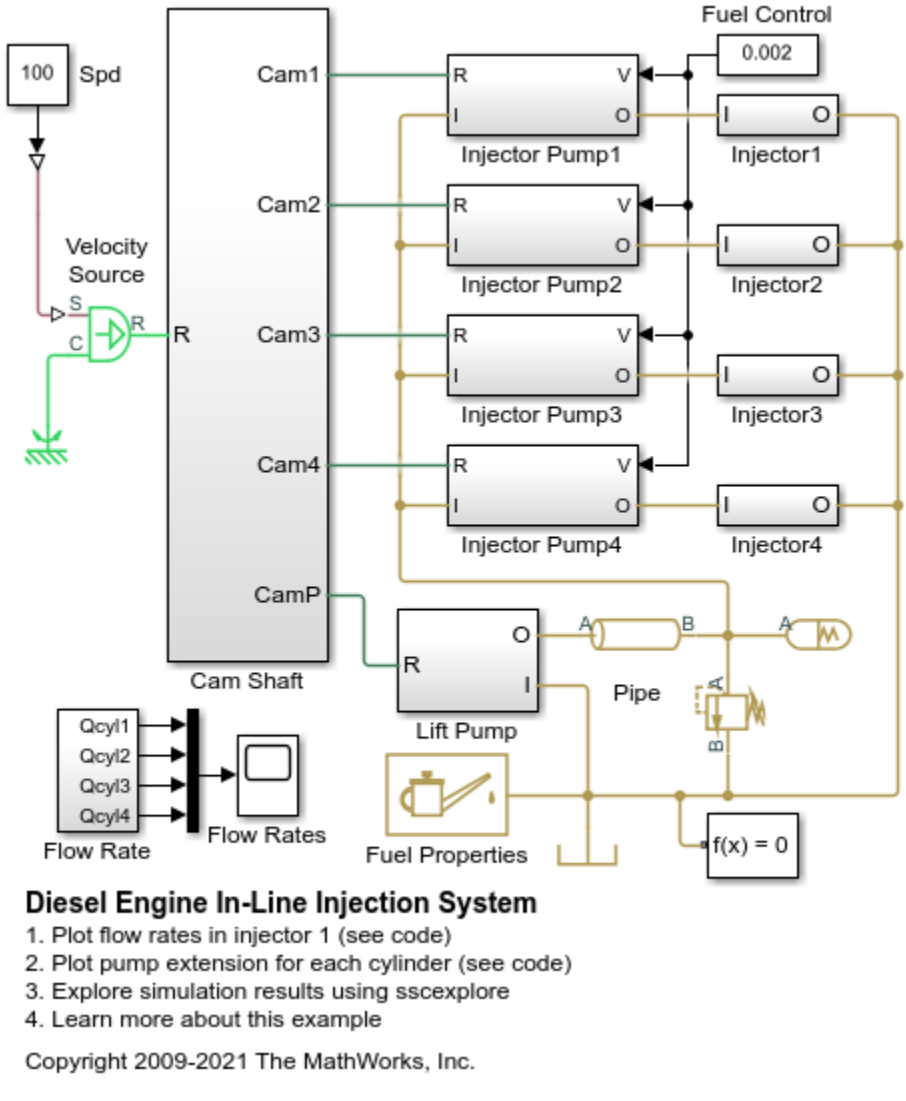
The plots below show the behavior of the 3-Position Valve Actuator block. The same pulses are applied to all three actuators, which each have different values for switching-on time, switching-off time, and stroke.


Valve Actuator Extension
Extension (m)
0.01
0.005
0
-0.005
-0.01
0
0.5
1
1.5
2
2.5
Actuator 1
Actuator 2
Actuator 3
Control Signals
Signal (V)
30
20
10
0
Input A
Input B
Time (s)

Diesel Engine In-Line Injection System

This example shows an in-line multi-element diesel injection system. It contains a cam shaft, lift pump, 4 in-line injector pumps, and 4 injectors.

Model



Injection System Description

The diesel injection system simulated by this model is shown in the schematic diagram below.

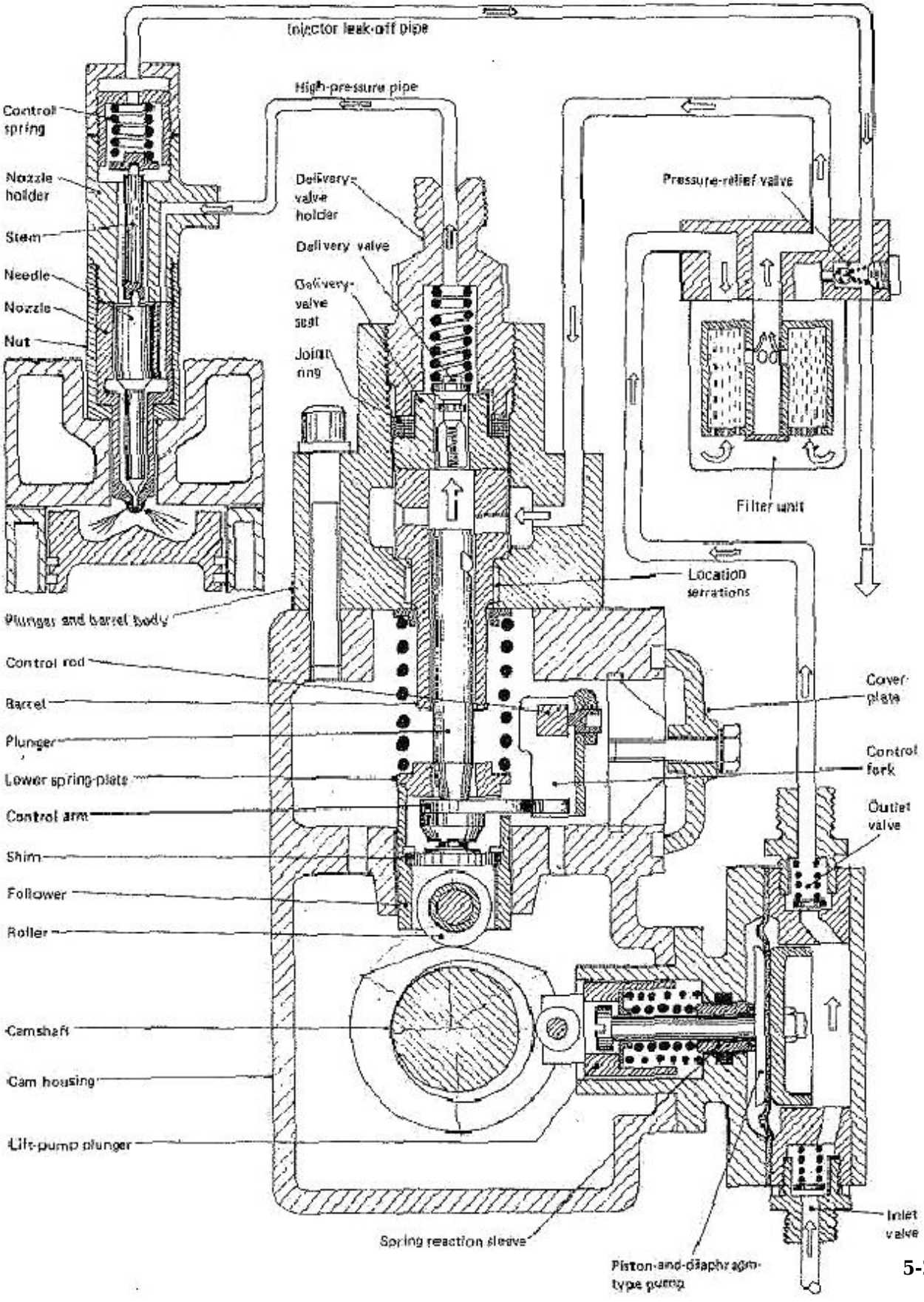

Injector leak-off pipe
High-pressure pipe
Control spring
Nozzle holder
Stem
Needle
Nozzle
Nut
Delivery-valve holder
Delivery valve
Delivery-valve seat
Joint ring
Pressure-relief valve
Filter unit
Location serrations
Plunger and barrel body
Control rod
Barrel
Plunger
Lower spring-plate
Control arm
Shim
Follower
Roller
Camshaft
Cam housing
Lift-pump plunger
Cover plate
Control fork
Outlet valve
Inlet valve
Spring reaction sleeve
Piston-and-diaphragm-type pump

Figure 1. Injection System Schematic Diagram

The system structure is reproduced from H. Heisler, *Vehicle and Engine Technology* (second edition), 1999, and is categorized as an in-line multi-element injection system. It consists of the following basic units:

- Cam shaft
- Lift pump
- In-line injector pump with four pumping elements, one element per cylinder
- Injectors

The cam shaft carries five cams. The first one is the eccentric cam to actuate the lift pump. The remaining four are intended to drive plungers of the pump. The cams are installed in such a way that pumping elements deliver fuel in the firing order and at the correct instant in the engine's cycle of operation. The lift pump supplies fluid to the intake of injector pump elements. Each element of the pump consists of a cam-driven plunger, delivery valve, and the governor assembly. The purpose of the governor is to control the volume of the fuel delivered by the plunger to a cylinder. It is attained by rotating the plunger with the helical groove with respect to the spill orifice. All the system units will be described in more details in the following sections.

The objective of the simulation is to investigate the entire system operation. The objective dictates the extent of idealization of every model in the system. If the objective were, for example, delivery valve or the injector investigation, the amount of factors taken into consideration and the scope of the element considered would be different.

Note: The model of the system does not represent any particular injection system. All the parameters have been assigned based on practical considerations, and do not represent any particular manufacturer parameters.

Cam Shaft

The model of a cam shaft is built of five cam models. There are four parabolic profile cams and one eccentric cam. Every cam contains a Simulink® masked subsystem that describes the cam's profile and generates motion profile for the position source, which is built out of Simscape™ blocks.

Cam Profile Simulation

The motion profile is generated as a function of the shaft angle, which is measured with the Angle Sensor block from the Pumps and Motors library. The sensor converts the measured angle to a value in the range from zero to 2*pi. After the cycle angle is determined, it is passed to the Simulink IF subsystem, which computes the profile. The cam that drives the plunger of the pump element is supposed to have a parabolic profile under which the follower moves back and forth at constant acceleration, as follows:

$$x = 2s \cdot \left(\frac{\theta - \theta_E}{\alpha} \right)^2, \qquad if \;\; \theta_E \le \theta < \frac{\alpha}{2} + \theta_E$$

$$x = s\left(1 - 2\left(1 - \frac{\theta - \theta_E}{\alpha}\right)^2\right), \qquad if \;\; \frac{\alpha}{2} + \theta_E \le \theta < \alpha + \theta_E$$

$$x = s\left(1 - 2\left(\frac{\theta - \theta_R}{\beta}\right)^2\right), \qquad if \;\; \theta_R \le \theta < \frac{\beta}{2} + \theta_R$$

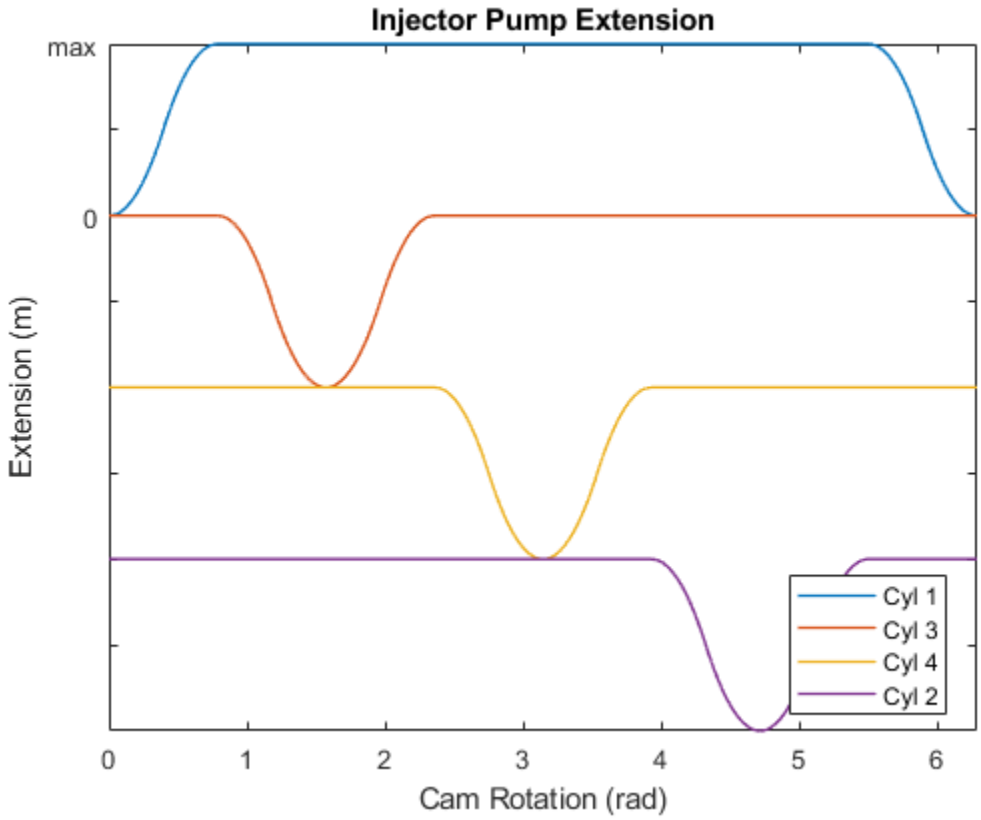
$$x = 2s\left(1 - \frac{\theta - \theta_R}{\beta}\right)^2, \qquad if \;\; \frac{\beta}{2} + \theta_R \le \theta < \beta + \theta_R$$

where

x	Follower position
s	Follower stroke
θ	Cam angular position (with phase angle accounted)
θ_E	Start extend angle
α	Extend angle
θ_R	Start retract angle
β	Retract angle

As a result, at *start extend angle* the follower starts moving up and reaches its top position after the shaft turns an additional *extend angle*. The follower starts the return stroke at *start retract angle* and it takes the *retract angle* to complete this motion. The difference between the *start retract angle* and the (*start extend angle* + *extend angle*) sets the *dwell angle* at fully extended position. The profile is implemented in the Simulink IF subsystem.

The firing sequence for the simulated diesel engine is assumed to be 1-3-4-2. The sequence of cam operation is shown in the figure below. The extend and return angles are set to pi/4. The dwell angle with fully extended follower is set to 3*pi/2 rad.



The profile of the eccentric cam is computed with the formula

$$x = e\left(1 - \cos(\theta)\right)$$

where e is eccentricity.

Position Source

The position source model, which generates position in mechanical translational motion following the Simulink signal at its input, is built of an Ideal Translational Velocity Source block, a PS Gain block, and a Translational Motion Sensor block installed in the negative feedback. The transfer function of the position source is

$$H(s) = \frac{1}{Ts + 1},$$

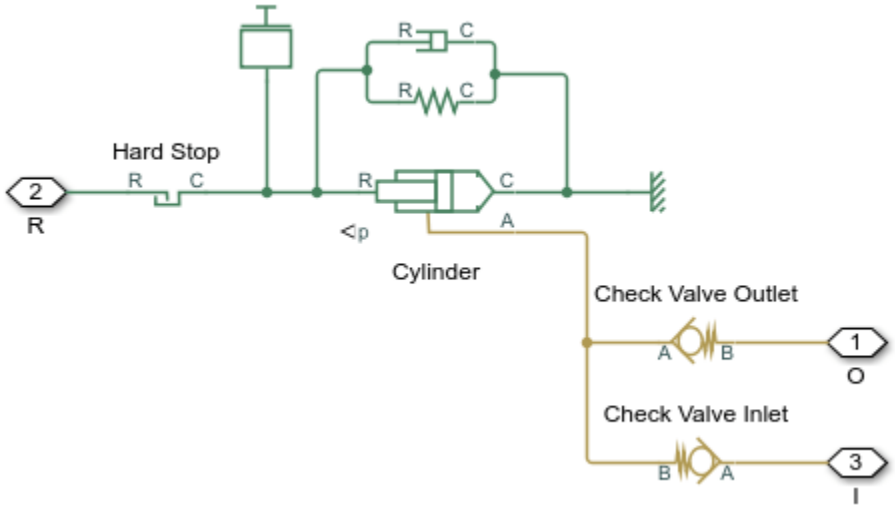
where

T - Time constant, equal 1/Gain,

Gain - Gain of the PS Gain block.

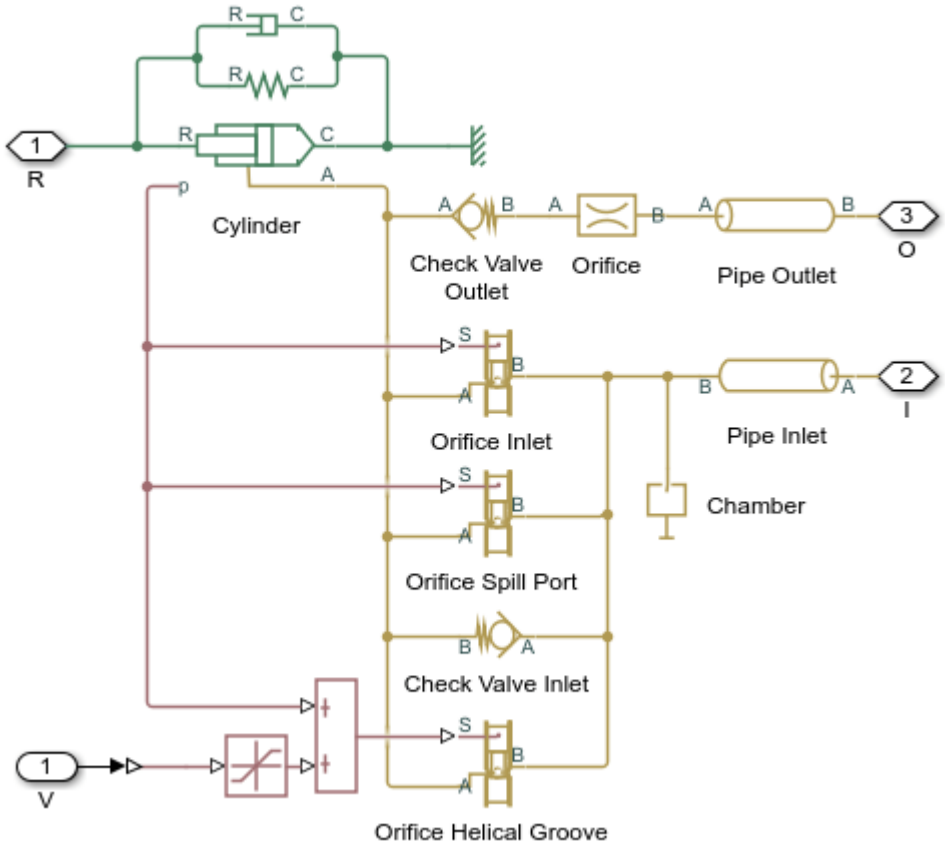
The gain is set to 1e6, which means that signals with frequencies up to 160 kHz are passed practically unaffected.

Lift Pump



The model of the lift pump, which is a piston-and-diaphragm type pump, is built of a Single-Acting Hydraulic Cylinder block and two Check Valve blocks. The check valves simulate inlet and outlet valves installed on both sides of the lift pump (see Figure 1). The contact between the pump rod roller and the cam is represented with the Translational Hard Stop block. The Translational Spring block simulates two springs in the pump that are supposed to maintain permanent contact between the roller and the cam.

Injection Pump



The in-line injection pump is a four-element pumping unit. Each element supplies fuel to its cylinder. All four elements are identical by design and parameters and simulated with the same model called Injection pump element. Each Injection pump element Injection pump element model contains two subsystems, named Pump and Injector, respectively. The Pump represents the pump plunger and the pump control mechanism, while the Injector simulates an injector installed directly on the engine cylinder (see Figure 1).

The pump plunger oscillates inside pump barrel driven by the cam (see Figure 1). The plunger is simulated with the Single-Acting Hydraulic Cylinder block. The Translational Hard Stop and Mass blocks represent the contact between the plunger roller and mass of the plunger, respectively. The contact is maintained by spring TS.

When the plunger moves down, the plunger chamber is filled with fuel under pressure developed by the lift pump. The fluid fills the chamber through two orifices, named Inlet port and Spill port (see Figure 2,a below).

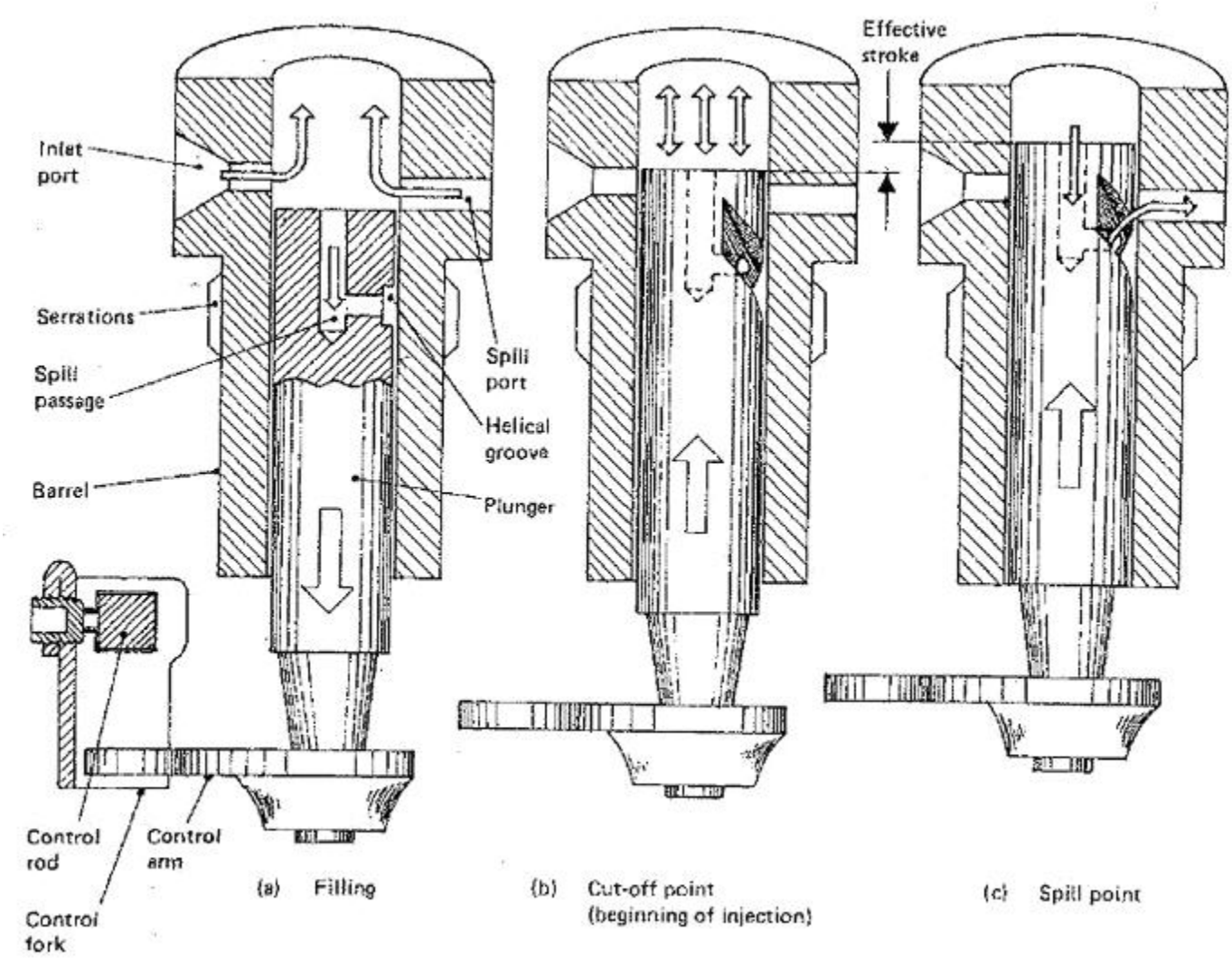


Figure 2. Plunger Interaction with Control Orifices in the Barrel

After the plunger moves towards its top position high enough to cut off both orifices from the inlet chamber, the pressure at the outlet starts building up. At a certain rise the injector at the engine cylinder is forced to open and fuel starts injecting in the cylinder (Figure 2,b).

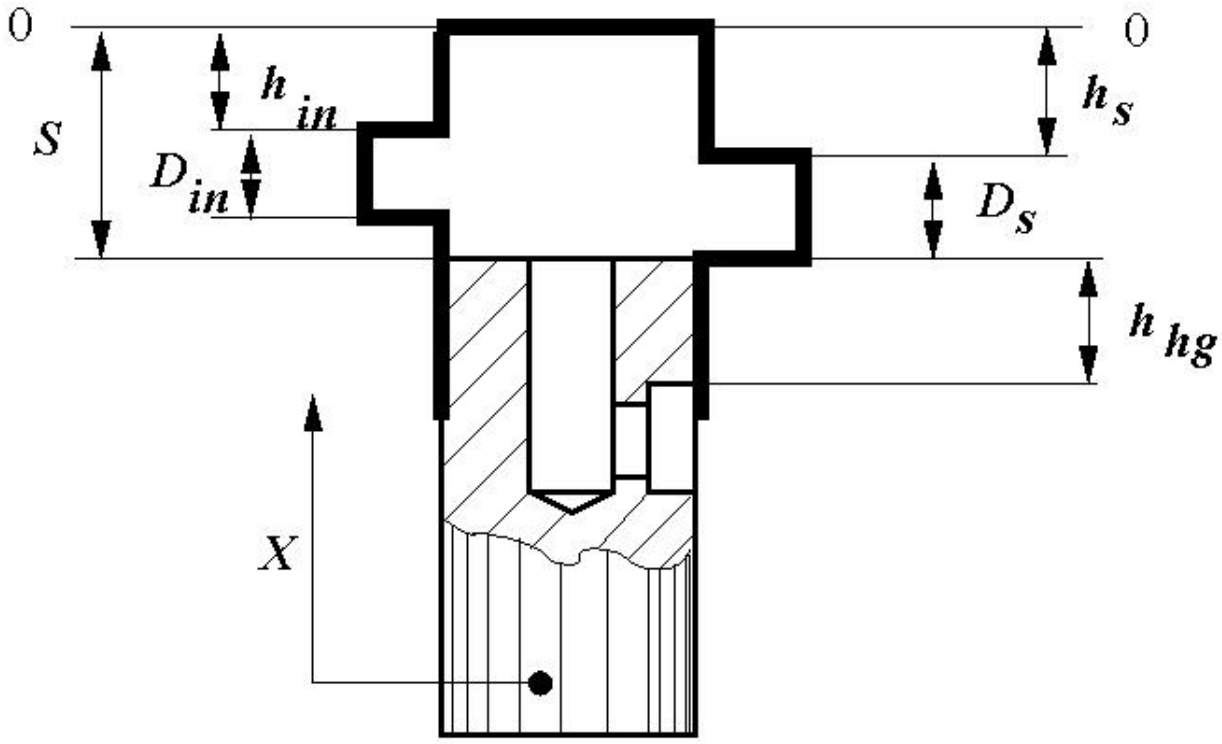
The injection stops as the helical groove formed on the side surface of the plunger reaches the Spill port, which connects the top chamber with the low pressure chamber through the orifice drilled inside the plunger (Figure 2,c). You can control the position of the helical groove with respect to the Spill port by rotating the plunger with the control fork, thus regulating the volume of fuel injected in the cylinder.

The model of the plunger control mechanism is based on the following assumptions:

1. There are three variable orifices in the control circuit: inlet port, spill port, and orifice formed by the helical groove and the spill port. The openings of the inlet and the spill orifices depend on the plunger motion, while the opening of the groove-spill port orifice is a function of the plunger motion and the plunger rotation. For the sake of simplicity, the displacement generated by the plunger

rotation is represented as a source of linear motion which is combined with the plunger displacement.

2. The figure below shows all the dimensions necessary to parameterize the orifices:



D_{in} - Inlet port orifice diameter

D_s - Spill port orifice diameter

S - Plunger stroke

h_{in} - Distance between the inlet orifice and the top plunger position

h_s - Distance between the spill port orifice and the top plunger position

h_{hg} - Distance between the spill port orifice and the upper edge of the helical groove

3. In assigning initial openings and orifice orientations, the plunger *top position is taken as the origin* and the motion in the upward direction is considered as the motion in positive direction. In other words, axis *X* is directed upward. Under these assumptions, the inlet and spill port orifice directions must be set to **Opens in negative direction**, while the groove-spill port orifice must be set to **Opens in positive direction** since it opens when the plunger moves upward. The table below shows the values assigned to initial openings and orifice diameters.

Notation	Name in parameter file	Value	Remarks
S	stroke	0.01 m	

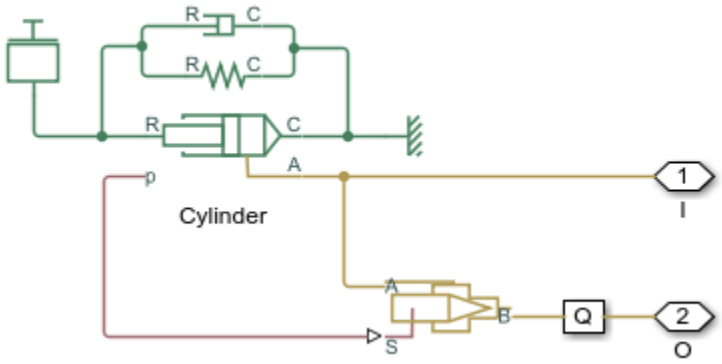
```
D_in       inlet_or_diameter                      0.003 m
D_s        spill_or_diameter                      0.0024 m
h_in       -stroke + inlet_or_diameter + 0.001             The inlet orifice is shifted upward f
h_s        -stroke + spill_or_diameter
h_hg       spill_or_diameter                               The spill orifice is assumed to be fu
```

4. The plunger effective stroke equals to

$$estroke = stroke - D_{in} - 1mm$$

The inlet orifice is generally located higher than the spill orifice. In the example, this distance is set to 1 mm. By rotating the plunger, you change the initial opening of the groove-spill port orifice. Since initial opening is a parameter and cannot be dynamically changed, the shift of the initial opening is simulated by addition of an equivalent linear displacement of the orifice control member. The larger the equivalent signal, the sooner the spill orifice is opened, thus decreasing the volume of fuel delivered to the cylinder. The maximum value of the equivalent signal is equal to the effective stroke. At this value, the spill orifice remains open all the time.

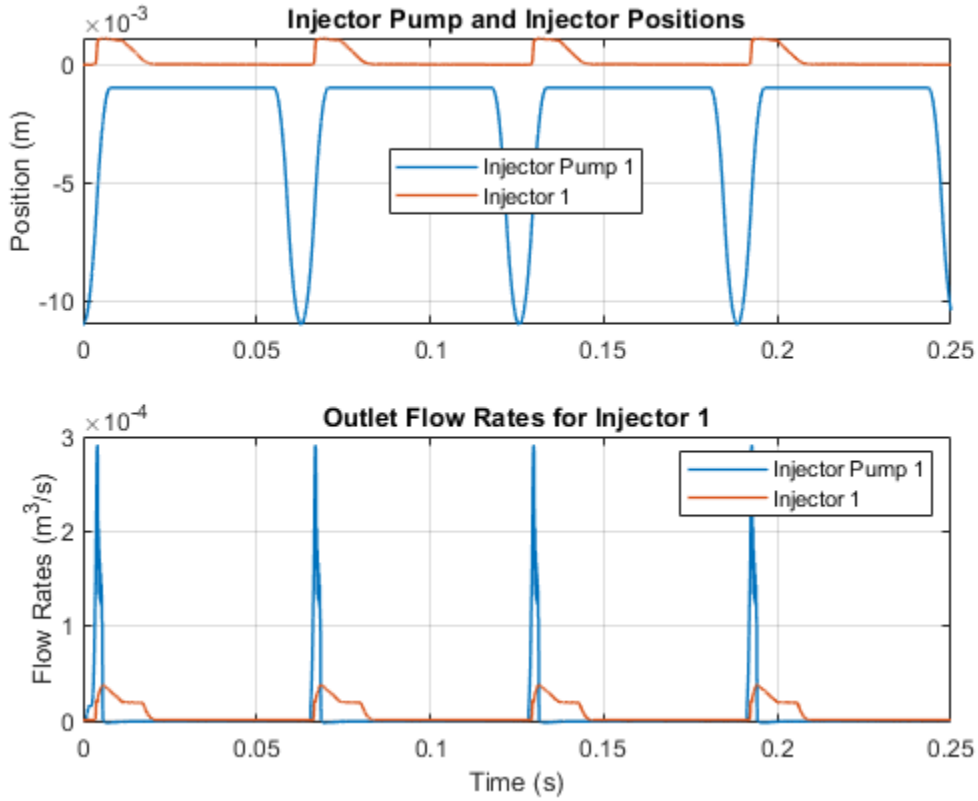
Injector



The model of the injector is based on the Single-Acting Hydraulic Cylinder block and the Needle Valve block. The needle valve is closed at initial position by the force developed by the preloaded spring. As the force developed by the cylinder overcomes the spring force, the injector opens and allows fuel to be injected in the cylinder. In the example, the injector is set to be opened at 1000 bar.

Simulation Results from Simscape Logging

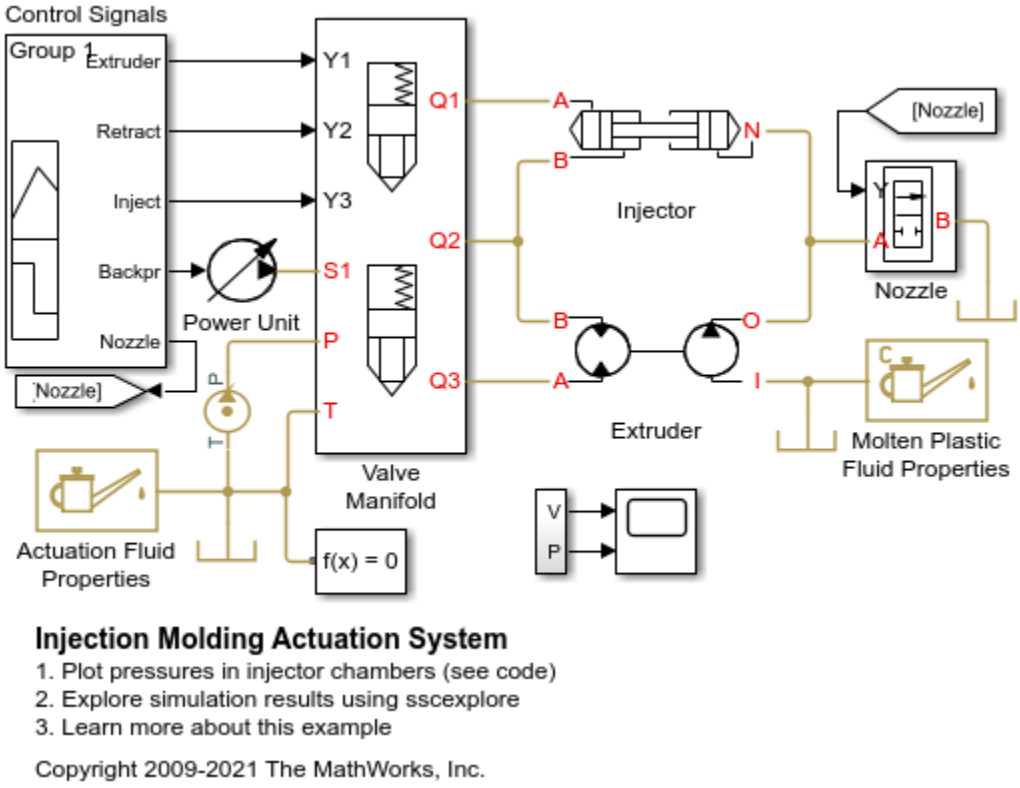
The plots below show the positions and outlet flow rates of injector pump 1 and injector 1. The effect of the cam profile is shown in the displacement of injector pump 1. During the second half of the cam stroke, fuel exits the injector pump and passes into the injector. The fuel exits the injector via a needle valve. The injector has chamber with a preloaded spring which acts to store the fluid from the pump temporarily and push it out of the injector more smoothly.


Injector Pump and Injector Positions
×10^-3
Position (m)
0
-5
-10
Injector Pump 1
Injector 1
0
0.05
0.1
0.15
0.2
0.25
Outlet Flow Rates for Injector 1
×10^-4
Flow Rates (m^3/s)
3
2
1
0
Injector Pump 1
Injector 1
0
0.05
0.1
0.15
0.2
0.25
Time (s)

Injection Molding Actuation System

This example shows an injection molding actuation system. The model contains a set of cartridge valves that control pumps, motors, and cylinders to execute the steps of an injection molding process.

Model



System Schematic

The system presented in the example is the actuation system of the molding machine injection unit described in full detail in the *Vickers Industrial Hydraulics Manual* (second edition, 1989), and shown in the schematic diagram below.

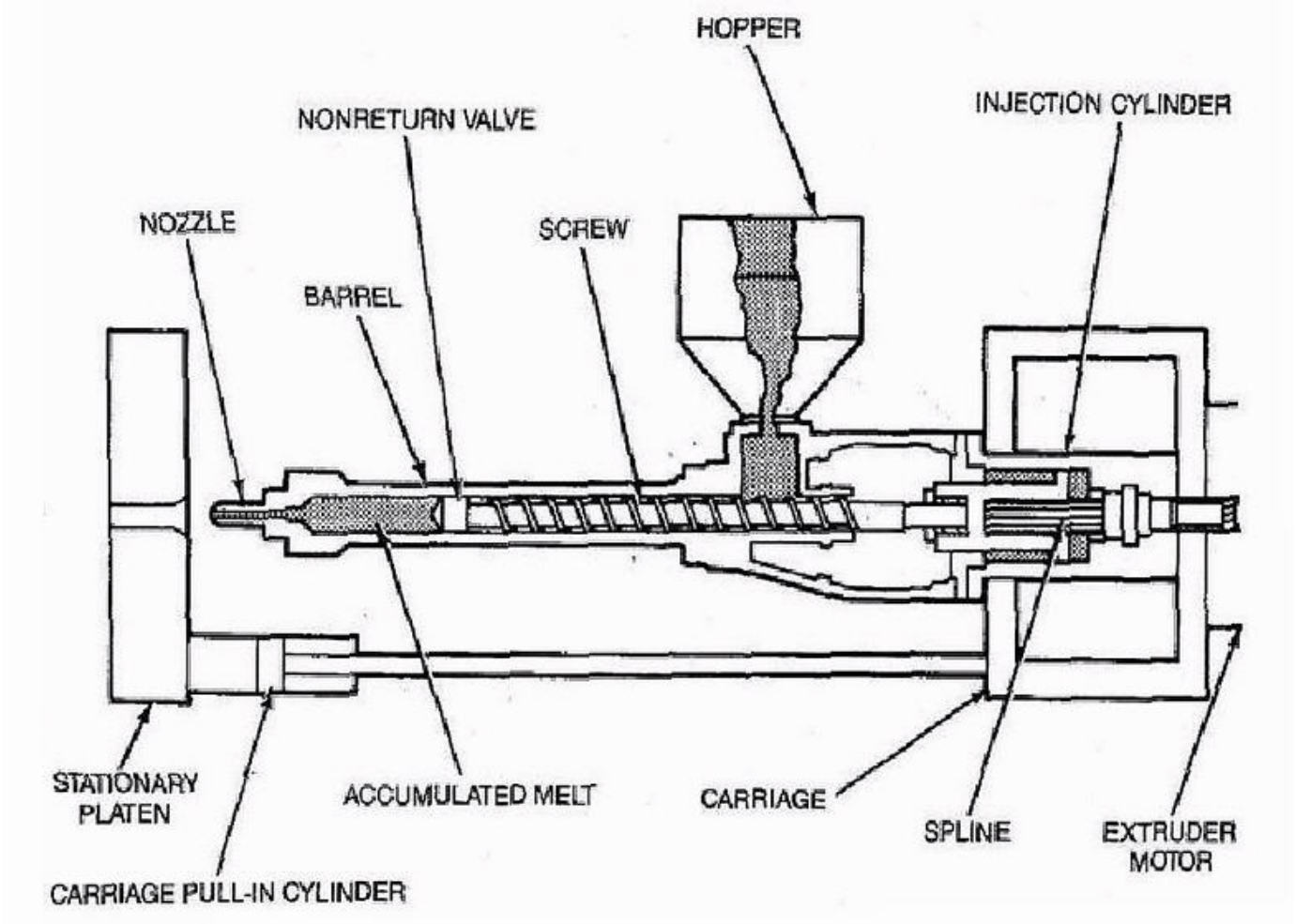


Figure 1. Working Members of the Molding Machine Injection Unit

The unit consists of a screw and barrel, hopper, extruder motor, and injection cylinder, which are all mounted on a carriage. The screw, extruder motor, and injection cylinder are arranged in a single unit that can slide along the carriage to eject the molten plastic from the barrel cylinder into the mold. The cycle considered in the example starts with the extruder rotation, during which molten plastic is pushed forward by the screw through the nonreturn (check) valve. The rotation is stopped when preset volume of molten plastic has been accumulated in the front chamber of the barrel. The injection maintains specified back-pressure while the chamber is filled and is pushed back by the molten plastic.

The next operation following the extruder run is called decompression. During this phase, the active chamber of the injection cylinder is decompressed by connecting it to the tank. The injector unit pauses after decompression for 2 seconds to allow the previously molded and hardened part to be removed from the mold. After the mold is closed, the injection cylinder goes forward and injects molten plastic into the mold.

Hydraulic Schematic

The schematic diagram of the unit hydraulic system is shown in Figure 2.

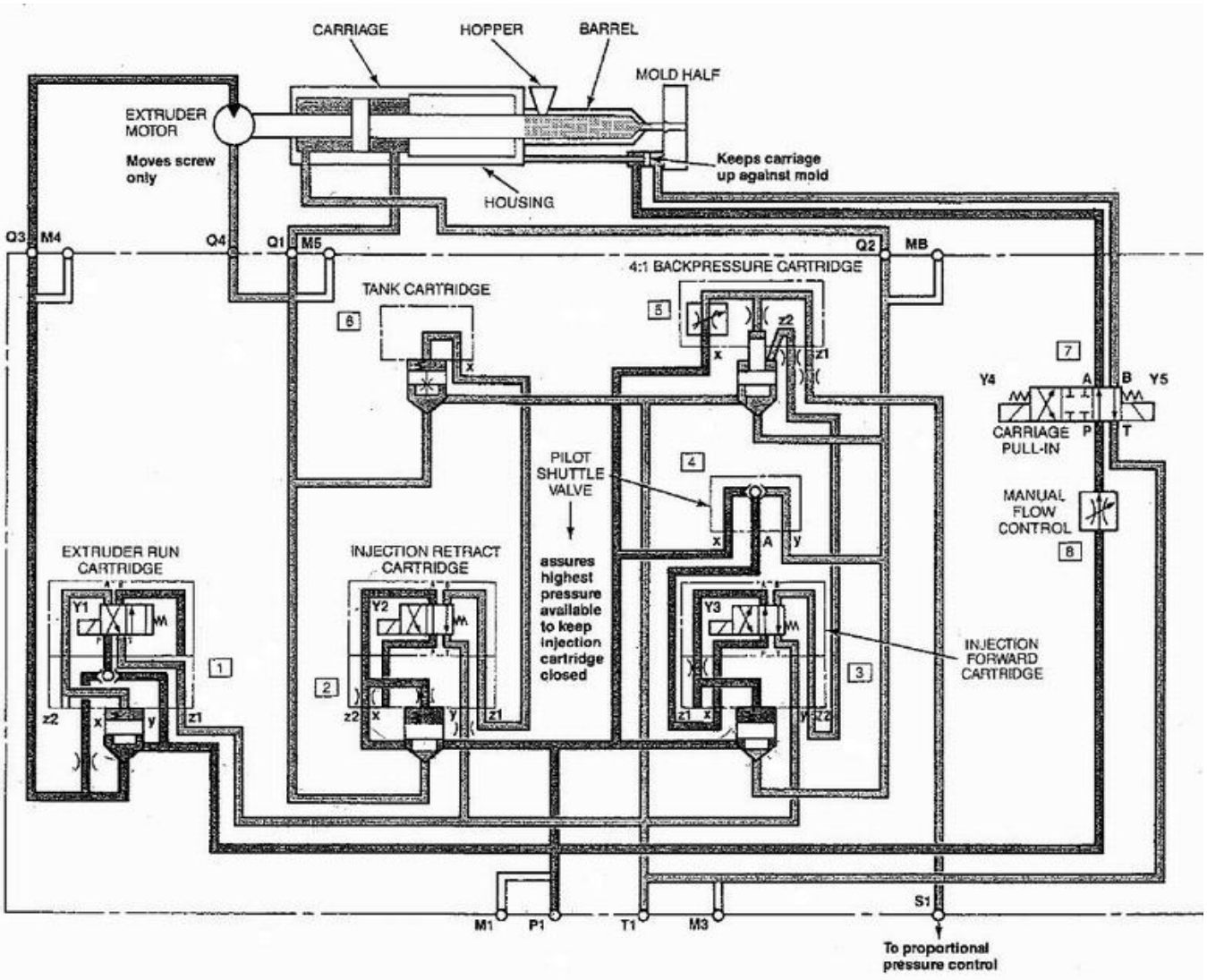
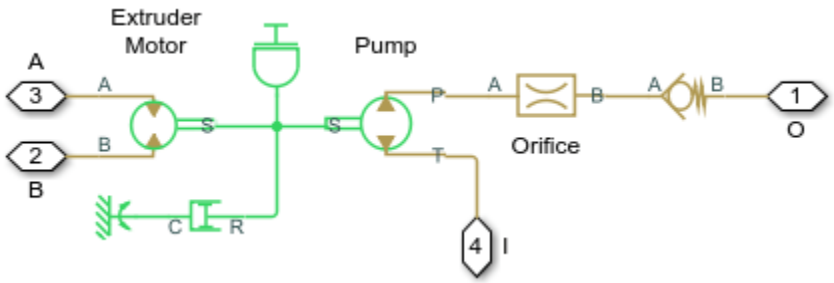
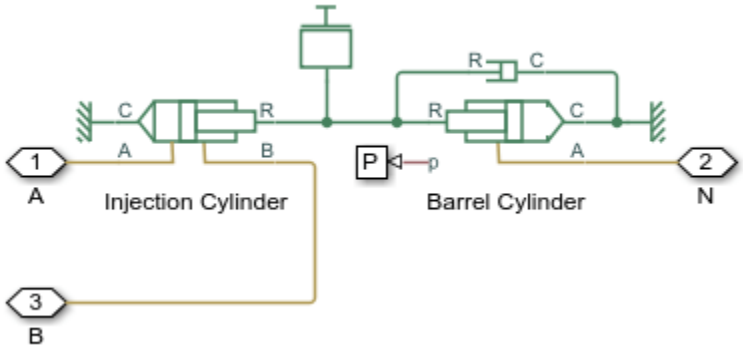


Figure 2. Injection Unit Hydraulic System Schematic Diagram

Extruder Subsystem

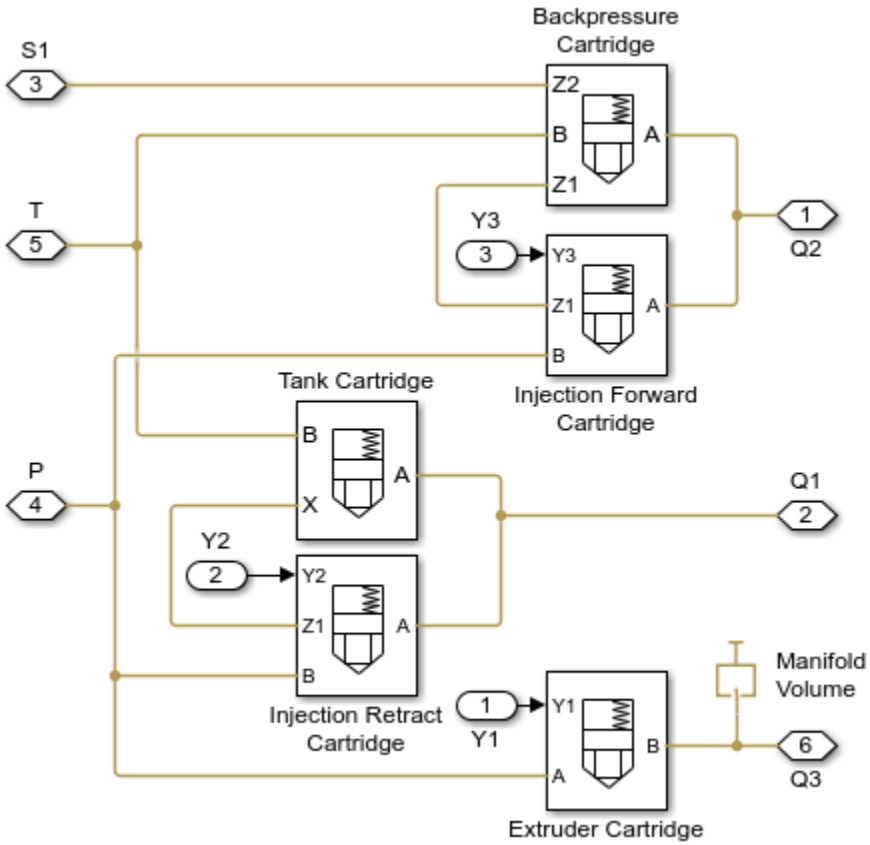


Injector Subsystem



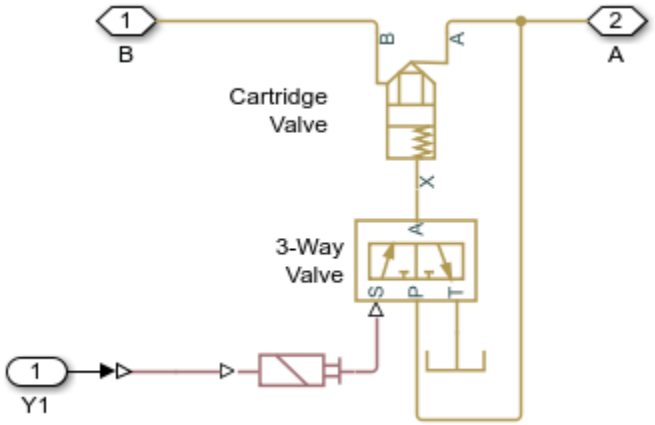
Valve Manifold Subsystem

The system is built of five cartridge valves that are arranged into the injection unit manifold:



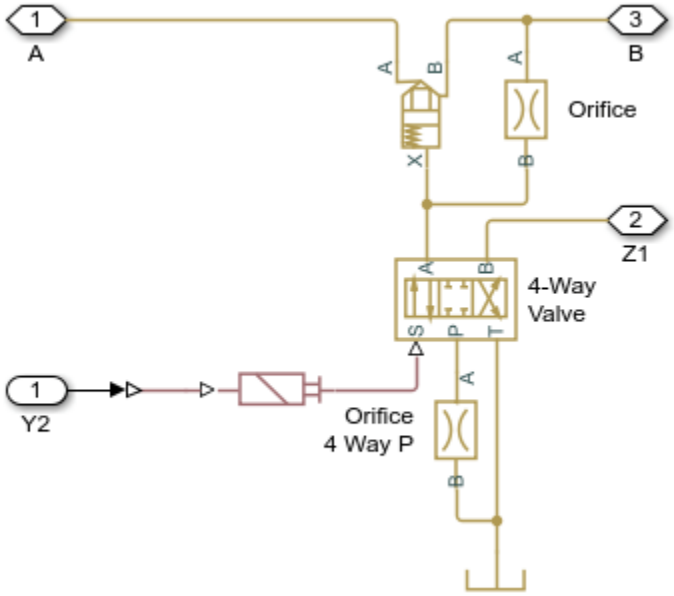
Extruder Cartridge Valve

The extruder cartridge valve controls rotation of the extruder motor. The motor starts rotating as electromagnet Y1 is energized.



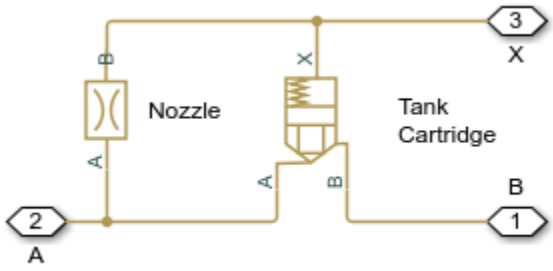
Injection Retract Cartridge Valve

The injection retract cartridge retracts the nozzle from the mold for a very short distance to allow the mold to be opened and the previously molded and hardened part to be removed. The cartridge poppet is opened as electromagnet Y2 is energized. When the electromagnet is deenergized, the poppet is closed and the control chamber of the tank cartridge valve is vented to allow free return flow from the extruder motor.



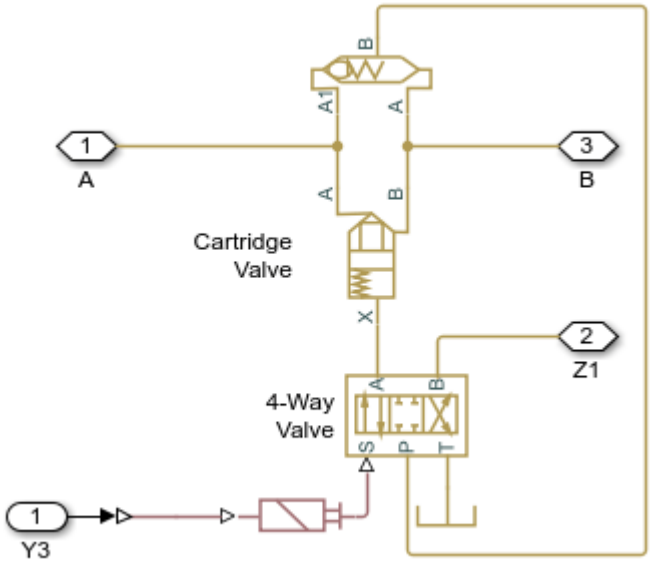
Tank Cartridge Valve

The tank cartridge valve connects extruder motor to the tank during the barrel fill



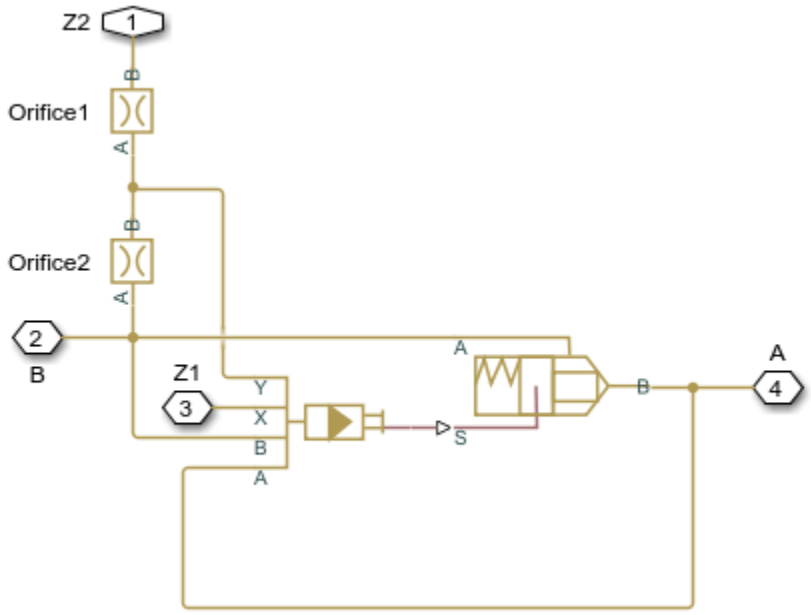
Injection Forward Cartridge Valve

The injection forward cartridge valve pushes the nozzle forward to contact the mold and injects molten plastic into the mold. The cartridge is controlled by electromagnet Y3. At neutral position, one of the control chambers of the back-pressure cartridge is vented to allow this cartridge to work as a pressure relief valve and maintain pressure set by the supply pressure at port Z1 and openings of orifices in the back-pressure cartridge covers.



Backpressure Cartridge Valve

The back-pressure cartridge valve maintains low back-pressure in the injection cylinder chamber to assure required quality of the molten plastic. The cartridge poppet is opened when electromagnet Y3 is deenergized. Simultaneously, the poppet of the back-pressure cartridge is closed by pressurizing the previously vented control chamber.



Control Signals

There are five signals that control this system:

1. Extruder - electromagnet Y1 signal

2. Retract - electromagnet Y2 signal

3. Inject - electromagnet Y3 signal

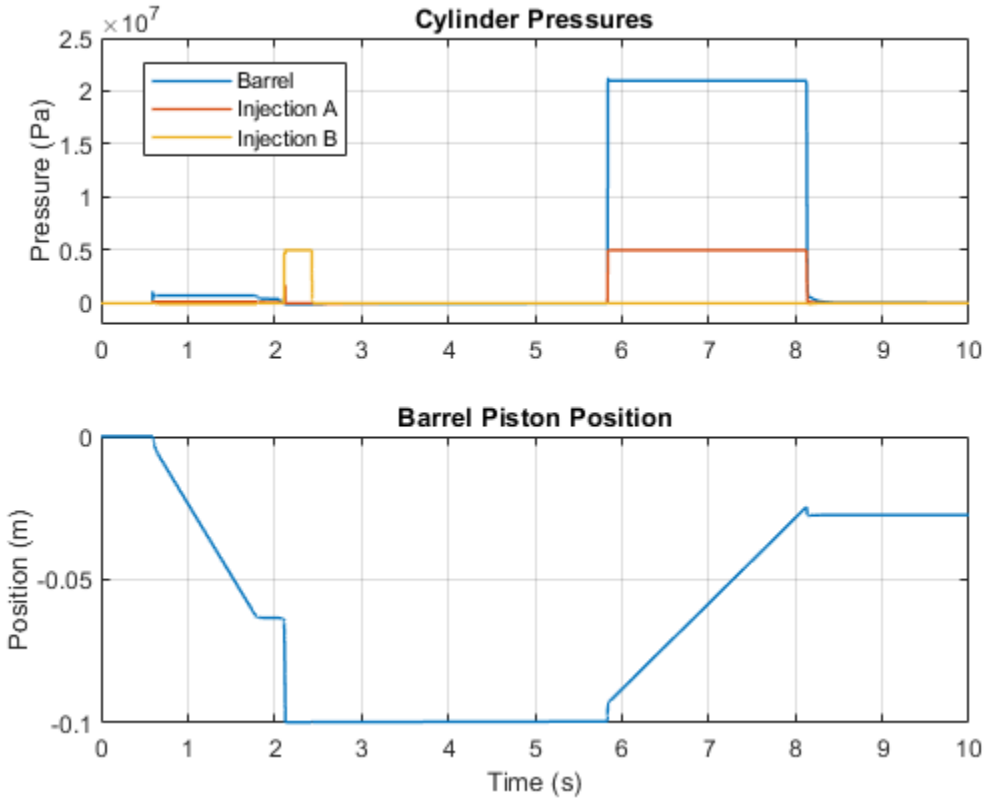
4. Nozzle - simulates the changing gap between the nozzle and the mold as a 2-way valve. In a real machine, the nozzle remains in contact with the mold while the extruder runs and the front portion of the barrel is filled. This contact prevents molten plastic from the spill. Since no mold is considered in the model, the nozzle is kept closed with the 2-way valve controlled by the signal 'Nozzle'. The valve is opened as the nozzle is moved away from the mold to keep things close to the real behavior.

5. Backpr - controls pressure level at port S1 (Figure 2) which establishes the pressure maintained by the back-pressure cartridge.

The purpose of remaining blocks is explained on the model schematic diagram (Figure 1).

Simulation Results from Simscape Logging

The plots below show the pressures in the injector chambers and the position of the barrel piston. The retract, decompression, and injection phases can be identified by the pressures and positions in the system.


Cylinder Pressures
×10^7
2.5
2
1.5
1
0.5
0
Pressure (Pa)
Barrel
Injection A
Injection B
0 1 2 3 4 5 6 7 8 9 10
Barrel Piston Position
0
-0.05
-0.1
Position (m)
0 1 2 3 4 5 6 7 8 9 10
Time (s)

Drill-Ream Actuator

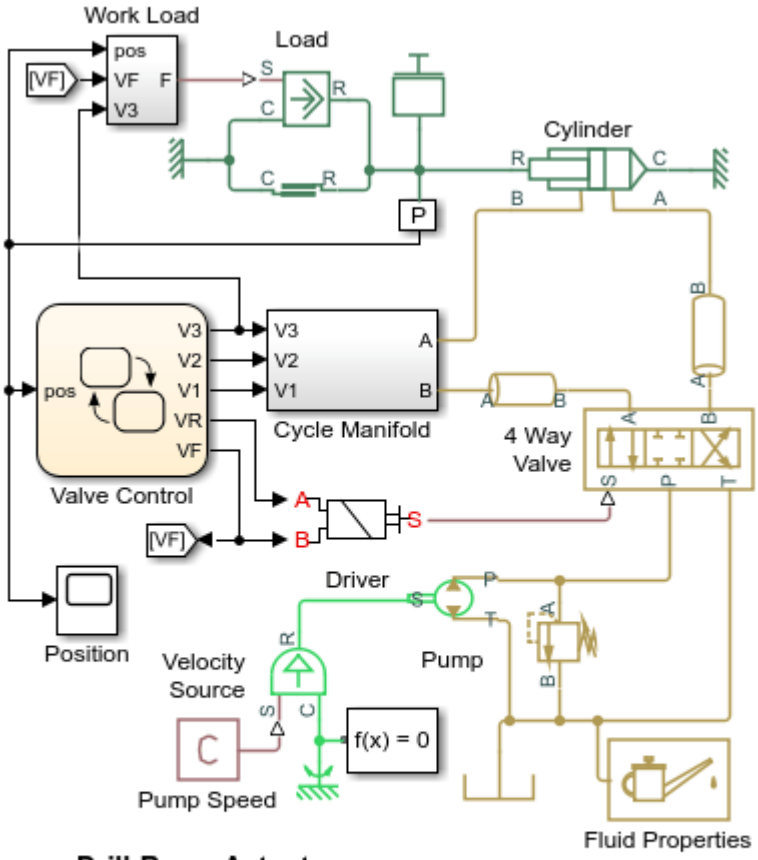
This example shows an actuator that drives a machine tool working unit performing a sequence of three technological operations: coarse drilling, fine drilling, and reaming. The actuator speed is controlled by one of three pressure-compensated flow control valves metering out return flow from the cylinder. The selection of an appropriate flow control is performed by directional valves that are activated by a control unit.

From the Valve Control, signals VF and VR are the signals that activate forward and return motions, respectively, by shifting the main directional valve. Signal V1 activates normally- closed valve Valve1 Retract. When opened, Valve1 Retract bypasses all flow controls and sets the fast traverse speed. V2 controls valve Valve2 Ream. When deenergized, this valve directs return flow through Flow Control Reaming, which is set to reaming speed. Signal V3 directs flow either through Flow Control Fine Drilling or Flow Control Coarse Drilling.

The load is simulated as a position-dependent force using a table lookup block.

Flow control valves are arranged as a subsystem comprised of a fixed orifice and a pressure compensator.

Model

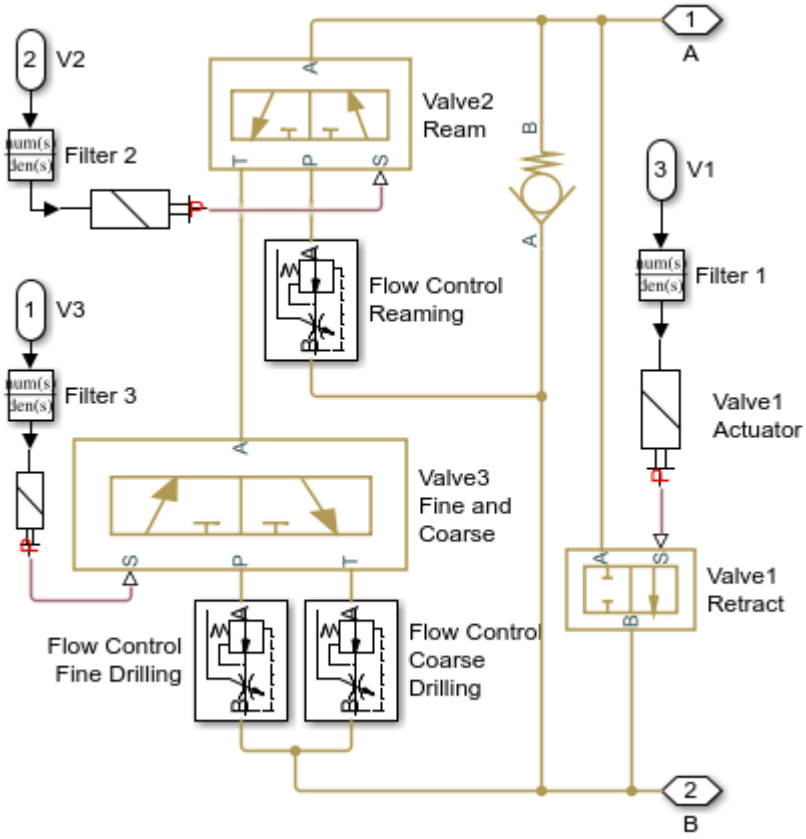


Drill-Ream Actuator

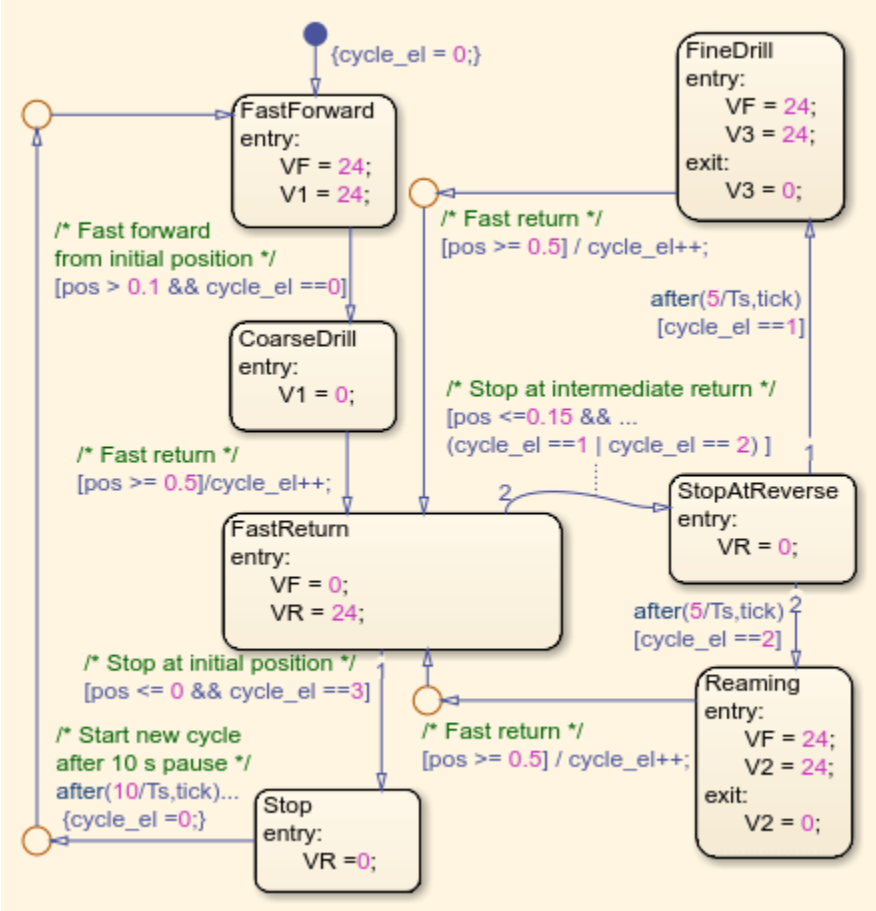
1. Plot pressures in cylinder (see code)
2. Plot valve states (see code)
3. Explore simulation results using sscexplore
4. Learn more about this example

Copyright 2005-2021 The MathWorks, Inc.

Cycle Manifold Subsystem

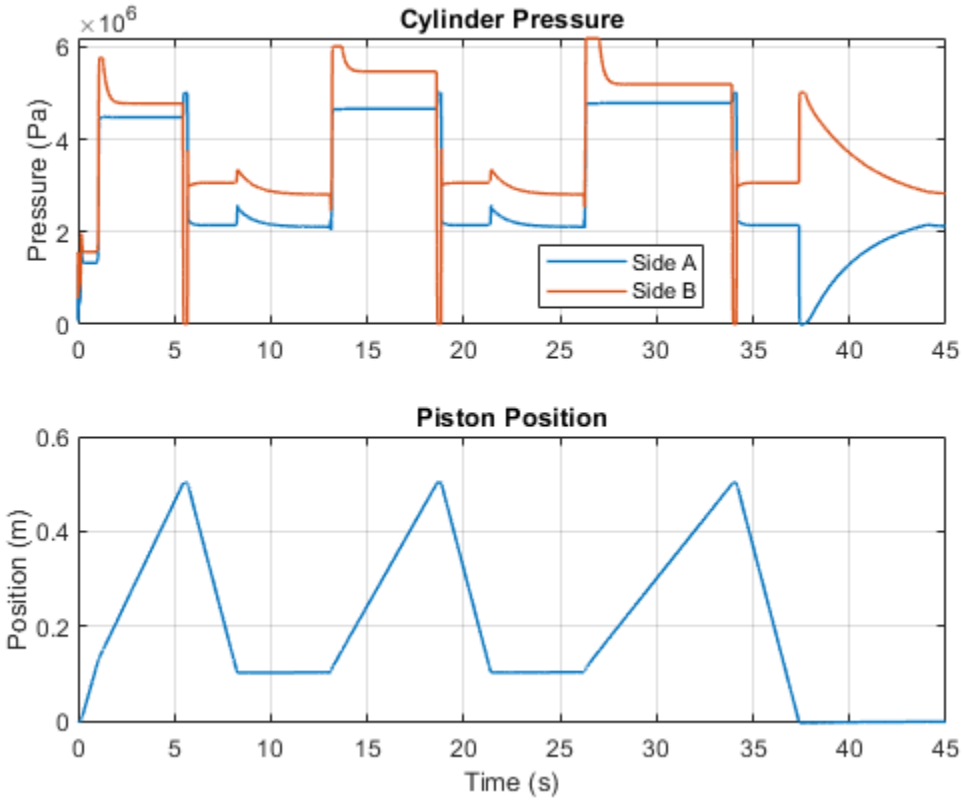


Valve Control Subsystem

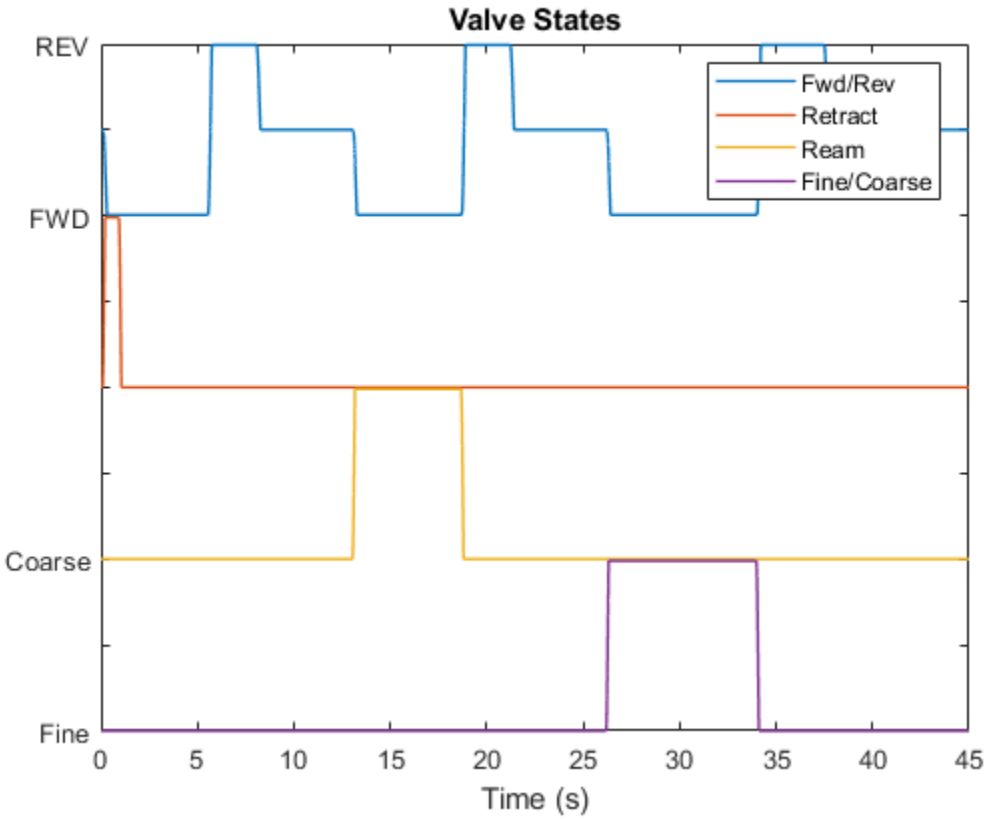


Simulation Results from Simscape Logging

The plots below show the pressures in the cylinder and the position of the piston. Three operating speeds are shown in the plots.



The plots below show the states of the different valves. Forward/Reverse has 3 states (Forward/Locked/Reverse), while the others have two states. Retract and Ream have states of on/off, and Fine/Coarse valve selects between two different flow restrictions.


Valve States
REV
FWD
Coarse
Fine
0
5
10
15
20
25
30
35
40
45
Time (s)
Fwd/Rev
Retract
Ream
Fine/Coarse

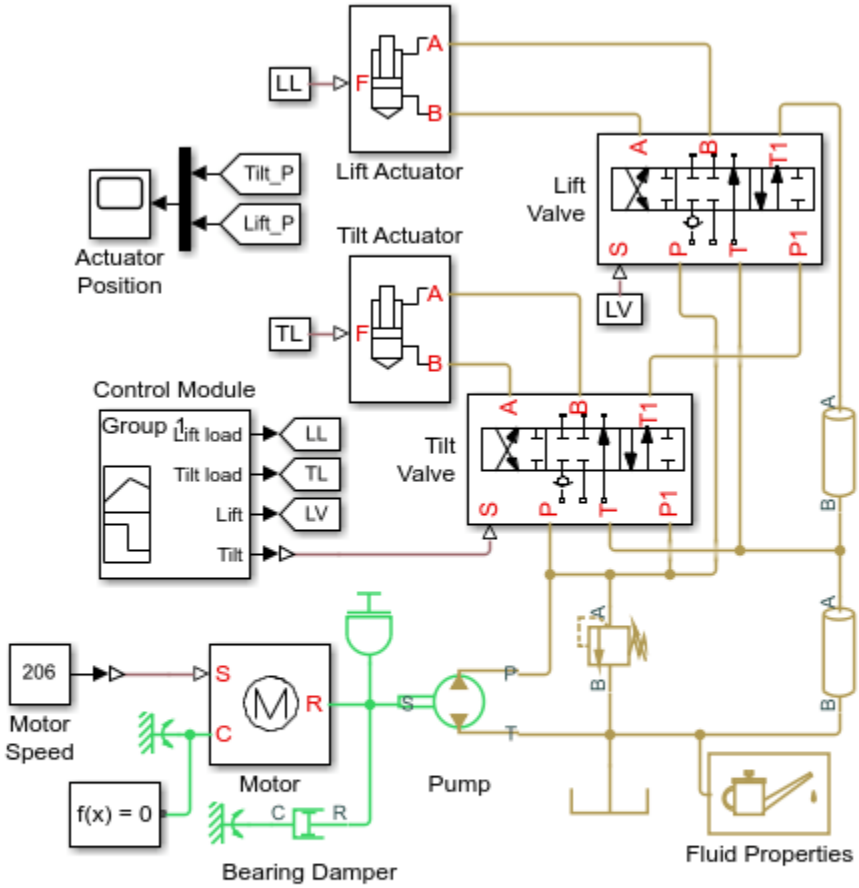
Front-Loader Actuation System

This example shows a simplified version of an actuation system consisting of the lift and tilt cylinders. Each cylinder is controlled by an open center, 6-way, 3-position directional valve. The valves are connected in series through their unloading branch such that the system pump is unloaded when both command levers are in neutral position. If either tilt or lift command is applied, the unloading path is closed.

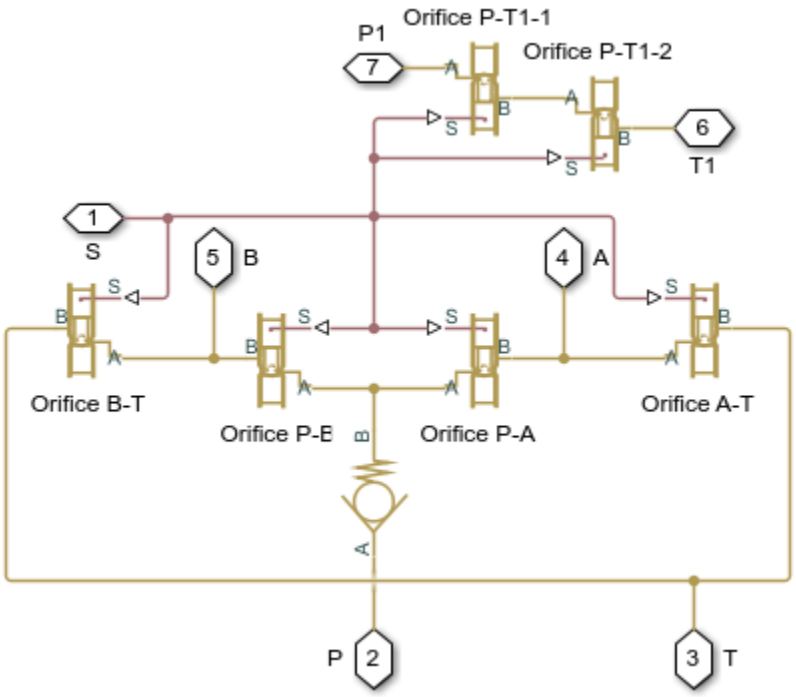
The pump is driven by a diesel engine simulated with a model that accounts for velocity change caused by the load on the output shaft. The directional valve is a custom model composed of Orifice with Variable Area Round Holes blocks and a Check Valve block.

A typical cycle of a front-loader actuation system is simulated with all the motion and load commands generated as a function of time.

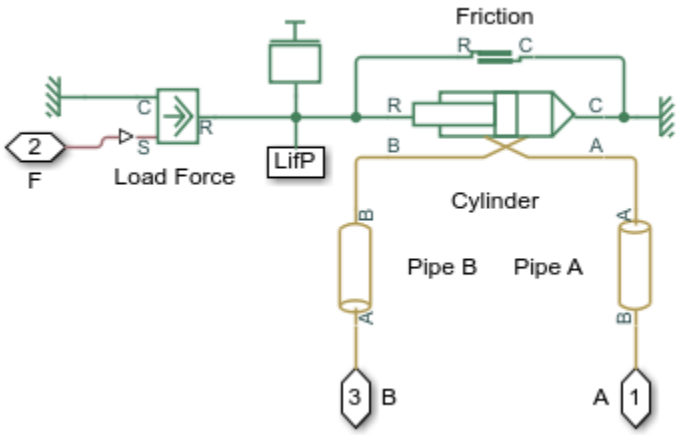
Model



Lift Valve Subsystem

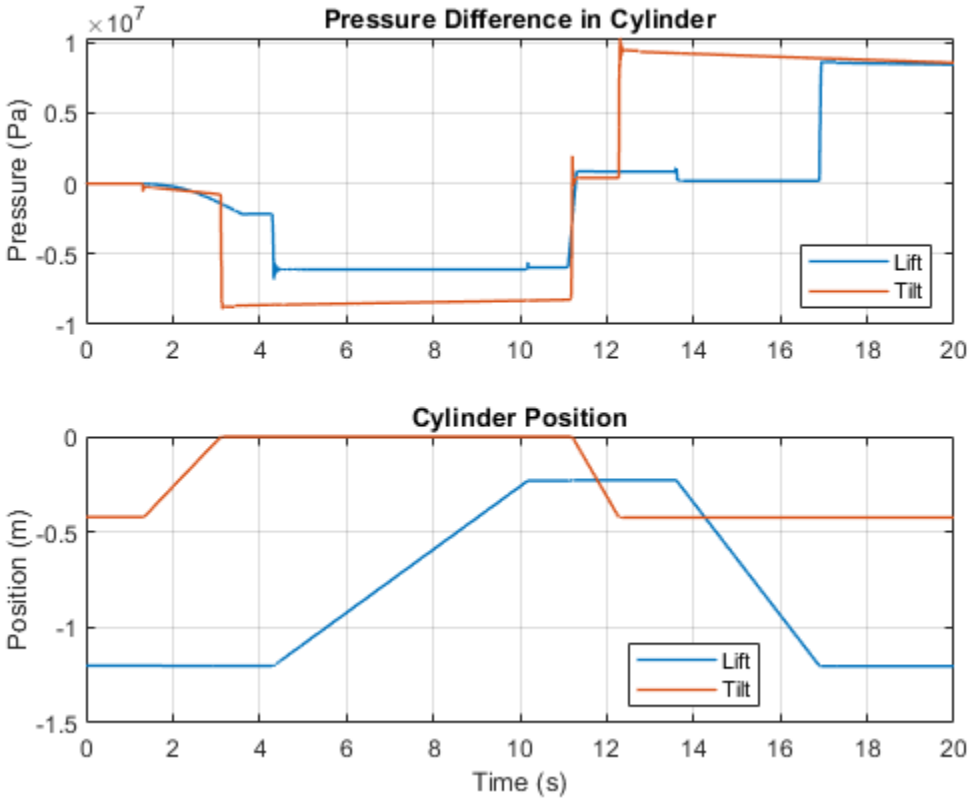


Lift Actuator Subsystem



Simulation Results from Simscape Logging

The plots below show the pressure difference in the lift and tilt cylinders and the position of the in each cylinder.


Pressure Difference in Cylinder
×10^7
Pressure (Pa)
1
0.5
0
-0.5
-1
0
2
4
6
8
10
12
14
16
18
20
Lift
Tilt
Cylinder Position
Position (m)
0
-0.5
-1
-1.5
Time (s)
Lift
Tilt

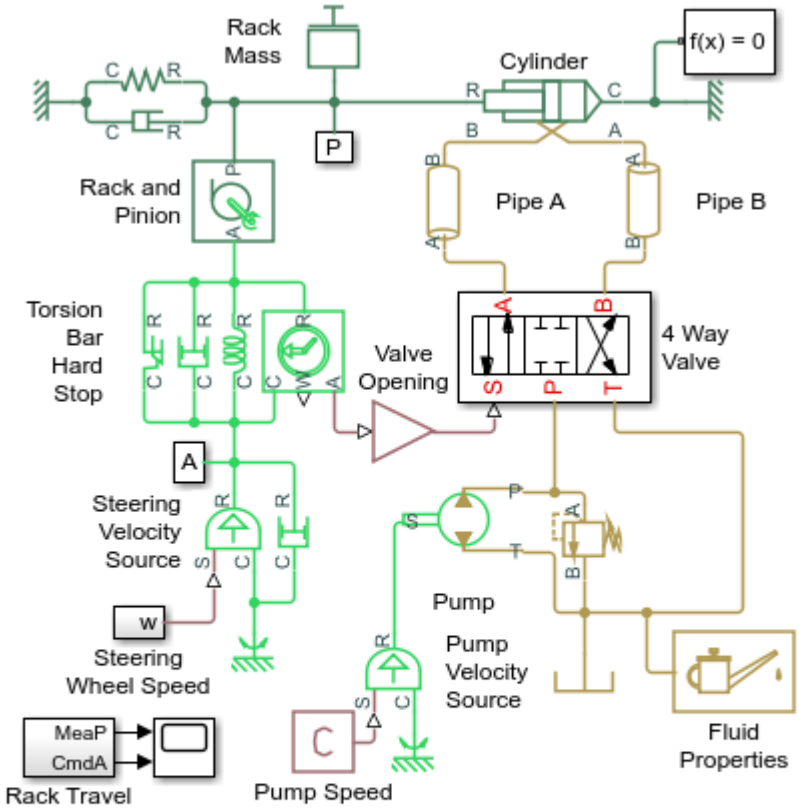
Power-Assisted Steering Mechanism

This example shows a simplified version of a power-assisted steering mechanism. The hydraulic actuation system includes a double-acting hydraulic cylinder, 4-way valve, fixed-displacement pump, and a pressure-relief valve. The steering rack acts against a load modeled by a spring and damper.

The valve opens based on the amount of twist of a torsion bar installed between the steering wheel and the pinion of a rack-and-pinion mechanism. The rotation of the steering wheel causes the torsion bar to twist relative to the pinion. The deformation of the bar is transformed into opening of the 4-way valve, which connects the chambers of the cylinder to pressure or return lines depending upon the direction of rotation. The PS Gain block Valve Opening converts torsion bar twist to the amount of opening for the 4-way valve. Setting the Gain parameter of this block to 0 disables the power steering.

If the torsion bar deformation exceeds 9 degrees the wheel is connected directly to the pinion through the hard stops installed in parallel with the torsion bar. The cylinder moves the steering rack which twists the torsion rod in the opposite direction until the valve is in the neutral position.

Model

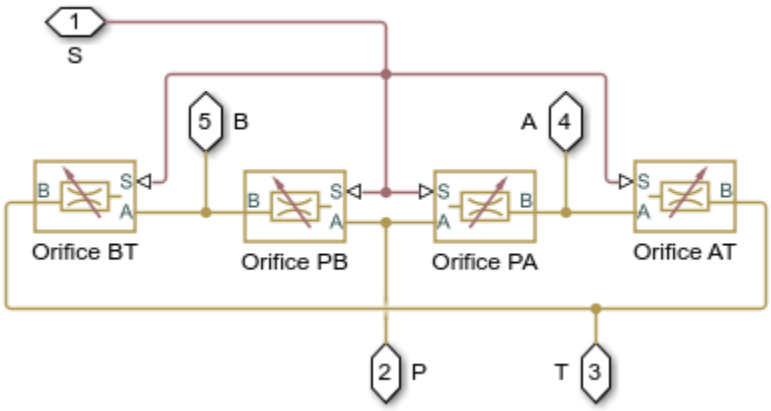


Power-Assisted Steering Mechanism

1. Plot pressures in cylinder (see code)
2. Compare effort with power steering on and off (see code)
2. Explore simulation results using sscexplore
3. Learn more about this example

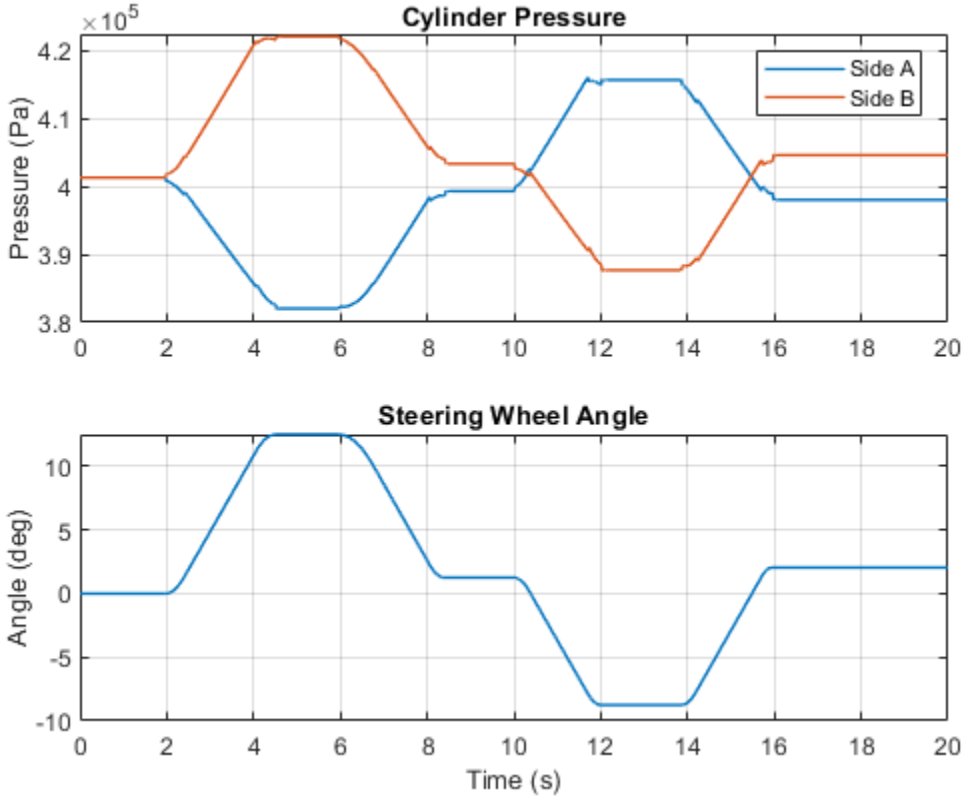
Copyright 2006-2021 The MathWorks, Inc.

4 Way Valve Subsystem

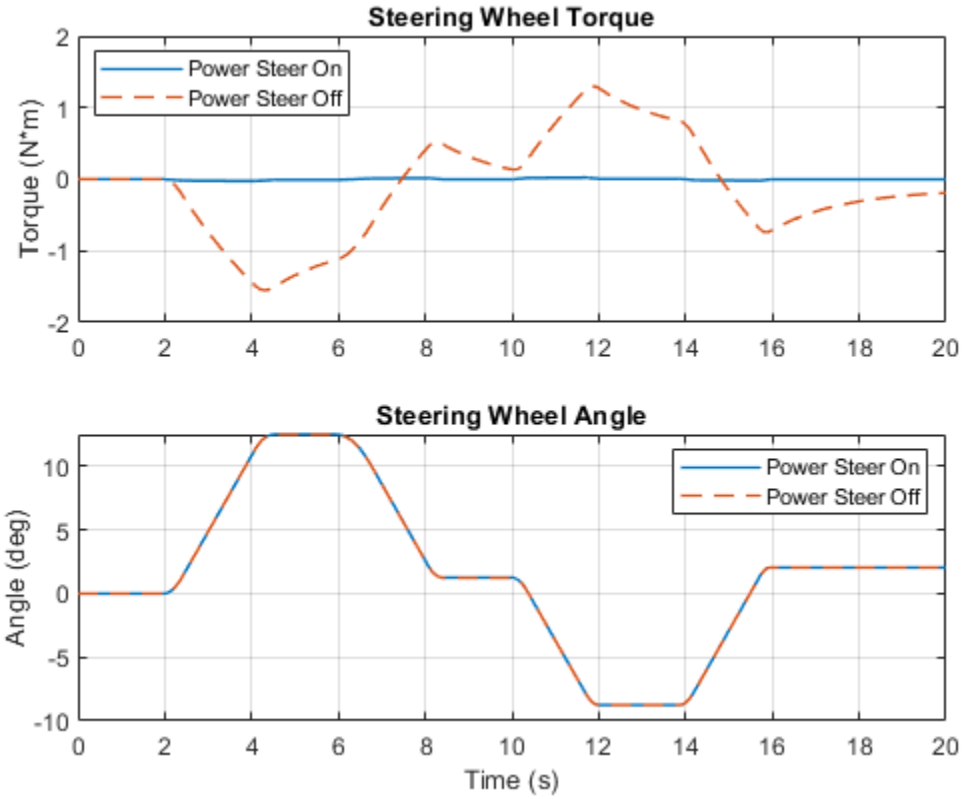


Simulation Results from Simscape Logging

The plots below show the amount of assistance the power steering system provides to the driver as the wheel is turned. The greater the difference in the cylinder chamber pressures, the more assistance that is provided.



The plots below compare the amount of effort required when the power steering is turned on and turned off. For the same steering sequence, the amount of torque required when the power steering is turned off is much higher. The power steering is turned off by setting the gain in Valve Opening to 0.

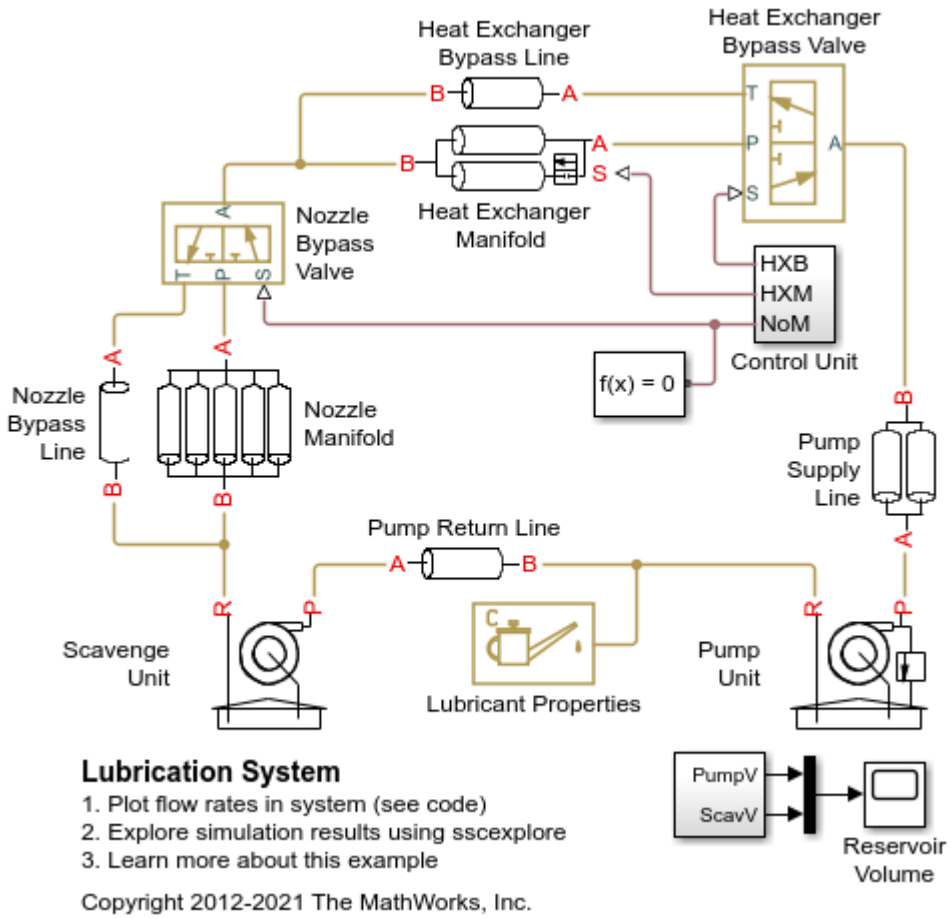

Steering Wheel Torque
Power Steer On
Power Steer Off
Torque (N*m)
2
1
0
-1
-2
0
2
4
6
8
10
12
14
16
18
20
Steering Wheel Angle
Power Steer On
Power Steer Off
Angle (deg)
10
5
0
-5
-10
Time (s)

Lubrication System

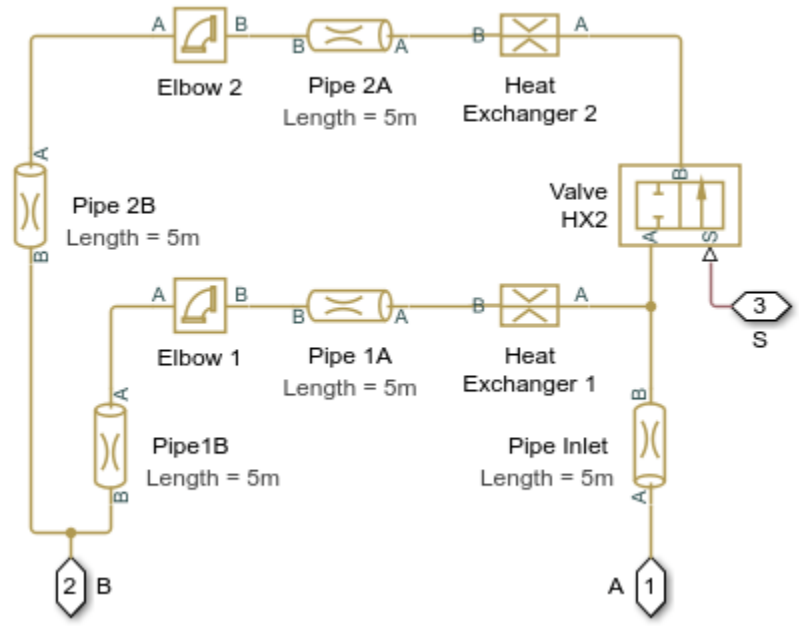
This example shows a simplified version of a lubrication system fed with the centrifugal pump. The system consists of five major units: Pump Unit, Scavenge Unit, Heat Exchanger Manifold, Nozzle Manifold, and Control Unit. Both the pump and the scavenge unit are built around the centrifugal pump. The Scavenge Unit collects fluid discharged by nozzles and pumps it back into the reservoir of the Pump Unit. The Control Unit generates commands to bypass either the heat exchanger, represented as a local resistance, or the nozzles block. In a real system, these commands are generated by temperature sensors installed in lubrication cavities.

The cycle simulated in the example allows you to observe pressures and flow rates at various points of the system. The example also shows the usage of local resistance blocks such as the Elbow, T-junction, and Sudden Area Change.

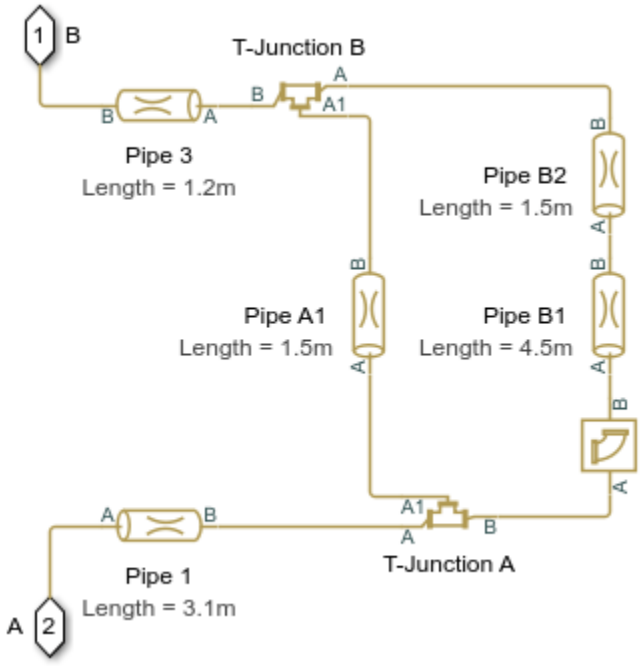
Model



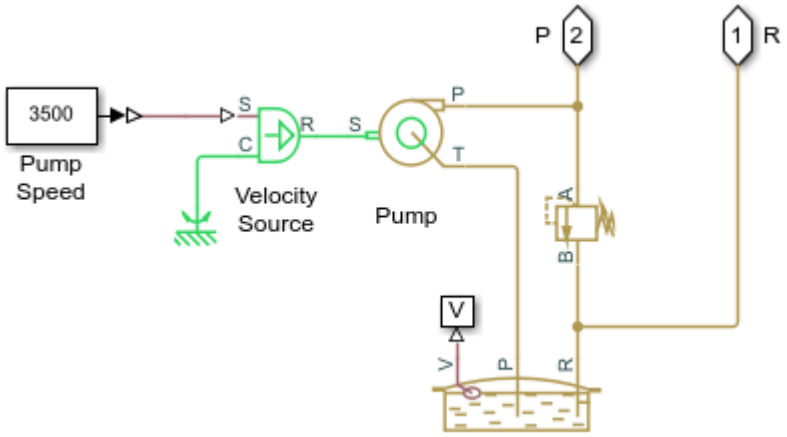
Heat Exchanger Manifold Subsystem



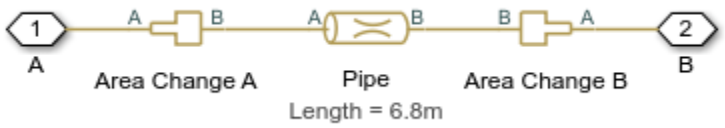
Pump Supply Line Subsystem



Pump Unit Subsystem

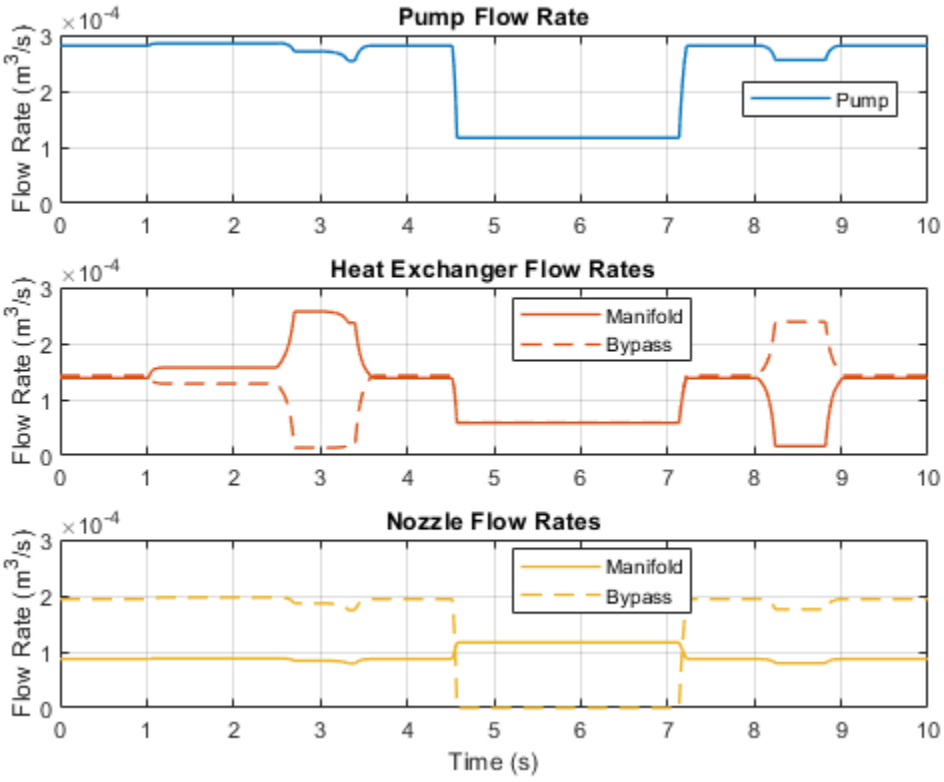


Pump Return Line Subsystem



Simulation Results from Simscape Logging

The plots below show the flow rates through different parts of the lubrication system. As the control member within the different valves are adjusted, flow rates change within the system.


Pump Flow Rate
×10^-4
Flow Rate (m^3/s)
Pump
Heat Exchanger Flow Rates
Manifold
Bypass
Nozzle Flow Rates
Manifold
Bypass
Time (s)

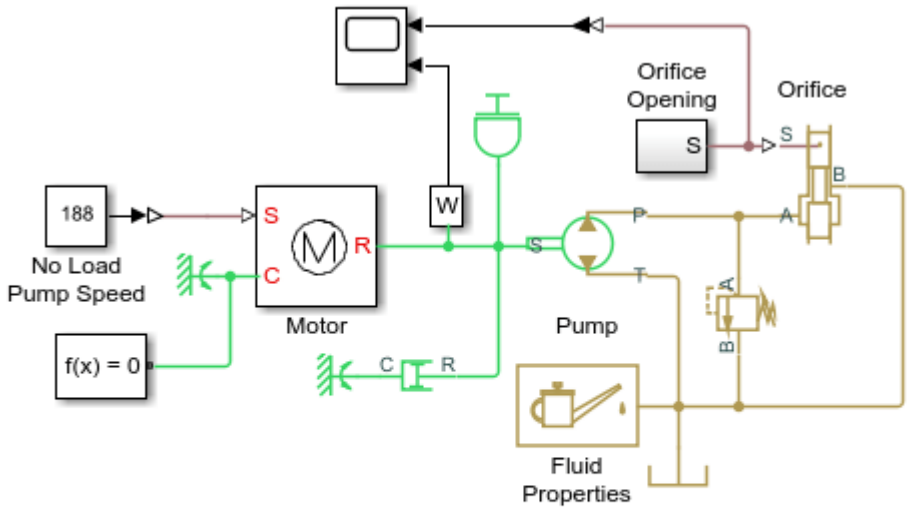
Closed-Loop Pump Control with Flexible Drive Shaft

This example shows a typical power unit consisting of a fixed-displacement pump driven by a motor through a flexible transmission and a pressure-relief valve. A variable orifice serves as a load for the system.

The motor model uses an Ideal Angular Velocity Source block. The load on the shaft decreases the velocity with a slip coefficient of 1.2 (rad/s)/Nm, resulting in a no-load speed for the motor of 188 rad/s. The load on the pump drive shaft is measured with the torque sensor. The shaft between the motor and the pump is assumed to be compliant and is simulated with a rotational spring and damper.

The simulation starts with the variable orifice open, which results in a low output pressure and the maximum flow rate going to the system. The orifice starts closing at 0.5 seconds and is closed completely at about 2.75 seconds. The output pressure builds up until it reaches 75e5 Pa, and is maintained at this level by the pressure-relief valve. Just after 3.2 seconds, the valve starts opening and the system returns to its initial state.

Model

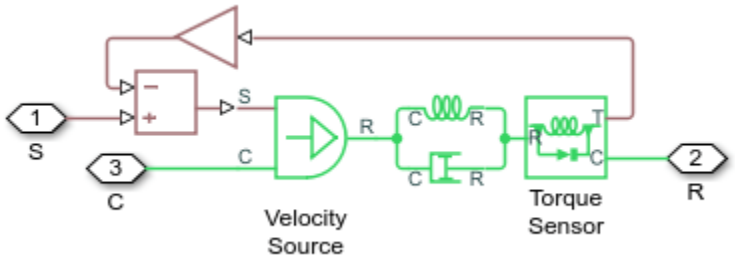


Closed-Loop Pump Control with Flexible Drive Shaft

1. Plot flow rates in system (see code)
2. Explore simulation results using sscexplore
3. Learn more about this example

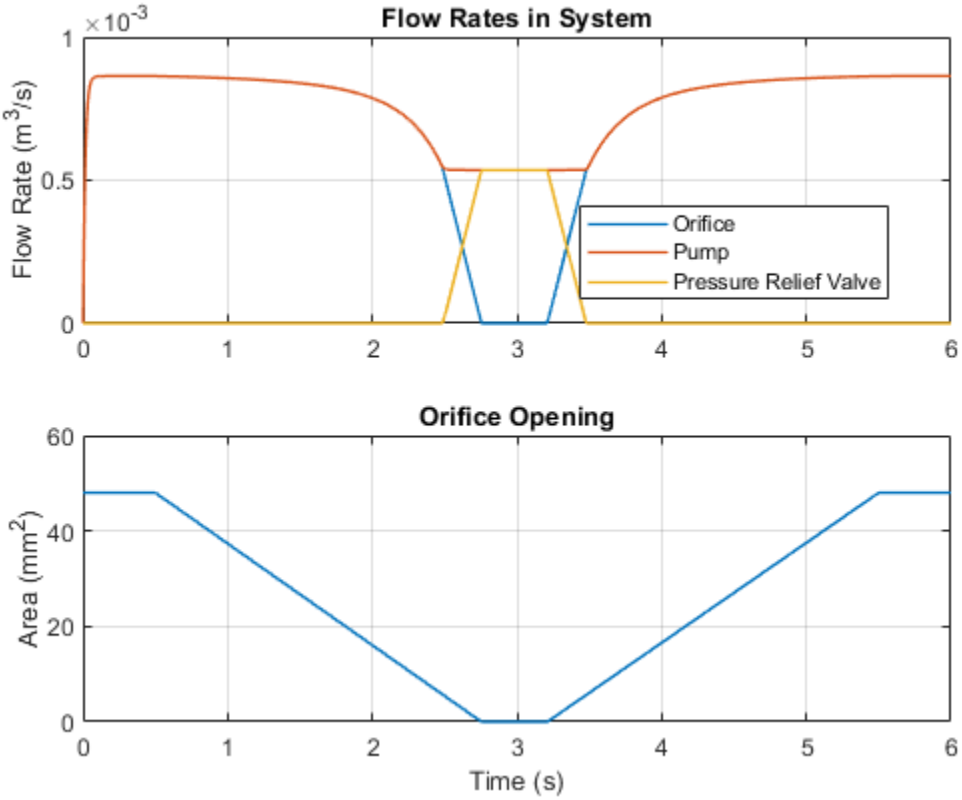
Copyright 2006-2021 The MathWorks, Inc.

Motor Subsystem



Simulation Results from Simscape Logging

The plots below show how flow rates vary in the system as the orifice is closed. All of the fluid exiting the pump goes through the orifice at the start. As the orifice is closed, flow gradually ends up being diverted through the pressure relief valve. Flow resumes through the orifice as it reopens.



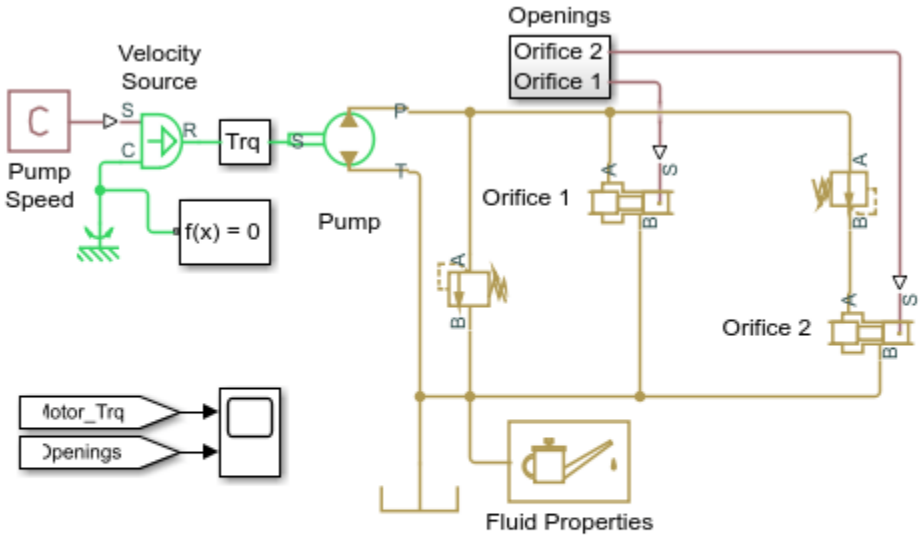
Pump with Pressure-Reducing Valve and Two Loads

This example shows a typical power unit consisting of a fixed-displacement pump driven by an angular velocity source, a pressure-relief valve, a pressure-reducing valve, and two variable orifices.

The variable orifices simulate fluid consumption in the main system and in the reduced pressure branch. The simulation starts with both variable orifices open, which results in a low output pressure and the maximum flow rate going to the system. The orifice for the main branch starts closing at 0.5 seconds and is closed completely at around 1.5 seconds.

The output pressure builds up until it reaches 75e5 Pa, and is maintained at this level by the pressure-relief valve. The pressure reducing valve in the reduced pressure branch is set to 20e5 Pa. It maintains this pressure at its output as long as the pressure upstream of the valve is higher than this setting.

Model



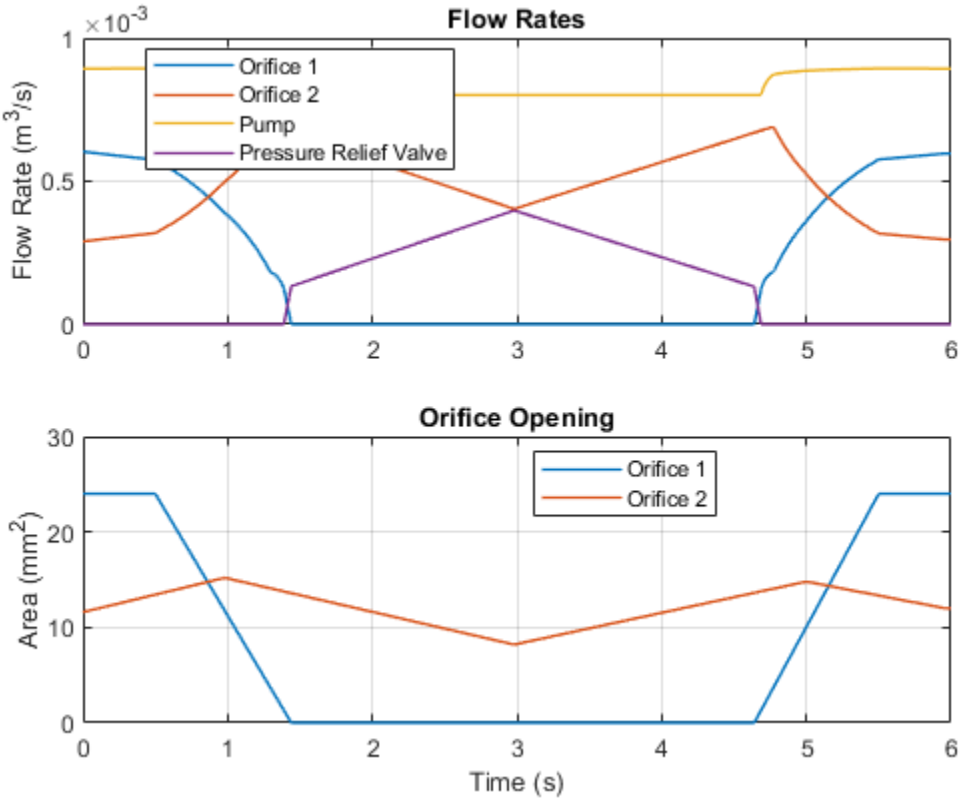
Pump with Pressure-Reducing Valve and Two Loads

1. Plot flow rates in system (see code)
2. Explore simulation results using sscexplore
3. Learn more about this example

Copyright 2005-2021 The MathWorks, Inc.

Simulation Results from Simscape Logging

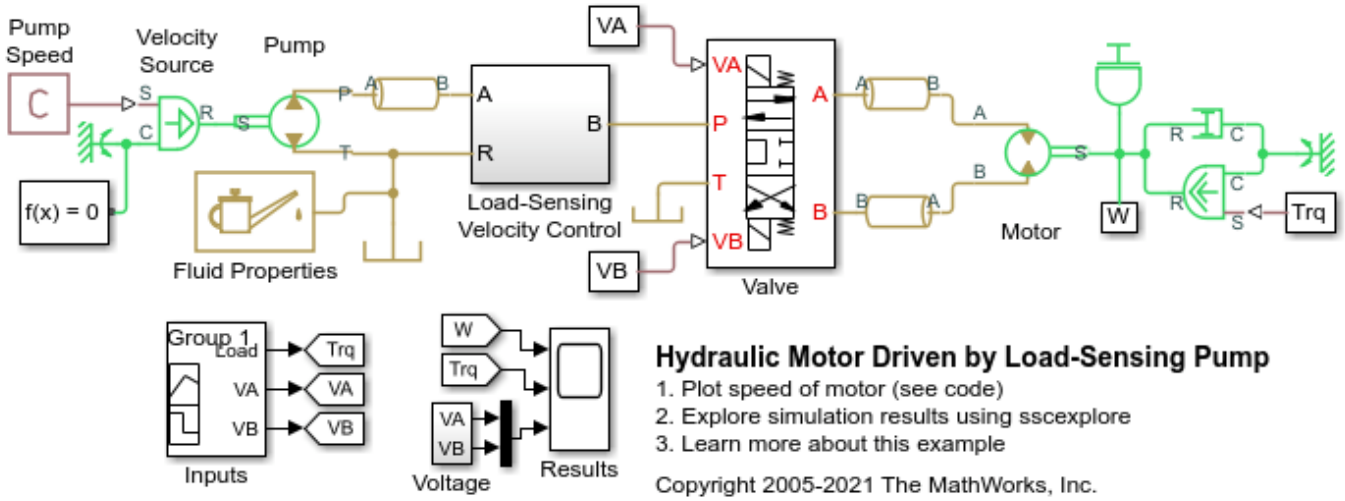
The plots below show how flow rates vary in the system as the orifices open and close. The fluid exiting the pump is divided between the orifices at the start. As the Orifice 1 and 2 is close, flow gradually ends up being diverted through the pressure relief valve.


Flow Rates
×10^-3
Flow Rate (m^3/s)
Orifice 1
Orifice 2
Pump
Pressure Relief Valve
Orifice Opening
Area (mm^2)
Orifice 1
Orifice 2
Time (s)

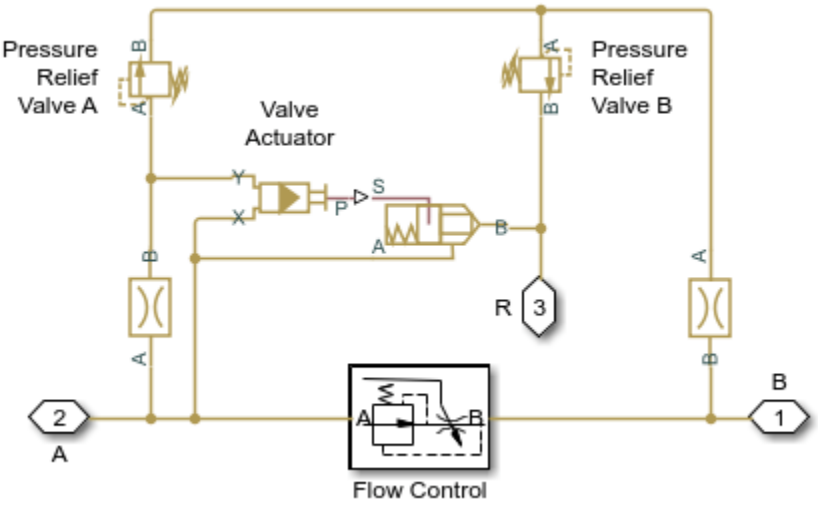
Hydraulic Motor Driven by Load-Sensing Pump

This example shows a circuit that is equipped with a load-sensing velocity regulator installed between the pump and directional valve. Unlike a conventional meter-in velocity control, the load-sensing device automatically adjusts output pressure of the pump in such a way that it equals the sum of the preset pressure drop across the pressure-compensated flow control valve and the pressure induced by load. The pilot-operated pressure-relief valve in the Load-Sensing Velocity Control block is built of the Poppet Valve and the Hydraulic Double-Acting Valve Actuator blocks.

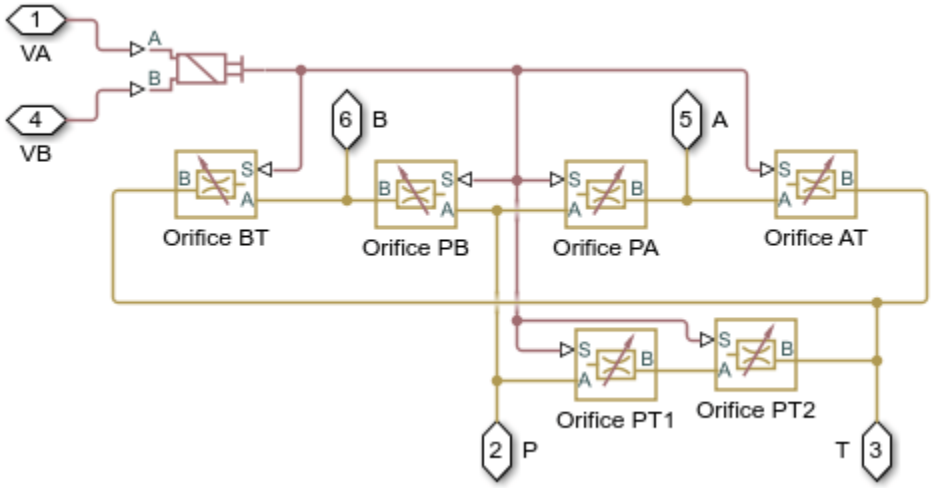
Model



Load-Sensing Velocity Control Subsystem

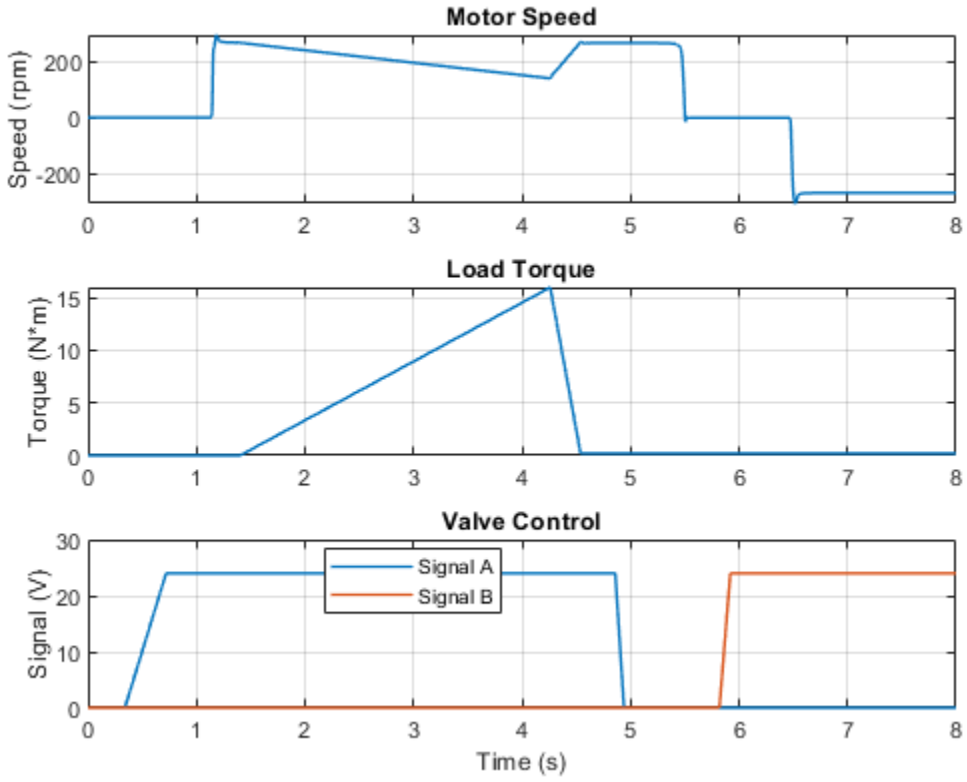


Valve Subsystem



Simulation Results from Simscape Logging

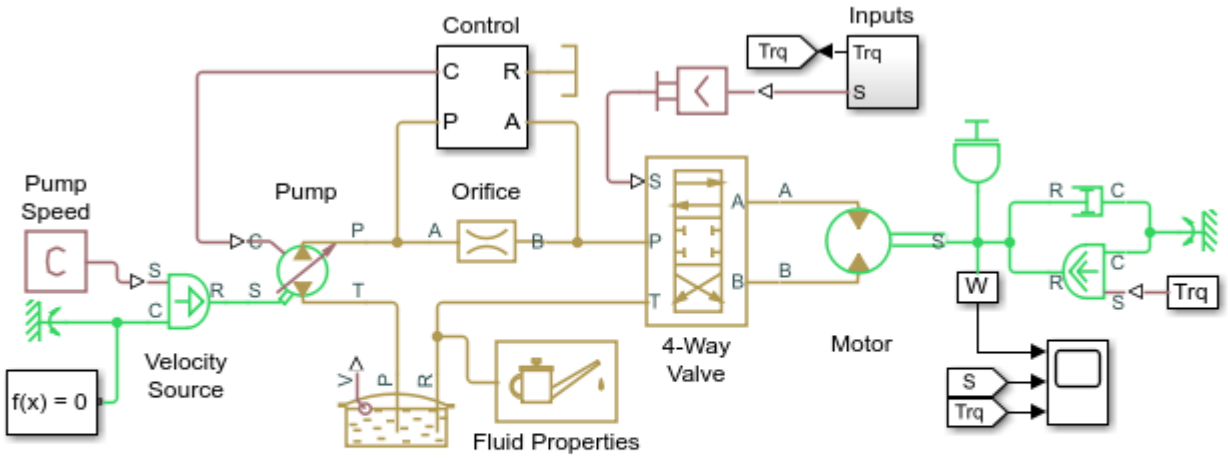
The plots below show the behavior of the motor. The motor speed varies as the load torque is applied. The valve controls pressure to the motor such that it can spin forward, backward, or be locked in place.



Hydraulic Motor Driven by Load-Sensing Variable-Displacement Pump

This example shows a circuit using a load-sensing and pressure-limiting unit in a conventional reciprocal system with variable load on the forward stroke. The unit limits output pressure to 300 bar and maintains a preset pressure drop in 10 bar across velocity control orifice on the discharge port of the pump. The unit is built of two 3-way, 2-position valves, two single-acting hydraulic valve actuators, and one double-acting hydraulic valve actuator.

Model

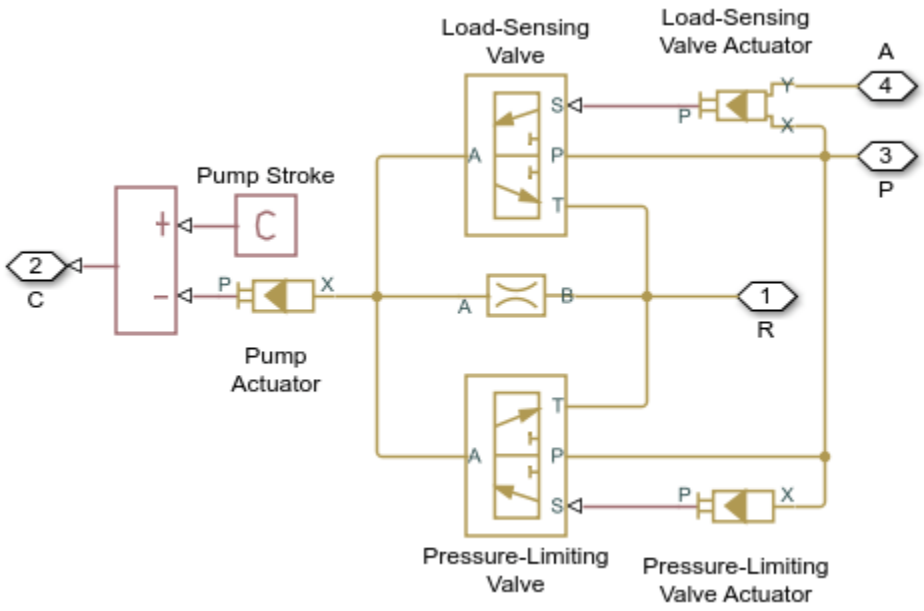


Hydraulic Motor Driven by Load-Sensing Variable-Displacement Pump

1. Plot speed of motor (see code)
2. Explore simulation results using sscexplore
3. Learn more about this example

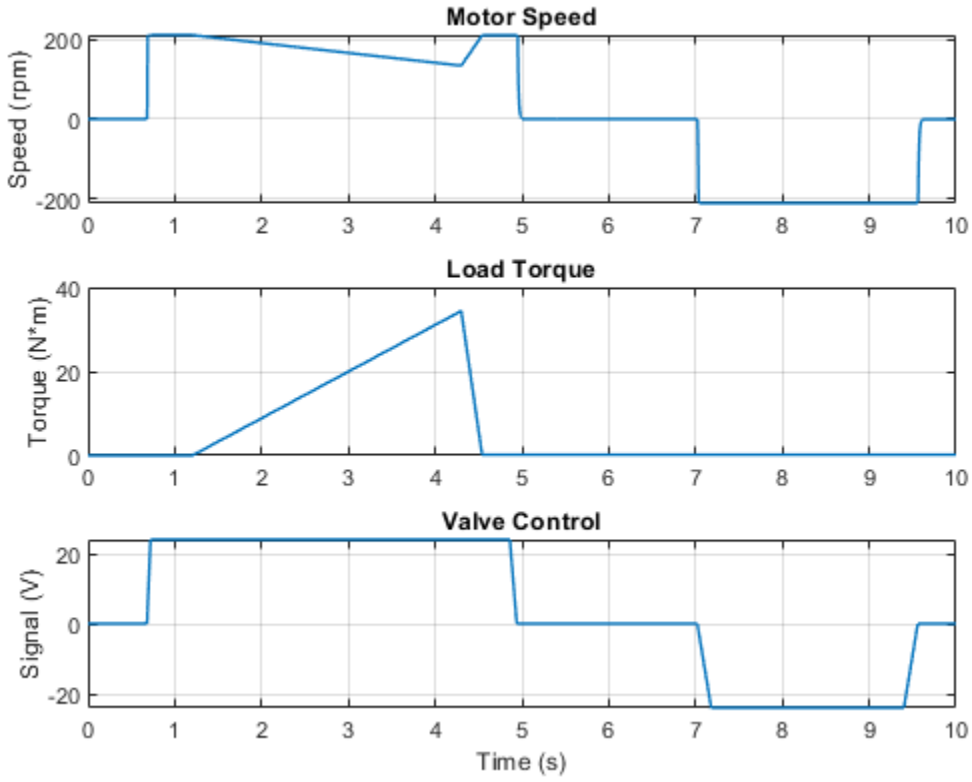
Copyright 2005-2021 The MathWorks, Inc.

Control Subsystem



Simulation Results from Simscape Logging

The plots below show the behavior of the motor as the valve inputs and load torque are varied.


Motor Speed
Speed (rpm)
200
0
-200
0 1 2 3 4 5 6 7 8 9 10
Load Torque
Torque (N*m)
40
20
0
0 1 2 3 4 5 6 7 8 9 10
Valve Control
Signal (V)
20
0
-20
0 1 2 3 4 5 6 7 8 9 10
Time (s)

Hydraulic Axial-Piston Pump with Load-Sensing and Pressure-Limiting Control

This example shows a test rig designed to investigate interaction between an axial-piston pump and a typical control unit, simultaneously performing the load-sensing and pressure-limiting functions. To assure required accuracy, the model of the pump must account for such features as interaction between pistons, swash plate, and porting plate, which makes it necessary to build a detailed pump model.

Model

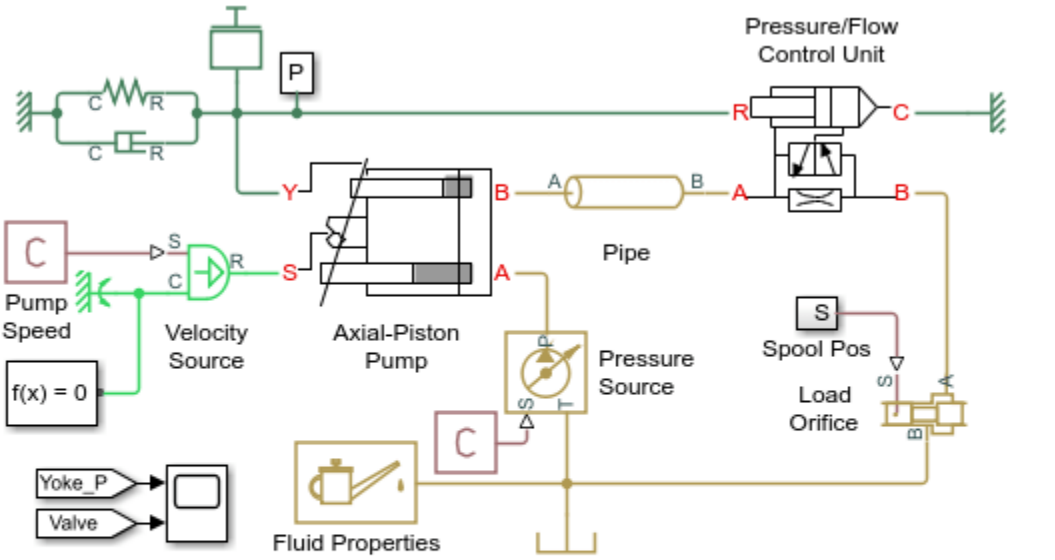


Figure 1. Test Rig Schematic

Test Rig Description

The pump model is represented by the subsystem named Axial-Piston Pump. The prime mover rotating the pump is simulated with the Ideal Angular Velocity source. The pump output passes through a pipeline, control unit, and a variable orifice that acts as a load. To test control unit response to variable load, the orifice changes its area during simulation. The change profile is implemented by the Signal Builder block.

The control unit in the test rig is represented by the subsystem named Pressure/Flow Control Unit. The load-sensing function of the pump control uses a fixed orifice. The control unit keeps the pressure differential across this orifice constant, regardless of pump loading. The control unit receives signals on pump output pressure and load pressure, measured after the flow control valve.

Based on these pressures, the unit produces yoke displacement, which affects the angular position of the angled swash plate in the pump. This helps maintain the specified pressure differential across the flow control valve and prevent pump pressure from exceeding the preset value.

Test rig basic parameters:

```
Pump maximum displacement                    7.8877e-6 m^3/rad
Pitch radius                                 0.04 m
Piston area                                  1.77e-4 m^2
Number of pistons                            5
Maximum piston stroke                        0.06 m
Swash plate maximum angle                    35 deg (0.6109 rad)
Arm length between the actuator
  and the swash plate pivoting point         0.055 m
Swash plate actuator stroke                  0.04 m
Diameter of the orifice at the bottom
  of the piston chamber                      0.007 m
Pump maximum rated speed                     260 rad/s (2482 RPM)
Maximum pressure                             270 bar
Rated flow                                   1.1e-3 m^3/s
```

All basic parameters are specified through the **Model Properties** > **Callbacks** > **InitFcn** option.

Axial-Piston Pump Model

The pump under investigation is an axial-piston pump with five pistons.

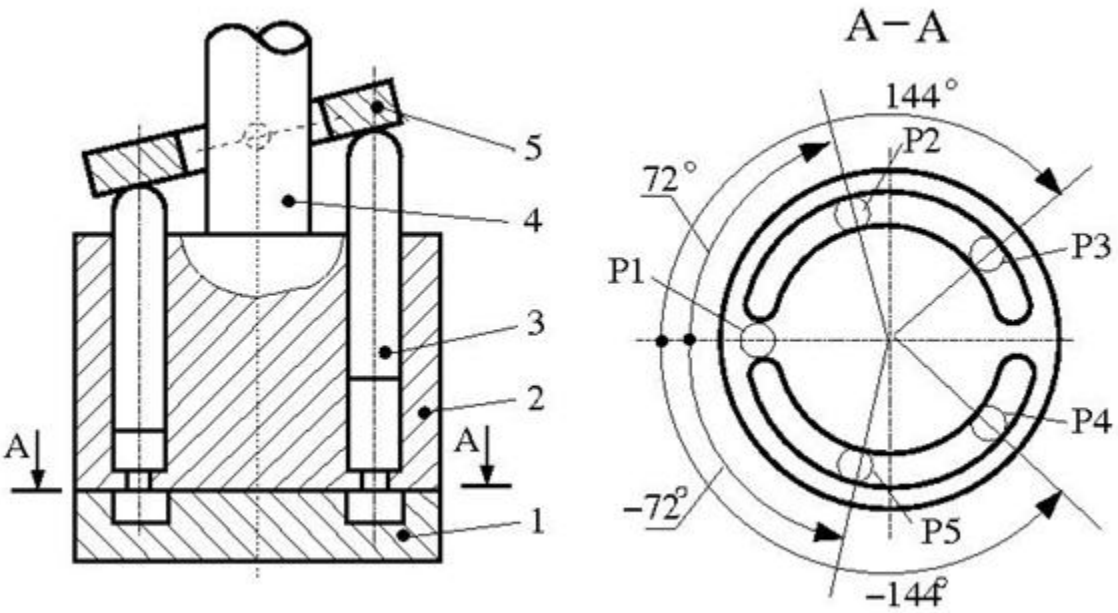


Figure 2. Axial-Piston Pump Schematic

The pump schematic diagram is shown in Figure 2, where:

1 - porting plate

2 - cylinder block (rotor)

3 - piston

4 - driving shaft

5 - swash plate

The block diagram of the pump model is shown in Figure 3.

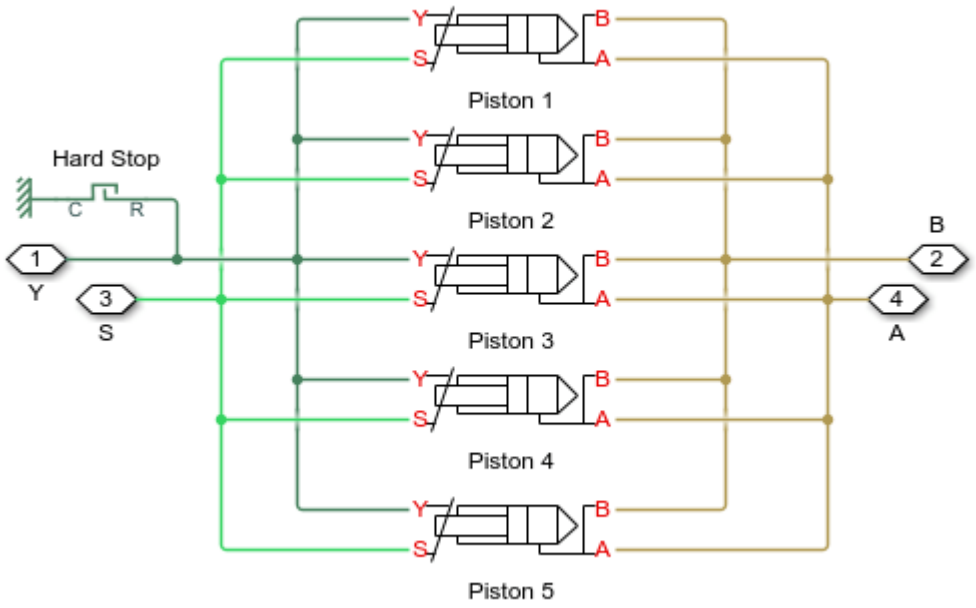


Figure 3. Pump Model

Every piston of the pump is represented by a subsystem called Piston. These subsystems are identical and are connected to the following external ports of the pump model:

- S - pump driving shaft
- Y - yoke connected with the inclined plate of the swash mechanism
- P - pump discharge

The suction ports of all pistons (ports A) are connected to the output of a low pressure booster pump, which is simulated with the Ideal Hydraulic Pressure Source block. The output pressure of the booster pump is set to 5e5 Pa.

The yoke is connected to the Y ports of all pistons, thus acting on the angled plate of the swash mechanism. The displacement of the yoke is limited by the hard stop.

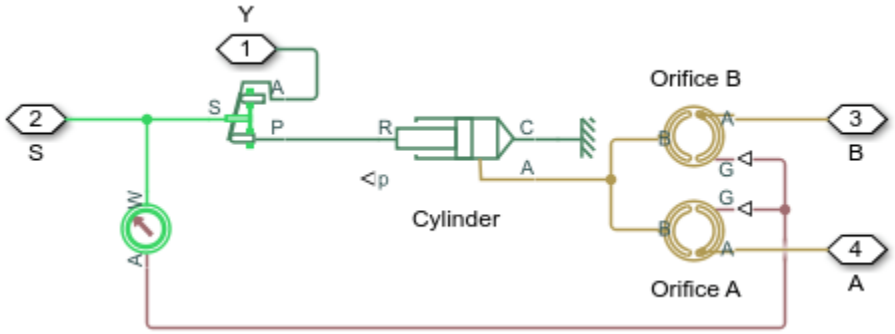


Figure 4. Piston Model

The model of the piston (Figure 4) is based on the Single-Acting Hydraulic Cylinder block, which is mechanically connected to the drive shaft through the Swash Plate block. The cylinder is also hydraulically connected to ports A and B through the Porting Plate Variable Orifice blocks. Ports A and B represent pump discharge and intake ports, respectively.

The port assignment is performed on the basis of the following considerations:

- The pistons are evenly spread along the pitch circle of the piston cylinder, as shown in Figure 2. This makes the angle between pistons 360/5 = 72 degrees.
- Let us assume that the first piston (marked P1 in the schematic) is located exactly at the reference point that corresponds to the lowest piston position. Let us further assume that port A represents the intake outlet of the pump. In other words, the piston moving along slot A in positive direction (clockwise in this case) goes up, and its chamber is filled with fluid by a booster pump. This means that the **Phase angle** parameter of the Porting Plate Variable Orifice A in Piston 1 must be set to zero. The same parameter of the Porting Plate Variable Orifice B in Piston 1 must be set to 180 degrees, because it starts interacting with slot B (pump discharge port) only after rotation by 180 degrees.

In the piston model, parameters **Phase angle** of each Porting Plate Variable Orifice block are denoted as **Phase angle A** and **Phase angle B**, respectively. The values of the phase angle for all five pistons are computed in the initialization section of the Axial-Piston Pump subsystem mask editor. The following table shows their values in degrees, with the corresponding values in radians given in parenthesis:

Piston name in the pump model	Phase angle A	Phase angle B	Phase angle in Swash Plate block
Piston_1	0	180 (pi)	0
Piston_2	72 (1.2566)	-108 (-1.885)	72 (1.2566)
Piston_3	144 (5.5133)	-36 (-0.8029)	144 (5.5133)
Piston_4	-144 (-5.5133)	36 (0.8029)	-144 (-5.5133)
Piston_5	-72 (-1.2566)	108 (1.885)	-72 (-1.2566)

The Swash Plate block in the piston model also requires phase angle to be assigned, to specify the position of a piston with respect to the inclined surface. With the selected reference point, the values of the swash plate phase angle coincide with the Phase angle A values, as shown in the table.

The Porting Plate Variable Orifice blocks require angular position of the respective piston at their input. This function is performed by the Angle Sensor block.

Other important parameters are the stroke of the cylinder and the initial position of the piston with respect to the cylinder cap. The stroke must be large enough to allow the piston to reciprocate even at the maximum angle of the swash plate

Stroke > 2 * *PitchRadius* * `tan`(*MaxAngle*),

where *PitchRadius* is the radius of the cylinder block pitch circle, and *MaxAngle* is the maximum allowed angle of the swash plate.

In the model, the maximum angle is set to 35 degrees (0.6109 rad) and pitch radius is set to 0.04 m, which makes the stroke to be greater than 0.056 m. The stroke is set 0.06 m. The piston initial positions must be equal to half of the stroke at zero initial swash plate angle. But the initial angle changes its value depending on the initial position of the actuator. As a result, the piston initial positions are computed with the equation

$$x_{p0_i} = x_{a_i} * \frac{r_{pc}}{r_a} \cos(\gamma_0), \qquad i = 1,\ldots,5$$

where

x_{p0}	piston initial position
x_a	actuator initial position
r_{pc}	pitch radius of the cylinder block
r_a	actuator radius
γ_0	phase angle

The calculation of the piston initial positions is performed in the initialization section of the subsystem mask editor.

Pressure/Flow Control Unit

The purpose of the control unit is to implement two functions: load sensing and pressure limiting. The load sensing is implemented by maintaining specified pressure differential across the flow control valve. In the model of the test rig (Figure 1), the flow control valve is simulated with the Orifice with Variable Area Slot block. Pressures upstream and downstream of the valve are conveyed to the Pressure/Flow Control unit via ports P and LSP (Figure 5).

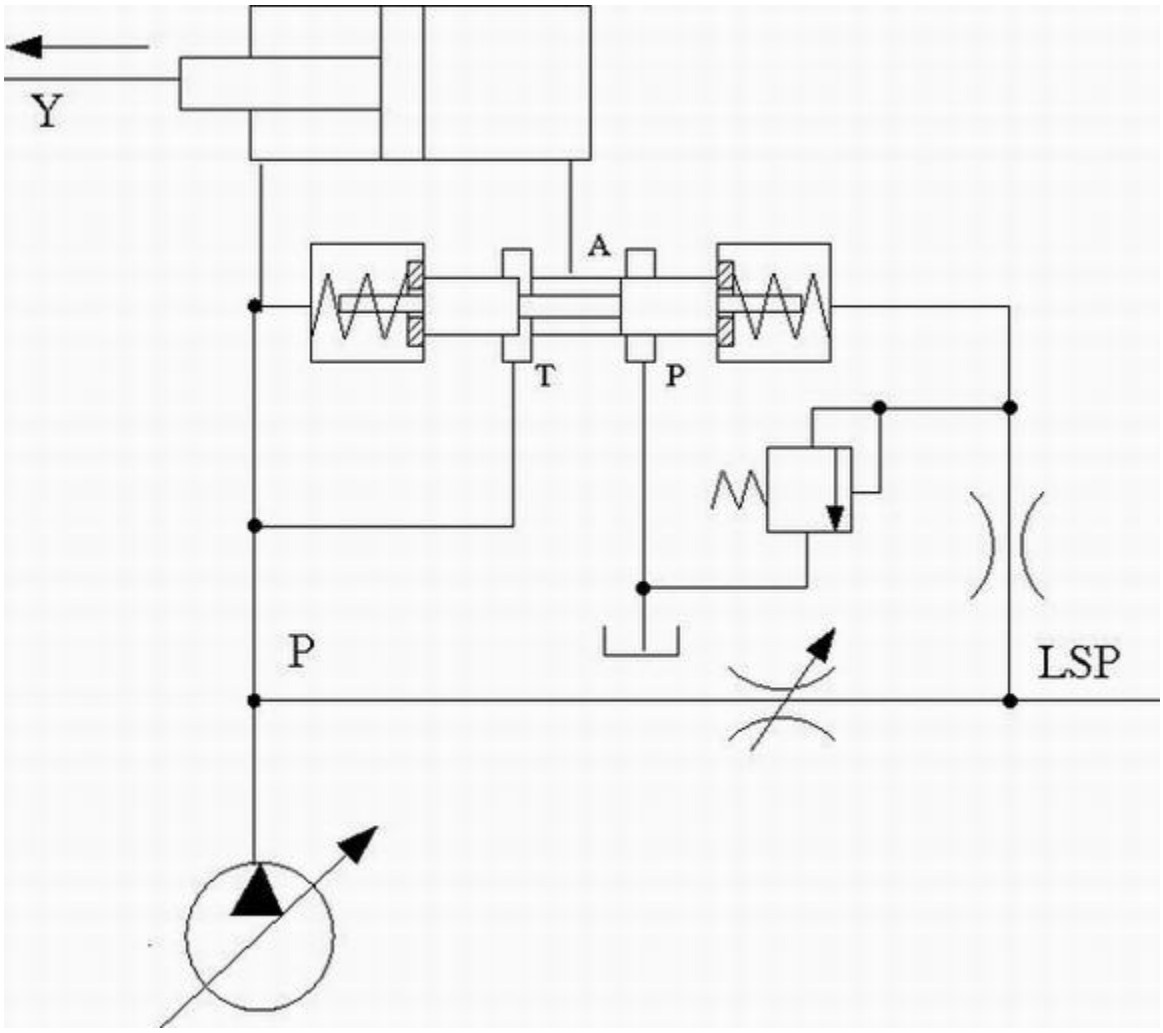


Figure 5. Pressure/Flow Control Unit Schematic

These pressures act on the side faces of the 3-way directional valve and shift the valve proportionally to the pressure difference and setting of the centering springs. The valve connections are selected in such a way that increased pressure differential opens path A-P and closes path A-T. The actuator is arranged as a single-rod differential hydraulic cylinder with rod connected to the pump yoke. The pump displacement is increased if the rod moves in the direction of the arrow shown in the schematic. Due to difference between cylinder effective areas, the displacement is increased if both cylinder chambers are connected to the pump, and decreased if the chamber without rod is connected to tank. As a result, increased pressure differential across the valve causes the pump to decrease its displacement until it returns to preset value. The spring preload of the valve is determined with the equation

$$f_0 \geq \Delta p \cdot A_{LSC}$$

where

f_0	spring preload of the load sensing valve
Δp	required pressure differential across the flow control valve

The purpose of the pressure limiting function is to prevent pump pressure from exceeding the preset value. It is implemented with the pressure-relief valve and the orifice in the LSP line. The pressure-relief valve is set to the desired maximum value. When pump pressure builds up to this value, the valve opens and causes pressure in the right chamber of the valve to decrease opening path A-P. The actuator shifts to the right until the pressure returns to the preset value.

The model of a load-sensing valve is built using the 3-Way Directional Valve, Hydraulic Double-Acting Valve Actuator, Pressure-Relief Valve, and Fixed Orifice blocks, as shown in the model diagram (Figures 6 and 7).

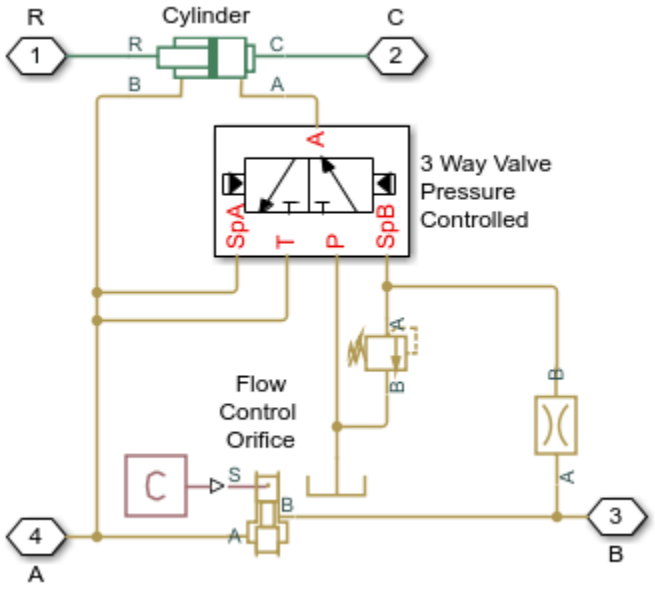


Figure 6. Pressure/Flow Control Unit Model

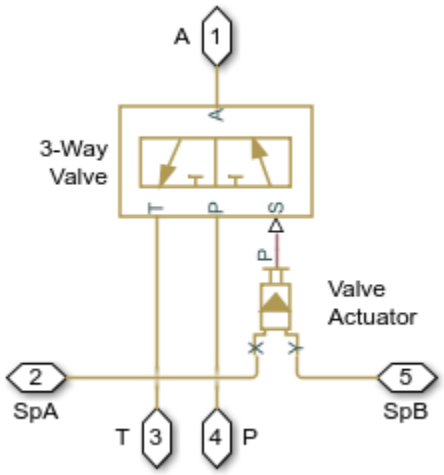


Figure 7. 3 Way Valve Pressure Controlled Model

The pressure differential is set to 20 bar. The 3-Way Directional Valve path A-T must be initially open, to force the pump to increase its displacement at the start of operation. To perform the load-sensing function, pressure increase at the B port (load-sensing port) must open the A-T path and close the A-P path. These are the reasons that determined the valve port connections to the system. The remaining load sensing control valve parameters, such as spring stiffness, valve stroke, valve orifice area, and so on, are tuned in the model to ensure required accuracy, stability, and numerical effectiveness.

The pressure-limiting function is implemented with the combination of the Fixed Orifice and Pressure-Relief Valve blocks. The valve is set to 250 bar. At this pressure, increased flow through the fixed orifice causes pressure at port Y of the Hydraulic Double-Acting Valve Actuator (block Valve Actuator in Figure 7) to drop, which eventually decreases the displacement of the pump.

Cycle Description

The simulated cycle consists of six elements characterized by different load conditions with the Variable Area Slot block.

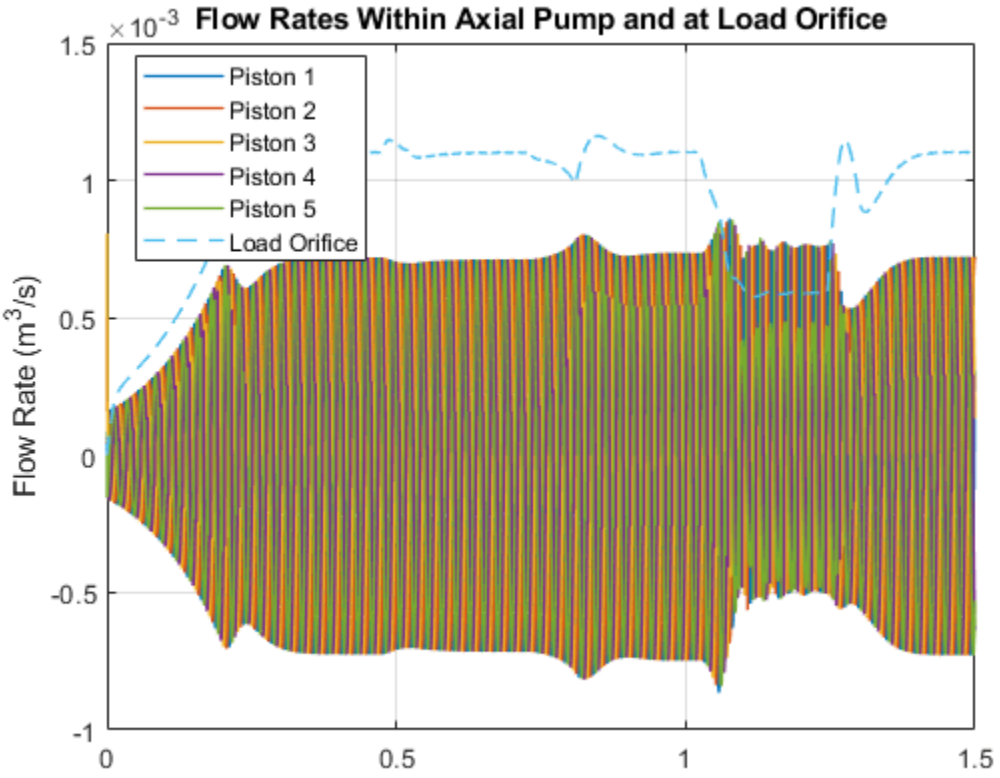
The cycle starts with zero opening signal, followed by opening of 2.8, 5.2, 1, -0.8, and, finally, 2.45 mm. At the start of the cycle, the pump shaft starts rotating at 260 rad/s (~2500 RPM) with the pump yoke initial position set to 5 mm. The servo cylinder starts increasing pump displacement, pump pressure slowly builds up, and the process settles down at ~0.35 s after the pressure differential across the flow control valve becomes close to preset value of 20 bar. The load-sensing valve is opened at this moment by ~1.2 mm.

During next three portions of the cycle, the pump maintains practically the same delivery despite changes of the load valve opening.

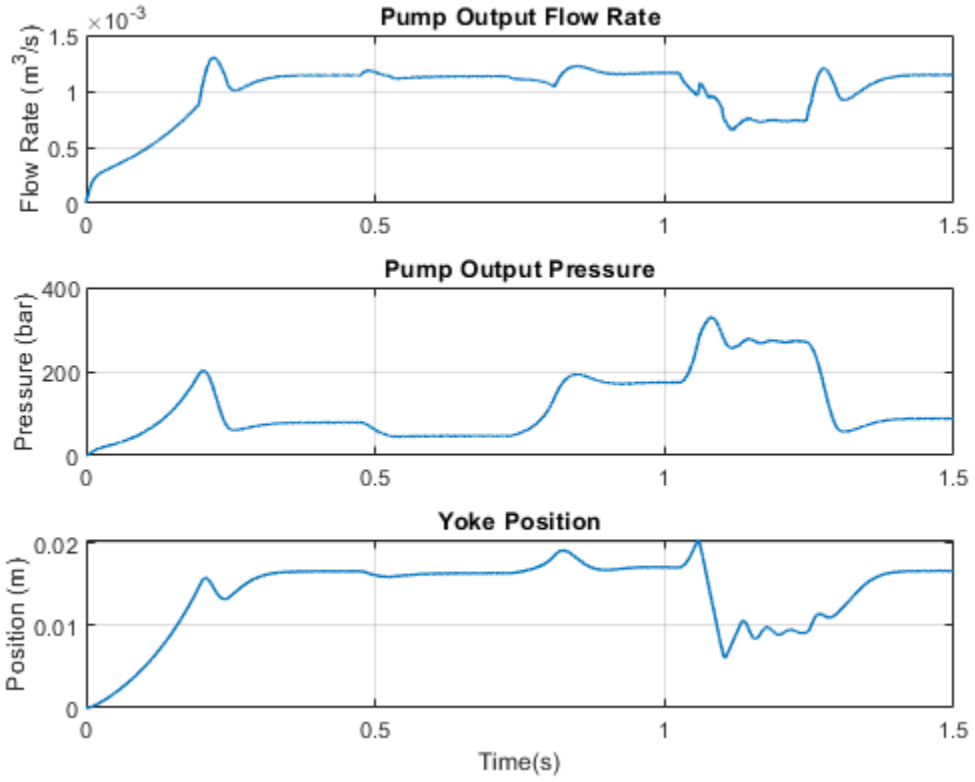
At 1 s, the load valve is practically fully closed, causing pump pressure to rise. The load-limiting function becomes dominant as the pressure reaches 270 bar. The pump returns to the load-sensing mode after the pressure falls below the preset value.

Simulation Results from Simscape Logging

The plot below shows the flow rate within the pump pistons and at the load. The cyclic nature of the piston pressures can be seen, as well as the overall behavior of the pump which stays close to its rated flow rate.



The plots show the load sensing and pressure limiting control. The pump maintains its rated flow rate of 1.1 m^3/s even as the load changes, as shown on the pump output pressure plot. However, as the pump output pressure rises to its maximum rated pressure, the pressure limiting control adjusts the yoke position is and the flow rate drops below its rated flow rate.

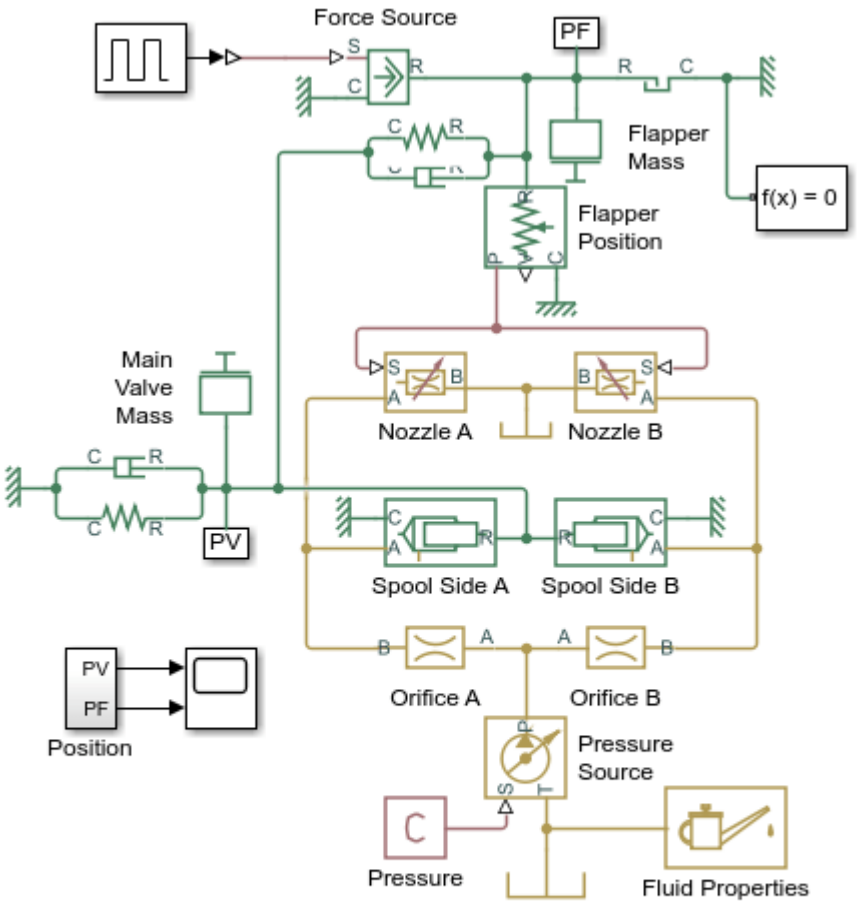

Pump Output Flow Rate
×10^-3
Flow Rate (m^3/s)
1.5
1
0.5
0
0
0.5
1
1.5
Pump Output Pressure
Pressure (bar)
400
200
0
0
0.5
1
1.5
Yoke Position
Position (m)
0.02
0.01
0
0
0.5
1
1.5
Time(s)

Hydraulic Flapper-Nozzle Amplifier

This example shows an amplifier that consists of two fixed orifices, two variable orifices representing nozzles, flapper, and main valve simulated with mass, viscous friction, and centering spring. Two translational converters represent servo-actuators on both sides of the main valve. A feedback spring connects the flapper and the main valve.

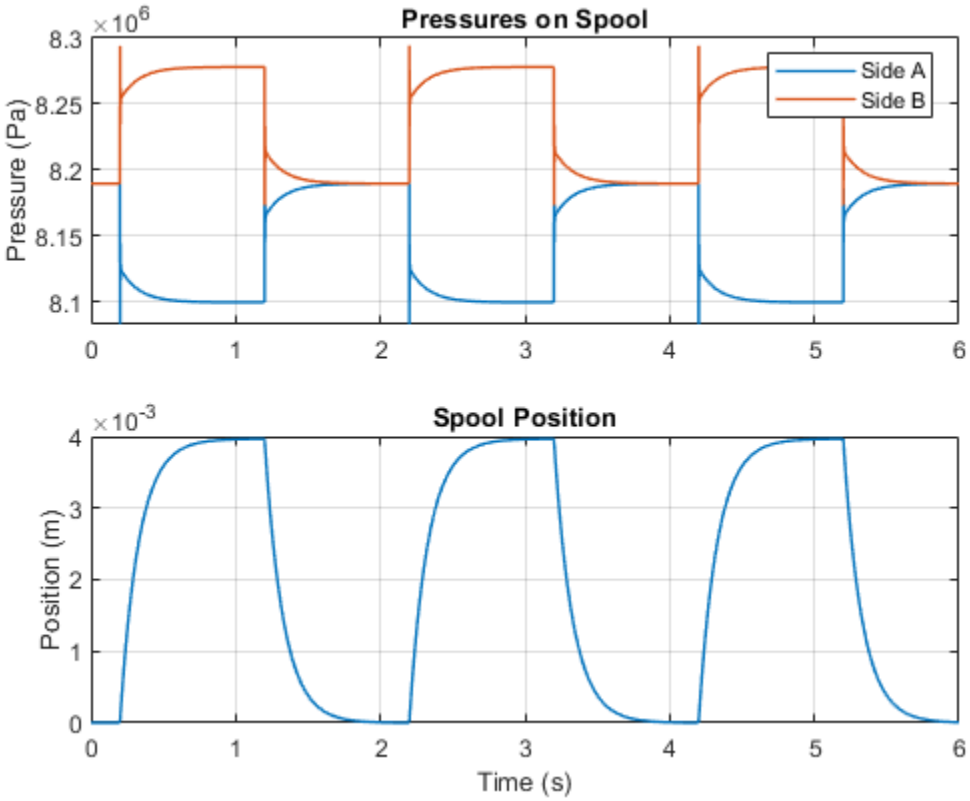
The amplifier is actuated with a force source, which represents the force generated by an electromagnet. The flapper displacement causes pressure difference between converter chambers, which moves the spool. The main valve acts on the flapper in the opposite direction via a feedback spring. The amplifier comes to a steady-state position as the force equilibrium on the flapper is reached.

Model



Simulation Results from Simscape Logging

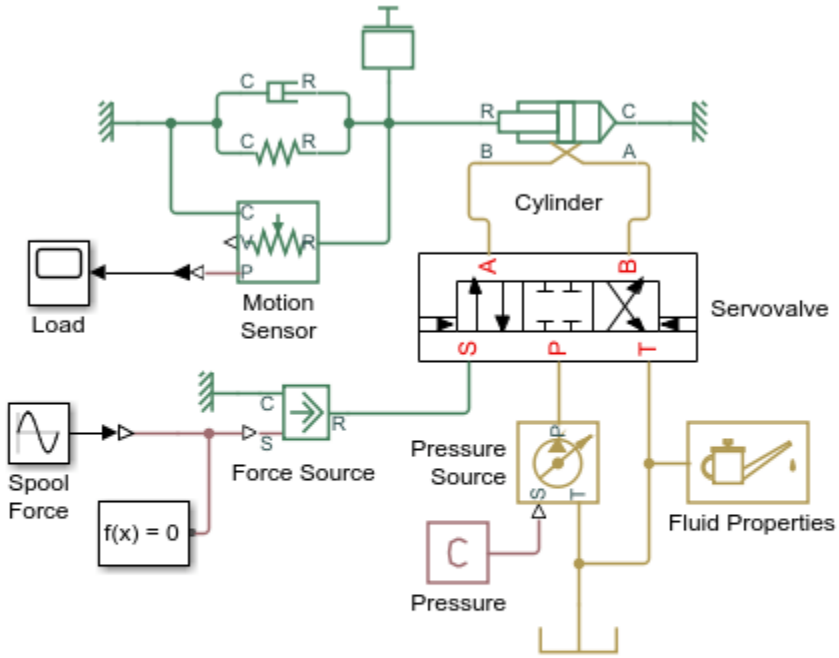
The plots below show the pressures acting on the spool and the position of the spool.



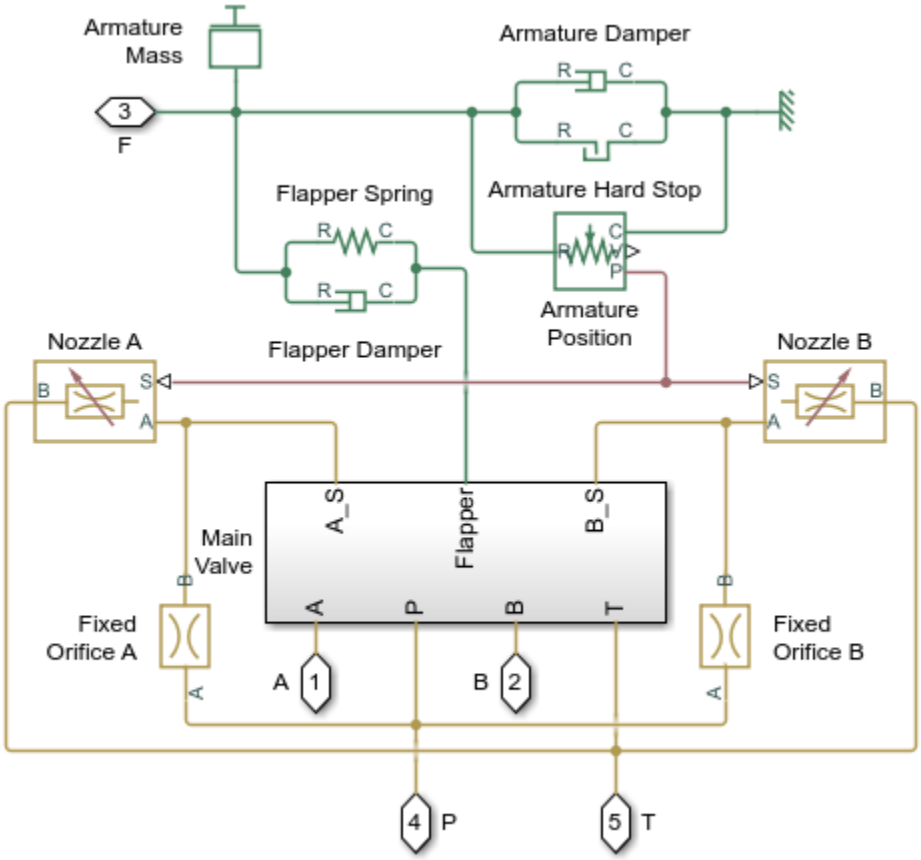
Servo-Valve Controlling Hydraulic Actuator

This example shows a model of a two-stage servo-valve with a 4-way, 3-position spool valve in the power stage and a flapper-nozzle amplifier in the pilot stage. The flapper is connected to the armature of a torque motor, which in the example is represented with an ideal translational force source. The servo-valve shown in the example is equipped with the spring feedback between the flapper and the spool of the main valve. To investigate the behavior of such a valve, axial hydraulic forces on all four spool orifices are accounted for in the model by using Spool Orifice Hydraulic Force blocks. The servo-valve controls a simple double-acting cylinder in an open-loop application.

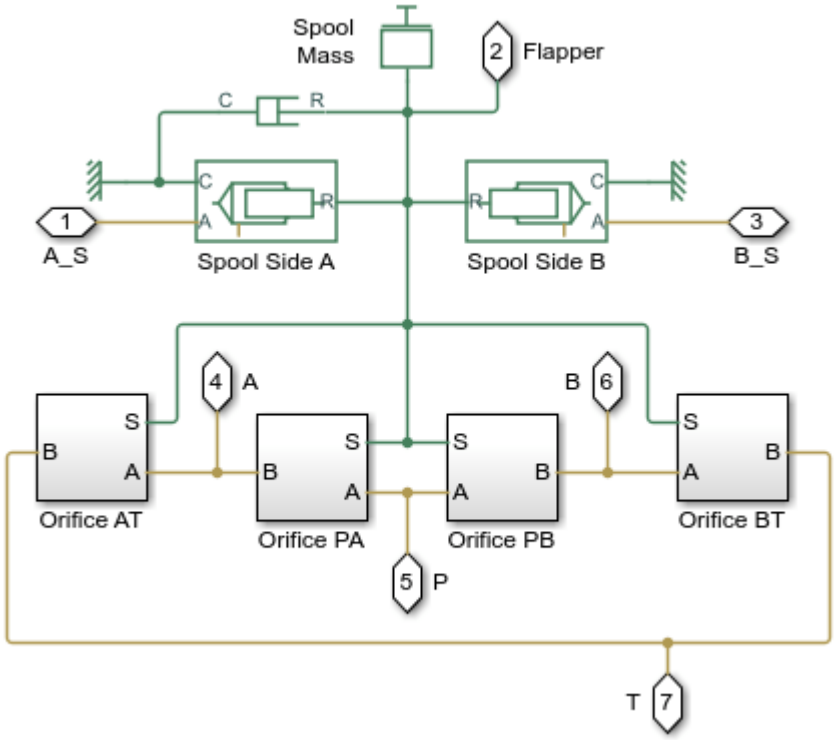
Model



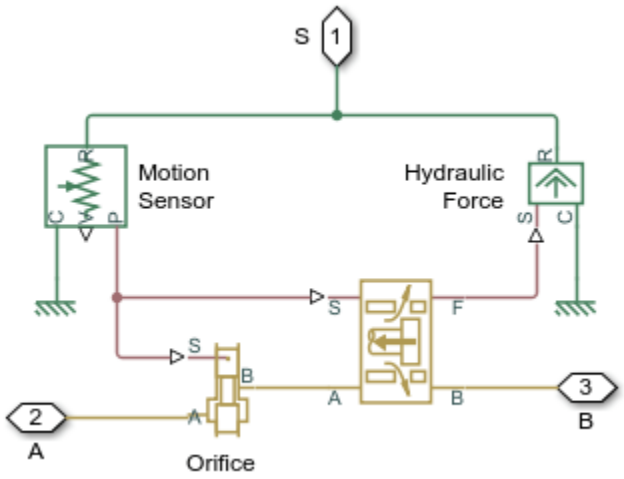
Servovalve Subsystem



Main Valve Subsystem

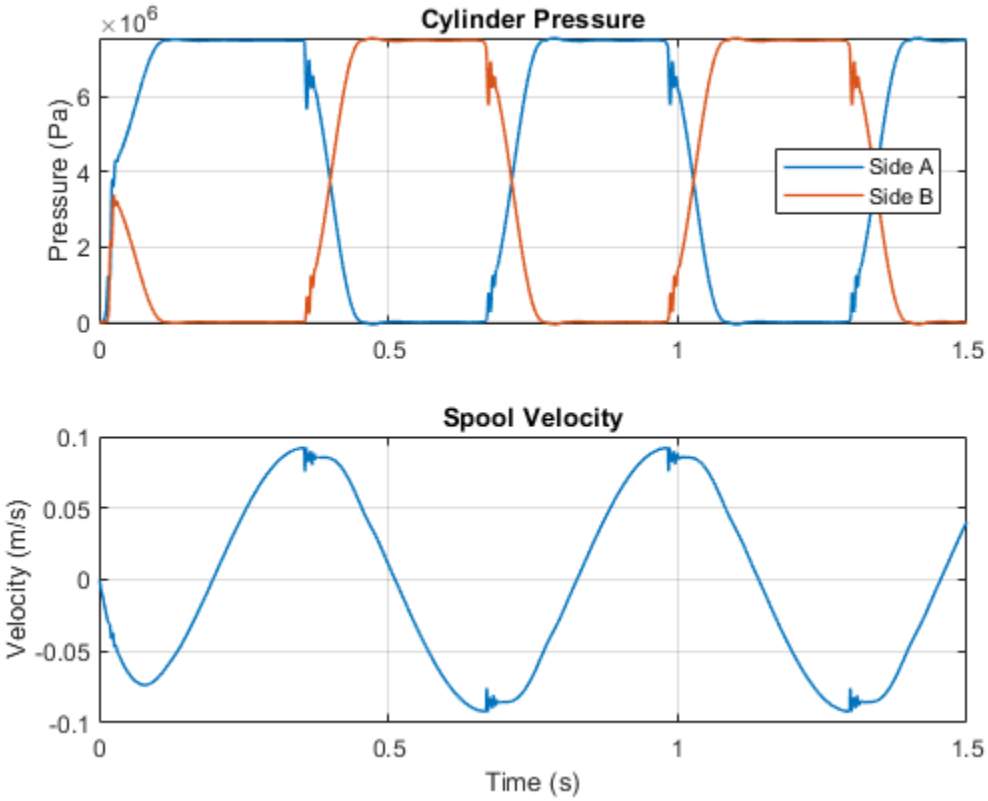


Orifice AT Subsystem

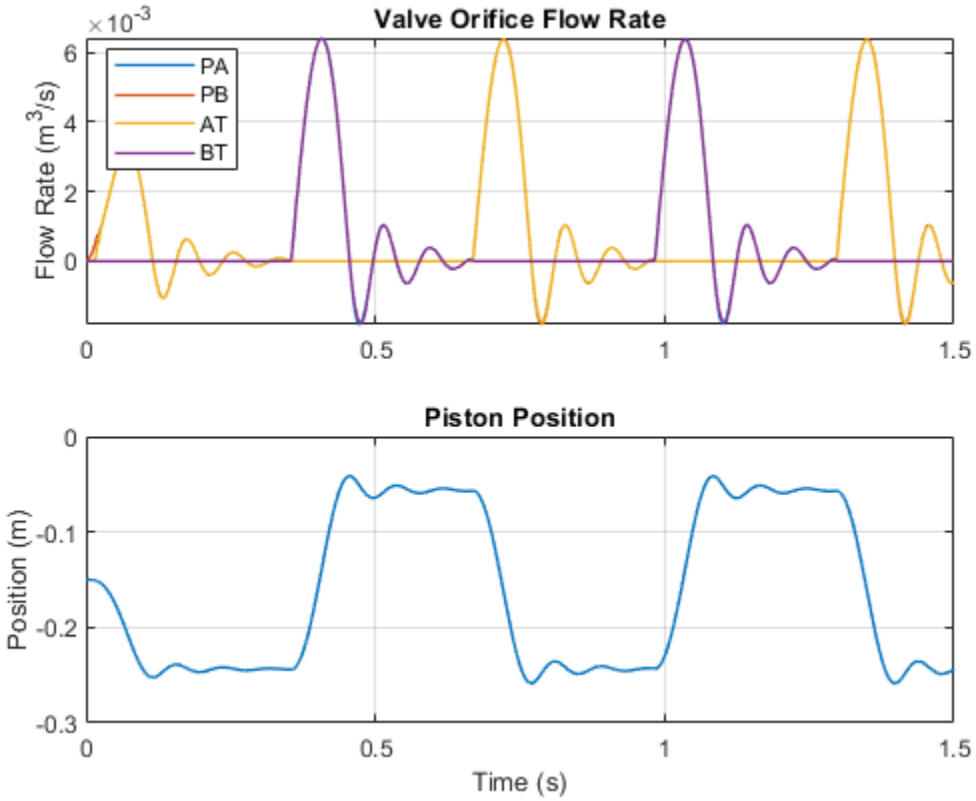


Simulation Results from Simscape Logging

The plots below show the pressures in the cylinder and the velocity of the spool in the pilot stage. The effect of the pilot stage spool oscillations are visible in the cylinder pressures.



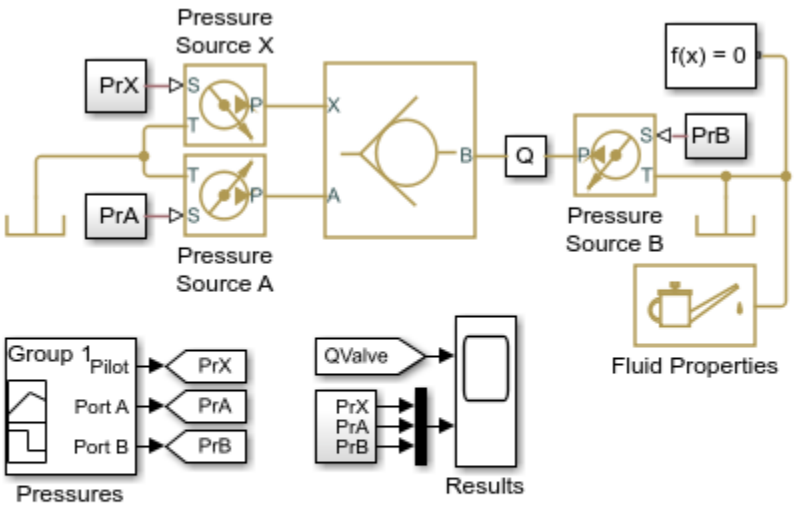
The plots below show the flow rates in the valve and the position of the piston.


Valve Orifice Flow Rate
×10^-3
Flow Rate (m^3/s)
PA
PB
AT
BT
6
4
2
0
0
0.5
1
1.5
Piston Position
Position (m)
0
-0.1
-0.2
-0.3
0
0.5
1
1.5
Time (s)

Pilot-Operated Check Valve Test Rig

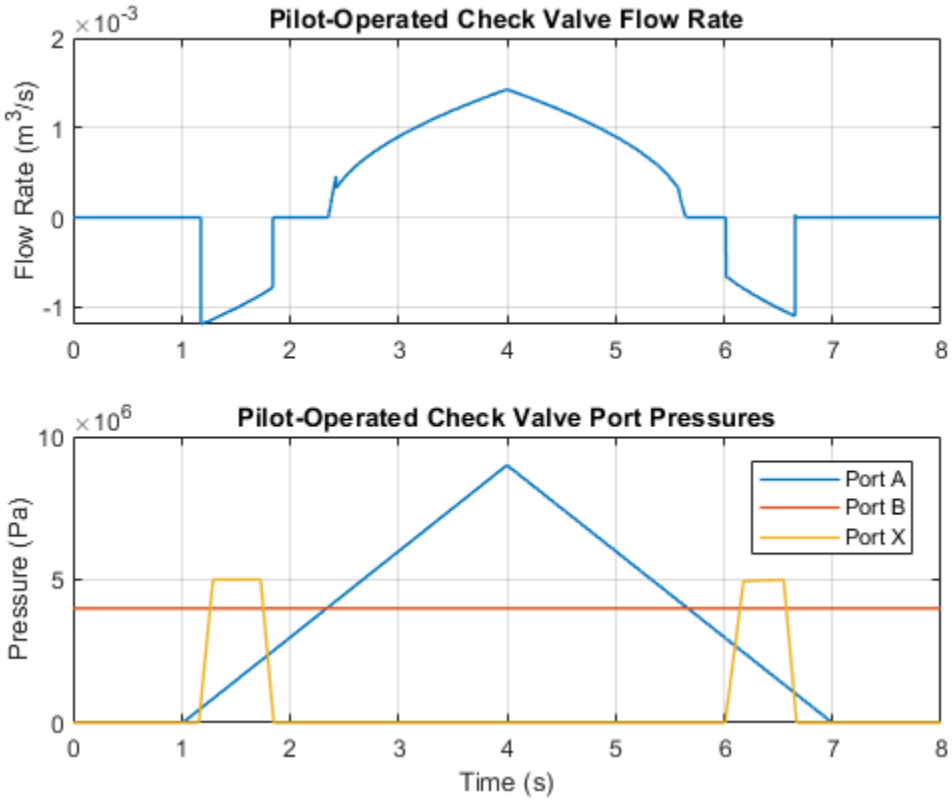
This example shows a test circuit built to check the characteristics of a pilot-operated check valve. The valve is loaded with three ideal pressure sources, two of which create pressure differential across the main flow line, while the third applies pressure to the pilot inlet X. The pilot pressure allows flow through the valve even if the main pressure differential is negative.

Model



Simulation Results from Simscape Logging

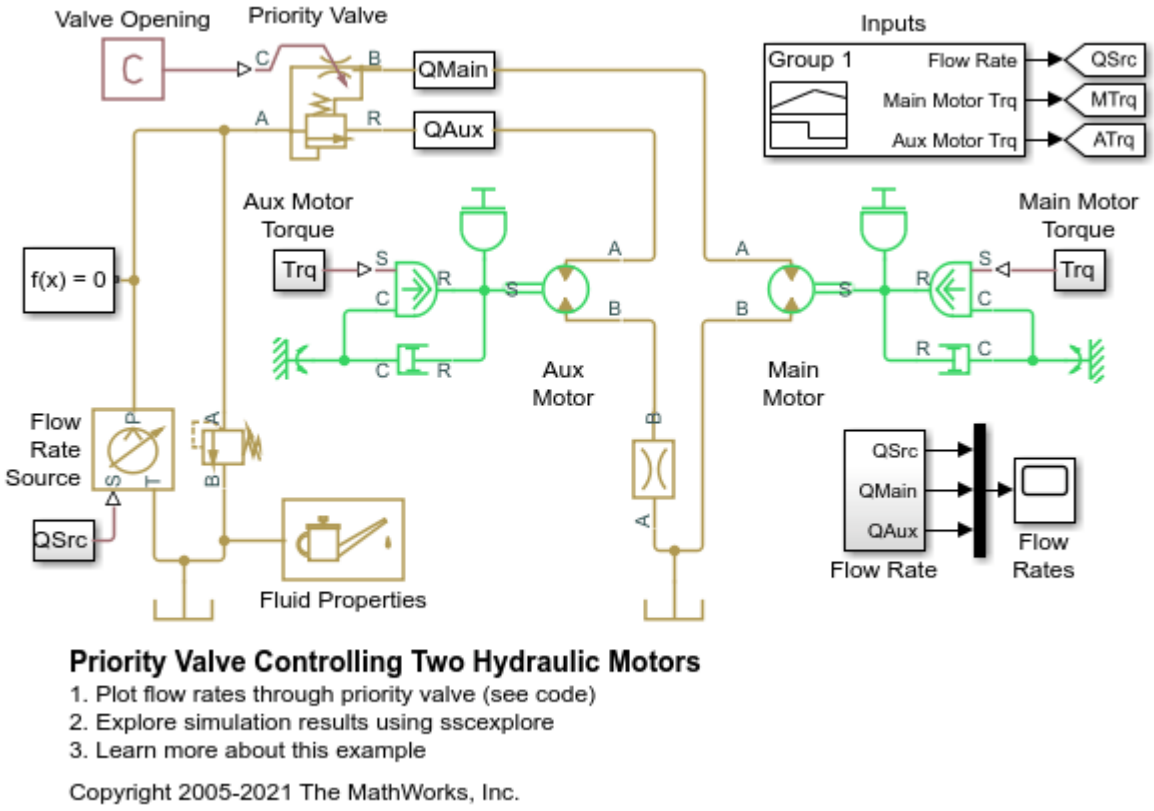
The plots below show the behavior of the Pilot-Operated Check Valve block under varying conditions. Flow is permitted from port A to port B (positive flow rate) when the pressure differential exceeds the cracking pressure. Flow is permitted from port B to port A (negative flow rate) when the pressure at port X is high enough to overcome the pressure differential between port B and port A.


Pilot-Operated Check Valve Flow Rate
×10^-3
Flow Rate (m^3/s)
2
1
0
-1
0
1
2
3
4
5
6
7
8
Pilot-Operated Check Valve Port Pressures
×10^6
Pressure (Pa)
10
5
0
Port A
Port B
Port X
Time (s)

Priority Valve Controlling Two Hydraulic Motors

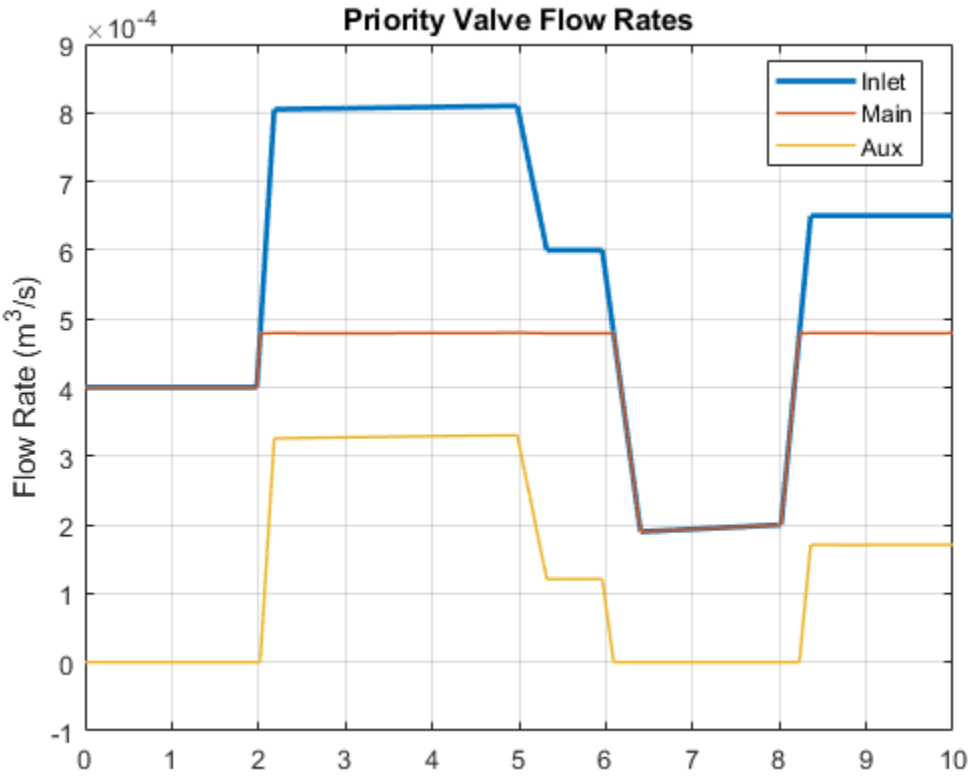
This example shows a pressure-compensated 3-way flow control valve. This valve maintains constant flow rate through the main hydraulic motor, which is connected to the pressure-compensated outlet of the flow control valve. It acts as a priority valve, diverting the excess flow to the auxiliary hydraulic motor if the main hydraulic motor receives enough fluid to maintain a preset angular velocity. The auxiliary motor is shut off completely if there is insufficient flow to power the main hydraulic motor.

Model



Simulation Results from Simscape Logging

The plot below shows the flow rates through the priority valve. As the flow rate at the inlet varies, the main motor is driven at the desired speed and excess flow is directed to the auxiliary hydraulic motor. When the inlet flow drops below the threshold to maintain the speed for the main hydraulic motor, the main motor slows down as well.

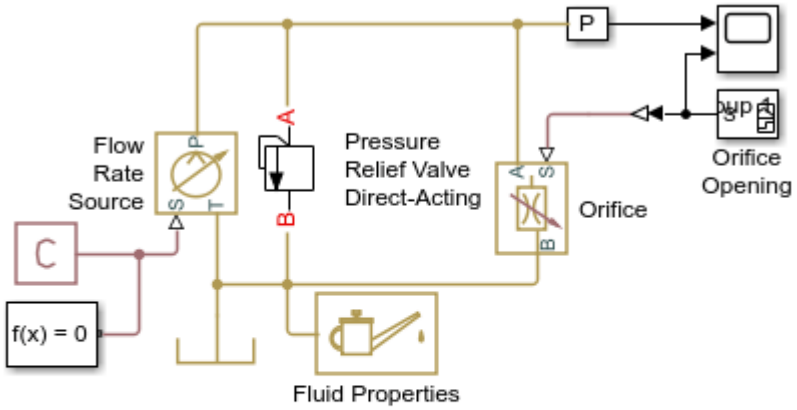

Priority Valve Flow Rates
×10^-4
Flow Rate (m^3/s)
Inlet
Main
Aux
9
8
7
6
5
4
3
2
1
0
-1
0
1
2
3
4
5
6
7
8
9
10

Direct-Acting Pressure Relief Valve Test Rig

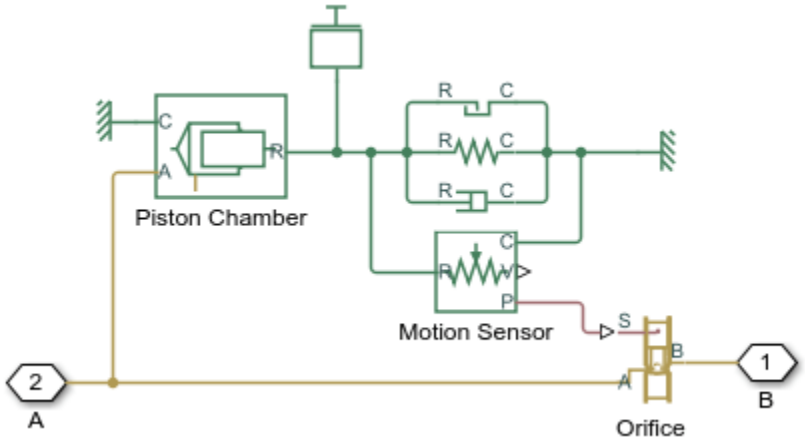
This example shows a test circuit built to check the pressure-flow characteristic of a direct-acting pressure-relief valve. The valve model consists of an orifice, a hydro-mechanical converter, a preloaded spring, and a hard stop.

An ideal flow rate source delivers fluid to a system through a variable orifice. As the orifice closes, pressure gradually builds up and eventually reaches the setting of the pressure-relief valve. At this pressure, the valve starts diverting flow to a tank and maintains preset pressure at the pump outlet. As the orifice is opened again, the pressure-relief valve is closed and the entire flow passes through the variable orifice.

Model

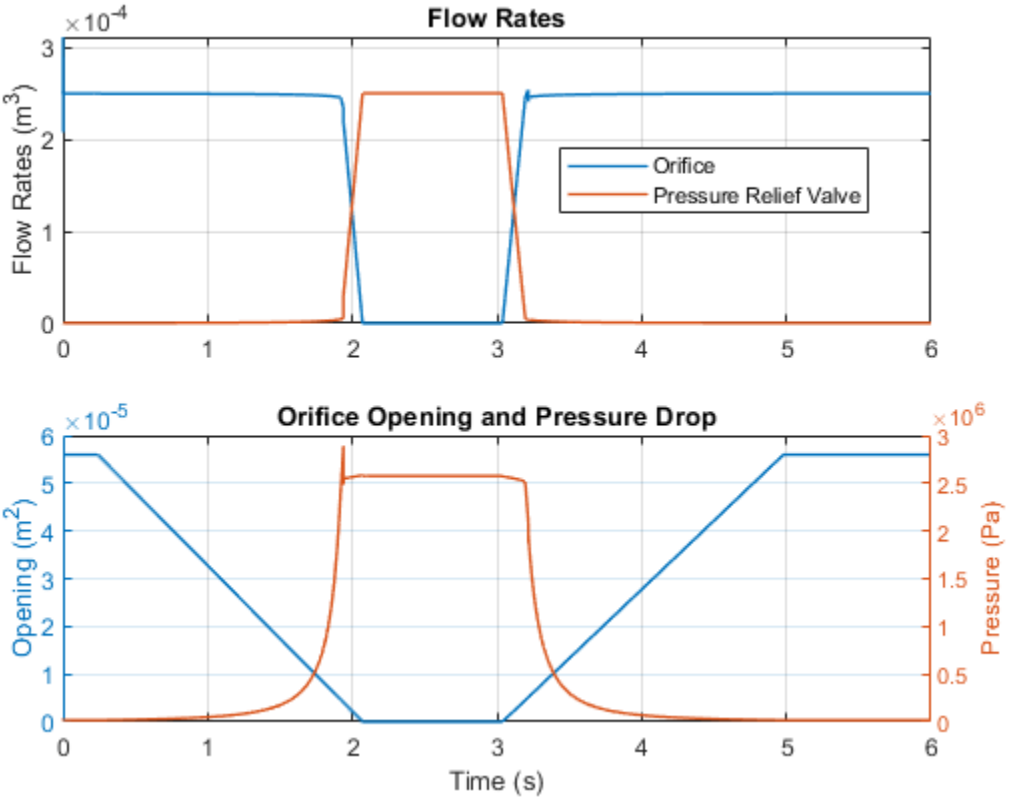


Pressure Relief Valve Direct-Acting Subsystem



Simulation Results from Simscape Logging

The plots below show the behavior of the direct-acting pressure relief valve. As the load orifice closes, pressure in the system rises. When the system pressure reaches the pressure setting for the pressure relief valve, the valve opens which diverts flow away from the orifice and allows the system to maintain the set pressure of the pressure relief valve.

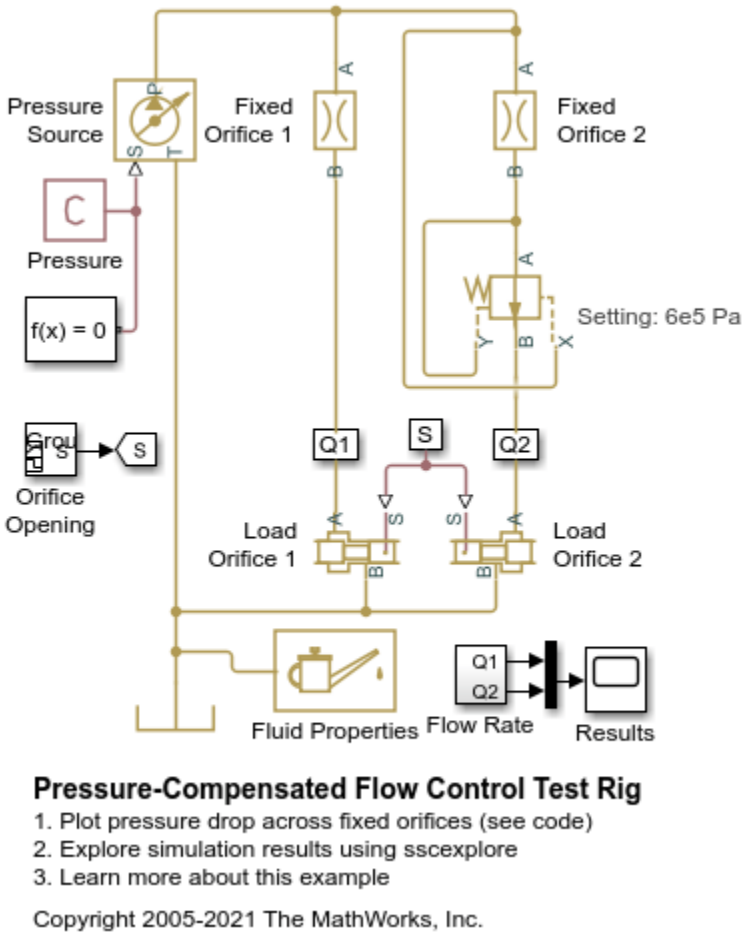


Pressure-Compensated Flow Control Test Rig

This example shows a test circuit built to check the characteristics of a pressure compensator. A pressure source is connected to two flow paths, each with a fixed orifice and a variable orifice that acts as a variable load. Along one flow path a pressure compensator is installed.

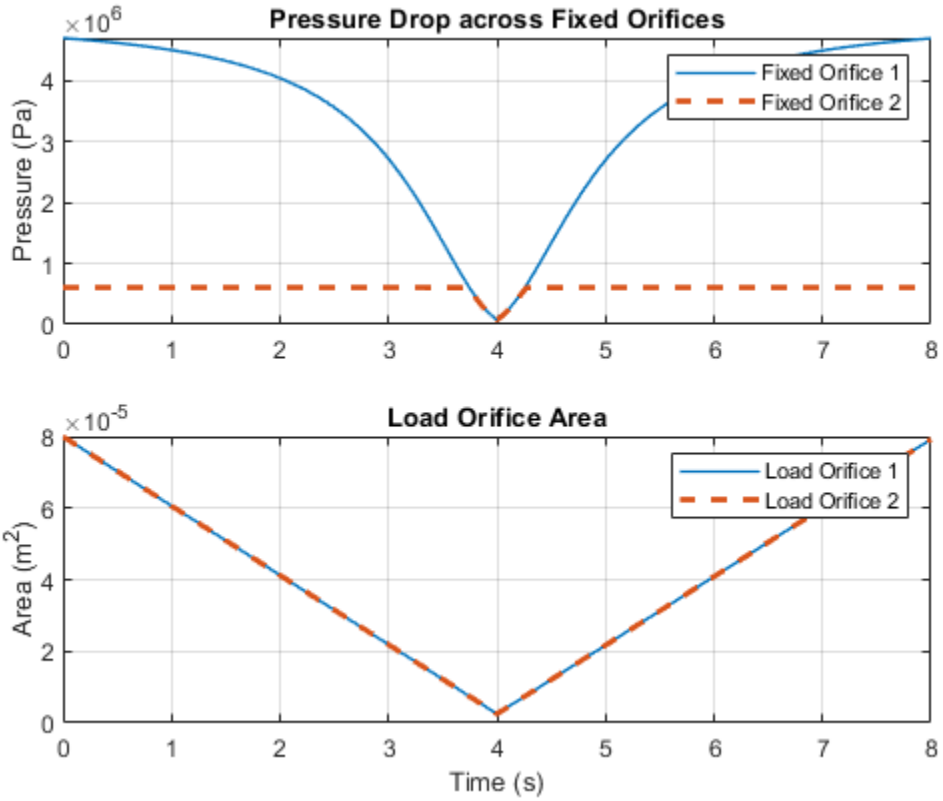
The pressure compensator is set to maintain a 6 bar pressure differential across the orifice. As a result, the flow rate along that flow path remains nearly constant as long as the pressure differential across the flow control is bigger than 6 bar. This is not the case for the flow path with no pressure compensator where the flow rate varies with the load.

Model



Simulation Results from Simscape Logging

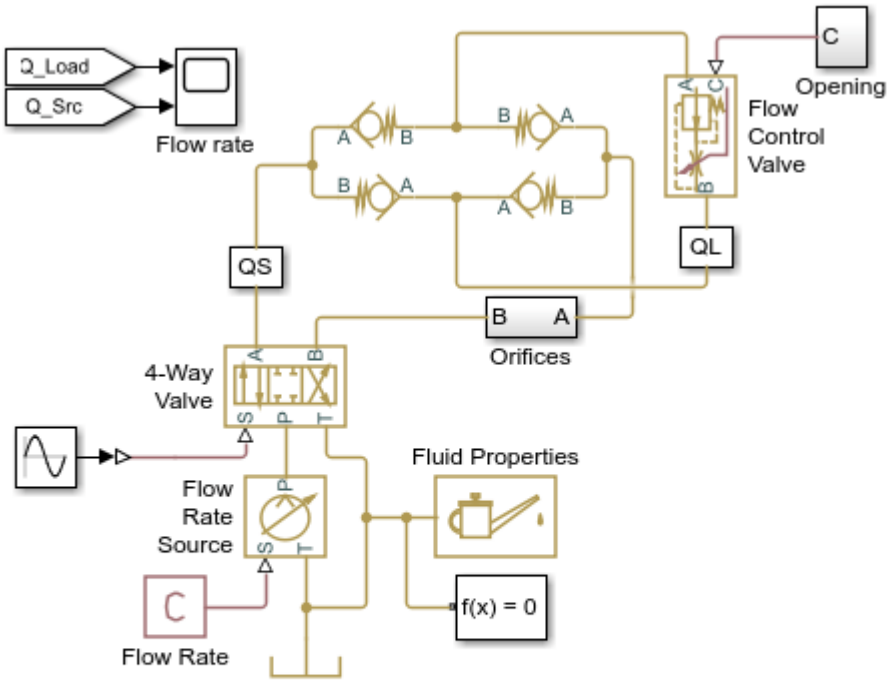
This plot shows the behavior of the pressure compensator, which controls the pressure drop across Fixed Orifice 2. As the downstream loads (variable orifices) are adjusted, the pressure drop across Fixed Orifice 2 stays constant at the set level in the pressure compensator. When the pressure drop across the loads gets too high, the pressure drop across Fixed Orifice 2 then falls below the set level in the pressure compensator.


Pressure Drop across Fixed Orifices
×10^6
Pressure (Pa)
Fixed Orifice 1
Fixed Orifice 2
Load Orifice Area
×10^-5
Area (m^2)
Load Orifice 1
Load Orifice 2
Time (s)

Hydraulic Flow Rectifier Circuit

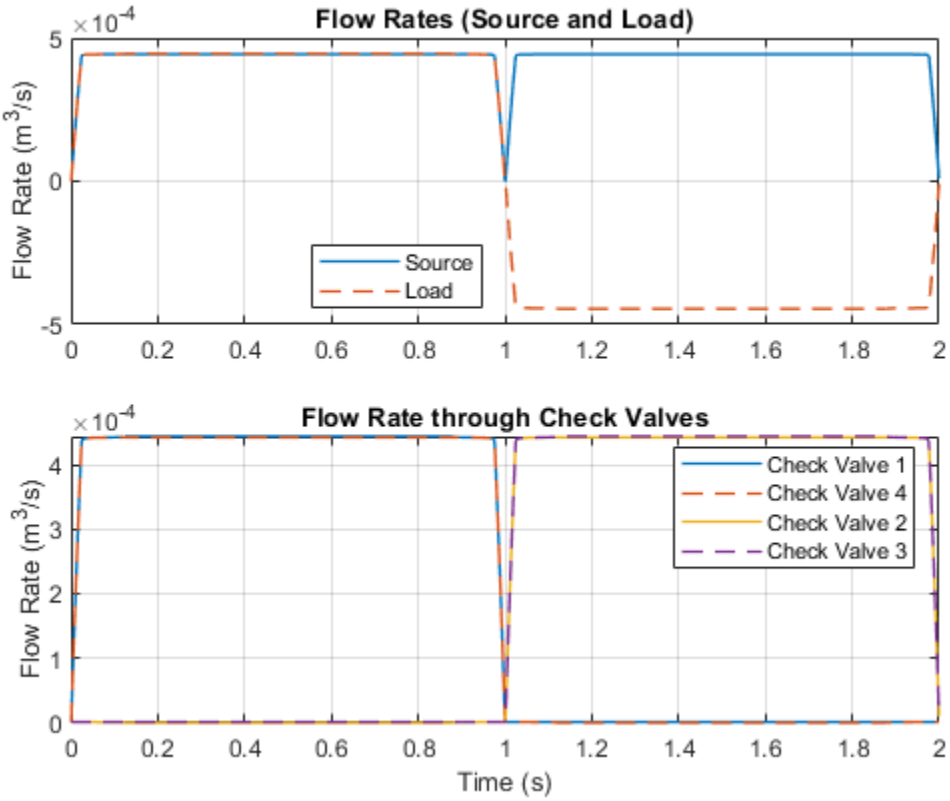
This example shows a flow rectifier circuit, which consists of four check valves and a flow control valve. It is used to control flow rate flowing in both directions with only one flow control valve. Similar to a Graetz circuit implemented with diodes, the check valves are arranged in such a way that flow always passes through the flow control valve in the same direction. Two other check valves, Flow BA Check Valve and Flow AB Check Valve, are used to select an orifice depending on the flow direction.

Model



Simulation Results from Simscape Logging

The plots below show the flow rate through various components in the system. Though the flow rate from the source (controlled by the 4-Way Valve) changes direction, the flow rate in the load (Flow Control Valve) remains in the same direction. The flow rates through the check valves are plotted to show how the flow proceeds through the flow rectifier.


Flow Rates (Source and Load)
×10^-4
5
0
-5
Flow Rate (m^3/s)
Source
Load
0
0.2
0.4
0.6
0.8
1
1.2
1.4
1.6
1.8
2
Flow Rate through Check Valves
×10^-4
4
3
2
1
0
Flow Rate (m^3/s)
Check Valve 1
Check Valve 4
Check Valve 2
Check Valve 3
Time (s)

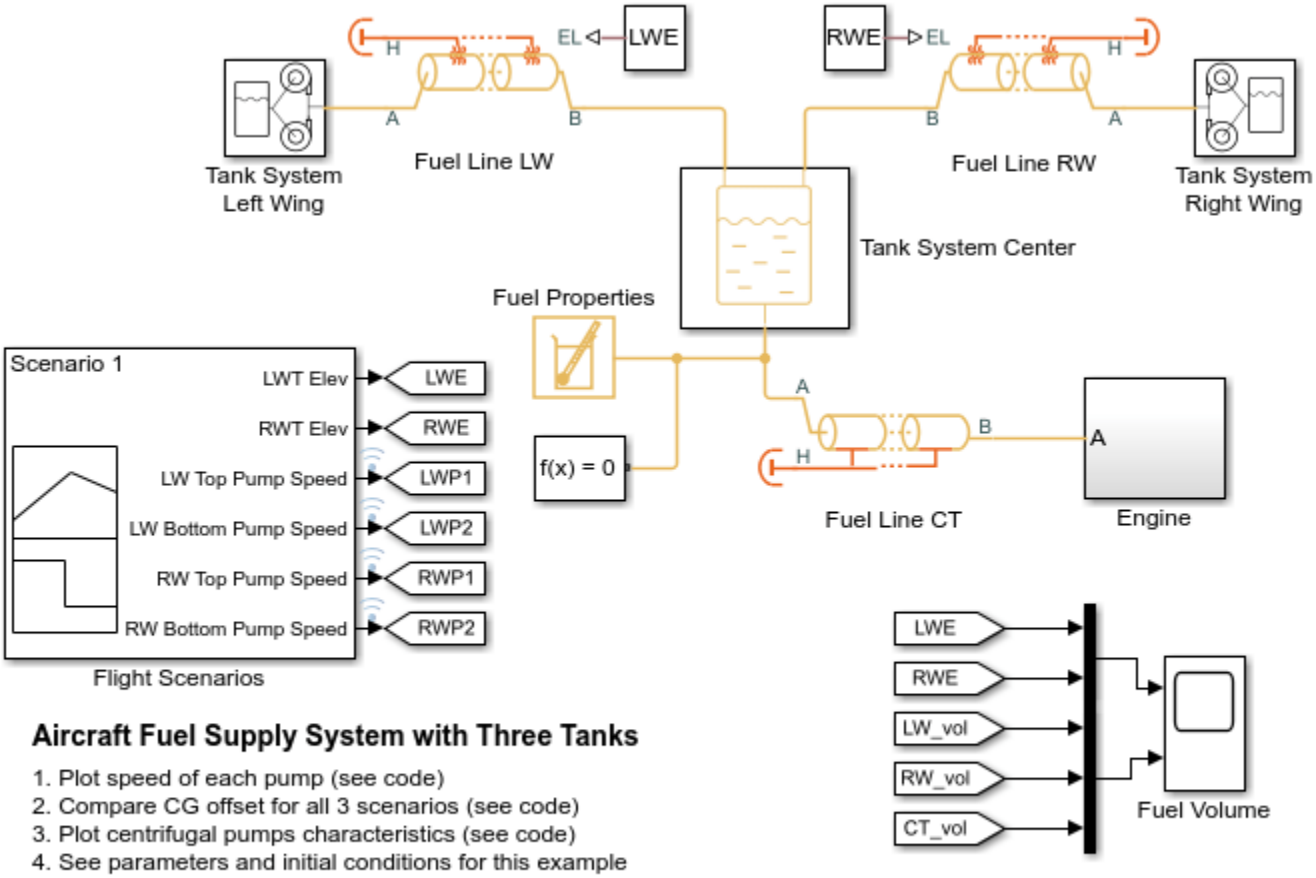
Aircraft Fuel Supply System with Three Tanks

The fuel supply system represented in the example consists of three tanks and an engine. The tanks are connected by the fuel lines and the pressure drop in the fuel lines depends on the bank angle of the aircraft or the relative elevation of the ends of the fuel lines. The central tank bottom is elevated by 36 in with respect to the reference plane, which is drawn horizontally through the entry point of the engine. The side tank bottoms are elevated by 4.2 in each at zero bank angle.

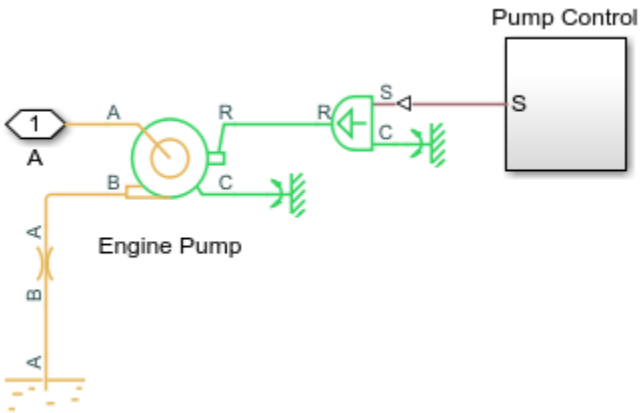
The engine is fed from the central tank, while fuel from the left wing tank and the right wing tank is pumped to the central tank with respective pumping stations. Each pumping station consists of two centrifugal pumps, connected in parallel, with check valves installed in the pump outlets to prevent back flow. The pumps are driven by prime movers at angular velocity of 120 rev/s. The movers are simulated with the ideal torque source. The fuel lines with varying elevation are represented by the Pipe Variable Elevation (TL) block.

In the left wing, right wing, and central tanks, the controllers cut the outlet off if fluid volume in the tank becomes less than preset volume. In the central tank, the controllers also prevent the tank from overflowing. In all tanks, the cutoff is performed with the 2-way valves. The model can be simulated under different flight scenarios including different pump failure.

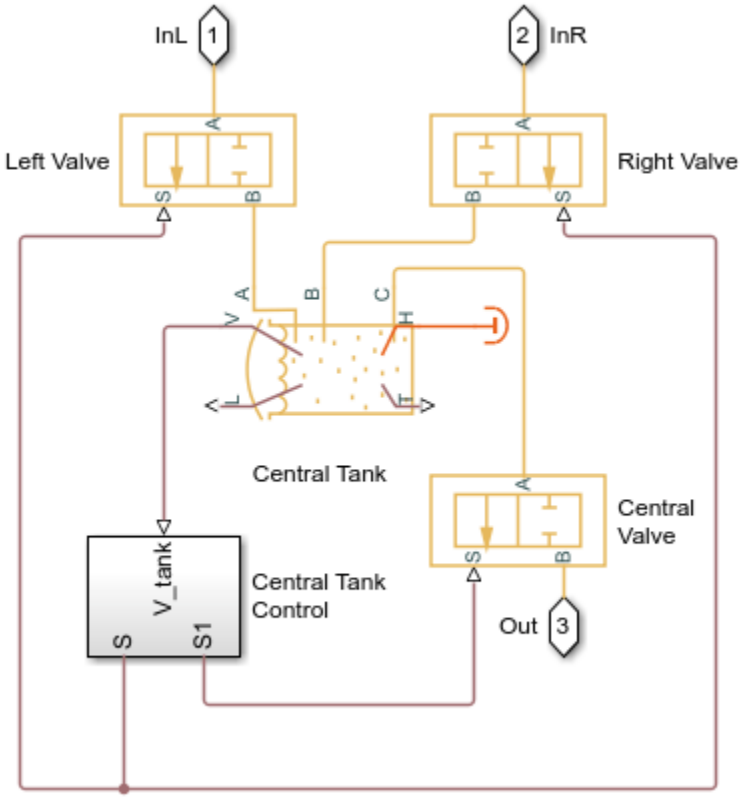
Model



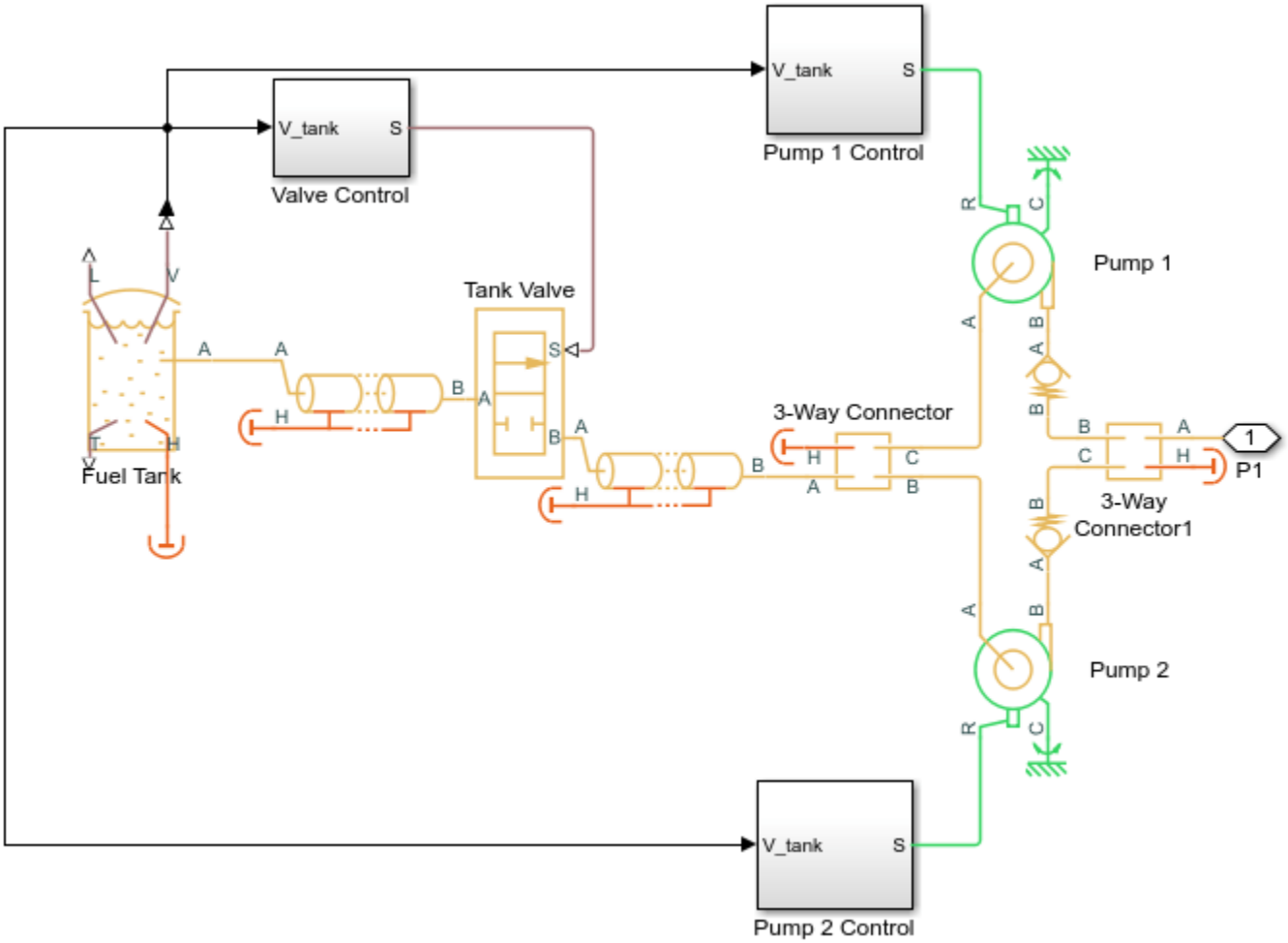
Engine Subsystem



Tank System Center Subsystem

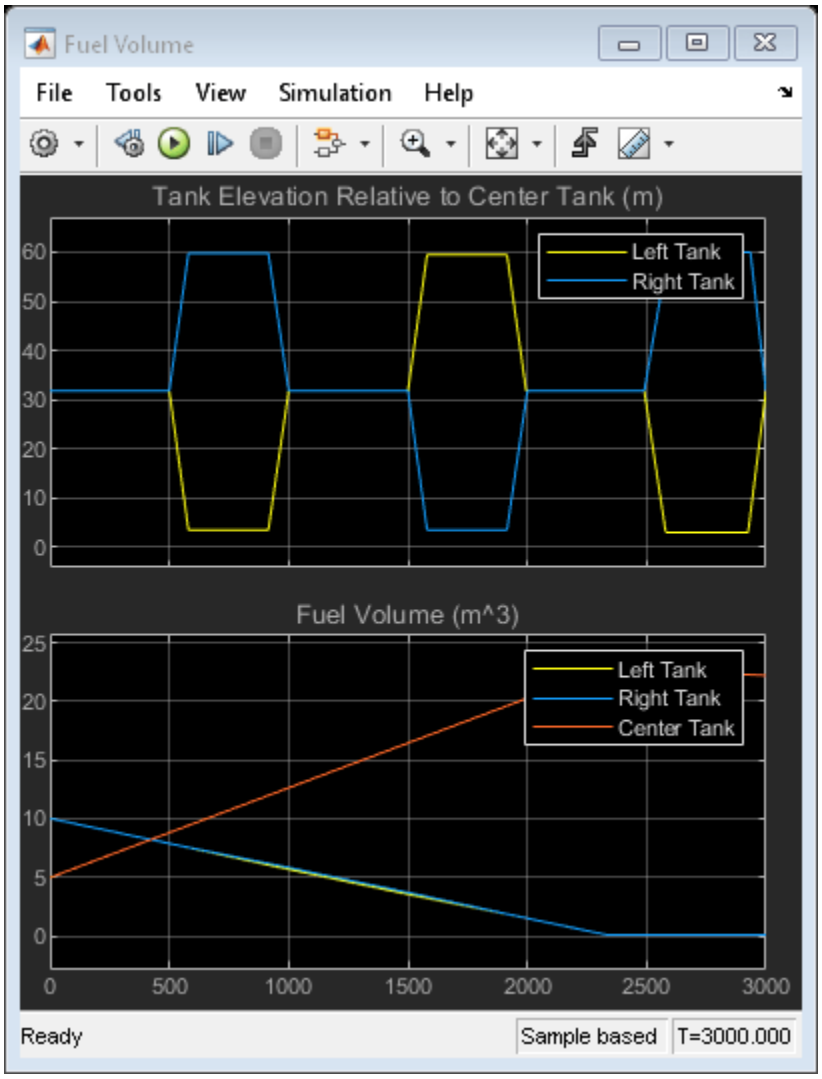


Tank System Left Wing Subsystem



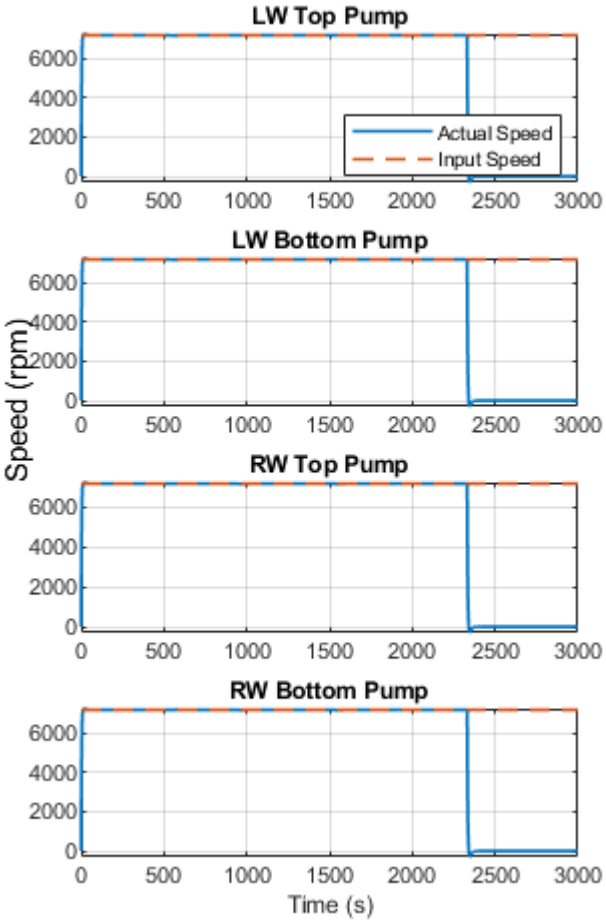
Simulation Results from Scopes

The plots below show the amount of fuel remaining in the tanks. The rate of fuel consumption from each tank is affected by the pressure drop across the fuel lines due to the elevation change relative to the center tank. Since the pump flow rate controllers keep flow rates independent of the aircraft bank angle, the effects of elevation on the tank flow rate will be small.

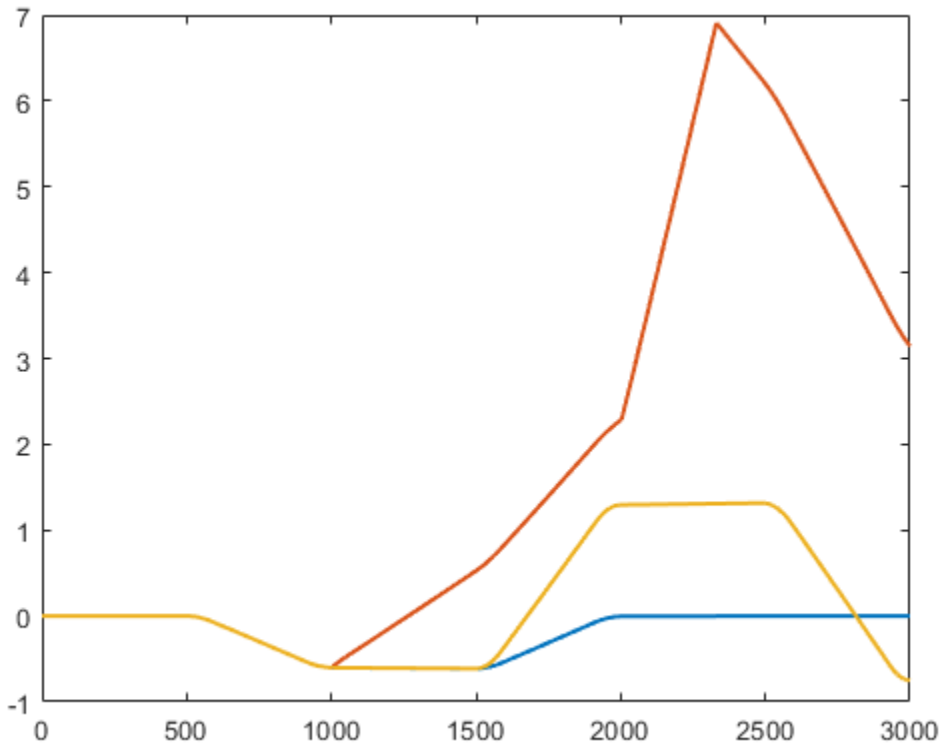


Simulation Results from Simscape Logging

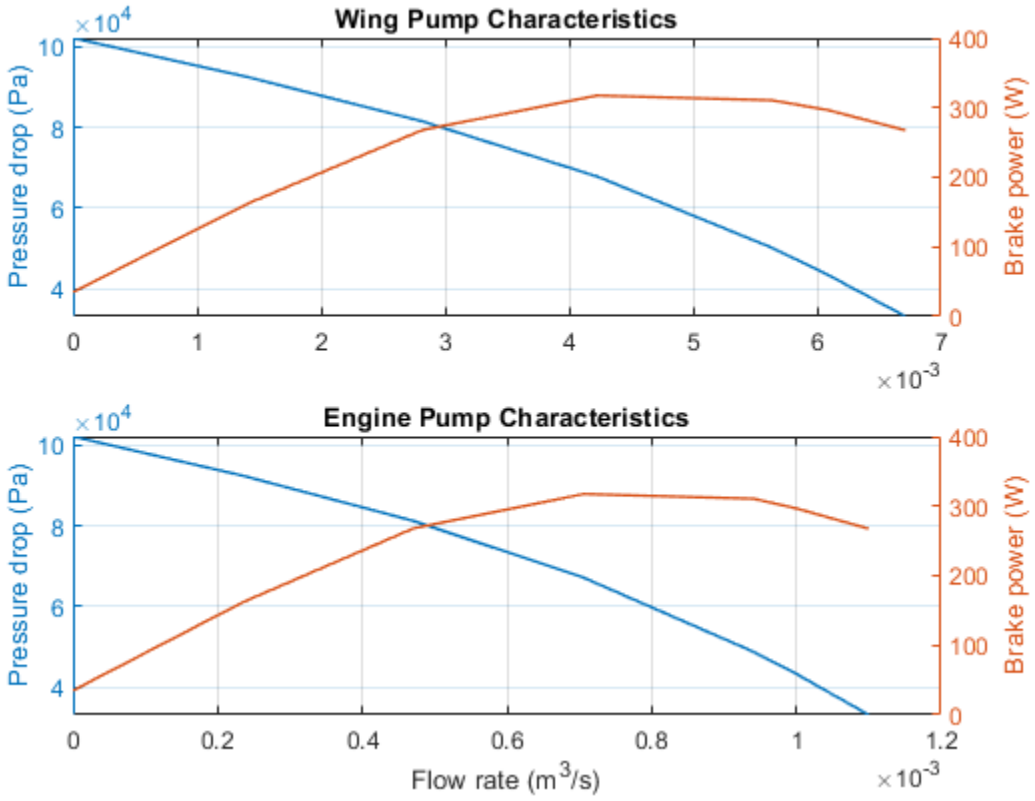
The plots below show the speed for all four pumps in the fuel supply system. For Flight Scenarios 2 and 3, pump speeds drop below the maximum speed to observe the effect on the fuel levels in the tanks.



The plot below shows the CG offset due to fuel distribution as a percentage of the half distance between left and right fuel tanks. The plots show the results for three tests where different combinations of pump failures are triggered.



The plots below show the characteristic curves of centrifugal pumps at reference condition in the fuel supply and engine systems.


Wing Pump Characteristics
×10^4
Pressure drop (Pa)
Brake power (W)
×10^-3
Engine Pump Characteristics
×10^4
Pressure drop (Pa)
Brake power (W)
Flow rate (m^3/s)
×10^-3

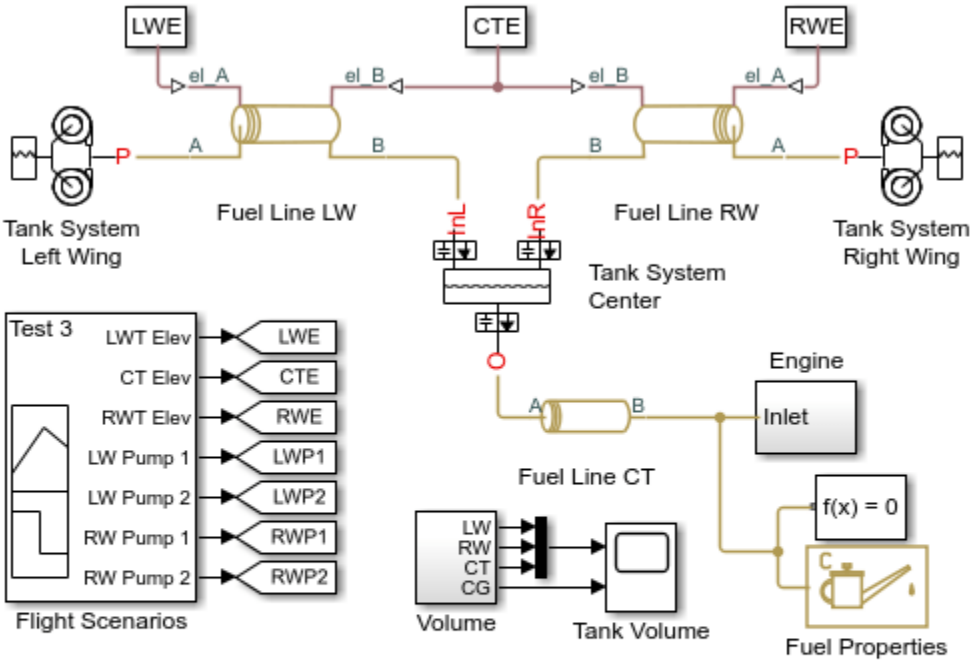
Aircraft Fuel Supply System with Three Tanks

The fuel supply system represented in the example consists of three tanks and an engine. The engine is fed from the central tank, while fuel from the left wing tank and the right wing tank is pumped to the central tank with respective pumping stations. Each pumping station consists of two centrifugal pumps, connected in parallel, with check valves installed in the pump outlets to prevent back flow. The pumps are driven by prime movers at angular velocity of 7200 rpm. The movers are simulated with the ideal angular velocity source.

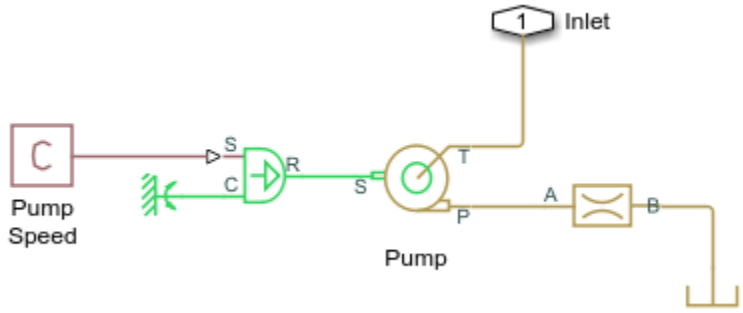
The central tank bottom is elevated by 36 in with respect to the reference plane, which is drawn horizontally through the entry point of the engine. The side tank bottoms are elevated by 4.2 in each.

The left wing and the right wing tanks are equipped with the volume control block intended to cut the outlet off if fluid volume in the tank becomes less than preset volume. A similar block in the central tank controls both the tank overfill and the exposure of its highest port. In all tanks, the cutoff is performed with the 2-way valves.

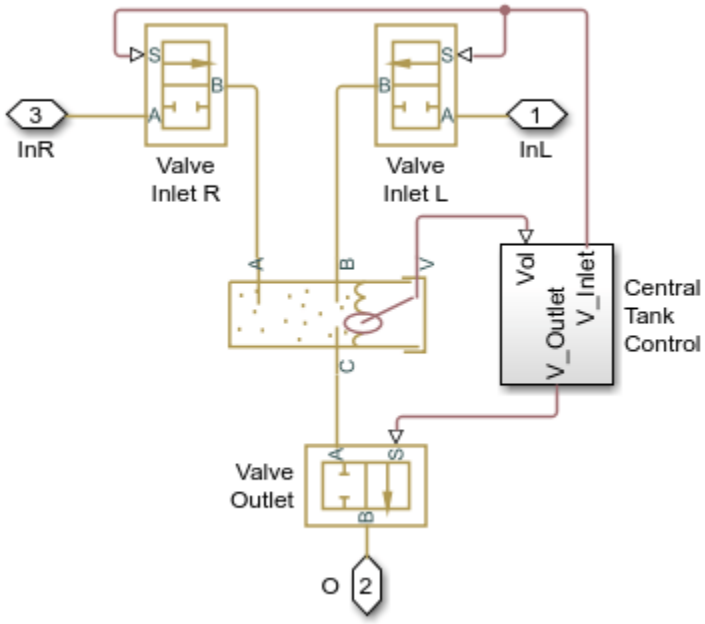
Model



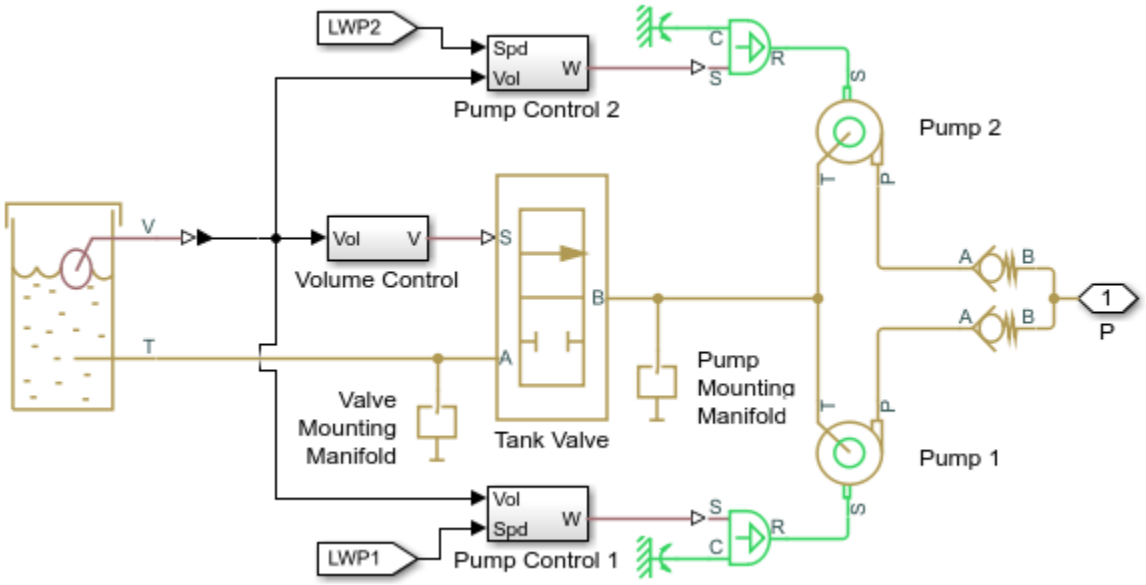
Engine Subsystem



Tank System Center Subsystem

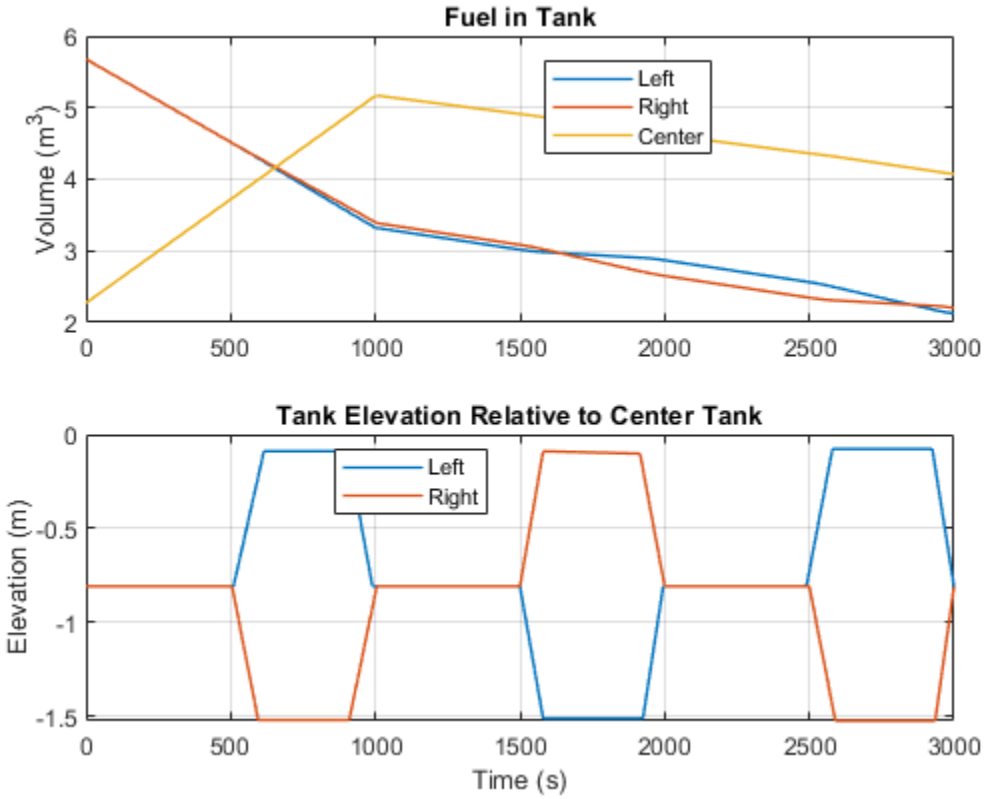


Tank System Left Wing Subsystem

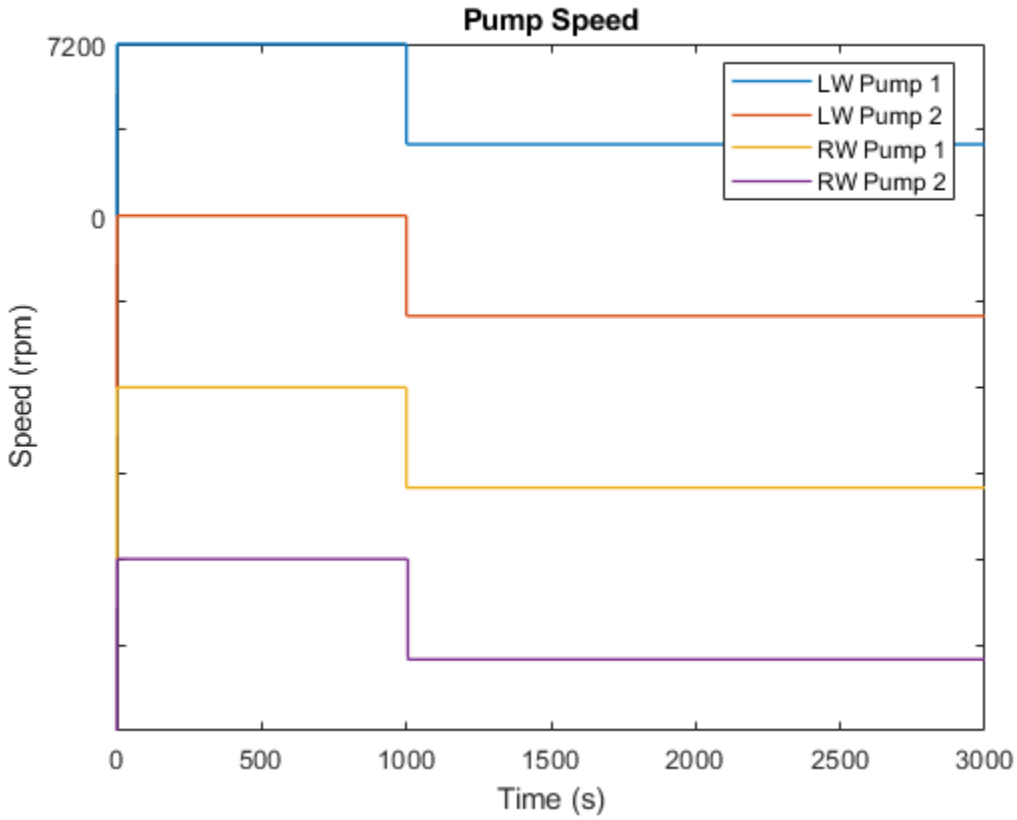


Simulation Results from Simscape Logging

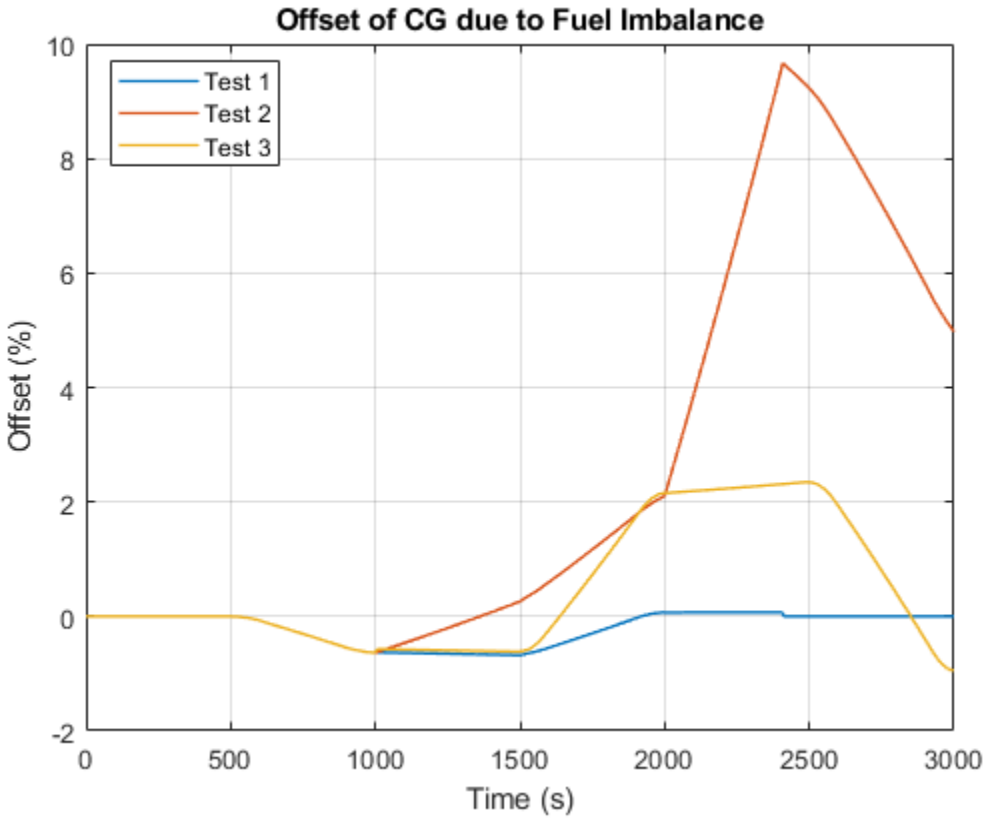
The plots below show the amount of fuel remaining in the tanks. The rate of fuel consumption from each tank is affected the pressure drop across the fuel lines due to the elevation change relative to the center tank.



The plots below show the speed for all four pumps in the fuel supply system. For some test sequences, pump speeds drop below the maximum speed to observe the effect on the fuel levels in the tanks.



The plot below shows the offset of the CG from the fuel on the aircraft as a percentage of the half the distance between the left and right fuel tanks. The plots show the results for three tests where different combinations of pump failures are triggered.

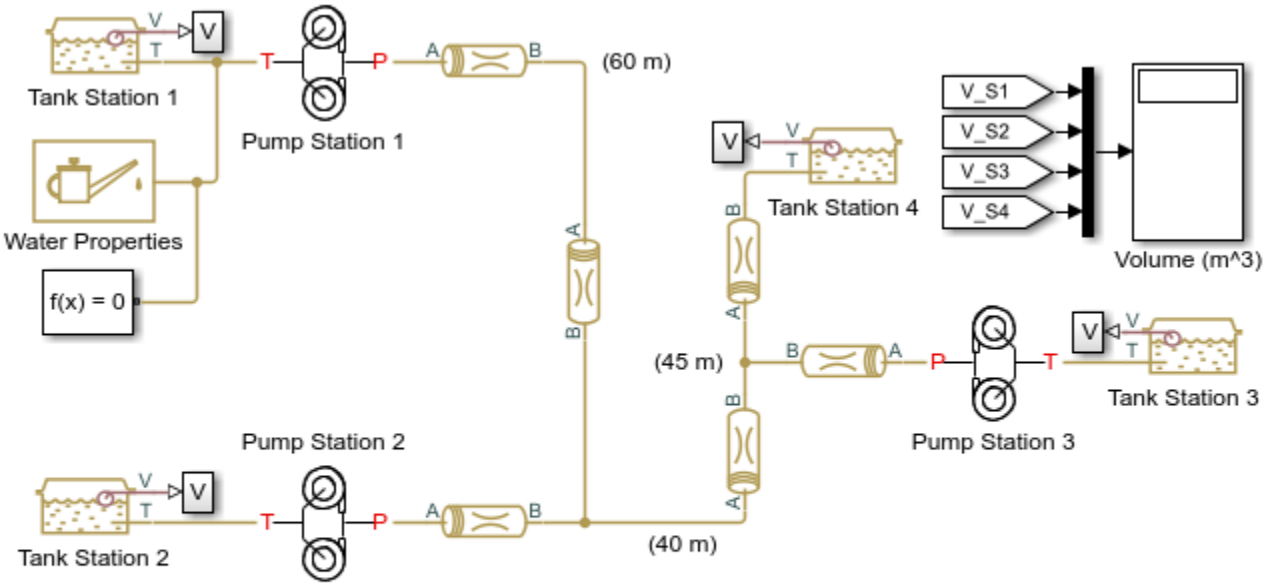

Offset of CG due to Fuel Imbalance
Test 1
Test 2
Test 3
Offset (%)
Time (s)
10
8
6
4
2
0
-2
0
500
1000
1500
2000
2500
3000

Water Supply System

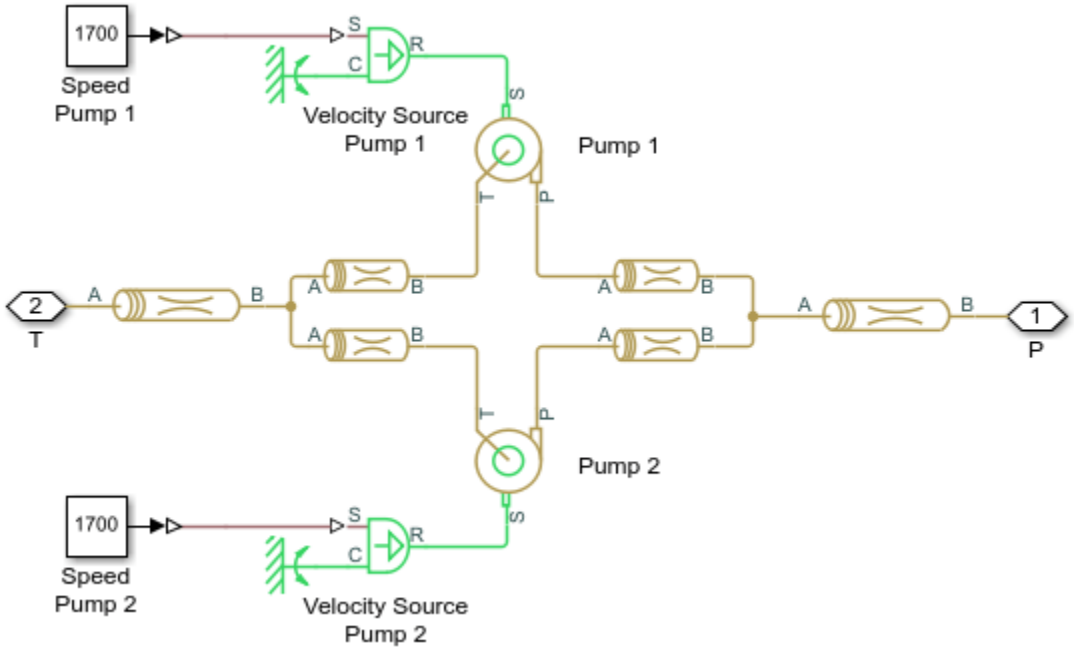
This example shows a water supply system consisting of three pumping stations located at 45, 25, and 30 m with respect to to the reference plane, respectively. All three stations are expected to pump water in a tank located at 61 m. All tanks are large enough to assume that the fluid level remains nearly constant, allowing them to be modeled with the Constant Head Tank block. The initial volume of water in each tank is set to 100 m^3. Each pumping station consists of a tank, two centrifugal pumps installed in parallel, and a prime mover rotating at 1700 rpm. The pump characteristics are specified using lookup tables.

The objective of the simulation is to determine steady-state flow rates and pressures. For this reason, all pipes are simulated with the Resistive Pipe LP block and no system dynamics are observed.

Model

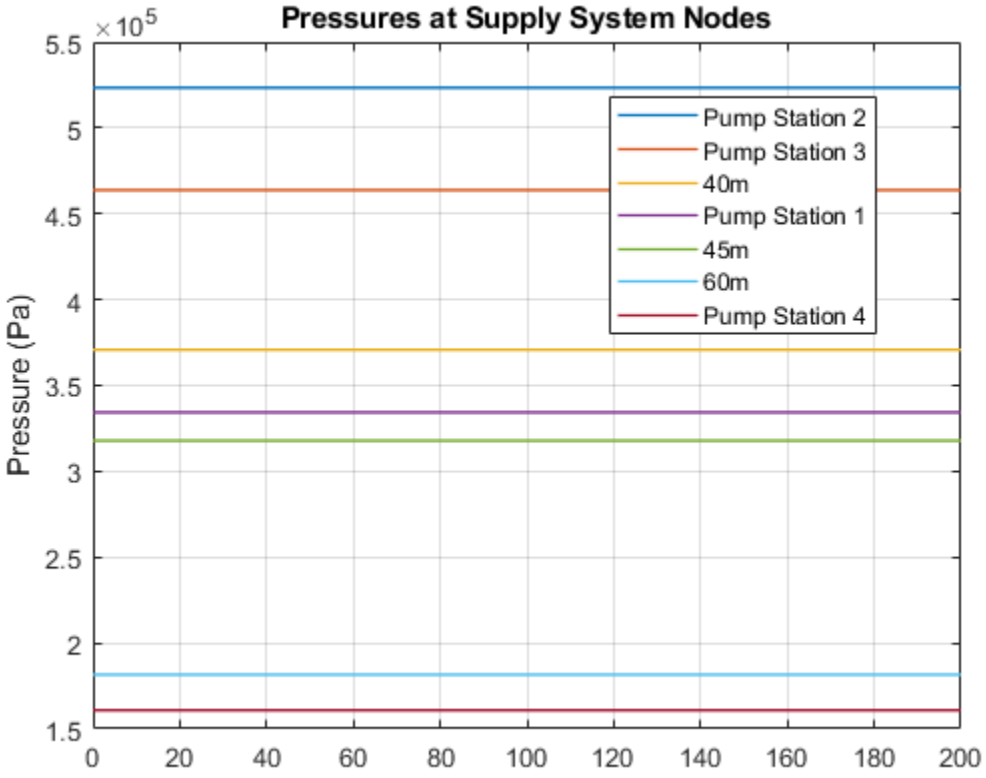


Pump Station 1 Subsystem



Simulation Results from Simscape Logging

The plot below shows the pressure at each node in the water supply system.

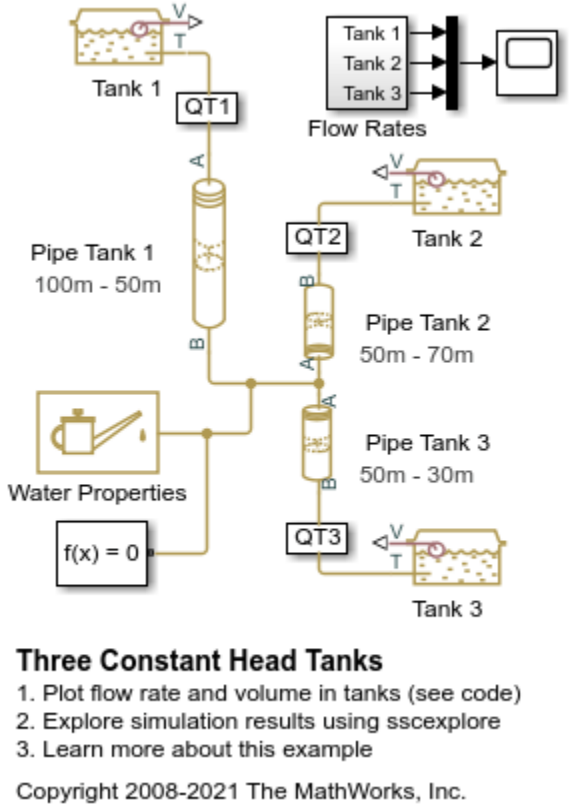

Pressures at Supply System Nodes
×10^5
5.5
5
4.5
4
3.5
3
2.5
2
1.5
Pressure (Pa)
0
20
40
60
80
100
120
140
160
180
200
Pump Station 2
Pump Station 3
40m
Pump Station 1
45m
60m
Pump Station 4

Three Constant Head Tanks

This example shows a classical problem of fluid transportation: to determine flow rates, pressures, and fluid volumes in a system built of three constant head tanks. The tanks are located at different elevations and connected with pipelines combined together in a common node located at 50 meters with respect to the reference plane. The simulation time is set to 50 seconds, which is enough for system variables to settle down and reach near steady-state values.

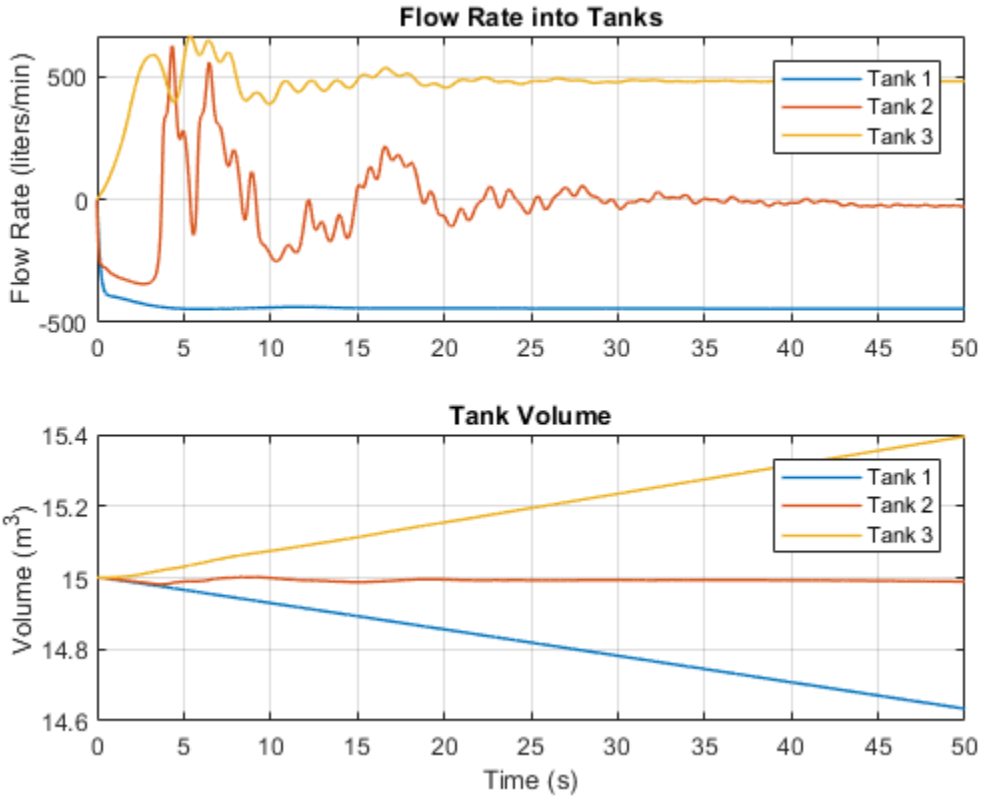
The pipelines are simulated with the Segmented Pipe LP block, which accounts for hydraulic losses, fluid inertia, and the head due to different node elevations.

Model



Simulation Results from Simscape Logging

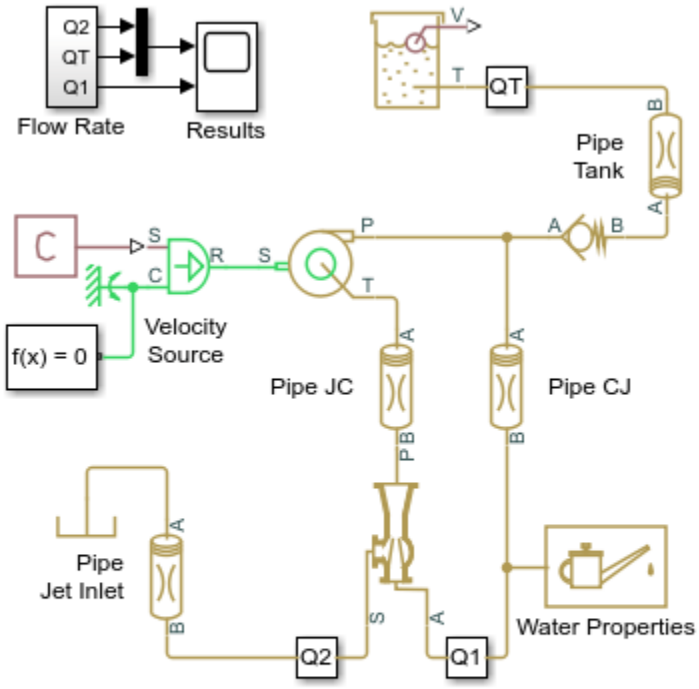
The plots below show the flow rate into and volume of water in each tank. Simulation results show that the pipelines account for fluid inertia, hydraulic losses, and head due to elevation.


Flow Rate into Tanks
Flow Rate (liters/min)
500
0
-500
0
5
10
15
20
25
30
35
40
45
50
Tank 1
Tank 2
Tank 3
Tank Volume
Volume (m3)
15.4
15.2
15
14.8
14.6
Time (s)

Well with Jet Pump

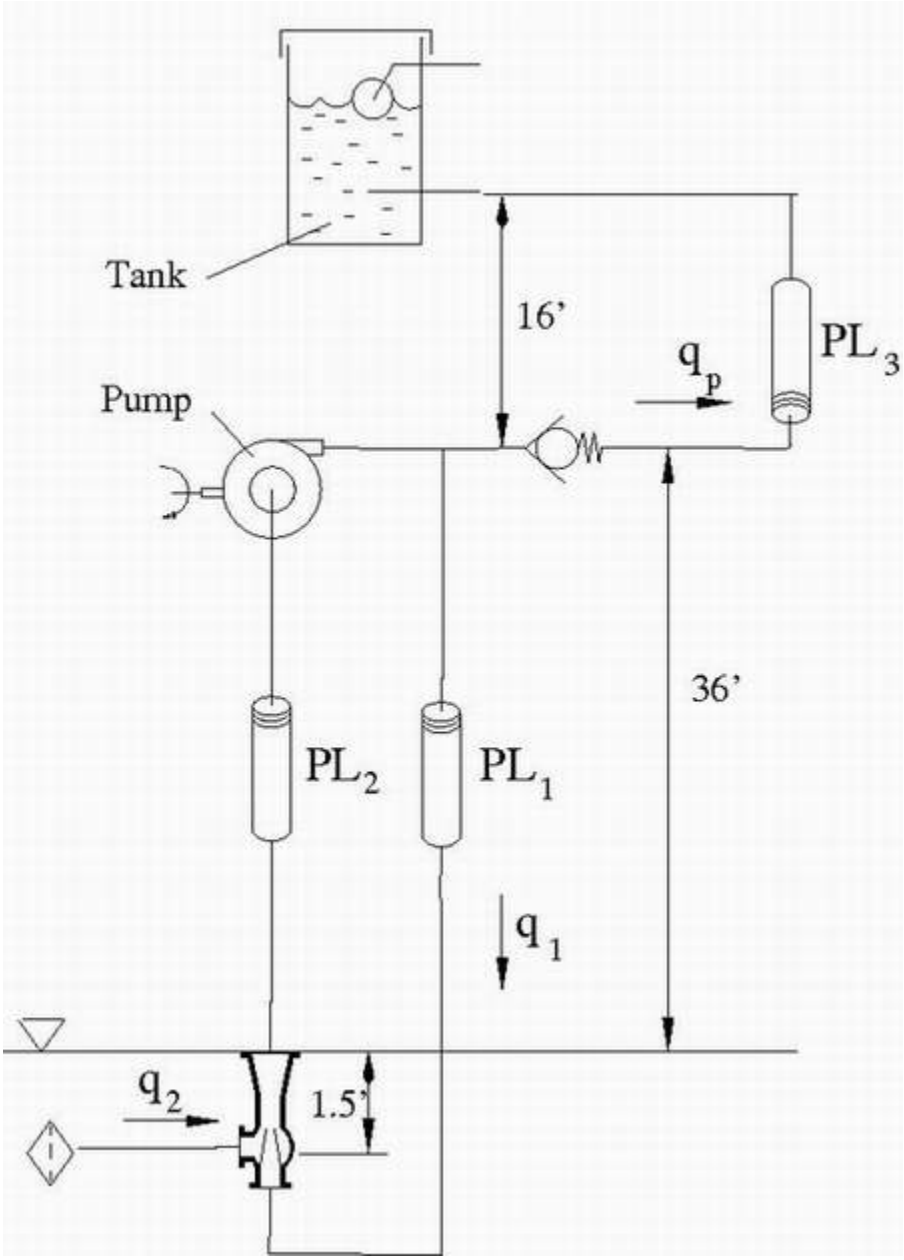
This example shows a well jet pump installation. The well jet pump installation presented in this example consists of a surface-mounted centrifugal pump and a jet pump installed in the well below the surface of the water.

Model



System Schematic

In this well jet pump installation, the distance between the centrifugal pump and water level is 36 feet. The jet pump installed in the well 1.5 feet below the surface of the water. See the schematic below.

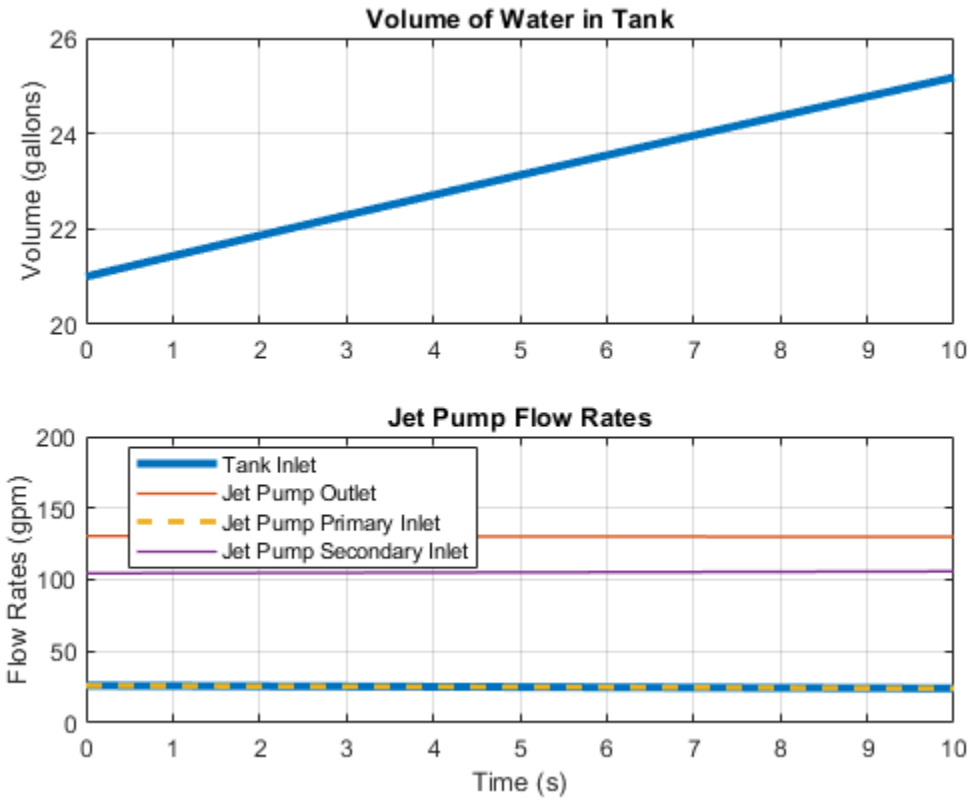


The centrifugal pump discharge pressure is 20 psi and its delivery at nominal conditions is expected to be about 130 gallons/min. The pipe between the centrifugal pump output and the jet pump inlet has a 2 inch diameter and is 37.5 feet long. The diameter of the pipe connecting the jet pump output and centrifugal pump input is 4 inches, and it is 36 feet long. The centrifugal pump output is divided into two parts: one goes through the check valve into the tank, and the rest is diverted to the inlet of the jet pump. The delivery of the pump assembly must be greater than 25 gallons/minute. The centrifugal pump NPSH must be greater than 2 psi.

The jet pump parameters were initially determined based on recommendations provided in the *Pump Handbook* by Karassic, I. J., et al (third edition, 2001), and then modified to simplify the calculations. As a result, the primary flow rate (q_1) of the jet pump is 104.5 gpm, the secondary flow rate (q_2) is 26.5 gpm, and the output of the pump assembly (q_p) is 26.5 gpm.

Simulation Results from Simscape Logging

The plots below show flow rates into and out of the jet pump and the volume of the water tank.

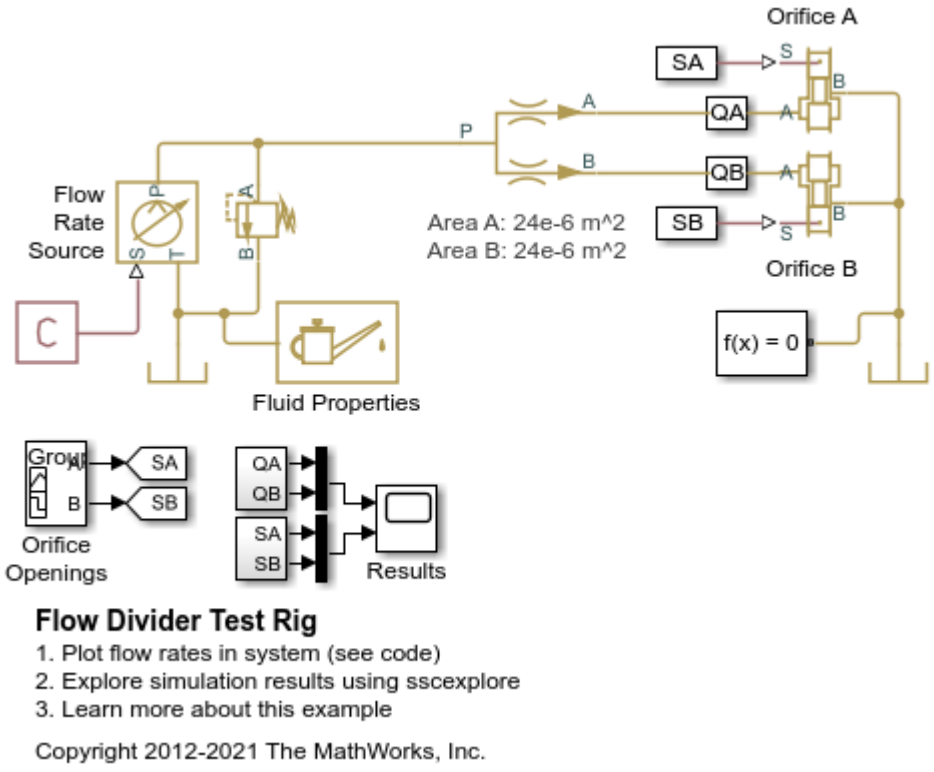


Flow Divider Test Rig

This example shows the behavior of a flow divider subjected to varying loads. The model consists of a power unit, flow divider, and two consumers. The power unit is modeled using a flow rate source and a pressure relief valve. The Flow Divider block splits the flow between the two variable orifices.

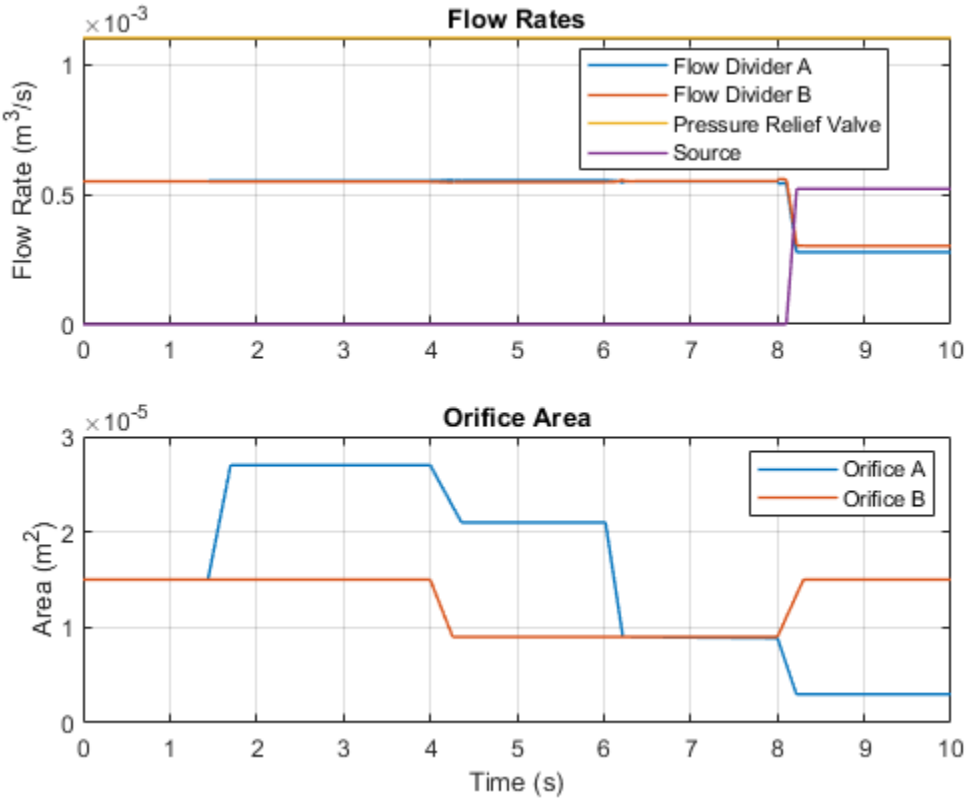
To test the ability of the Flow Divider block to split the flow equally between the two consumers, the load in the branches is varied by changing the passage area of the load orifices.

Model



Simulation Results from Simscape Logging

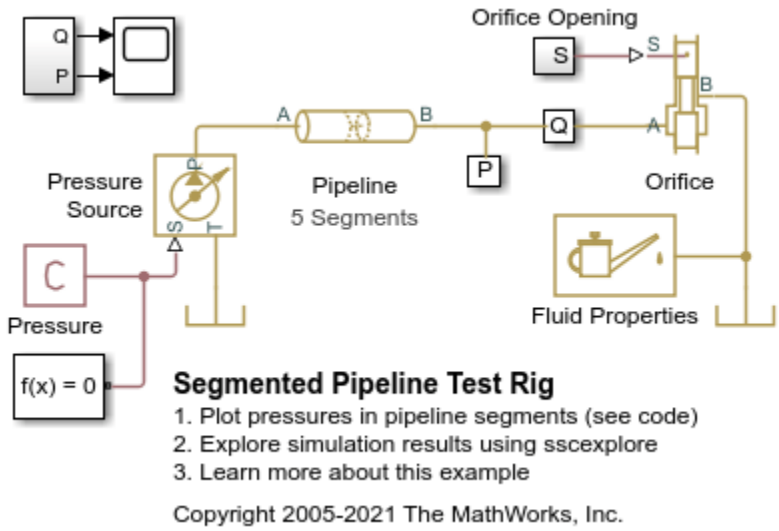
The plots below show the behavior of the flow divider as the load on each branch is varied. The flow divider manages to divide the flow nearly equal to the ratio of the flow divider orifice areas for the entire simulation. Near the end, the restrictions downstream of the flow divider become high enough that flow is diverted through the pressure relief valve.


Flow Rates
×10^-3
Flow Rate (m^3/s)
Flow Divider A
Flow Divider B
Pressure Relief Valve
Source
Orifice Area
×10^-5
Area (m^2)
Orifice A
Orifice B
Time (s)

Segmented Pipeline Test Rig

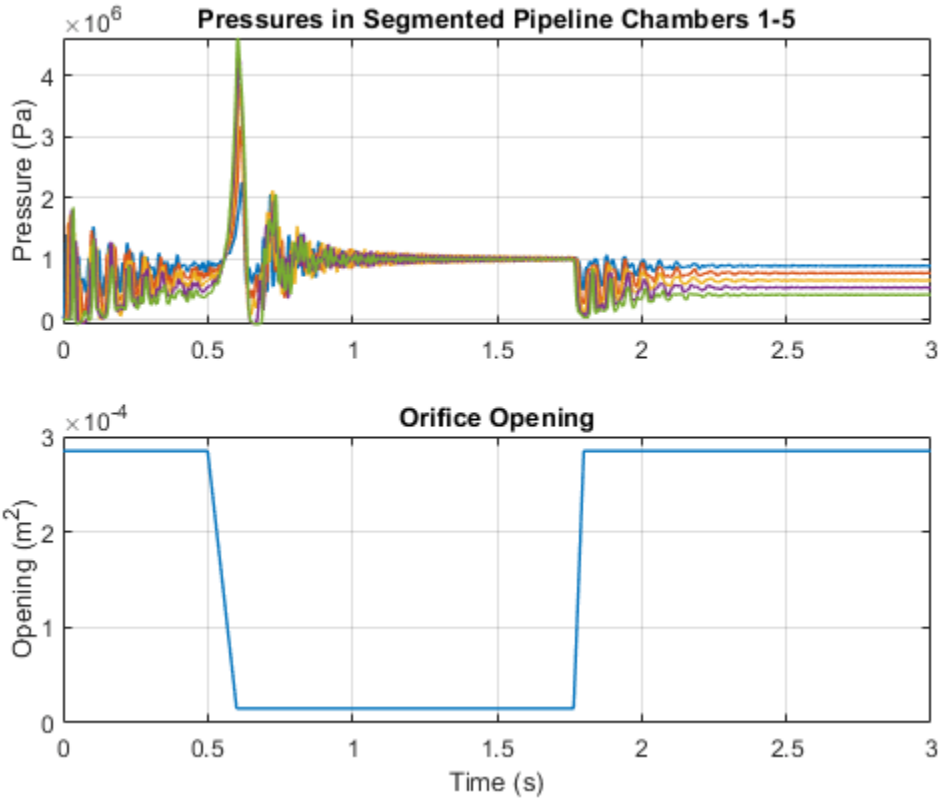
This example shows a test rig used to investigate the water hammer effect, which occurs when a variable orifice is abruptly shut off with full flow rate flowing through the branch. The water hammer behavior of the pipeline can be investigated by changing pipeline dimensions, number of segments in the pipeline model, fluid properties, valve closure speed, and minimum opening.

Model



Simulation Results from Simscape Logging

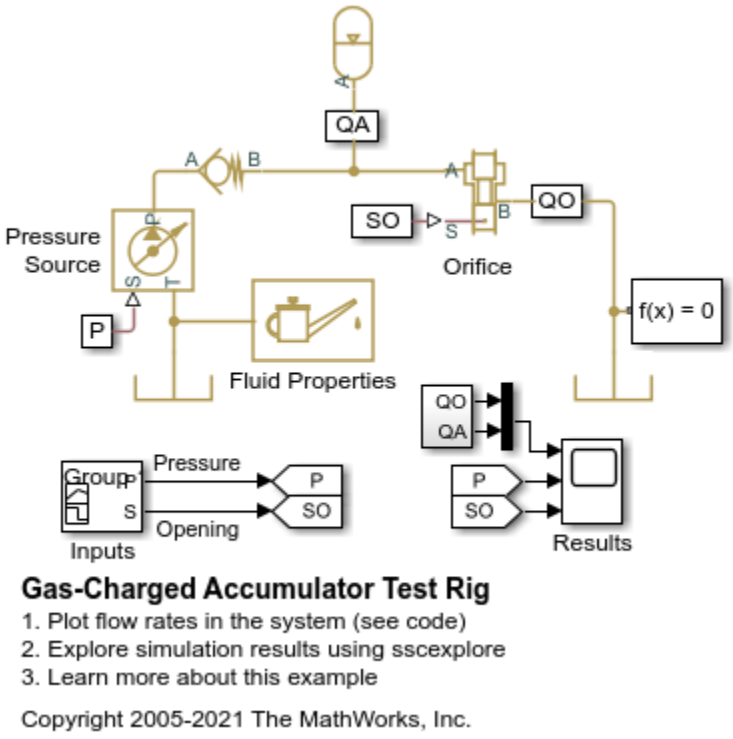
The plot below shows the pressure in each of the chambers in the segmented pipeline. The orifice at the outlet of the pipeline closes suddenly, creating pressure waves within the pipeline (water hammer effect).


Pressures in Segmented Pipeline Chambers 1-5
×10^6
Pressure (Pa)
0
1
2
3
4
0
0.5
1
1.5
2
2.5
3
Orifice Opening
×10^-4
Opening (m^2)
0
1
2
3
Time (s)

Gas-Charged Accumulator Test Rig

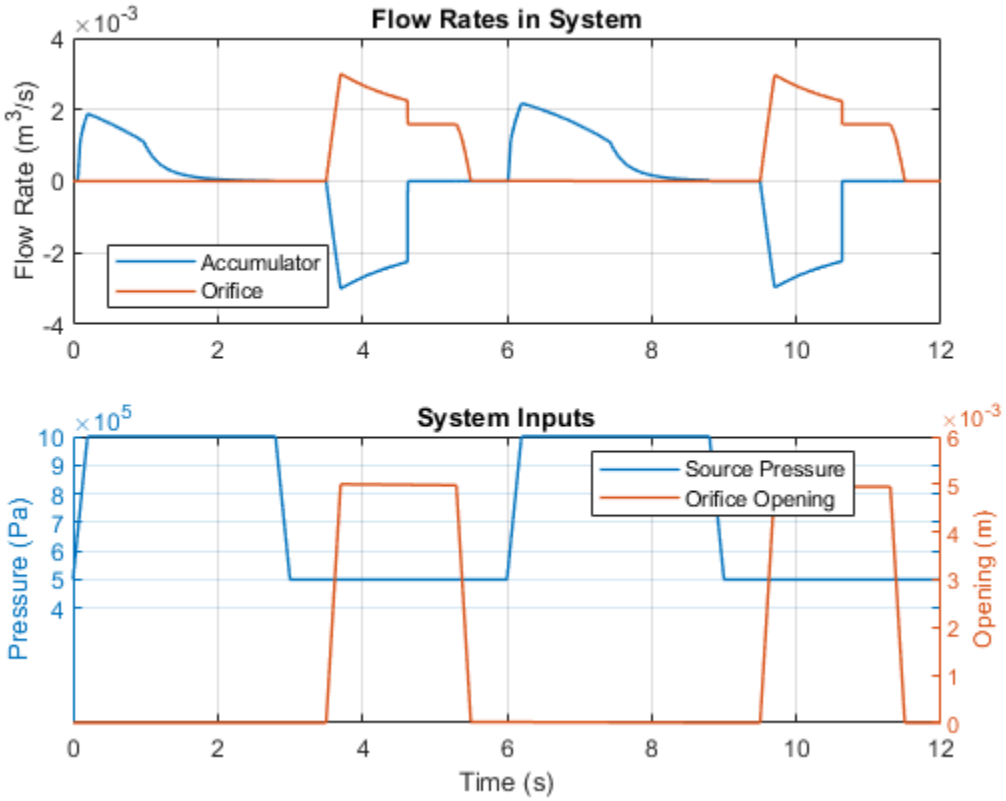
This example shows a testrig for a gas-charged accumulator. The accumulator is charged by the pressure source while the orifice is closed, held at its charged pressure by the check valve, and discharged as the orifice opens.

Model



Simulation Results from Simscape Logging

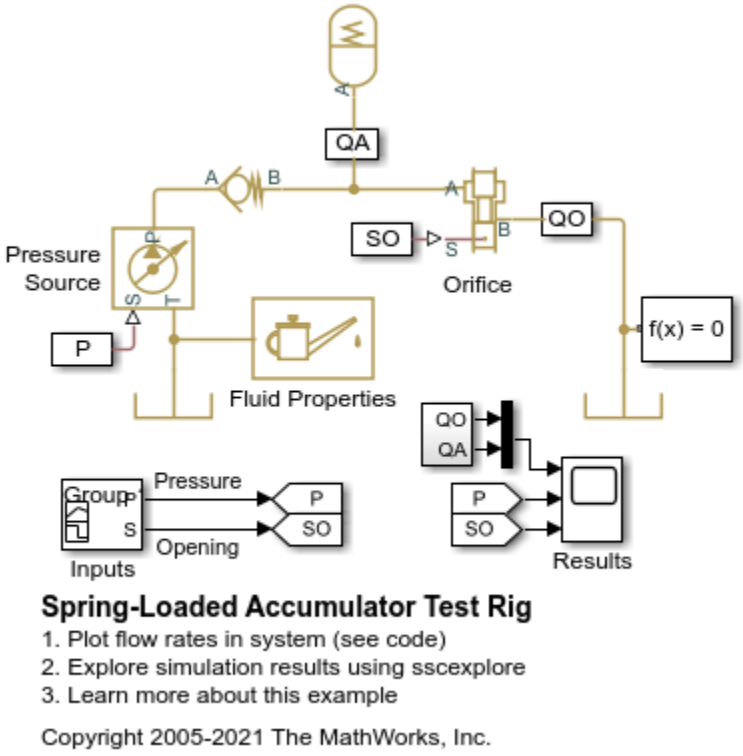
The plots below show two cycles of charging and discharging the accumulator. While the orifice is closed, the supply pressure is increased, charging the accumulator. The check valve holds the pressure in the accumulator as the supply pressure is dropped. The orifice opens, allowing the accumulator to discharge. Once the accumulator is empty, the flow rate through the orifice stays steady at a lower value due to the lower supply pressure. The orifice closes, and the cycle repeats.


Flow Rates in System
×10^-3
Flow Rate (m^3/s)
Accumulator
Orifice
System Inputs
×10^5
Pressure (Pa)
×10^-3
Opening (m)
Source Pressure
Orifice Opening
Time (s)

Spring-Loaded Accumulator Test Rig

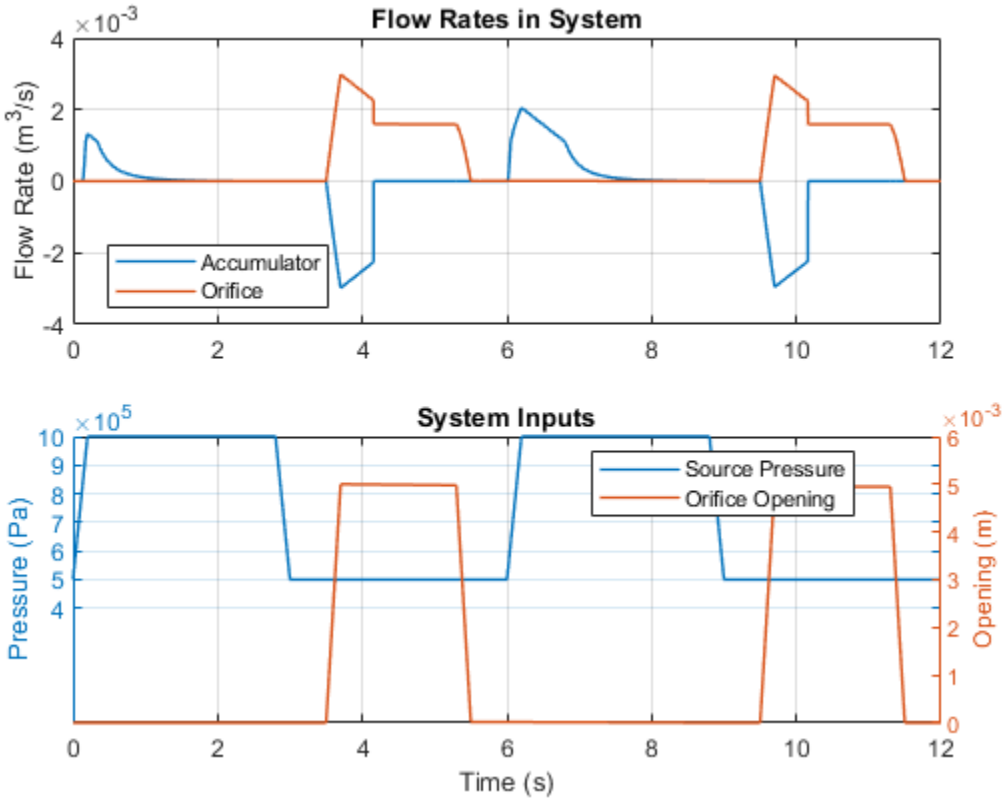
This example shows a testrig for a spring-loaded accumulator. The accumulator is charged by the pressure source while the orifice is closed, held at its charged pressure by the check valve, and discharged as the orifice opens.

Model



Simulation Results from Simscape Logging

The plots below show two cycles of charging and discharging the accumulator. While the orifice is closed, the supply pressure is increased, charging the accumulator. The check valve holds the pressure in the accumulator as the supply pressure is dropped. The orifice opens, allowing the accumulator to discharge. Once the accumulator is empty, the flow rate through the orifice stays steady at a lower value due to the lower supply pressure. The orifice closes, and the cycle repeats.

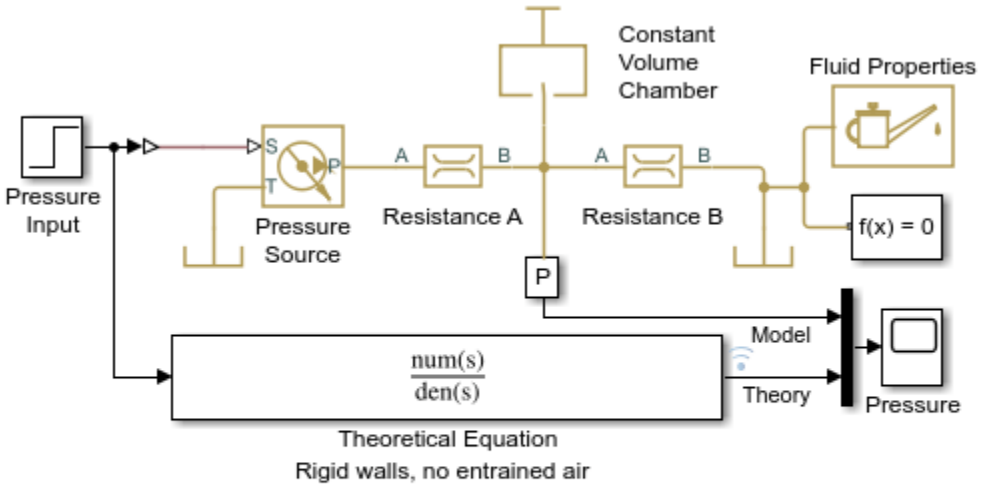

Flow Rates in System
×10^-3
Flow Rate (m^3/s)
Accumulator
Orifice
System Inputs
×10^5
Pressure (Pa)
×10^-3
Opening (m)
Source Pressure
Orifice Opening
Time (s)

Constant Volume Chamber Test Rig

This example shows the behavior of a constant volume chamber exposed to a step increase in pressure. The chamber is placed between two linear hydraulic resistances and subjected to an abrupt change in supply pressure. The chamber is a piece of a steel cylindrical pipe of 0.03 m internal diameter, 0.036 m external diameter, and 16.5 m length. If the chamber wall is set to rigid and the amount of entrapped air is zero, the pressure change at the chamber inlet can be determined analytically.

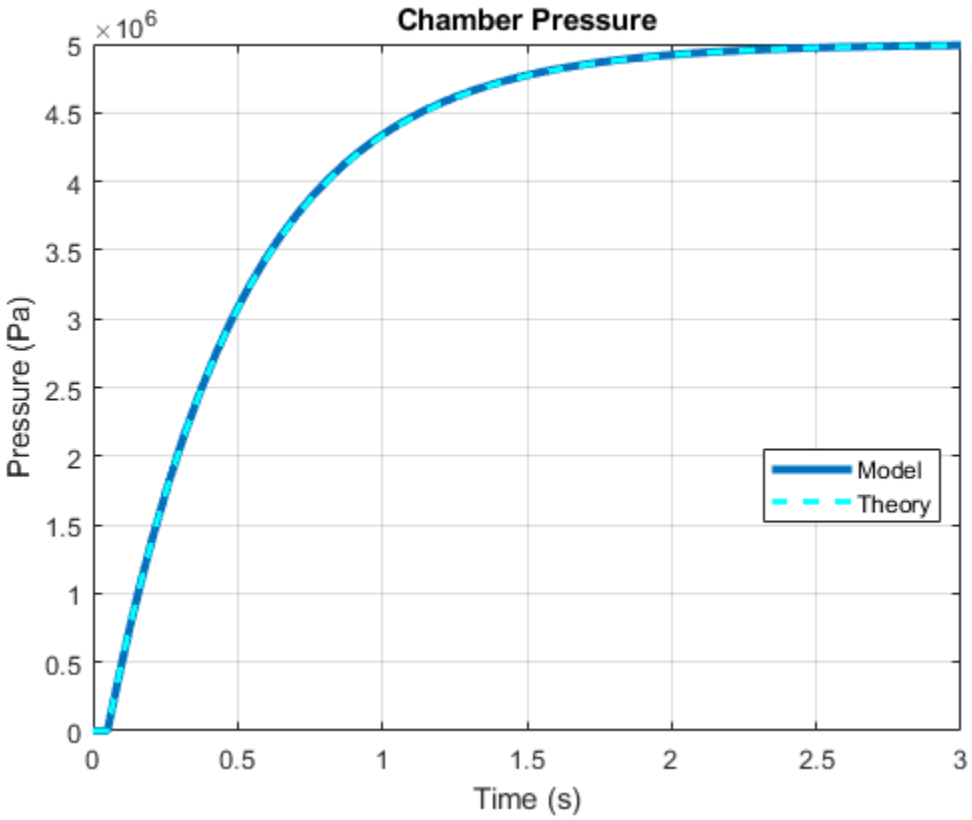
The chamber characteristics can be investigated by changing air content in the fluid, by switching from rigid to compliant walls, and by changing wall compliance and pipe viscoelasticity time constant.

Model



Simulation Results from Simscape Logging

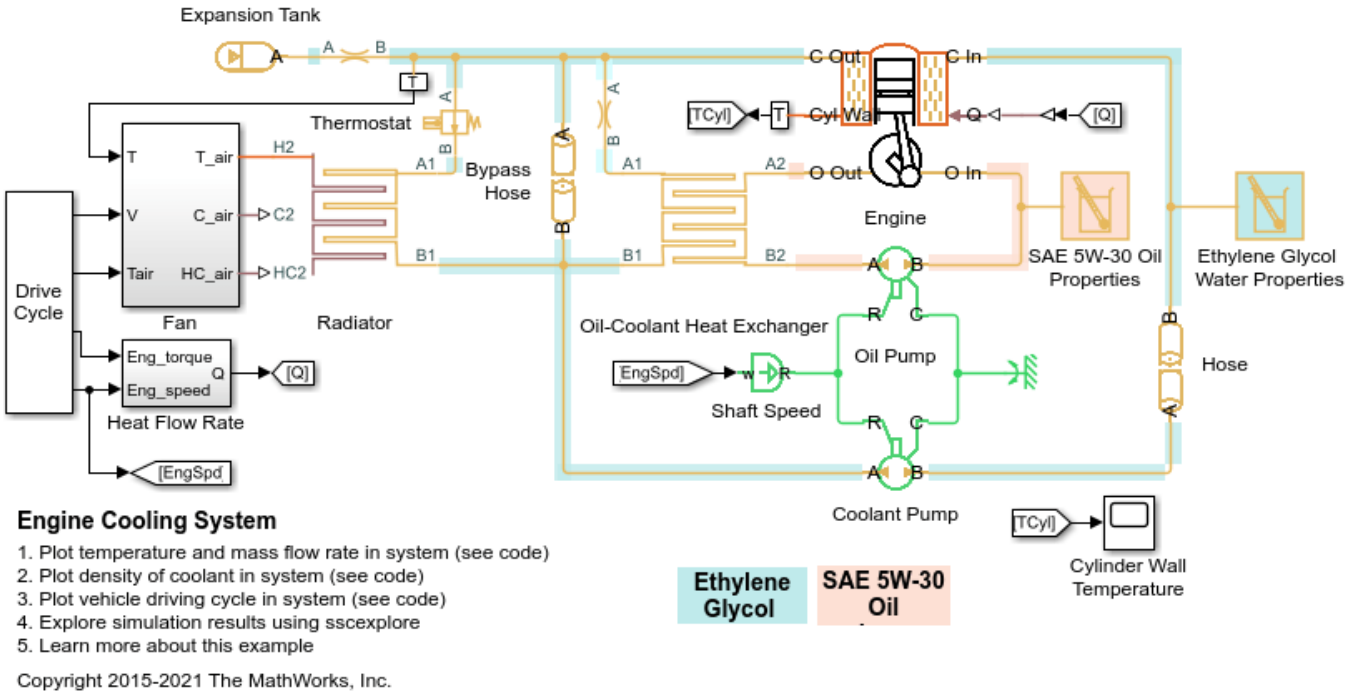
The plot below shows how the pressure in a constant volume chamber varies with time.


Chamber Pressure
×10^6
5
4.5
4
3.5
3
2.5
2
1.5
1
0.5
0
Pressure (Pa)
0
0.5
1
1.5
2
2.5
3
Time (s)
Model
Theory

Engine Cooling System

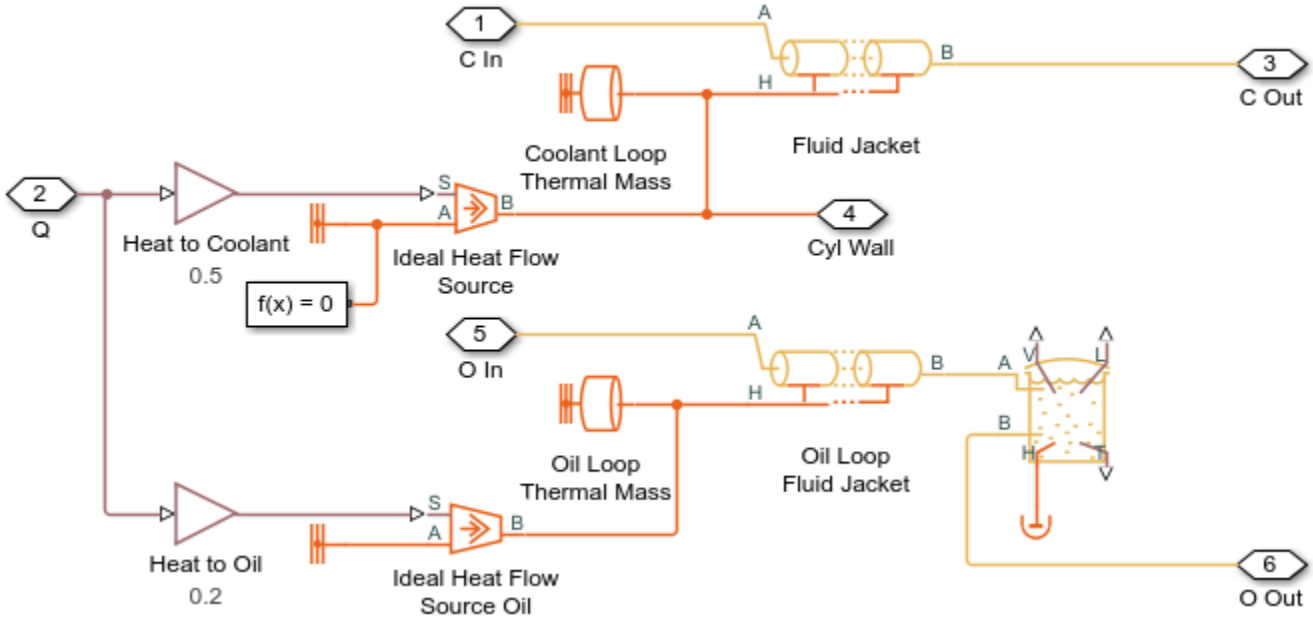
This example shows how to model an engine cooling system with an oil cooling circuit using Simscape™ Fluids™ Thermal Liquid blocks. The system includes a coolant circuit and an oil cooling circuit. A fixed-displacement pump drives coolant through the cooling circuit. The main portion of heat from the engine is absorbed by the coolant and dissipated through the radiator. The system temperature is regulated by the thermostat, which diverts flow to the radiator only when the temperature is above a threshold. The oil cooling circuit also absorbs some of the heat from the engine. The heat added to the oil is transferred to the coolant by the oil-coolant heat exchanger. The Radiator is the E-NTU Heat Exchanger (TL) block with the air-side flow controlled by physical signal inputs. The oil-coolant heat exchanger is the E-NTU Heat Exchanger (TL-TL) block. Both coolant pump and oil pump are driven by the engine speed.

Model

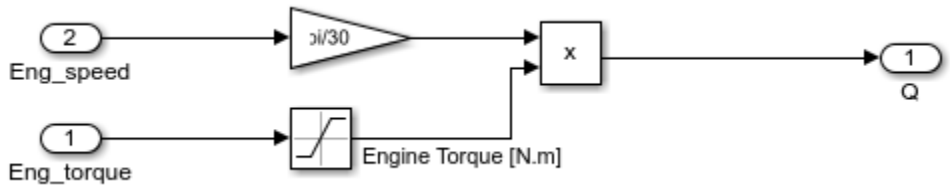


Engine Subsystem

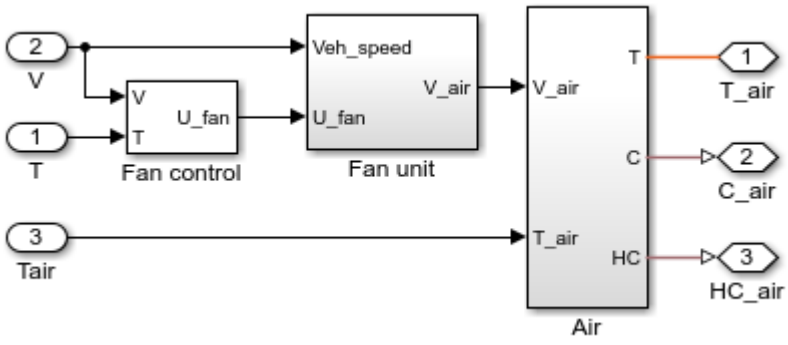
The thermal power generated by the engine is calculated as a function of the instantaneous engine speed and the engine torque. This power is separated in two parts going to the coolant and the oil circuit. It is assumed that 50% of the amount of heat rejected from the engine is added to the coolant and 20% of the heat rejected from the engine is added to the oil.



Heat Flow Rate in Engine Subsystem

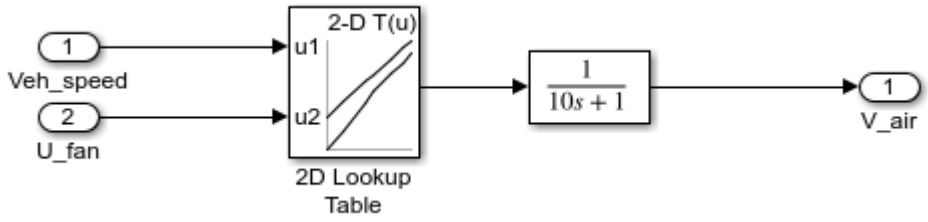


Fan Subsystem



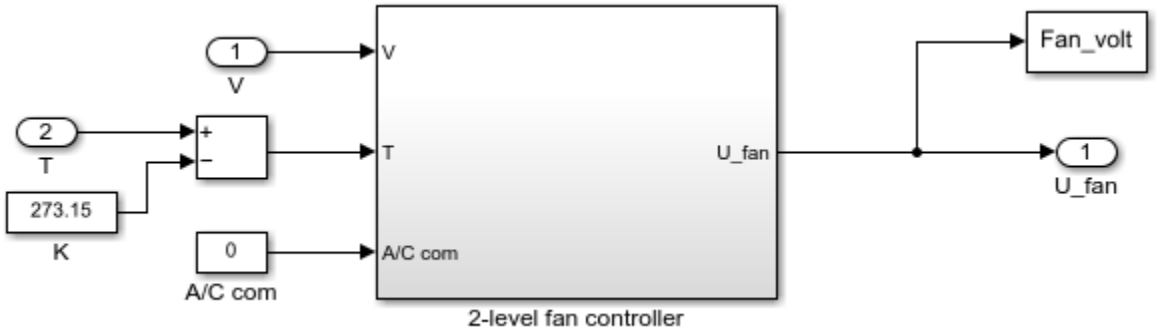
Fan Unit Subsystem

The cooling air speed in the radiator is modeled with a 2D lookup table as a function of instantaneous vehicle speed and the fan controller signal.

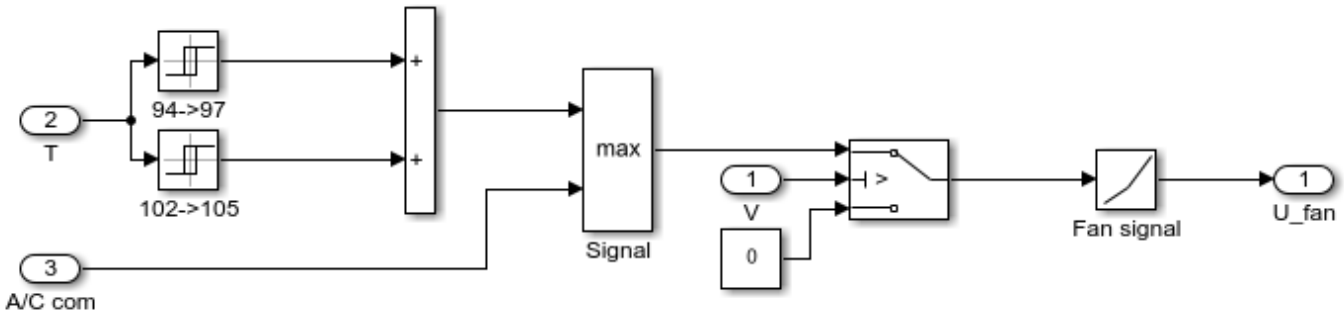


Fan Control Subsystem

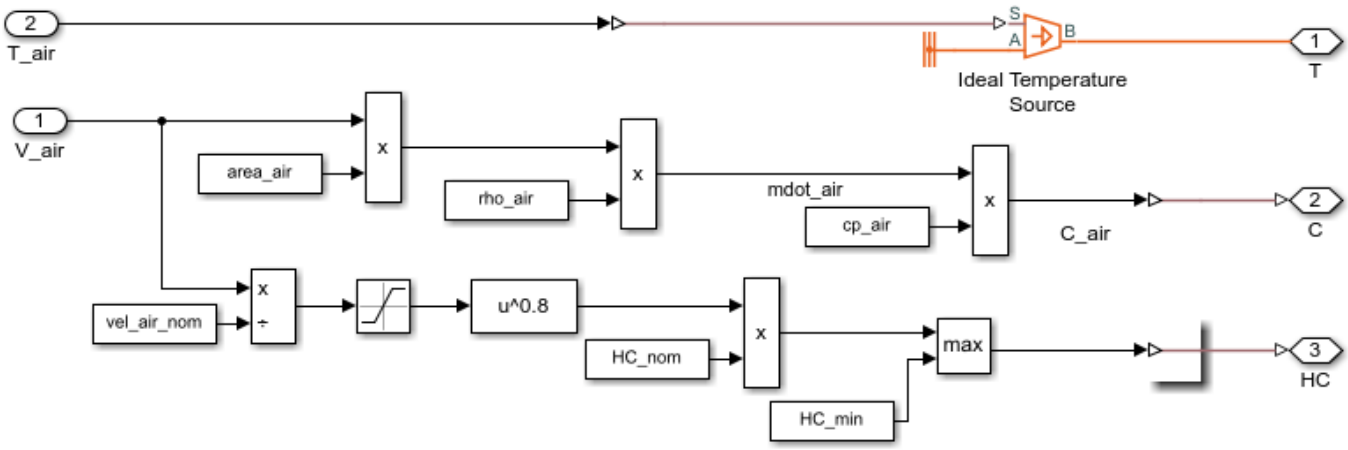
The fan controller unit includes two control levels. The primary level operates at coolant temperatures higher than the primary control target temperature. Once the coolant temperature exceeds a temperature threshold, the secondary level is activated.



2-Level Fan Controller Subsystem

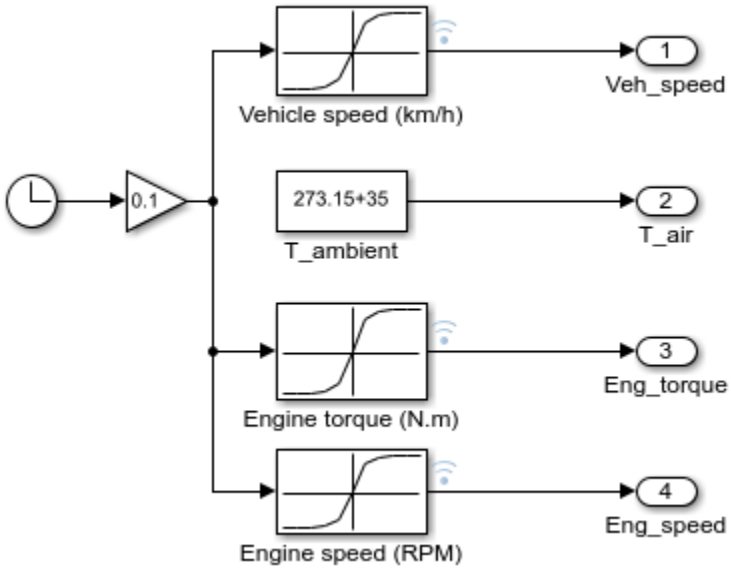


Air Subsystem

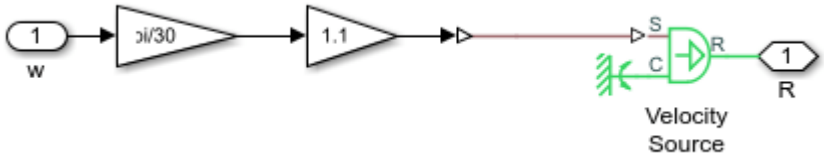


Driving Cycle Subsystem

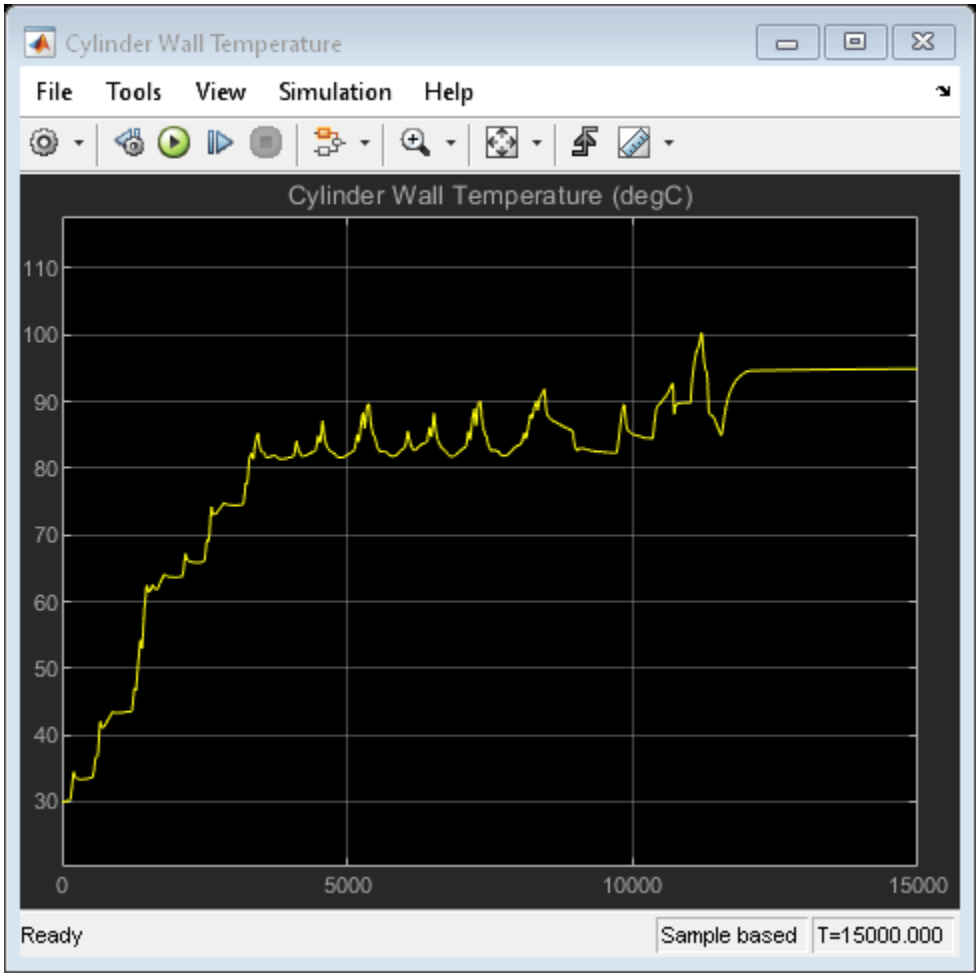
The real vehicle driving cycle is presented based on the instantaneous vehicle speed, the engine speed, and the engine torque inputs.



Shaft Speed Subsystem

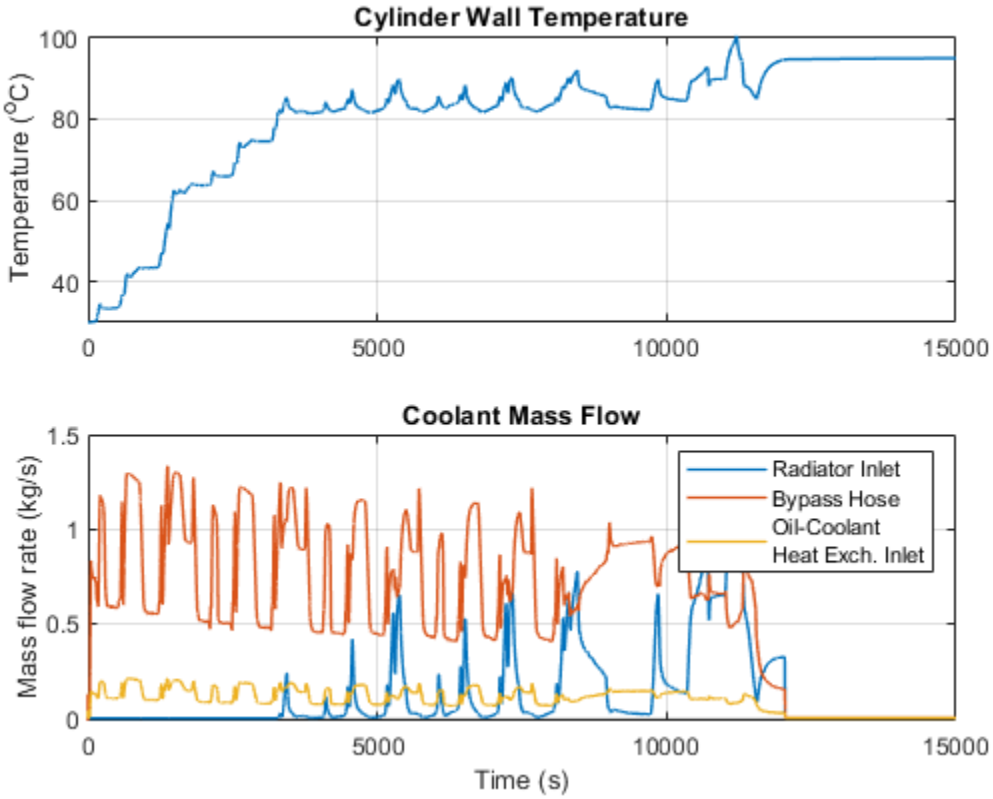


Simulation Results from Scopes

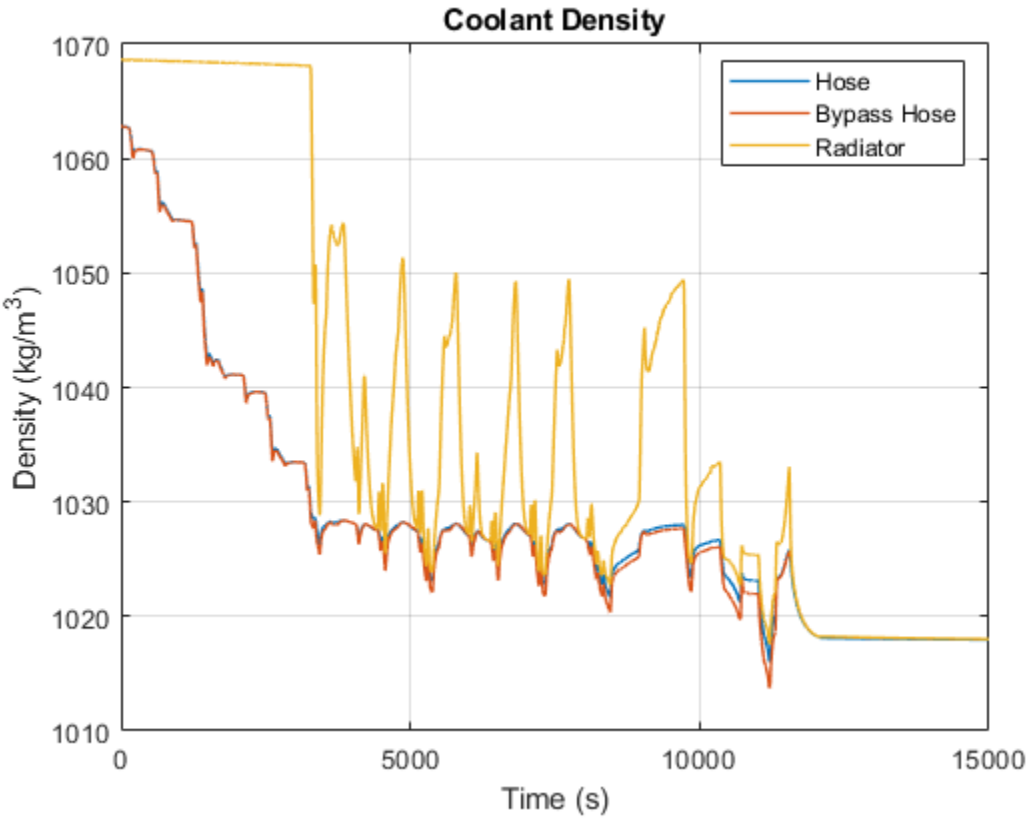


Simulation Results from Simscape Logging

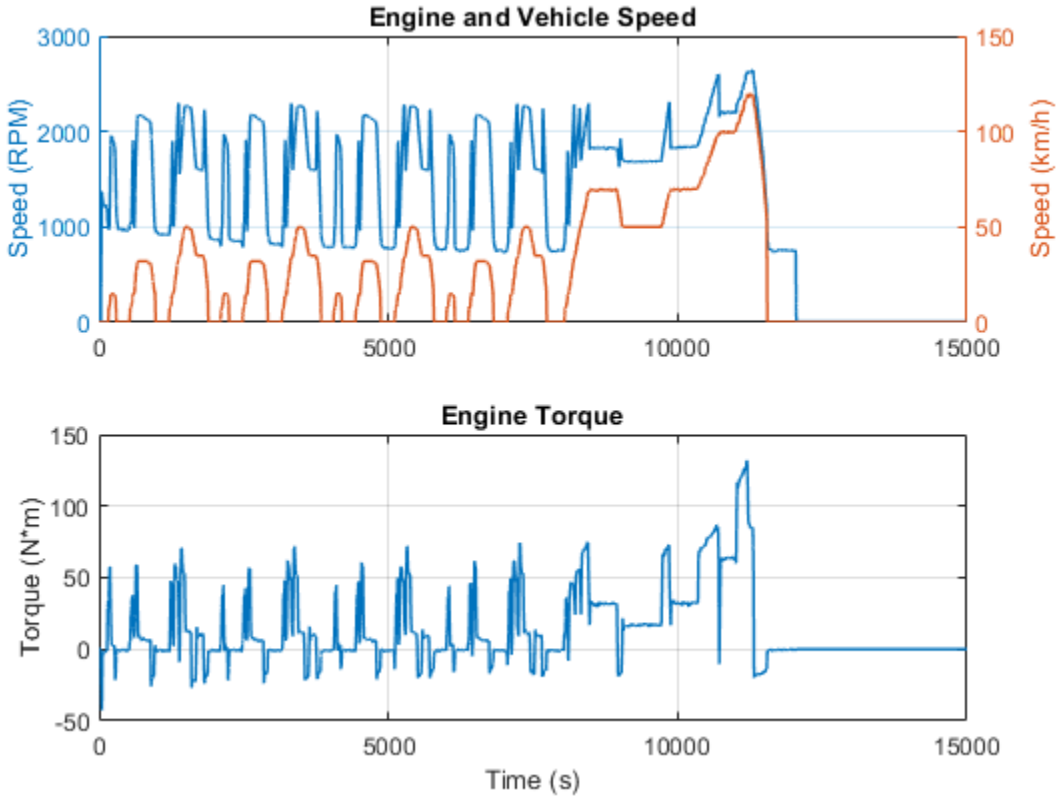
These plots show the effect of opening the thermostat in the engine cooling system. The temperature of the engine block climbs steadily until the thermostat opens. At that point, the flow of coolant through the radiator climbs sharply and the flow of coolant through the bypass hose decreases. Because coolant passing through the radiator releases heat to the atmosphere, the engine block temperature rises more slowly.



This plot shows the density of the coolant at different locations in the cooling system over time. The density of the coolant varies throughout the network based on the local temperature and pressure.



These plots show the instantaneous vehicle speed, engine speed and torque input profiles. The vehicle starts from rest accelerating to near its maximum speed. Subsequently, the vehicle decelerates till it fully stops.

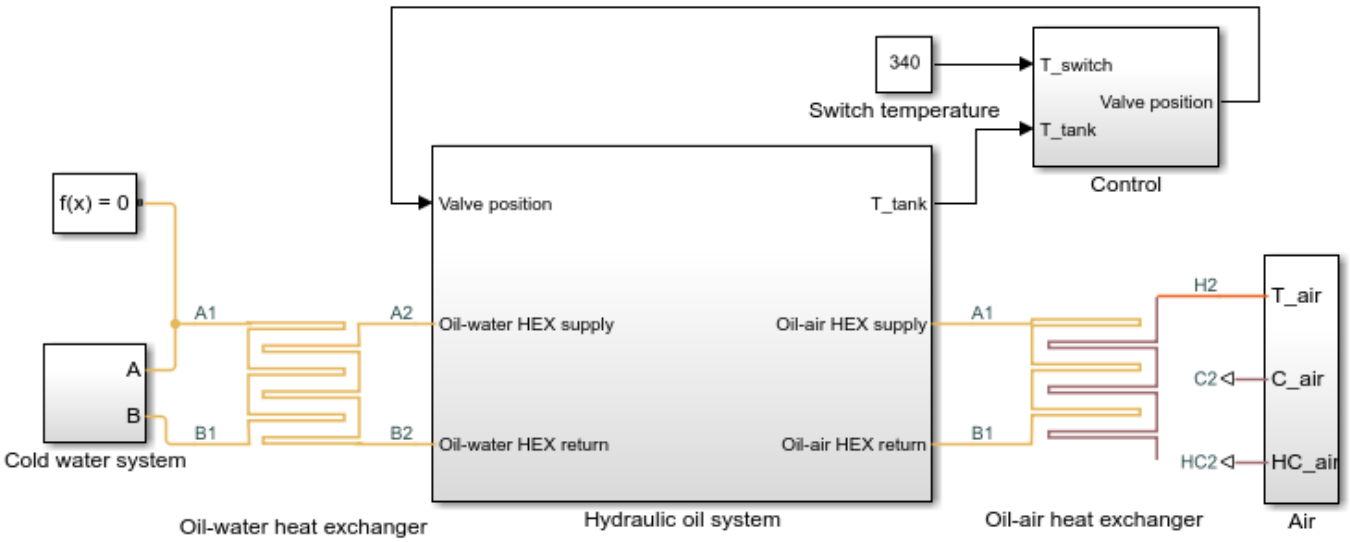

Engine and Vehicle Speed
Speed (RPM)
Speed (km/h)
3000
2000
1000
0
150
100
50
0
0
5000
10000
15000
Engine Torque
Torque (N*m)
150
100
50
0
-50
0
5000
10000
15000
Time (s)

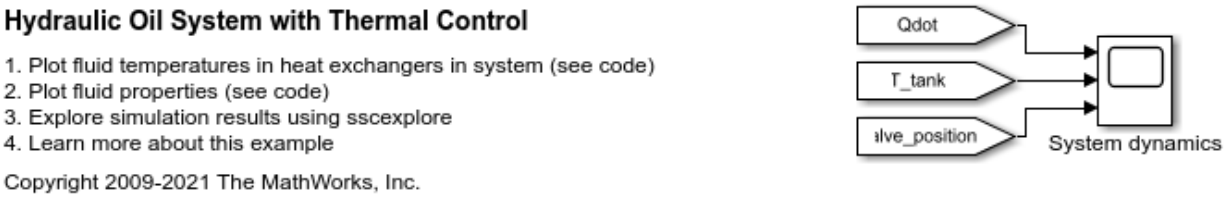
Hydraulic Oil System with Thermal Control

This example shows a hydraulic oil system with a thermal control using Simscape™ Fluids™ Thermal Liquid blocks. The hydraulic oil system consists of an oil storage tank represented by the Tank (TL) block with two inlets, a pump represented by a Mass Flow Rate Source (TL) block, and pipelines represented by Pipe (TL) block.

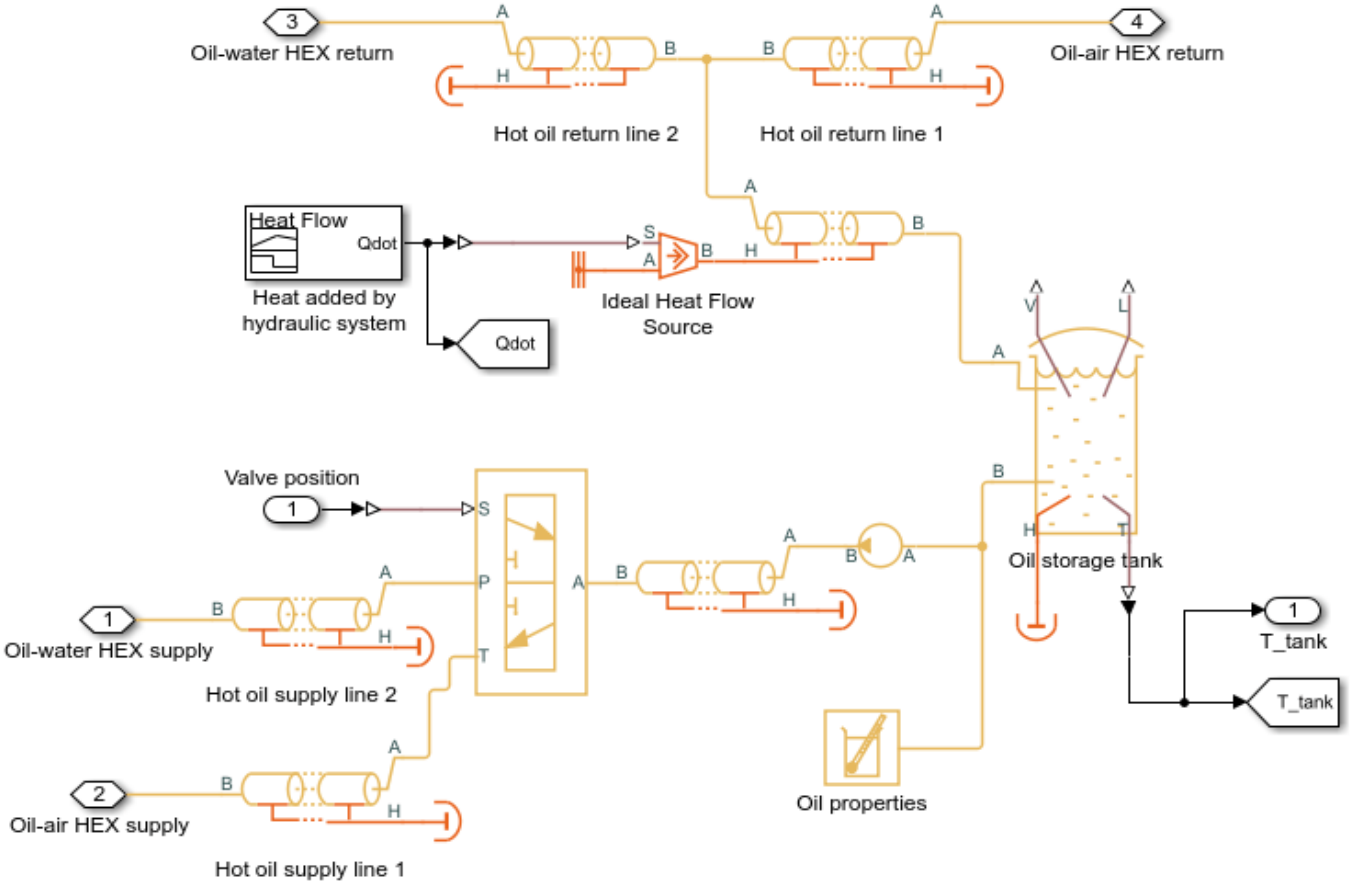
The system is subjected to intensive heating and as the result, the oil temperature in the tank keeps rising. To prevent the oil from overheating, the primary oil-air heat exchanger absorbs heat from the system. In the case when the oil-air heat exchanger cannot maintain the tank temperature below a threshold, the second oil-water heat exchanger is added to the system to further absorb heat. The oil-air heat exchanger is the E-NTU Heat Exchanger (TL) block with the air-side flow controlled by physical signal inputs. The flow of the oil into each heat exchanger is regulated through the 3-Way Directional Valve (TL) block.

Model

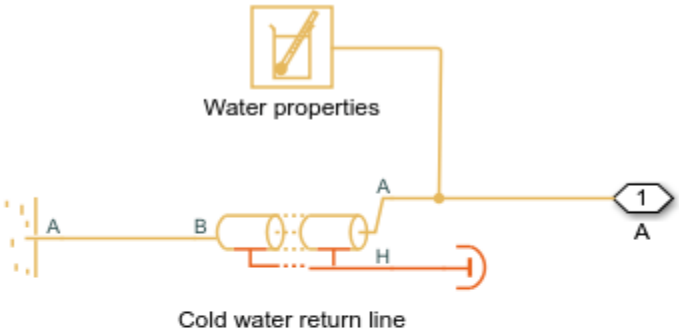


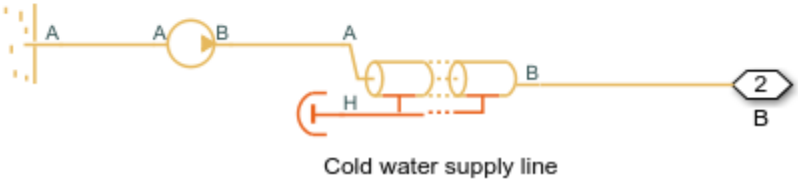


Hydraulic Oil Subsystem

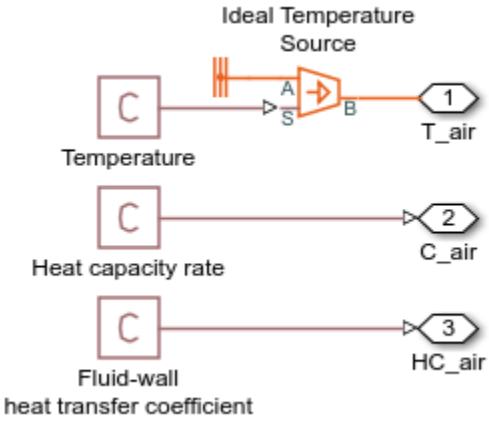


Cold Water Subsystem

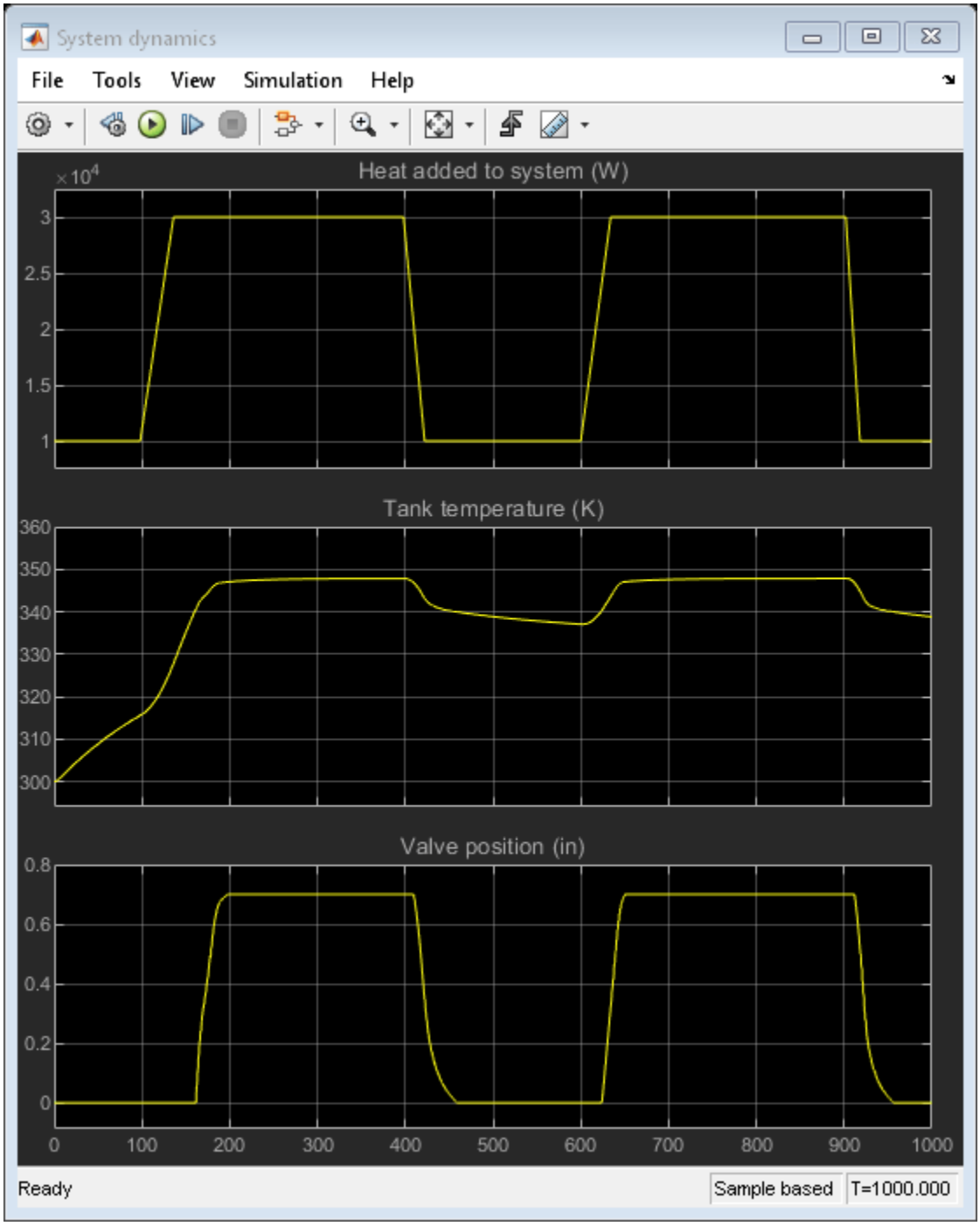




Air Subsystem

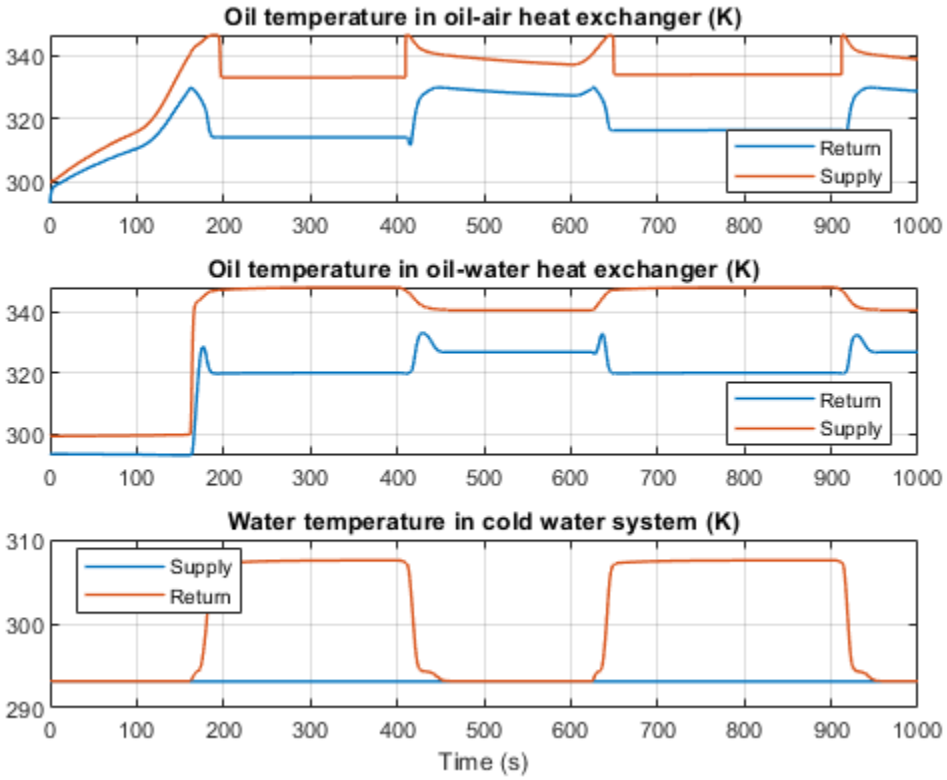


Simulation Results from Scopes



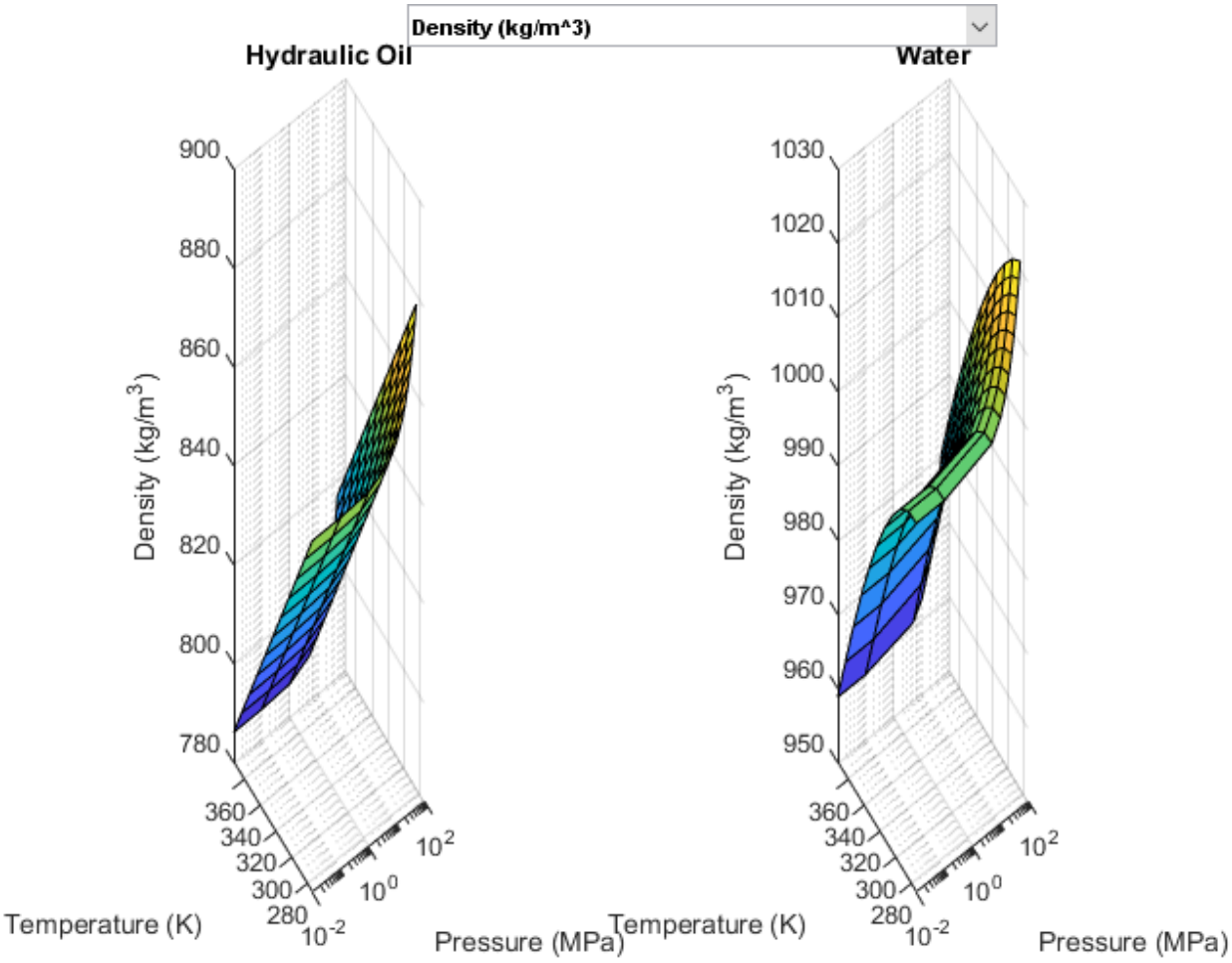
Simulation Results from Simscape Logging

These plots show the temperatures of hydraulic oil and water in the two heat exchangers. The oil-air heat exchanger primarily absorbs heat from the system. In the case when the oil temperature in the tank reaches a threshold, the 3-way directional valve diverts most of the flow to the oil-water heat exchanger to further lower the temperature.



Fluid Properties

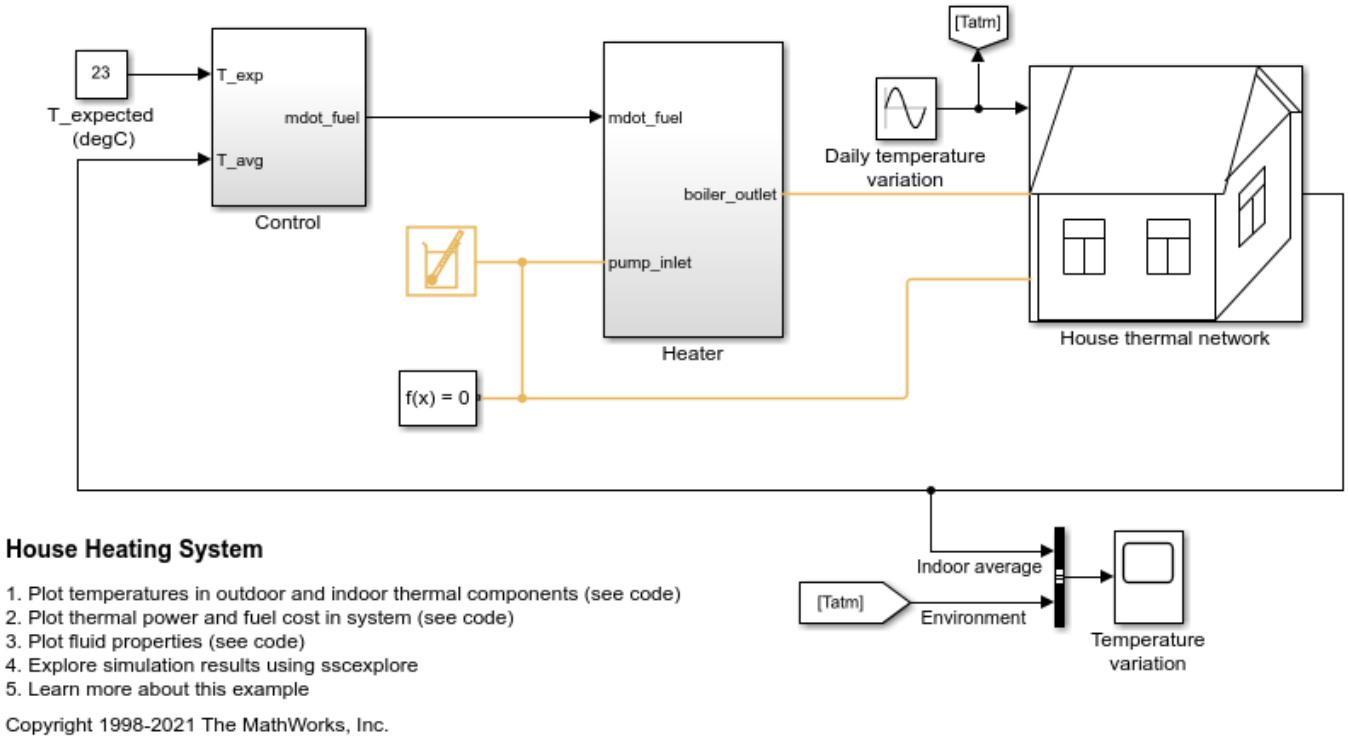
This plots show the fluid properties of hydraulic oil and water as functions of pressure and temperature.


Density (kg/m^3)
Hydraulic Oil
Density (kg/m^3)
900
880
860
840
820
800
780
360
340
320
300
280
Temperature (K)
10^{-2}
10^{0}
10^{2}
Pressure (MPa)
Water
Density (kg/m^3)
1030
1020
1010
1000
990
980
970
960
950
360
340
320
300
280
Temperature (K)
10^{-2}
10^{0}
10^{2}
Pressure (MPa)

House Heating System

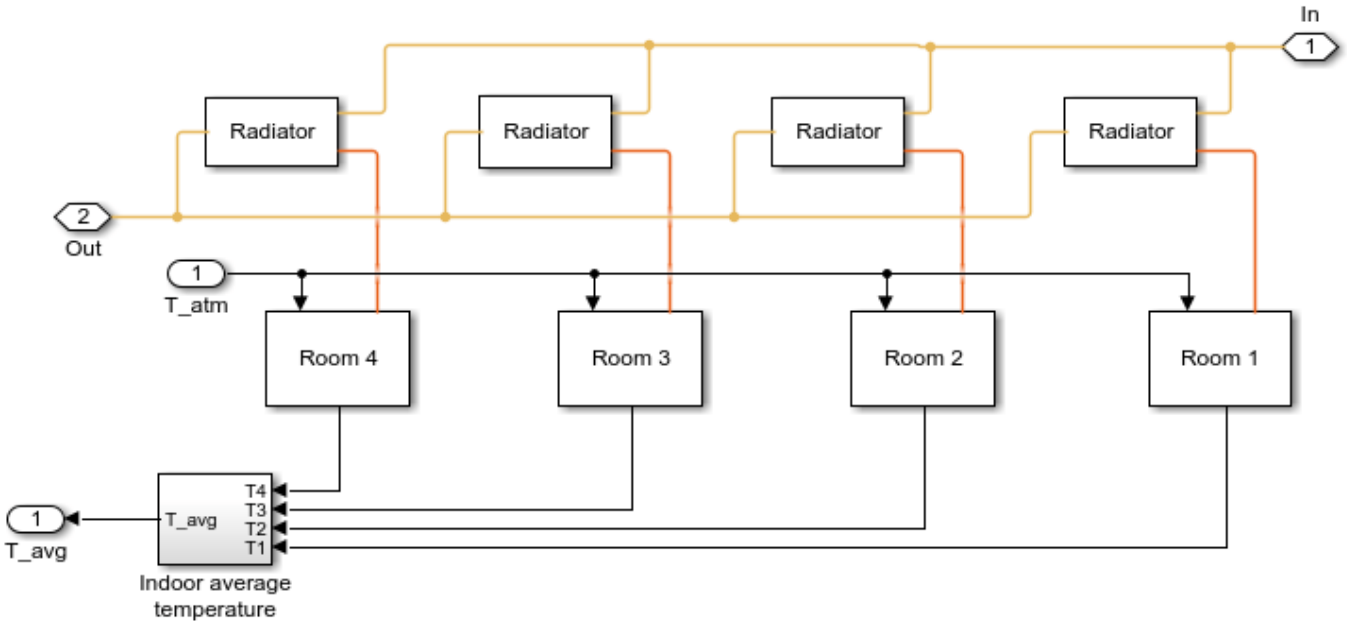
This example shows how to model a simple house heating system. The model contains a heater, a controller, and a house structure with four radiators and four rooms. Each room exchanges heat with the environment through its exterior walls, roof, and windows. Each path is simulated as a combination of a thermal convection, thermal conduction, and the thermal mass. It is assumed that heat is not transferred internally between rooms. The heater consists of a furnace, a boiler, an accumulator, and a pump to circulate hot water in the system. The controller starts admitting fuel into the furnace if the overall average temperature of rooms falls below 21 degree C and it stops if the temperature exceeds 25 degree C. The simulation calculates the heating cost and indoor temperatures.

Model

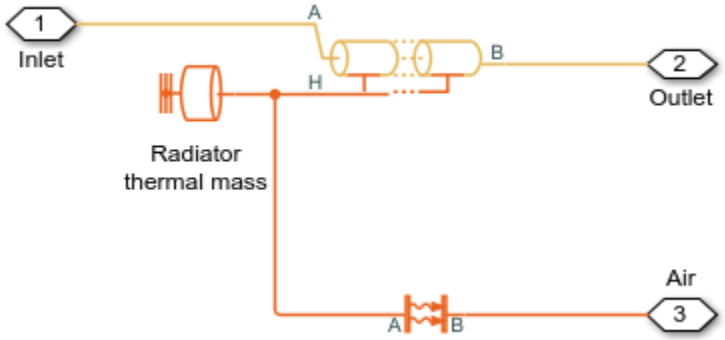


House Thermal Network

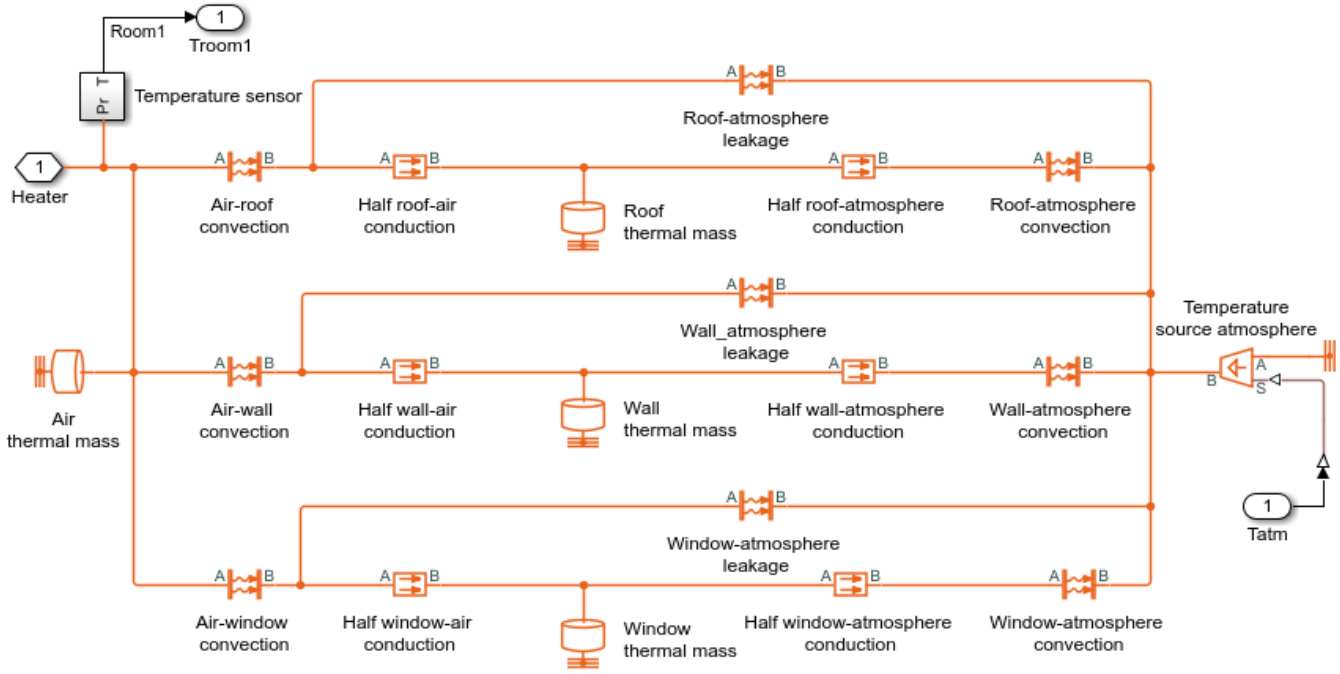
It consists of four radiators and four rooms.



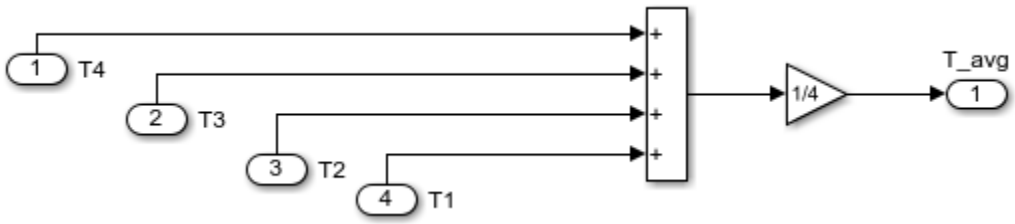
Radiator Subsystem



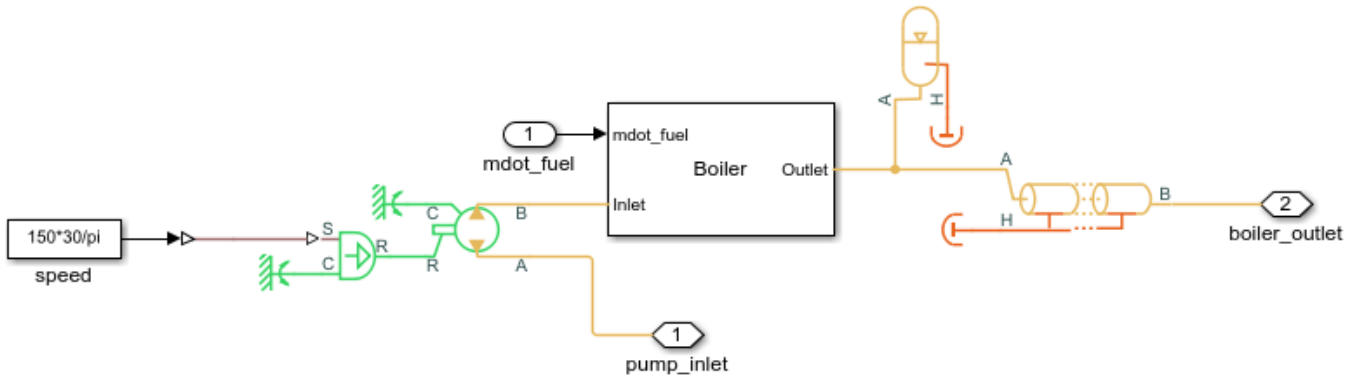
Room Subsystem



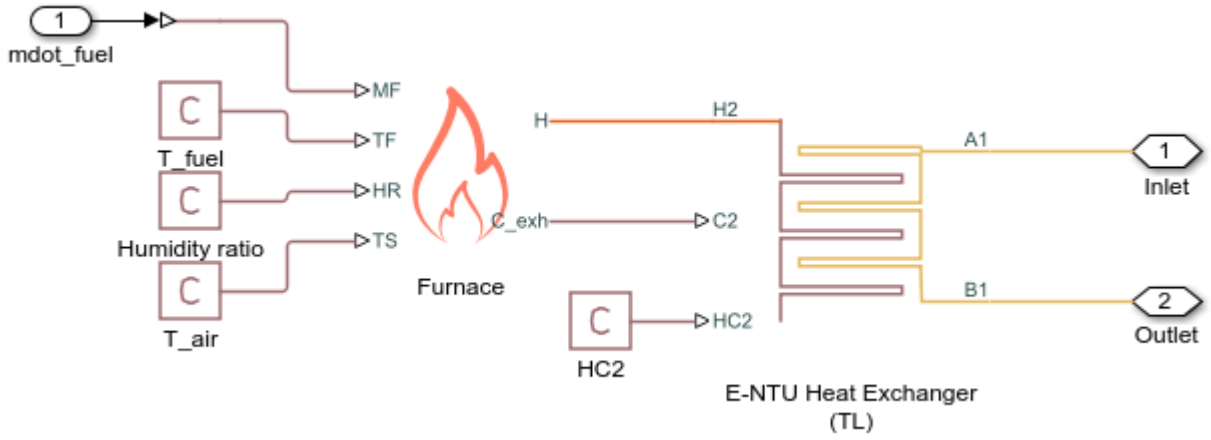
Indoor Average Temperature Subsystem



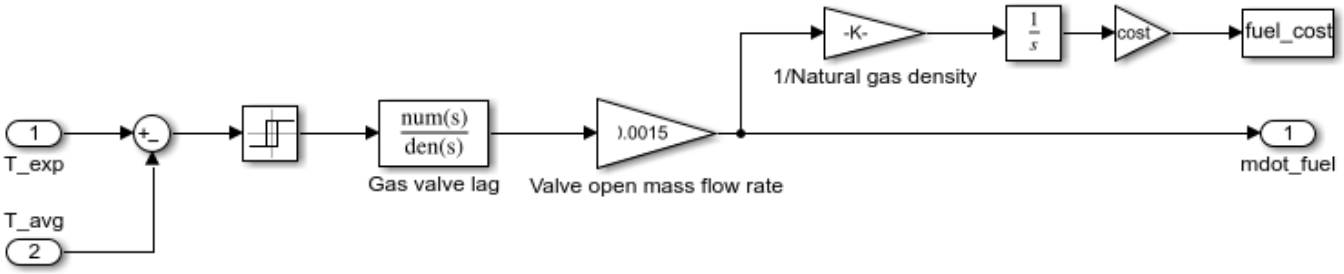
Heater Subsystem



Boiler Subsystem

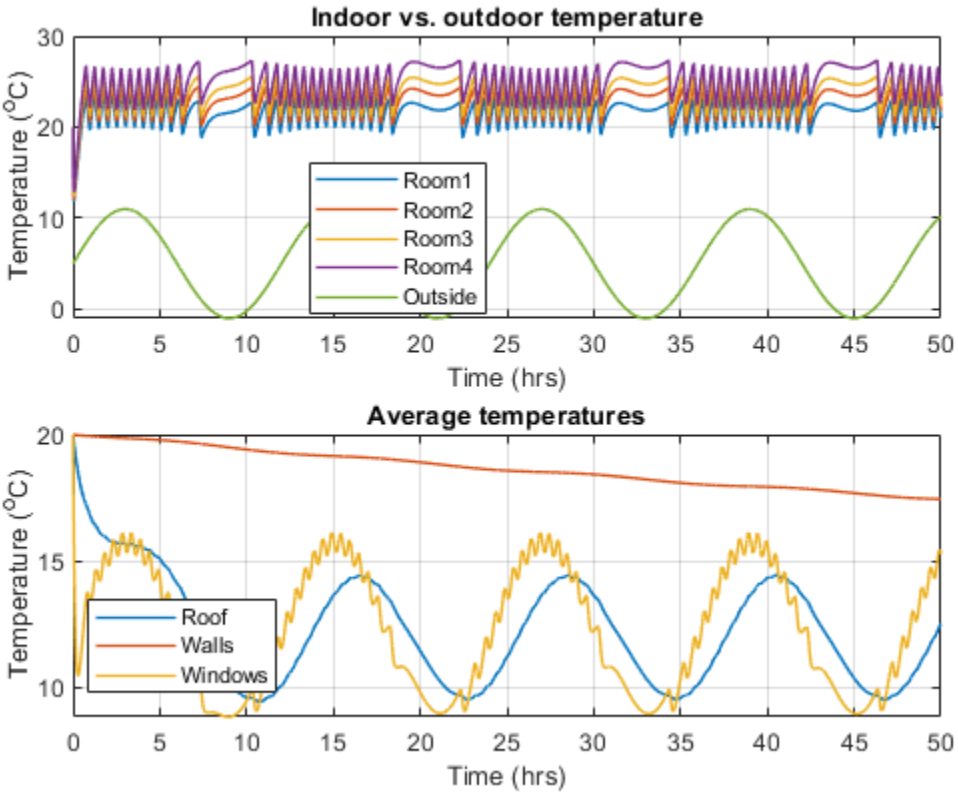


Control Subsystem

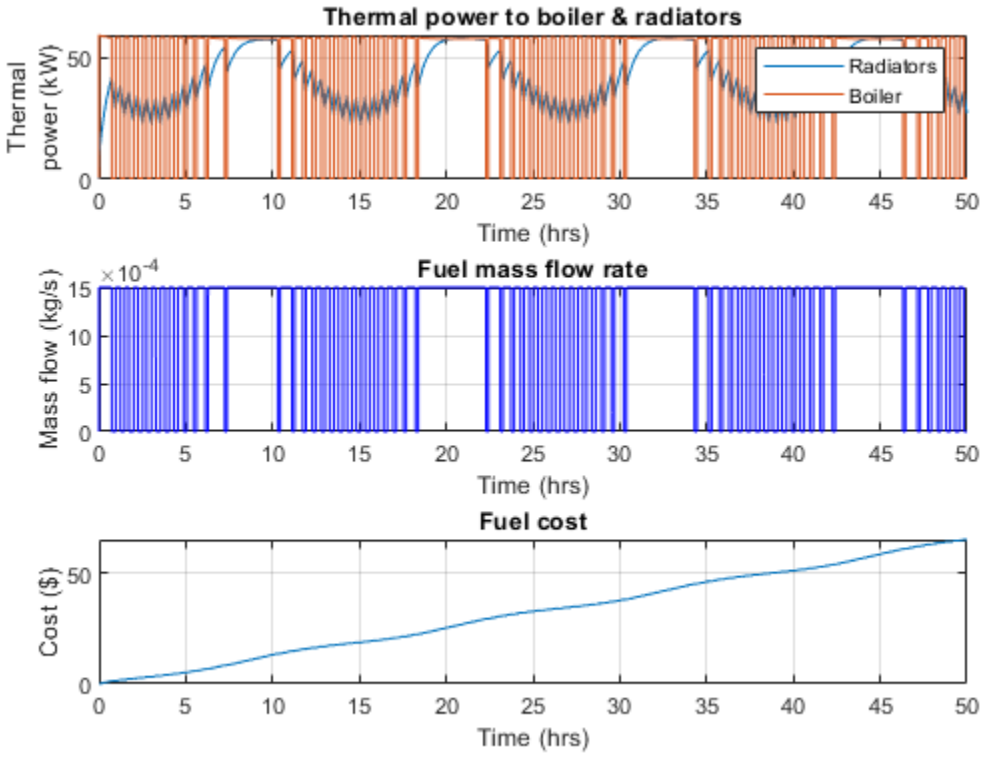


Simulation Results from Simscape Logging

The first plot shows the temperature variation of each room compared to the outside temperature fluctuations. The second plot shows the average overall temperatures of roof, walls, and windows.

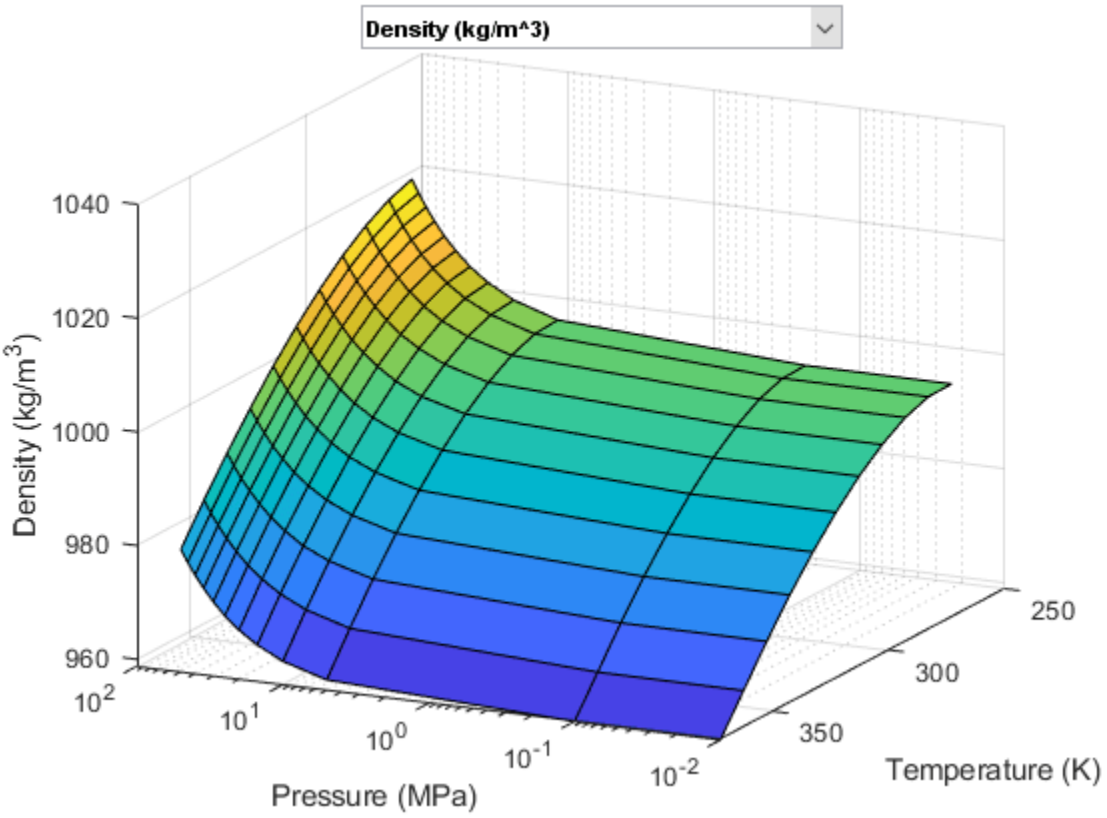


These plots show heat flow rate from furnace to the boiler and to radiators. The cost of fuel is calculated based on the fuel mass flow rate through the furnace. It is assumed that the fuel has the density of natural gas (~ 0.9 kg/m^3) and the cost of natural gas is $8.50 per thousand cubic feet (Mcf): $8.50 per 1000 cf.



Fluid Properties

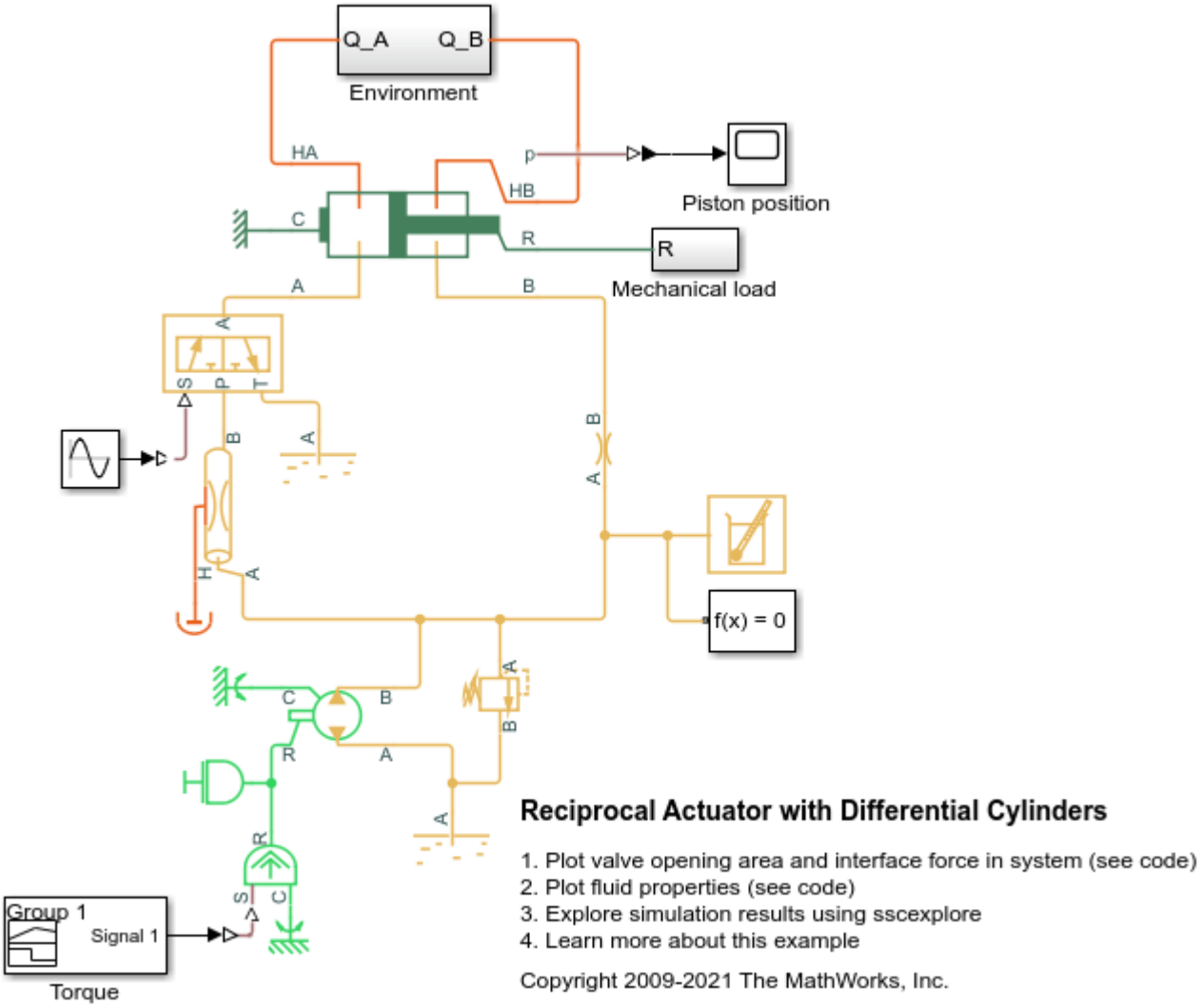
This figure plots the fluid properties of water as functions of pressure and temperature.


Density (kg/m^3)
1040
1020
1000
980
960
Density (kg/m^3)
10^2
10^1
10^0
10^-1
10^-2
Pressure (MPa)
250
300
350
Temperature (K)

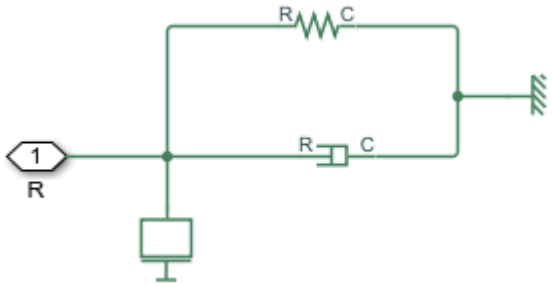
Reciprocal Actuator with Differential Cylinders

This example shows a double-acting actuator with differential cylinders. The pump output is connected to cylinder B of the actuator while cylinder A of the actuator can be connected to either the pump or the reservoir through the 3-way directional valve. When cylinder A is connected to the pump, pressures at both cylinders become equal. Because of the larger effective piston area in cylinder A, the interface force in cylinder A is larger than that of in cylinder B which causes the piston to extend. When cylinder A is connected to the reservoir, the piston starts to retract. The 3-way directional valve is controlled by a sinusoidal signal to achieve repeating reciprocal motion in the actuator.

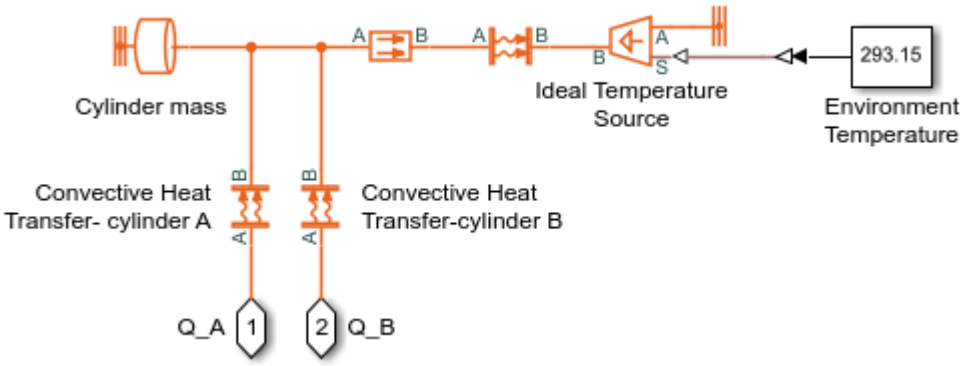
Model



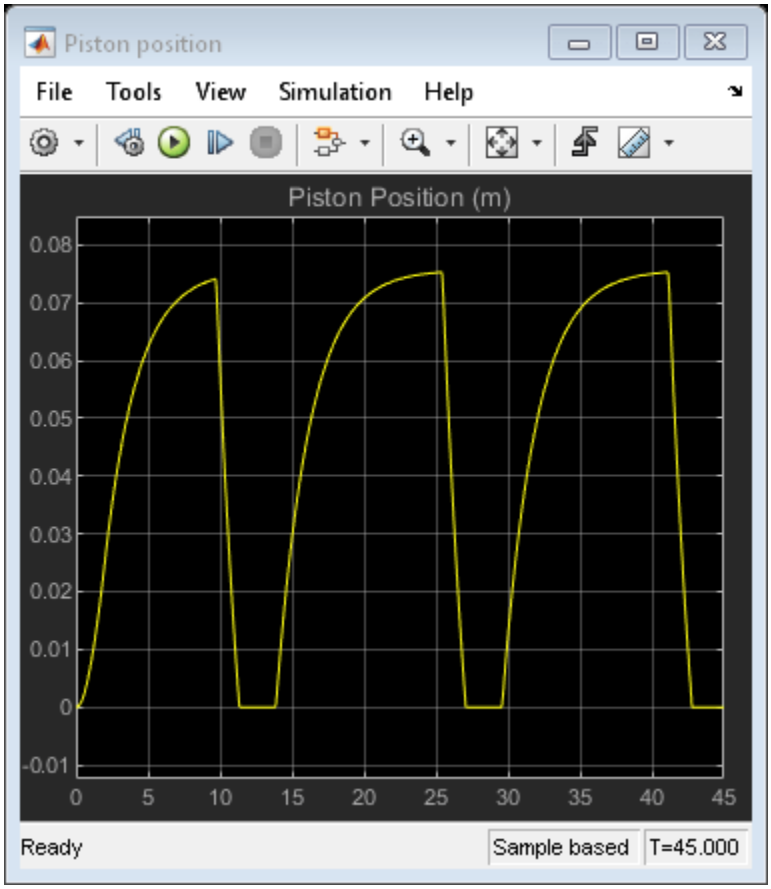
Mechanical Load Subsystem



Environment Subsystem

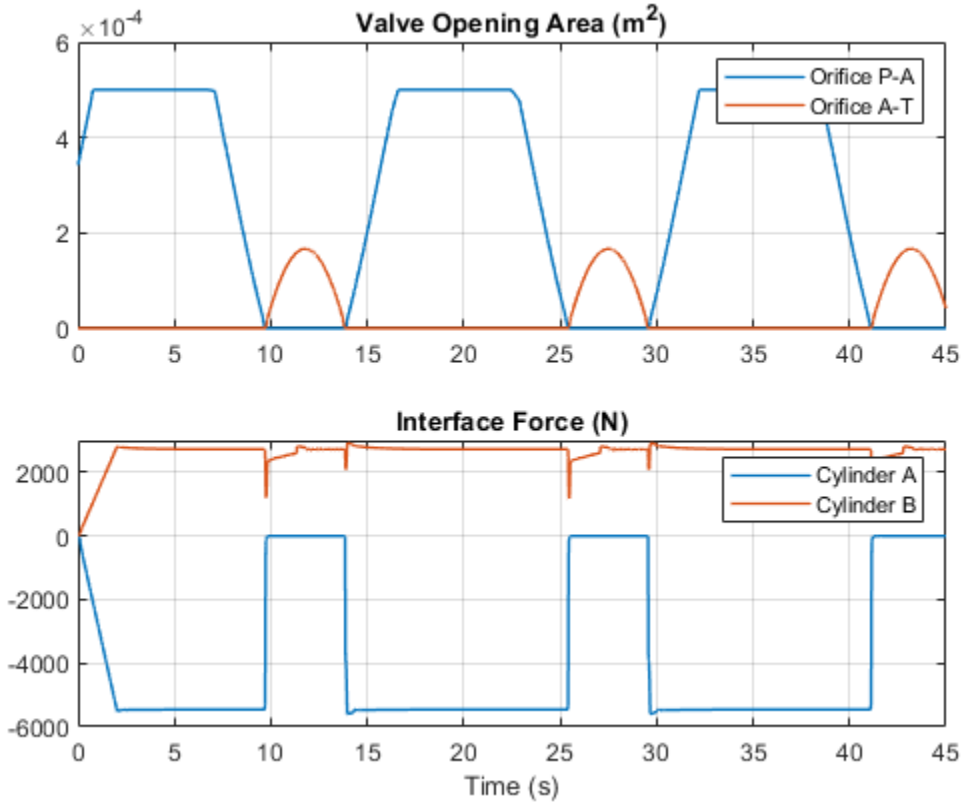


Simulation Results from Scopes

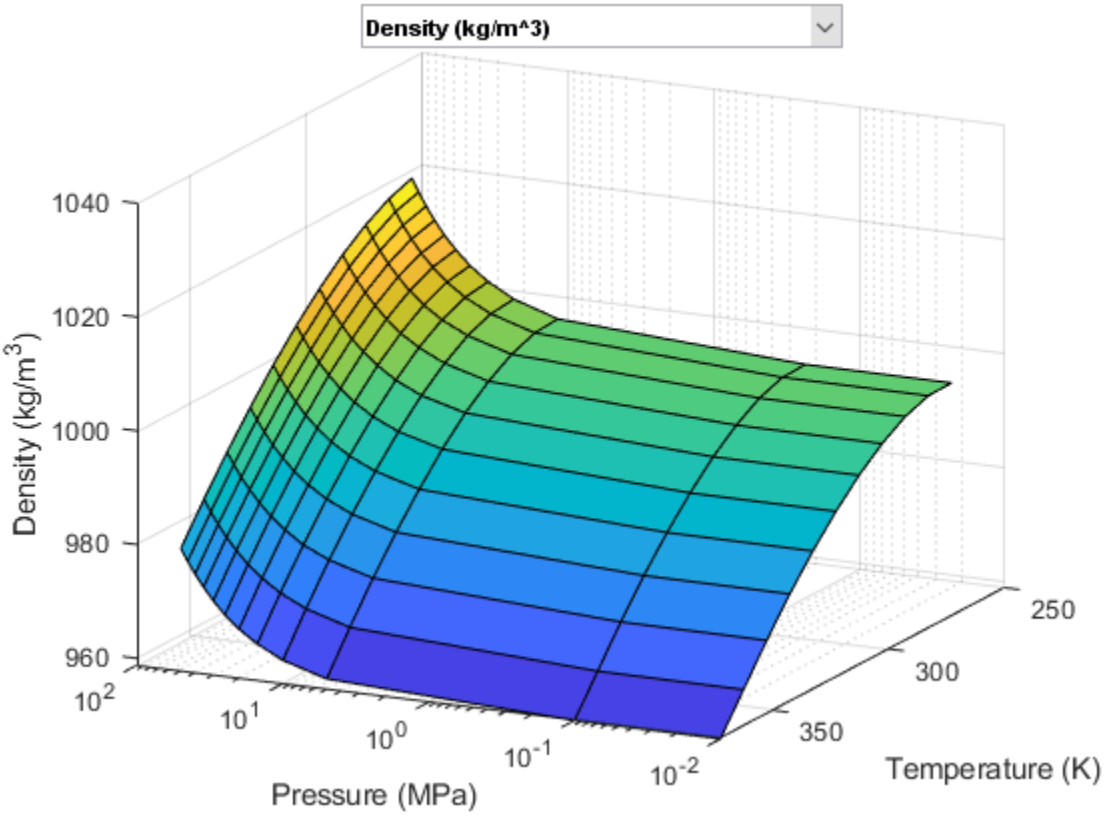


Simulation Results from Simscape Logging

These plots show the 3-way valve opening area and the interface forces generated on each side of the actuator during the reciprocal motion.


Valve Opening Area (m^2)
×10^-4
6
4
2
0
0
5
10
15
20
25
30
35
40
45
Orifice P-A
Orifice A-T
Interface Force (N)
2000
0
-2000
-4000
-6000
Cylinder A
Cylinder B
Time (s)

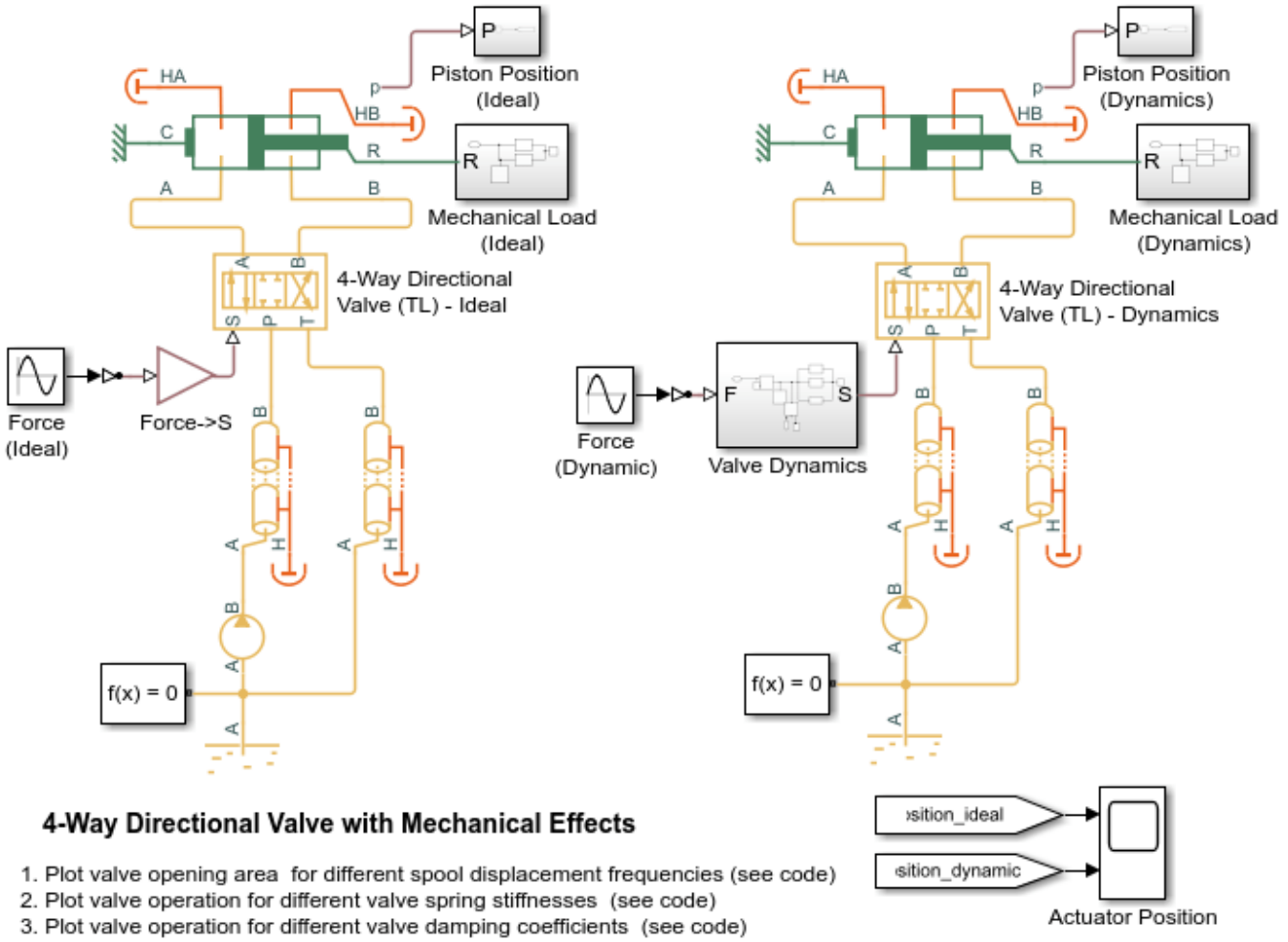
Fluid Properties



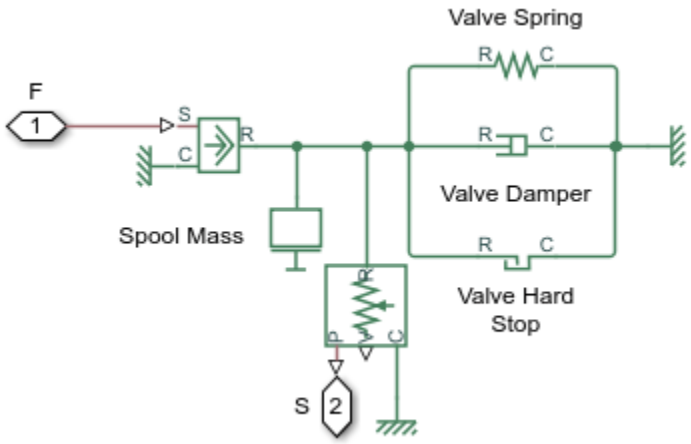
4-Way Directional Valve with Mechanical Effects

This example compares two 4-way directional valves, one with mechanical components to account for spool mass, valve spring and damper and one without (ideal valve). The input force to both valves is a sine wave. The operation of two valves are compared for different displacement frequencies, valve spring stiffness and damper coefficients.

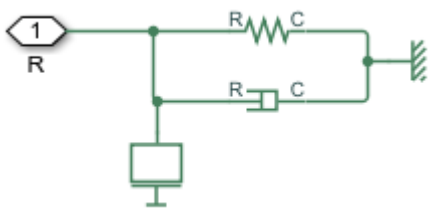
Model



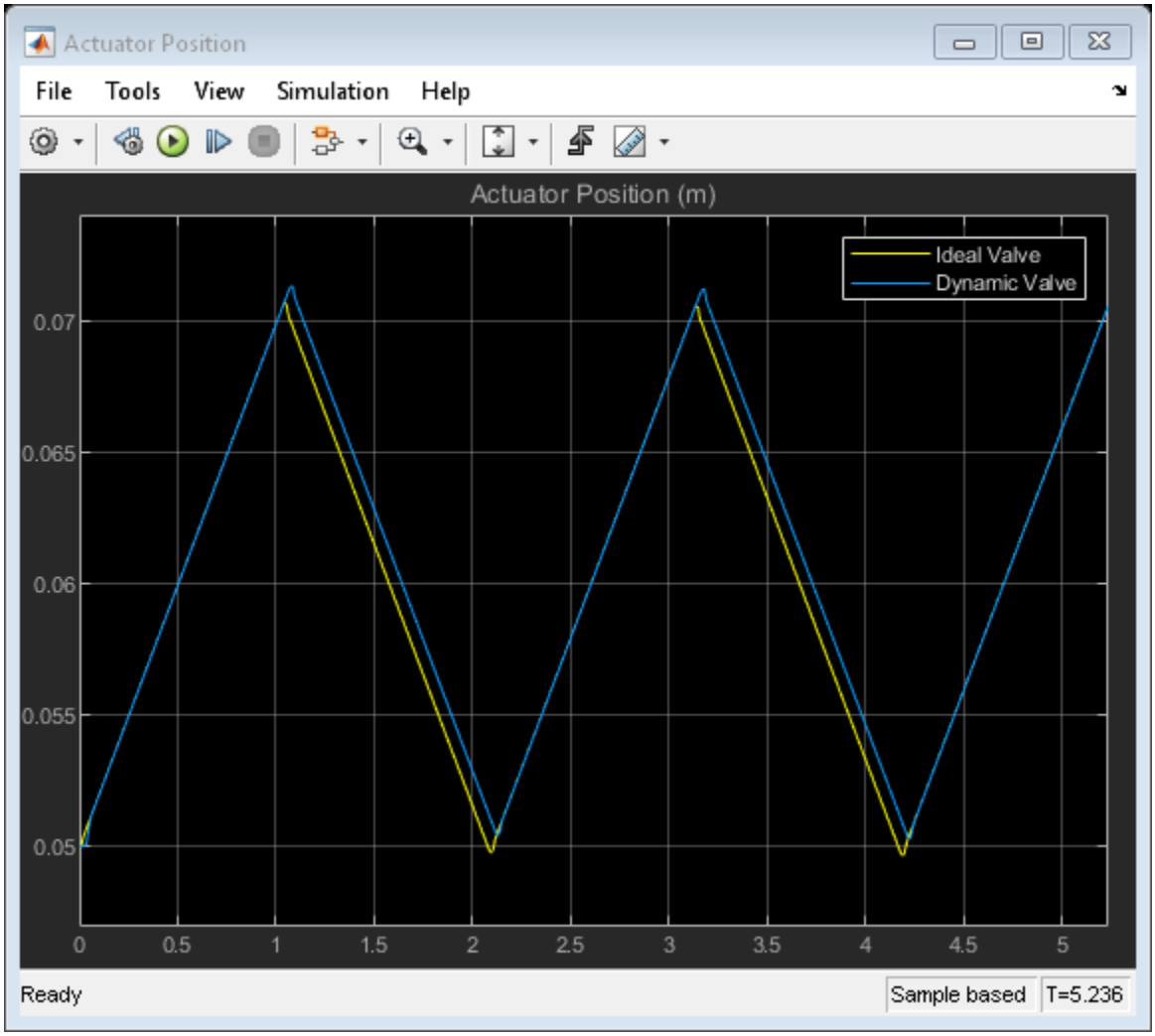
Valve Dynamics Subsystem



Mechanical Load Subsystem

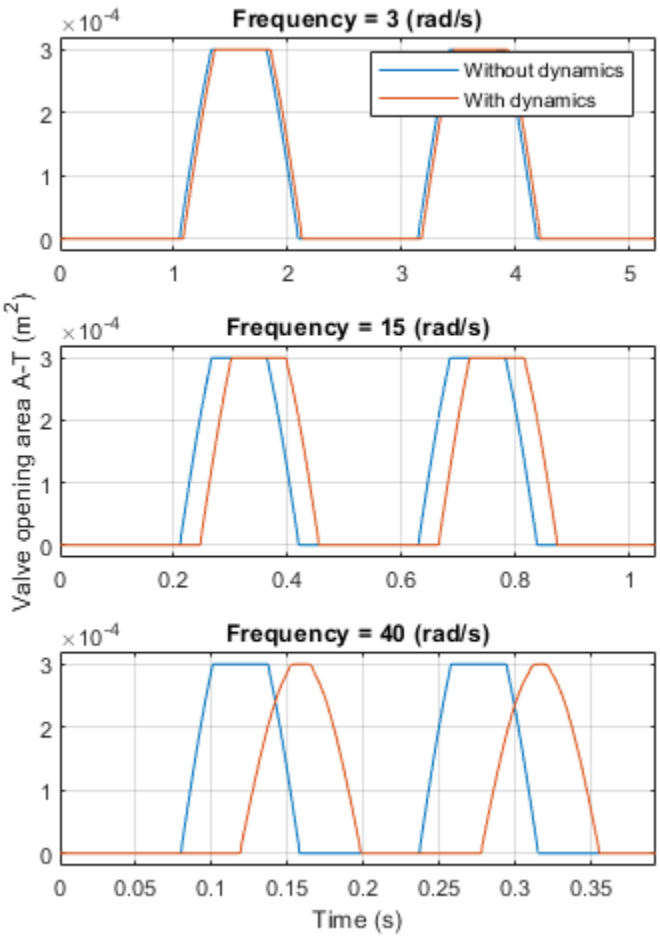


Simulation Results from Scopes

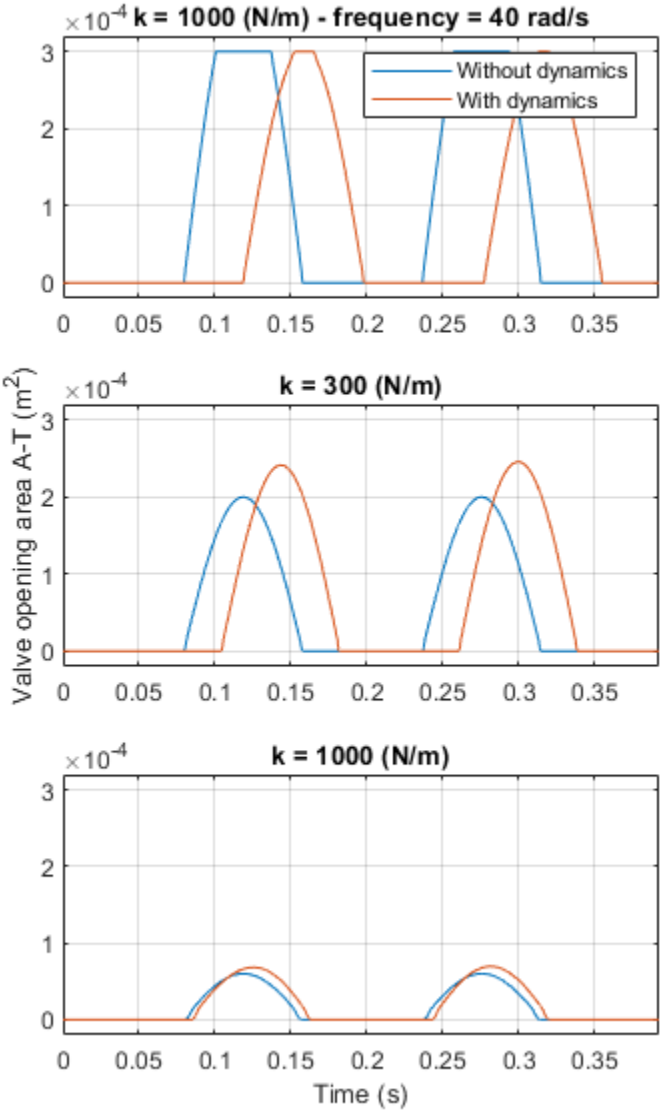


Simulation Results from Simscape Logging

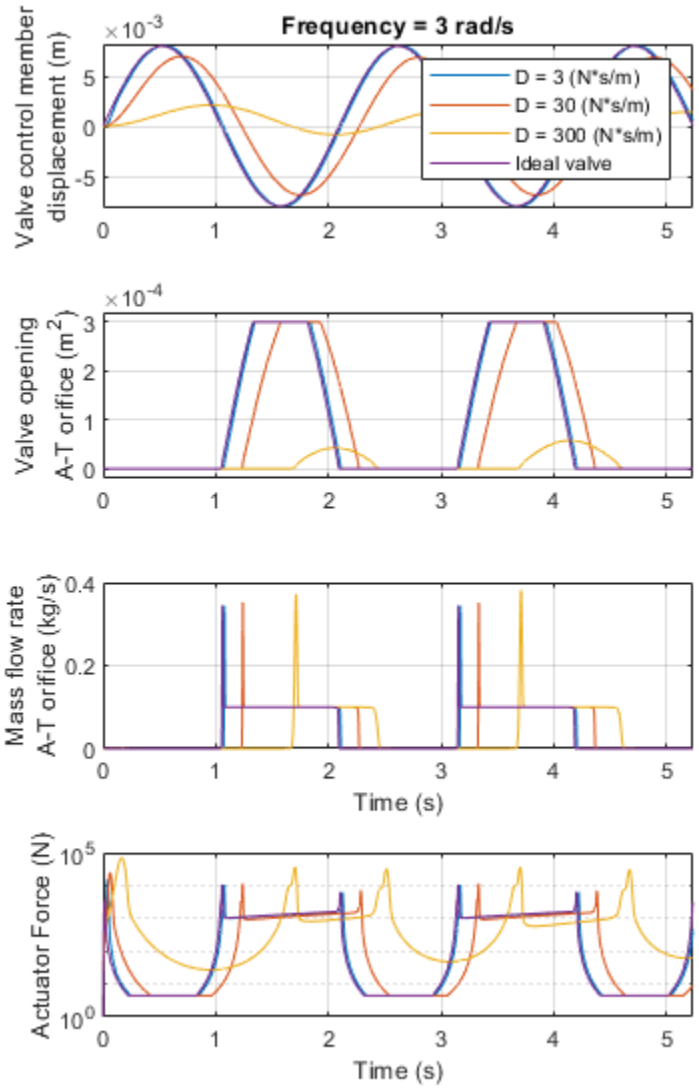
The plots below show the valve opening area (orifice A-T) for ideal valve and valve with dynamics for different input force frequencies.



The plots below show the valve opening area (orifice A-T) for ideal valve and valve with dynamics for different spring stiffnesses.



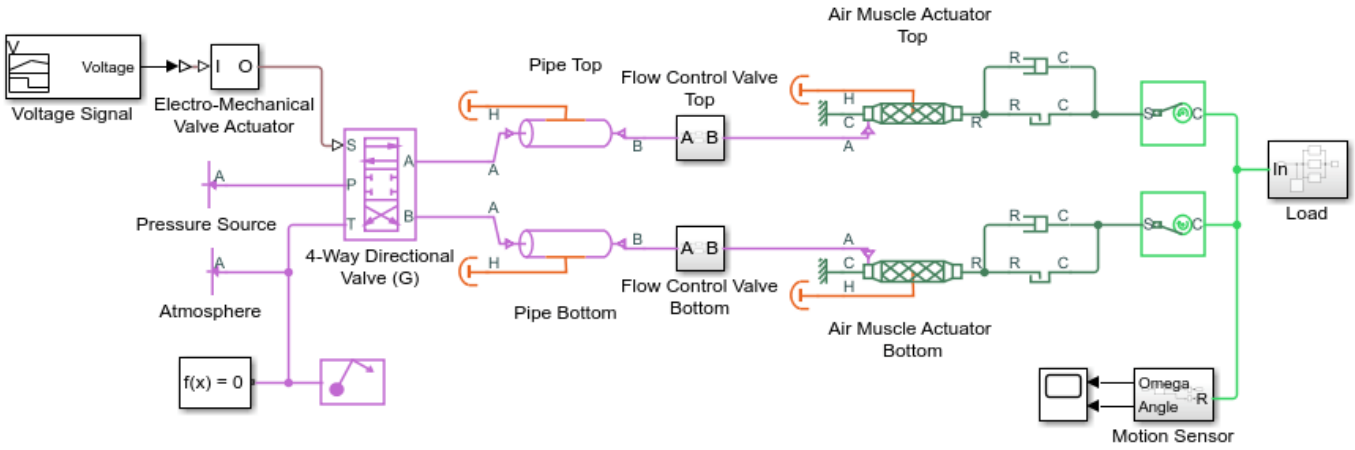
The plots below show the valve opening area (orifice A-T), control member displacement, and mass flow rate for ideal valve and valve with dynamics for different valve damping coefficients.


Frequency = 3 rad/s
Valve control member displacement (m)
×10^-3
D = 3 (N*s/m)
D = 30 (N*s/m)
D = 300 (N*s/m)
Ideal valve
Valve opening A-T orifice (m^2)
×10^-4
Mass flow rate A-T orifice (kg/s)
Time (s)
Actuator Force (N)
Time (s)

Antagonistic McKibben Muscle Actuator

This demo shows a muscle actuation based on two air muscle actuators (or McKibben artificial muscles) in antagonistic connection. The air muscle actuators are connected to the opposite sides of a lever. The 4-way directional valve is controlled by an electro-mechanical valve actuator. In the 4-way directional way when the high-pressure path P-A and return line B-T are open, the top air muscle actuator contracts and forces the bottom air muscle actuator on the opposite side to extend. Similarly, as the high-pressure path P-B and return line A-T open, the bottom air muscle actuator starts to contract and forces the top air muscle actuator to extend. The oscillating motions of the muscles are converted into the angular rotation of the output load connected to the mechanical linkage modeled with the slider-cranks.

Model

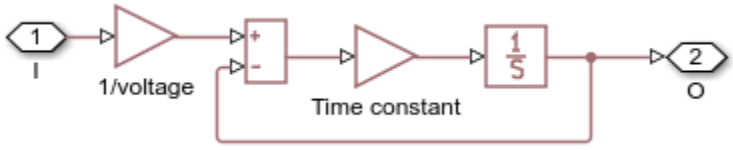


Antagonistic McKibben Muscle Actuator

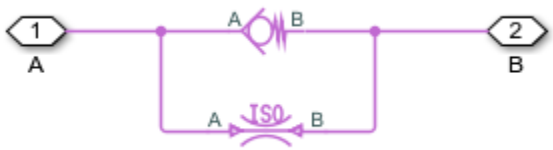
1. Plot pressures and mass flow rates in 4-way directional valve (G) (see code)
2. Plot actuators lengths and pressures in air muscle actuators (see code)
3. Explore simulation results using sscexplore
4. Learn more about this example

Copyright 2018-2021 The MathWorks, Inc.

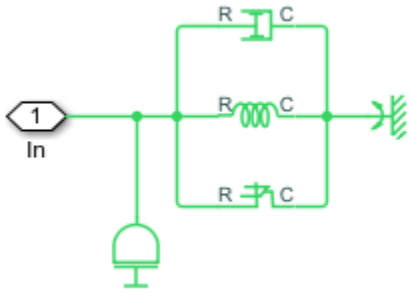
Electro-Mechanical Valve Actuator Subsystem



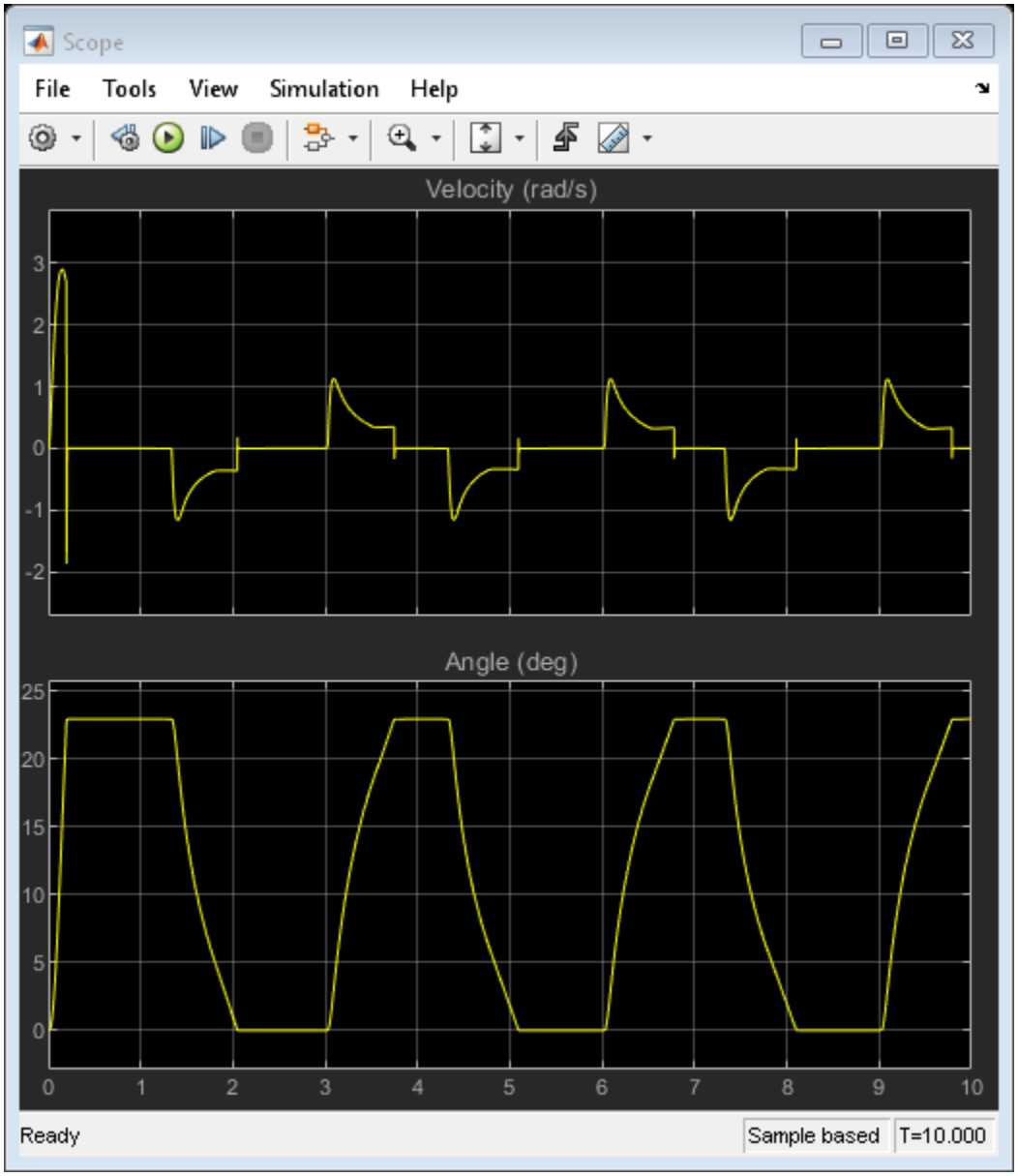
Flow Control Valve Subsystems



Load Subsystem

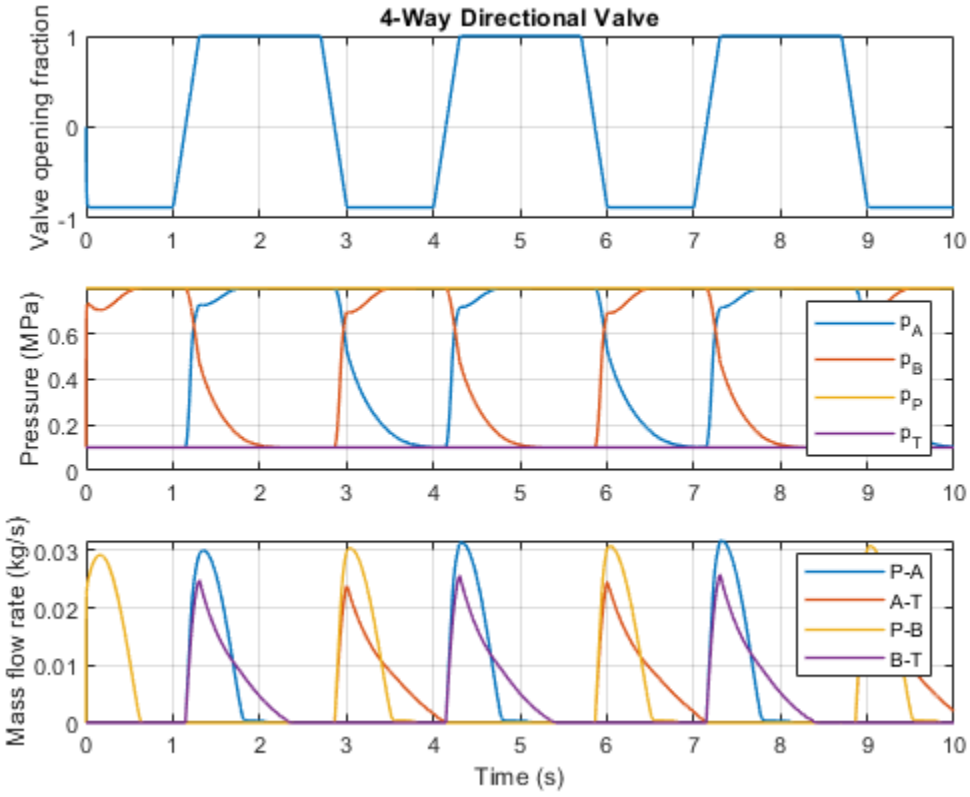


Simulation Results from Scopes

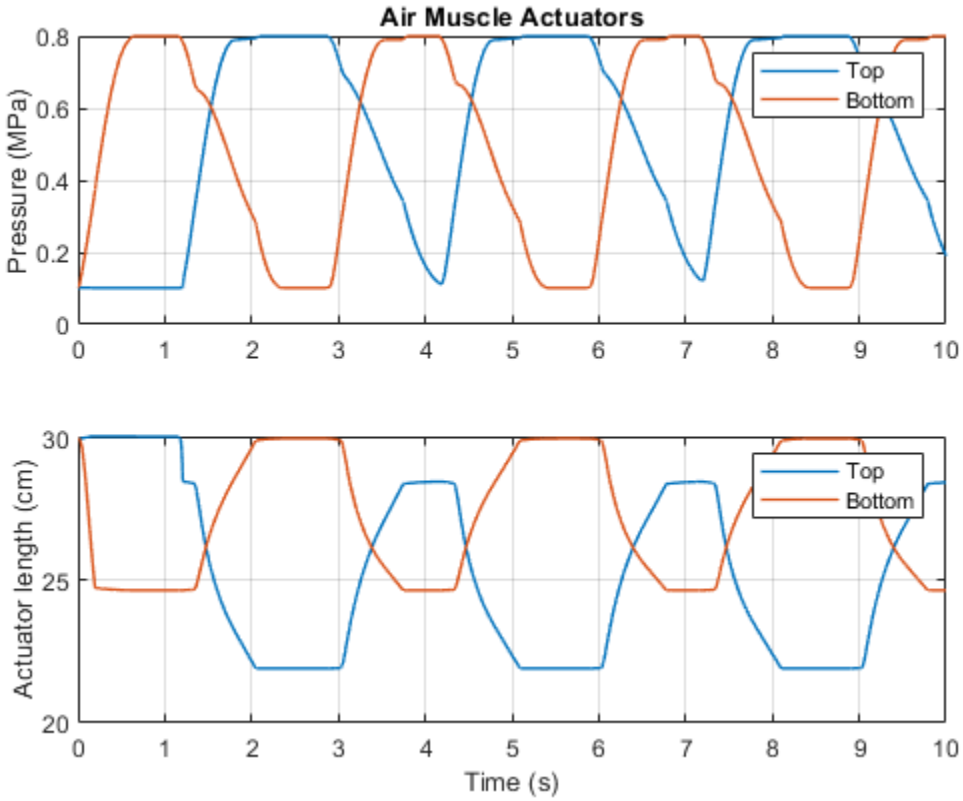


Simulation Results from Simscape Logging

The plots below show the pressures across and mass flow rates through the 4-Way Directional Valve (G) for a given voltage signal.



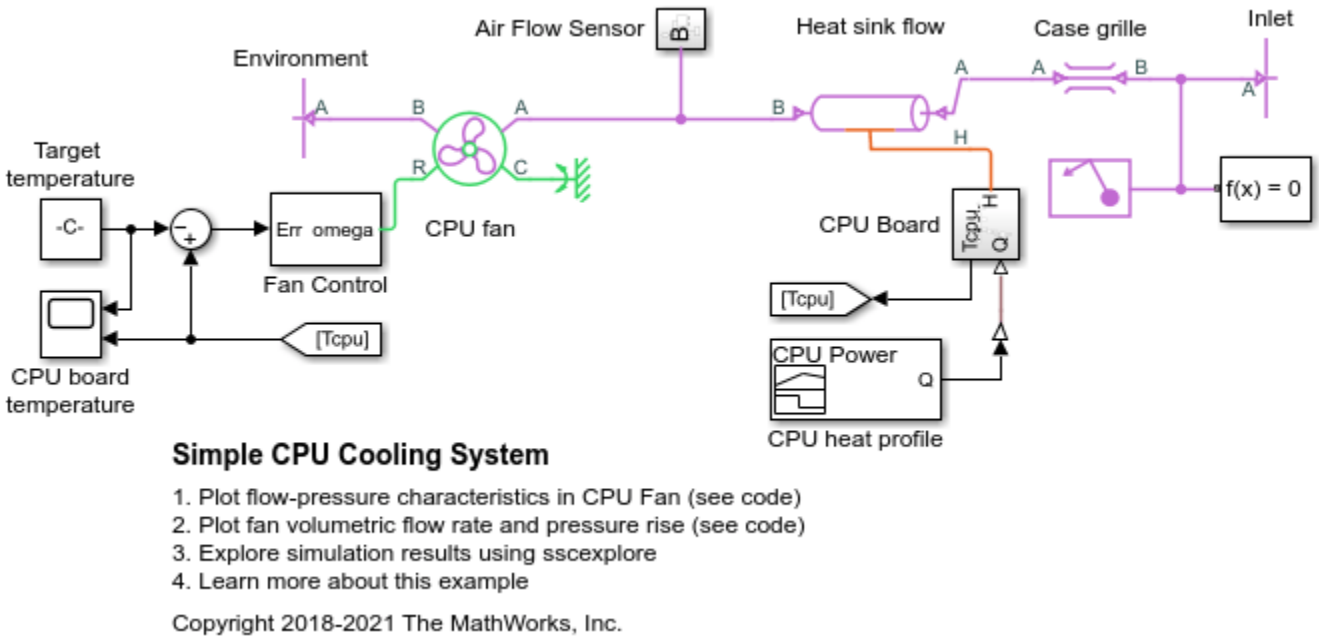
The plot below shows the pressure and mass flow rates in air muscle actuators.


Air Muscle Actuators
Pressure (MPa)
0
0.2
0.4
0.6
0.8
Top
Bottom
0
1
2
3
4
5
6
7
8
9
10
Actuator length (cm)
20
25
30
Top
Bottom
Time (s)

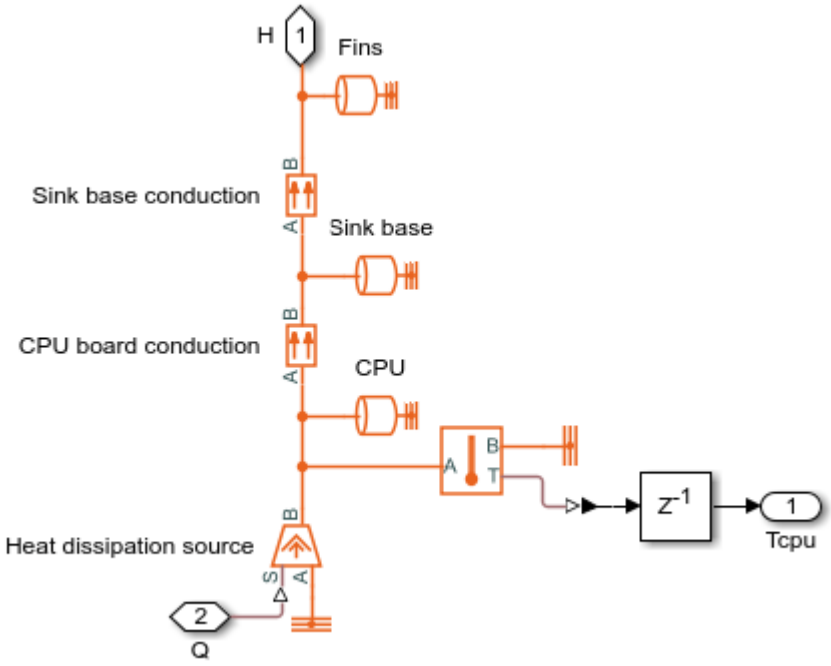
Simple CPU Cooling System

This example shows a simple CPU cooling system consists of a heat sink, a CPU fan, and fan controllers. The heat generated by the CPU is transferred to the heat sink by conduction and it is dissipated to the cooling air by forced convection mechanism. The heat sink is a parallel plate fin with rectangular fins between the plate fins. The CPU fan moves air across the heat sink by drawing air into the case grille from the outside and ejecting warm air from inside. The fan controller unit includes three control levels, representing a 3-speed switch, according to the CPU temperature. For lower CPU temperature, the fan speed is decreased and for higher temperature it is increased.

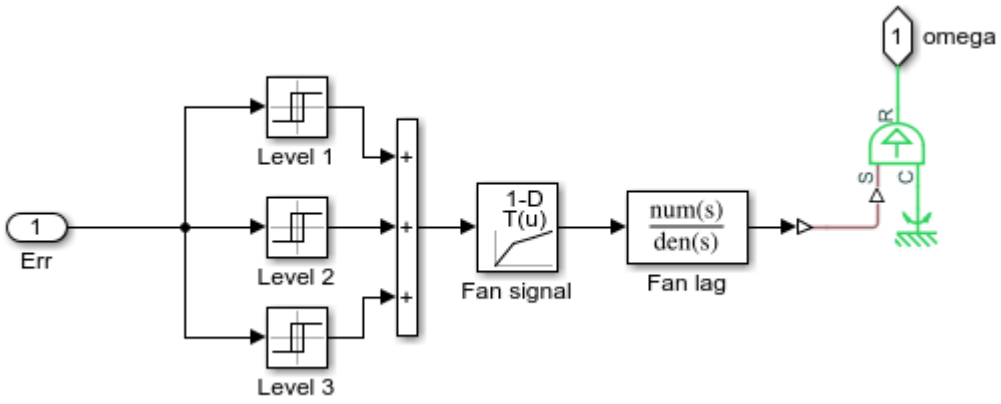
Model



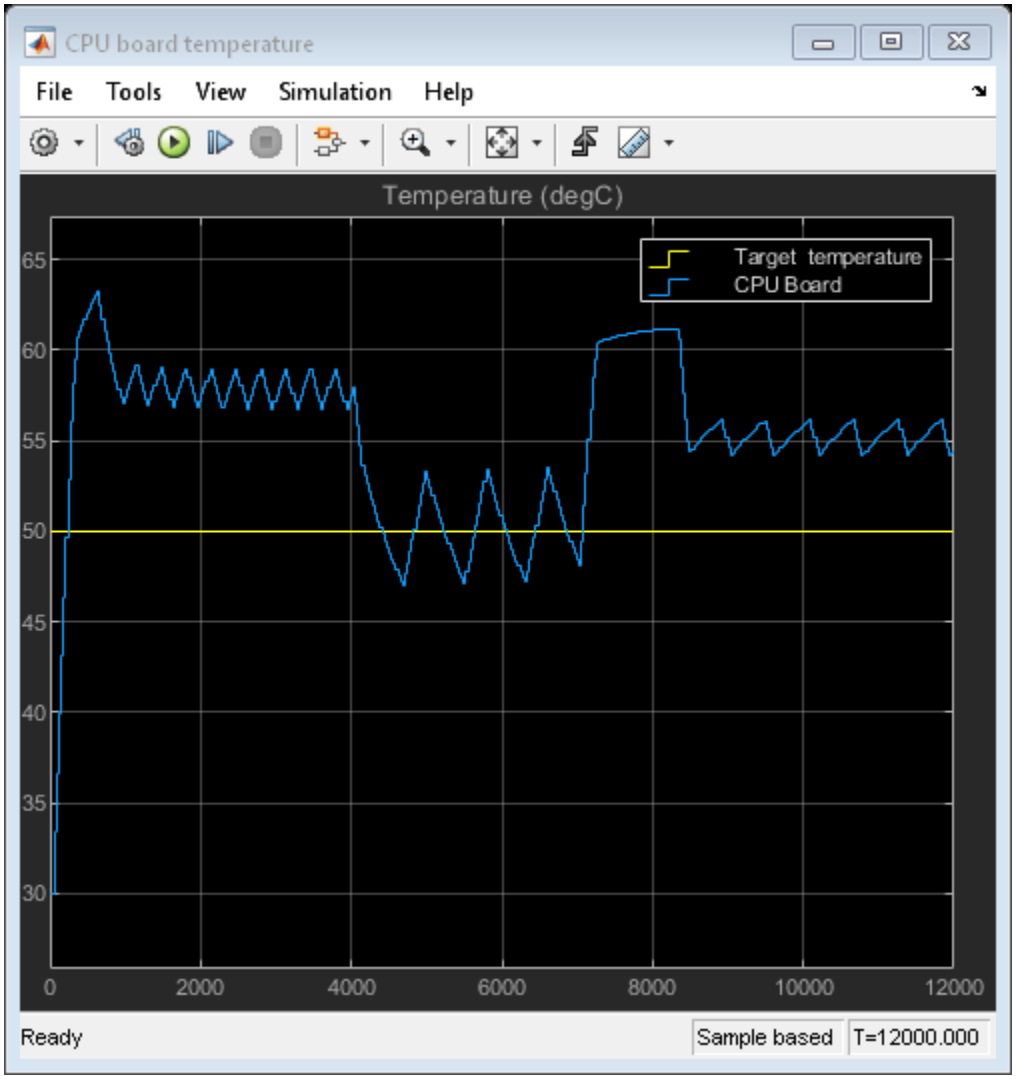
CPU Board Subsystem



Fan Control Subsystem

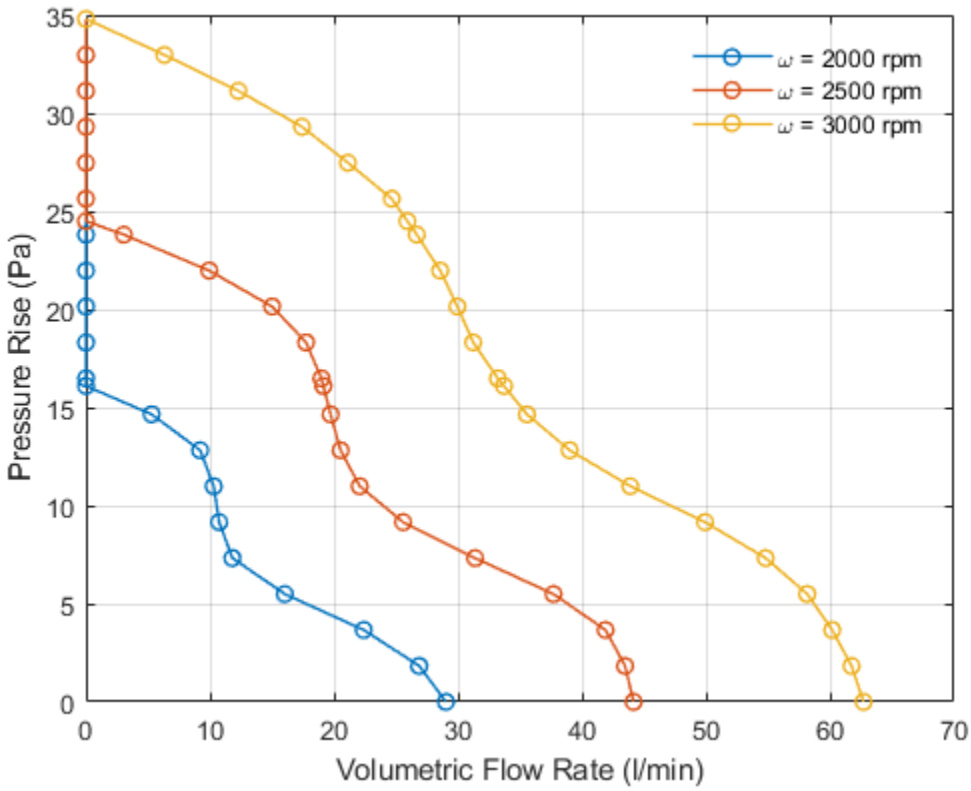


Simulation Results from Scopes

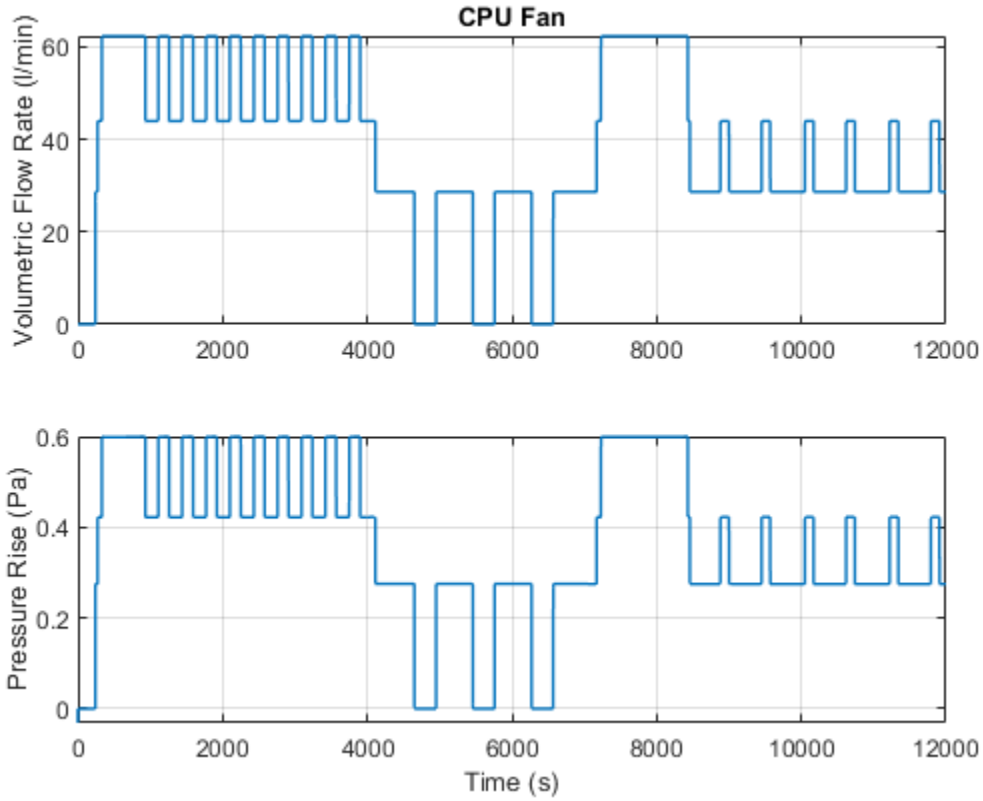


Simulation Results from Simscape Logging

The plot below shows the volumetric flow rate - pressure rise characteristics of CPU fan.



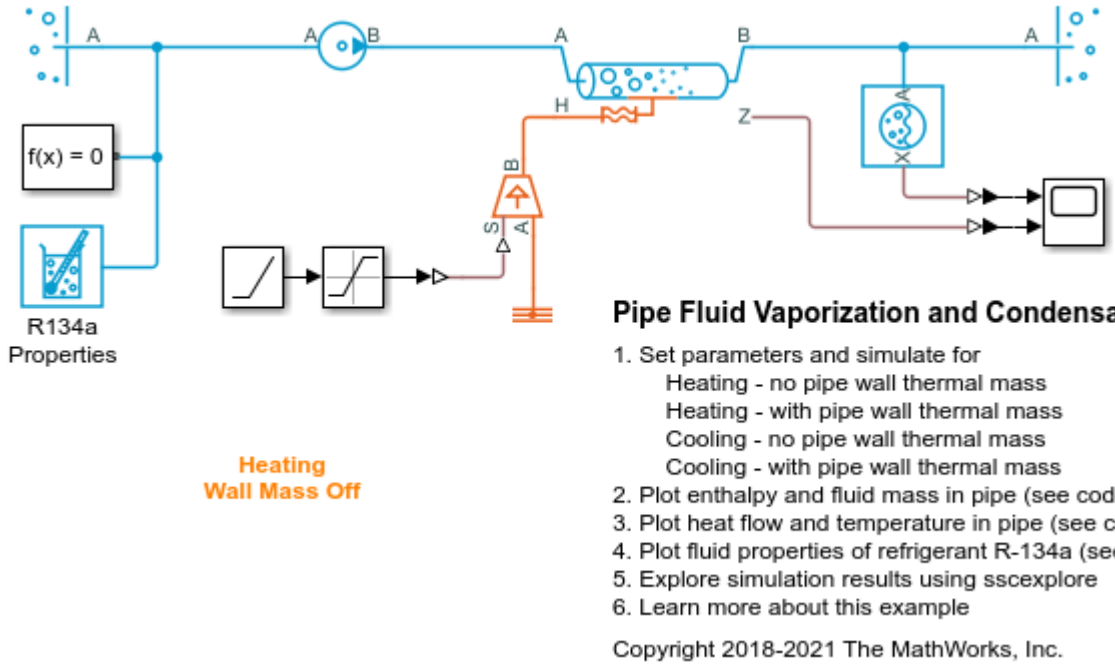
The plots below show the volumetric flow rate through and pressure rise across the CPU fan.


CPU Fan
Volumetric Flow Rate (l/min)
60
40
20
0
0
2000
4000
6000
8000
10000
12000
Pressure Rise (Pa)
0.6
0.4
0.2
0
0
2000
4000
6000
8000
10000
12000
Time (s)

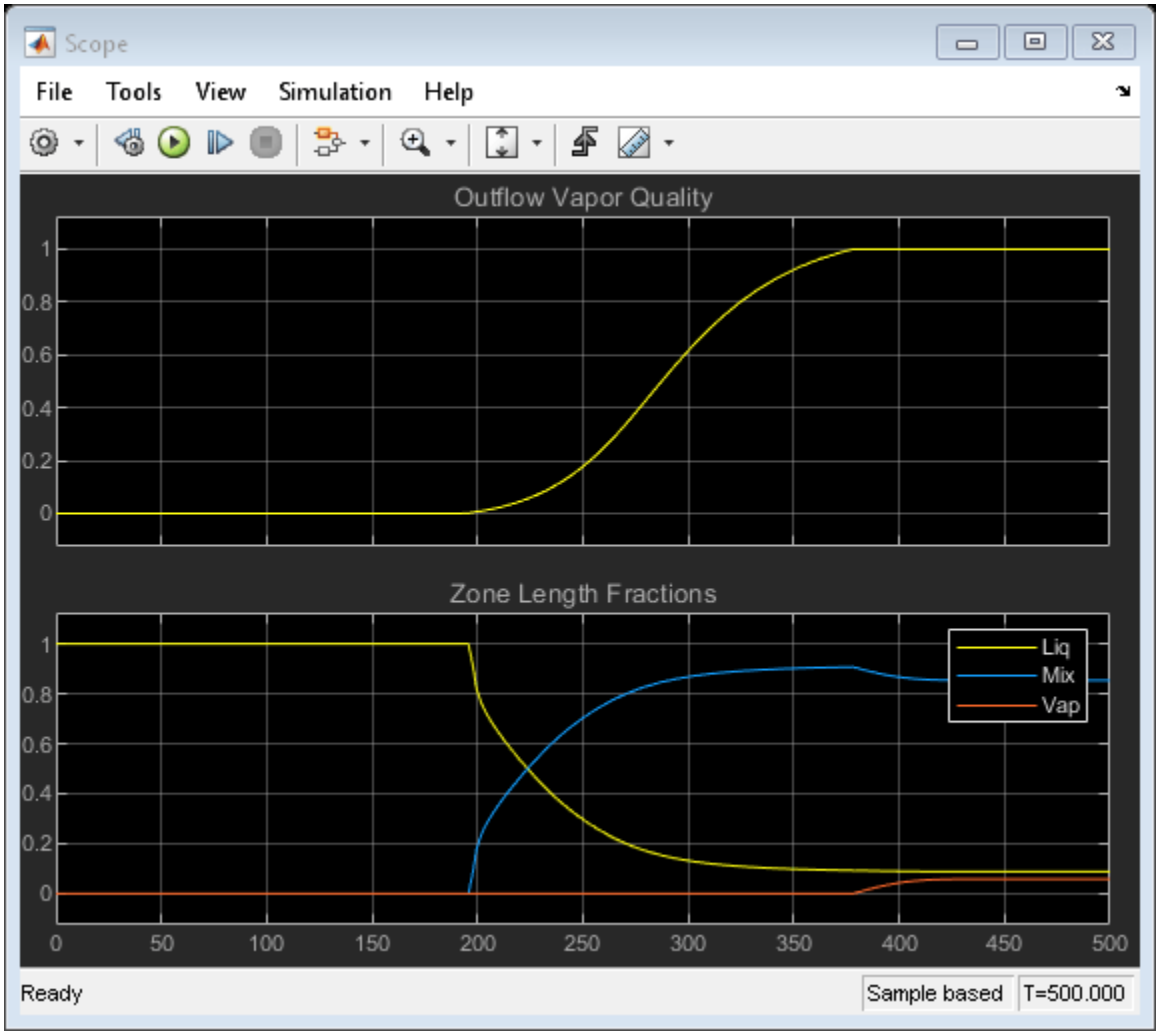
Pipe Fluid Vaporization and Condensation

This example shows the 3-Zone Pipe (2P) block used to model vaporization or condensation of fluid flow in a pipe. The block divides the internal fluid volume into up to three zones: liquid zone, mixture zone, and vapor zone, depending on the state of the fluid along the pipe. As fluid flows through the pipe, heat is transferred between the environment external to the pipe and the fluid inside the pipe, causing it to change from liquid to mixture to vapor for the heating case or from vapor to mixture to liquid for the cooling case. The effect of thermal storage in the pipe wall can be optionally turned on by specifying a nonzero pipe wall thickness.

Model

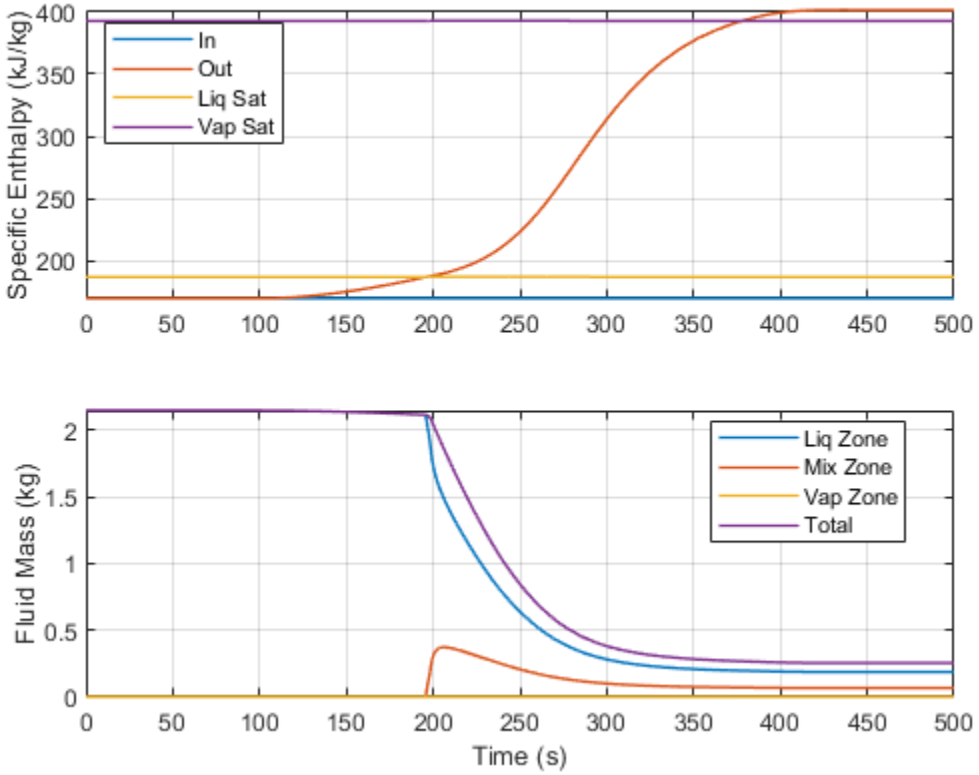


Simulation Results from Scopes

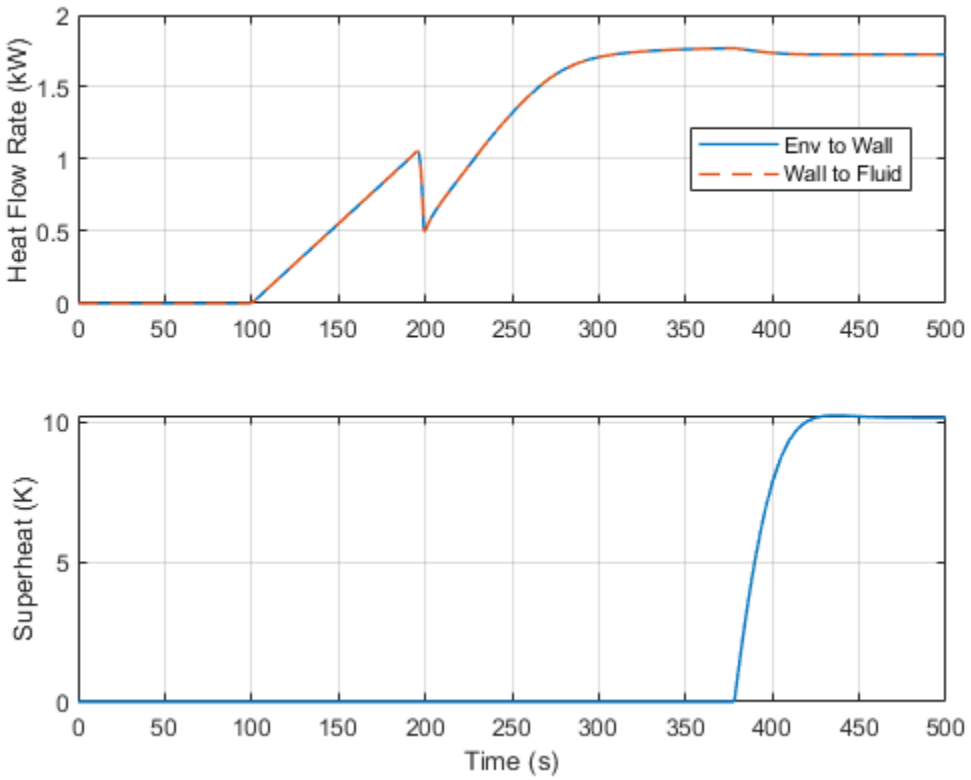


Simulation Results from Simscape Logging

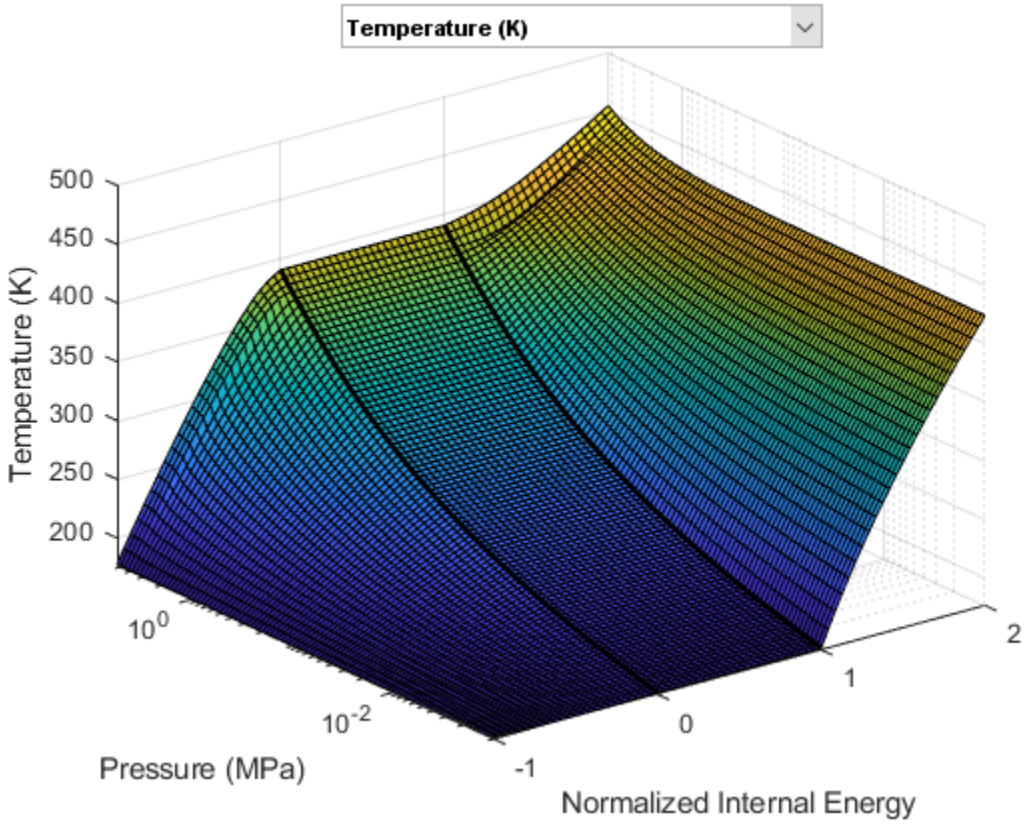
This plot shows specific enthalpy at the beginning and at the end of the pipe. The appearance and disappearance of the liquid, mixture, and vapor zones occurs when the specific enthalpies cross a saturation boundary. The plot also shows the amount of fluid mass within each zone.

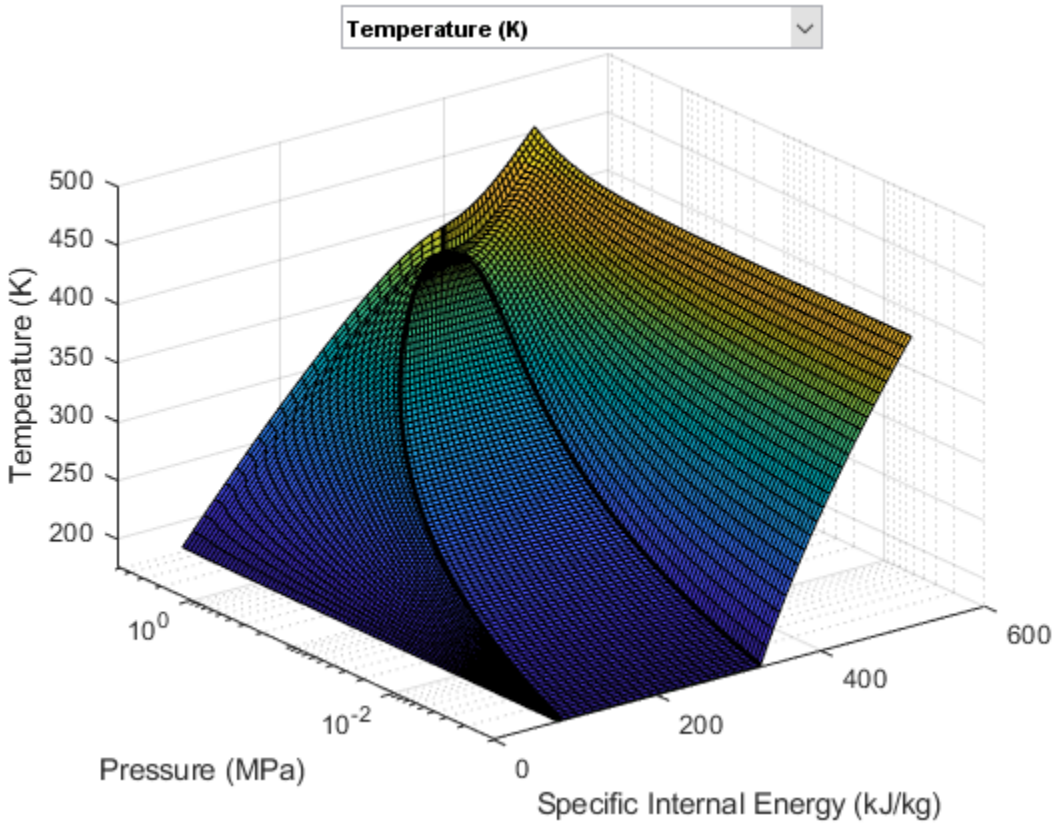


This plot shows the rate of heat transfer, which consists of the heat flow rate from the external environment to the pipe wall and the heat flow rate from the pipe wall to the fluid. When the pipe wall thermal mass is omitted, the two heat flow rates are equal. When there is nonzero pipe wall thermal mass, the two heat flow rates are not equal until steady-state is reached due to thermal storage in the segment of the pipe wall over each zone. The plot also shows the amount of superheat in the outflow for the heating case or the amount of subcooling in the outflow for the cooling case.


Heat Flow Rate (kW)
2
1.5
1
0.5
0
0
50
100
150
200
250
300
350
400
450
500
Env to Wall
Wall to Fluid
Superheat (K)
10
5
0
0
50
100
150
200
250
300
350
400
450
500
Time (s)

Fluid Properties

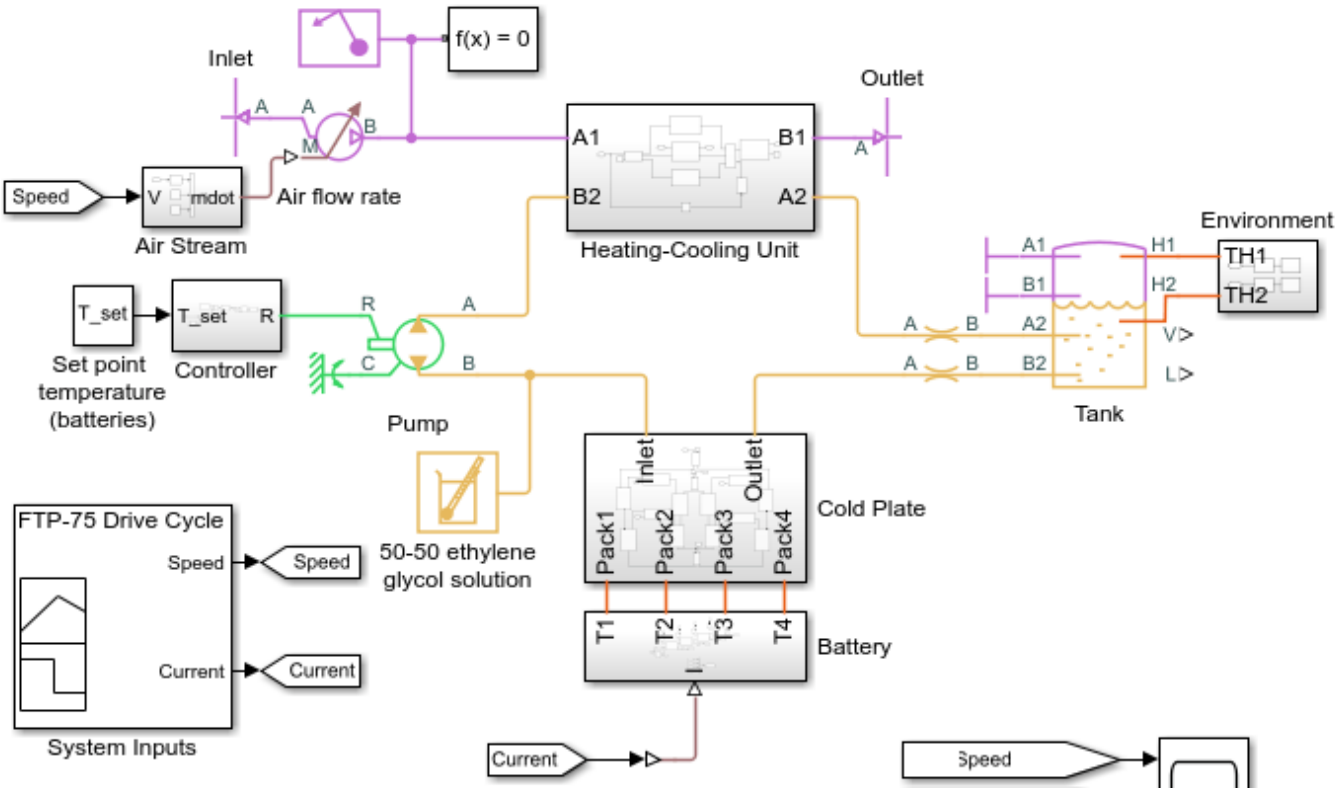



Temperature (K)
500
450
400
350
300
250
200
Temperature (K)
10^0
10^{-2}
Pressure (MPa)
0
200
400
600
Specific Internal Energy (kJ/kg)

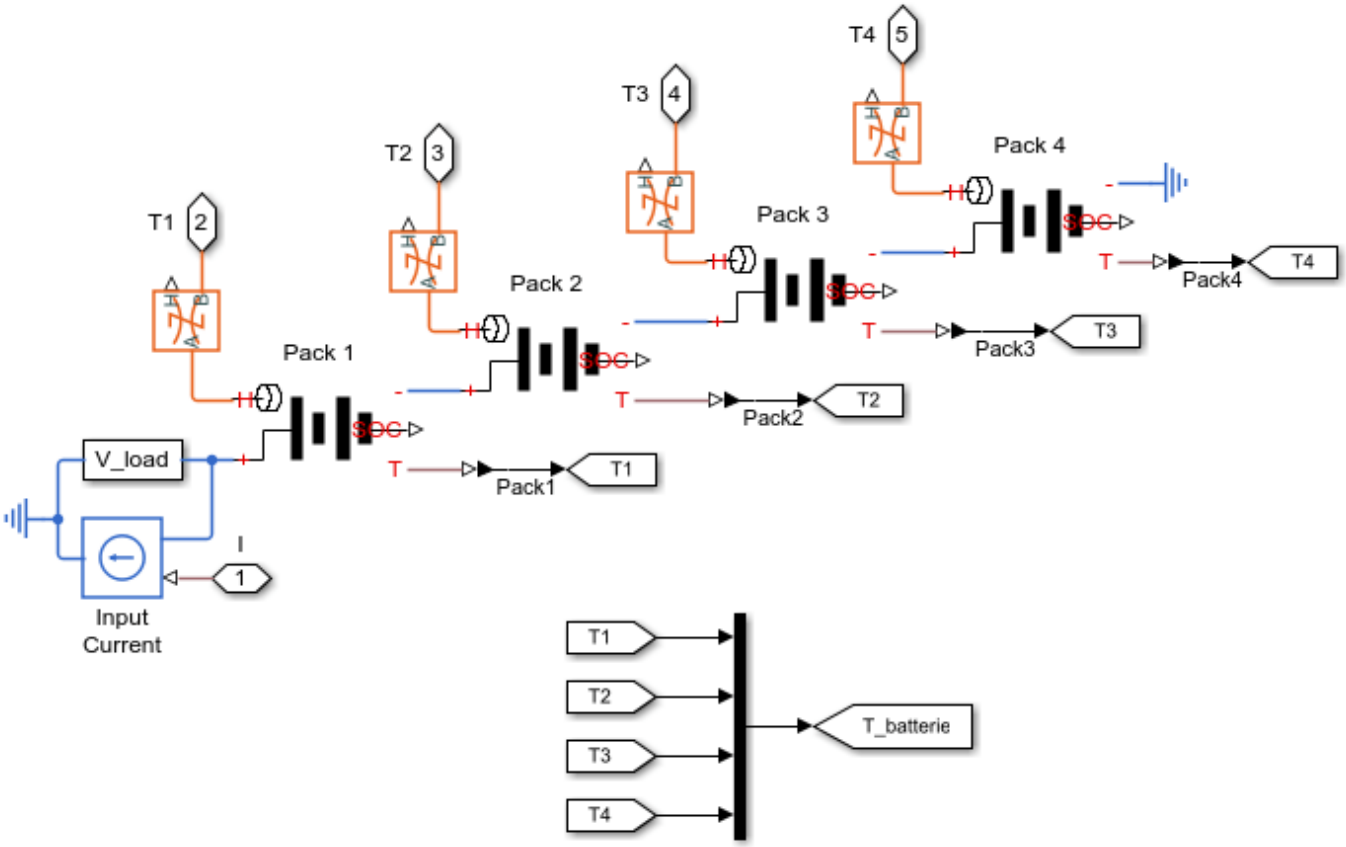
EV Battery Cooling System

This demo shows an Electric Vehicle (EV) battery cooling system. The battery packs are located on top of a cold plate which consists of cooling channels to direct the cooling liquid flow below the battery packs. The heat absorbed by the cooling liquid is transported to the Heating-Cooling Unit. The Heating-Cooling Unit consists of three branches to switch operating modes to cool and heat the battery. The Heater represents an electrical heater for fast heating of the batteries under low temperature conditions. The Radiator uses air-cooling and/or heating when the batteries are operated stably. The Refrigerant system is used for cooling the overheated batteries. The refrigeration cycle is represented by the amount of heat flow extracted from the cooling liquid. The system is simulated under either FTP-75 drive cycle or fast charge scenarios with different environment temperatures.

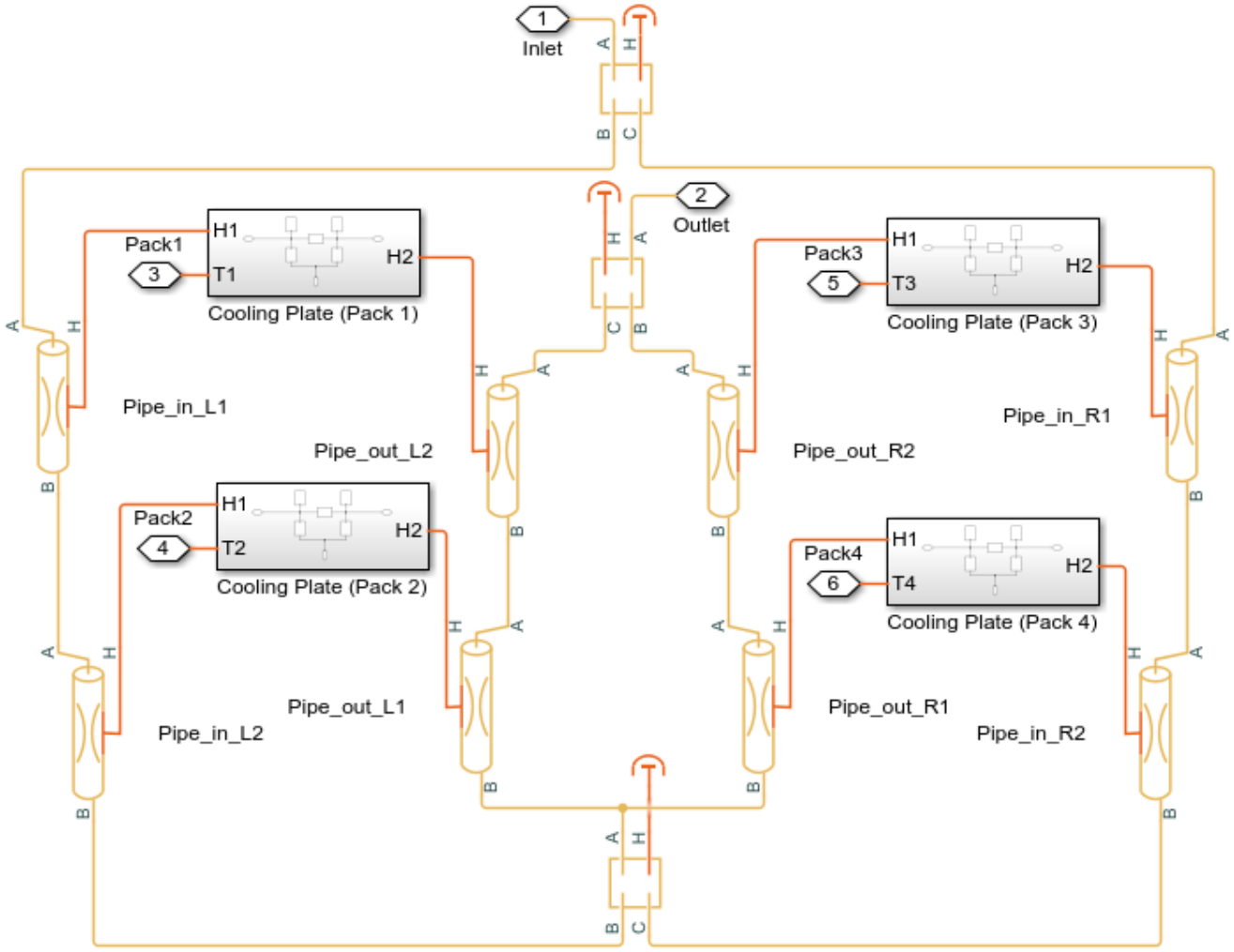
Model



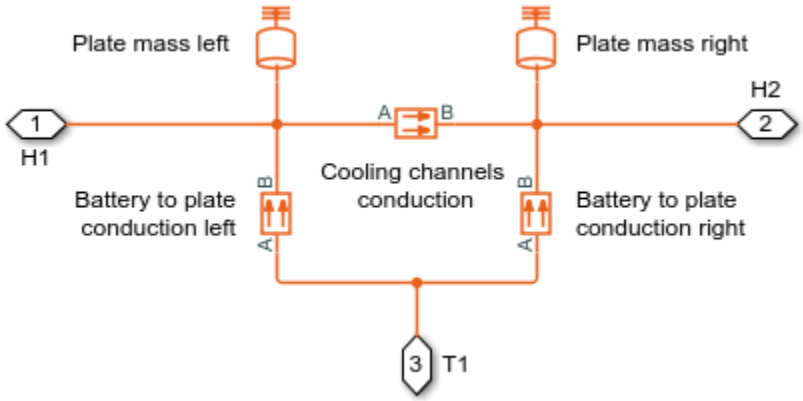
Battery Subsystem



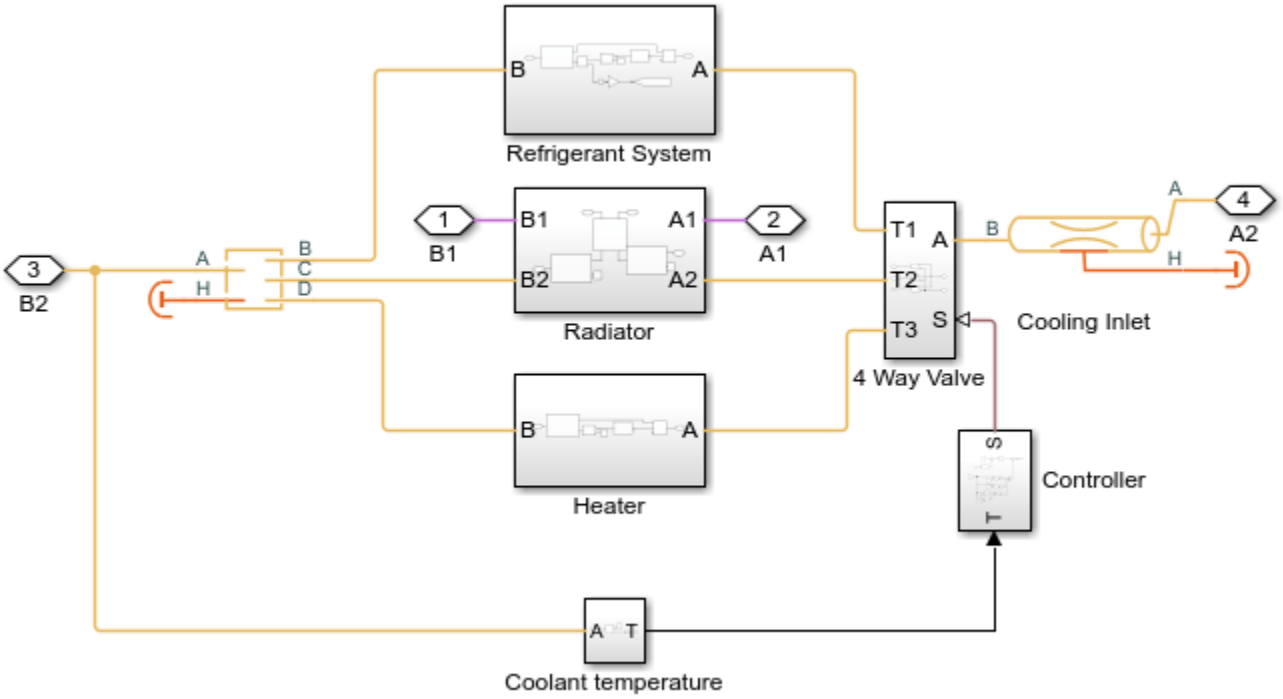
Cold Plate Subsystem



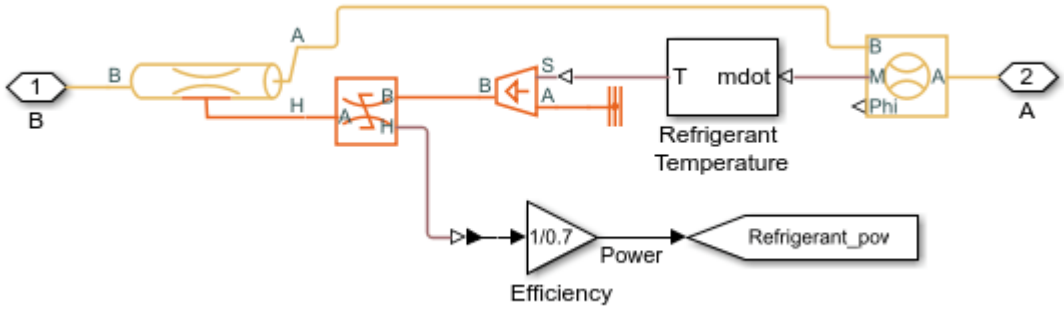
Cooling Plate (Pack 1) Subsystem



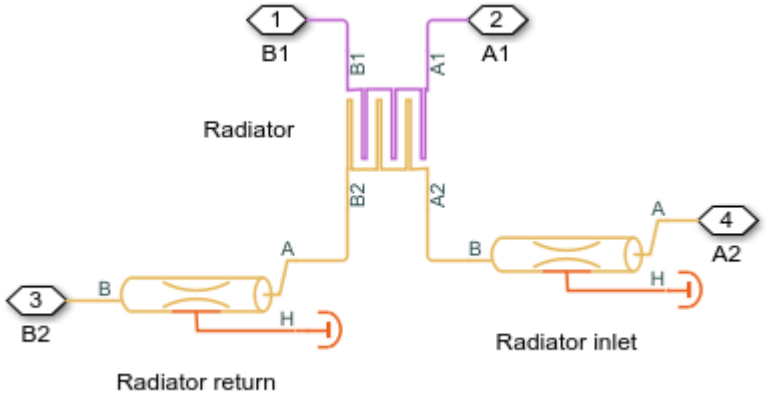
Heating-Cooling Unit Subsystem



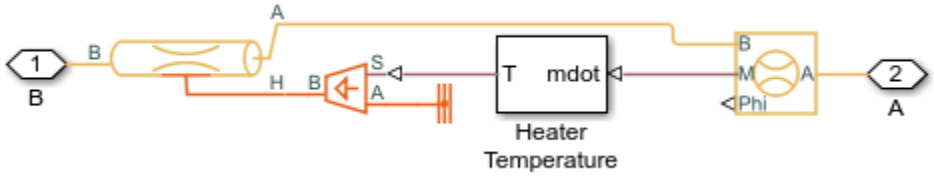
Refrigerant Subsystem



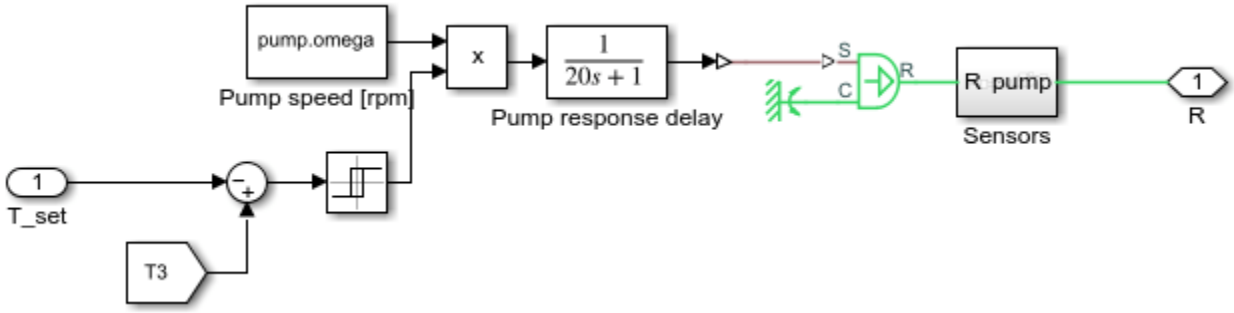
Radiator Subsystem



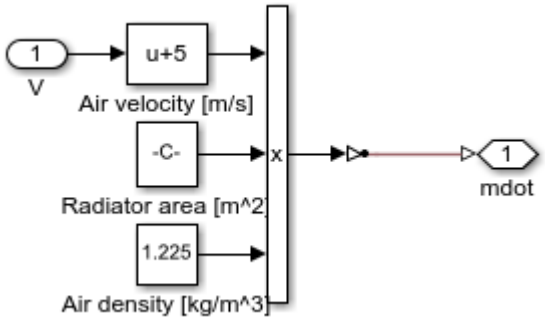
Heater Subsystem



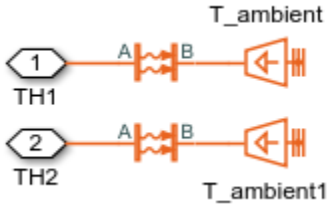
Pump Controller



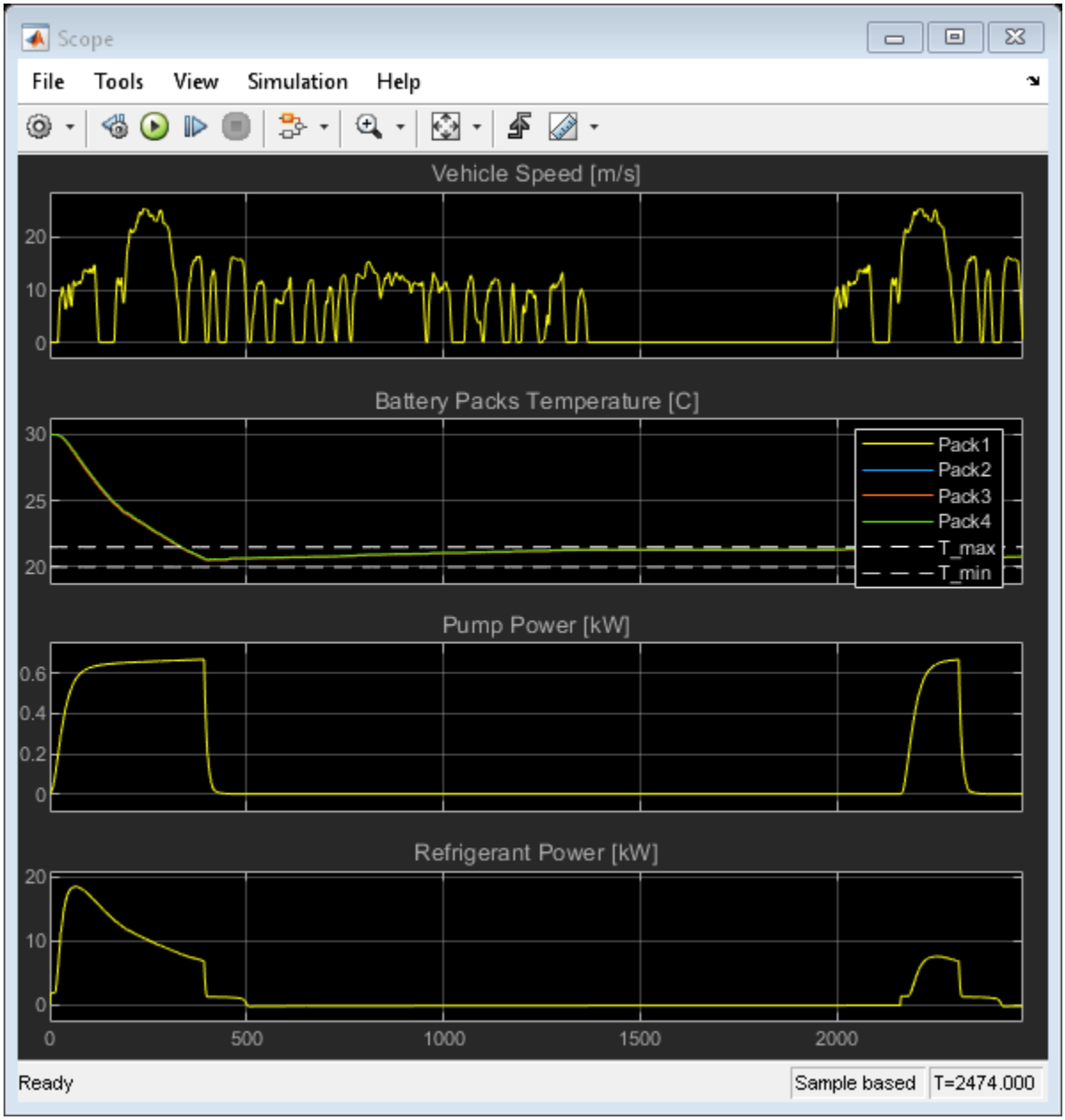
Air Stream



Environment

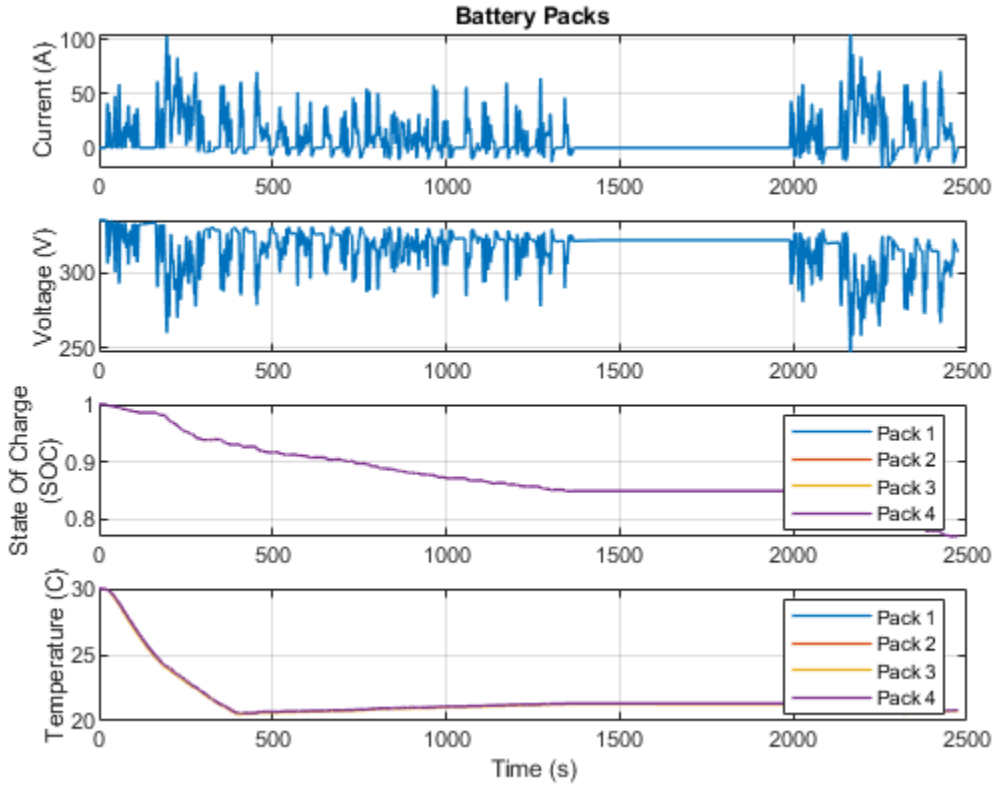


Simulation Results from Scopes

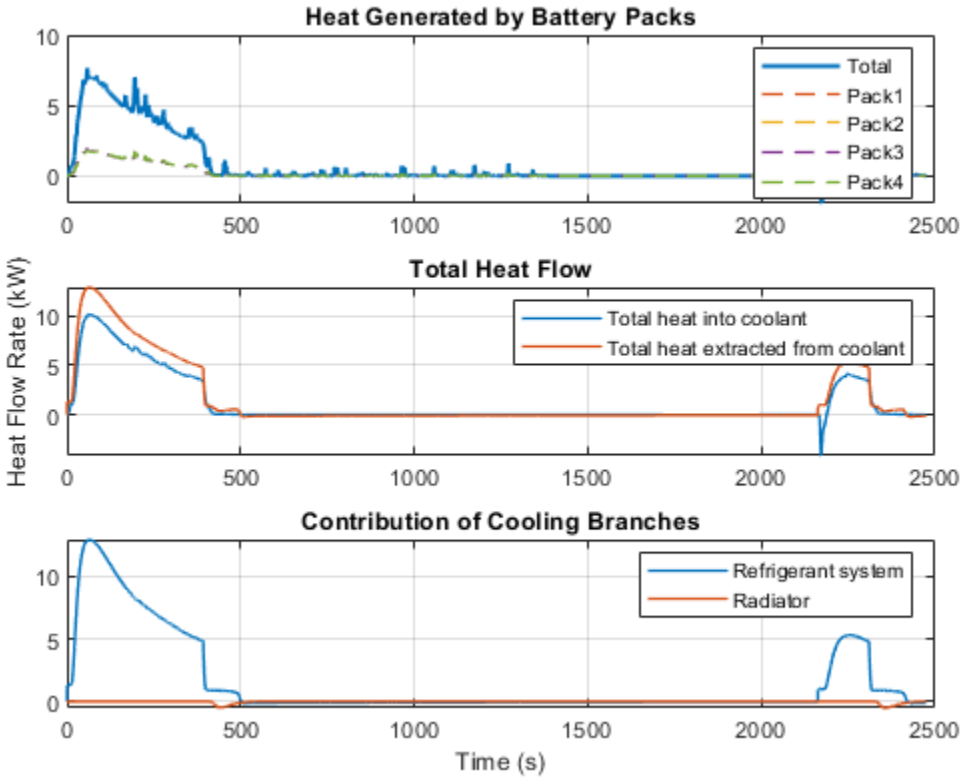


Simulation Results from Simscape Logging

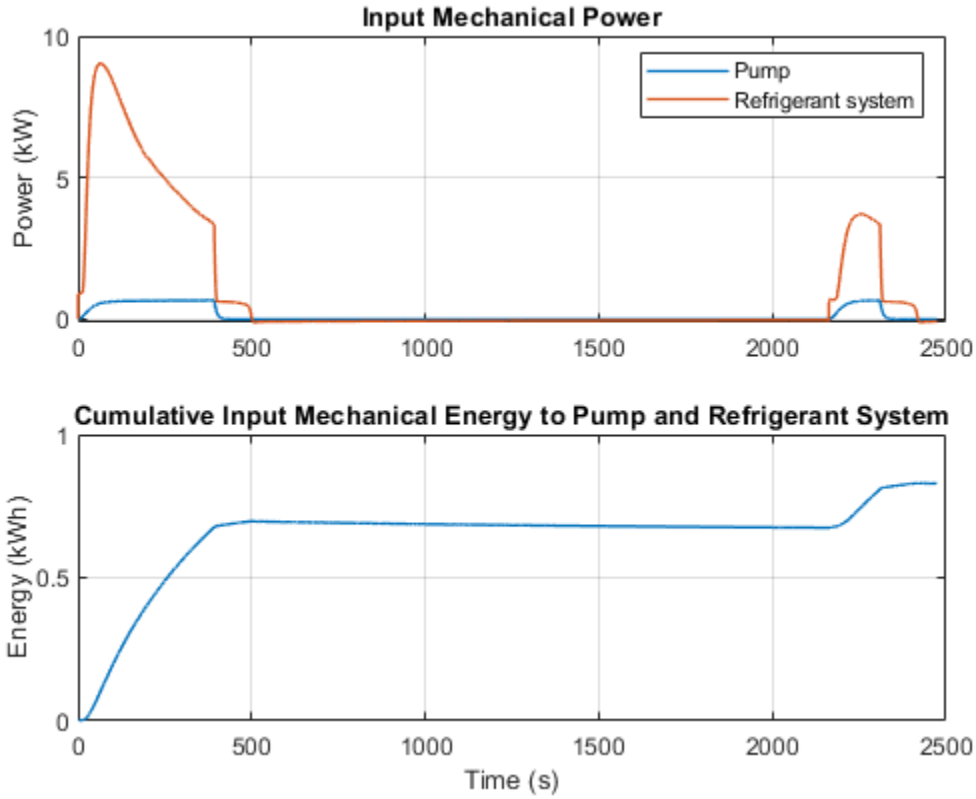
This figure shows the performance of series of four lithium-ion battery packs. The data used for each cell is based on T. Huria, M. Ceraolo, J. Gazzarri,R. Jackey. "High Fidelity Electrical Model with Thermal Dependence for Characterization and Simulation of High Power Lithium Battery Cells," IEEE International Electric Vehicle Conference, March 2012.



This figure shows the total heat flow rate generated by batteries, the total heat flow into to the coolant and the total heat flow extracted from the coolant, and the contribution of each cooling system.



This figure shows the mechanical energy input for the cooling system.

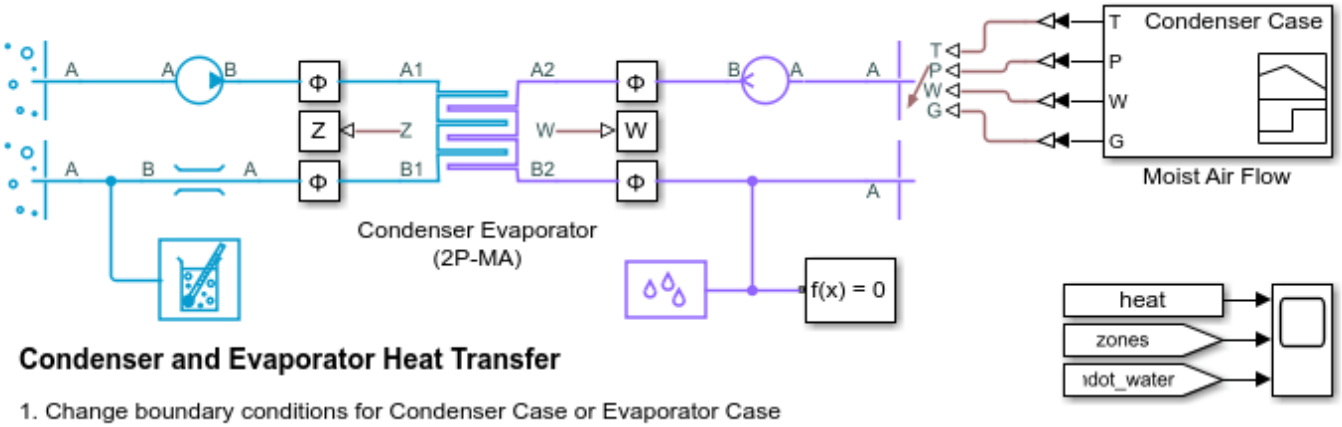

Input Mechanical Power
Power (kW)
10
5
0
0
500
1000
1500
2000
2500
Pump
Refrigerant system
Cumulative Input Mechanical Energy to Pump and Refrigerant System
Energy (kWh)
1
0.5
0
0
500
1000
1500
2000
2500
Time (s)

Condenser and Evaporator Heat Transfer

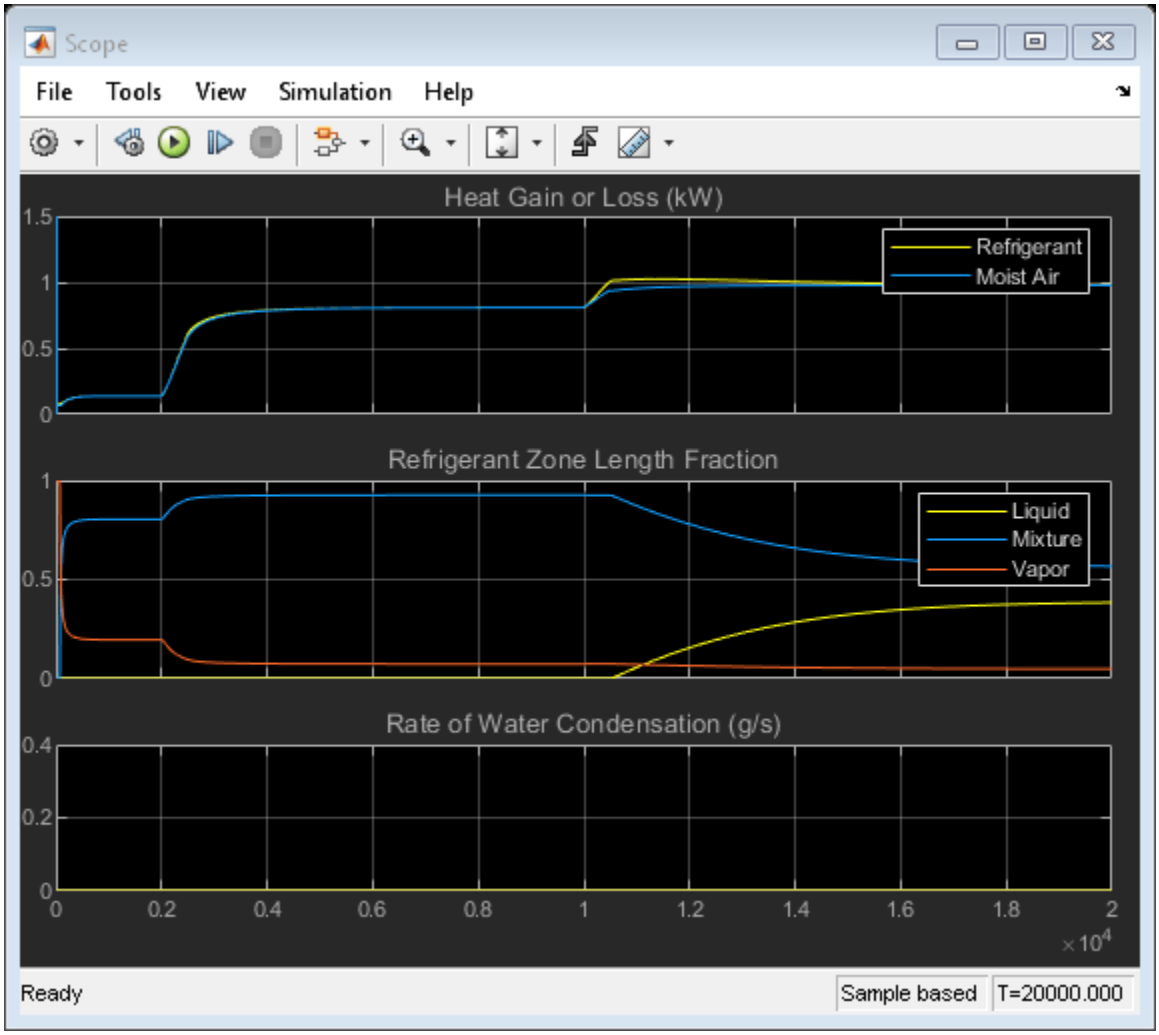
This example models a condenser or an evaporator in simple test setup with R134a refrigerant on the left side and moist air on the right side. It has a cross flow arrangement with the moist air blowing across tube banks filled with the refrigerant.

The Condenser Evaporator (2P-MA) block can operate as either a condenser or an evaporator. In the condenser case, heat flows from the refrigerant to the moist air. This causes the refrigerant to condense from a superheated vapor to a two-phase mixture to a subcooled liquid, resulting in up to three fluid zones along the length of the condenser tubes. In the evaporator case, heat flows from the moist air to the refrigerant. If the moist air is sufficiently wet, water vapor condenses on the surface and is removed from the rest of the moist air flow.

Model

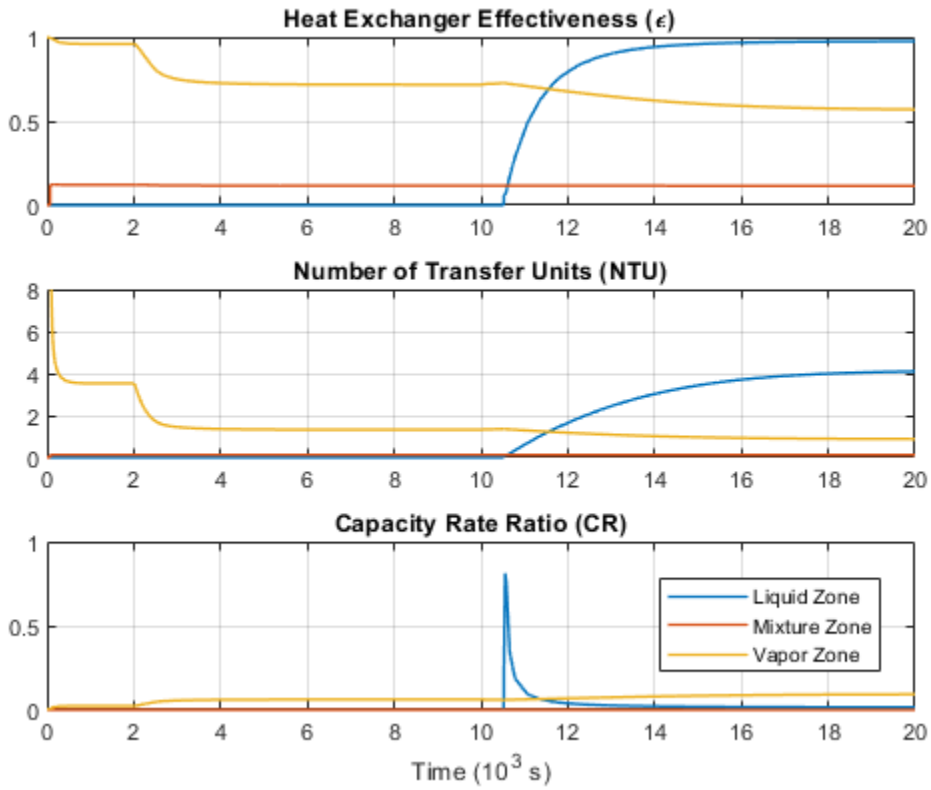


Simulation Results from Scopes

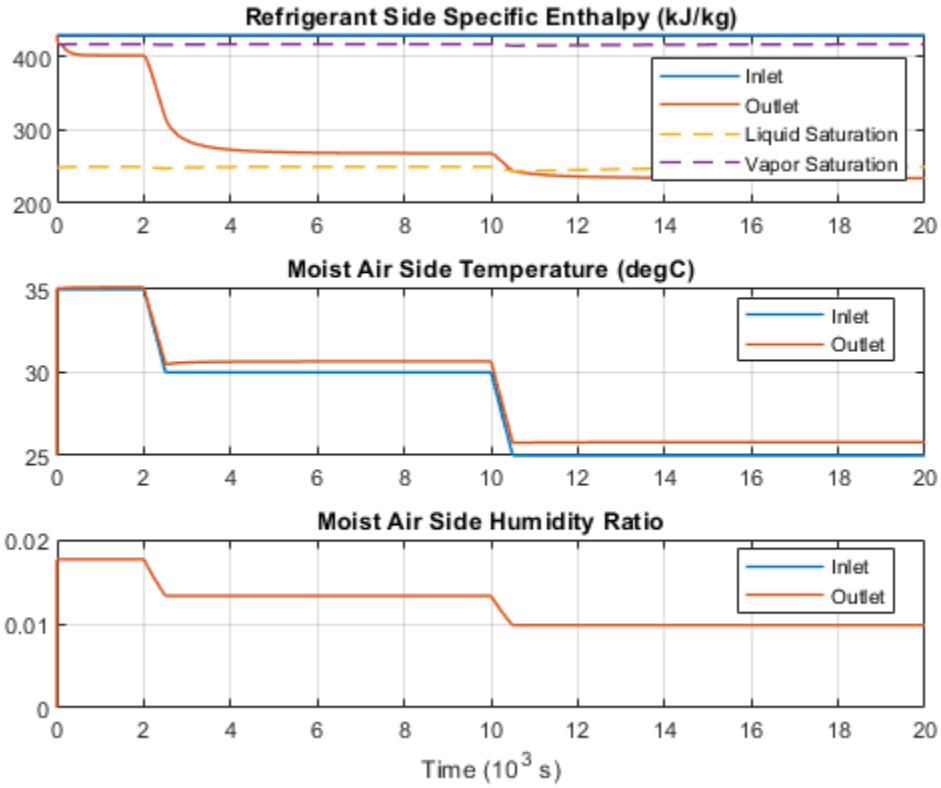


Simulation Results from Simscape Logging

Heat transfer between the refrigerant and the moist air is computed with the ϵ-NTU method. The liquid zone, mixture zone, and vapor zone along the refrigerant tubes have different heat transfer coefficients, heat transfer surface areas, and inlet temperatures. Therefore, the heat exchanger effectiveness, number of transfer units, and the capacity rate ratio are different in each zone. If a zone does not exist, a default value of 0 is shown.



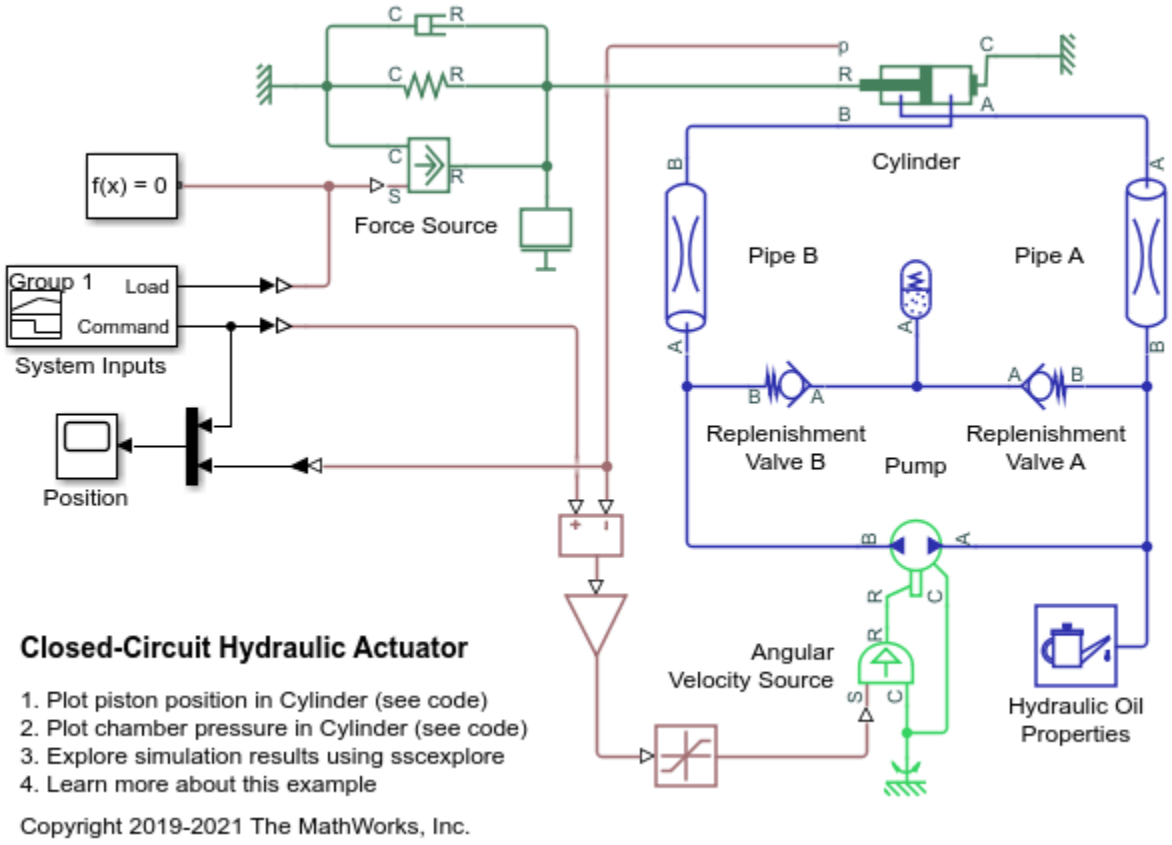
This plot shows the fluid state at the inlets and outlets of the condenser or the evaporator. Liquid, mixture, and vapor zones appears and disappears as the specific enthalpy crosses the saturation boundaries.


Refrigerant Side Specific Enthalpy (kJ/kg)
400
300
200
0
2
4
6
8
10
12
14
16
18
20
Inlet
Outlet
Liquid Saturation
Vapor Saturation
Moist Air Side Temperature (degC)
35
30
25
Inlet
Outlet
Moist Air Side Humidity Ratio
0.02
0.01
0
Inlet
Outlet
Time (10^3 s)

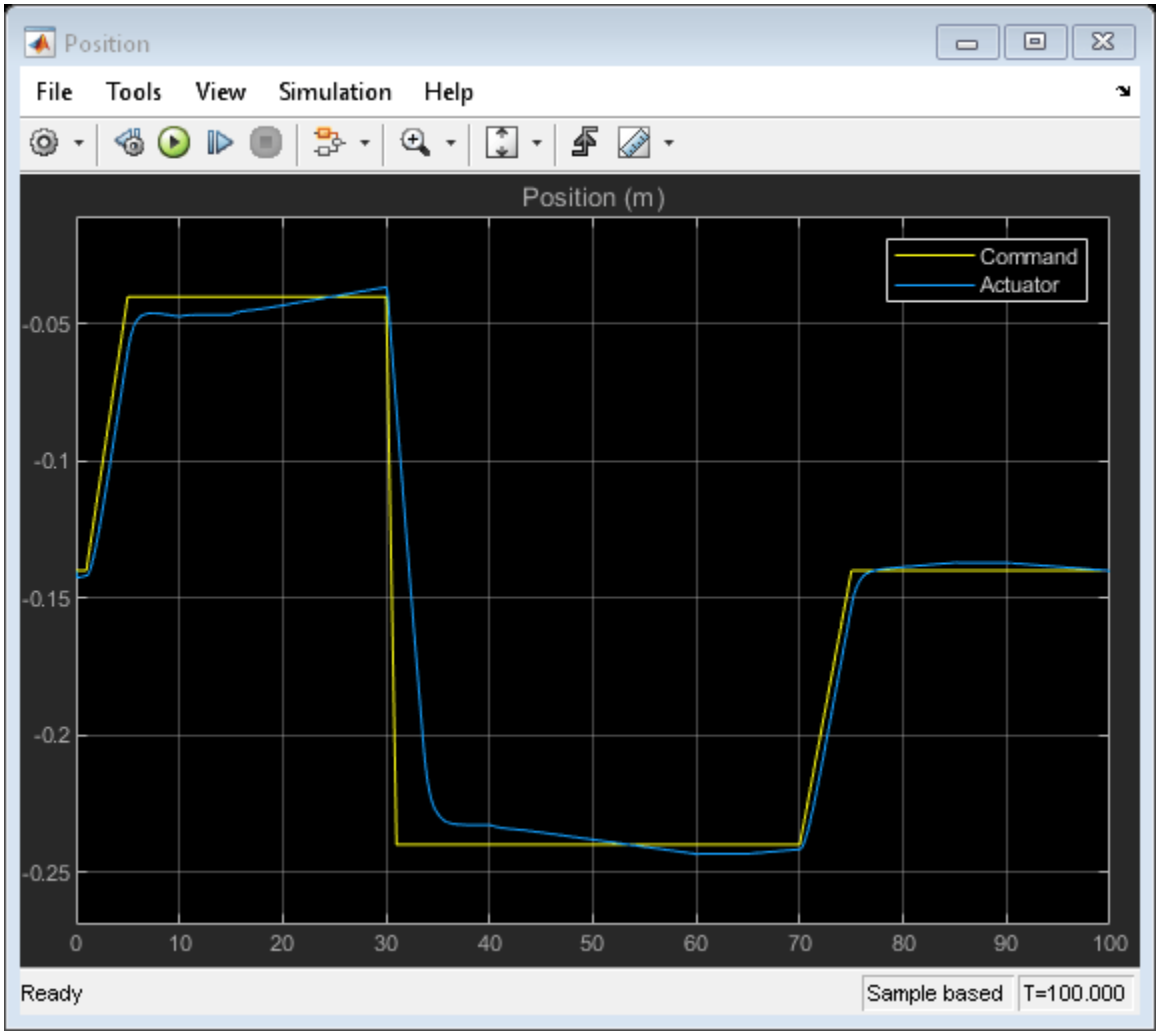
Closed-Circuit Hydraulic Actuator

This example shows a closed-circuit hydraulic actuator driven by a variable-speed pump. The actuator is arranged as a closed fluid system with two replenishment valves (check valves) and a spring-loaded accumulator serving as a replenishment reservoir. The pump speed is controlled by the difference between the commanded and measured piston position. The actuator acts against a spring, a damper, and a time-varying load.

Model

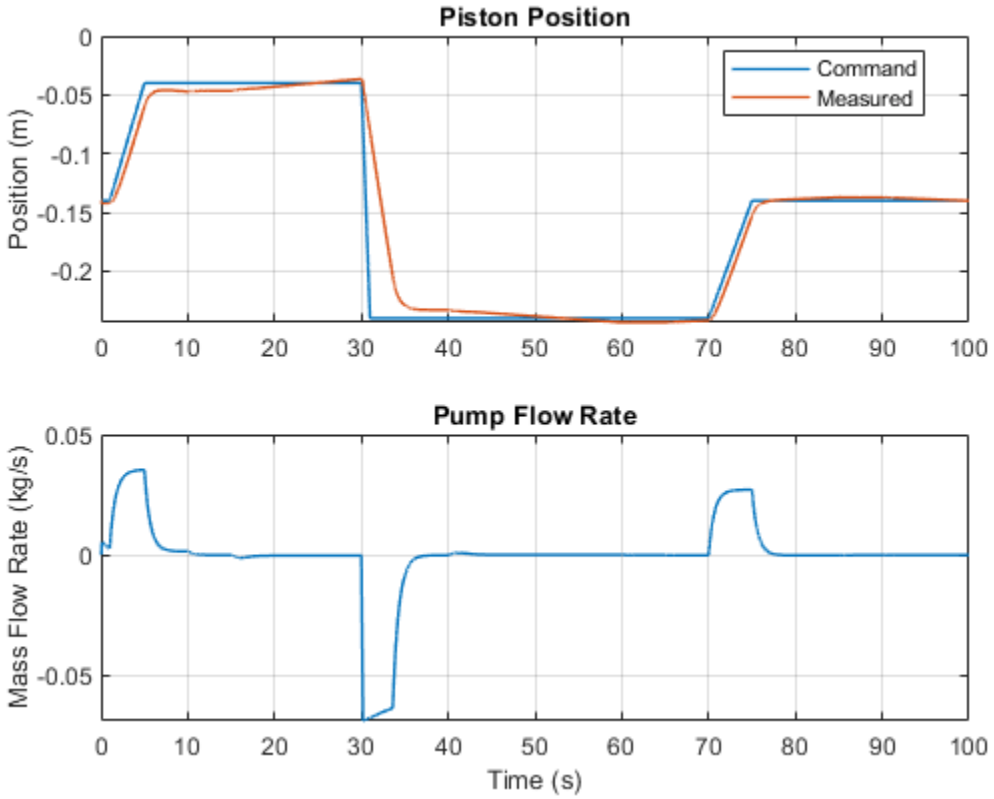


Simulation Results from Scopes

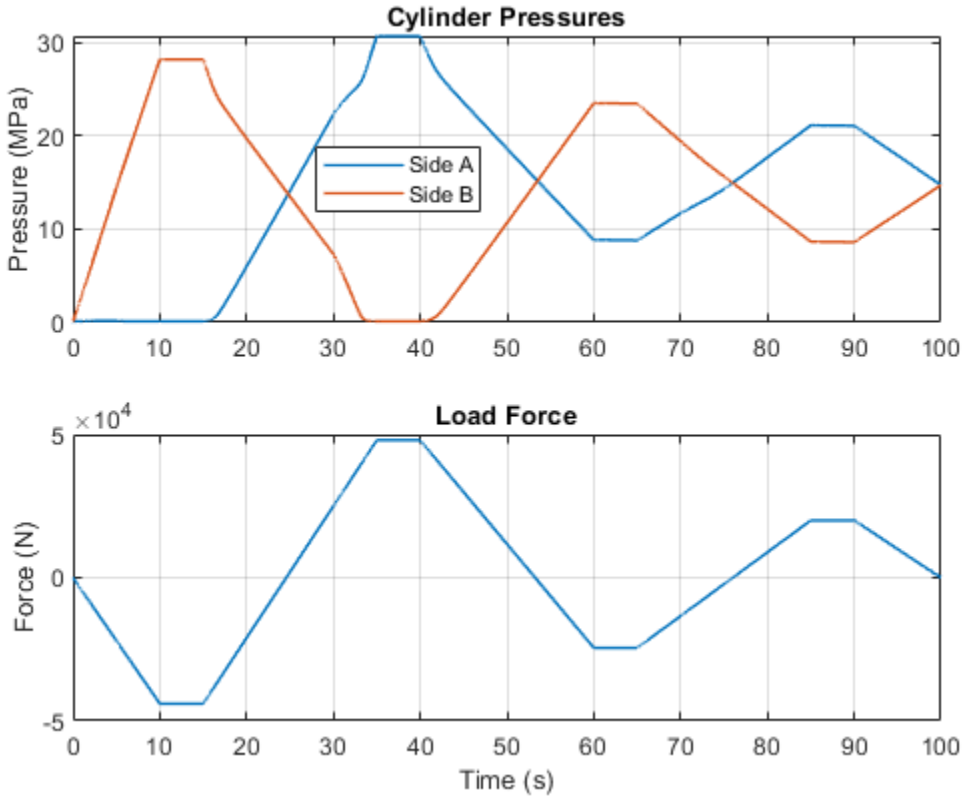


Simulation Results from Simscape Logging

This plot shows the position of the piston and the flow rate through the pump.



This plot shows the pressures inside the cylinder chambers and the time-varying load on the piston.


Cylinder Pressures
Pressure (MPa)
Side A
Side B
Load Force
×10^4
Force (N)
Time (s)

Photovoltaic Thermal (PV/T) Hybrid Solar Panel

This example shows how to model the cogeneration of electrical power and heat using a hybrid PV/T solar panel. The generated heat is transferred to water for household consumption.

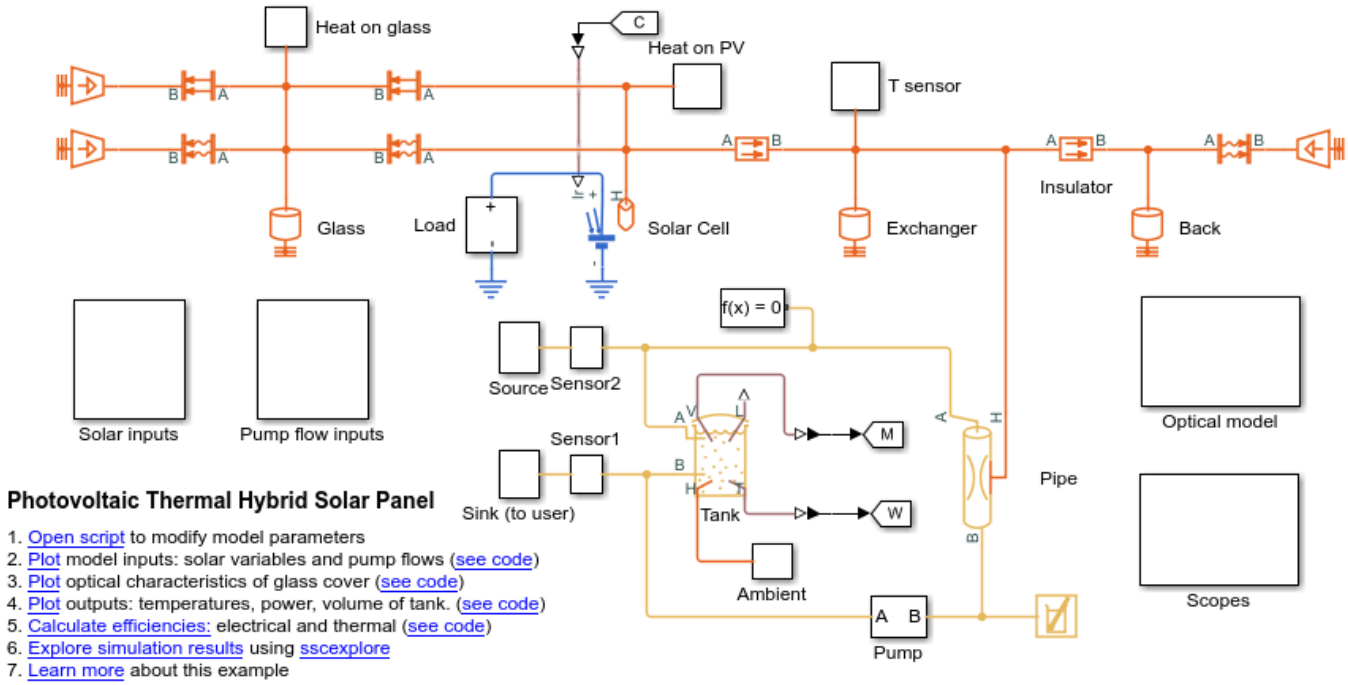
It uses blocks from the Simscape™ Foundation™, Simscape Electrical™, and Simscape Fluids™ libraries. The electrical portion of the network contains a Solar Cell block, which models a set of photovoltaic (PV) cells, and a Load subsystem, which models a resistive load. The thermal network models the heat exchange that occurs between the physical components of the PV panel (glass cover, heat exchanger, back cover) and the environment. Heat is exchanged through conduction, convection, and radiation. The thermal-liquid network contains a pipe, a tank, and pumps. The pumps control the flows of the liquids through the system.

To model the reflection, absorption and transmission of light in the glass cover, an optical model is embedded in a MATLAB® Function block.

Model overview

Open the model to view its structure:

```
open_system('sscv_hybrid_solar_panel');
```



The thermal network is in red, the electrical network in blue and the thermal liquid network in yellow. There are subsystems for the solar and pump inputs. There is also a subsystem that contains scopes for visualizing the simulation results. Another subsystem contains the function for the optical model.

Parameters

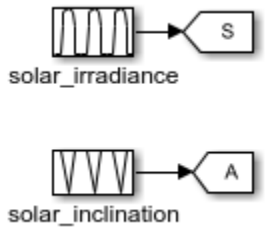
You can use the hybrid_solar_panel_data.m script to change the parameter values that this example uses for components such as the load, solar cell, pipe, and tank.

```
edit sscv_hybrid_solar_panel_data;
```

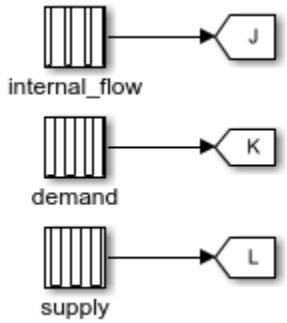
Inputs

The inputs of the model are the pump flows and the solar variables for irradiance and incidence angle. A repeating sequence block is used to define the inputs because they follow a 24-hour periodic cycle.

```
open_system('sscv_hybrid_solar_panel/Solar inputs');
```




```
open_system('sscv_hybrid_solar_panel/Pump flow inputs');
```

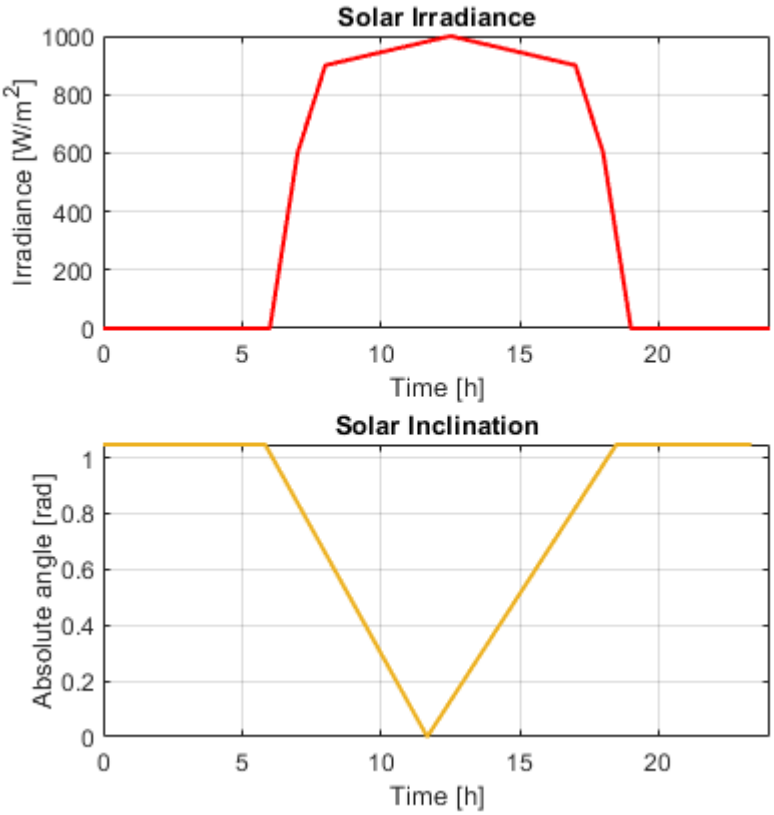


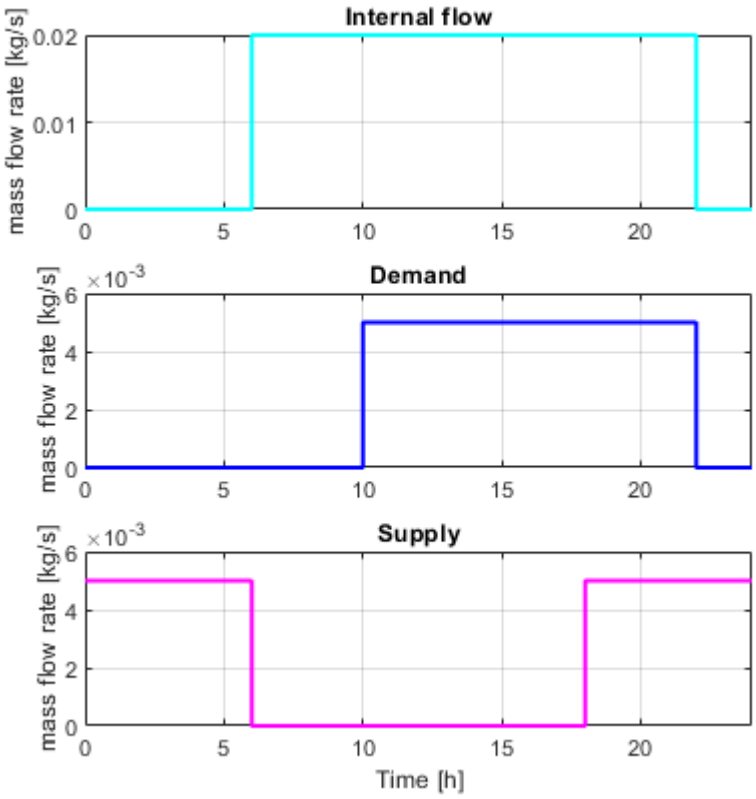
The sun rises at 6:00 and sets at 19:00. The irradiance follows a bell curve that peaks at 12:30. The incidence angle changes from pi/3 to 0.

There are three pumps. One pump models user demand, another models source supply, and a third models internal flow that forces convection in the pipe. The demand is constant and only non-zero from 10:00 to 22:00. The supply is constant and only non-zero from 18:00 to 6:00. The internal flow is also constant and only non-zero from 6:00 to 22:00. This model is used for the internal flow because it is not efficient to force heat exchange during the night when the ambient temperature is low.

You can use the hybrid_solar_panel_plot_inputs.m script to plot the inputs:

```
sscv_hybrid_solar_panel_plot_inputs;
```

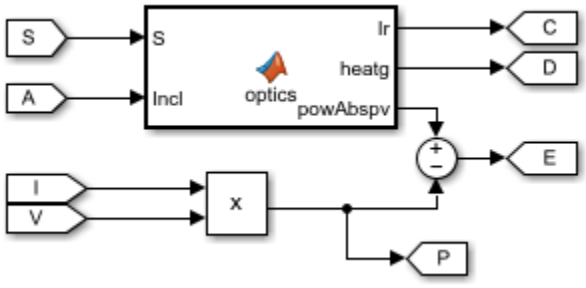

Solar Irradiance
Irradiance [W/m^2]
1000
800
600
400
200
0
0
5
10
15
20
Time [h]
Solar Inclination
Absolute angle [rad]
1
0.8
0.6
0.4
0.2
0
0
5
10
15
20
Time [h]



Optical model for the glass cover

The optical model is inside a subsystem:

```
open_system('sscv_hybrid_solar_panel/Optical model');
```



It consists of a MATLAB® Function block, with the 2 solar inputs, and 3 outputs: the transmitted irradiance on the PV cells, the heat absorbed by the glass, and the radiative power absorbed by the PV cells. Part of it will be transformed into electrical power (V*I) and the rest will be heat absorbed by the PV cells.

From an optical point of view, the glass consists of 2 parallel boundaries (air-glass, glass-air), each one of those reflects and transmits light. The reflection coefficient in a boundary is obtained from the

Fresnel equations. r_p is for P-polarization and r_s for S-polarization. The total reflection is the average of both, and the transmittance is $1 - r$ as there is no absorption so far:

$$r_p = \left(\frac{n_{rel}^2 \cos(\theta_i) - \sqrt{n_{rel}^2 - \sin(\theta_i)^2}}{n_{rel}^2 \cos(\theta_i) + \sqrt{n_{rel}^2 - \sin(\theta_i)^2}} \right)^2$$

$$r_s = \left(\frac{\cos(\theta_i) - \sqrt{n_{rel}^2 - \sin(\theta_i)^2}}{\cos(\theta_i) + \sqrt{n_{rel}^2 - \sin(\theta_i)^2}} \right)^2$$

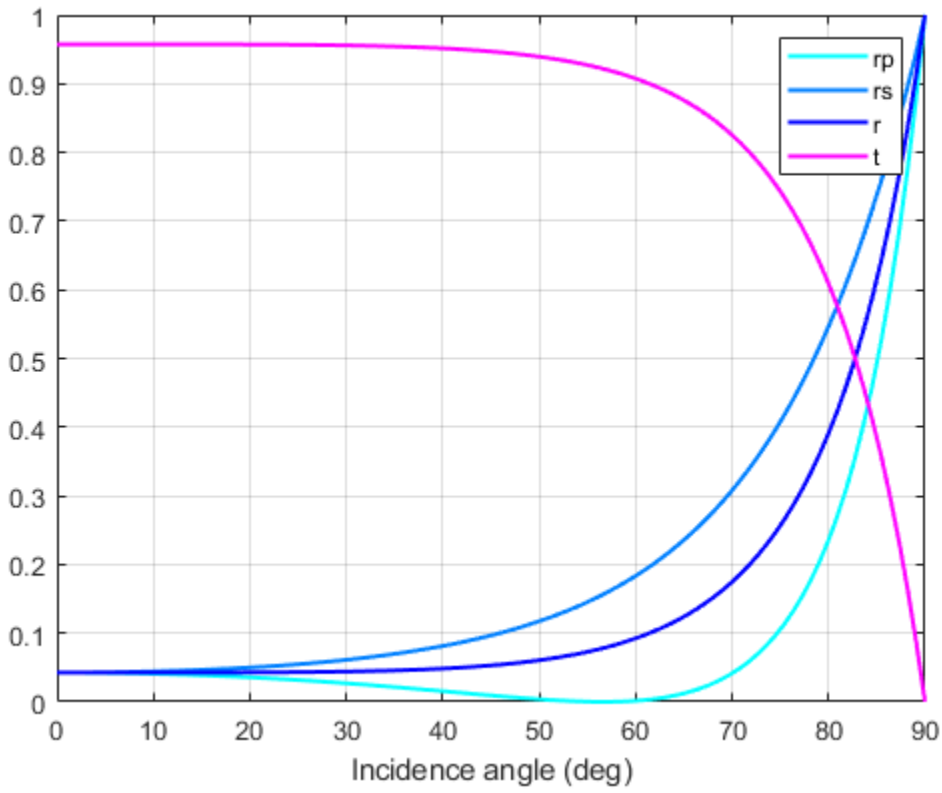
$$r = \frac{1}{2}(r_p + r_s)$$

$$t = 1 - r$$

This is an example of the optical coefficients rp, rs, r and t in function of incidence angle:

```
nrel = 1.52; %Optical index from air to glass
theta = linspace(0, pi/2, 100);
rp = ( nrel^2*cos(theta) - sqrt(nrel^2 - sin(theta).^2) ).^2./...
    ( nrel^2*cos(theta) + sqrt( nrel^2 - sin(theta).^2 ) ).^2 ;
rs = ( cos(theta) - sqrt(nrel^2 - sin(theta).^2) ).^2./...
    ( cos(theta) + sqrt( nrel^2 - sin(theta).^2 ) ).^2 ;
r = 0.5*(rp + rs);
t = 1 - r;

figure();
plot(theta*180/pi, rp, 'Color', [0 1 1], 'LineWidth', 1.5);
hold on
plot(theta*180/pi, rs, 'Color', [0 0.5 1], 'LineWidth', 1.5);
plot(theta*180/pi, r, 'Color', [0 0 1], 'LineWidth', 1.5);
plot(theta*180/pi, t, 'Color', 'm', 'LineWidth', 1.5);
legend('rp','rs','r','t');
xlabel('Incidence angle (deg)');
grid on
box on
```



This is what happens in one boundary, but the glass has 2 parallel boundaries separated by d_g. The angle after the 1st boundary is the incidence angle on the 2nd boundary and is calculated from **Snell's Law**:

$$n_1 \sin(\theta_1) = n_2 \sin(\theta_2)$$

When the light enters the glass, it absorbs part of it with a constant probability per unit length (alpha_g), resulting in an exponential decay from distance travelled for the transmittance coefficient in the glass:

$$\tau_g = \exp\left(\frac{-\alpha_g d_g}{\cos(\theta_2)}\right)$$

Then, when it arrives at the 2nd boundary, it reflects and transmits again with Fresnel equations. The reflected light is trapped inside the glass, reflecting infinite times between the 2 boundaries until completely absorbed. The total reflection and transmission coefficients of the system are then the sum of an infinite geometrical series, for which the result is:

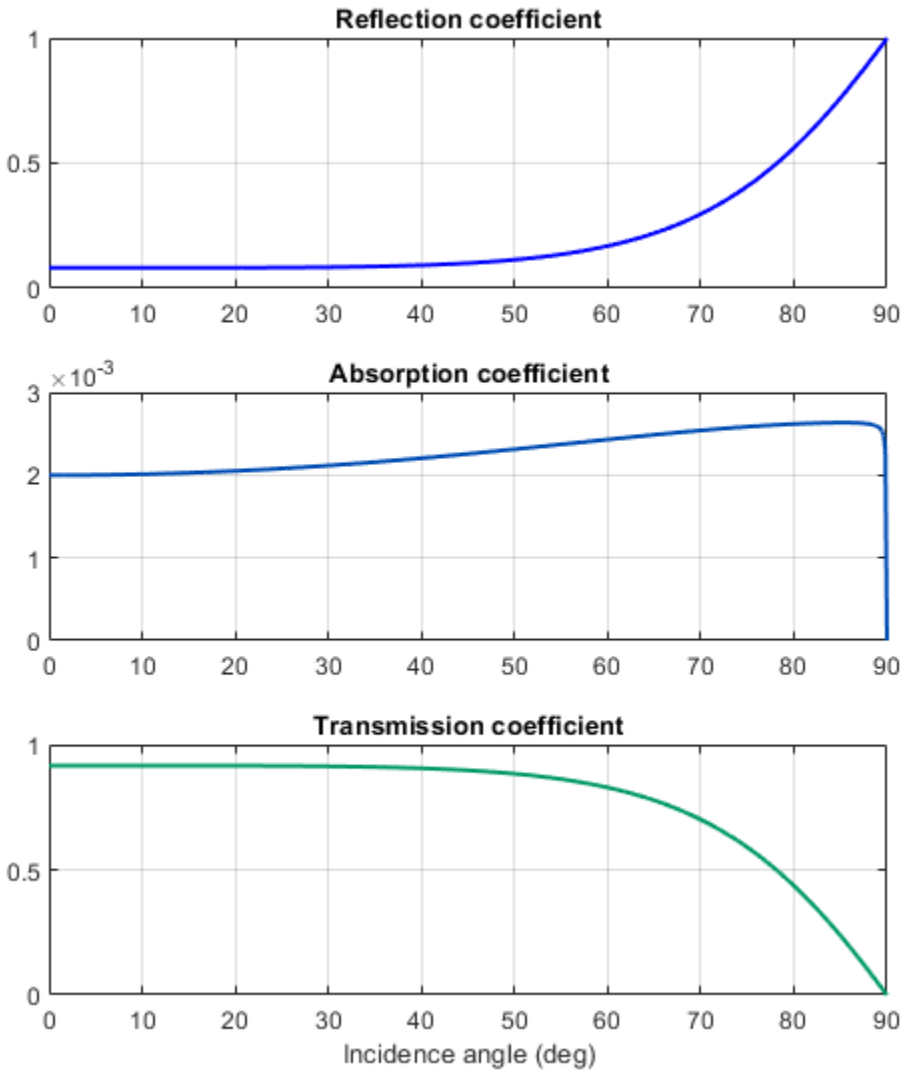
$$T_g = \frac{t_1 \tau_g t_2}{1 - r_1 r_2 \tau_g^2}$$

$$R_g = r_1 + \frac{t_1^2 \tau_g^2 r_2}{1 - r_1 r_2 \tau_g^2}$$

$$A_g = 1 - T_g - R_g$$

Finally, the total optical coefficients for the glass are:

```
sscv_hybrid_solar_panel_plot_optics;
```

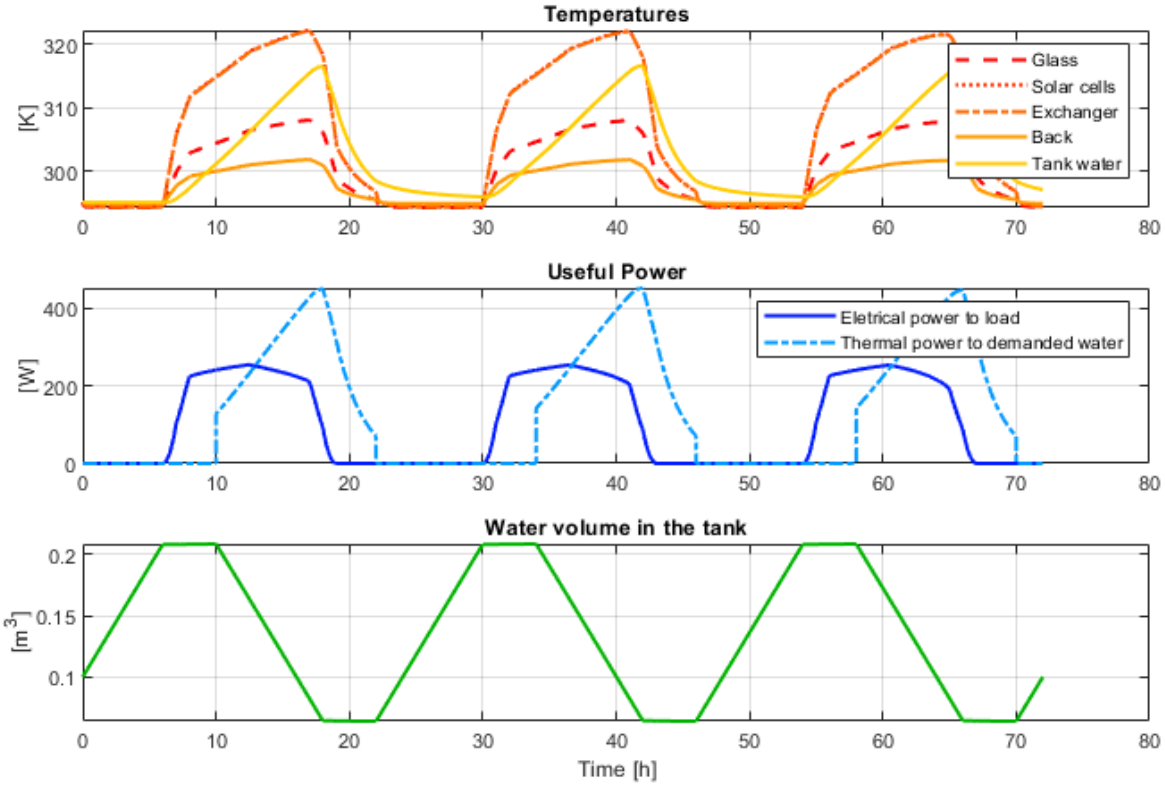


Outputs

The outputs of the model are the temperatures of all components of the panel, the electrical and thermal power, and the volume in the tank.

You can use the script hybrid_solar_panel_plot_outputs to plot the solution:

```
sscv_hybrid_solar_panel_plot_outputs;
```



Efficiency calculation

From the outputs it is possible to calculate the electrical, thermal, and total efficiency of the panel:

```
sscv_hybrid_solar_panel_efficiency;


****** Efficiency Calculation *********

Total input energy from the sun in the period: 43.7828 kWh
Average input energy from the sun per day: 14.5943 kWh/day

Total electrical energy supplied to the load: 7.5326 kWh
Average electrical energy supplied per day: 2.5109 kWh/day

Total absolute thermal energy in the water supplied to the user: 26.4752 kWh
Total absolute thermal energy in the water extracted from the source: 16.5049 kWh

Total used thermal energy (sink - source): 9.9703 kWh
Average used thermal energy per day (sink - source): 3.3234 kWh/day

Electrical efficiency: 0.17205
Thermal efficiency: 0.22772
Total efficiency: 0.39977

**************************************
```

The electrical efficiency is on the order of standard PV cells, but adding the thermal efficiency the production of energy is significantly better, with a system efficiency on the order of a cogeneration plant.

A further analysis could use Simulink® Design Optimization™ or other optimization tools to find optimal values for certain parameters eligible for control, maximizing total efficiency.

Another improvement would be the addition of controllers to the pumps and the electrical load, in order to drive the system to different operating points and optimize the performance.