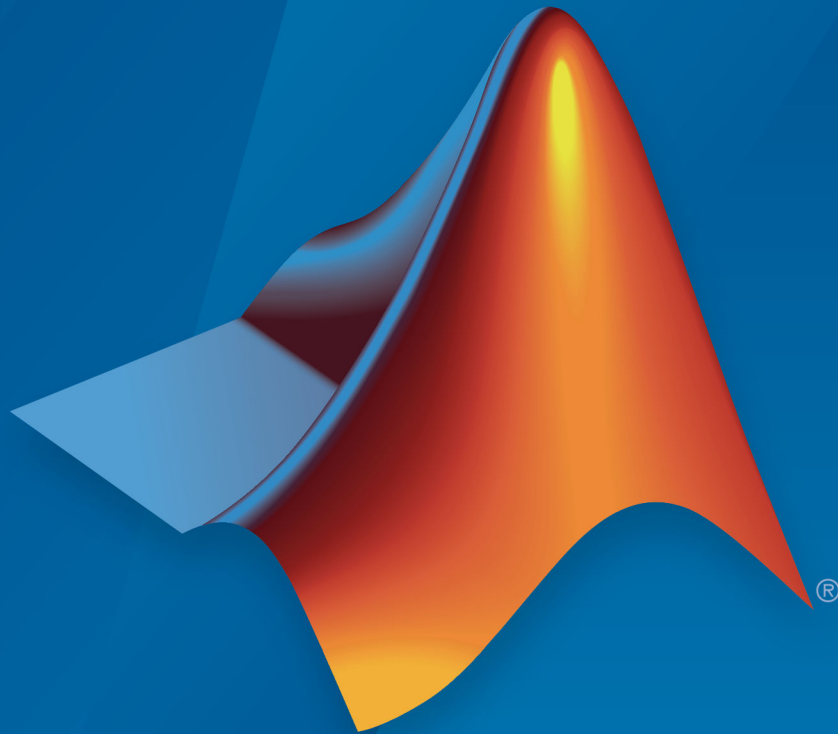**Simscape™**

Getting Started Guide

MATLAB®&SIMULINK®

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

Revision History

September 2007	Online only	New for Version 2.0 (Release 2007b)
March 2008	Online only	Revised for Version 2.1 (Release 2008a)
October 2008	Online only	Revised for Version 3.0 (Release 2008b)
March 2009	Online only	Revised for Version 3.1 (Release 2009a)
September 2009	Online only	Revised for Version 3.2 (Release 2009b)
March 2010	Online only	Revised for Version 3.3 (Release 2010a)
September 2010	Online only	Revised for Version 3.4 (Release 2010b)
April 2011	Online only	Revised for Version 3.5 (Release 2011a)
September 2011	Online only	Revised for Version 3.6 (Release 2011b)
March 2012	Online only	Revised for Version 3.7 (Release 2012a)
September 2012	Online only	Revised for Version 3.8 (Release 2012b)
March 2013	Online only	Revised for Version 3.9 (Release 2013a)
September 2013	Online only	Revised for Version 3.10 (Release 2013b)
March 2014	Online only	Revised for Version 3.11 (Release 2014a)
October 2014	Online only	Revised for Version 3.12 (Release 2014b)
March 2015	Online only	Revised for Version 3.13 (Release 2015a)
September 2015	Online only	Revised for Version 3.14 (Release 2015b)
October 2015	Online only	Rereleased for Version 3.13.1 (Release 2015aSP1)
March 2016	Online only	Revised for Version 4.0 (Release 2016a)
September 2016	Online only	Revised for Version 4.1 (Release 2016b)
March 2017	Online only	Revised for Version 4.2 (Release 2017a)
September 2017	Online only	Revised for Version 4.3 (Release 2017b)
March 2018	Online only	Revised for Version 4.4 (Release 2018a)
September 2018	Online only	Revised for Version 4.5 (Release 2018b)
March 2019	Online only	Revised for Version 4.6 (Release 2019a)
September 2019	Online only	Revised for Version 4.7 (Release 2019b)

Contents

Product Fundamentals

Creating a New Simscape Model

In this section...
"Domain Types and Recommended Blocks" on page 1-2
"Data Logging Settings for New Model" on page 1-4

Domain Types and Recommended Blocks

Simscape models require certain blocks to be present in the model configuration, such as a Solver block, or domain-specific reference blocks. Other blocks, although not required, are highly likely to be needed, such as Simulink-PS Converter and PS-Simulink Converter blocks. An easy way to start a new Simscape model is by using the `ssc_new` function.

When you type `ssc_new` at the MATLAB® Command prompt, the software creates a new model prepopulated with certain blocks.
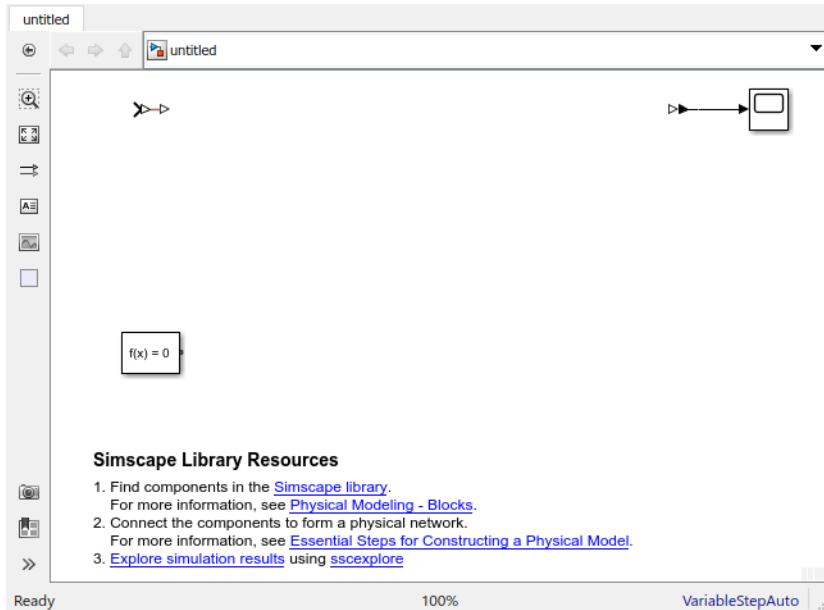
By default, the model name is not specified, the model contains a Solver Configuration block, a Simulink-PS Converter block, and a PS-Simulink Converter block connected to a

Scope block. The default solver is `VariableStepAuto`. The resources section at the bottom of the model window provides links to the relevant block libraries and documentation topics.

You can use the `ssc_new` function arguments to specify the model name and add domain-specific blocks. See the `ssc_new` reference page for details. For example, typing

```
ssc_new('pneumatic_actuator','gas')
```

creates the following model.
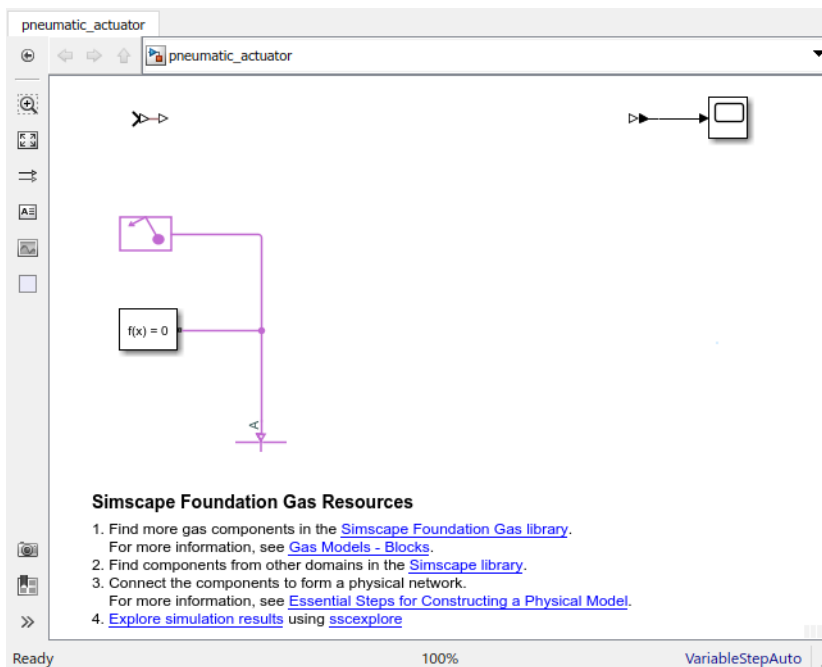
After using `ssc_new`, continue developing your model by copying the blocks, as needed, and adding other blocks from the Simscape libraries.

For electrical models, you can also use the Creating A New Circuit example as a template for a new model. This example creates a new electrical model and opens an Electrical Starter Palette, which contains links to the most often used electrical components. Open the example by typing `ssc_new_elec` in the MATLAB Command Window and use **File > Save As** to save it under the desired model name. Then delete the unwanted components and add new ones from the Electrical Starter Palette and from Simscape libraries.

Data Logging Settings for New Model

Using data logging is a best practice for Simscape models because it provides access to important simulation and analysis tools. Therefore, the ssc_new function automatically turns on data logging for the whole model. It uses the default workspace variable name simlog to store simulation data, and limits the data to the last 10000 points to avoid slowing down simulation. At the same time, it raises MaxDataPoints to 10000, so that Simulink® data logging history matches Simscape data logging history.

When you create a new model using ssc_new, the model has the following data logging configuration:

- **Log simulation data** — All.
- **Log simulation statistics** — Off.
- **Record data in Simulation Data Inspector** — Off.
- **Open viewer after simulation** — Off.
- **Workspace variable name** — simlog.
- **Decimation** — 1.
- **Limit data points** — On.
- **Data history (last N steps)** — 10000.

For information on what these settings mean and how to change them, see "Data Logging Options".

See Also
ssc_new

More About
- "Select Solver Using Auto Solver" (Simulink)
- "Making Optimal Solver Choices for Physical Simulation"
- "Data Logging Options"

Evaluating Performance of a DC Motor

This example shows how to simulate systems that span electrical and mechanical domains. You learn how to model physical components with Simscape blocks, connect them into a realistic model, use Simulink blocks as well, and then simulate and modify a motor model.

The model is based on a Faulhaber Series 0615 DC-Micromotor. The model uses equivalent circuit parameters for the 1.5V motor to verify manufacturer-quoted no-load speed, no-load current, and stall torque. You can use the model to assess motor performance in a given application by adding the requisite mechanical load model.

Explore the Model

1   To open the Permanent Magnet DC Motor example model, type `ssc_dcmotor` in the MATLAB Command Window.
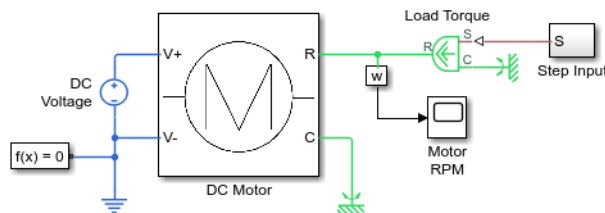
Main Model

The model contains a DC Motor subsystem with two electrical and two mechanical rotational ports.

For improved readability of block diagrams, each Simscape domain uses a distinct default color and line style for the connection lines. In this block diagram, for example, the electrical circuit is indicated by the dark-blue color of the connection lines, while the connection lines between the mechanical rotational ports are light-green. Physical signal lines are brown.

The electrical ports of the motor connect to the electrical circuit, which consists of an Electrical Reference block, representing an electrical ground, and a 1.5 V DC voltage source.

On the mechanical side, a Mechanical Rotational Reference block represents a reference point for the other elements.

The motor load is represented by the block named Load Torque, which is an Ideal Torque Source block. On one side it is connected to a Mechanical Rotational Reference block and on the other side to the motor shaft. The load torque is specified by the Step Input subsystem, connected through a physical signal to the control port S of the Load Torque block.

The Step Input subsystem contains a regular Simulink Step source, which provides the control signal. A Simulink-PS Converter block converts the control signal into a physical signal and applies it to the control port of the Load Torque block through the Connection Port block S.

The diagram also contains a Solver Configuration block, which is required in any Simscape model. It contains parameters relevant to numerical algorithms for Simscape simulations.

2   Double-click the DC Motor subsystem to open it.
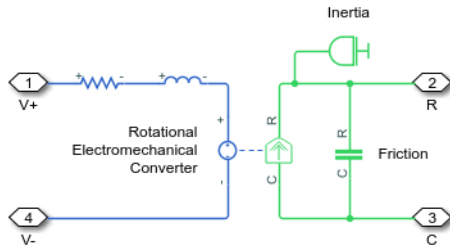
DC Motor Subsystem

The motor consists of an electrical circuit and a mechanical rotational circuit, connected by the Rotational Electromechanical Converter block. The electrical circuit consists of a Resistor block and an Inductor block. It contains two electrical ports, corresponding to the V+ and V- electric terminals of the motor. The mechanical circuit contains a Rotational Friction block, an Inertia block, and two mechanical

rotational ports, C and R, corresponding to the motor case and rotor, respectively. Notice how the C and R ports of the Rotational Friction block and the Rotational Electromechanical Converter block are connected to the C and R ports of the motor, to preserve the correct direction of variables in the physical network.

Run the Model

1   Double-click the Motor RPM scope to open it. During simulation, this window displays the shaft speed as a function of time.

2
Click ▶ to start the simulation. The Simscape solver evaluates the model, calculates the initial conditions, and runs the simulation. This process might take a few seconds. The message in the bottom-left corner of the model window provides the status.

3   Examine the simulation results in the Motor RPM scope window.
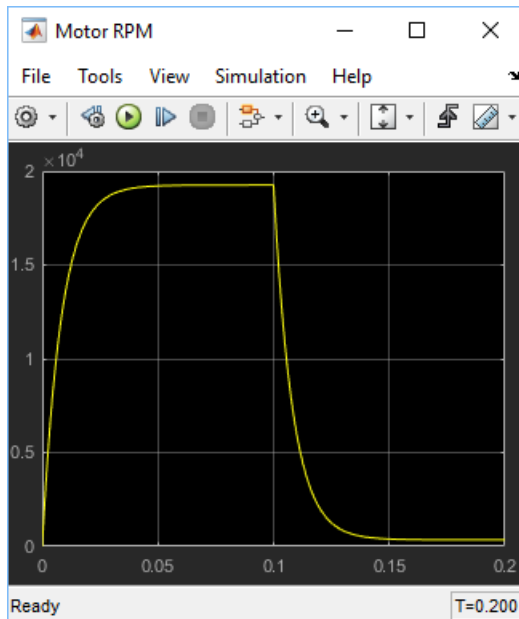
For the first 0.1 seconds, the motor has no external load, and the speed builds up to the no-load value. Then at 0.1 seconds, the stall torque is applied as a load to the motor shaft. Zooming in on the Motor RPM scope shows that the model matches the manufacturer parameters for no-load speed and stall torque.

1-7

4   The example model also shows how you can use MATLAB code to analyze the
    simulation results. To plot the current and load torque, click the `Plot` hyperlink in
    the model annotation. The current shown in the figure matches the manufacturer
    parameters for no-load current.
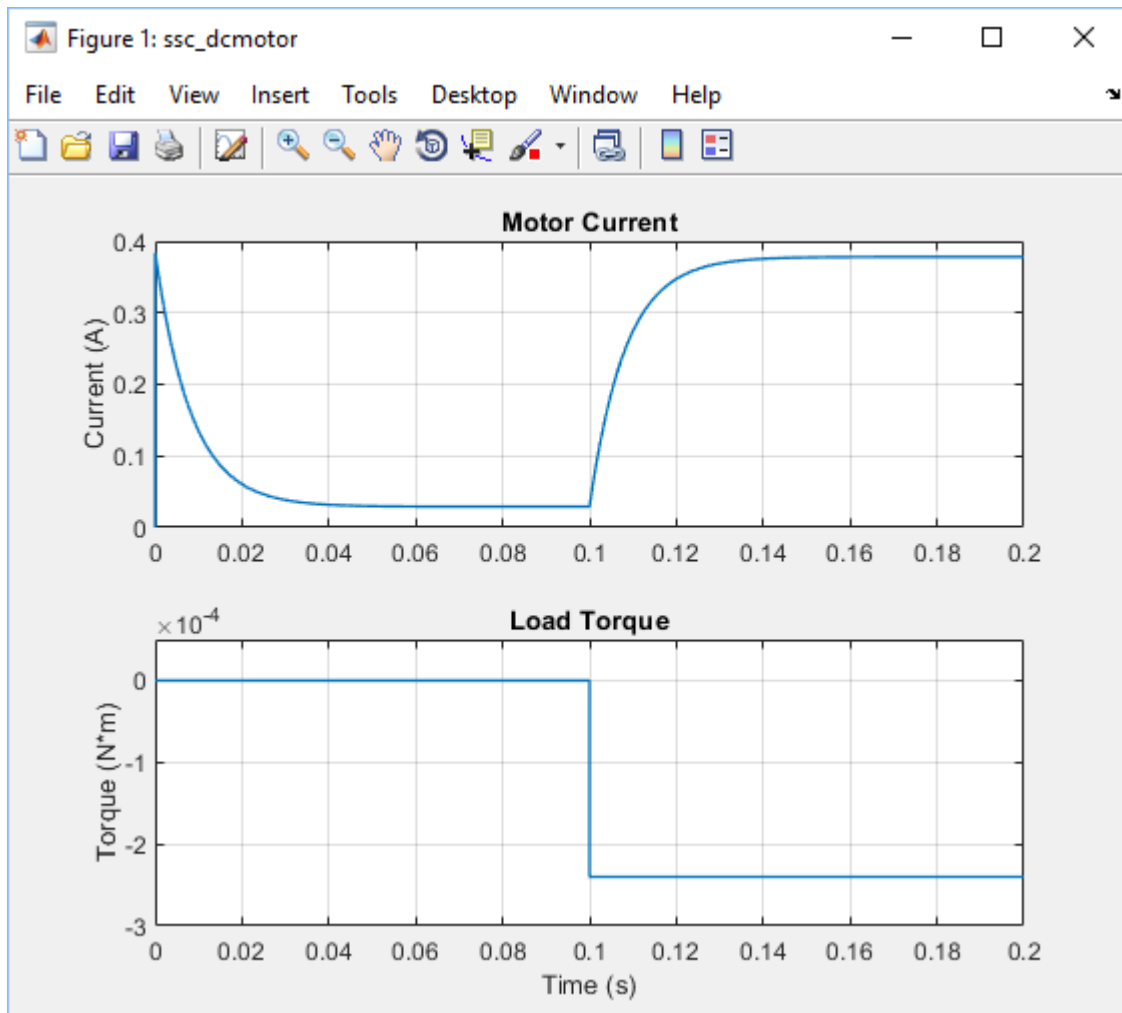
Change the Supply Voltage

Reduce the supply voltage to 1.25 volts (to simulate the battery running down) and vary the load torque to find the maximum torque at this reduced voltage.

1   Double-click the DC Voltage Source block. Set **Constant voltage** to 1.25 V.
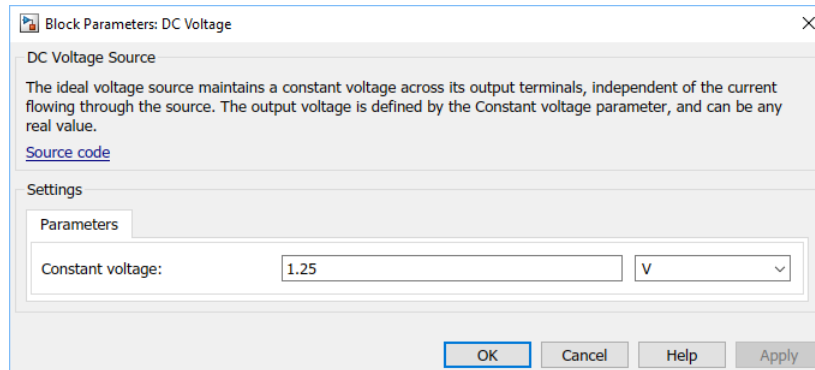
2   Run the simulation. Note the effect of reduced voltage on the no-load speed.
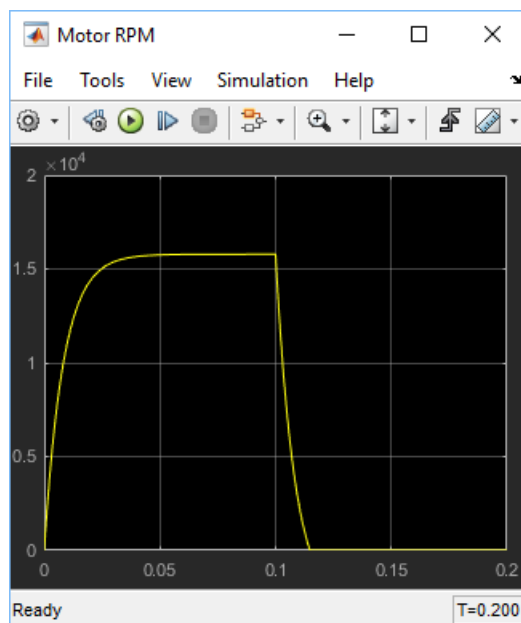
3 Try varying the load torque to find the maximum torque at this reduced voltage. Open the Step Input subsystem and double-click the Step source block. Enter different final values for the input signal and rerun the simulation.

The next illustration shows the simulation results for **Final value** set to -0.2e-3, which corresponds to (1.25/1.5)*0.24mNm, as the magnitude of the torque-speed curve is proportional to voltage for a DC motor.
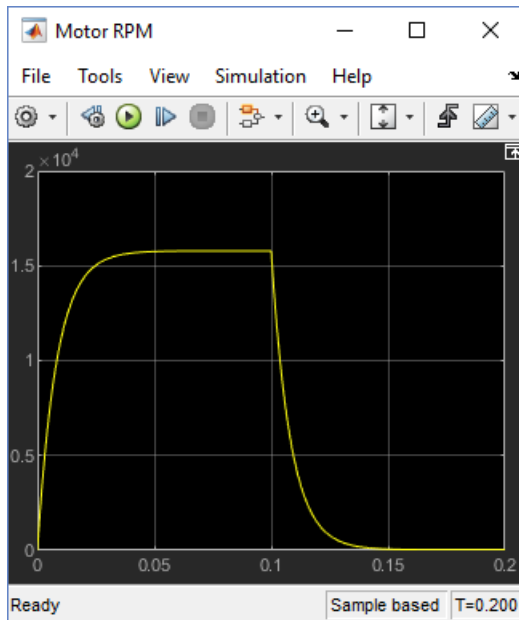
Change the Motor Load

Replace the torque source with a simple mechanical load, for example, a fan, for which the torque is defined by alpha*speed^2, where alpha is -1e-10 Nm/(rad/s)^2.

1 Delete the Step Input subsystem from the model.

2 In the **Simscape** block library, open **Foundation Library > Mechanical > Mechanical Sensors**.

3 Drag the Ideal Rotational Motion Sensor block to the model window.

4 Open **Foundation Library > Physical Signals > Functions**.

5 Drag the PS Product block and the PS Gain block to the model window.

6 Connect the blocks as shown in the following illustration. To rotate a block, select it and press **Ctrl+R**.
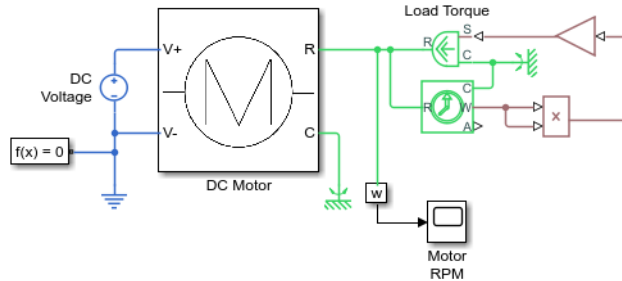
7 Double-click the PS Gain block to open its dialog box. In the first edit box next to the **Gain** parameter, enter the value of `-1e-10`. In the second combo box, with the unit drop-down list, type the unit expression `N*m/(rad/s)^2` and click **OK**.

8 Run the simulation and assess motor performance with the new load.
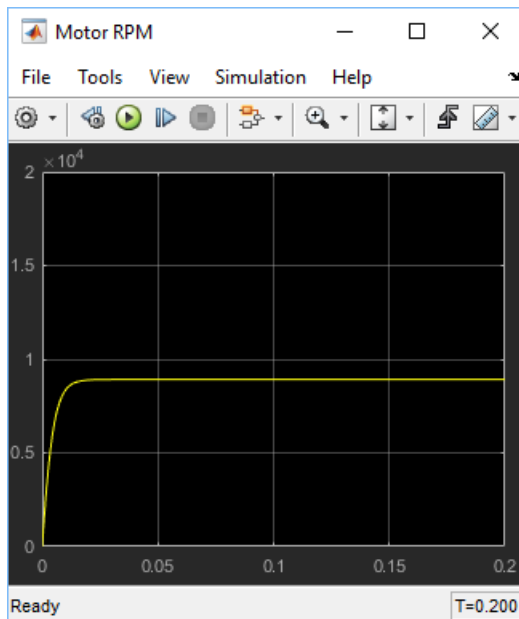
Essential Steps for Constructing a Physical Model

In this section...
"Workflow Overview" on page 1-12
"Step 1: Create New Model Using ssc_new" on page 1-14
"Step 2: Assemble Physical Network" on page 1-15
"Step 3: Adjust Block Parameters and Variable Targets" on page 1-17
"Step 4: Add Sources" on page 1-20
"Step 5: Add Sensors" on page 1-21
"Step 6: Connect to Simulink with Interface Blocks" on page 1-22
"Step 7: Simulate Model" on page 1-27
"Step 8: View Simulation Results" on page 1-28

Workflow Overview

The table lists the essential major steps for building and simulating a physical model, along with related documentation topics that provide background information on each step. Detailed step descriptions follow.

Key Step	Related Topics
"Step 1: Create New Model Using ssc_new" on page 1-14	"Setting Up Solvers for Physical Models" Solver Configuration
"Step 2: Assemble Physical Network" on page 1-15	"Basic Principles of Modeling Physical Networks" "Essential Physical Modeling Techniques" "Modeling Best Practices"
"Step 3: Adjust Block Parameters and Variable Targets" on page 1-17	"Block-Level Variable Initialization" "Variable Viewer"

Key Step	Related Topics
"Step 4: Add Sources" on page 1-20	"Basic Principles of Modeling Physical Networks" "Connecting Simscape Diagrams to Simulink Sources and Scopes"
"Step 5: Add Sensors" on page 1-21	"Basic Principles of Modeling Physical Networks" "Connecting Simscape Diagrams to Simulink Sources and Scopes"
"Step 6: Connect to Simulink with Interface Blocks" on page 1-22	"Connecting Simscape Diagrams to Simulink Sources and Scopes"
"Step 7: Simulate Model" on page 1-27	"Important Concepts and Choices in Physical Simulation" "Troubleshooting Simulation Errors"
"Step 8: View Simulation Results" on page 1-28	"About Simulation Data Logging" "About the Simscape Results Explorer"

The graphic illustrates the key workflow steps as applied to the Mass-Spring-Damper with Controller example model. Step descriptions below include detailed instructions on how to create this model.
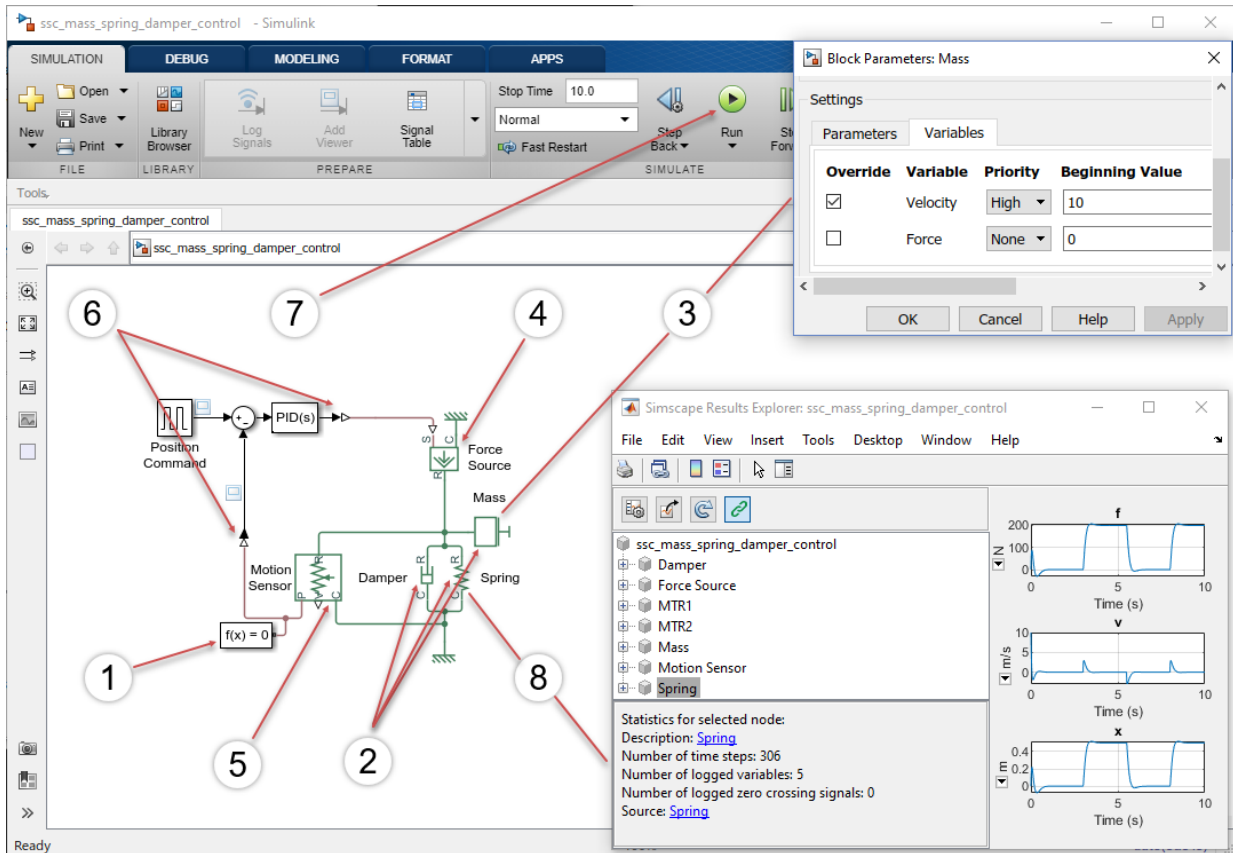
Step 1: Create New Model Using ssc_new

Using `ssc_new` is the best way to start building Simscape models. It helps to ensure that you use recommended default settings for your model. When you use `ssc_new`, it automatically:

- Creates a new Simscape model, with required and commonly used blocks already on the model canvas
- Selects recommended solver and tolerance settings
- Enables data logging for the whole model

Detailed Instructions

In the MATLAB Command Window, type:

```
ssc_new
```
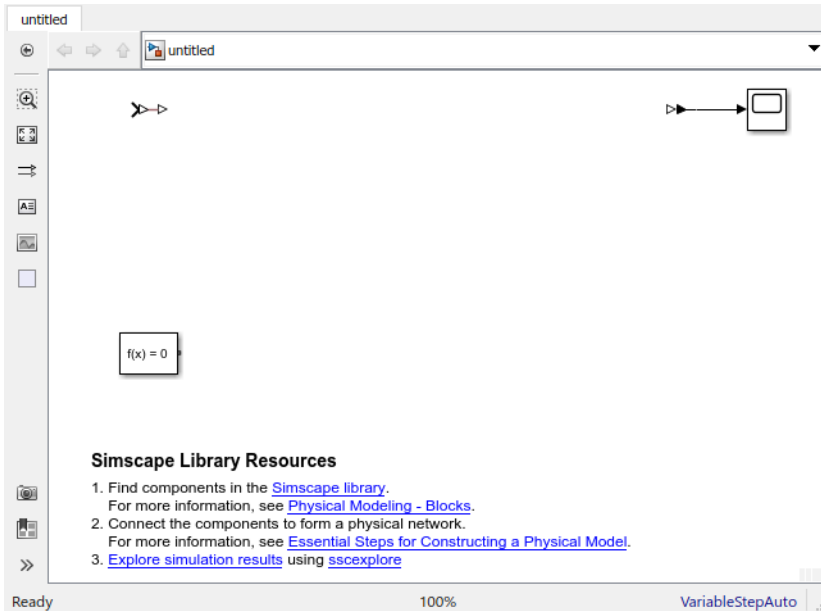

The new untitled model, with the default solver set to `VariableStepAuto`, contains a Solver Configuration block, a Simulink-PS Converter block, and a PS-Simulink Converter block connected to a Scope block.

The resources section at the bottom of the model window provides links to the relevant block libraries and documentation topics.

Step 2: Assemble Physical Network

To model your system, you add blocks from the Simscape libraries to a model, and then connect them into a physical network. The lines connecting the blocks in the schematic you create represent the physical connections that exist between the components in a real system that you are modeling. In other words, Simscape diagrams mimic the physical system layout.
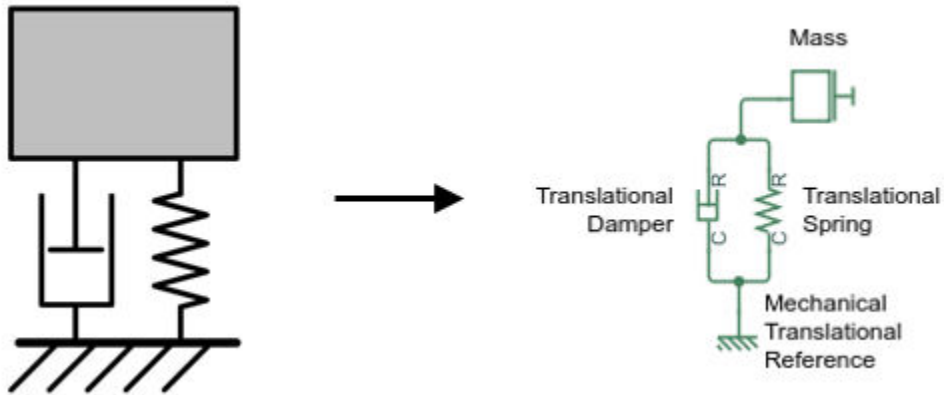
When constructing your network, it is important to include domain-specific reference blocks, such as Electrical Reference, Mechanical Translational Reference, and so on. Depending on domain, these blocks represent connection to ground, frame, or atmosphere. For more information, see "Grounding Rules".

Detailed Instructions

1   Open the Simscape > Foundation Library > Mechanical > Translational Elements library.

2   Drag the Mass, Translational Spring, Translational Damper, and Mechanical Translational Reference blocks into the model window.

3   Connect the Translational Spring, Translational Damper, and Mass blocks to the Mechanical Translational Reference block as shown in the next illustration. To rotate a block, select it and press **Ctrl+R**.

   To adjust visual presentation of the model, shorten the names of the Spring and Damper blocks, as shown. To have the name of the Mass block appear along with Spring and Damper, right-click the Mass block and, from the context menu, select **Format > Show Block Name > On**.
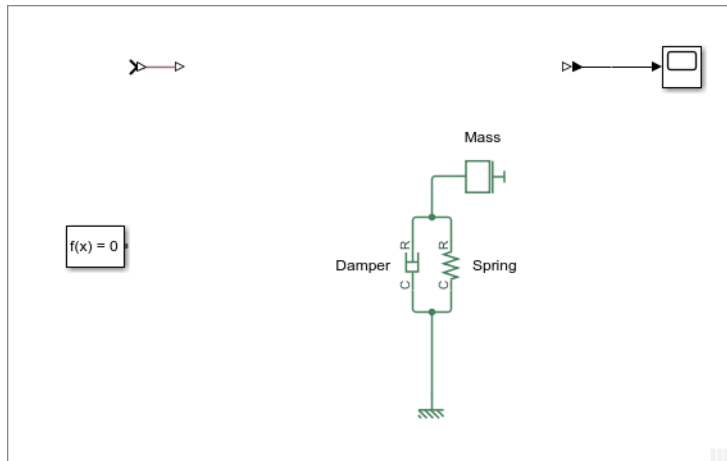
Step 3: Adjust Block Parameters and Variable Targets

Simscape blocks represent generic components that have default initial values for block parameters and variables. You can adjust these values to suit your application or match the manufacturer data sheet.

Detailed Instructions

In this model, you will adjust parameters that control the mass, the spring stiffness, and the damping coefficient. You will also specify the initial velocity of the mass by adjusting the respective variable target.

1 Double-click the Spring block. Set its **Spring rate** to `400 N/m`.
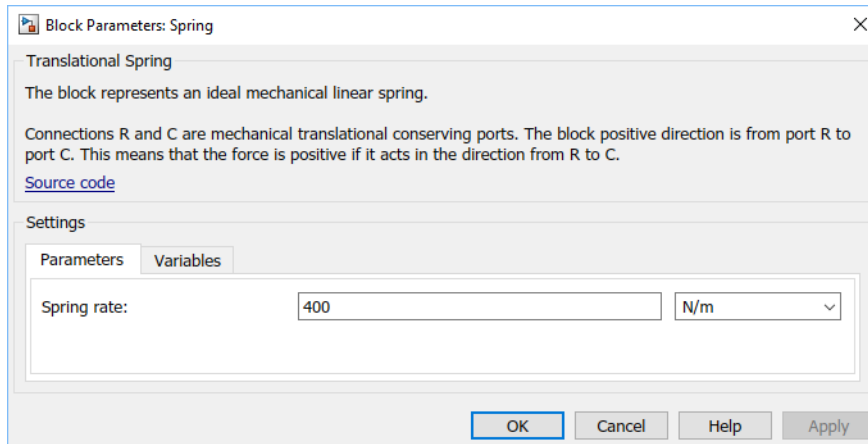
2 Double-click the Damper block. Set its **Damping coefficient** to `100 N/(m/s)`.
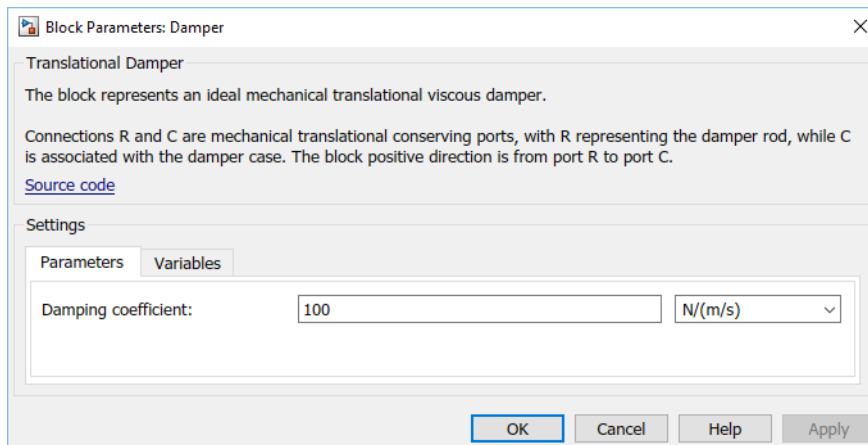
3 Double-click the Mass block. Set its **Mass** to `3.6` kg.
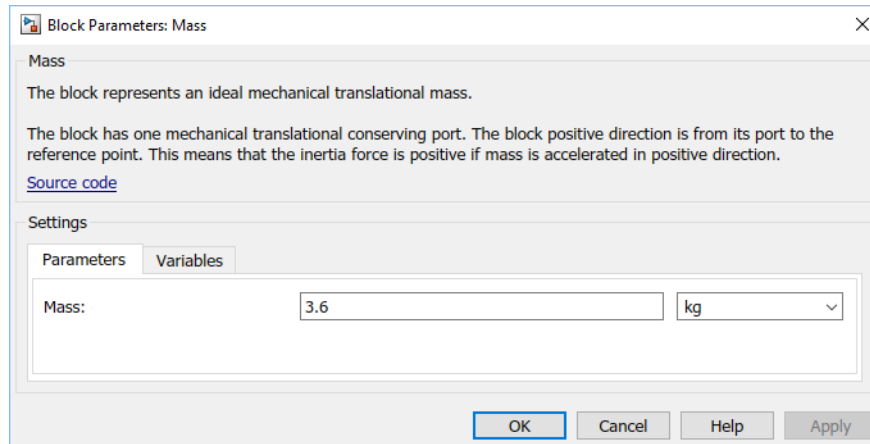
4 To specify the desired initial value for mass velocity, click the **Variables** tab. The **Velocity** variable already has its **Priority** specified as `High`. In the **Beginning Value** field, type `10`, then click **Apply**.
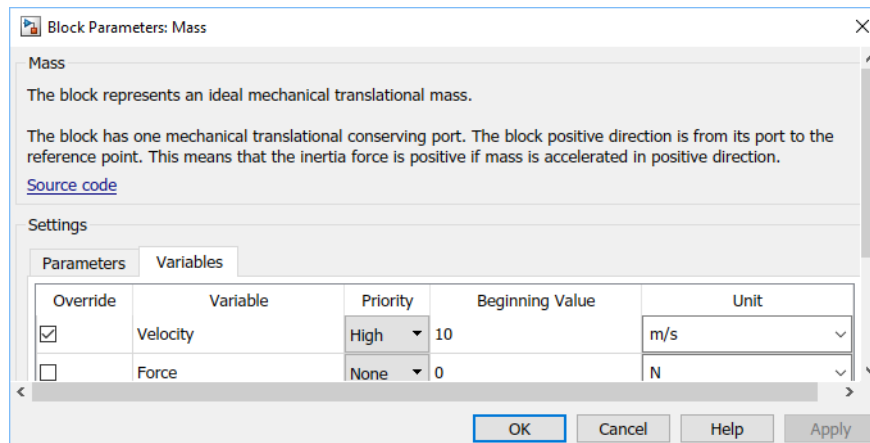
Note that the **Override** check box next to the **Velocity** variable is now selected, which means that you have overridden the default variable initialization value. This variable has high priority, therefore, the solver will try to exactly satisfy this beginning value when it computes the initial conditions to determine the simulation starting point.

Step 4: Add Sources

You can drive Simscape models using input signals. This technique allows you to represent physical effects, such as forces, voltages, or pressures, that act on your system. You can also specify other quantities that flow through your system, such as current, mass flow rate, and heat flux. You add signal input connections to your physical network by using Simscape source blocks.

Detailed Instructions

To add the representation of the force acting on the mass, use the Ideal Force Source block.

1   Open the Simscape > Foundation Library > Mechanical > Mechanical Sources library.

2   Add the Ideal Force Source block to your diagram. Shorten the block name, as shown.

3   To reflect the correct direction of the force shown in the original schematic, flip the block orientation. With the Force Source block selected, on the **Format** tab of the Simulink Toolstrip, under **Arrange**, click **Flip up-down**.

4   Copy the Mechanical Translational Reference block by right-clicking it and dragging to a new location. Then flip the orientation of the new block using the same technique as with the Force Source block.

5   Connect port C of the Force Source block to this second Mechanical Translational Reference block and port R to the Mass block, as shown below.

    The input signal for the force profile will be supplied through port S, after you connect the physical network to a Simulink source (see "Step 6: Connect to Simulink with Interface Blocks" on page 1-22). A positive signal at port S will specify a force that acts from port C to port R.
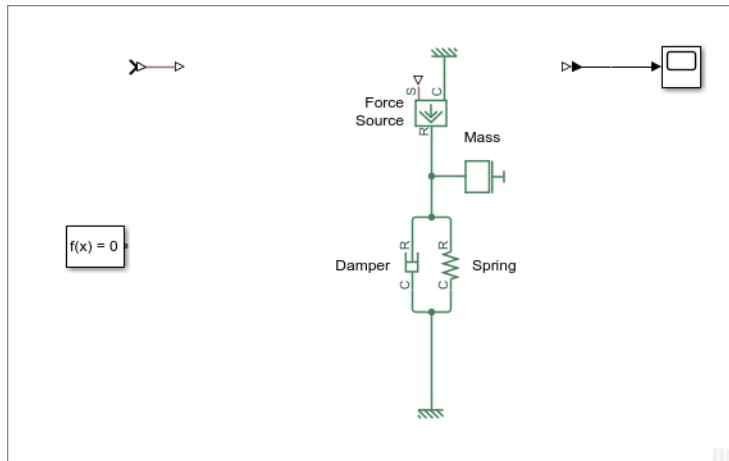
Step 5: Add Sensors

You can measure quantities from your physical network and use them in other locations in your model. Some common uses of those quantities include feedback for a control algorithm, modeling physical components whose behavior depends on other physical quantities (such as temperature-dependent resistor), or simply viewing the results during simulation.

You measure quantities using sensor blocks, connected in series or in parallel depending on the measured value. To measure a quantity defined by a Through variable (such as current, flow rate, force), connect the sensor in series. To measure a quantity defined by an Across variable (such as voltage, pressure, velocity), connect the sensor in parallel. For more information on Through and Across variables, see "Variable Types".

Detailed Instructions

To measure spring deformation, connect an Ideal Translational Motion Sensor block in parallel with the spring.

1 Open the Simscape > Foundation Library > Mechanical > Mechanical Sensors library.

2 Add the Ideal Translational Motion Sensor block to your diagram.

3 To rotate the block, select it and press **Ctrl+R**.

4   Connect the block as shown in the next illustration. Shorten the block name, as
    shown.
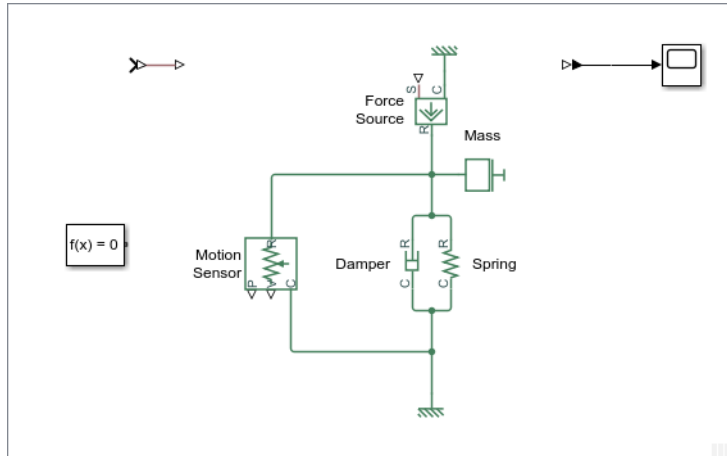
Step 6: Connect to Simulink with Interface Blocks

Equations in a Simscape network are solved simultaneously, while Simulink blocks are
evaluated sequentially. Interface blocks, such as Simulink-PS Converter and PS-Simulink
Converter, handle the boundary between these two modeling conventions. You need
interface blocks when Simulink signals specify quantities in a Simscape network, or when
passing Simscape quantities to Simulink for control design or other purposes. Every time
you connect a Simulink block to a Simscape physical network, you have to use an
appropriate converter block.

Detailed Instructions

You are now going to connect your physical network to a controller built out of regular
Simulink blocks. First, prepare the physical network to be connected to Simulink signals:

1   Delete the Scope block.

    While you can use scopes to view simulation results, they add clutter to the block
    diagram. A more efficient way to view and analyze simulation results is to use the
    Simscape Results Explorer, as described in "Step 8: View Simulation Results" on page
    1-28.

2 Connect the physical signal output port of the Simulink-PS Converter block to port S of the Force Source block.

3 Connect the output port P of the Motion Sensor block to the physical signal input port of the PS-Simulink Converter block.

4 Connect the Solver Configuration block to the circuit and hide the converter block names. Your diagram now looks like this.
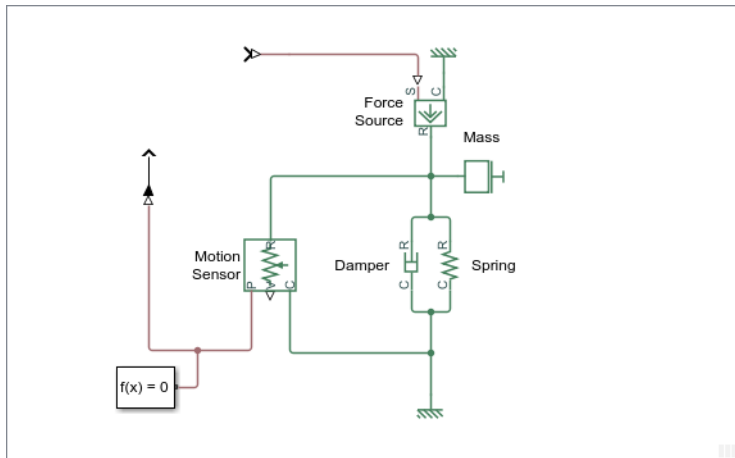
Then, build and connect the controller:

1 Open the Simulink > Sources library and drag the Pulse Generator block into the model. Change the block name to Position Command. Set the block parameters as shown.

Block Parameters: Position Command ✕

Pulse Generator

Output pulses:

```
if (t >= PhaseDelay) && Pulse is on
  Y(t) = Amplitude
else
  Y(t) = 0
end
```

Pulse type determines the computational technique used.

Time-based is recommended for use with a variable step solver, while Sample-based is recommended for use with a fixed step solver or within a discrete portion of a model using a variable step solver.

Parameters

Pulse type: Time based ▼

Time (t): Use simulation time ▼

Amplitude:

0.5

Period (secs):

5

Pulse Width (% of period):

50

Phase delay (secs):

3

☑ Interpret vector parameters as 1-D

OK   Cancel   Help   Apply

2 Open the Simulink > Math Operations library and drag the Sum block into the model. In the **List of signs**, replace the second plus sign with a minus sign, as shown.
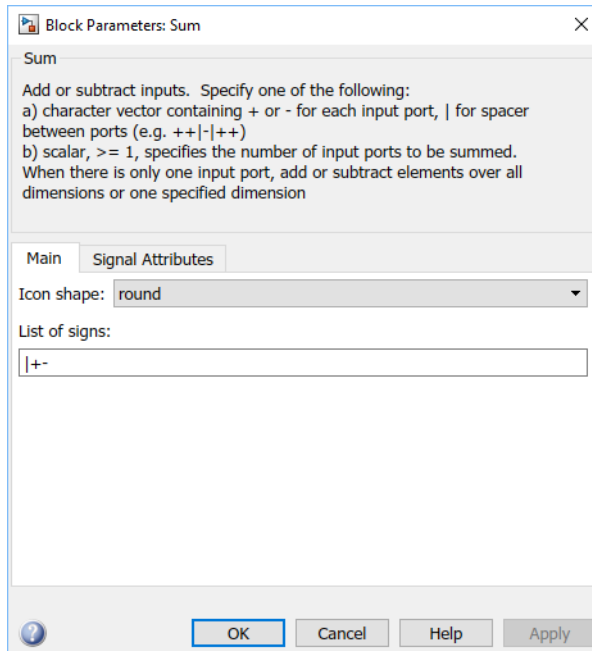
3   Open the Simulink > Continuous library and drag the PID Controller block into the model. Set the **Proportional (P)**, **Integral (I)**, and **Derivative (D)** parameter values as shown.

Block Parameters: PID Controller   ✕

PID 1dof (mask) (link)
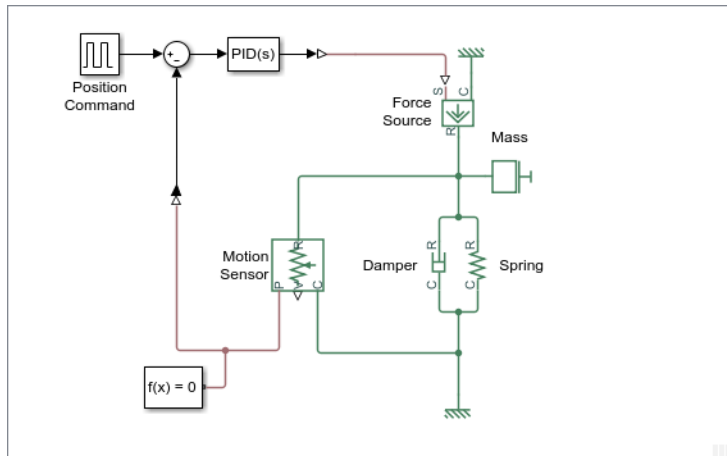
This block implements continuous- and discrete-time PID control algorithms and includes advanced features such as anti-windup, external reset, and signal tracking. You can tune the PID gains automatically using the 'Tune...' button (requires Simulink Control Design).

Controller: PID   ▼   Form: Parallel   ▼

Time domain:
- ⦿ Continuous-time
- ◯ Discrete-time

Discrete-time settings

Sample time (-1 for inherited): -1

▼ Compensator formula

$$P + I\frac{1}{s} + D\frac{N}{1+N\frac{1}{s}}$$

| Main | Initialization | Output Saturation | Data Types | State Attributes |

Controller parameters

Source: internal   ▼

Proportional (P): 700

Integral (I): 4000

Derivative (D): 15

☑ Use filtered derivative

Filter coefficient (N): 100

Automated tuning

Select tuning method: Transfer Function Based (PID Tuner App)   ▼   Tune...

☑ Enable zero-crossing detection

OK   Cancel   Help   Apply

4 Connect the blocks as shown in the following diagram.

The control Simulink signal goes to the input port of the Simulink-PS Converter block, where it is converted into a physical signal driving the force profile of the Ideal Force Source block.

The output port P of the Ideal Translational Motion Sensor block, which measures the spring deformation, connects to the PS-Simulink Converter block. This block converts the physical signal into a feedback Simulink signal for the controller.

To compare the input and feedback signals, connect them to a signal viewer:

1 Right-click the Simulink signal that goes from the Pulse Generator block to the Sum block. From the context menu, select **Create & Connect Viewer** > **Simulink** > **Scope**.
2 Then, right-click the Simulink signal that goes from the PS-Simulink Converter block to the Sum block. From the context menu, select **Connect To Viewer** > **Scope**.

Step 7: Simulate Model

You run simulations by clicking the Run button in the Simulink Toolstrip (at the top of the model window) or in a Scope Viewer toolbar. The Simscape solver evaluates the model, calculates the initial conditions, and runs the simulation. For a detailed description of this process, see "How Simscape Simulation Works". The message in the bottom-left corner of the model window provides the status update.

Detailed Instructions

To run the simulation, click (▶) in the Simulink Toolstrip.

The Scope Viewer displays overlaid plots of the input and feedback signals.
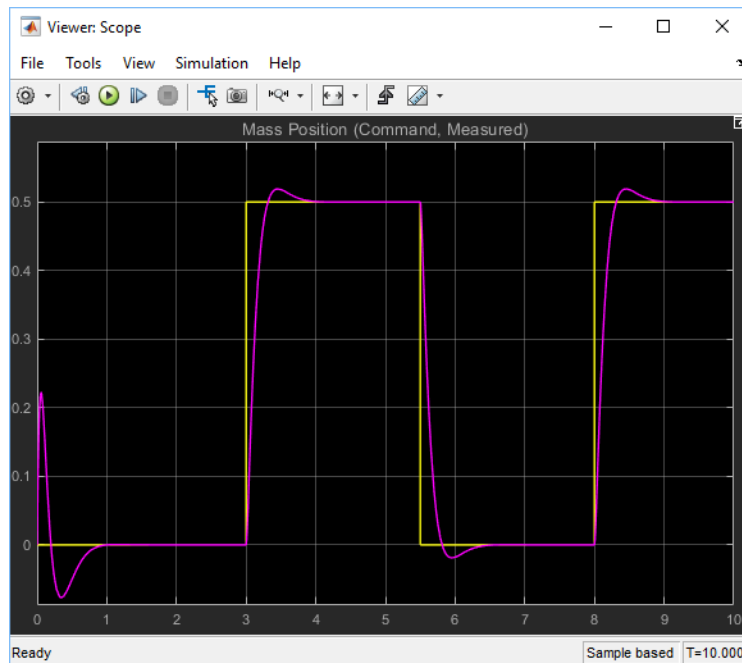

Step 8: View Simulation Results

The Simscape Results Explorer lets you view and analyze simulation data by using the data logging functionality. For example, you can compare two simulation runs to analyze how changing the mass affects the spring deformation.

Detailed Instructions

1   Right-click the Spring block.

2   From the context menu, select **Simscape** > **View simulation data** > **simlog**.

> **Note** `simlog` is the default name of the log variable. If you change the variable name using the **Workspace variable name** model configuration parameter, the context menu lists that name as the log variable associated with the block.

The Simscape Results Explorer window opens, with the node corresponding to the Spring block highlighted in the left pane. The right pane displays the simulation data plots for the three variables associated with the block.
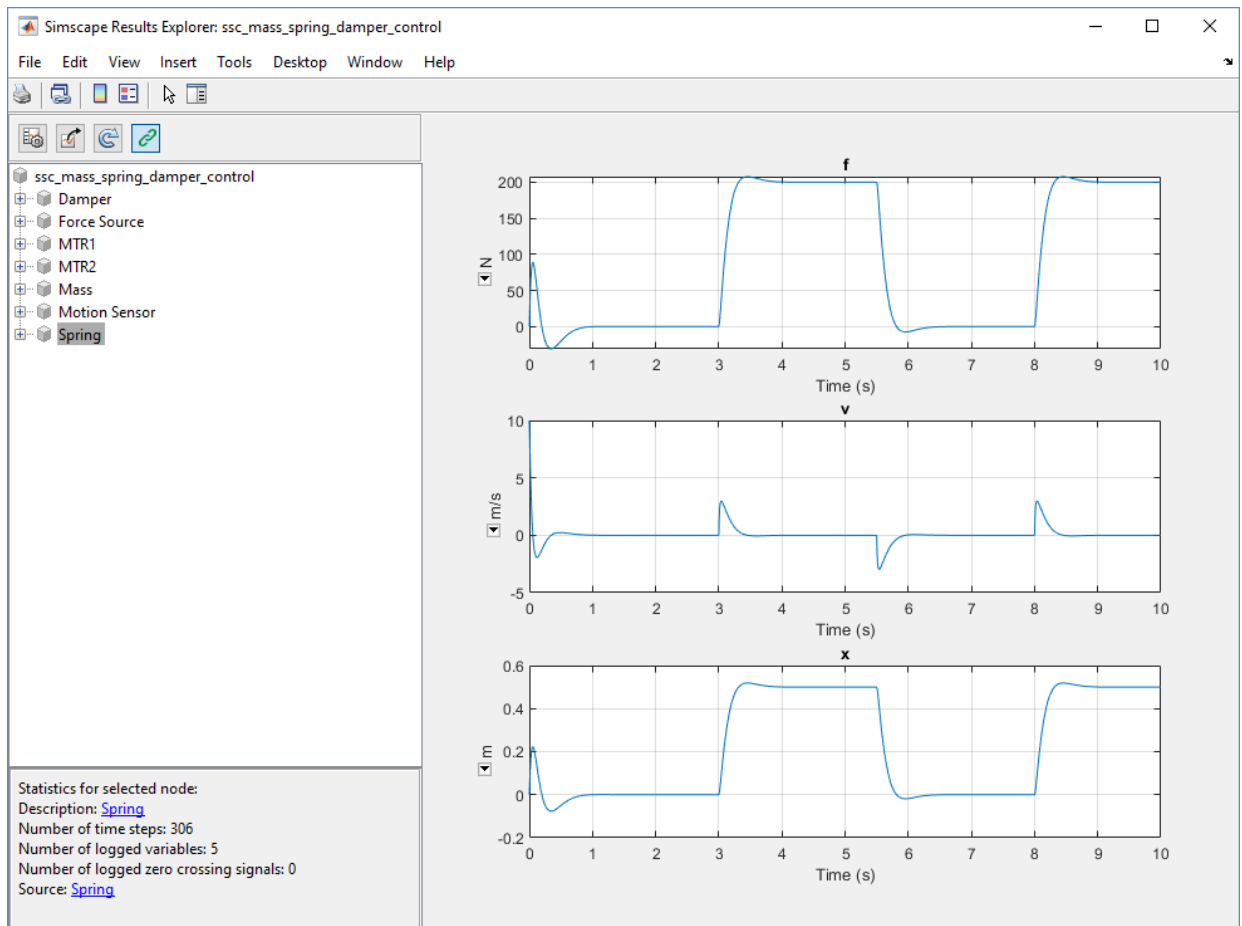
3   Expand the `Spring` node and select node `x`. The right pane displays the spring deformation data plotted over time.
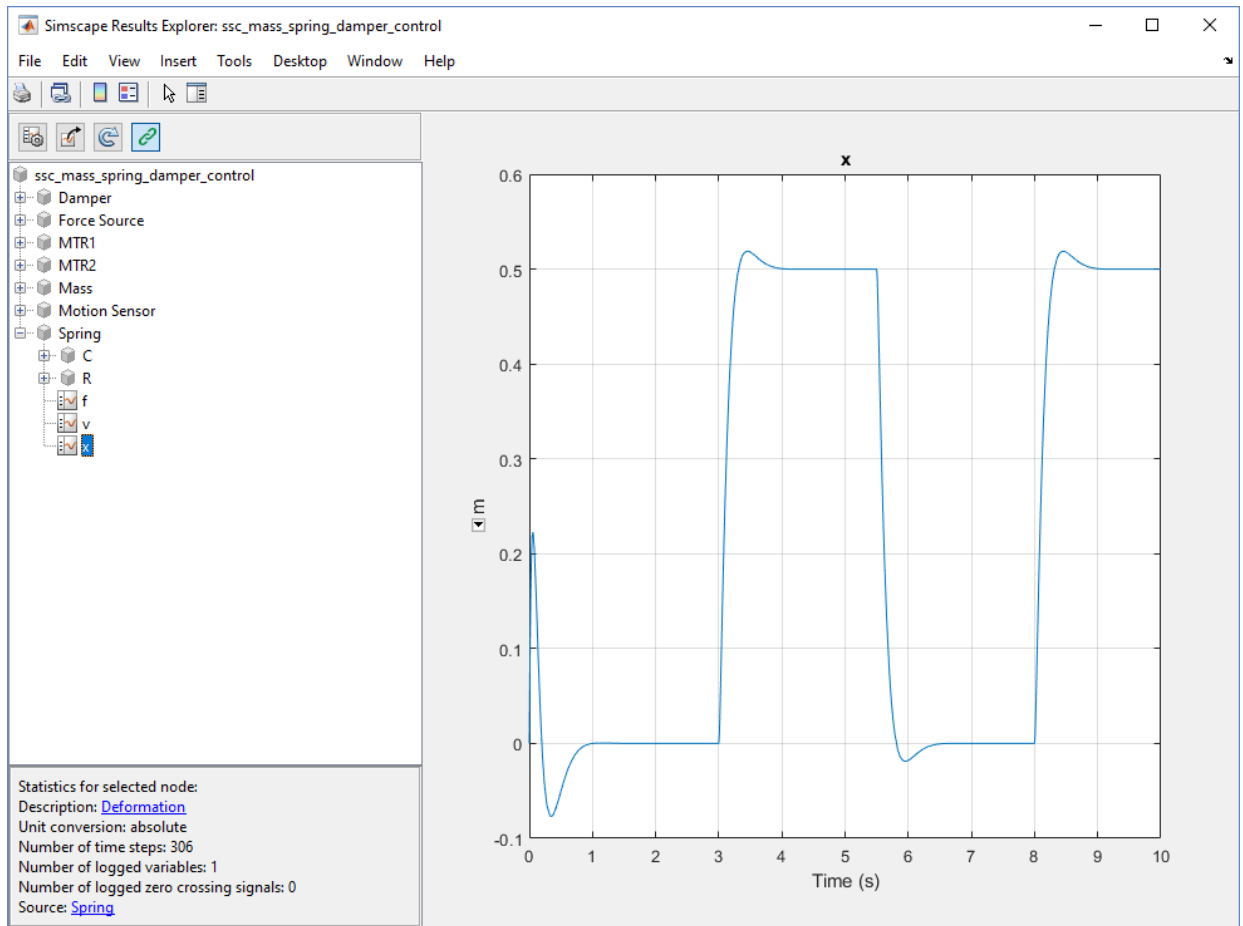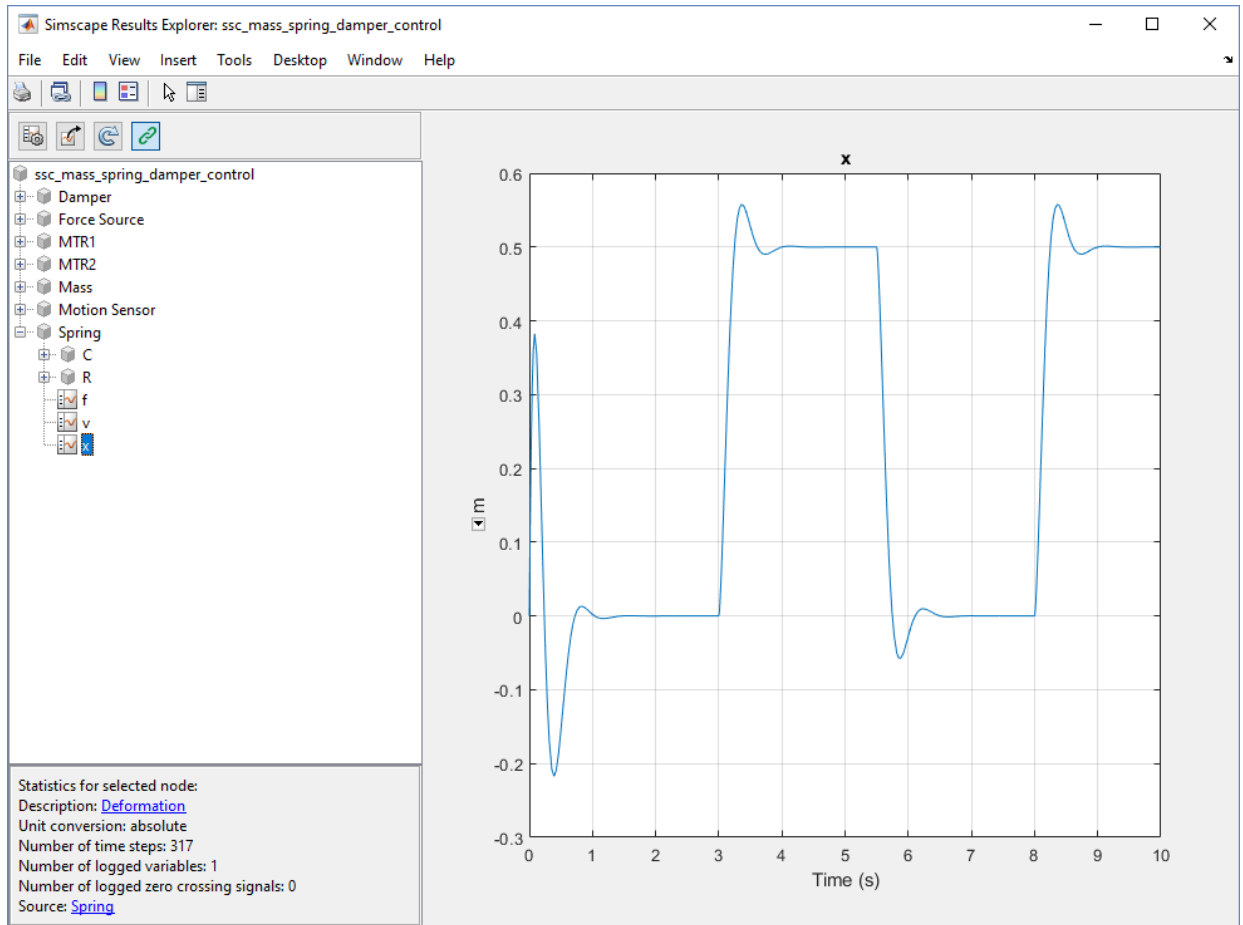
4   Double-click the Mass block. Set its **Mass** to 7.2 kg.

5   Rerun the simulation.

6

    To reload the logged data, click the button located in the toolbar of Simscape Results Explorer window. In the pop-up window displaying the variable name, click **OK**.

Notes

- To have the data reload automatically after each simulation run, in the Configuration Parameters dialog box, in the **Simscape** pane, select the **Open viewer after simulation** check box.

- To explore the simulation results side by side, in two separate Simscape Results Explorer windows, use the button after the first simulation run. For more information, see "About the Simscape Results Explorer".