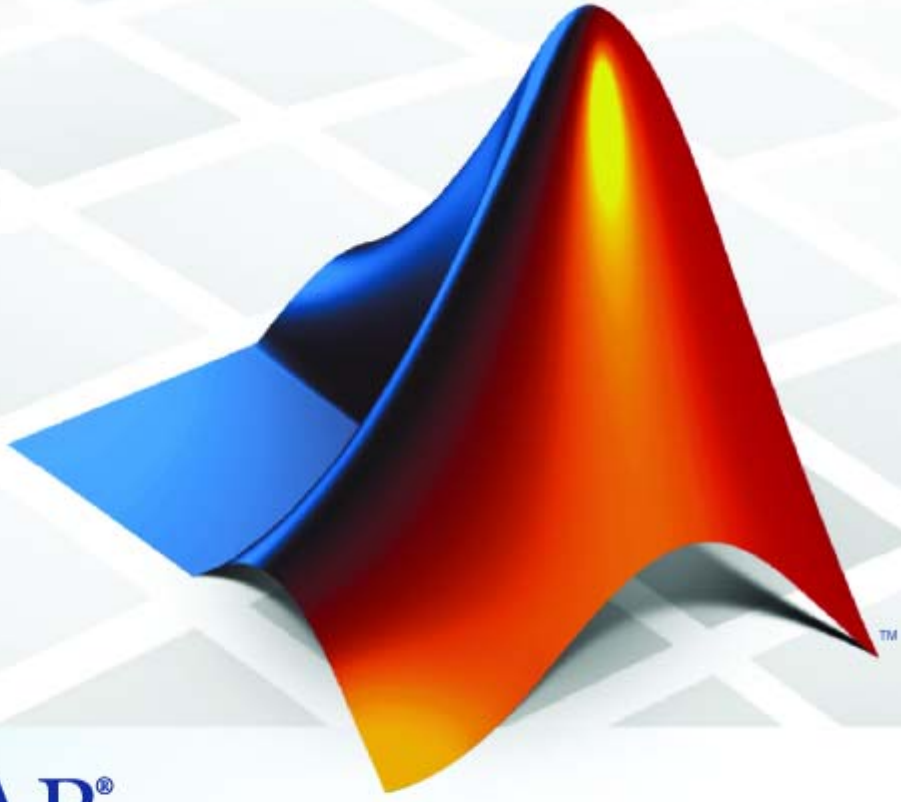
Simscape™ 3

Getting Started Guide



MATLAB®
&SIMULINK®

The MathWorks™
Accelerating the pace of engineering and science

How to Contact The MathWorks

`www.mathworks.com`	Web
`comp.soft-sys.matlab`	Newsgroup
`www.mathworks.com/contact_TS.html`	Technical Support

`suggest@mathworks.com`	Product enhancement suggestions
`bugs@mathworks.com`	Bug reports
`doc@mathworks.com`	Documentation error reports
`service@mathworks.com`	Order status, license renewals, passcodes
`info@mathworks.com`	Sales, pricing, and general information

508-647-7000 (Phone)

508-647-7001 (Fax)

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

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

Simscape™ Getting Started Guide

© COPYRIGHT 2007–2010 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

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

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)

Contents

1 Introduction to Simscape Software

Product Overview 1-2
Product Definition 1-2
Simscape Block Libraries 1-2
Physical Modeling Product Family 1-4

Prerequisites 1-6
Background Knowledge 1-6
Required Products 1-6

Opening Simscape Libraries 1-7

Creating a New Simscape Model 1-8

Working with a Simscape Demo Model 1-11
About This Tutorial 1-11
Exploring the Model 1-12
Running the Model 1-14
Modifying the Model 1-15

Learning More 1-21
Next Steps 1-21
Product Help 1-21
The MathWorks Online 1-22

1

Introduction to Simscape Software

- "Product Overview" on page 1-2
- "Prerequisites" on page 1-6
- "Opening Simscape Libraries" on page 1-7
- "Creating a New Simscape Model" on page 1-8
- "Working with a Simscape Demo Model" on page 1-11
- "Learning More" on page 1-21

Product Overview

In this section...
"Product Definition" on page 1-2
"Simscape Block Libraries" on page 1-2
"Physical Modeling Product Family" on page 1-4

Product Definition

Simscape™ software extends the Simulink® product line with tools for modeling and simulating multidomain physical systems, such as those with mechanical, hydraulic, pneumatic, thermal, electrical, and electromagnetic components. Unlike other Simulink blocks, which represent mathematical operations or operate on signals, Simscape blocks represent physical components or relationships directly. With Simscape blocks, you build a model of a system just as you would assemble a physical system.

Simscape models employ a Physical Network approach to model building: components (blocks) corresponding to physical elements such as pumps, motors, and op-amps, are joined by lines corresponding to the physical connections that transmit power. This approach lets you describe the physical structure of a system rather than the underlying mathematics. From your model, which closely resembles a schematic, Simscape technology automatically constructs equations that characterize the behavior of the system. These equations are integrated with the rest of the Simulink model.

Simscape software runs within the Simulink environment and interfaces seamlessly with the rest of Simulink product family and with the MATLAB® technical computing environment.

Simscape Block Libraries

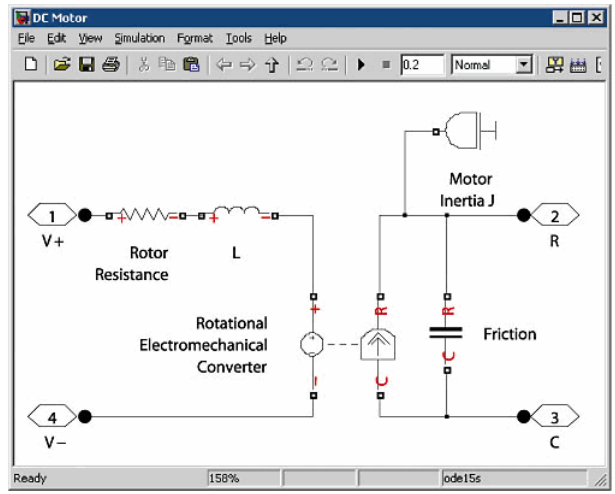
Simscape Foundation libraries contain a comprehensive set of basic elements and building blocks, such as:

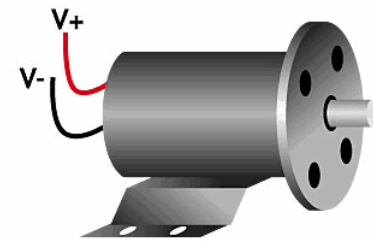
- Mechanical building blocks for representing one-dimensional translational and rotational motion

- Electrical building blocks for representing electrical components and circuits
- Magnetic building blocks that represent electromagnetic components
- Hydraulic building blocks that model fundamental hydraulic effects and can be combined to create more complex hydraulic components
- Pneumatic building blocks that model fundamental pneumatic effects based on the ideal gas law
- Thermal building blocks that model fundamental thermal effects
- Physical Signals block library that lets you perform math operations on physical signals, and graphically enter equations inside the physical network

Using the elements contained in these Foundation libraries, you can create more complex components that span different physical domains. You can then group this assembly of blocks into a subsystem and parameterize it to reuse and share these components.

Shown below is a Simscape model of a DC motor that accounts for friction and includes electrical and mechanical components. You can easily match the model to the motor illustration on the right. The model contains two electrical ports, corresponding to the V+ and V- electric terminals of the motor, and two mechanical rotational ports, C and R, corresponding to the motor case and rotor, respectively. It is built of Simscape foundation blocks. You can use this model to create a masked subsystem of a DC motor, to reuse and share in electric diagrams.





In addition to Foundation libraries, there is also a Simscape Utilities library, which contains utility blocks, such as:

- Solver Configuration block, which contains parameters relevant to numerical algorithms for Simscape simulations. Each Simscape diagram (or each topologically distinct physical network in a diagram) must contain a Solver Configuration block.
- Simulink-PS Converter block and PS-Simulink Converter block, to connect Simscape and Simulink blocks. Use the Simulink-PS Converter block to connect Simulink outports to Physical Signal inports. Use the PS-Simulink Converter block to connect Physical Signal outports to Simulink inports.

For examples of using these blocks in a Simscape model, see the following section, “Working with a Simscape Demo Model” on page 1-11.

Physical Modeling Product Family

Simscape functions and utilities support functionality common to other Simulink products that use physical connections between their blocks. Simscape serves as the platform product for these add-on products of the Physical Modeling family:

- SimHydraulics®, for modeling and simulating hydraulic systems

- SimDriveline™, for modeling and simulating powertrain systems
- SimMechanics™, for modeling and simulating general mechanical systems
- SimElectronics®, for modeling and simulating electromechanical and electronic systems

The Physical Modeling product family also contains SimPowerSystems™, for modeling and simulating electrical power systems. Use these products together to model physical systems in the Simulink environment.

Prerequisites

In this section...
“Background Knowledge” on page 1-6
“Required Products” on page 1-6

Background Knowledge

This Getting Started guide requires that you have some experience with modeling physical systems, as well as with building and running models in the Simulink environment.

Required Products

Simscape software requires these products:

- MATLAB
- Simulink

Opening Simscape Libraries

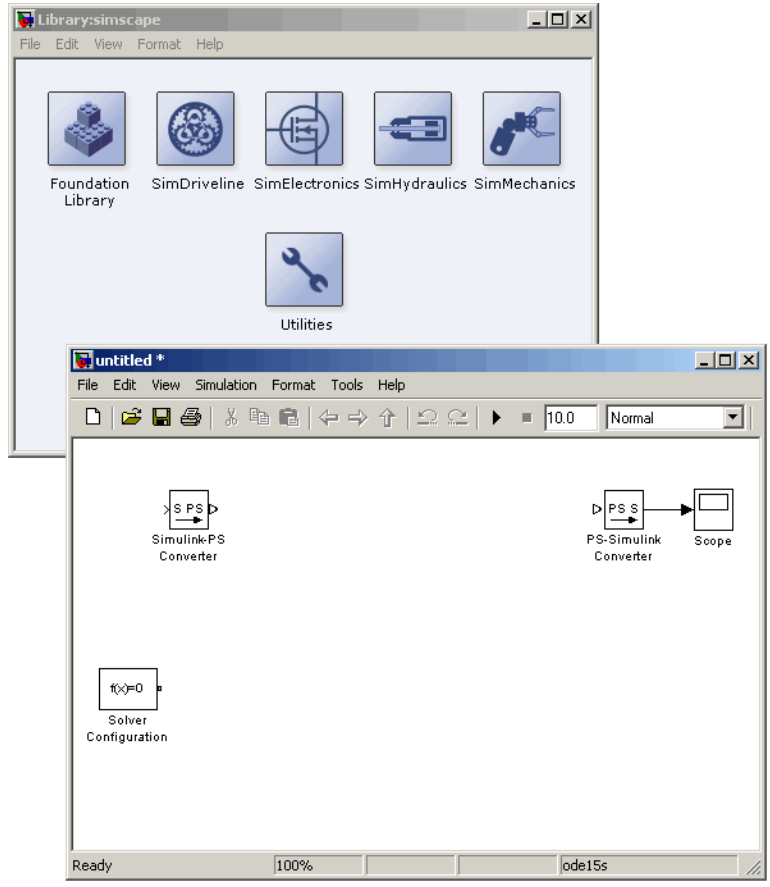
Simscape software is part of your MATLAB installation.

To open Simscape block libraries, type `simulink` in the MATLAB Command Window to display the Simulink Library Browser, and then expand the **Simscape** entry in the contents tree.

Creating a New Simscape Model

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` command.

When you type `ssc_new` at the MATLAB Command prompt, the software opens the main Simscape library and creates a new model prepopulated with certain blocks, as shown in the following illustration.

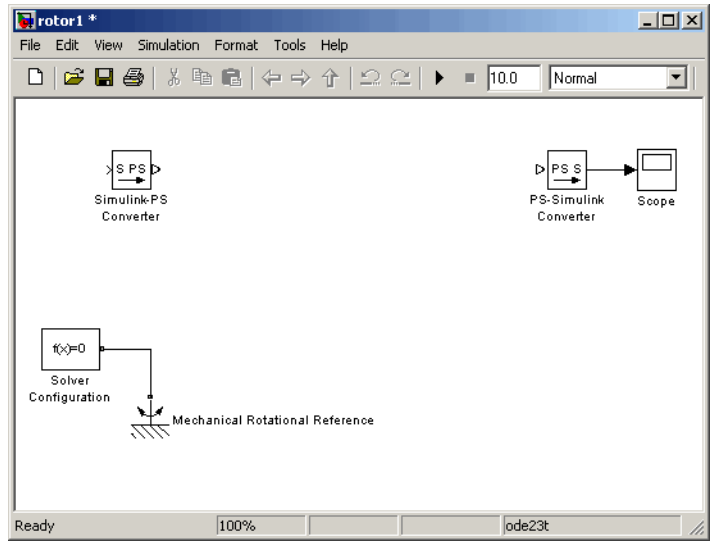


By default, the model name is not specified, the model contains a Solver Configuration block with the default solver set to `ode15s`, a Simulink-PS Converter block, and a PS-Simulink Converter block connected to a Scope block.

You can use the `ssc_new` command arguments to specify the model name, add a domain-specific reference block, and change the default solver. See the `ssc_new` command reference page for details. For example, typing

```
ssc_new('rotor1','rotational','ode23t')
```

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 demo as a template for a new model. This demo also opens an Electrical Starter Palette, which contains links to the most often used electrical components. Open the demo 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.

Working with a Simscape Demo Model

In this section...
"About This Tutorial" on page 1-11
"Exploring the Model" on page 1-12
"Running the Model" on page 1-14
"Modifying the Model" on page 1-15

About This Tutorial

This tutorial provides you with hands-on experience of using Simscape software. You will open a demo model, examine how it is built, and run the simulation. You will then perform simple modifications of the demo and view their effect on the simulation results.

The Permanent Magnet DC Motor demo shows how Simscape models can simulate systems that span electrical and mechanical domains. In this demo, 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. For the 1.5V variant, equivalent circuit parameters are given as:

```
Rotor resistance R=3.9 Ohms
Rotor inductance L=12mH
Back emf constant K=0.072mV/rpm
Rotor inertia J=0.01gcm^2
Friction torque Mr=0.02mNm
```

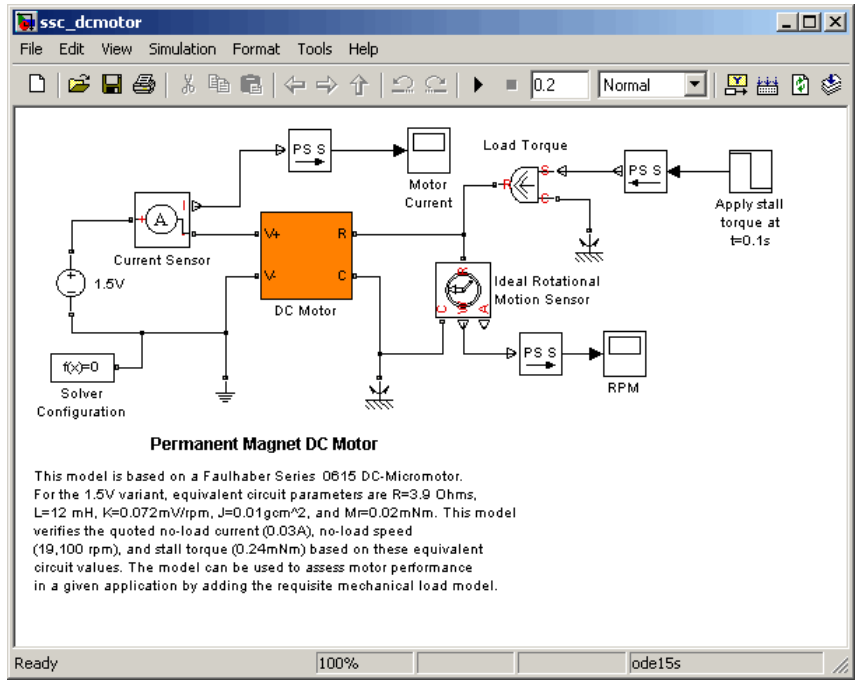
The model uses these parameters to verify manufacturer-quoted no-load speed, no-load current, and stall torque, which are:

```
No-load speed = 19,100rpm
No-load current = 0.03A
Stall torque = 0.24mNm
```

You can use the model to assess motor performance in a given application by adding the requisite mechanical load model.

Exploring the Model

1 To open the Permanent Magnet DC Motor demo, type `ssc_dcmotor` in the MATLAB Command Window.



Main Model Window

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

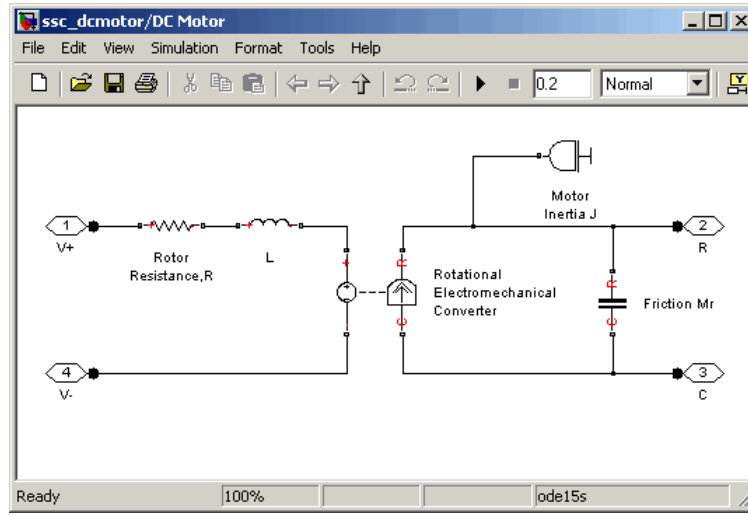
The electrical ports connect to the electrical circuit, which consists of an Electrical Reference block, representing an electrical ground, a 1.5 V DC voltage source, and a current sensor. The current sensor connects, through a PS-Simulink Converter block, to a Simulink scope labeled Motor Current.

On the mechanical side, a Mechanical Rotational Reference block represents a reference point for the other elements. An ideal rotational motion sensor connects, through a PS-Simulink Converter block, to a Simulink scope labeled RPM.

The motor load is represented by an Ideal Torque Source block, which on one side connects to a Mechanical Rotational Reference block, and on the other side to the motor shaft. A regular Simulink Step source 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 Ideal Torque Source block.

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 circuit, connected by the Rotational Electromechanical Converter block. The electrical circuit consists of a Rotor Resistance block and an Inductance block L. It contains two electrical ports, corresponding to the V+ and

V- electric terminals of the motor. The mechanical circuit contains a Rotational Friction block, Motor Inertia J, 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 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.

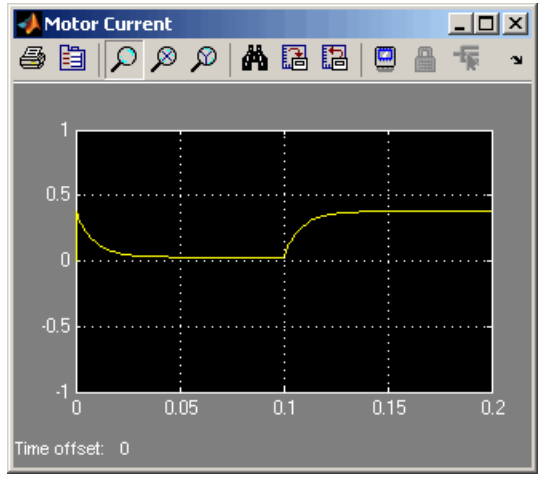
Running the Model

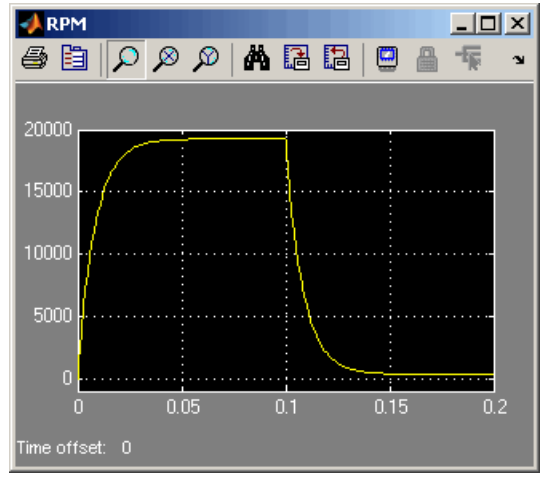
After you have reviewed the block diagram of the demo model, you can run the simulation.

1. Double-click the Motor Current and RPM scopes to open them. During simulation, these windows display the motor current and shaft speed, respectively, as functions of time.

2. In the toolbar of the model window, 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 Current and RPM scope windows.





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 RPM and Motor Current scopes shows that the model matches the manufacturer parameters for no-load speed, no-load current, and stall torque.

Modifying the Model

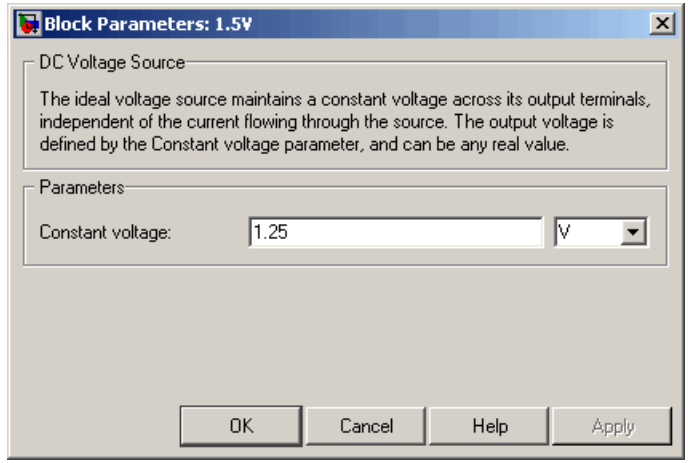
After running the initial simulation, you can experiment with adjusting various inputs and block parameters. Try the following adjustments:

1. Change the supply voltage.
2. Change the motor load.

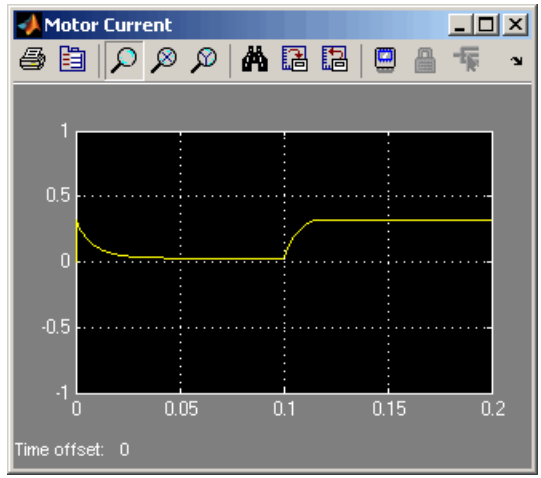
Changing the Supply Voltage

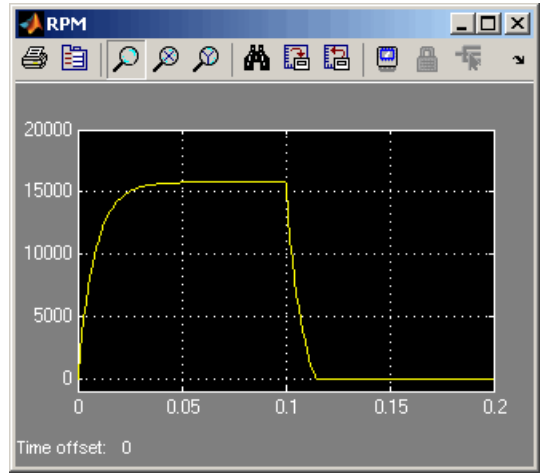
Reduce the supply voltage to 1.25 volts (to simulate the battery running down) and vary the load torque to find the maximum torque at this reduced voltage.

1. Double-click the 1.5V DC Voltage Source block. Set **Constant voltage** to `1.25 V`.



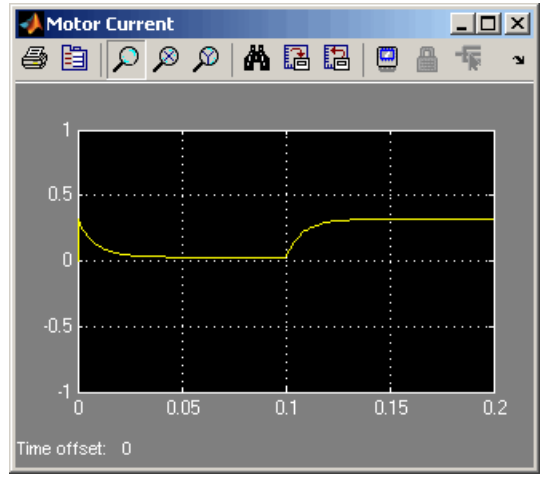
2 Run the simulation. Note the effect of reduced voltage on the no-load speed.

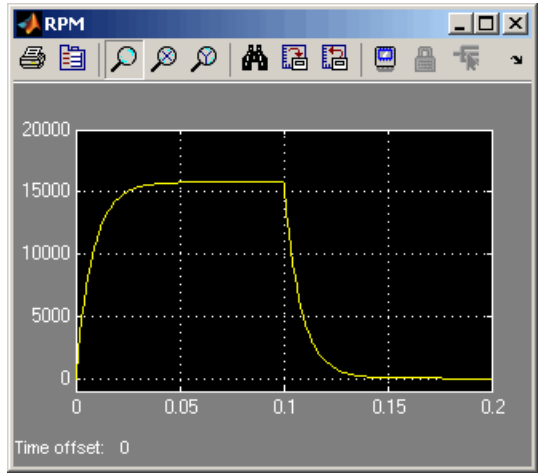




3 Try varying the load torque to find the maximum torque at this reduced voltage. Double-click the Step source block, enter different final values for the input signal, and rerun the simulation.

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.

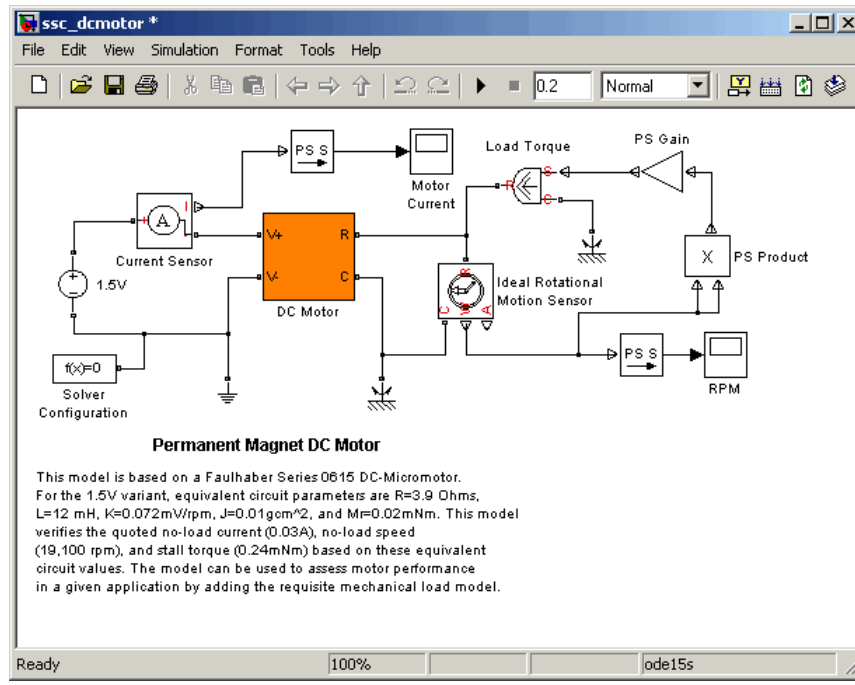




Changing the Motor Load

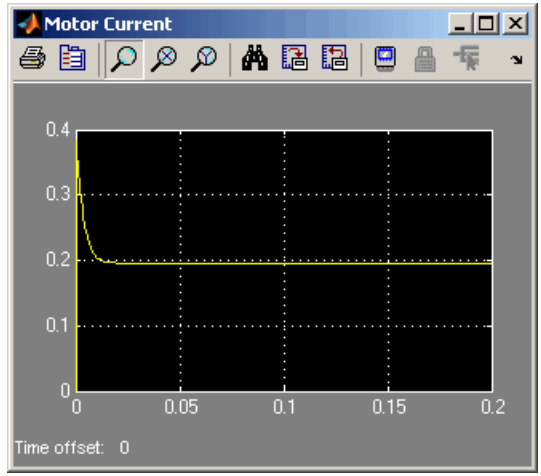
Replace the torque source with a simple mechanical load, for example, a fan, for which the torque is defined by `alpha*speed^2`, where `alpha` is `-1e-10` Nm/(rad/s)^2.

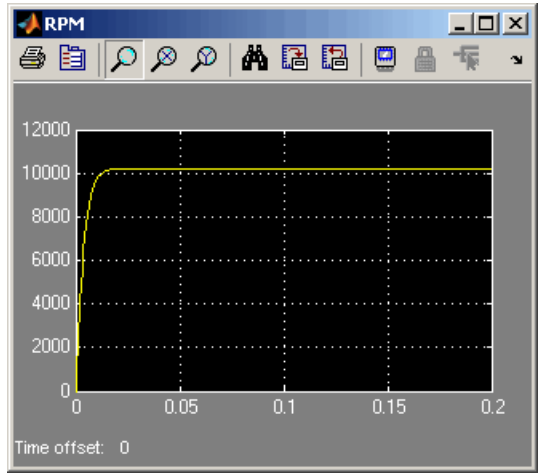
1. Delete the Step source and the Simulink-PS Converter block from the model.
2. In the **Simscape** block library, open **Foundation Library** > **Physical Signals** > **Functions**.
3. Drag the PS Product block and the PS Gain block to the model window.
4. Connect the blocks as shown in the following illustration. To rotate a block, select it and press **Ctrl+R**.



5 Double-click the Gain block to open its dialog box. Enter **Gain** value of `-1e-10` and click **OK**.

6 Run the simulation and assess motor performance with the new load.


Motor Current
0.4
0.3
0.2
0.1
0
0
0.05
0.1
0.15
0.2
Time offset: 0


RPM
12000
10000
8000
6000
4000
2000
0
0
0.05
0.1
0.15
0.2
Time offset: 0

Learning More

In this section...
"Next Steps" on page 1-21
"Product Help" on page 1-21
"The MathWorks Online" on page 1-22

Next Steps

To take the next lesson on using the product, follow the steps in "Creating a Simple Model" in the *Simscape User's Guide*. Also see the following topics to continue your exploration:

For...	See...
Rules of building models with physical components	"Basic Principles of Modeling Physical Networks"
Variables available with each domain	"Variable Types"
Block orientation in a model	"Direction of Variables "
Overview of the simulation process	"How Simscape Simulation Works"
Code generation	"Generating Code"

Product Help

More information is available with your product installation. In the MATLAB desktop, click for help, and then click the product name in the **Contents** pane.

For...	See...
List of blocks	"Block Reference"
Advanced tutorials	Examples
More product demonstrations	Simscape Demos
What's new in this product	Release Notes

The MathWorks Online

Point your Internet browser to the MathWorks Web site for additional information and support at http://www.mathworks.com/products/simscape/.