MapleSim User's Guide

MapleSim User's Guide

Copyright

This document was produced using a special version of Maple and DocBook.

Printed in Canada

ISBN

Contents

Introduction

MapleSim Overview

MapleSim™ is a complete environment for modeling and simulating complex multi-domain physical systems. It allows you to build component diagrams that represent physical systems in a graphical form. Using both symbolic and numeric approaches, MapleSim generates model equations from a component diagram and runs high-fidelity simulations.

Complex Multi-domain Modeling

Using MapleSim, you can build physical models that integrate components from various engineering fields into a complete system. MapleSim features an extensive library of over 500 modeling components, including electrical, mechanical, and thermal devices; sensors and sources; and signal blocks. You can modify exising model libraries or create custom libraries to suit your modeling and simulation needs. You can also import modeling components available from the Modelica Association (www.modelica.org) or other specialized library vendors.

Advanced Symbolic and Numeric Capabilities

MapleSim uses the advanced symbolic and numeric capabilities of Maple™ to generate the mathematical models that simulate the behavior of a physical system. You can, therefore, apply simplification techniques to equations to create concise and numerically-efficient models.

Accessible Technical Document Features

You can use the technical document features in Maple to work with model equations and perform advanced analysis, such as parameter optimization. To analyze your physical model and present your simulation results in a readable and interactive format, you can use Maple features such as embedded components, plotting tools, and document creation tools. These features can help you to disseminate your simulation results and reduce work in future projects. Finally, you can use the code generation feature in Maple to work with your models in other applications and tools, including real-time simulation.

Related Products

To use MapleSim, you must first install Maple 12 on your computer.

Maplesoft™ offers a suite of toolboxes, packages, and other applications that extend the capabilities of Maple and enhance MapleSim functionality. For a complete list of products, visit the Maplesoft Web site at www.maplesoft.com/products.

Related Resources

For additional resources, visit www.maplesoft.com/site_resources.

Resource	**Description**
MapleSim Installation Guide	Provides system requirements and installation instructions for MapleSim. The **MapleSim Installation Guide** is available in the **Install.html** file on your MapleSim installation CD.
MapleSim Help System	Provides the following information: • **MapleSim User's Guide**, which contains conceptual information about MapleSim and tutorials to help you get started using the software • **MapleSim Library Reference Guide**, which contains descriptions of the modeling components available in MapleSim • Help pages for model building, simulation, and analysis tasks

Resource	Description
MapleSim Examples	Provides sample pre-built models from various domains. These examples are available in the MapleSim **Examples** palette.

Getting Help

To open the MapleSim help system, press **Ctrl + F1**.

Customer Feedback

Maplesoft welcomes your feedback. For comments related to the MapleSim product documentation, contact doc@maplesoft.com.

1 Getting Started with MapleSim

In this chapter:

1.1 Physical Modeling in MapleSim

Physical modeling, or physics-based modeling, incorporates mathematics and physical laws to describe the behavior of an engineering component or a system of interconnected components. Since most engineering systems have associated dynamics, the behavior is typically defined with ordinary differential equations (ODEs).

To help you develop physical models quickly and easily, MapleSim provides the following features:

Topological or "Acausal" System Representation

Unlike the signal-flow approach, which requires system inputs and outputs to be defined explicitly, a topological representation allows you to connect interrelated components without having to consider how signals flow between them.

Mathematical Model Formulation and Simplification

A topological representation maps readily to its mathematical representation and the symbolic capability of MapleSim automates the generation of equations.

Also, when MapleSim formulates the equations, several mathematical simplification tools remove any redundant equations and multiplication by zero or one. The simplification tools, then, combine and reduce the expressions to get the minimal set of equations required to represent a system without losing fidelity.

Advanced Differential Algebraic Equation Solvers

Algebraic constraints are introduced in the topological approach to model definition, and problems that combine ODEs with these algebraic constraints are called Differential Algebraic Equations (DAEs). Depending on the nature of these constraints, the DAE problem increases in complexity, which is usually indicated by an increase in the “index” of the DAEs.

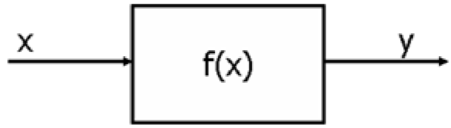
The development of generalized solvers for complex DAEs is the subject of much research in the symbolic computation field. As a result, MapleSim provides advanced DAE solvers that incorporate leading-edge symbolic and numeric techniques for solving high-index DAEs.

About Acausal and Causal Modeling

Since most engineering systems consist of physical components, system models can be defined in terms of their components and how they are connected. For example, if you want to define an electrical circuit model, you can construct it either as an electrical circuit or as a mechanical system, depending on the connections you specify. In other words, you can construct a system topology using blocks to represent components and lines to represent the relationships between those components.

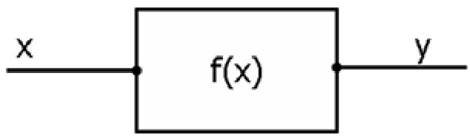
MapleSim allows you to use this topological approach to defining a model. The underlying physical laws are defined mathematically within the model components, and the connections between the components determine the system behavior. This approach is described as “acausal.”

In contrast, the signal-flow block-diagram approach is described as “causal,” meaning that each block requires an input as a signal flowing in one direction. A mathematical operation is performed on the inputs and the result is sent out as a signal (or set of signals). In programming terms, this approach is analogous to an assignment, $y := f(x)$, where a calculation is performed on a known variable or set of variables on the right hand side and the result is assigned to another variable on the left:



This approach is useful for modeling systems that handle signals, for example, controllers, digital filters, and other signal processing systems.

Conversely, an acausal model is simply a statement of the relationship between a block and its connecting ports, which are bi-directional. The programming analogy would be a simple equality statement, $y = f(x)$:

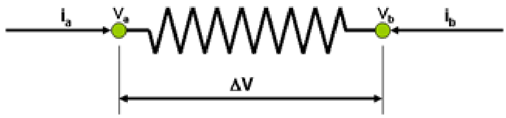


The mathematical representation within a block determines the physical conservation law to which the component must comply. This law states that the sum of all the energies, currents, torques, heat flows, or masses -- depending on the purpose of a component -- is equal to zero at the connection ports. You can use MapleSim to create causal models or a model that mixes both approaches.

Through and Across Variables

To understand these concepts further, it is useful to identify the *through* and *across* variables of the component you are modeling. In general terms, an across variable represents the driving force in a system and a through variable represents the flow of a conserved quantity:

For example, in an electrical circuit, the through variable, *i*, is the current and the across variable, *V*, is the voltage drop:



The following table lists some examples of through and across variables for other domains:

Domain	**Through**	**Across**
Electrical	Current (*A*)	Voltage (*V*)
Mechanical (translational)	Force (*N*)	Velocity (*m/s*)
Mechanical (rotational)	Torque (*N.m*)	Angular Velocity (*rad/s*)
Hydraulic	Flow (m^3 /*s*)	Pressure (*N/* m^2)
Heat flow	Heat flow (*W*)	Temperature (*K*)

As a simple example, the form of the governing equation for a resistor is

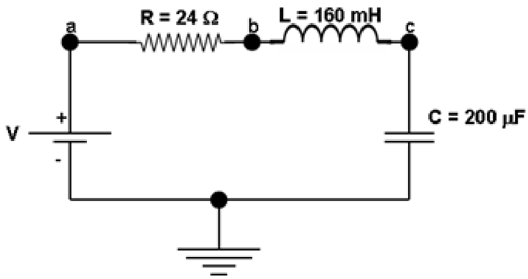
$$V = R \cdot i$$

For the purpose of an acausal representation, the equation would be rewritten as

$$R \cdot i_b = V_b - V_a \text{ and } i_b + i_a = 0$$

This equation, in conjunction with Kirchhoff's conservation of current law, allows a complete representation of a circuit.

To extend this example, the following schematic diagram descibes an RLC circuit:



If you wanted to model this circuit manually, it can be represented with the following characteristic equations for the resistor, inductor, and capacitor respectively:

$$R \cdot i_R = V_b - V_c$$

$$L\frac{d}{dt}i_L = \left(V_a - V_b\right)$$

$$i_c = C \cdot \frac{d}{dt}V_c$$

By applying Kirchhoff's law, the following conservation equations are at points *a*, *b*, and *c*:

$$i_V - i_R = 0$$

$$i_R - i_L = 0$$

$$i_L - i_C = 0$$

These equations, along with a definition of the input voltage

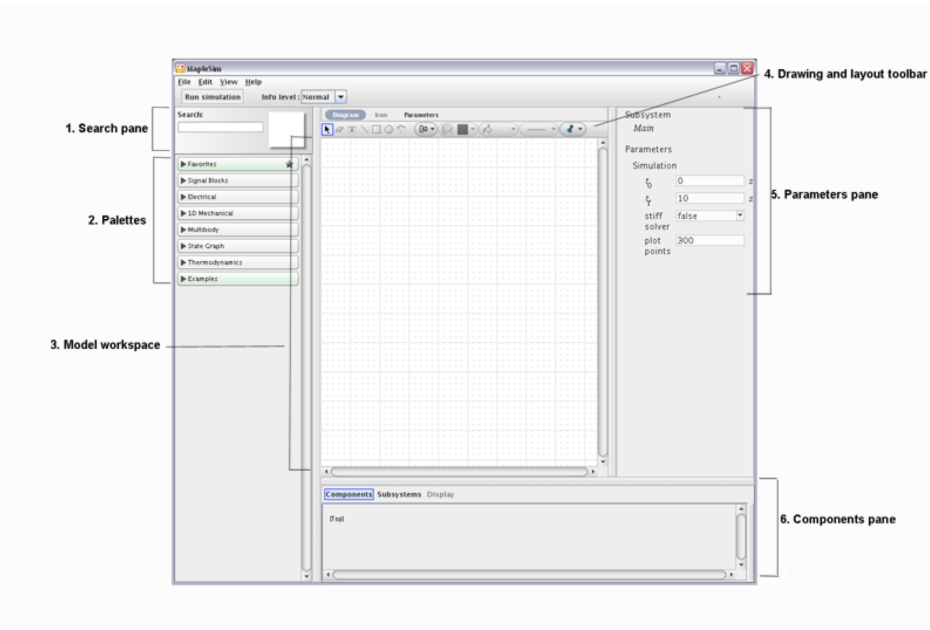
$$V_a = \begin{cases} 0.0 & 0.0 \leq t < 1.0 \\ 1.0 & t \geq 1.0 \end{cases}$$

provide enough information to define the model and solve for the voltages and currents through the circuit.

In MapleSim, all of these calculations are done for you automatically: you need only to draw the circuit and provide the component parameters. These principles can be applied equally to all engineering domains in MapleSim and allow you to connect components in one domain with components in others easily.

1.2 The MapleSim Screen

The MapleSim screen contains the following panels and components:



Component	**Description**
1. Search pane	Allows you to search for and then add a modeling component directly to a physical model.
2. Palettes	Expandable menus containing sample models and domain-specific components that you can add to a physical model.
3. Model workspace	The area in which you build and edit a physical model.
4. Drawing and layout toolbar	Contains tools that you use to lay out objects in the model workspace and add illustrations and annotations to a physical model.
5. Parameters pane	Allows you to view and change the parameter values and names of components in a physical model, and specify simulation values. The contents of this pane change depending on your selection in the model workspace.
6. Components pane	Allows you to manage subsystems and navigate the hierarchy of your physical model after you add components to the model workspace.

1.3 Basic Tutorial: Modeling an RLC Circuit and DC Motor

In this tutorial, you will perform the following tasks:

1. Build an RLC circuit.
2. Set component parameters to specify simulation conditions.
3. Add probes to specify the values to include in the simulation.
4. Simulate the RLC circuit model.
5. Modify the RLC circuit diagram to create a simple DC motor model.
6. Simulate the DC motor model under different conditions.

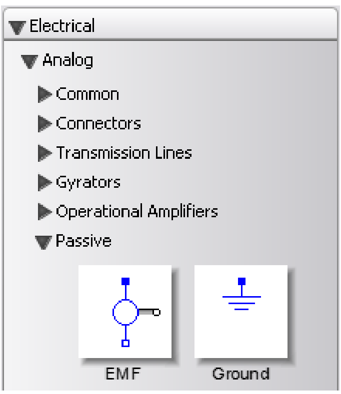
This tutorial is intended to familiarize you with some of the modeling components available in the MapleSim library and basic model development tools. It illustrates the ability to mix causal models with acausal models. The RLC circuit is an acausal representation because you connect modeling components to define the system topology.

Building an RLC Circuit Model

To build a physical model, you add components to the model workspace and then connect them in a system. In this example, the RLC circuit model contains ground, resistor, inductor, capacitor, and signal voltage components from the Electrical library. It also contains a step source, which is a signal generator that drives the voltage level in the source.

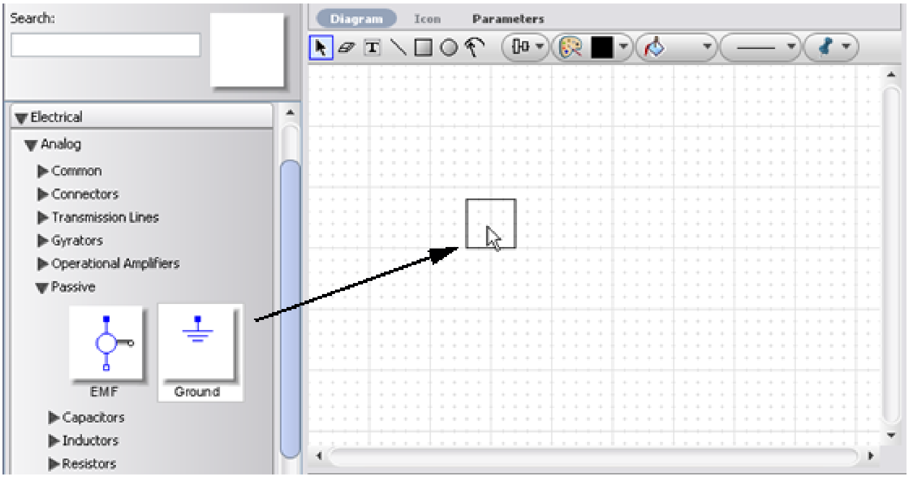
To build an RLC circuit model:

1. On the left side of the screen, click the triangle beside **Electrical** to expand the palette. In the same way, expand the **Analog** menu and then expand the **Passive** submenu.



Tip: To view the help topic associated with a modeling component, right-click a component icon in a palette and select **Help**.

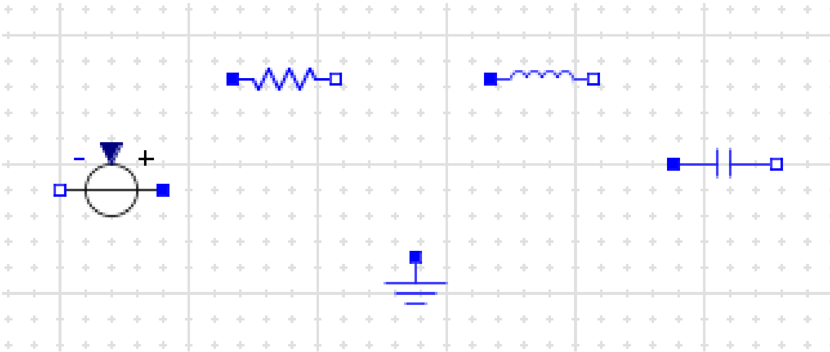
2. From the **Electrical** → **Analog** → **Passive** menu, click and drag the **Ground** icon to the model workspace.



3. Using the process described in steps 1 and 2, add the remaining electrical components to the model workspace:

- From the **Electrical** → **Analog** → **Passive**→ **Resistors** menu, add the **Resistor** component.
- From the **Electrical** → **Analog** → **Passive** → **Inductors** menu, add the **Inductor** component.
- From the **Electrical** → **Analog** → **Passive** → **Capacitors** menu, add the **Capacitor** component.
- From the **Electrical** → **Sources** → **Voltage** menu, add the **Signal Voltage** component.

4. To position the modeling components in the model workspace, click and drag the components in the arrangement as shown below.

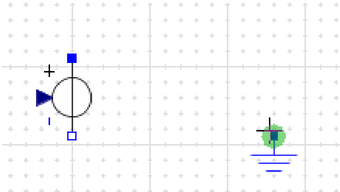


5. To rotate the **Signal Voltage** component clockwise and then flip it horizontally, right-click the **Signal Voltage** icon in the model workspace and select **Rotate Clockwise**. Right-click the component again and select **Flip Horizontal**.

6. To rotate the **Capacitor** component clockwise, right-click the **Capacitor** icon in the model workspace and select **Rotate Clockwise**.

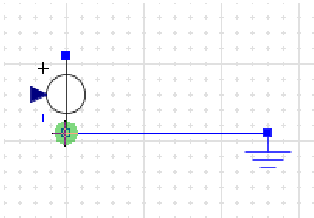
You can now connect the modeling components in your physical model to represent interactions in your system.

7. Hover your mouse pointer over the input port of the **Ground** component. The port is highlighted.



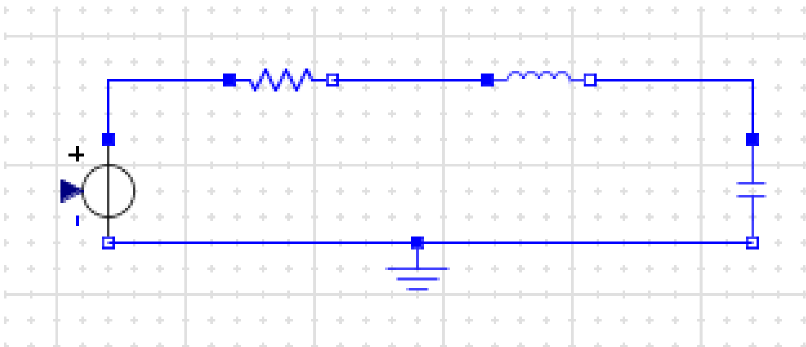
8. Click the **Ground** input port once to start the connection line.

9. Hover your mouse pointer over the input port of the **Signal Voltage** component. The port is highlighted.



10. Click the **Signal Voltage** input port. The **Ground** component is connected to the **Signal Voltage** component.

11. Connect the remaining components in the arrangement as shown below.



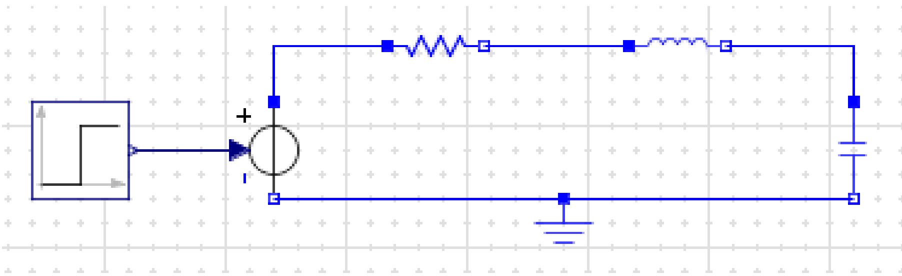
You can now add a source to your physical model.

12. Expand the **Signal Blocks** palette. Expand the **Sources** menu and then expand the **Real** submenu.

13. Click and drag the **Step** source from the palette to the left of the **Signal Voltage** component in the model workspace.

The step source has a specific signal flow, which is represented by the arrows on the connections. This flow causes the circuit to respond to the input signal.

14. Connect the **Step** source to the **Signal Voltage** component. The complete RLC circuit model is displayed below.



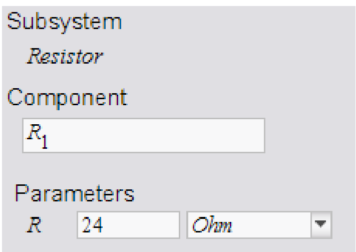
15. Save the physical model using the file name **RLC_Circuit1.msim**.

Setting Component Parameters

To specify simulation conditions, you set parameter values for components in your model.

To set component parameters:

1. In the model workspace, click the **Resistor** component. The Parameters panel on the right side of the screen displays the name and parameter values of the resistor.



2. In the **R** field, enter **24**, and press **Enter**. The resistance is changed to 24 Ω.

3. Using the process described in steps 1 and 2, specify the following parameter values for the other components:

- For the **Inductor**, specify an inductance of $\mathbf{160 \cdot 10^{-3}}$ H.

- For the **Capacitor**, specify a capacitance of $\mathbf{200 \cdot 10^{-6}}$ F.
- For the **Step** source, specify a T_0 value of **0.1** s.
- **Tip:** To enter a superscript, press **Shift + 6** (key combination for the caret symbol, ^) and then type the value to include in the superscript. To move the cursor out of the superscript region, press the right arrow key on your keyboard.

Adding Probes to the RLC Circuit Model

To specify the quantities to include in a simulation graph, you attach probes to lines or ports in your physical model. In this example, you will measure the voltage of the RLC circuit.

To attach a probe:

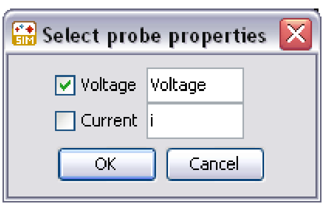
1. From the drawing and layout toolbar, click the probe icon .

2. Hover your mouse pointer over the line that connects the **Inductor** and **Capacitor** components. The line is highlighted.

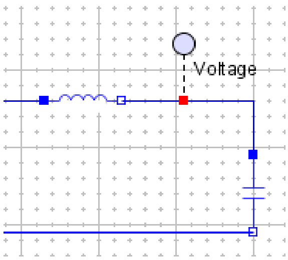
3. Click the line once. The **Select probe properties** dialog box is displayed.

4. To include the voltage value in the simulation graph, select the **Voltage** check box.

5. To display a customized name for this quantity in the model workspace, in the **Voltage** field, enter **Voltage**.



6. Click **OK**. The probe is added to the connection line.



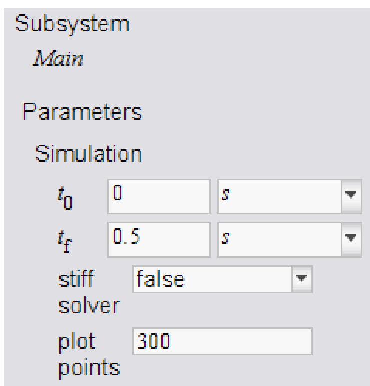
7. Click the probe once to position it on the line.

Simulating the RLC Circuit Model

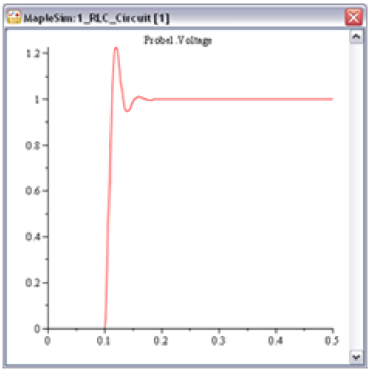
To simulate the physical model, you specify the duration for which to run the simulation.

To simulate an RLC circuit model:

1. To specify the duration of the simulation, in the Parameters pane, set the t_f parameter to **0.5** seconds and press **Enter**.



2. Click the **Run Simulation** button located in the top-left corner of the window. MapleSim generates the system equations and then simulates the response to the step input. When the simulation is complete, the voltage response is plotted in a graph.



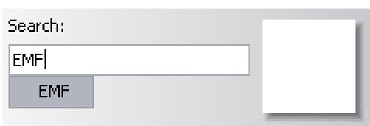
3. Save the physical model using the file name **RLC_Circuit2.msim**. The probes and modified parameter values are saved as part of your physical model.

Building a Simple DC Motor Model

You will now add an electromotive force (EMF) and a mechanical intertia component to the RLC circuit model to create a simple DC motor model. In this example, you will add components to the RLC circuit model using the search feature. This feature provides a quick and alternate means of adding a component to a physical model.

To build a simple DC motor model:

1. In the search pane located above the palettes, in the **Search** field, type **EMF**. A drop-down menu displays your search results.



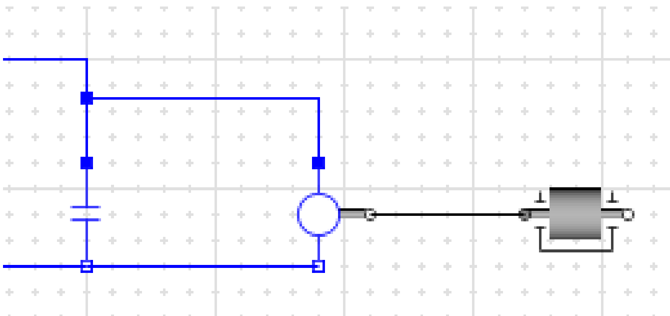
2. Select **EMF** from the drop-down menu. The **EMF** component is displayed in the square beside the search field.

3. From the square in the search pane, click and drag the **EMF** icon to the modeling workspace, placing it to the right of the **Capacitor** component. The **EMF** component is added to the physical model.

4. In the search pane, search for **Inertia**.

5. Add the **Inertia** component to the model workspace, placing it to the right of the **EMF** component.

6. Connect the components as shown below. To connect the positive port of the **EMF** component, click that port once, drag your mouse pointer to the line connecting the capacitor and inductor, and click the line.



7. In the model workspace, click the **EMF** component and change the value of the parameter, **k**, to **10.**

8. Click the **Step** component and change the value of the parameter, T_0, to **1**.

9. Save the physical model using the file name **DC_Motor1.msim**.

Simulating the DC Motor Model

To simulate the DC motor model:

1. From the drawing and layout toolbar, click the probe icon .

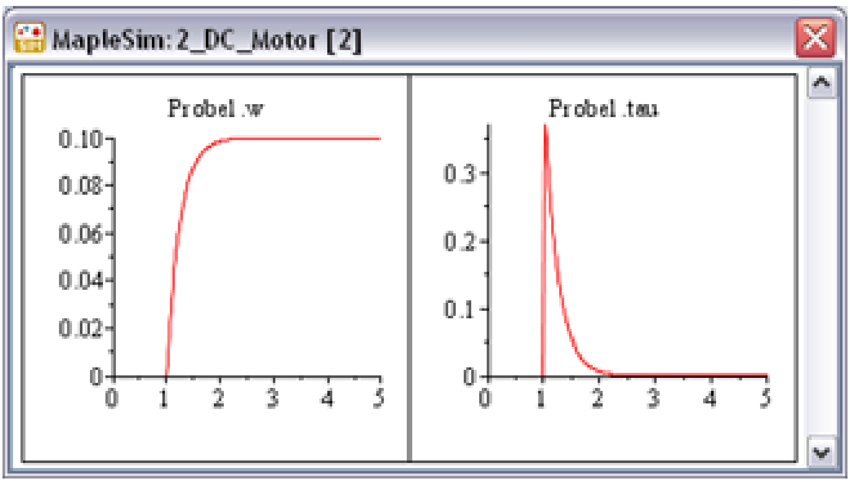
2. Hover your mouse pointer over the line that connects the **EMF** and **Inertia** components. The line is highlighted.

3. Click the line once. The **Select probe properties** dialog box is displayed.

4. To include the speed and torque values in the simulation graphs, select the **Speed** and **Torque** check boxes.

5. Click **OK**. The probe, with an arrow indicating the direction of the conserved quantity flow, is added to the physical model.

6. In the Parameters panel, set the t_f parameter to **5** seconds and then click **Run Simulation**. The following graphs are displayed.



7. Save the physical model using the file name **DC_Motor2.msim**.

2 Building a Physical Model

In this chapter:

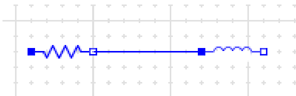
2.1 About the MapleSim Library

The MapleSim library contains over 500 components that you can use to build a physical model. These components, which are based on the Modelica Standard Library, are organized in palettes according to their respective domains: electrical, 1-D and multibody mechanical, thermal, state graph, and signal blocks. The library also contains sample models that you can view and simulate, for example, complete electric circuits and filters. For more information about the MapleSim library structure and modeling components, see the **MapleSim Library Reference Guide** in the help system.

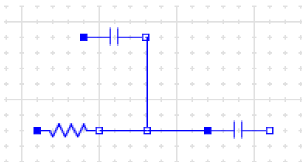
To extend the default library, you can also create custom modeling components from a mathematical model and define new libraries. For more information, see *Creating Custom Modeling Components (page 22)* and *About Custom Libraries (page 24)*.

2.2 Defining How Components Interact in a System

To define how modeling components interact, you connect them in a system. You can draw a connection line between two connection ports



or between a port and another connection line.



By default, each connection type is displayed in a specific color:

Connection Type	Color
Mechanical 1-D rotational	Black
Mechanical 1-D translational	Green
Mechanical multibody	Black
Electrical	Blue
Digital logic	Purple
Boolean signal	Pink
Causal signal	Navy blue
Integer signal	Orange
State Graph	Black
Thermal	Red

For more information, see the **Using MapleSim** → **Building a Model** → **Connecting Modeling Components** topic in the MapleSim help system.

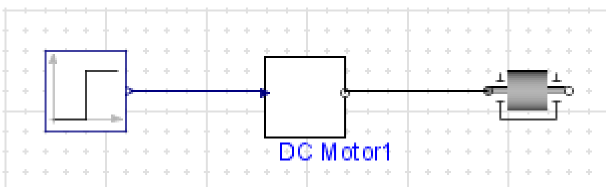
2.3 Specifying Simulation Conditions

To specify simulation conditions for your physical model, you set the parameter values for individual modeling components. When you select a component in the model workspace, the configurable parameter values for that component are displayed in the **Parameters** panel. Not all modeling components contain parameter values that you can edit. To view a short description for a parameter, click your mouse pointer once in a parameter field.

For more information, see the **Using MapleSim** → **Building a Model** → **Specifying Parameter Values** topic in the MapleSim help system.

2.4 About Subsystems

A subsystem is a set of modeling components that are grouped within a single block component. For example, you can group components that form a complete system, such as a tire or DC motor, or common components according to their domains. A sample DC motor subsystem is shown in the following image:



You can connect a subsystem to other components and use various tools to reuse subsystems and manage complex systems more efficiently. For more information, see the **Using MapleSim** → **Building a Model** → **Working with Subsystems** section in the MapleSim help system.

2.5 Creating Custom Modeling Components

To extend the MapleSim library, you can create a custom component that is based on a mathematical model. For example, you might want to create a custom component to contain a subsystem or to provide specialized functionality.

Using the Custom Component template included with MapleSim, you perform the following tasks in Maple to create a custom component:

- Define the component equations and properties that determine the behavior of the component (for example, parameters and port variables)
- Test and analyze your mathematical model
- Define ports and add them to the component
- Generate the component and make it available in MapleSim

You must create a separate worksheet for each custom component that you want to generate.

For more information about opening the custom component template, see the **Using MapleSim → Analyzing a Model → Performing Analysis Tasks in Maple** topic in the MapleSim help system.

Opening Sample Custom Component Worksheets

You may want to review sample custom component worksheets before creating custom components. The following examples are available:

Worksheet	Description
Algebraic Equation	Custom component defined using an algebraic equation
DC Motor	Sample DC motor component defined using a differential equation
Transfer Function	Custom component defined using a transfer function

To open a sample custom component worksheet:

1. In MapleSim, from the **View** menu, select **Document Folder...**

2. In the **Document Folder for <file_name>** dialog box, click **More Templates...**

3. In the **Browse Templates** dialog box, browse to the **Component Templates** folder.

4. Select the sample worksheet that you want to open, and click **Attach Template...**

5. In the **Enter Document Name** dialog box, enter a unique name for the worksheet.

6. Click **OK**. An instance of the worksheet is added to the MapleSim document folder.

7. In the document list, select the worksheet entry that you just created and click **Open Selected**. The sample Custom Component worksheet is opened in Maple.

Working with Custom Components in MapleSim

After making a custom component available in MapleSim, you can perform the following tasks:

Add an Icon to the Custom Component

By default, a custom component is displayed as a white box in the MapleSim window. Using the drawing and layout tools provided in MapleSim, you can add an icon as a unique identifier for the custom component.

For more information, see the **Using MapleSim → Building a Model → Adding Lines and Shapes → Adding an Icon to a Subsystem or Custom Component** topic in the MapleSim help system.

Save the Custom Component as a Part of the Current Model

To save a custom component as a part of the current model, add the component by dragging it into the model workspace and save the model. The next time you open the file, the custom component will be displayed in the model workspace and in the **Subsystems** panel.

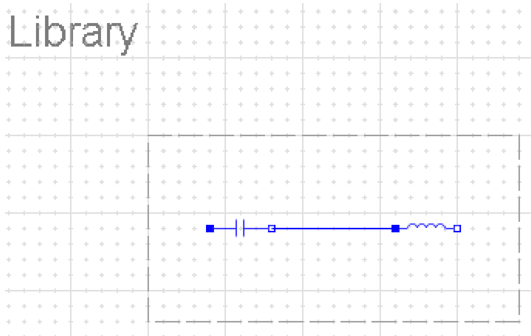
Add the Custom Component to a Custom Library

If you want to reuse a custom component in projects other than the current model, you can add the component to a custom library. For more information, see *About Custom Libraries (page 24)*.

2.6 About Custom Libraries

You can create a custom library to save a collection of subsystems or custom modeling components that you plan to reuse in future projects. Custom libraries are displayed in custom palettes below the Examples palette and saved on your hard drive with the file extension .msimlib: they will be displayed in those palettes the next time you start MapleSim.

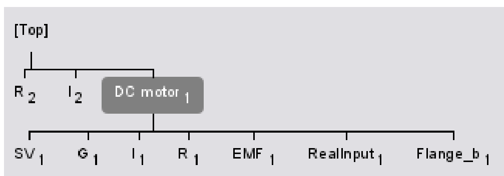
Subsystems or components cannot be modified after they have been added to a custom library (that is, you cannot modify elements such as connection lines, parameters, or icons). To find out whether a subsystem or component is part of a custom library, right-click the subsystem or component in the model workspace and select **Open Component**. A **Library** caption is displayed in the top-left corner of the model workspace if that component or subsystem is part of a custom library.



For more information, see the **Using MapleSim** → **Building a Model** → **Working with Subsystems** → **Creating a Custom Library** topic in the MapleSim help system.

2.7 Navigating a Physical Model

When you add a modeling component to the model workspace or create a subsystem, a node is added to the model tree in the **Components** panel below the model workspace. For example, the navigation tree of a DC motor model appears as follows:

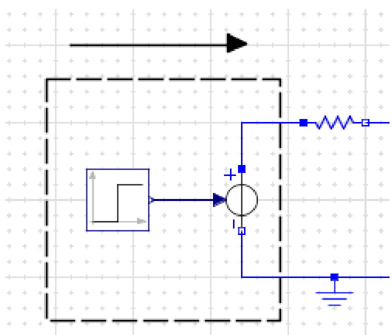


Each node in the model tree represents a modeling component, subsystem, or connection port in your physical model. To browse your model, click the nodes in the model tree. You can click the **[Top]** node to view the top level of your model or the child nodes to view individual modeling components, subsystems, or connection ports in detail. Alternatively, to view the contents of a node in detail, right-click a modeling component or subsystem in the model workspace and select **Open Component**.

The model tree can help you to create hierarchical levels of modeling components. For example, you can navigate to a child node and then add and connect modeling components at that sub-level.

2.8 Adding Text and Illustrations to a Physical Model

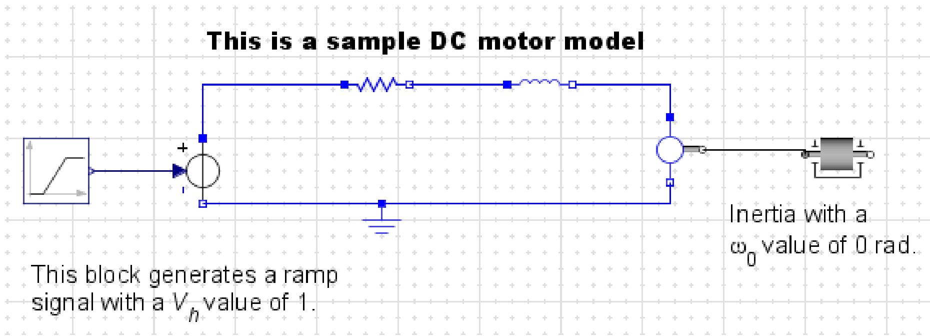
You can use the tools in the drawing and layout toolbar to draw lines, arrows, and shapes in the model workspace. MapleSim also provides many tools that you can use to customize the color, line style, and fill of lines and shapes, and lay out objects in the model workspace.



For more information, see the **Using MapleSim** → **Building a Model** → **Adding Lines and Shapes** section in the MapleSim help system.

Adding Text

You can use the text tool ( ) in the drawing and layout toolbar to annotate your physical model. In annotations, you can format expressions in 2-D math notation and modify the style or font of the text.



For more information, see the **Using MapleSim** → **Building a Model** → **Annotating a Model** section in the MapleSim help system.

3 Simulating a Physical Model

In this chapter:

3.1 Adding Probes to a Physical Model

To specify the quantities to include in a simulation, you add probes to connection lines or ports in your physical model. When you run a simulation, a separate graph is displayed for each value that you specify. If you add a probe to measure a flow, an arrow indicates the direction of the positive flow in the model workspace.

To attach a probe to a physical model:

1. From the toolbar, click the probe icon .

2. In the model workspace, hover your mouse pointer over the connection line or port to which you want to attach a probe. When that line or port is highlighted, click it once.

3. In the **Select Probe Properties** dialog box, select the quantities that you want to include in the simulation graphs.

4. (Optional) To display a custom name for a quantity in the model workspace, type a new name in the corresponding field for that quantity.

5. Click **OK**.

6. Position the probe on the line or port, and then click the probe once.

Alternatively, to attach a probe, you can right-click a connection line or port, and select **Attach Probe**. You can save probes as a part of your physical model.

3.2 Setting the Progress Information Level

During a simulation, progress messages are displayed in a **Progress Information** pop-up window. These progress messages inform you of the status of the MapleSim engine as it generates a mathematical model. Optionally, you can specify the amount of detail displayed in these messages. The following options are available:

- **Quiet**
- **Normal**
- **Verbose**

By default, the progress information level is set to **Normal**.

To set the progress information level:

From the **Info level** drop-down menu above the toolbar, select the information level that you want.

3.3 Running a Simulation

To run a simulation:

1. In the model tree, click **[Top]** to browse to the top level of the physical model.

2. In the **Parameters** panel, specify the simulation values. You can specify the following parameter values for all model types:

Parameter	Default	Description
t_0	0	Start time of the simulation. You can specify any positive value, including floating-point values.
t_f	10	End time of the simulation. You can specify any positive value, including floating-point values.
stiff solver	false	Integrator method used during the simulation. **true**: use a stiff DAE solver. **false**: use a non-stiff DAE solver.
plot points	200	Number of points to plot in the simulation graphs.

You can specify the following parameter values for models containing multibody mechanical components:

Parameter	Default	Description
$\widehat{e}_g$	$[0, -1, 0]$	Direction of gravity.
g	9.81	Gravitational acceleration.

3. In the top-left corner of the window, click **Run Simulation**. The **Progress Information** pop-up window displays the status of the MapleSim engine as it generates the mathematical model. When the simulation is complete, a graph for each specified quantity is displayed in a pop-up window.

After running a simulation, you can change the original probe or parameter values and run the simulation again. When you run the simulation again, a second set of graphs is displayed in a new pop-up window. For more information on changing physical model parameters, see Specifying Parameter Values for a Modeling Component.

3.4 Editing Probe Values

To edit probe values:

1. In the model workspace, right-click a probe that you want to edit, and select **Edit Probe**.

2. In the **Select Probe Properties** dialog box, edit the values or names.

3. Click **OK**.

Changing the Flow Direction of a Probe

To change the flow direction of a probe:

- In the model workspace, right-click the probe for which you want to change the flow direction, and select **Reverse Probe**.

4 Analyzing and Manipulating a Physical Model

In this chapter:

4.1 Working with Model Properties and Equations in Maple

Because MapleSim is fully integrated with the Maple environment, you can use Maple technical document features and commands to work with the properties and equations of your MapleSim model. In particular, the **MapleSim** package provides various commands that you can use to analyze and work with model properties and equations. For more information, see the **?MapleSim** topic in the Maple 12 help system.

You can also use the Data Analysis and Parameter Optimization templates to open a MapleSim model in Maple and perform analysis tasks using pre-built tools. For more information about using the templates, see the **Using MapleSim** → **Analyzing a Model** → **Performing Analysis Tasks in Maple** topic in the MapleSim help system.

Analyzing and Editing a Physical Model Using Maple Embedded Components

Embedded components are graphical user interface elements that you can add to a Maple Standard worksheet to visualize and analyze mathematical expressions.

In Maple, the MapleSim Model embedded component is available in the **Components** palette. You can use it to view and edit the properties of MapleSim models programmatically and analyze your model in a Maple worksheet. You can also associate model properties with other Maple embedded components, including sliders and plots.

For more information about the MapleSim Model component, see the **?MapleSimModel** topic in the Maple 12 help system. For more information about embedded components, see the **?EmbeddedComponents** topic in the Maple 12 help system.

4.2 Generating Code from a Physical Model

If you want to use or test your physical model in Simulink®, you can use the Code Generation Template to generate code from your physical model. For more information about using the template, see the **Using MapleSim** → **Analyzing a Model** → **Performing Analysis Tasks in Maple** topic in the MapleSim help system.

4.3 Working with Attachments

If you create an external document that relates to a physical model (for example, a design document, code generation document, or Maple worksheet), you can attach it to a physical model and save it as a part of the model using the MapleSim document folder. You can attach a file in any format and there are no size limitations on attachments.

For more information, see the **Using MapleSim** → **Analyzing a Model** → **Attaching a File to a Model** topic in the MapleSim help system.

5 Advanced Tutorial

5.1 Modeling a DC Motor with a Gearbox

In this tutorial, you will perform the following tasks:

1. Add a gearbox to the DC motor that you created in the basic tutorial.

2. Simulate the DC motor with a gearbox.

3. Group the DC motor components into a subsystem.

4. Assign global parameters to the model.

5. Add signal block components and a PI controller to the model.

6. Simulate the modified DC motor model under different conditions.

Adding a Gearbox to the DC Motor Model

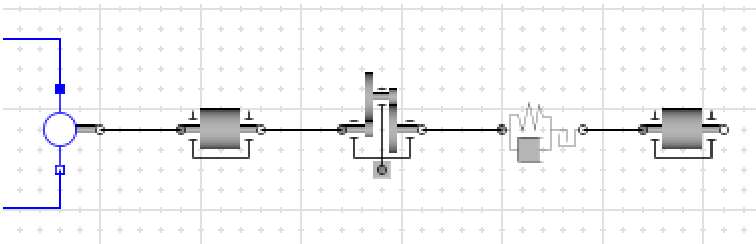
To add a gearbox to the DC motor model:

1. Open the **DC_Motor2.msim** file that you created in the basic tutorial.

2. To add components to the model, perform the following tasks:

- From the **1-D Mechanical** → **Rotational** → **Bearings and Gears** menu, add the **Ideal Gear** component to the model workspace and place it to the right of the **Inertia** component.
- From the **1-D Mechanical** →**Rotational** → **Springs and Dampers** menu, add the **Elasto Backlash** component to the model workspace and place it to the right of the **Ideal Gear** component.
- From the **1-D Mechanical** →**Rotational** →**Common** menu, add another **Inertia** component to the model workspace and place it to the right of the **Elasto Backlash** component.

3. Connect the components to the first **Inertia** component as shown below.



4. In the model workspace, click the **Ideal Gear** component.

5. In the Parameters panel, in the **r** field, enter **10** and press **Enter**. The transmission ratio between the flanges of the gearbox is changed to 10.

6. Using the process described in steps 4 and 5, specify the following parameter values for the other components:

- For the **Elasto Backlash** component, in the **b** field, specify a total backlash of **0.3** *rad*. In the **d** field, specify a damping constant of **10000** $\frac{N.m.s}{rad}$.

- For the **Inertia** component, in the **J** field, specify a moment of inertia of **10** *kg.m2*.

Simulating the DC Motor with a Gearbox Model

To simulate the DC motor with a gearbox model:

1. From the drawing and layout toolbar, click the probe icon .

2. Hover your mouse pointer over the line that connects the **Elasto-Backlash** and **Inertia** components. The line is highlighted.

3. Click the line once. The **Select probe properties** dialog box is displayed.

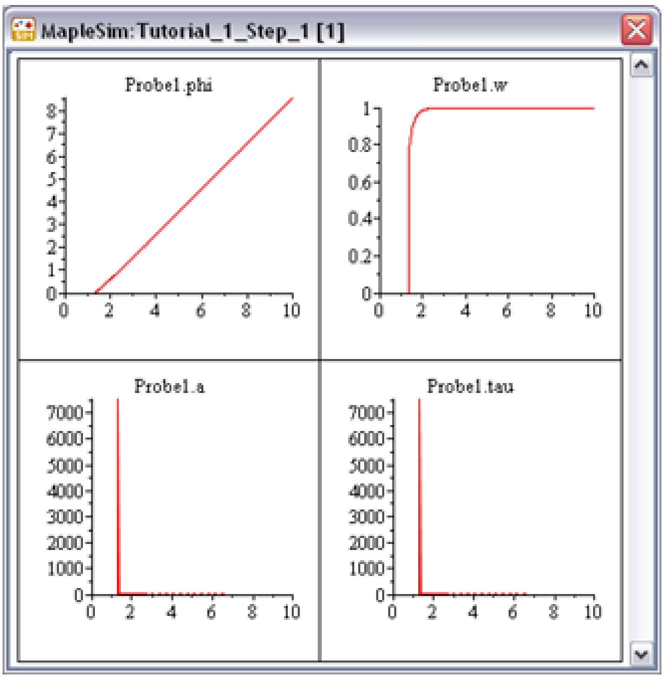
4. To include the angle (ϕ), speed (w), acceleration (a), and torque (τ) values in the simulation graphs, select **Angle**, **Speed**, **Acceleration**, and **Torque**.

5. Click **OK**. The probe is added to the connection line.

6. Click the probe once to position it on the line.

7. In the **Parameters** panel, in the t_f field, specify a simulation end time of **10** seconds.

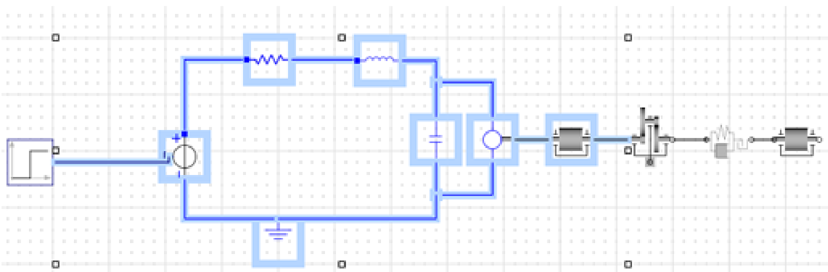
8. Click **Run Simulation**. When the simulation is complete, the following graphs are displayed:



Grouping the DC Motor Components into a Subsystem

To group the DC motor components into a subsystem:

1. Using the selection tool in the drawing and layout toolbar (), draw a box around the electrical components and the first inertia component.



2. From the **Edit** menu, select **Create Subsystem**.

3. In the **Create Subsystem** dialog box, enter **DC motor**.

4. Click **OK**. A white box, which represents the DC motor, is displayed in the model workspace.

Tip: To view the components in the subsystem, right-click the **DC motor** subsystem in the model workspace, and select **Open Component**. To navigate back to the top level of the physical model, in the model tree below the model workspace, click **[Top]**.

Assigning Global Parameters to a Physical Model

If your physical model contains multiple instances of a parameter value, you can create a global parameter. A global parameter allows you to define a common parameter value in one location and apply that value to multiple modeling components that use that parameter.

To assign global parameters:

1. Navigate to the top level of the physical model.

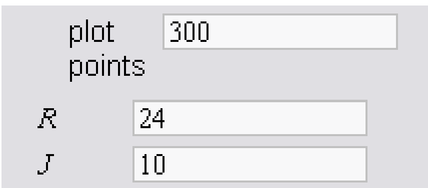
2. Click the **Parameters** button above the toolbar. The Parameters view is displayed.

3. In the first row of the **Main subsystem default settings** table, define a parameter called **R**. Specify a default value of **24** and enter **Global resistance variable** as the description.

4. In the second row of the table, define a parameter called **J**. Specify a default value of **10** and enter **Global moment of intertia value** as the description.

5. Click anywhere outside of the table.

6. To navigate back to the Diagram view, click the **Diagram** button above the toolbar. The new **R** and **J** parameters are displayed in the **Parameters** pane below the **plot points** field.



You can now assign these values to other components in your model

7. Navigate back to the Parameters view.

8. In the I_3 **component** table, in the value field for the moment of inertia parameter, enter **J** and click anywhere outside of the table.

I_3 component

Name	Type	Value	Units	Description
J	Inertia	J	$kg\ m^2$	Moment of inertia

The moment of intertia parameter now inherits the numeric value of the global parameter, **J** (in this example, 24).

9. In the Diagram view, right-click the DC Motor subsystem and select **Open Component**.

10. Click **Parameters**.

11. In the $\boldsymbol{R_1}$ **component** table, in the value field for the resistance parameter, enter **R** and click anywhere outside of the table.

12. In the $\boldsymbol{EMF_1}$ **component** table, in the value field for the transformation coefficient, enter $\boldsymbol{R \cdot J}$ and click anywhere outside of the table. **Note:** This value is an approximation of the transformation coefficient.

13. Save the physical model using the file name **DC_Motor3.msim**.

Changing Input and Output Values

To change the input and output values:

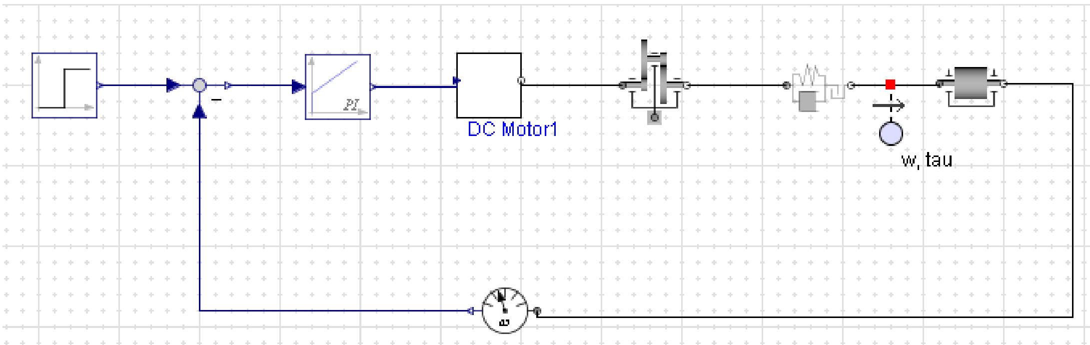
1. From the **1-D Mechanical** → **Rotational** →**Sensors** menu, add the **Rotational Speed Sensor** component to the model workspace and place it below the gearbox components.

2. Delete the connection line between the **Step** and **Signal Voltage** components.

3. From the **Signal Blocks** →**Controllers** menu, add the **PI** component to the model workspace and place it to the left of the **Signal Voltage** component.

5. From the **Mathematical** → **Operators** menu, add the **Feedback** component to the model workspace and place it to the left of the **PI** component.

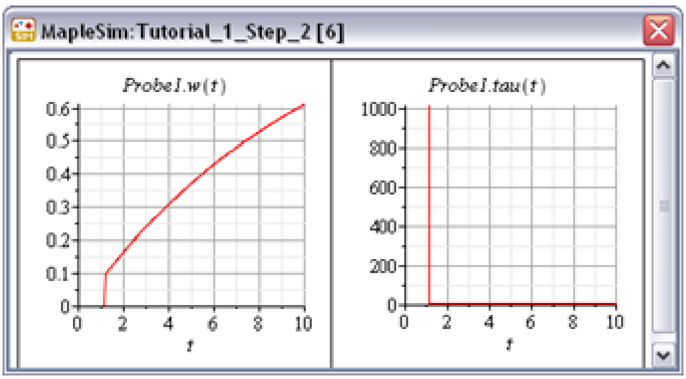
6. Connect the components as shown below.



7. Click the **PI** component in the model workspace.

8. In the **Parameters** panel, specify a gain of **20** in the **k** field, and a time constant of **3** in the **T** field.

9. Run the simulation again. When the simulation is complete, the following graphs are displayed:



10. Save the physical model using the file name **DC_Motor4.msim**.

6 Reference: MapleSim Keyboard Shortcuts

Opening, Closing, and Saving a Physical Model

Task	Key Combination
Create a new physical model	**Ctrl** + **N**
Open an existing physical model	**Ctrl** + **O**
Close the active document	**Ctrl** + **Shift** + **F4**
Save a physical model as an .msim file	**Ctrl** + **S**

Exporting and Printing a Physical Model

Task	Key Combination
Export a physical model as an image	**Ctrl** + **E**
Print a physical model	**Ctrl + P**

Building a Physical Model

Task	Key Combination
Rotate the selected modeling component 90 degrees clockwise	**Ctrl** + **R**
Rotate the selected modeling component 90 degrees counter-clockwise	**Ctrl** + **L**
Flip the selected modeling component vertically	**Ctrl** + **F**
Flip the selected modeling component horizontally	**Ctrl** + **H**
Group the selected modeling components into a sub-model	**Ctrl** + **G**
Display or hide probes in the model workspace	**Ctrl** + **D**

Navigating a Physical Model

Task	**Key Combination**
View the selected modeling component or subsystem in detail	**Ctrl** + **M**
Zoom in to the model workspace	**Ctrl** + <*number*> + plus sign key (+)
Zoom out from the model workspace	**Ctrl** + <*number*> + hyphen key (-)

Drawing and Layout Tools

	Key
Switch to the selection tool	**S**
Switch to the eraser tool	**E**
Switch to the text box tool	**T**
Switch to the line tool	**L**
Switch to the rectangle tool	**R**
Switch to the oval tool	**O**

Index