1. Introduction to MapleSim

In this chapter, you will be introduced to the MapleSim environment. Using examples included with MapleSim, you will learn how to run simulations and customize your results.

User Interface

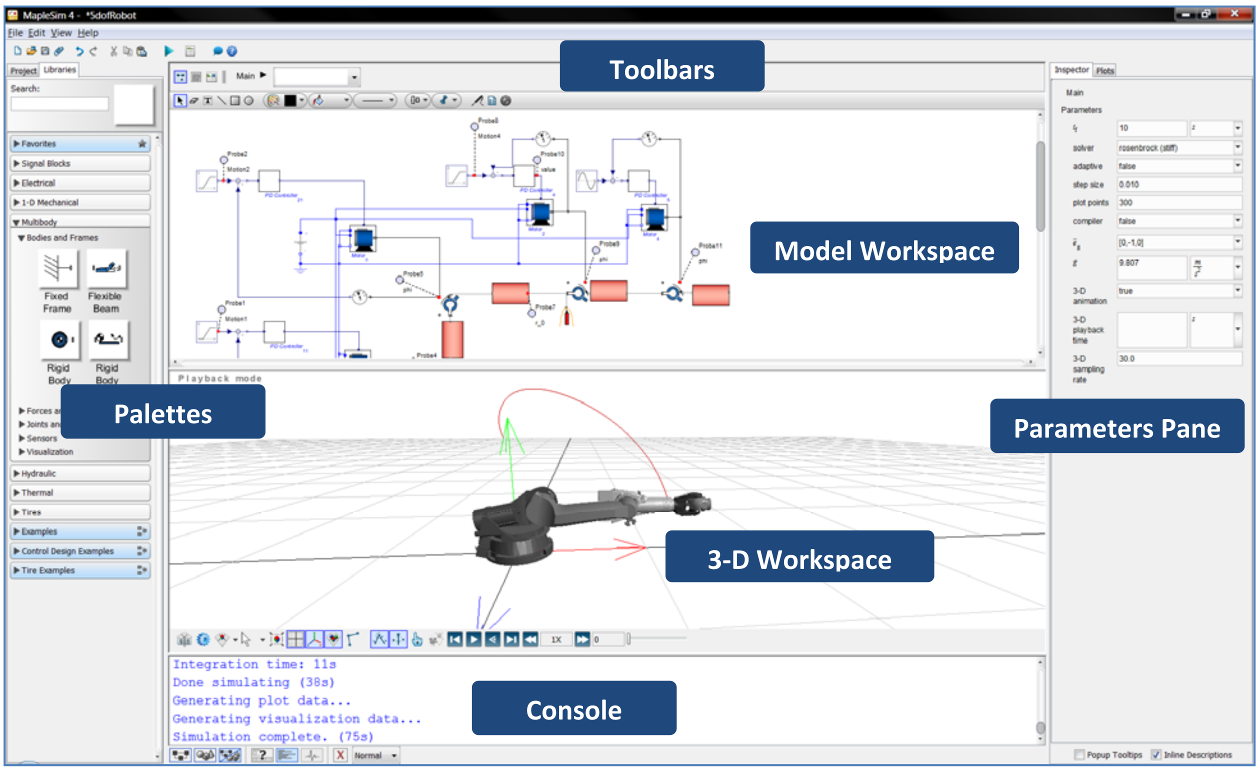


figure 1.1: MapleSim environment

MapleSim has two workspaces for assembling your models – the **Model Workspace** and **3-D Workspace**. You can toggle between these views using . The Model Workspace is the main work area for creating your MapleSim models. The 3-D Workspace has two operating modes, **Constructor** and **Playback**. The Constructor mode gives you a preview of your multibody models as you are building them in the model workspace, and also allows you to build models directly in the 3-D space as well. The Playback mode is used to display finished animations of you multibody models. You can toggle between construction and playback mode using the icons  and .

The **Palettes** bar contains expandable menus with tools to help build a model and manage your MapleSim project. This pane contains two tabs, **Components** and **Project**. The Components tab contains palettes with domain specific components to build your models, and sample models. The Project tab contains palettes with tools to help browse your model, and manage your results.

Toolbars contain tools for running and viewing simulations, viewing attached documents, navigating your model hierarchy, and laying out your model.

The **Console** contains three panes: **Help, Debugging,** and **Message Console**. The Help pane displays context-sensitive help topics associated with a modeling component. The Debugging pane displays diagnostic messages as you build your model. The Message Console displays progress messages indicating the status of the MapleSim engine during a simulation. You can change between these panes using the icons .

The **Parameters Pane** contains two tabs, **Inspector** and **Plots**, which change depending on your selection in the Model Workspace. The Inspector tab allows you to view and edit modeling component properties, such as names and parameter values, and specify simulation parameters. The Plots tab allows you to define custom layouts for simulation graphs and plot windows.

2. Working with a Sample Model

Included with MapleSim is a collection of example models from different engineering domains and that are available from the **Examples Palettes**. To open a model, expand the **Examples** palette and then expand **Visualization**. Select **Robot with Five Degrees of Freedom**. This will load the model into the workspace.

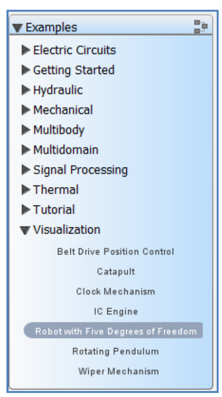
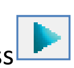


figure 2.1: examples palette

Running a Simulation

To run a simulation, press  found in the toolbar. The progress of your simulation is displayed in the Message Console. Once the simulation is completed, the results are displayed along with the 3D visualization.

Graphical Output

The graphical output displays the results that were probed in the model. Results can be manipulated much like any graph created in Maple. You can change the way results are displayed by **right-clicking** on a plot to bring up the context sensitive menu. You can also drag one plot onto another by left-clicking on the plot and dragging it into another plot area. Holding **Ctrl** will create a copy of it in the new plot area.

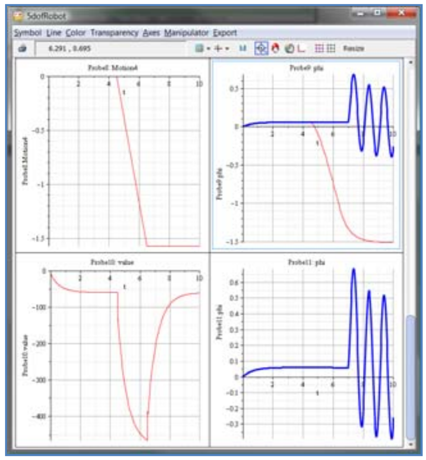


figure 2.2: graphical output

3D Visualization

For multibody models, 3D animations are created of your simulation. To view the animation, press the play button. You can customize the view of your animation by doing the following:

- Rotate: Hold **[Ctrl]** + left mouse button while moving the mouse
- Pan: Hold **[Shift]** + left mouse button while moving the mouse
- Zoom: Hold **[Alt]** + left mouse button while moving the mouse

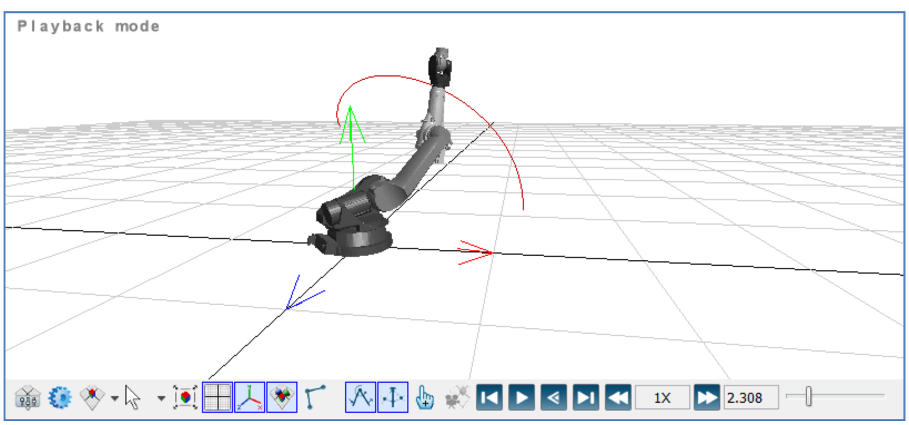


figure 2.3: 3D animation

3. Building Models: Creating a Controlled Arm

In this example, you will use multibody components to build a single link pendulum. You will then expand the example to create a multidomain controlled arm as shown below.

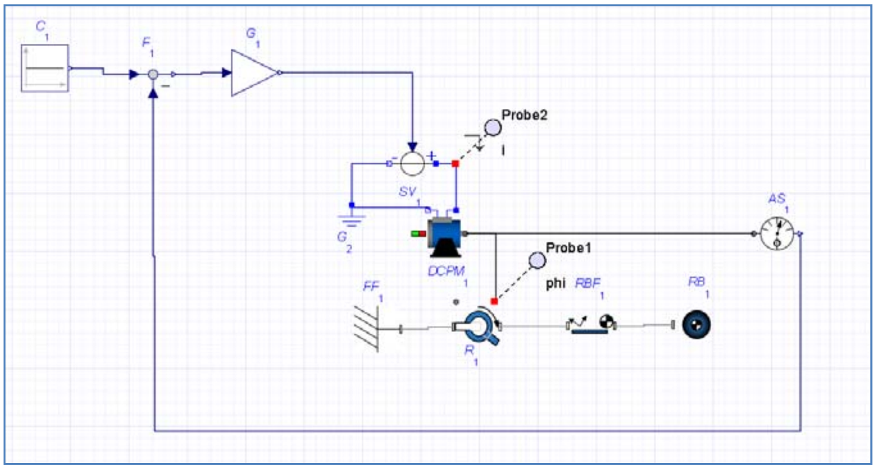
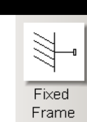
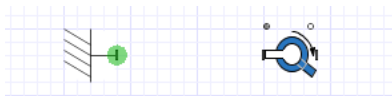


figure 3.1: finished model

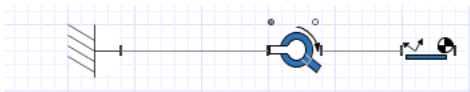
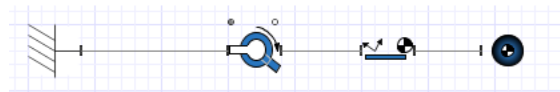
Before you begin the demo, make sure that MapleSim is in the split view that shows both the block diagram and 3-D construction environment. Click the split view button  found at the bottom left to turn on the 3-D model construction view. You will build this demo in the block diagram environment and use the 3-D view as a real-time previewer. This is one of the most common ways of using the new 3-D viewer to accelerate your model development. You could also build the multibody portions of the model in the 3-D view directly.

First, use multibody components to build a single link pendulum.

Number of Components	Component Name and Location	Symbol
1	Multibody > Bodies and Frames > Fixed Frames	
1	Multibody > Joints and Motions > Revolute	
1	Multibody > Bodies and Frames > Rigid Body Frame	
1	Multibody > Bodies and Frames > Rigid Body	



(a) *(b)* *(c)*

(d) *(e)*

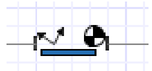
figure 3.2: connecting components

1. Drag a **Fixed Frame** component into the work area. Drag a **Revolute** component into the work area to the right of the Fixed Frame.
2. Hover the mouse over the port of the Fixed Frame (***figure 3.2a***) and a green dot will appear. Press the left mouse button.
3. Move the mouse pointer towards the left port of the Revolute (***figure 3.2b***).
4. When the mouse is over the port, a green dot will appear (***figure 3.2c***). Press the left mouse button to make the connection.
5. Drag a **Rigid Body Frame** component into the work area to the right of the Revolute.
6. Flip the Rigid Body Frame by right-clicking on the component and selecting **Flip Horizontally.**
7. Connect the Rigid Body Frame to the Revolute (***figure 3.2d***).
8. Drag a **Rigid Body** component into the work area to the right of the Rigid Body Frame.
9. Rotate the Rigid Body by right-clicking on the component and selecting **Flip Horizontally.**
10. Connect the Rigid Body to the Rigid Body Frame (***figure 3.2e****)*.

With the first section of the model complete, you will now connect a probe to record data during the simulation.

11. To connect a probe, right-click on the top right port of the Revolute and select **Attach Probe**.
12. Move the probe to the desired location and click to place the probe.
13. In the **Inspector** pane, check the box next to Angle, and change **phi** to **Angle**.

Once you have completed creating the model as shown above, change the following parameters. To change a parameter, select a component. This will open the **Inspector** tab on the right hand side of the screen. In this tab you will find all parameters associated with the given component.

Component	Parameter Change
Rigid Body Frame 	$\overline{r}_{XYZ}$=[-1, 0, 0] *m* This makes the link 1 m in length in the negative x direction from the center of mass

Running a Simulation

To run a simulation, press found in the toolbar. The progress of your simulation is displayed in the Message Console. Once the simulation is completed, the results are displayed along with the 3D visualization.

Graphical Output

The graphical output can be manipulated much like any graph created in Maple. To manipulate a graph, **right-click** on the plot to bring up the context sensitive menu.

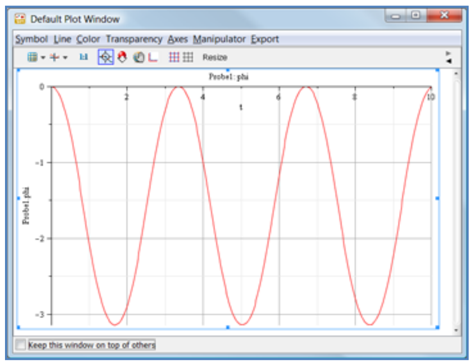


figure 3.3: graph output

3D Visualization

To view the 3D visualization animation, press . You can see the arm swinging like a pendulum, as you would expect. You can control the view of your animation through the following:

- Rotate: Hold **[Ctrl]** + left mouse button while moving the mouse
- Pan: Hold **[Shift]** + left mouse button while moving the mouse
- Zoom: Hold **[Alt]** + left mouse button while moving the mouse

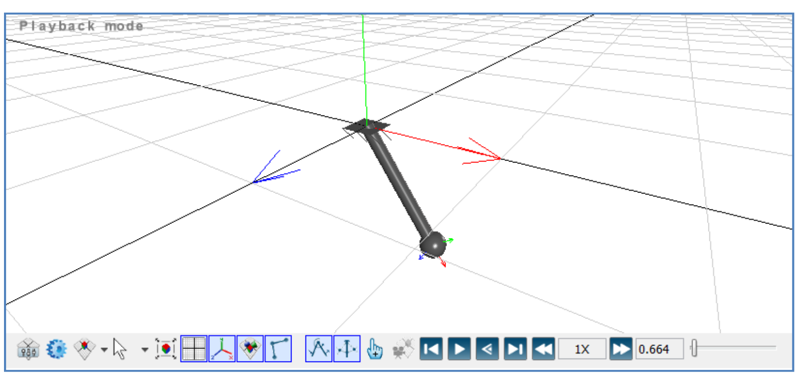


figure 3.4: animation of arm

Creating a Custom Component

You will now add friction to your model by creating a custom component.

1. Open the document folder by pressing .
2. From the dropdown menu, select **Custom Component.**
3. Change the document name to **Friction Component,** and press **Create Attachment.** Maple will now open the Custom Component Template.
4. Under **Component Description**, change the component name to **MyFriction**.

Under **Component Equations**, you will now enter in the equations to define our component, along with any parameters and initial conditions.

5. For the equation, enter $eq := [\tau(t) = f * \dot{\theta}(t)]$

To enter Greek symbols, such as θ (theta) and τ (tau), you can use the **Greek** palette.

To insert dot notation, press **[Ctrl]+[Shift]+[']** to move the cursor over the variable, then use a period to insert the dot.

6. For the parameters, enter $params := [f = 0.5]$
7. To build the system object, enter
   $sys := DynamicSystems[DiffEquation](eq, inputvariable = [\tau(t)], outputvariable = [\theta(t)])$

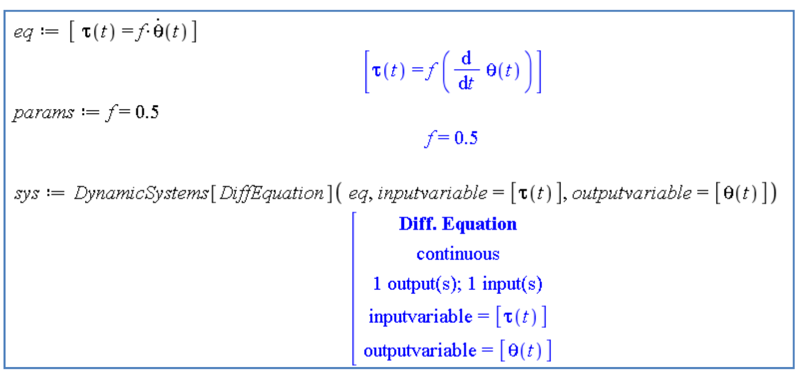


figure 3.5: custom component equations

8. Under **Component Ports**, press **Clear All Ports**. This will remove all ports from the component.
9. Press **Add Port**. Left-click on the port and drag it to the bottom of the component.

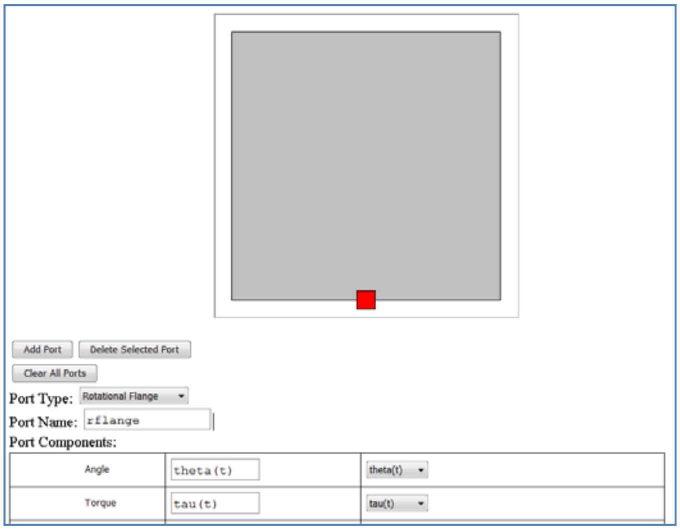


figure 3.6: defining component ports

10. With the port selected, from the **Port Type** dropdown box, select **Rotational Flange.**
11. From the **Angle** dropdown box, select **theta(t)**.
12. From the **Torque** dropdown box, select **tau(t).**
13. Press Generate MapleSim Component to create your component block. This will bring you back into the MapleSim environment.

The custom component will now appear in the Project Manager under **Library Models > User**. Drag the custom component into your model area and attach it to the top-right port of the revolute. Run the simulation. You will notice the effects of the friction component in the animation and plot.

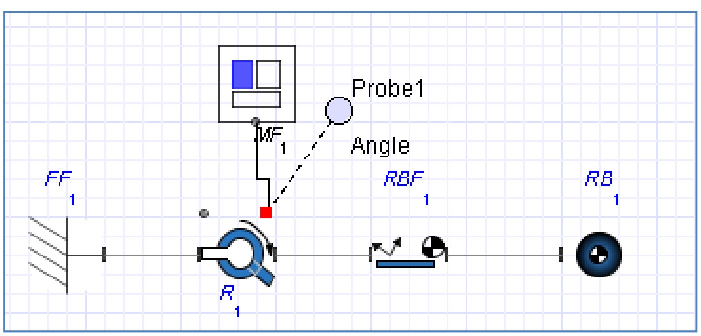
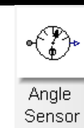
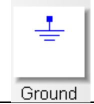
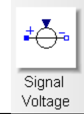
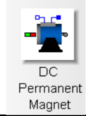
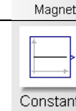
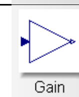
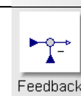


figure 3.7: model with custom component block

Completing the Model

Add the following components to your model to create a multidomain model of a controlled motor and arm.

Number of Components	Component Name and Location	Symbol
1	1-D Mechanical > Rotational > Sources > Angle Sensor	
1	Electrical > Analog > Common > Ground	
1	Electrical > Analog > Sources > Voltage > Signal Voltage	
1	Electrical > Machines > DC Machines > DC Permanent Magnet	
1	Signal Blocks > Common > Constant	
1	Signal Blocks > Common > Gain	
1	Signal Blocks > Common > Feedback	

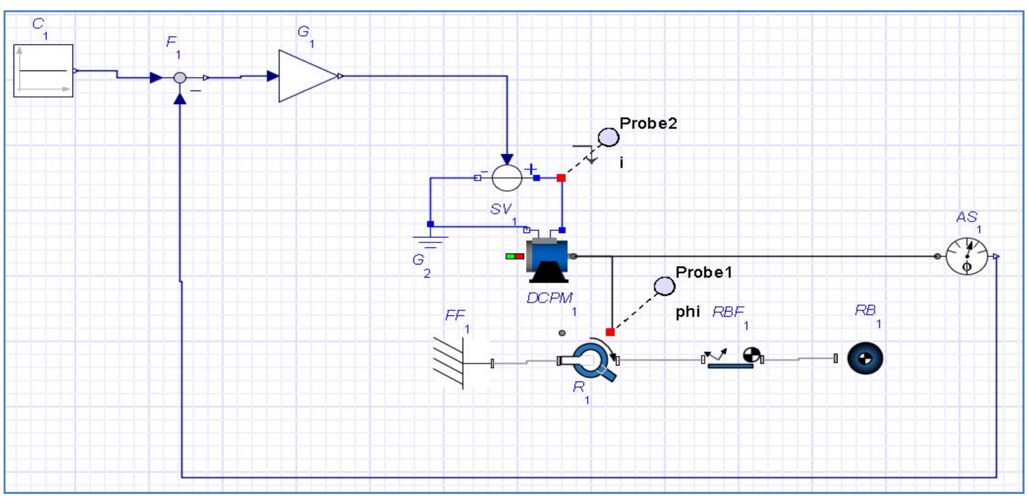
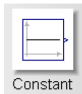


figure 3.8: model with controller

Once you have completed creating the model as shown above, change the following component parameter:

Component	Parameter Change
Constant Signal 	k=0 This specifies the controller to hold the angle the link makes with the x-axis about the z-axis as close to 0 as possible

Attach a probe to the line connecting the DC Motor to the Signal Voltage. In the **Inspector** pane, check the box next to **Current**.

Creating a Subsystem

Creating a subsystem allows you to group components togehter into a single block. This helps to organize your model both visually and by function. We will create a subsystem of the arm.

1. Drag a box around the revolute, rigid body, and rigid body frame components.
2. Press **Ctrl+G** to create the subsystem. Name the subsystem **Arm**. Press **OK**.

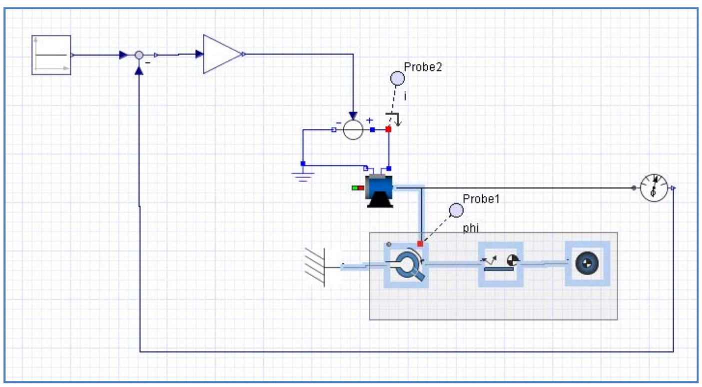


figure 3.9: creating a subsystem

You can explore the subsystem by double-clicking on it, or by using the drop-down model navigator and selecting *Arm*$_1$.

Creating a Custom Plot

Creating custom plots allows you to manage the way that your simulation results are displayed.

1. Click the **Plots** tab found in the upper right hand corner of the screen.
2. From the drop down menu, select **Add Window**.
3. Enter the window name **Current vs Angle**.

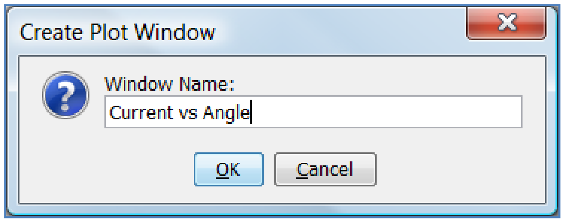


figure 3.10: entering the name of a custom plot

4. Click Empty to bring up the plot options.
5. For the X-axis, select **i** (current).
6. For the Y-axis, select **phi** (angle).
7. Press  to run the simulation.

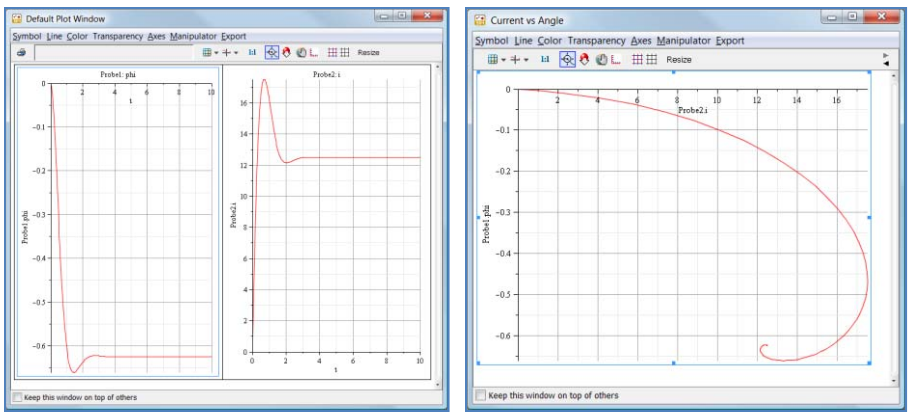


figure 3.11: default and custom plot output

Managing Results

You can quickly save simulation results to recall or export at any time using the Project tab.

1. Click on the **Project** tab located near the top left corner.
2. Expand the palette **Stored Results**. Here is where you will find the results of your previous simulation, along with any previous saved results.
3. Select **Rename this result to save it**. The Inspector tab is now displayed on the right.
4. In the Inspector window under **Result**, rename the file to **Gain=1** and press .

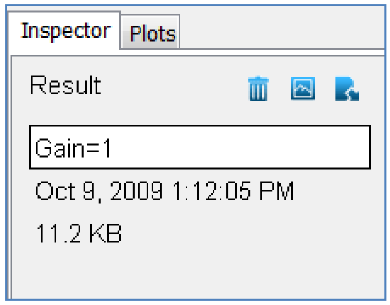
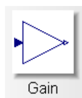


figure 3.12: saving results

You will now return to the model and change the value of the gain, run the simulation, and compare the results.

5. Change the following component parameters:

Component	Parameter Change
Gain 	k=2 Increasing the gain will reduce the error.

6. Press  to run the simulation. The new results are displayed.
7. Return to the **Project** tab and select **Gain = 1**, then under the **Inspector** tab
8. Right-click on the results **Gain=1** and select **View**. The results from this simulation should now be displayed.
9. Left-click on the line found in the **phi** plot in **Gain=1**. While selected, hold down **Ctrl** and drag the plot into the phi plot window of your last simulation run. You should now have two red plots displayed in one plot area.
10. To help distinguish between the two plots, right-click on one of the lines and select **Color > Blue**. Repeat these steps for the current plot (i). You can further manipulate your plots if you wish using the right-click options.

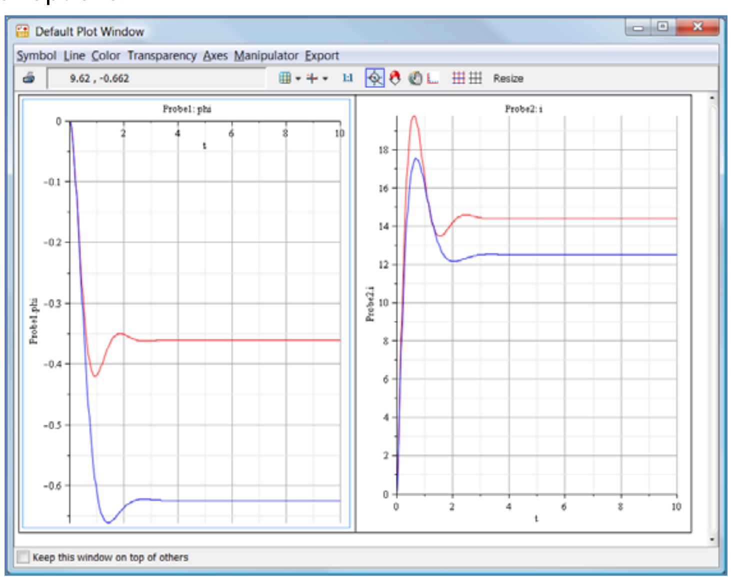


figure 3.13: comparison of results