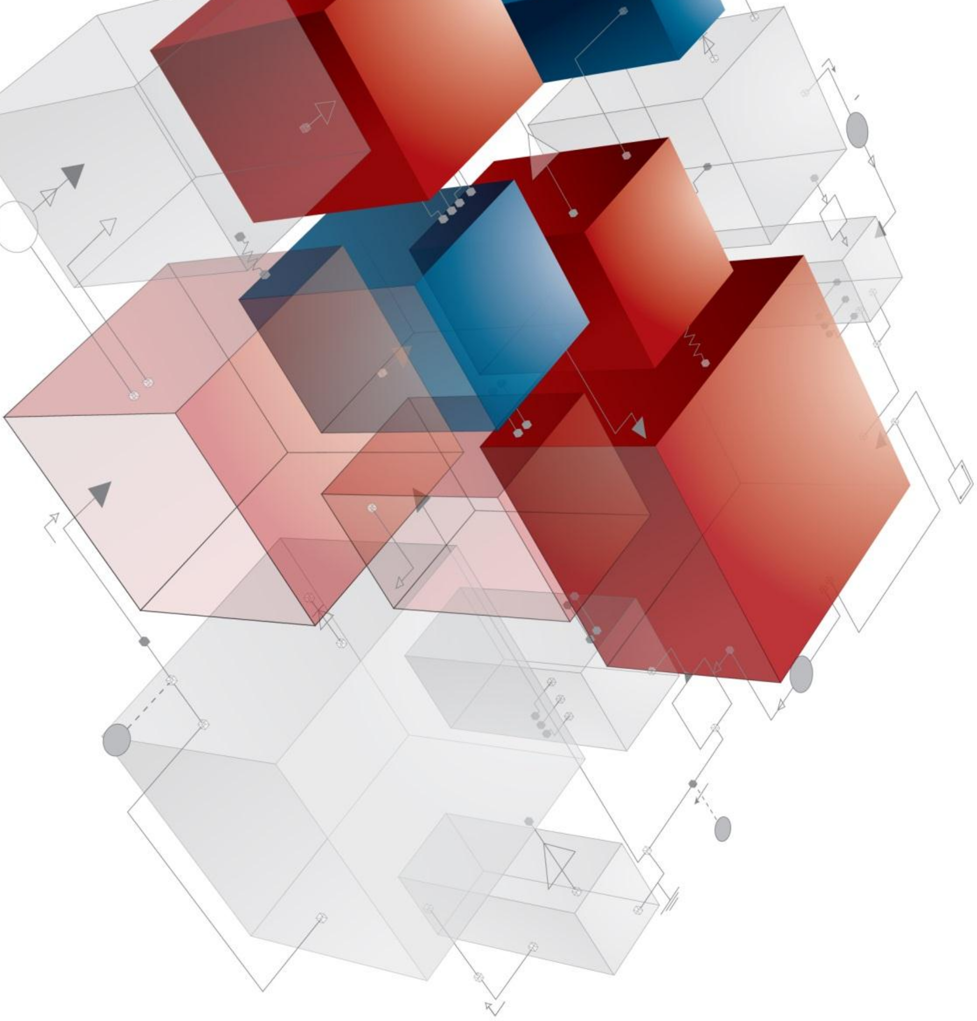


MapleSim™

Advanced System-Level Modeling

Training Manual

Table of Contents

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.

1.1 User Interface

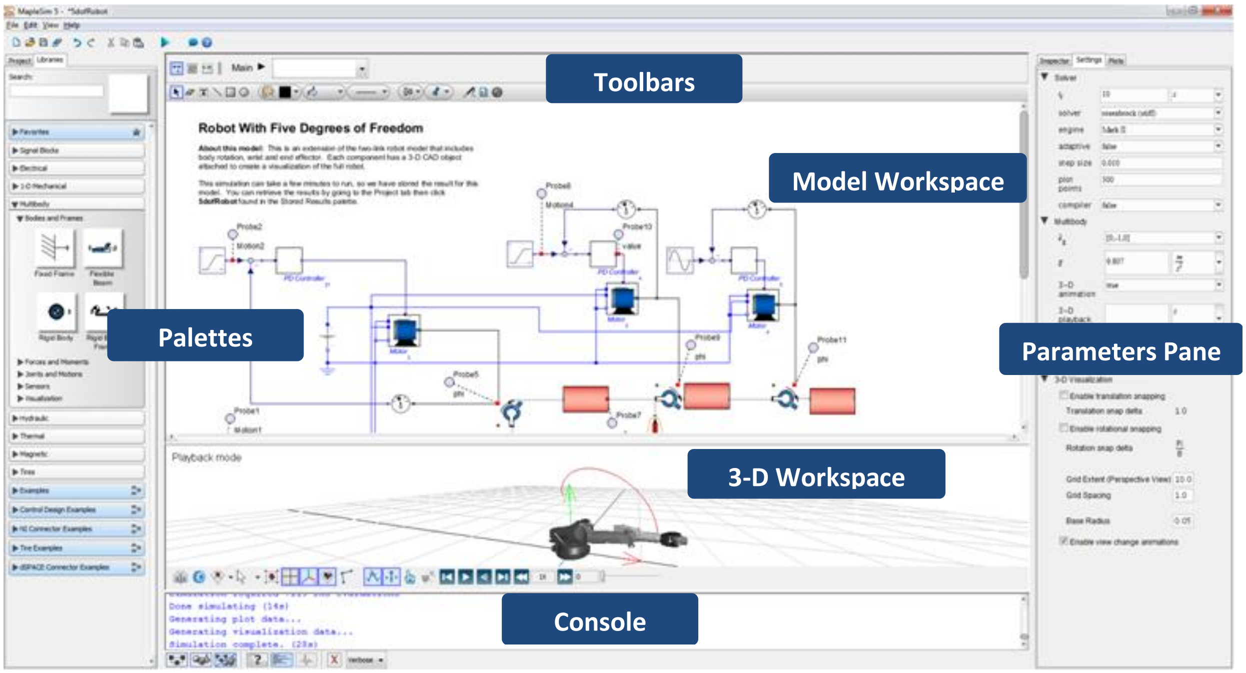


Figure 1.1. The MapleSim environment

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, **Libraries** and **Project**. The Libraries 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 simulations, navigating your model hierarchy, and attaching Maple templates.

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

The **Parameters Pane** contains three tabs, **Inspector, Settings** 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. The Settings tab allows you to 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.

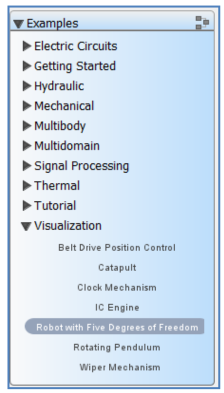


Figure 2.1. Examples palette

2.1 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.

2.2 Graphical Output

The graphical output displays the results that were probed in the model. Results can be manipulated much like any plot 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.

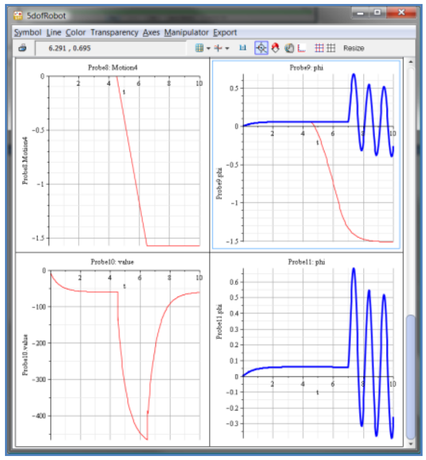


Figure 2.2. Output plots

2.3 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 (or use the mouse wheel)

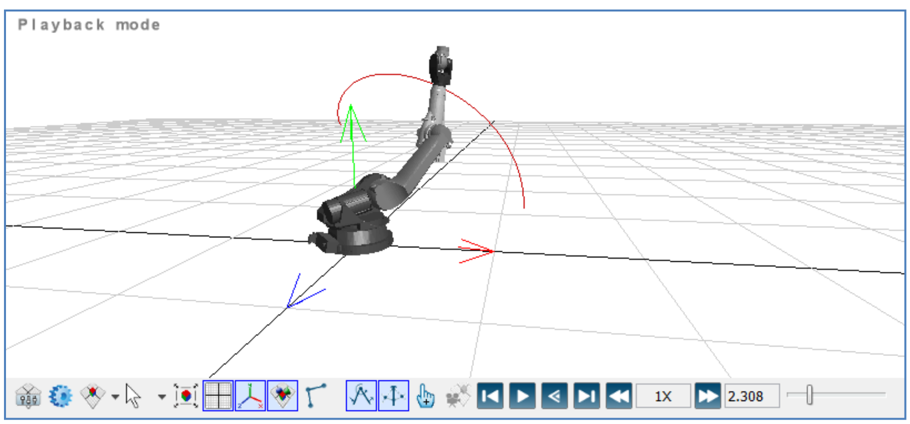


Figure 2.3. 3D animation

3 Building Models: Creating a DC Motor

In this example, you will use components from the electrical, mechanical, and signal domain to build a DC Motor. It should be noted that there are many motor models in the Electrical > Machines palette, and so the purpose of this exercise is to become familiar with the MapleSim environment.

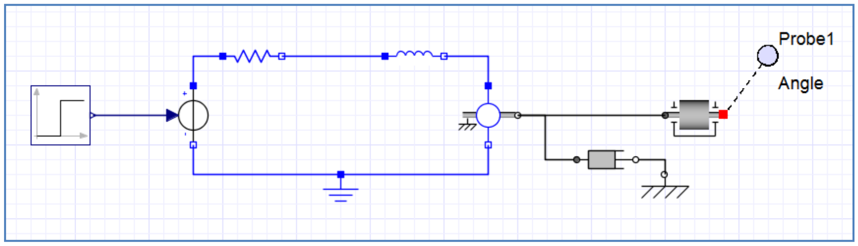
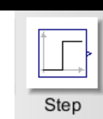
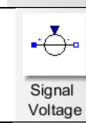
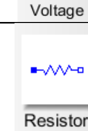
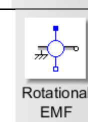
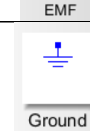
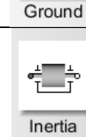
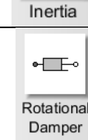
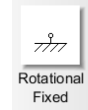
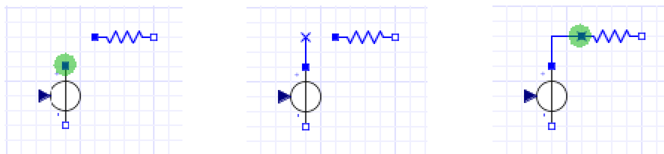


Figure 3.1. DC Motor

Before you begin the demo, make sure that MapleSim is in either the 2D view  or split view . Drag the following components into the modeling environment, either by locating them in the component library, or searching for them using the **Search** area in the same tab:

Number of Components	Component Name and Location	Symbol
1	Signal Blocks > Common > Step	
1	Electrical > Analog > Sources > Voltage > Signal Voltage	
1	Electrical > Analog > Common > Resistor	
1	Electrical > Analog > Common > Inductor	
1	Electrical > Analog > Common > Rotational EMF	
1	Electrical > Analog > Common > Ground	
1	1-D Mechanical > Rotational > Inertia	
1	1-D Mechanical > Rotational > Common > Rotational Damper	

Number of Components	Component Name and Location	Symbol
1	1-D Mechanical > Rotational > Rotational Fixed	



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

Figure 3.2. Connecting components

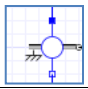
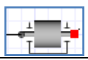
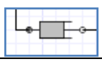
1. Drag a **Signal Voltage** component into the work area to the right of the Fixed Frame.
2. Right-click on the **Signal Voltage** and select **Rotate Clockwise**, or press [**Ctrl+r**].
3. Right-click on the **Signal Voltage** and select **Flip Horizontal**, or press [**Ctrl+h**].
4. Drag a **Resistor** into the work area to the right of the **Signal Voltage**.
5. Hover the mouse over the positive pin (blue) of the **Signal Voltage** (Figure 3.2a) and a green dot will appear. Press the left mouse button.
6. Move the mouse pointer towards the positive pin of the **Resistor** (Figure 3.2b).
7. When the mouse is over the pin, a green dot will appear (Figure 3.2**c**). Press the left mouse button to make the connection.
8. Drag an **Inductor** into the work area to the right of the **Resistor**.
9. Connect the negative pin (white) of the **Resistor** to the positive pin of the **Inductor**.
10. Drag a **Rotational EMF** component into the work area to the right of the **Inductor**.
11. Connect the negative pin of the **Inductor** to the positive pin of the **Rotational EMF**.
12. Drag a **Ground** component to the left of the **Rotational EMF**.
13. Connect the negative pin of the **Rotational EMF** to the **Ground**.
14. Connect the negative pin of the **Signal Voltage** to the **Ground**.
15. Drag a **Step** to the left of the **Signal Voltage**.
16. Connect the **Step** to the **Voltage Input**.
17. Drag an **Inertia** to the right of the **Rotational EMF**.
18. Connect the flange of the **Rotation EMF** to flange_a of the **Inertia**.
19. Drag a Rotational Damper below the Inertia.
20. Connect flange_a of the **Rotational Damper** to the flange of the **Rotational EMF**.
21. Drag a **Rotational Fixed** to the right of the **Rotational Damper**.
22. Connect flange_b of the **Rotational Damper** to the **Rotational Fixed**.

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

23. To connect a probe, right-click on the right flange of the **Inertia** and select **Attach Probe**.

24. Move the probe to the desired location and click to place the probe.
25. 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
Inductor 	$L = 0.5\ H$
Rotational EMF 	$k = 0.01\ \frac{N\ m}{A}$
Inertia 	$J = 0.01\ kg\ m^2$
Rotational Damper 	$d = 0.1\ \frac{N\ m\ s}{rad}$

3.1 Running a Simulation

To run a simulation, press  found in the toolbar. The progress of your simulation is displayed in the Message Console.

3.1.1 Graphical Output

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

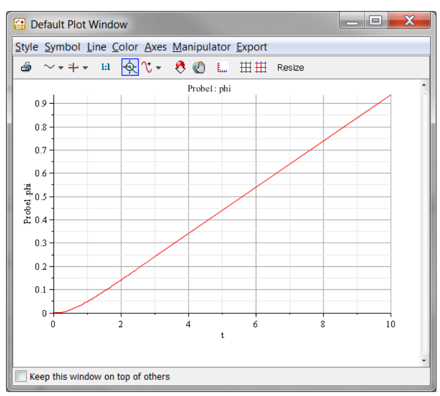


Figure 3.3. Output plot

4 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.

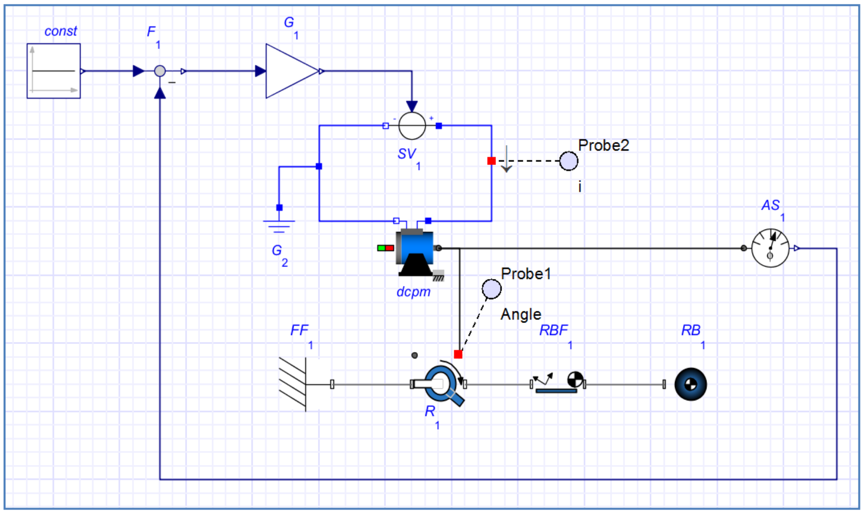
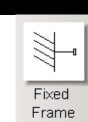
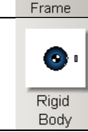


Figure 4.1. Complete model

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	

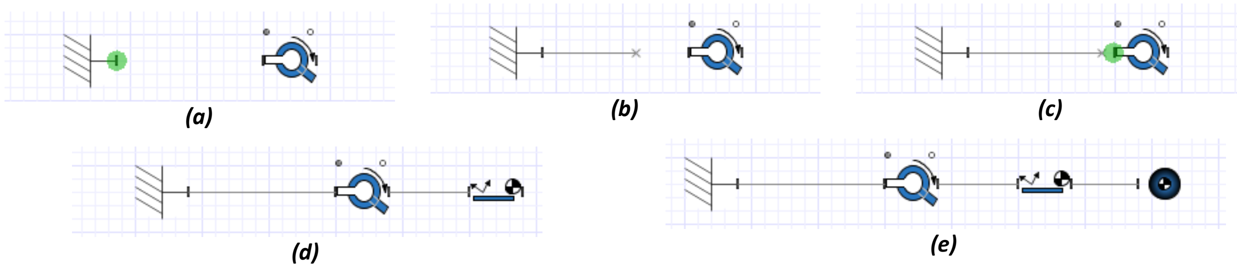


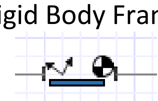
Figure 4.2. Connecting components

26. Drag a **Fixed Frame** component into the work area. Drag a **Revolute** component into the work area to the right of the Fixed Frame.
27. Hover the mouse over the port of the Fixed Frame (Figure 4.2a) and a green dot will appear. Press the left mouse button.
28. Move the mouse pointer towards the left port of the Revolute (Figure 4.2b).
29. When the mouse is over the port, a green dot will appear (Figure 4.2c). Press the left mouse button to make the connection.
30. Drag a **Rigid Body Frame** component into the work area to the right of the Revolute.
31. Flip the Rigid Body Frame by right-clicking on the component and selecting **Flip Horizontal.**
32. Connect the Rigid Body Frame to the Revolute (Figure 4.2d).
33. Drag a **Rigid Body** component into the work area to the right of the Rigid Body Frame.
34. Rotate the Rigid Body by right-clicking on the component and selecting **Flip Horizontal.**
35. Connect the Rigid Body to the Rigid Body Frame (Figure 4.2e*).*

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

36. To connect a probe, right-click on the top right port of the Revolute and select **Attach Probe**.
37. Move the probe to the desired location and click to place the probe.
38. 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 	$\bar{r}_{XYZ}$=[-1, 0, 0] *m* This makes the link 1 m in length in the negative x direction from the center of mass

4.1 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.

4.1.1 Graphical Output

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

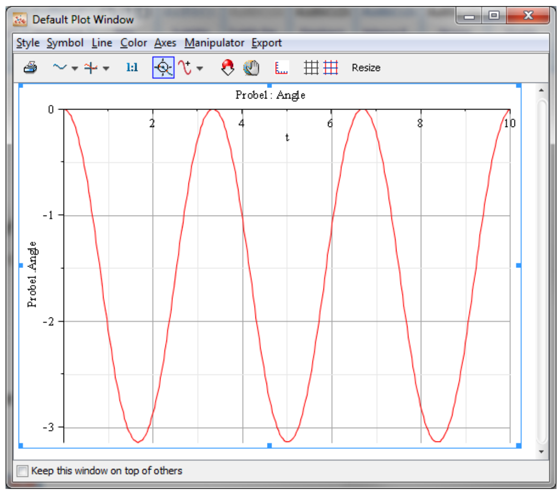


Figure 4.3. Output plot

4.1.2 3D Visualization

To view the 3D visualization animation, press ▶. You can see the arm swinging like a pendulum. 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 (or use the mouse wheel)

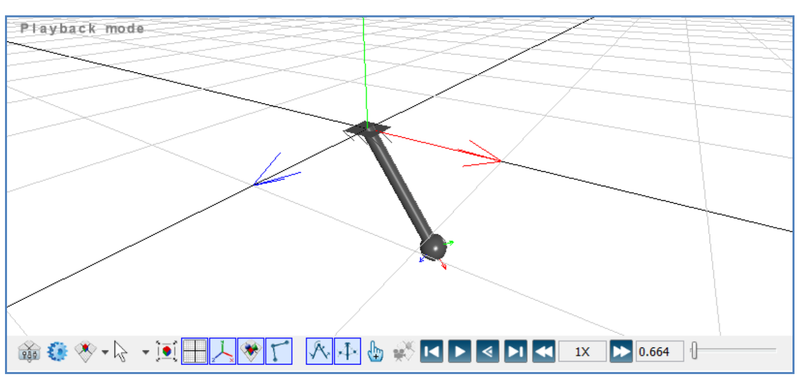


Figure 4.4. Animation of arm

4.2 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 $params := [f = 0.5]$

$$eq := [\tau(t) = f \cdot \dot{\theta}(t)]$$

$$params := [f = 0.5]$$

$$initialconditions := [\ ]$$

$$eq := [\tau(t) = f\dot{\theta}(t)]$$

$$params := [f = 0.5]$$

$$initialconditions := [\ ]$$

Figure 4.5. Custom component creation

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

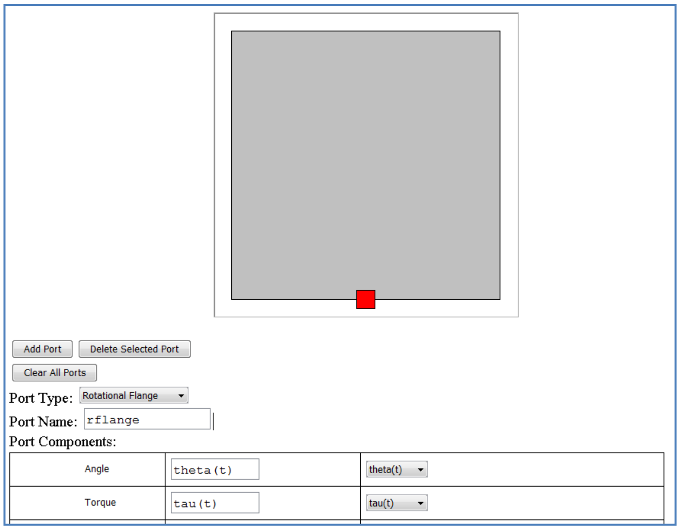


Figure 4.6. Defining component ports

9. With the port selected, from the **Port Type** dropdown box, select **Rotational Flange.**
10. From the **Angle** dropdown box, select **theta(t)**.
11. From the **Torque** dropdown box, select **tau(t).**
12. 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.

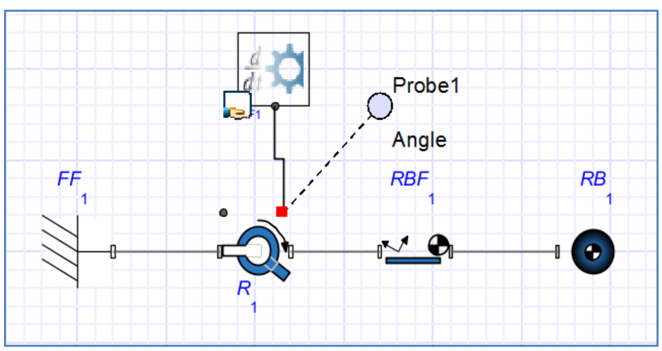
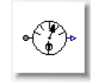
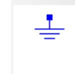
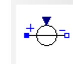
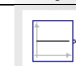
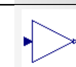
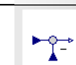


Figure 4.7. Model with custom component block

4.3 Completing the Model

Remove the custom component from your model, and add the following components 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	 Angle Sensor
1	Electrical > Analog > Common > Ground	 Ground
1	Electrical > Analog > Sources > Voltage > Signal Voltage	 Signal Voltage
1	Electrical > Machines > DC Machines > DC Permanent Magnet	 DC Permanent Magnet
1	Signal Blocks > Common > Constant	 Constant
1	Signal Blocks > Common > Gain	 Gain
1	Signal Blocks > Common > Feedback	 Feedback

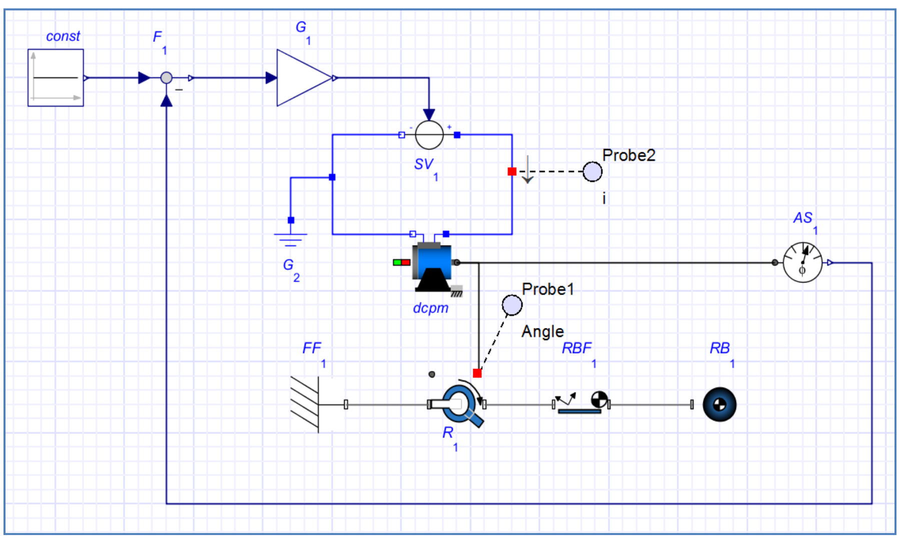
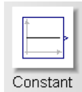


Figure 4.8. Model with controller

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**.

4.4 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**.

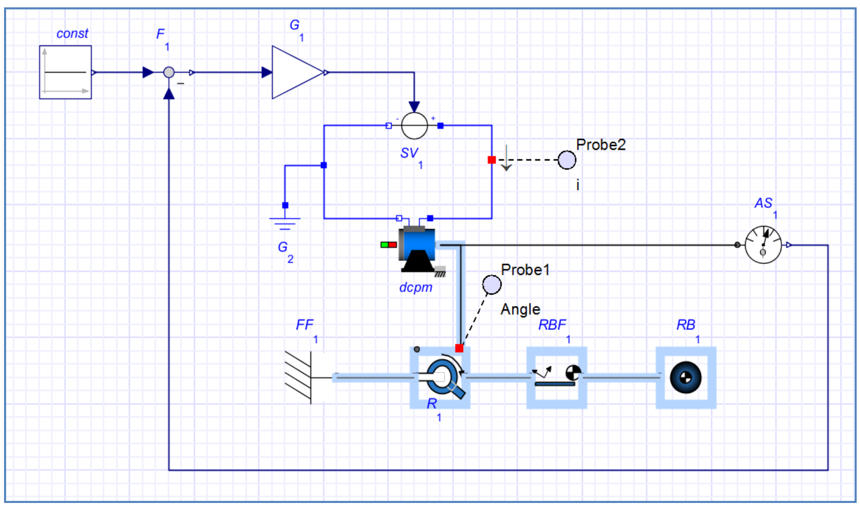


Figure 4.9. Creating a subsystem

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

4.5 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**.

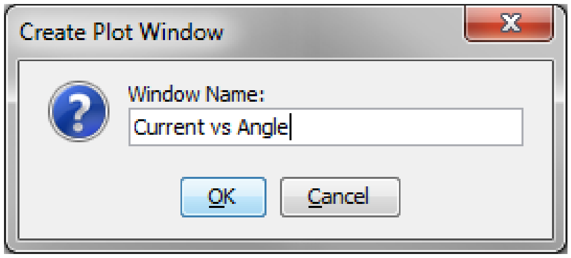


Figure 4.10. Entering the name of a custom plot

4. Click Empty to bring up the plot options.
5. For the X-axis, select **Probe2: i** (current).
6. For the Y-axis, select **Probe1: Angle**.
7. Press  to run the simulation.

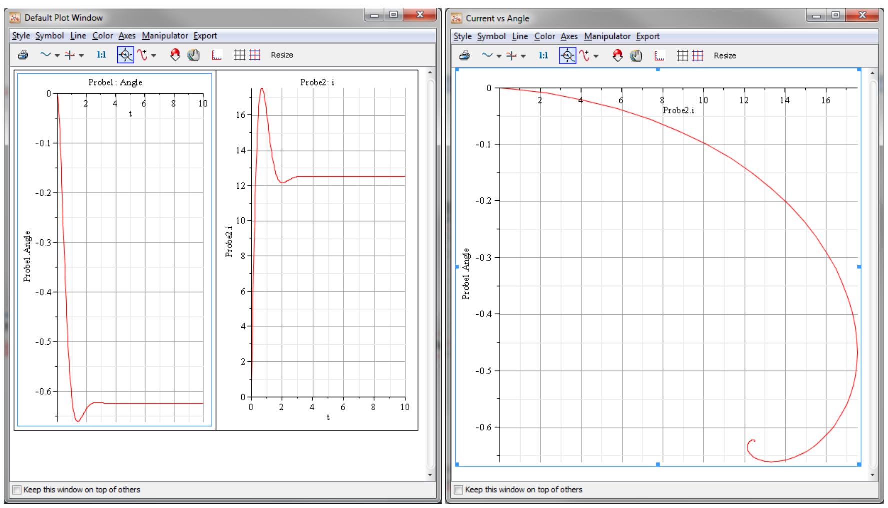


Figure 4.11. Default and custom plot output

4.6 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 .

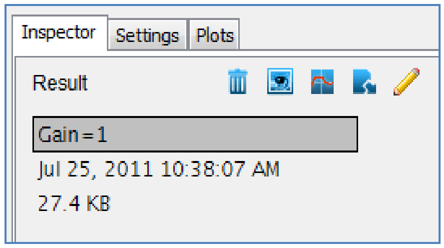
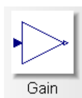


Figure 4.12. Saving results

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**.
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 **Angle** 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.

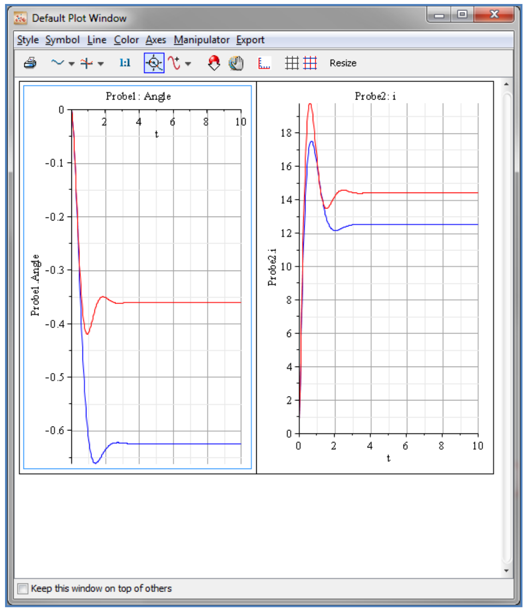


Figure 4.13. Comparison of results

5 Building Models: Creating a Slider-Crank

Use multibody components to build a double pendulum.

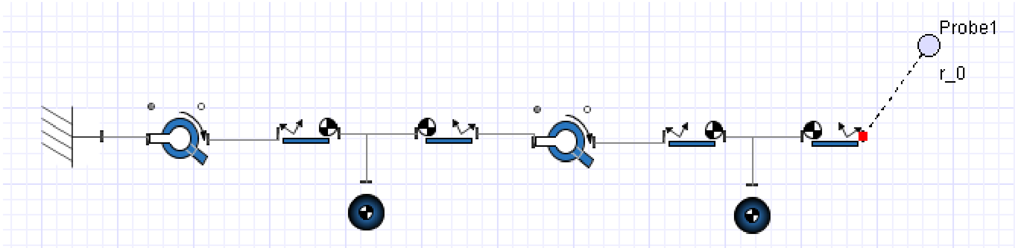
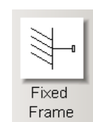


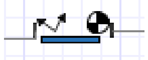
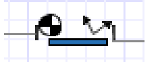
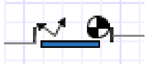
Figure 5.1. Double pendulum

1. In a new model environment, drag the following components into the model workspace:

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

Note: some components require you to rotate or flip them. To do so, right-click on a component and select the required manipulation.

2. Connect the components as shown in Figure 5.1
3. Change the following component parameters:

Component	Parameter Change
1st Rigid Body Frame 	$\bar{r}_{XYZ}$=[-1/2, 0, 0]
2nd Rigid Body Frame 	$\bar{r}_{XYZ}$=[1/2, 0, 0]
3rd Rigid Body Frame 	$\bar{r}_{XYZ}$=[-1, 0, 0]

4. To connect a probe, right-click on the top right port of the fourth Rigid Body Frame and select **Attach Probe**.
5. Check the boxes next to **Length[1]** and **Length[2]**.
6. Press **OK**.
7. Move the mouse to the location that you would the probe, then left-click to place the probe.
8. Click the **Plots** tab found in the upper right hand corner of the screen.
9. From the drop down menu, select **Add Window**.
10. Enter the window name **End Point Motion.**
11. Click Empty to bring up the plot options.
12. For the X-axis, select **Probe 1: r_0[1].**
13. For the Y-axis, select **Probe 1: r_0[2]**.
14. Press  to run the simulation.

5.1 Finishing the Model

You can now modify the pendulum into a slider crank driven by a torque component.

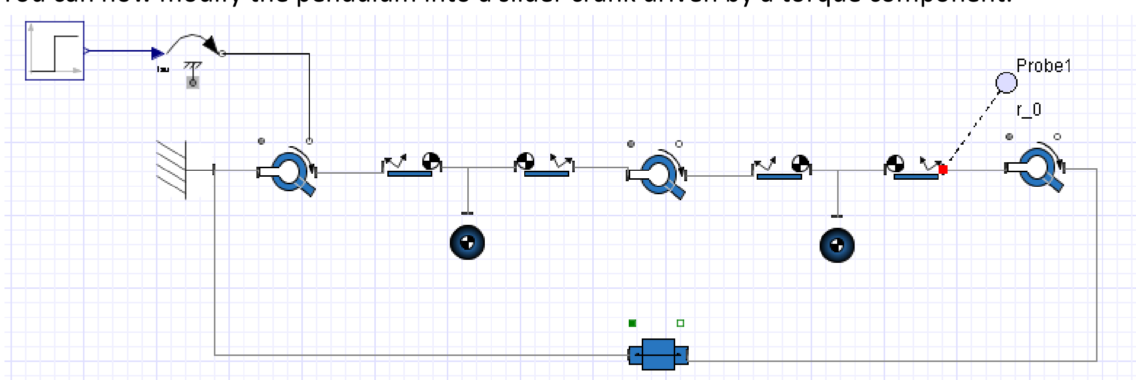
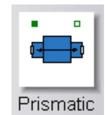
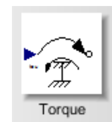
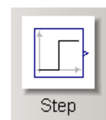
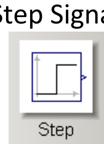


Figure 5.2. Slider-crank

Number of Components	Component Name and Location	Symbol
1	Multibody > Joints and Motions > Revolute	
1	Multibody > Joints and Motions > Prismatic	
1	1-D Mechanical > Rotational > Torque Drives > Torque	
1	Signal Blocks > Common > Step	

For the Step Signal, change the **height** to 10.

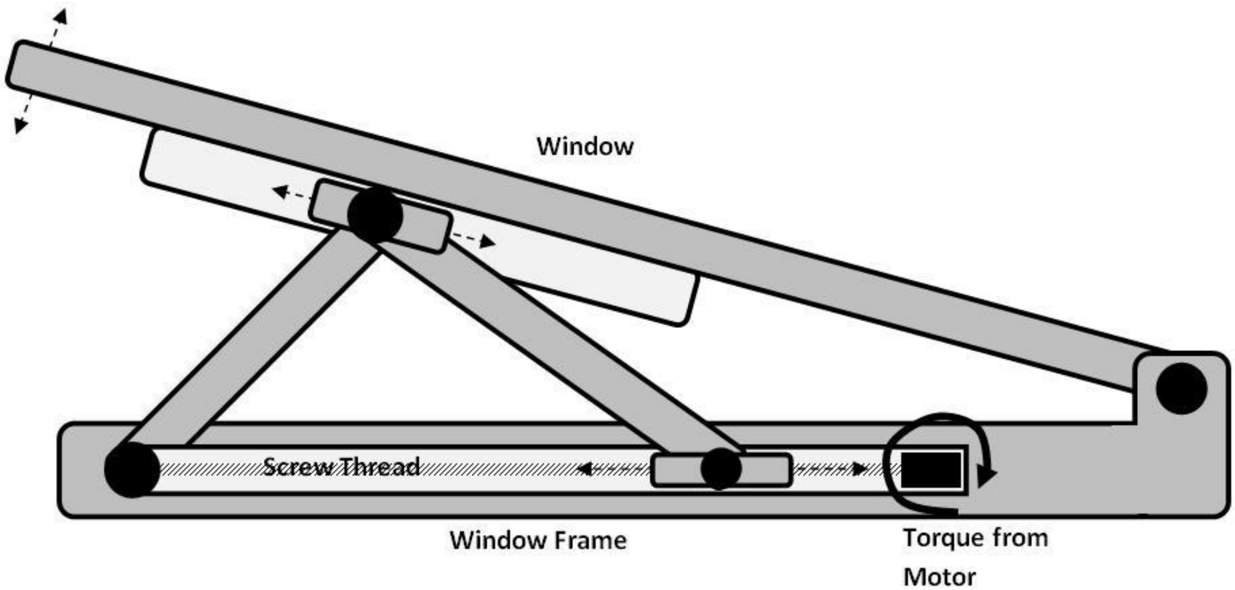
Component	Parameter Change
Step Signal 	*height* = 10

Press  to run the simulation.

6 Design Project: Motorized Window Assistant

6.1 Introduction

The aging European demographic demands that homes are designed with the elderly and infirm in mind. The opening and closing of a window may seem trivially easy, but can often represent a challenge to senior citizens. An engineer has designed a motorized window frame that can be easily retrofitted to existing windows.



6.2 Tasks

Create a MapleSim model of this system and generate a 3D animation. Make reasonable assumptions about unspecified design parameters.

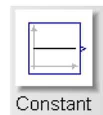
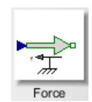
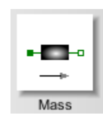
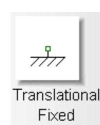
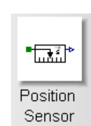
Use a proportional controller to regulate the torque applied to open or close the window to a set point. Consider using a DC Permanent Magnetic motor and regulating the applied voltage for this purpose.

Tune the controller so that the window opens or closes in a reasonable amount of time.

7 Generating Parameterized Transfer Functions from Plant Models in MapleSim

In this tutorial you will generate a parameterized transfer function from a spring-mass-damper system. You will start by creating the plant model, then extract the system equations in a Maple worksheet and convert them into a transfer function.

1. In a new model environment, drag the following components into the model workspace:

Number of Components	Component Name and Location	Symbol
1	Signal Blocks > Common > Constant	
1	1-D Mechanical > Translational > Common > Force	
1	1-D Mechanical > Translational > Common > Mass	
1	1-D Mechanical > Translational > Common > Translational Spring Damper	
1	1-D Mechanical > Translational > Common > Translational Fixed	
1	1-D Mechanical > Translational > Sensors > Position Sensors	

2. Connect the model as shown in Figure 7.1**.** For help on building your model, see 4*. Building Models: Creating a Controlled Arm*.

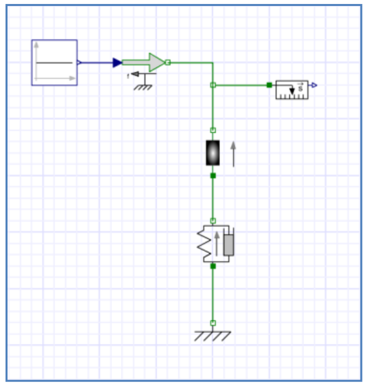
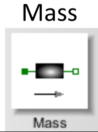
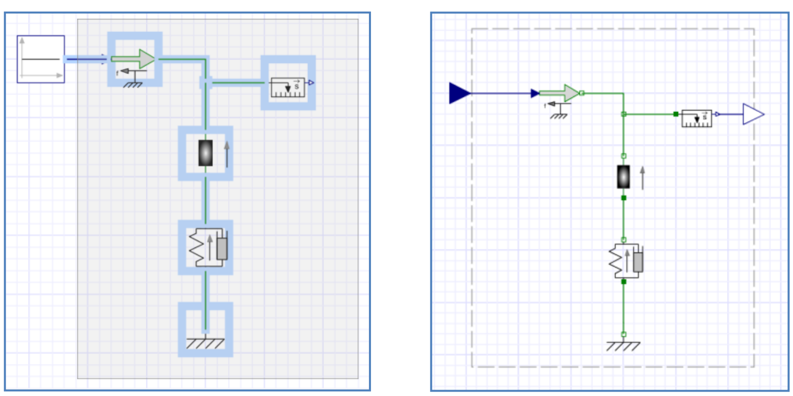


Figure 7.1. Sliding mass-spring -damper model

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

Component	Parameter Change
Translational Spring Damper  	c = 3000 N/m d = 300 N s/m This changes the spring stiffness and damping constant
Mass  	m = 100 kg Changes the mass for the component

3. Drag a box around all of the components, excluding the constant signal.



(a) *(b)*

Figure 7.2. a) Creating a subsystem and b) connecting components

4. Press **Ctrl + G** to create the subsystem. Name the subsystem **Plant**. Press **OK**. Note that the input to the subsystem is named u1(t)

5. Double-click on the **Plant** subsystem. This will bring you to the subsystem level.
6. Connect the Position Sensor to the frame of the subsystem by left-clicking on the right port of the sensor and connecting it to the frame of the subsystem (Figure 7.2***b***). This will make the port accessible at the Main level.
7. Click **Main** in the subsystem browser to return to the top level of the model.
8. Right-click on the right-hand port on the Plant subsystem and select **Attach Probe**.
9. Change ***value*** to ***Position***. Press **OK**.
10. Press  to run the simulation.

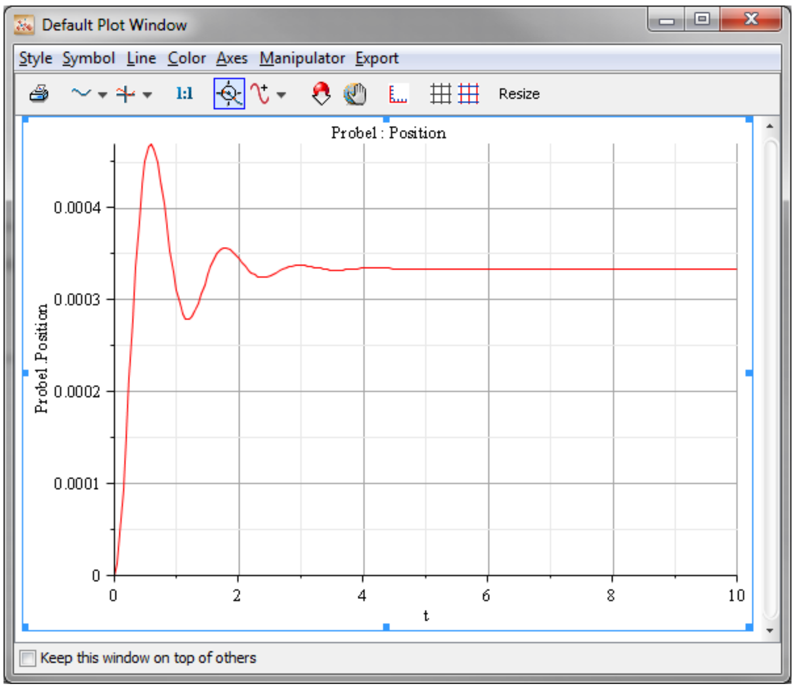


Figure 7.3. Simulation results

You will now attach an equation template to your model so you can access the model equations.

11. Click  to open the **Document Folder.**
12. Selection **Equations**, and click **Create Attachment**. This will now launch Maple.
13. Select the Plant subsystem in the MapleSim component, and click **Load Selected Subsystem**
14. Under **Parameter and Variable Manipulation**, click **Toggle Symbolic**
15. Click **Reassign Equations**

16. Scroll down to **View Core Equations**. These are the dynamic equations for the Plant subsystem and have been assigned to the variable **DAEs**.

$$\left[\begin{array}{c} M1\_m\,M1\_a(t) + SD1\_c\,SD1\_s\_rel(t) - SD1\_c\,SD1\_s\_rel0 + SD1\_d\,SD1\_v\_rel(t) - u1(t) = 0 \\ \frac{d}{dt} SD1\_s\_rel(t) = SD1\_v\_rel(t) \\ \frac{d}{dt} SD1\_v\_rel(t) = M1\_a(t) \end{array}\right]$$

Figure 7.4. Equation generation template

17. Scroll to the bottom of the worksheet and create new execution groups by pressing **Ctrl+K** a few times.
18. Excecute the following code to convert the equation system into a transfer function

```
sys := DynamicSystems[TransferFunction](DAEs, [u1(t)],
[SD1_s_rel(t)]);
```

$$sys := \left[\begin{array}{c} \textbf{Transfer Function} \\ \text{continuous} \\ \text{1 output(s); 1 input(s)} \\ \text{inputvariable} = [u1(s)] \\ \text{outputvariable} = [SD1\_s\_rel(s)] \end{array}\right.$$

```
sys:-tf;
```

$$\left[\frac{1}{M1\_m\,s^2 + SD1\_d\,s + SD1\_c}\right]$$

Figure 7.5. Code (red) with output (blue) to convert equations to transfer function

The first line converts the equations into a transfer function, whose input is the force, *u1*(*s*), and whose output is the displacement of the mass, *SD1_s_rel*(*s*).

8 Creating Custom Components

8.1 Understanding Custom Components

When you create custom components based on physical connections, each port has two variables associated with it; the *across variable* and the *through variable*.

The through variable has a direction: flow into the component is positive, and flow out of the component is negative.

Let's assume that you wanted to model a simple resistor. This would requires several variables

i(t)	Current
R(t)	Resistance
V(t)	Voltage difference
Vleft(t)	Voltage on the left port
Vright(t)	Voltage on the right port

These are the equations that would be implemented

$$v(t) = Vright(t) - Vleft(t)$$

$$v(t) = I(t)\, R(t)$$

This is how the equations would be mapped to ports on the custom component

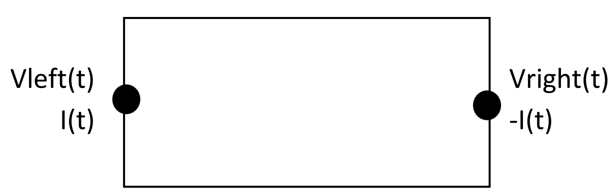


Figure 8.1. How variables map to physical ports in a resistor

The current on the right port has a negative sign; this is because it represents flow *out* of a component. Conversely, the current on the left port is positive this is because it represents flow *into* a component

8.2 Modeling a Temperature-Dependent Resistor

In this tutorial, you will create a temperature dependent resistor whose resistance varies as

$$r(t) = 0.1\,T(t)^2$$

1. In a new model, click to **View Document Folder.**
2. Select the template **Custom Component**. Press **Create Attachment**. This will now launch Maple.
3. Under **Component Description**, change the component name to **TempResistor**.

Under **Component Equations**, you will now enter in the equations to define your component, along with any parameters and initial conditions. Be sure to hit enter at the end of each line.

4. For the equation, enter $eq := [v(t) = vp(t) - vn(t),\ r(t) = 0.1 * T(t)^2,\ v(t) = i(t) * r(t),$ $q(t) = i(t) * v(t)]$
5. For the parameters, enter $params := [\,]$
6. For the initial conditions, enter $initialconditions := [\,]$

$$eq := [v(t) = vp(t) - vn(t), r(t) = 0.1 \cdot T(t)^2, v(t) = i(t) \cdot r(t), q(t) = i(t) \cdot v(t)]$$

$$eq := [v(t) = vp(t) - vn(t), r(t) = 0.1\,T(t)^2, v(t) = i(t)\,r(t), q(t) = i(t)\,v(t)]$$

$$params := [\,]$$

$$params := [\,]$$

$$initialconditions := [\,]$$

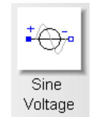
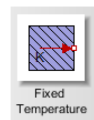
$$initialconditions := [\,]$$

Figure 8.2. Custom component equations

7. Scroll down to **Component Ports**
8. Under **Component Ports**, press **Clear All Ports**. This will remove all ports from the component.
9. Press **Add Port** three times.
10. With the **bottom** port selected, from the **Port Type** dropdown box, select **Heat Port.**
11. From the **Temperature** dropdown box, select **T(t)**.
12. From the **Heat Flow Rate** dropdown box, select **q(t).**
13. With the **right** port selected, from the **Port Type** dropdown box, select **Negative Pin.**
14. From the **Voltage** dropdown box, select **vn(t)**.
15. From the **Current** dropdown box, select **i(t)** and change this to **–i(t).**
16. With the **left** port selected, from the **Port Type** dropdown box, select **Positive Pin.**
17. From the **Voltage** dropdown box, select **vp(t)**.
18. From the **Current** dropdown box, select **i(t).**
19. 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 **Definitions > Components**. Drag the custom component into your model area.

20. Under the Project tab, drag the following components into your workspace:

Number of Components	Component Name and Location	Symbol
1	Electrical > Analog > Common > Sine Voltage	
1	Electrical > Analog > Common > Ground	
1	Electrical > Analog > Common > Inductor	
1	Thermal > Boundary Condition Controls > Fixed Temperature	

21. Connect the model as shown in **figure 7.2** and attach the appropriate probes. For help on building your model, see Section 3.

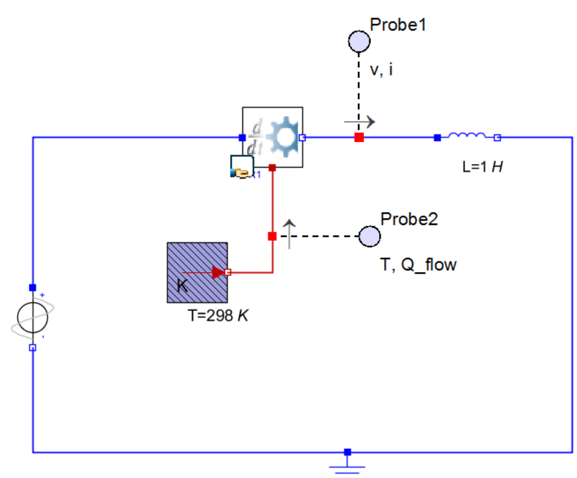
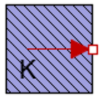


Figure 8.3. Complete model with custom component

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

Component	Parameter Change
Fixed Temperature 	T = 298 K

22. Press  to run the simulation.

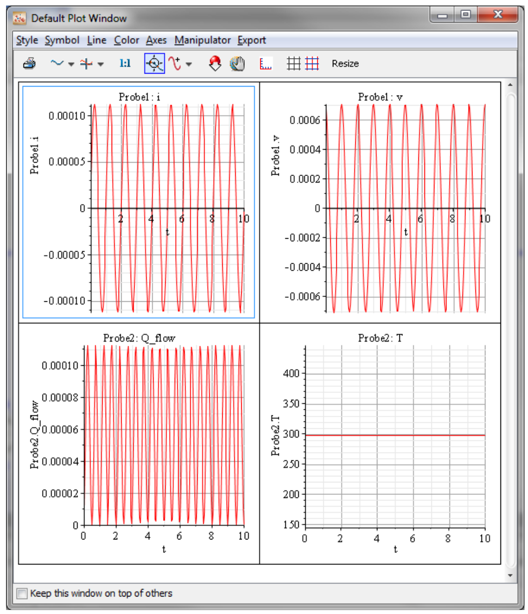


Figure 8.4. Simulation results

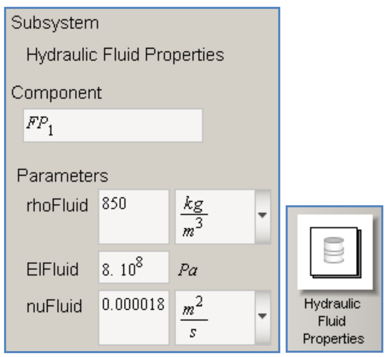
9 Understanding Hydraulics in MapleSim

9.1 Introduction

This section describes how to model hydraulic systems in MapleSim. While the hydraulic blocks are designed primarily to convert hydraulic flow into mechanical motion, they can also be used to model pure hydraulic flow.

9.1.1 Basic Hydraulic Blocks

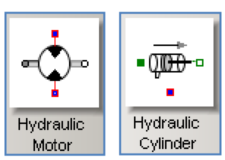
Hydraulic Fluid Properties



All hydraulic models need a Hydraulic Fluid Properties block. It defines the properties of the hydraulic fluid, and is essentially a Parameter block.

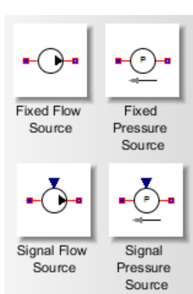
- rhoFluid: liquid density
- ElFluid: Bulk Modulus (this defines the compressibility of the fluid)
- nuFluid: Kinematic Viscosity (the dynamic viscosity divided by the liquid density)

Hydraulic Actuator



You need an actuator to convert hydraulic flow into motion of a mechanical body. MapleSim offers a Hydraulic Cylinder (for translational motion) and a Hydraulic Motor (for rotational motion).

Hydraulic Source



You can either specify the flowrate or the pressure of the hydraulic source. If a Pressure Source is used, then MapleSim balances the load in the hydraulic system against the pressure source to find the flowrate.

9.2 Example 1 – Simulating Translational Motion with a Fixed Flowrate Source

We will start by creating this model. It converts flow from a fixed pressure source to translational motion.

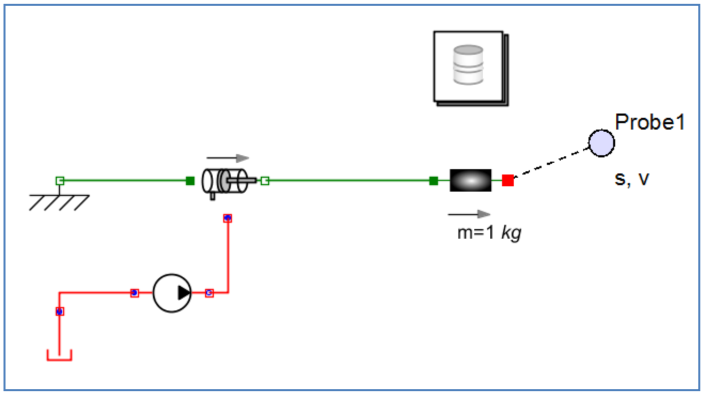
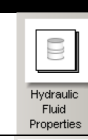
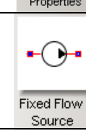
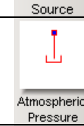
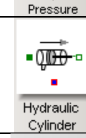
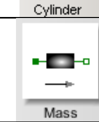
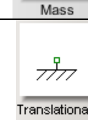


Figure 9.1. Basic hydraulic piston with constant flow rate

Number of Components	Component Name and Location	Symbol
1	Hydraulic > Reference Components > Hydraulic Fluid Properties	
1	Hydraulic > Sources > Fixed Flow Source	
1	Hydraulic > Reference Components > Atmospheric Pressure	
1	Hydraulic > Actuators > Hydraulic Cylinder	
1	1-D Mechanical > Translational > Common > Mass	
1	1-D Mechanical > Translational > Common > Translational Fixed	

Note that the Hydraulic cylinder has a cross-sectional area A of 1 m^2, while the Fixed Flowrate source has a flow Q of 1 $m^3\ s^{-1}$.

The sliding mass will hence be pushed along by the cylinder at a speed of

$$v = \frac{Q}{A} = \frac{1\,m^3\,s^{-1}}{1\,m^2} = 1\,m\,s^{-1}$$

This is confirmed by running the simulation and probing the speed of the sliding mass.

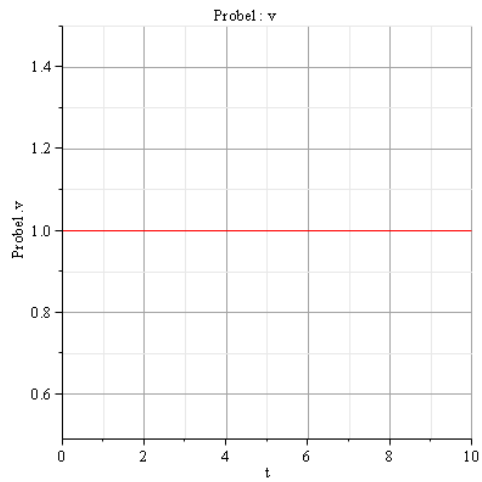


Figure 9.2. Simulation results for constant flow rate

9.3 Example 2 – Simulating Translational Motion with a Fixed Pressure Source

Now replace the Fixed Flow Source with a Fixed Pressure Source

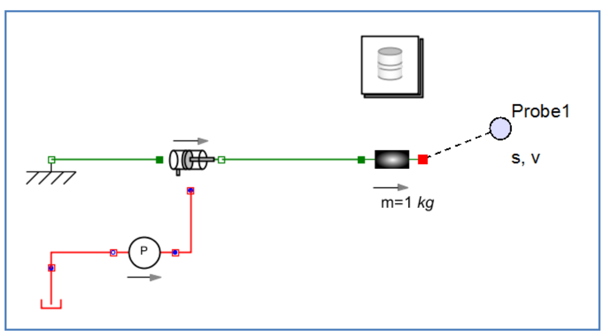
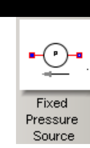


Figure 9.3. Basic hydraulic piston with constant pressure

Number of Components	Component Name and Location	Symbol
1	Hydraulic > Sources > Fixed Pressure Source	

The force on the Sliding Mass is equal to the cross-sectional of the hydraulic cylinder A multiplied by the pressure P of the hydraulic fluid.

$$F = A \times P = 1m^2 \times 1Pa = 1N$$

The acceleration of the Sliding Mass is giving by

$$F = m\,a$$

$$1N = 1kg \times a$$

Therefore

$$a = 1\,m\,s^{-1}$$

Again, this is confirmed by probing the acceleration, speed and displacement of the Sliding Mass

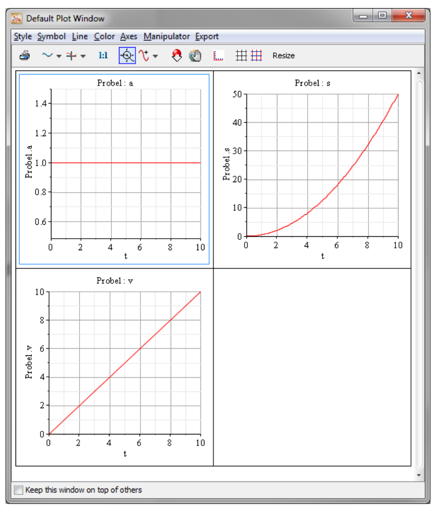


Figure 9.4. Simulation results for constant pressure

9.4 Understanding Pressure Losses in a Hydraulic Network

Pressure must be applied to overcome internal frictional effects within the liquid (in laminar flow), and the effect of the surface roughness of the pipe (in turbulent flow).

In laminar flow, the internal frictional f effect is determined by the following equations

$$f = \frac{64}{Re}$$

$$Re = \frac{D\,V}{\nu}$$

where D is the pipe diameter, V is the fluid velocity, and ν is the dynamic viscosity.

In turbulent flow, the frictional effects of the surface roughness of the pipe are characterised by

$$f = \frac{1}{\left(1.8\, log10\left(\frac{6.9}{Re} + \left(\frac{\epsilon}{3.7\, D}\right)^{1.11}\right)\right)^2}$$

In either laminar or turbulent flow, the frictional losses have to be balanced against the applied pressure to determine the flowrate.

9.5 Example 3 – Modeling the Flowrate through a Pipe

Figure 9.5 is a constant pressure applied to a pipe. We wish to know the resulting flow rate.

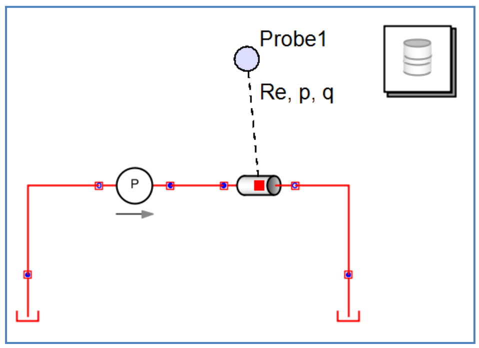
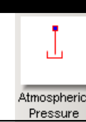
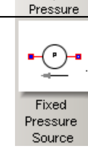
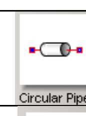
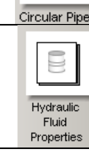


Figure 9.5. Constant pressure applied to a pipe

Number of Components	Component Name and Location	Symbol
2	Hydraulic > Reference Components > Atmospheric Pressure	
1	Hydraulic > Sources > Fixed Pressure Source	
1	Hydraulic > Pipes and Valves > Circular Pipe	
1	Hydraulic > Reference Components > Hydraulic Fluid Properties	

Running the MapleSim model predicts the following flowrate – about 3.2 x 10^{-9} m^3 s^{-1}.

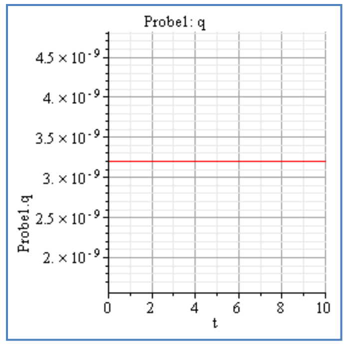


Figure 9.6. Simulation results for flow rate

9.5.1 Hand Calculations and Results

If we were to analyze this system by hand, we would apply the Bernoulli Equation.

$$\frac{\Delta P}{\rho\, g} = f\, \frac{L}{D}\, \frac{V^2}{2\, g}$$

Assuming that the system is in laminar flow, then $f = \frac{64}{Re}$ and hence

$$\frac{1}{850 \times 9.81} = \frac{64}{\frac{0.01 \times V}{0.000018}} \times \frac{5}{0.01} \times \frac{V^2}{2 \times 9.81}$$

$$\Rightarrow V = 0.0000408\, m\, s^{-1}$$

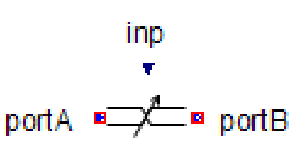
Hence

$$Q = V \times \frac{\pi\, D^2}{4} = 0.0000408 \times \frac{\pi \times 0.01^2}{4} = 3.21 \times 10^{-9}\, m^3 s^{-1}$$

This is the same value given by MapleSim. Using the calculated value of V gives Re=0.02. This is far less than the critical value of 2000, and hence we have confirmed the system is in laminar flow.

9.6 Spool Valves and controlling the path of fluid flow

Spool valves allow you to switch flow path from one part of a model to another. A spool valve has three ports.



The top port accepts a signal input that is equal to the open valve area. By regulating valve area, we can switch flow on or off.

The left and right ports are hydraulic connectors

Spool valves can be used with Boolean blocks to switch fluid flow from one pipe to another. For example, the model in Figure 9.7 switches flow from the top leg to the bottom leg when the simulation time reaches five seconds.

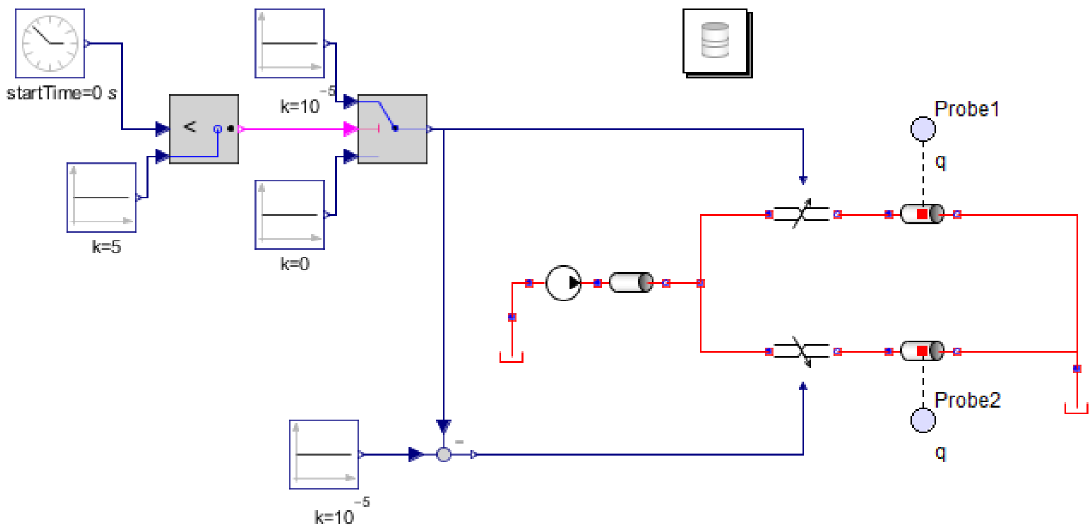


Figure 9.7. Model of a hydraulic switching circuit