Getting Started with System Generator for DSP

Lab 1 - Using Simulink

XILINX®

Using Simulink Lab

Introduction

In this lab you will learn the basics of Simulink. You will use a Simulink blockset to generate a simple design and take it through simulation. You will then change the sampling settings to see its effect on the output. You will then learn how to create a subsystem.

Objectives

After completing this lab, you will be able to:

- Use the Simulink tool to create a simple design
- Create a subsystem and simulate
- Describe the effect of the sampling period

Lab Setup

Please check the System Generator release notes to insure that the proper versions of the ISE Design Suite and MATLAB are installed on your machine. Failure to have the proper tool versions installed may result in unexpected behavior.

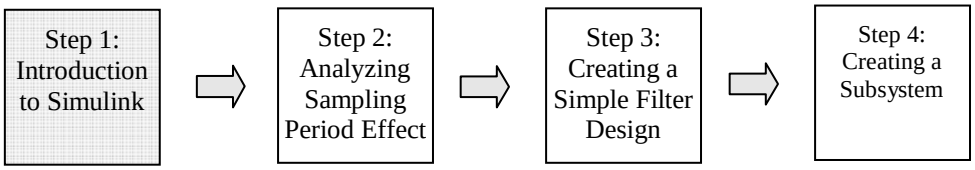
Procedure

This lab has four primary steps. In Step 1, you are introduced to the Simulink environment. In Step 2, you will analyze the effect of the sampling period. In Step 3, you will create a simple filter

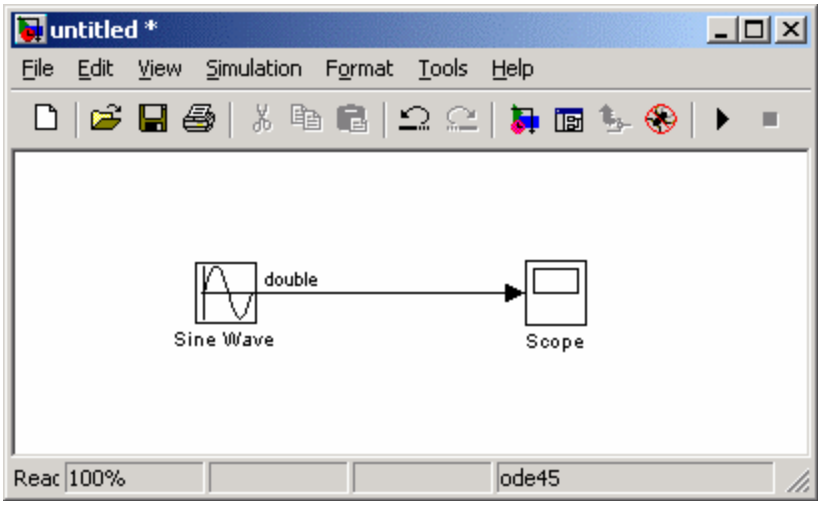
design using a Simulink blockset. Finally, in Step 4, you will create a subsystem of the design and perform simulation.

Introduction to Simulink — Step 1

General Flow for this Lab:



In this step you will become familiar with the MATLAB and Simulink environments (software tools from The MathWorks suite). You will create a blank worksheet, add a **Sine Wave** source element, add a **Scope** sink element, and wire the two, as shown in the figure below:

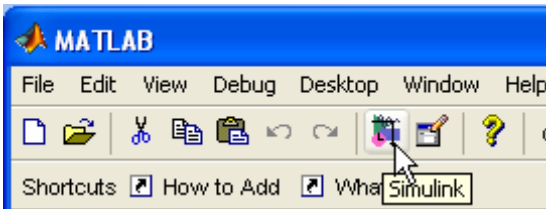


❶ Select **Start → Programs → MATLAB → MATLAB** or double-click the **MATLAB** shortcut on the desktop if available.

❷ Navigate to the **lab1** folder from the MATLAB command prompt line. For example, find the **sysgen** subdirectory under the ISE Design Suite tree (specified by the $XILINX environment variable), then proceed to the folder **...sysgen/examples/getting_started_training/lab1**.

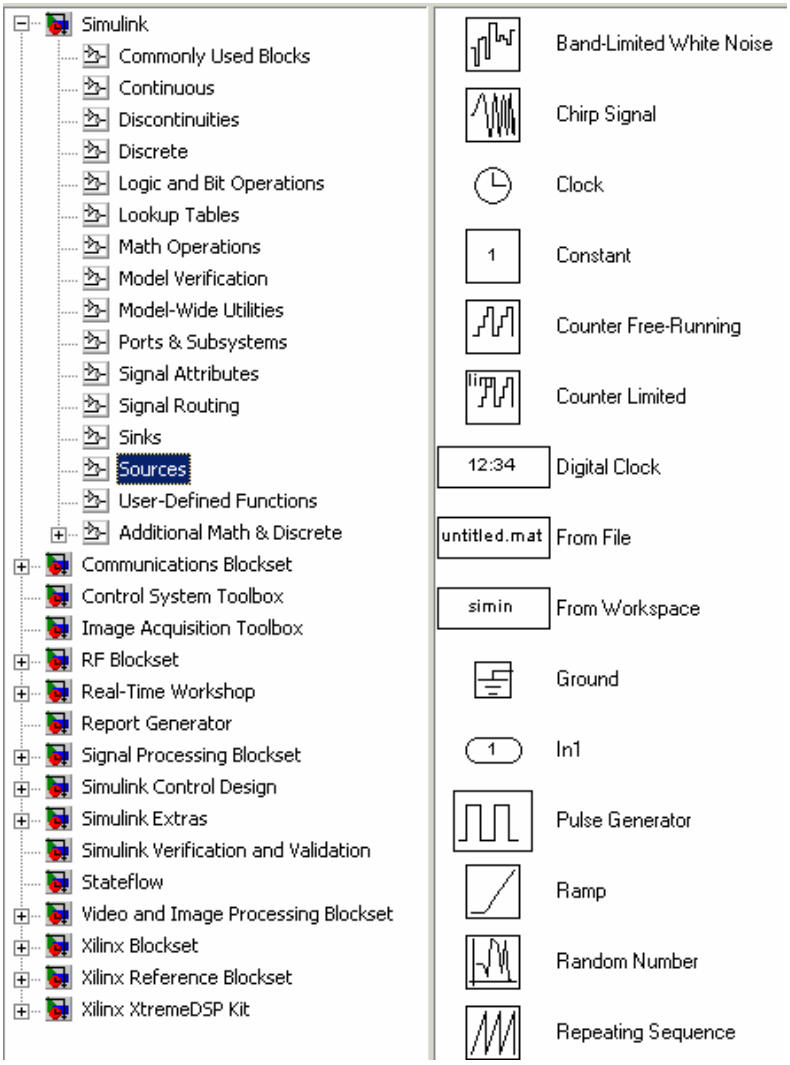
You can view the directory contents in the MATLAB Current Directory window, or type *ls* at the command line prompt. Many UNIX-type shell commands work from the MATLAB command window.

❸ Type ***simulink*** at the MATLAB command prompt or click the **Simulink** button in the MATLAB toolbar to open the Simulink Library Browser.

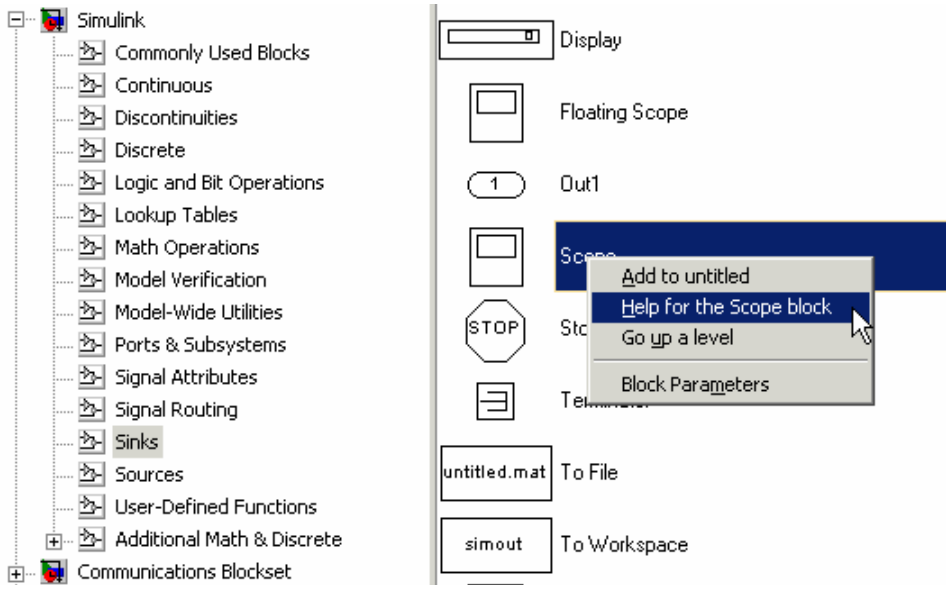


❹ Examine the available blocks in the Simulink Library Browser

The following elements, among others, should appear:
- Simulink (sources and sinks)
- Xilinx Blockset
- Xilinx Reference Blockset
- Xilinx XtremeDSPKit

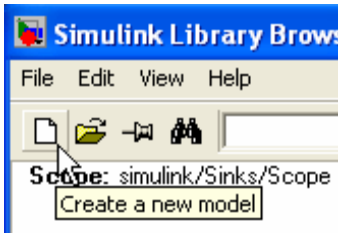


❺ As shown below, right-click on any **block** in the Library Browser and select **Help** from the MATLAB menu



This provides details about the block. You can also select **Help** from the Xilinx blockset elements.

❻ As shown below, click the **New model** button in the Simulink Library Browser to create a new model blank sheet



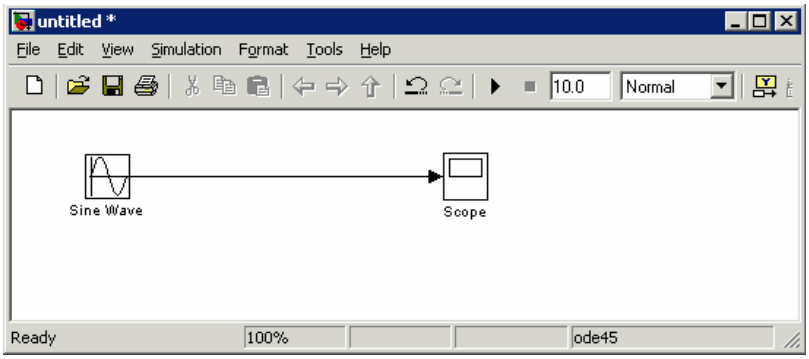
❼ In the Library Browser window, expand the **Simulink Library** and click **Sources**

❽ Scroll through the library to find the **Sine Wave** source. Select and drag **Sine Wave** onto the worksheet

❾ From the Simulink Browser, select **Simulink** → **Sinks**, add the **Scope** block, and draw a **wire** from the **Sine Wave** to the **Scope** block. An automatic block connection tip appears.

If you are performing this task for the first time, select the **Do not show this message again** option and click **Close**

Note: To draw a wire, move the cursor to the source output port (the cursor will become a cross-hair). Click and drag your mouse to an input port of destination. Alternatively, you can select a source block by clicking once on that block, moving the cursor over to the destination block, pressing Ctrl, and clicking on the destination port. With this method, the first available output port of the source block will be connected to the first available input port of the destination block.



Next, you will assign a 2*pi*(1/150) frequency to the **Sine Wav**e element and show port data types. You will change the stop time of the simulation to 150 and set the Solver options.

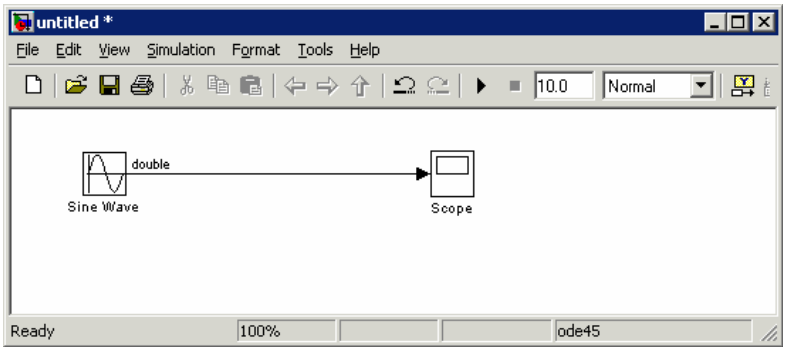
❶ Double-click the **Sine Wave** block.

The Block Parameters dialog box opens.

❷ Change the frequency to **2*pi*(1/150)**, as shown below, then click **OK** to close the dialog box.

❸ On the worksheet, select **Format** → **Port/Signal Displays** → **Port Data Types**

The signal type is displayed on the wire, as shown below:



❹ From the project sheet, select **Simulation** → **Configuration Parameters**.

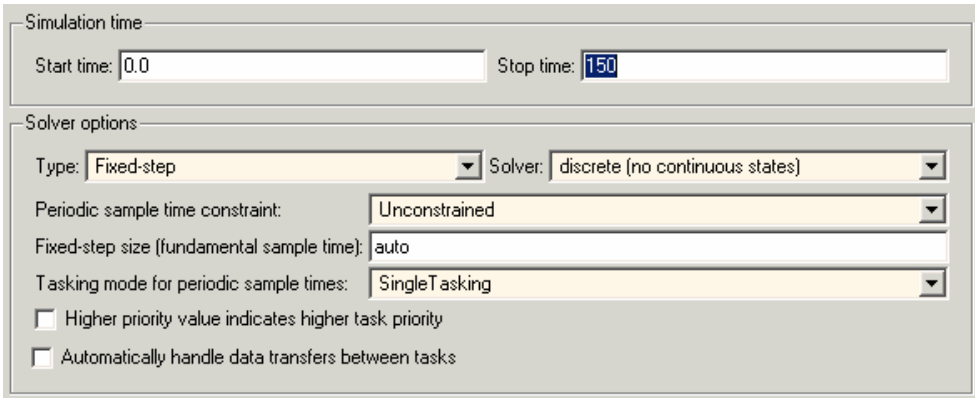
❺ From the **Configuration Parameters** dialog box, enter **150** in the Stop time field, and set the following Solver options:

Type: **Fixed-step**

Solver: **Discrete (no continuous states)**

Tasking mode: **SingleTasking**

Setting these parameters allows your simulation to run for 150 time units.

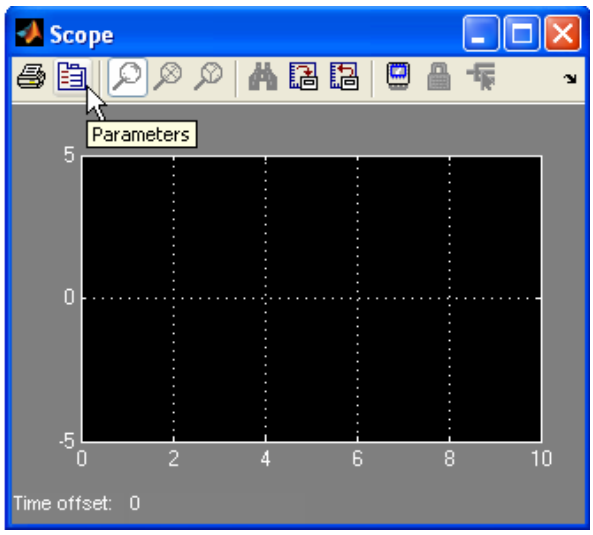


❻ Click **OK**

Next, you will parameterize the **Scope** block and run the simulation.

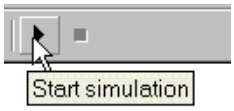
❶ Double-click the **Scope** block.

❷ Click the **Scope Parameters** button.

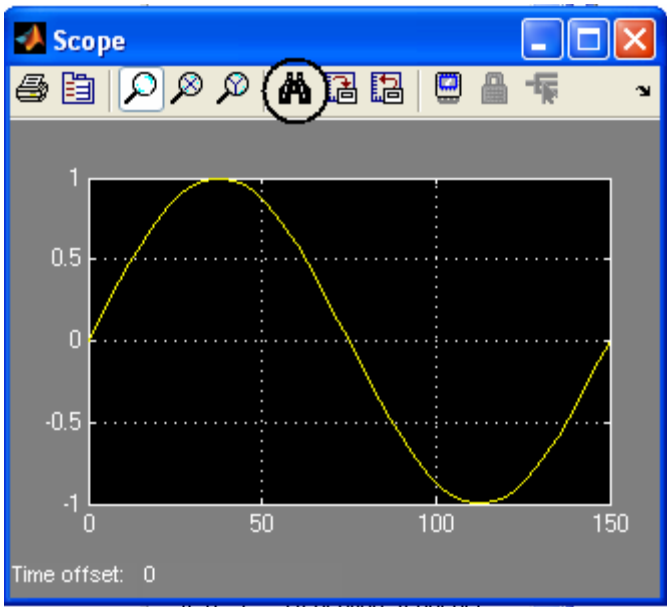


❸ In the Scope Parameters dialog box, set the **Time range** to **150**, then click **OK.**

❹ Run the simulation. From your Simulink project worksheet, select **Simulation → Start** or click the **Start Simulation** button



❺ On the **Scope** display, click the **Autoscale** button so that the output will fit into the Scope

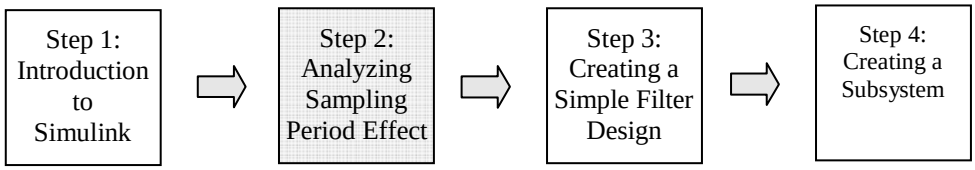


❻ View the Scope output

A *smooth* sine wave should fit into your scope window, which is what you would expect because you are running a double-precision software simulation.

Analyzing the Sampling Period Effect — Step 2

General Flow for this Lab:

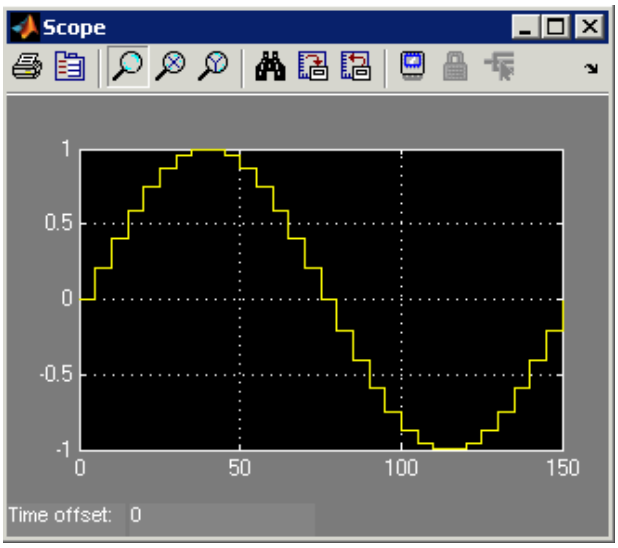


In this step you will run the simulation at a slower sampling period (sample period of 5) of the sine wave source and analyze the sampling period effect. You will then change the stop time to 500, run the simulation, and observe the output. Finally, you will change the stop time back to 150.

❶ Double-click the **Sine Wave** block to open its parameter dialog box.

❷ Change the **Sample time** of the Sine Wave source to **5**, click **OK**, and run the **simulation**.

Note: As the **Sample time** is increased (fewer times sampled), the quantization error is increased.



❸ Type **500** in the project sheet toolbar window and press **Enter** to change the simulation stop time.

❹ Run the **simulation** and observe the output in the scope window.

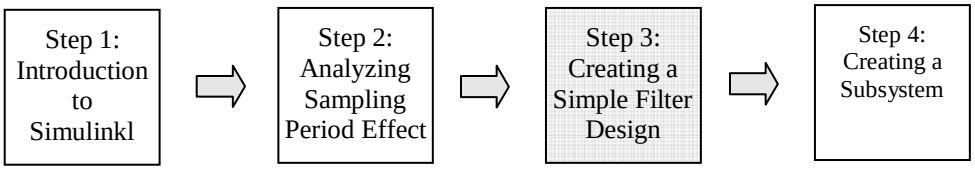
You may have to click the **Autoscale** button to see three complete cycles.

❺ Change the **simulation stop time** back to 150.

❻ Change the **Sample time** of the Sine Wave source back from 5 to 0 and click **OK**.

Creating a Simple Filter Design Using Simulink Blocks Step 3

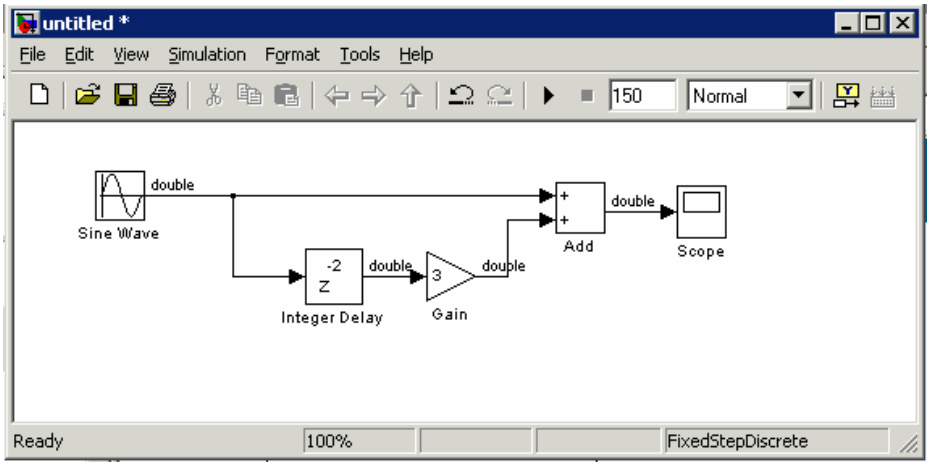
General Flow for this Lab:



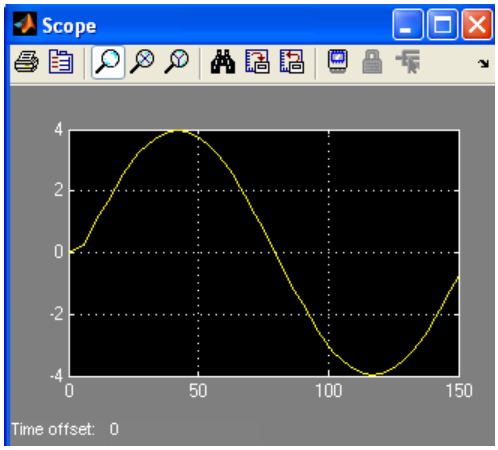
In this step you will build a simple design to implement the following function by using the appropriate blocks from the Simulink blockset.

$Y(n+1) = X(n+1) + 3 * X(n-1)$.

1. From the Simulink Library Browser, select the **Discrete** library. Select and add the **Integer Delay** block to the design.
2. Double-click the **Integer Delay** block and change the Number of delays to **2**.
3. From the Simulink Library Browser, select the Math Operations library. Select and add the **Gain** block to the design.
3. Double-click the **Gain** block and change the Gain to **3**.
5. From the Simulink Library Browser, select the Math Operations library. Select and add the **Add** block to the design.
6. Connect the blocks as shown below:

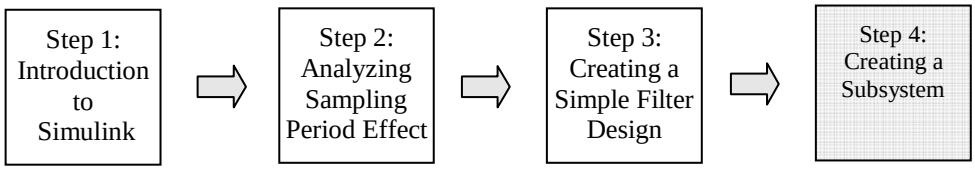


❼ Run the **simulation** and observe the output.



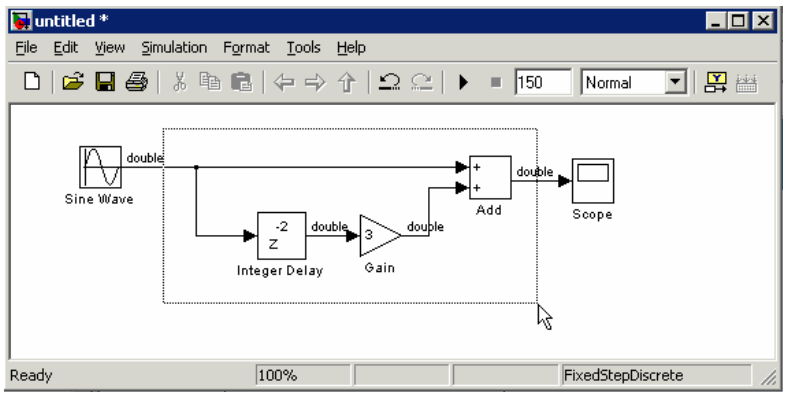
Creating a Subsystem — Step 4

General Flow for this Lab:

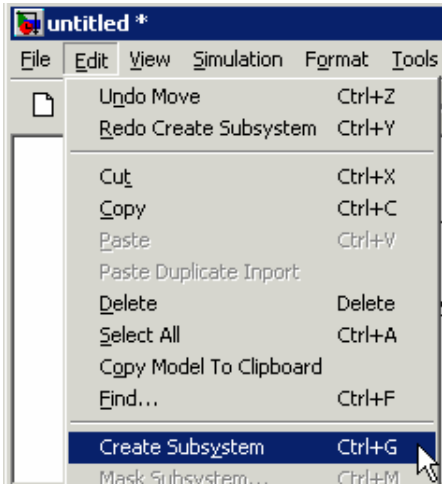


In this step you will select all the blocks between the source and sink, then create a subsystem. Name the subsystem Filter. Run the simulation and verify that the output is still the same. Save the model as **lab1.mdl** in the current working directory …/sysgen/examples/getting_started_training/lab1). Exit MATLAB.

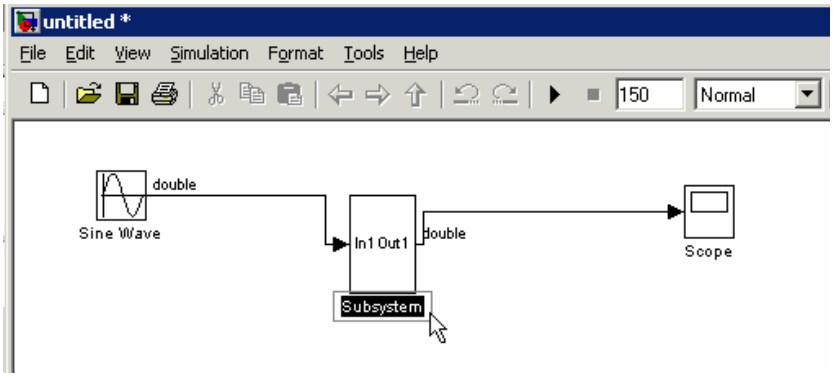
❶ Click in a white space and create a rectangle enclosing all relevant blocks and connections to select all the blocks between the source and sink



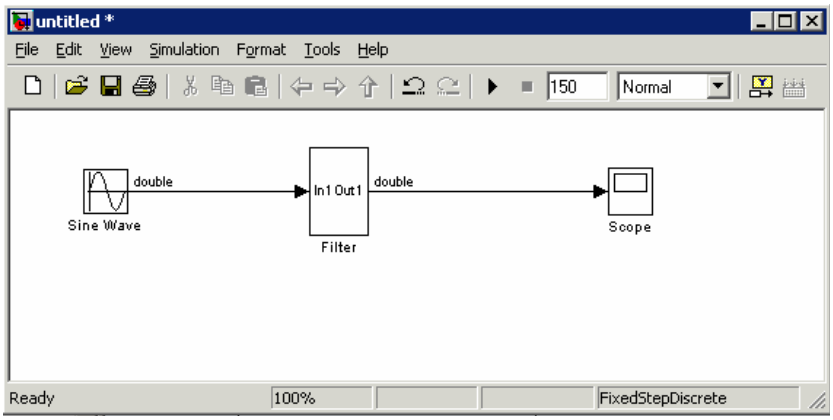
❷ Select **Edit → Create Subsystem** or press **Ctrl+g**



❸ Name the subsystem ***Filter*** by selecting and typing over the title



You can adjust block placements so that the design appears lined up. The design should look similar to the figure below:



❹ Click **File** → **Save** and enter ***lab1.mdl*** as the filename

❺ Click **Save** to save the file in the current directory (...sysgen/examples/getting_started_training/lab1)

❻ Type ***exit*** in the MATLAB command window to close MATLAB

Conclusion

In this lab, you learned the basic design flow involved in Simulink using the Simulink blockset. You observed the effect of a sampling period. You simulated the design using the Simulink simulator and observed the output using the **Scope** block. Finally, you created a subsystem.