INTRODUCTION TO SIMULINK

Introduction:

Simulink is a software package that enables you to model, simulate, and analyze systems whose outputs change over time. Such systems are often referred to as dynamic systems. Simulink can be used to explore the behavior of a wide range of real-world dynamic systems, including electrical circuits, shock absorbers, braking systems, and many other electrical, mechanical, and thermodynamic systems. This section explains how Simulink works.

Simulating a dynamic system is a two-step process with Simulink. First, a user creates a block diagram, using the Simulink model editor, that graphically depicts time-dependent mathematical relationships among the system's inputs, states, and outputs. The user then commands Simulink to simulate the system represented by the model from a specified start time to a specified stop time.

In general, block and lines can be used to describe many "models of computations." One example would be a flow chart. A flow chart consists of blocks and lines, but one cannot describe general dynamic systems using flow chart semantics.

The term "time-based block diagram" is used to distinguish block diagrams that describe dynamic systems from that of other forms of block diagrams. In Simulink, we use the term block diagram (or model) to refer to a time-based block diagram unless the context requires explicit distinction.

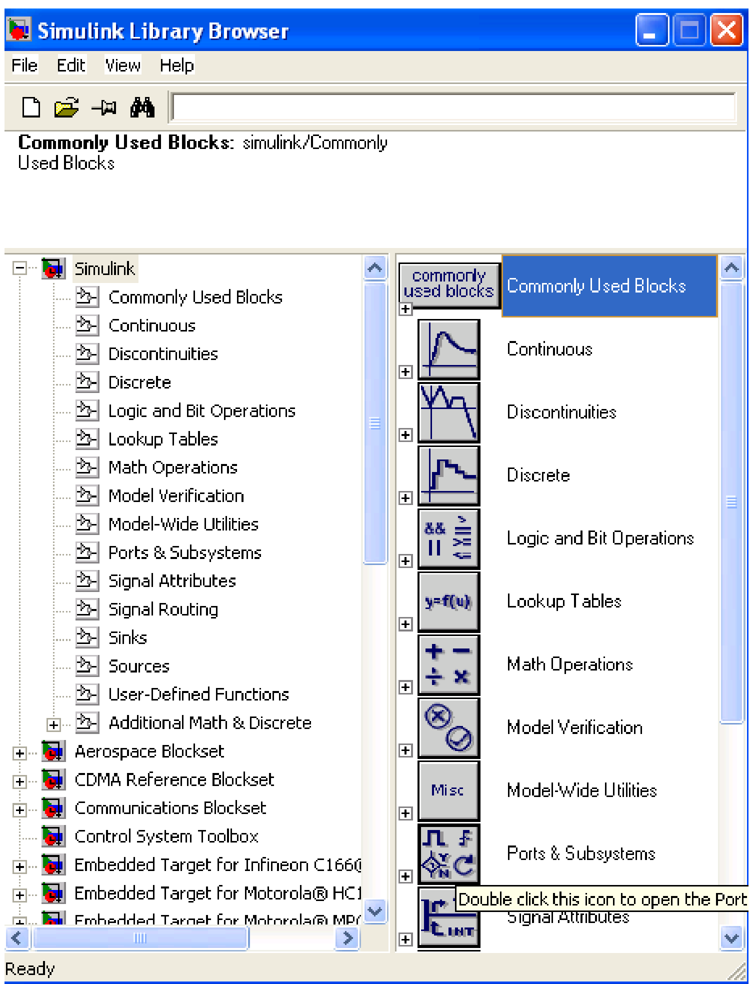
Simulink block diagrams define time-based relationships between signals and state variables. The solution of a block diagram is obtained by evaluating these relationships over time, where time starts at a user specified "start time" and ends at a user specified "stop time." Each evaluation of these relationships is referred to as a time step. Signals represent quantities that change over time and are defined for all points in time between the block diagram's start and stop time. The relationships between signals and state variables are defined by a set of equations represented by blocks. Each block consists of a set of equations (block methods). These equations define a relationship between the input signals, output signals and the state variables. Inherent in the definition of a equation is the notion of parameters, which are the coefficients found within the equation.

Starting Simulink

To start Simulink, you must first start MATLAB. You can then start Simulink in two ways:

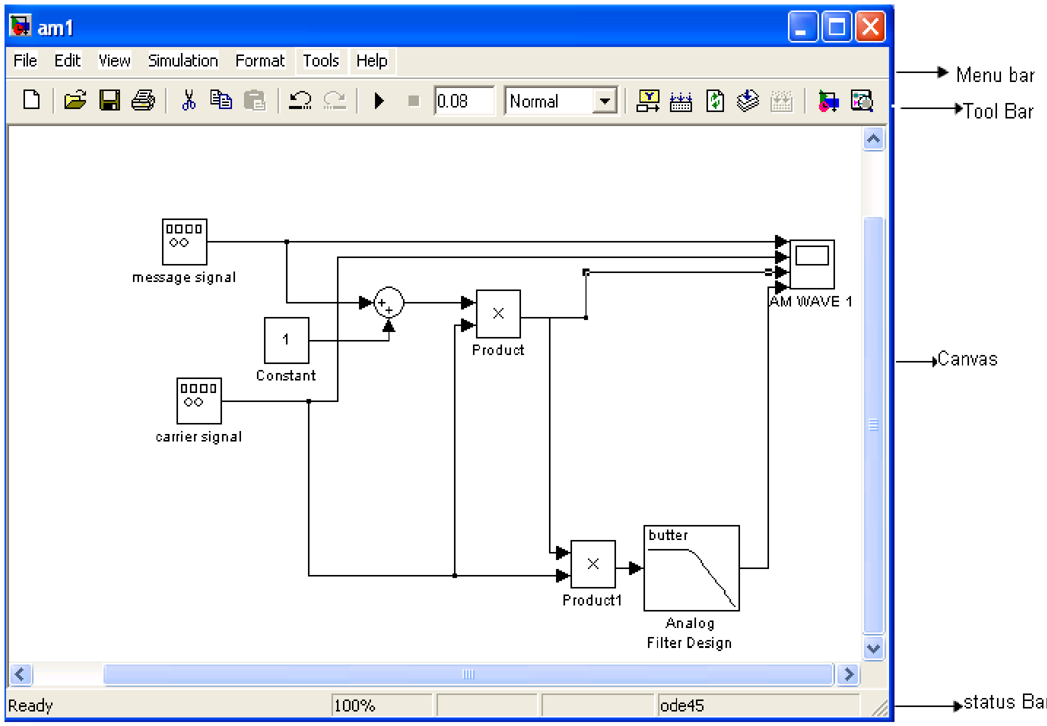
- Click the Simulink icon  on the MATLAB toolbar.
- Enter the simulink command at the MATLAB prompt.

On Microsoft Windows platforms, starting Simulink displays the Simulink Library Browser.



SIMULINK EDITOR:

When you open a Simulink model or library, Simulink displays the model or library in an instance of the Simulink Editor.



Editor Components:

The Simulink Editor includes the following components.

Menu Bar

The Simulink menu bar contains commands for creating, editing, viewing, printing, and simulating models. The menu commands apply to the model displayed in the editor. See Creating a Model and Running Simulations for more information.

Toolbar

The toolbar allows you to execute Simulink's most frequently used Simulink commands with a click of a mouse button. For example, to open a Simulink model, click the open folder icon on the toolbar. Letting the mouse cursor hover over a toolbar button or control causes a tooltip to appear. The tooltip describes the purpose of the button or control. You can hide the toolbar by clearing the Toolbar option on the Simulink View menu.

Canvas

The canvas displays the model's block diagram. The canvas allows you to edit the block diagram. You can use your system's mouse and keyboard to create and connect blocks, selelect and move blocks, edit block labels, display block dialog boxes, and so on. See Working with Blocks for more information.

Context Menus

Simulink displays a context-sensitive menu when you click the right mouse button over the canvas. The contents of the menu depend on whether a block is selected. If a block is selected, the menu displays commands that apply only to the selected block. If no block is selected, the menu displays commands that apply to a model or library as a whole.

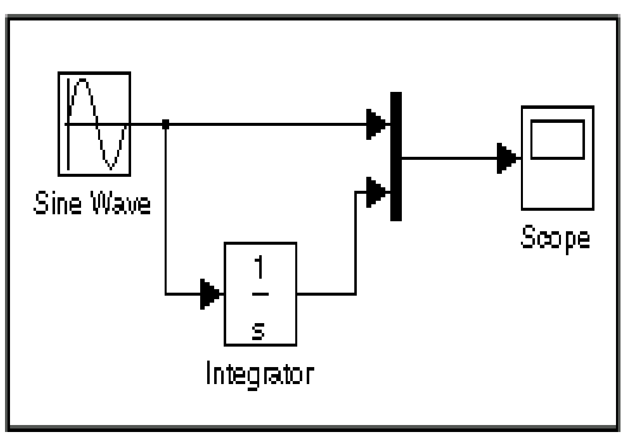
Status Bar

The status bar appears only in the Windows version of the Simulink Editor. When a simulation is running, the status bar displays the status of the simulation, including the current simulation time and the name of the current solver. You can display or hide the status bar by selecting or clearing the Status Bar option on the Simulink View menu.

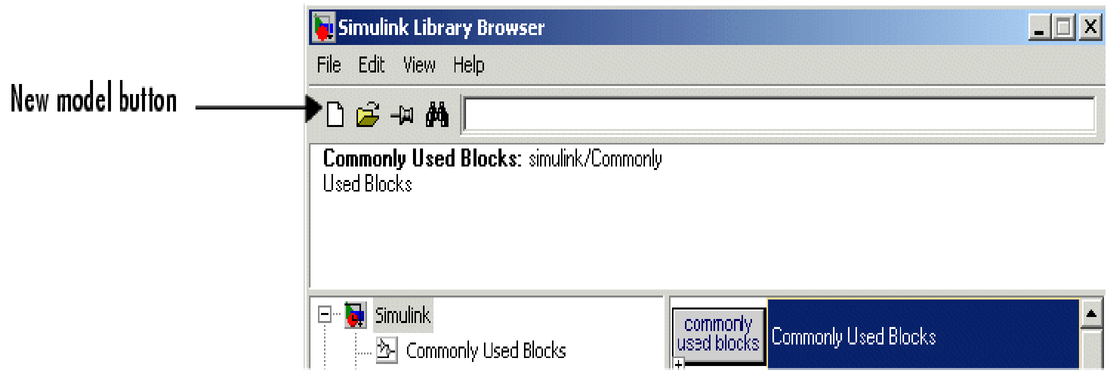
Building a Model

This example shows you how to build a model using many of the model-building commands and actions you will use to build your own models.

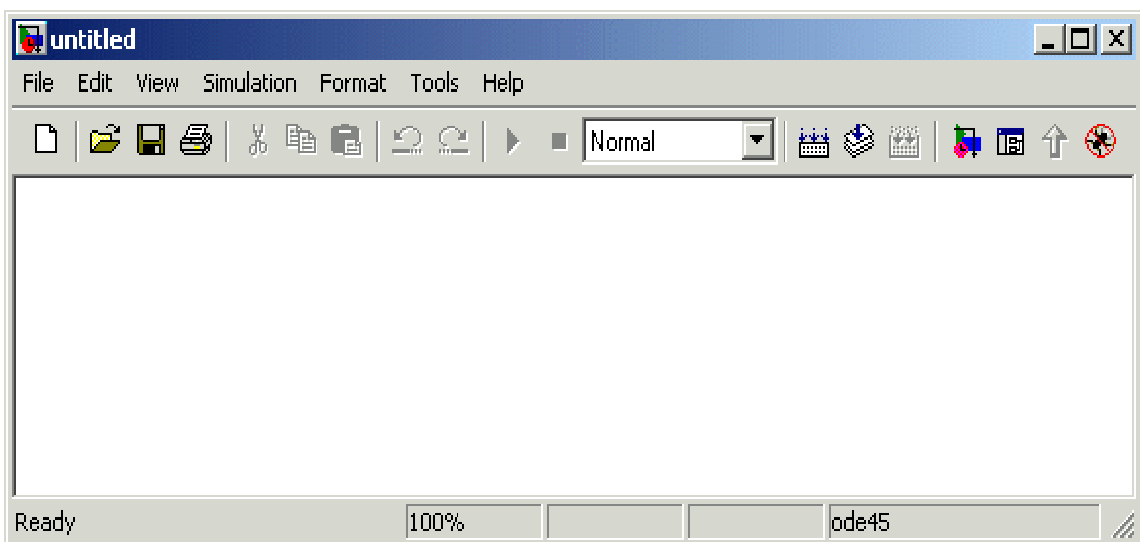
The model integrates a sine wave and displays the result along with the sine wave. The block diagram of the model looks like this.



To create a new model , click the **New Model button** on the Library Browser's toolbar.



Simulink opens a new model window.



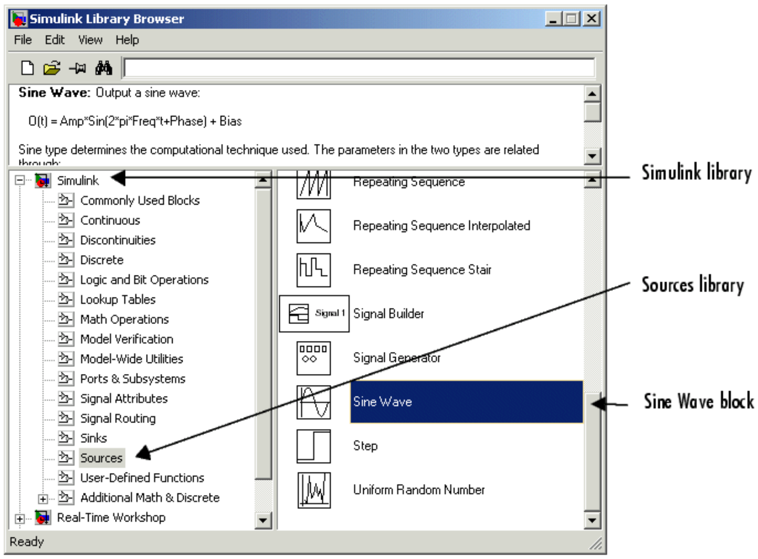
To create this model, you need to copy blocks into the model from the following Simulink block libraries:

- Sources library (the Sine Wave block)
- Sinks library (the Scope block)
- Continuous library (the Integrator block)
- Signal Routing library (the Mux block)

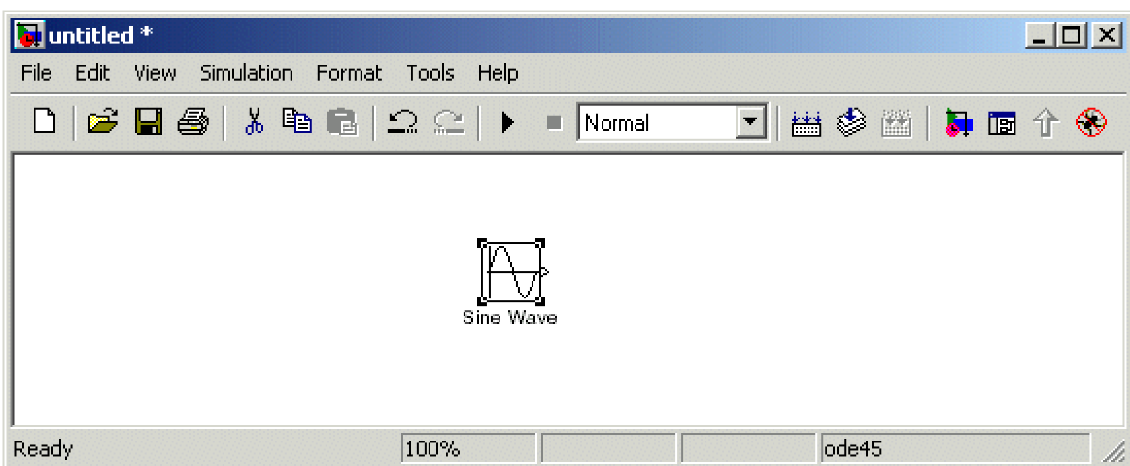
To copy the Sine Wave block from the Library Browser, first expand the Library Browser tree to display the blocks in the Sources library. Do this by clicking the

Sources node to display the Sources library blocks. Finally, click the Sine Wave node to select the Sine Wave block.

Here is how the Library Browser should look after you have done this

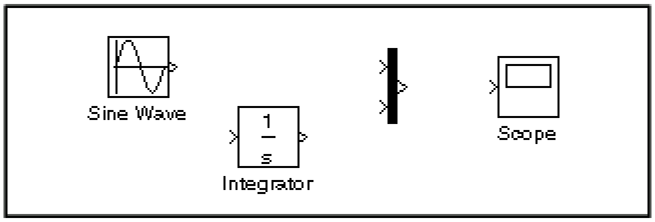


Now drag a copy of the Sine Wave block from the browser and drop it in the model window.

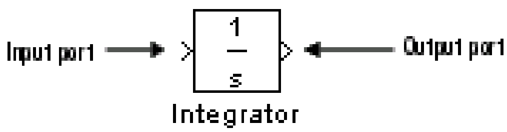


Copy the rest of the blocks in a similar manner from their respective libraries into the model window. You can move a block from one place in the model window to another by dragging the block. You can move a block a short distance by selecting the block, then pressing the arrow keys.

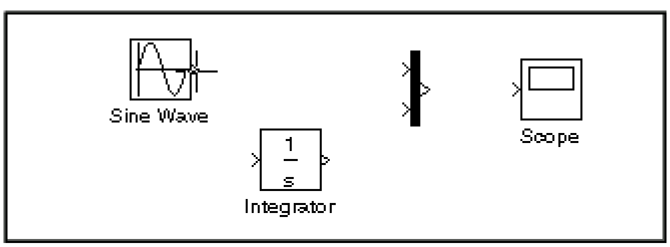
With all the blocks copied into the model window, the model should look something like this.



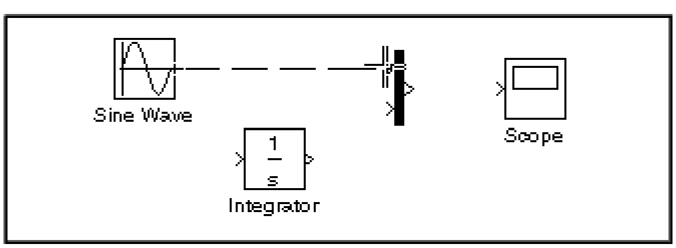
If you examine the blocks, you see an angle bracket on the right of the Sine Wave block and two on the left of the Mux block. The > symbol pointing out of a block is an output port; if the symbol points to a block, it is an input port. A signal travels out of an output port and into an input port of another block through a connecting line. When the blocks are connected, the port symbols disappear.



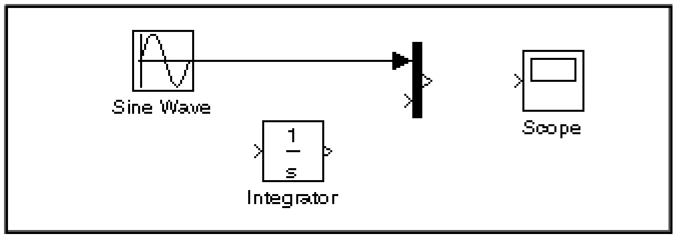
Now it's time to connect the blocks. Connect the Sine Wave block to the top input port of the Mux block. Position the pointer over the output port on the right side of the Sine Wave block. Notice that the cursor shape changes to crosshairs.



Hold down the mouse button and move the cursor to the top input port of the Mux block.
Notice that the line is dashed while the mouse button is down and that the cursor shape changes to double-lined crosshairs as it approaches the Mux block.

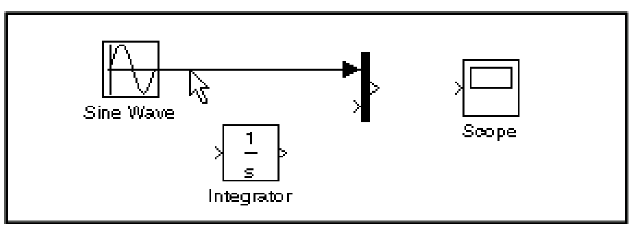


Now release the mouse button. The blocks are connected. You can also connect the line to the block by releasing the mouse button while the pointer is over the block. If you do, the line is connected to the input port closest to the cursor's position.

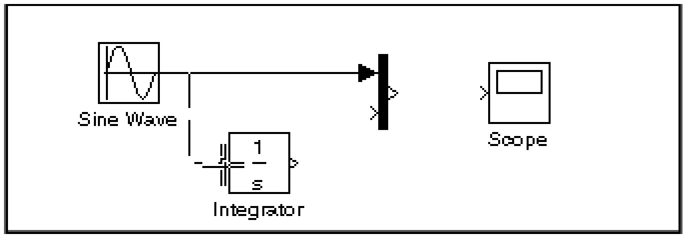


If you look again at the model at the beginning of this section, you'll notice that most of the lines connect output ports of blocks to input ports of other blocks. However, one line connects a line to the input port of another block. This line, called a branch line, connects the Sine Wave output to the Integrator block, and carries the same signal that passes from the Sine Wave block to the Mux block. Drawing a branch line is slightly different from drawing the line you just drew. To weld a connection to an existing line, follow these steps:

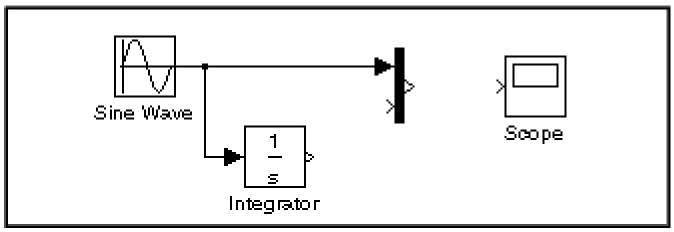
1. First, position the pointer on the line between the Sine Wave and the Mux block.



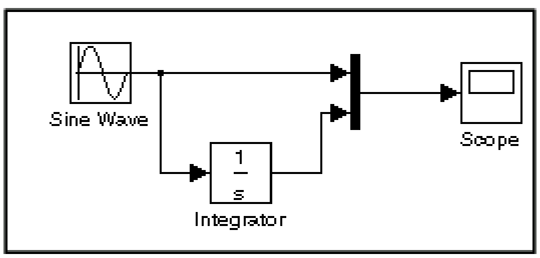
2. Press and hold down the Ctrl key (or click the right mouse button).Press the mouse button, then drag the pointer to the Integrator block's input port or over the Integrator block itself.



3. Release the mouse button. Simulink draws a line between the starting point and the Integrator block's input port.



Finish making block connections. When you're done, your model should look something like this.

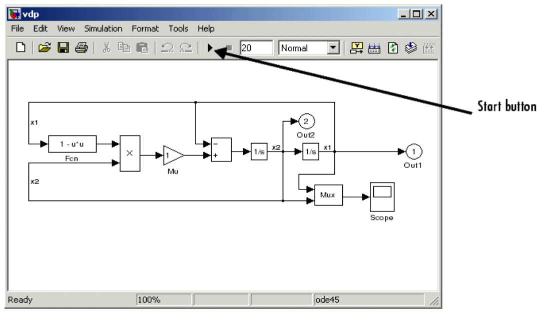


Controlling Execution of a Simulation

The Simulink graphical interface includes menu commands and toolbar buttons that enable you to start, stop, and pause a simulation.

Starting a Simulation

To start execution of a model, select Start from the model editor's Simulation menu or click the Start button on the model's toolbar.

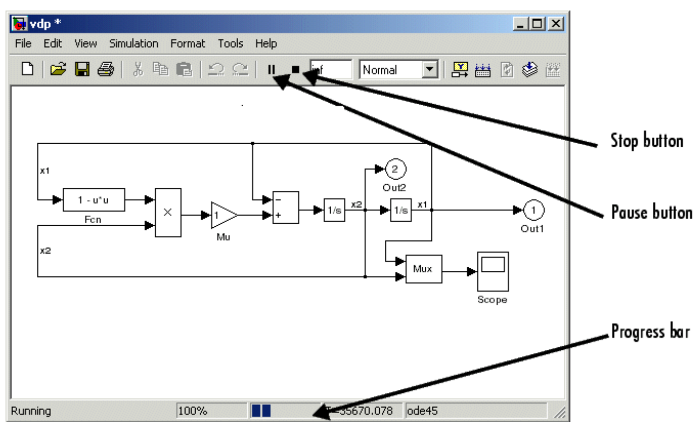


You can also use the keyboard shortcut, **Ctrl+T,** to start the simulation.

Note: A common mistake that new Simulink users make is to start a simulation while the Simulink block library is the active window. Make sure your model window is the active window before starting a simulation.

Simulink starts executing the model at the start time specified on the Configuration Parameters dialog box. Execution continues until the simulation reaches the final time step specified on the Configuration Parameters dialog box, an error occurs, or you pause or terminate the simulation.

While the simulation is running, a progress bar at the bottom of the model window shows how far the simulation has progressed. A **Stop** command replaces the **Start** command on the Simulation menu. A **Pause** command appears on the menu and replaces the **Start** button on the model toolbar.



Your computer beeps to signal the completion of the simulation.

Ending a Simulink Session

Terminate a Simulink session by closing all Simulink windows.

Terminate a MATLAB session by choosing one the **File** menu and **Exit MATLAB**.

Amplitude Modulation & Demodulation

Aim:

To generate amplitude modulated wave using simulink and demodulate the modulated wave.

Software Required:

MATLAB 7.0.4

Simulink

Theory:

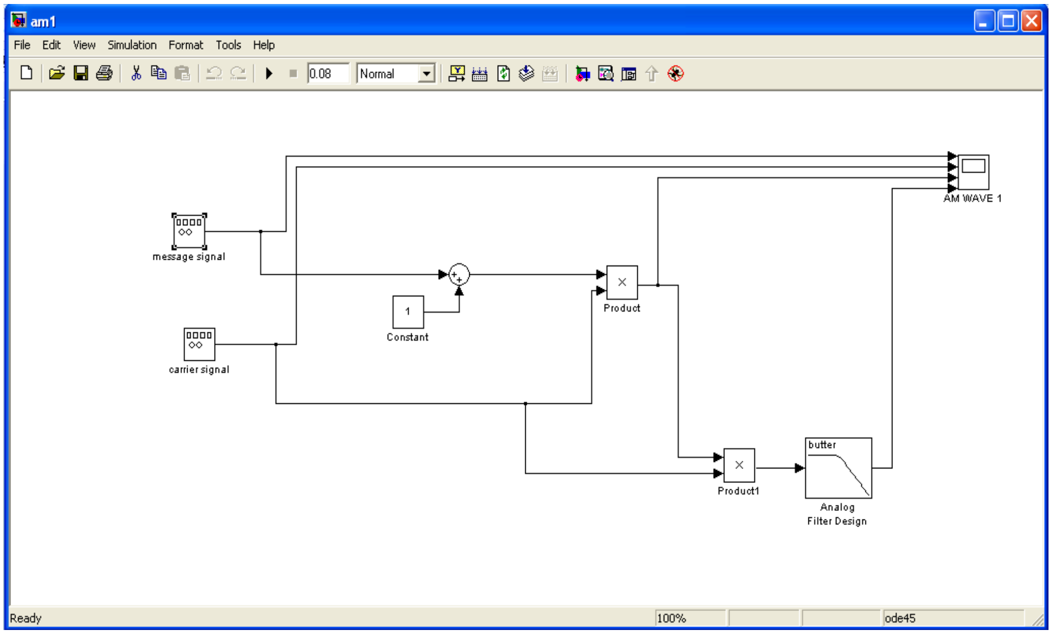
Amplitude Modulation is defined as a process in which the amplitude of the carrier wave c(t) is varied linearly with the instantaneous amplitude of the message signal m(t).The standard form of an amplitude modulated (AM) wave is defined by

$$s(t) = A_c\left[1 + K_a m(t)\cos(2\pi f_c t)\right]$$

Where K_a is a constant called the amplitude sensitivity of the modulator.

Basically amplitude modulated signal is generated by product modulator.The inputs to the product modulator are message signal and carrier signal. Demodulation is the process of extracting the baseband message signal from the carrier so that it may be processed at the receiver. For that purpose various methods are used like diode detector method, product detector method, filter detector etc. The same has been implemented on simulink model. Low pass filter has been implemented to extract the carrier from the modulated signal. Low pass filter (LPF), filters out the high frequency component and allows the low frequency component to pass. Since the carrier signal is of relatively much higher frequency than that of message signal, carrier signal is attenuated while the message signal is received at the receiver.

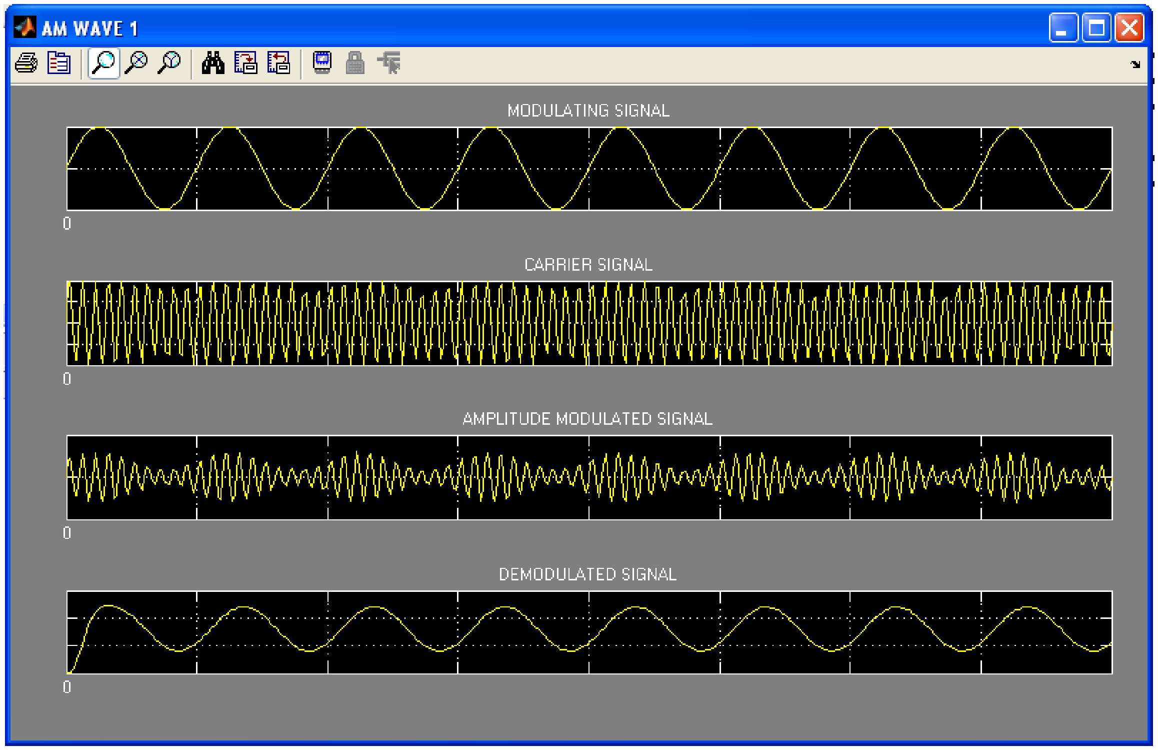
Circuit diagram:



Procedure:

1. open the MATLAB window and then select a simulink
2. select Create a new blank model and open the Simulink Library browser
3. select Signal generator from sources of simulink and drag it to the New model
4. Select the sine wave as message signal and set the input voltage signal to 5Vp-p and signal frequency to 500Hz
5. Again select the signal generator then sine wave. Give the name as Carrier signal. Set the carrier voltage 8Vp-p,frequency 1KHz
6. Select constant from commonly used block of simulink
7. Select Add, Product Blocks from Math Operations
8. All the above blocks connect as per the diagram shown to get the Amplitude modulation signal. observe the output in scope
9. For demodulation select Analog Filter Design block from Filter Designs Library Links of Simulink
10. Connect the filter output to the scope and observe the results

Model Waveform:



Results:

DSB-SC MODULATION AND DEMODULATION

Aim:

To generate DSB-SC Modulated wave using simulink and demodulate the modulated signal

Software Required:

MATLAB 7.0.4
SIMULINK

Theory:

In the double-sideband suppressed-carrier transmission (DSB-SC) modulation, unlike AM, the wave carrier is not transmitted; thus, a great percentage of power that is dedicated to it is distributed between the sidebands, which imply an increase of the cover in DSB-SC, compared to AM, for the same power used.

The DSB-SC modulator output as follows

$$v(t) = m(t)\cos \omega_c t$$

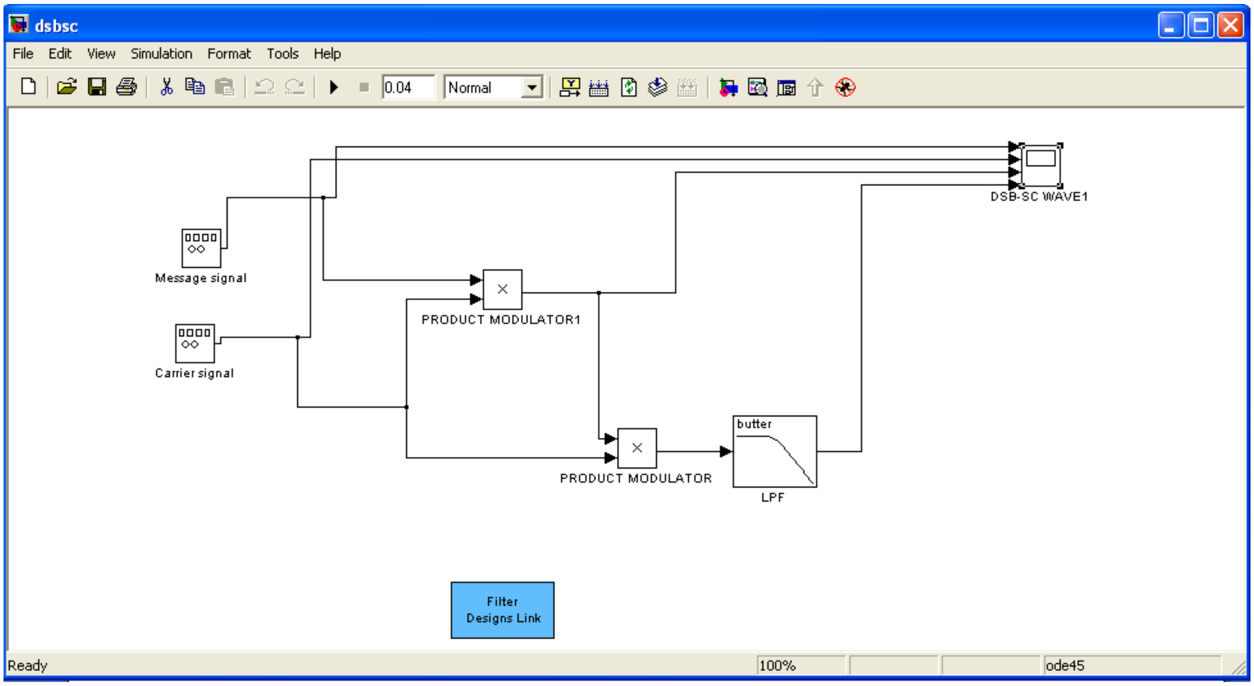
The coherent DSB-SC requires a synchronized local oscillator and works on following
principle.

$$[m(t)\cos \omega_c t]\cos \omega_c t = m(t)\cos^2 \omega_c t = m(t)\left(\frac{1}{2} + \frac{1}{2}\cos 2\omega_c t\right)$$

$$= \frac{m(t)}{2} + \frac{m(t)}{2}\cos 2\omega_c t$$

A low pass filter filters out the message signal from above.

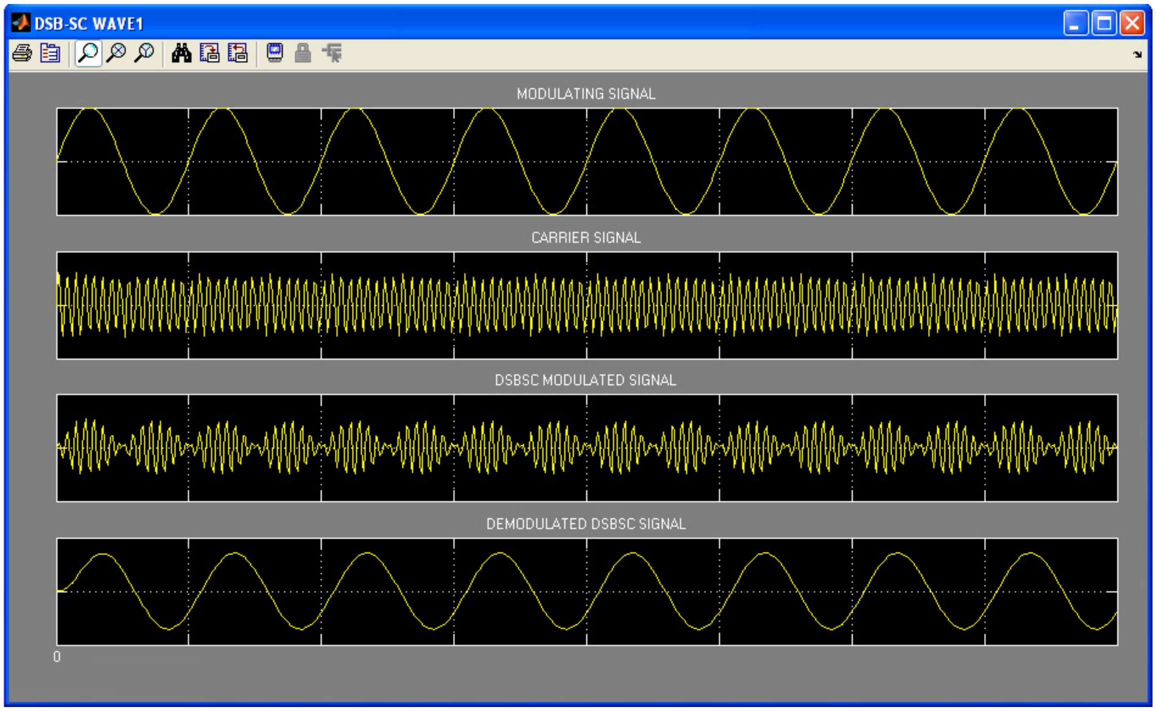
Circuit diagram:



Procedure:

1. open the MATLAB window and then select a simulink
2. select Create a new blank model and open the Simulink Library browser
3. select Signal generator from sources of simulink and drag it to the New model
4. Select the sine wave as message signal and set the input voltage signal to 5Vp-p and signal frequency to 500Hz
5. Again select the signal generator then sine wave. Give the name as Carrier signal. Set the carrier voltage 8Vp-p,frequency 1KHz
6. Select Product Block from Math Operations
7. All the above blocks connect as per the diagram shown to get the Amplitude modulation signal. observe the output in scope
8. For demodulation select Analog Filter Design block from Filter Designs Library Links of Simulink
9. Connect the filter output to the scope and observe the results

Model waveform:



Result:

Frequency Modulation

Aim:

To generate frequency modulated signal using communication block set of SIMULINK

Software Required:

MATLAB 7.0.4

SIMULINK

Theory:

In Frequency Modulation (FM), the amplitude of the sinusoidal carrier wave was modulated in AM, this time the instantaneous frequency of a sinusoidal carrier wave will be modified proportionally to the variation of amplitude of the message signal.

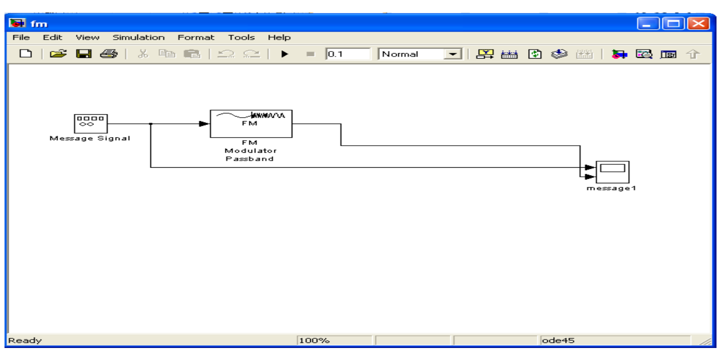
The FM signal is expressed as

$$s(t) = A_c \cos(2\pi f_c + \beta \sin(2\pi f_m t))$$

Where A_C is amplitude of the carrier signal, f_C is the carrier frequency

β is the modulation index of the FM wave

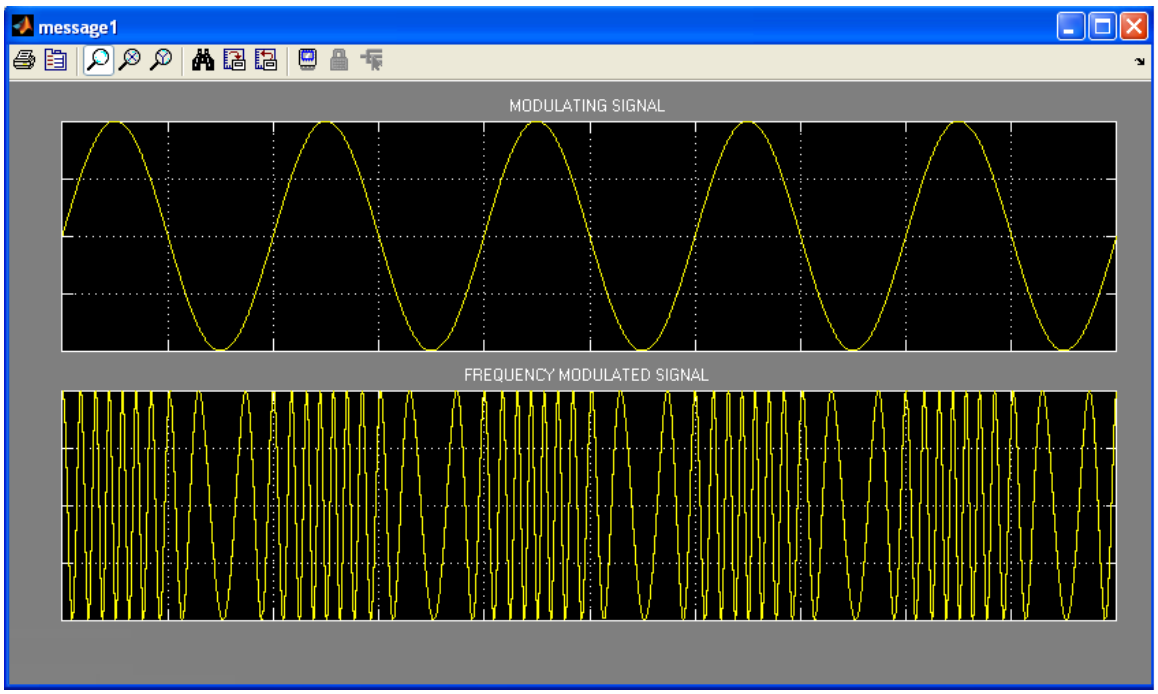
Circuit diagram:



Procedure:

1. open the MATLAB window and then select a simulink
2. select Create a new blank model and open the Simulink Library browser
3. select Signal generator from sources of simulink and drag it to the New model
4. Select FM modulator from Communication Block set of Simulink Library Browser
5. Observe FM modulated output in scope

Model waveform:



Results: