

BASIC TUTORIAL FOR THE USE OF SIMULINK

Overview

Simulink is a powerful system modeling tool which accompanies MATLAB. It allows the user to graphically model equations and analyze the results. By allowing the user to work at a higher level, a system can be better understood and "what-if" scenarios can be more readily explored. This particular tutorial is designed to familiarize the student with the basic tools needed to develop and run a Simulink model.

Tutorial Conventions

The following conventions will be used in this tutorial when describing command sequences:

- A comma is used to separate sequential commands or entries.
- The down arrow symbol ▼ indicates that a pull-down menu must be accessed to find the appropriate menu item.
- For Radio Buttons the symbol Ⓒ will be used to indicate a selection.
- Menu bar items, and keyboard strokes will be *italicized*.
- Custom entered commands will be enclosed in parenthesis i.e. “**type here**”

Note: The input boxes within parameter windows always require numerical input. For this tutorial, however, you may see variables in these input boxes. You must input the numerical value associated with that variable for your model to run properly.

Getting Started

To begin, we will discuss some of the basics of using Simulink.

Opening a Simulink model

Double click on the MATLAB icon on your desktop; the command prompt window will appear as in Fig. 1 below.

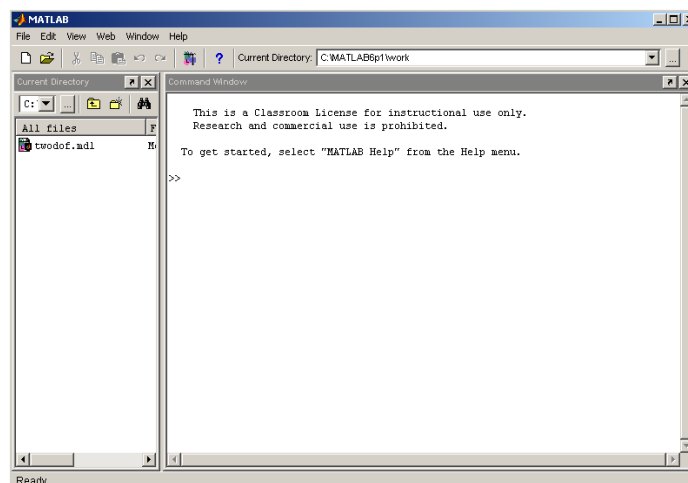


Fig. 1. The MATLAB® command window

At the prompt, type *Simulink*, *return*. You will see the Simulink library browser appear as in Fig. 2 below.

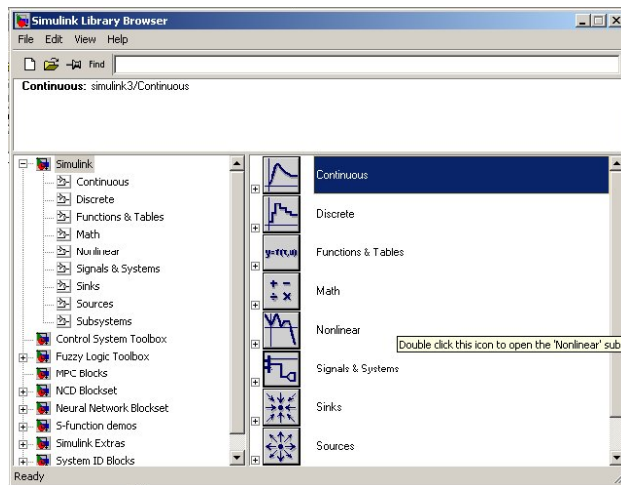


Fig. 2. The Simulink library browser

From the **File** menu, select **new, model**. A new untitled window will appear. This will be the model space in which you will construct your Simulink model.

Bringing blocks into the model

To build a model, drag and drop blocks from the library into the model space. For example, to drag a Scope block into the model space, click on the Simulink library browser tree, then click on the **Sinks** submenu (you will see a series of icons appear on the right), now click and drag the **Scope** icon onto the model space. Refer to Fig. 3 below.

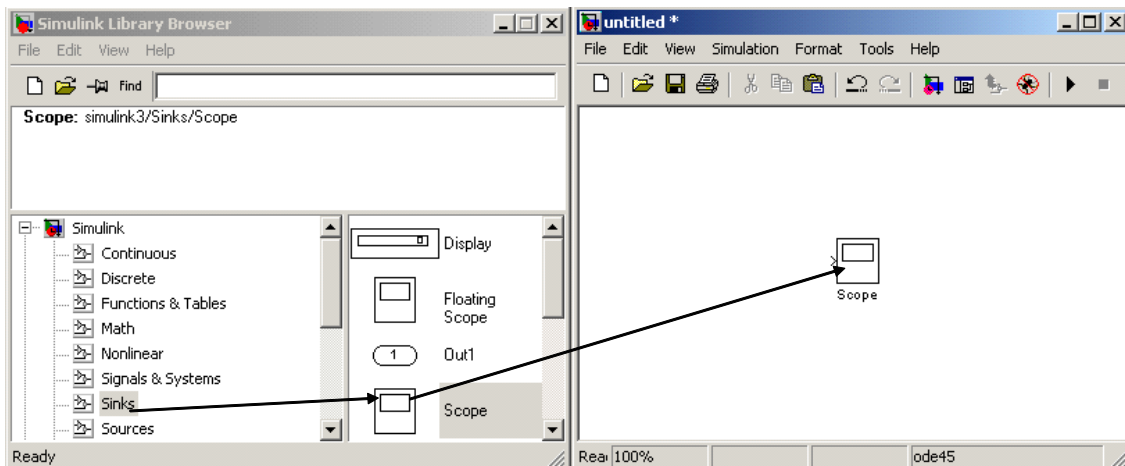


Fig. 3. Sequence for dragging and dropping Simulink blocks.

Other blocks can be accessed under the Simulink submenu.

Connecting and disconnecting blocks

Lines are used to connect blocks and allow signal transmission between blocks. Drag and drop a **Sine-wave** block from the **Sources** submenu into the model space. Notice the output of the sign block and the input of the Scope block in Fig. 4 below.



Fig. 4. Blocks prior to connection

Left-click on the output port of the *Sine-Wave* block (the pointer will become a cross-hair), click and drag the line towards the input terminal of the next block as in Fig. 5 below.

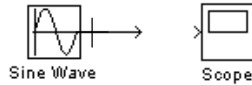


Fig. 5. Click and drag Line

When you reach the input port of the next block the pointer will become a double cross-hair. Release the left mouse button. The connected blocks will appear as in Fig. 6 below.



Fig. 6. Blocks attached.

Note: It is also possible to start with your cursor at the input port of a box and draw a line to the output port of another box.

Creating branch lines

You can also create a branch line off of the signal line just created. To do this, press and hold the **Ctrl** key, now with your cursor positioned anywhere on the line, left-click and create a branch line as in Fig. 7 below.

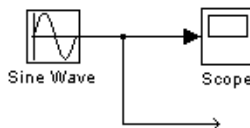


Fig. 7. Creating a branch line

Changing block directions

Sometimes it will become necessary to flip a block around. This can be done easily by right-clicking on the block, selecting **Format, Flip Block** or **Rotate Block**. The results of these choices can be seen in Fig. 8 below.

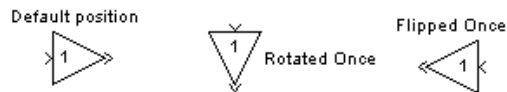


Fig. 8. Possible block positions

Copying and moving blocks

To create additional instances of a block, simply hold down the **Ctrl** key, click on the block, and drag off a copy. To move a block, simply left-click on the block, hold down on the mouse key, and drag the block to a new desired location.

Changing block labels and inserting text

To change block labels, simply click on the label and edit as usual. To add a text box to the workspace, double-click anywhere on the workspace and a textbox will appear, type into the textbox as usual, when done, click anywhere outside the textbox.

Changing block parameters

Block parameters are used to control block input and output. The parameter window for any block can be accessed by double-clicking on the block. For example, double-clicking on a **Sum** block will open the following window, shown in Fig. 9 below.

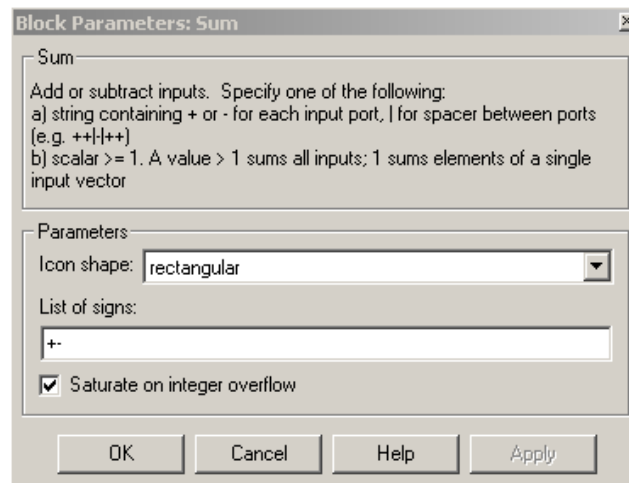


Fig. 9. Sum block parameter window

Here, it is possible to change the shape of the icon and the number of input ports. Also, the input ports can be adjusted to act as a summer (+) or a comparator (-). After changing the block parameters as required select **Apply**, **OK** to activate the new parameters. See the Summary Table for details on the parameters associated with different blocks. To aid in understanding how Simulink works, suggested values will be given for the examples in this tutorial.

Running the Simulation

Once the block parameters have been specified, it is time to run the model simulation. Select **Simulation** from the menu bar, **Simulation Parameters**, a window will appear as in Fig. 10 below.

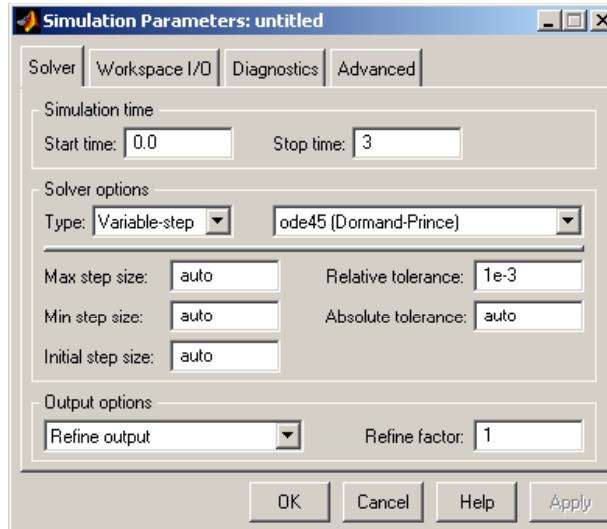


Fig. 10. Simulation parameters

In general, ode45 is the standard solver and will be sufficient for most of the simulations you will be doing. For most cases the start time should be left at the default of 0, the stop time, however, should be adjusted as required to see the system response (i.e. if the stop time is set too short you may not see the full response of the system). For the remaining fields, keep the default values. Once the fields have been updated select *Apply*, *OK*.

To run the simulation go to the menu bar and select the ► button in Fig. 11 below.



Fig. 11. Simulation button viewing block output

Viewing the Simulation Results

To view the output of any block simply drag and drop a scope block into the model space as shown in the example above and tap off of the line of interest. After running the simulation you can view the results by double-clicking on the Scope block. A window will appear as in Fig. 12 below.

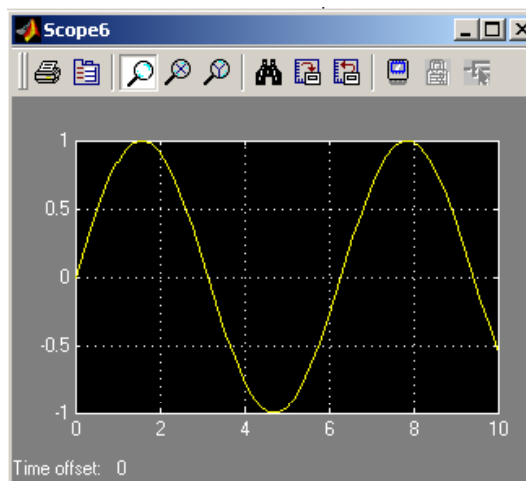


Fig. 12. Typical scope window

Click on the binocular icon to auto-size the window. You can also zoom in and out on the signal by selecting the appropriate buttons on the menu bar.

This concludes our basic introduction to using Simulink. You should now have the basic skills necessary to construct some simple dynamic systems models, run these models, and analyze the results.