

Starting Simulink

Simulink is started from the MATLAB command prompt by entering the following command:

Simulink

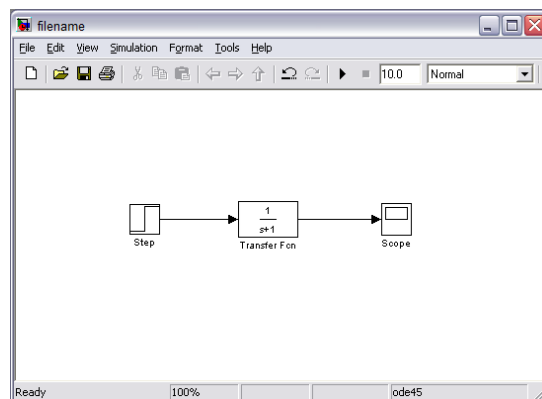
Alternatively, you can click on the **Simulink** icon at the top of the MATLAB window as shown here:



A new model can be created by selecting **New** from the **File** menu in any Simulink window (or by typing **Ctrl-N**).

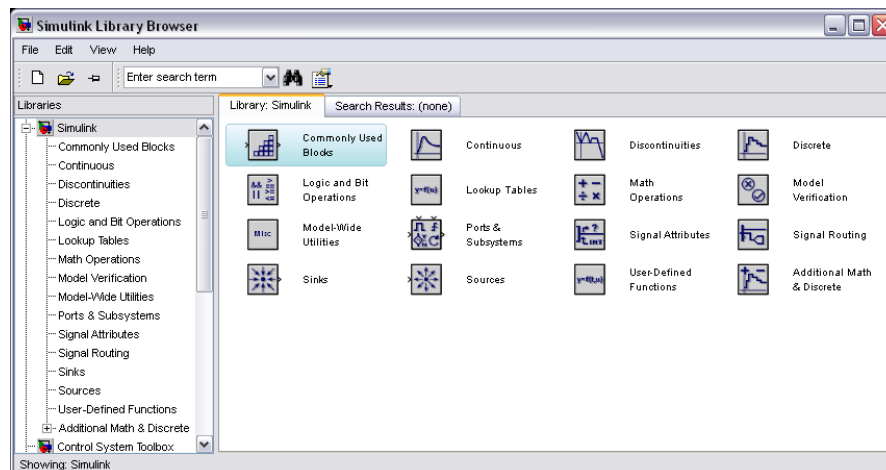
Once you have created a file, it can be opened in Simulink by entering the following command in the MATLAB command window. (Alternatively, you can load this file using the **Open** option in the **File** menu in Simulink, or by typing **Ctrl-O** in Simulink). For example, **filename**

The model window should then appear.



Libraries

When it starts, Simulink brings up a single window, entitled **Simulink Library Browser** which can be seen here. Open the libraries that contain the blocks you will need. These usually will include the **Sources**, **Sinks**, **Math** and **Continuous** libraries, and possibly others.



Drag the needed blocks from their library folders to that window. The Math library, for example, contains the **Gain** and **Sum** blocks. The **Continuous** library has the **Transfer Function** block. The **Source** library contains the **Step** block.

Arrange the blocks that you need to use in an orderly way corresponding to the equations to be solved.

Interconnect the blocks by dragging the cursor from the output of one block to the input of another block. Interconnecting branches can be made by right-clicking on an existing branch.

Double-click on any block having parameters that must be established, and set these parameters. For example, the gain of all **Gain** blocks must be set. The number and signs of the inputs to a **Sum** block must be established. The parameters of any source blocks should also be set in this way.

To Run Simulations

Before running a simulation of any system that contains a **Scope**, open the scope window by double-clicking on the scope block. Follow the instructions posted at Electronics Believer to obtain a Bode plot using the **Gain and Phase Margin Plot** block (<http://electronicsbeliever.com/how-to-setup-bode-plot-in-simulink/>). Note that it is possible to obtain a pole-zero plot using **Pole-Zero Plot** block in **Simulink**. The data generated during a simulation can be exported to MATLAB using the **To Workspace** block, the results of the calculation can then be used with the **bode** function to obtain a Bode plot or the **nyquist** function to obtain a Nyquist plot.

It is necessary to specify a stop time for the solution. This is done by clicking on the Simulation > Parameters entry on the Simulink toolbar.

At the **Simulation > Parameters** entry, several parameters can be selected in this dialog box. If the response before time zero is needed, it can be obtained by setting the **Start** time to a negative value. It may be necessary to reduce the maximum integration step size used by the numerical algorithm for some simulations. If the plots of the results of a simulation appear “choppy” or composed of straight-line segments when they should be smooth, reducing the max step size permitted can solve this problem.

Then, to start the simulation, either select **Start** from the **Simulation** menu, click the **Play** button at the top of the screen, or enter **Ctrl-T**.

There is an **Autoscale** icon that will rescale the vertical axis of the plot that is generated. To rescale the horizontal axis, you must change the run time in the **Simulation > Parameter** pop-up window.

Compiled from:

- CISE 302 Linear Control Systems Laboratory Manual, Systems Engineering Department, King Fahd University of Petroleum & Minerals.
<http://www.kfupm.edu.sa/departments/se/SiteAssets/Lab%20Manuals/CISE-302-Linear-Control-Systems-Lab-Manual.pdf>
- Controls Tutorials for MATLAB and Simulink, College of Engineering, University of Michigan
<http://ctms.engin.umich.edu/CTMS/index.php?aux=Home>
- MathWorks website www.mathworks.com/

