Chapter 5: Getting started with SIMULINK

Chapter 5: Getting started with SIMULINK

5.1-Introduction

Simulink is the MATLAB graphics extension for representing functions mathematics and systems in block diagram form, and to simulate the operation of these systems. The objectives of this course are:

- Get to grips with the Simulink Matlab simulation software;
- See some essential blocks to use in practical exercises;
- Learn how to modify block parameters and simulate a Simulink model.

5.2- SIMULINK libraries

To open Simulink type "simulink" in the matlab command window or click on the icon present in the main MATLAB window. A dialog box containing the list of libraries appears on the screen. Each library contains a set of blocks used to create diagrams, or block diagrams [9].

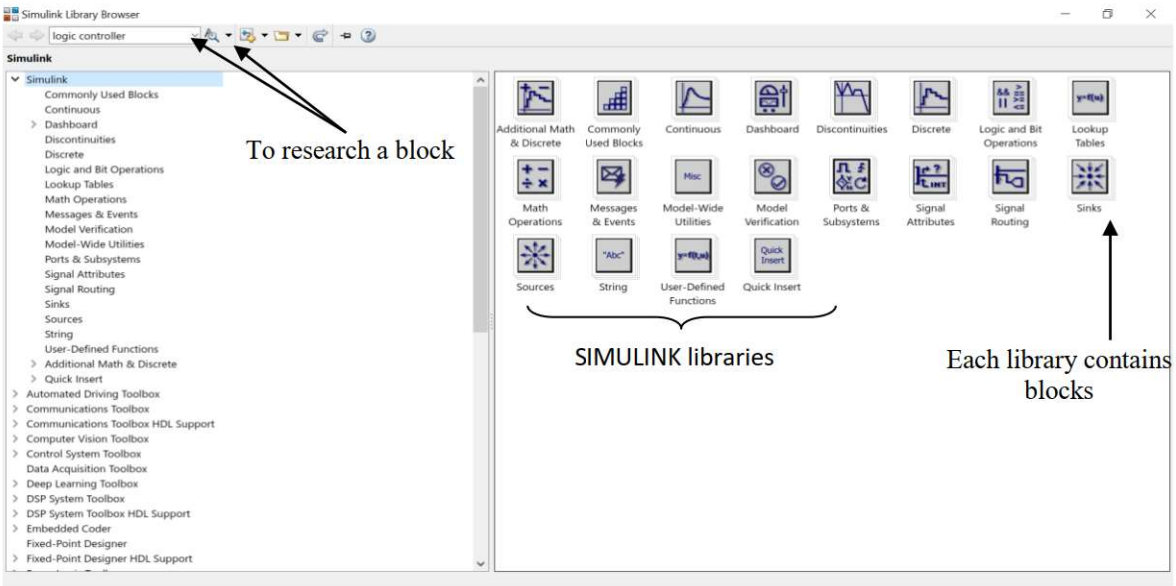


Figure 5.1 *Simulink library browser*

5.3- Quick start

Simulink provides a graphical editor, customizable block libraries, and solvers for modeling and simulating dynamic systems. It is integrated with MATLAB, enabling you to incorporate

MATLAB algorithms into models and export simulation results to MATLAB for further analysis.

To create a Simulink model follow the following steps:

- From the Simulink Library Browser menu select File > New > Model or click on the icon, an Untitled working window will open.
- Open the block collections by double clicking (You can search for any block by typing its name in the search bar of the Simulink Library Browser);
- Drag the blocks you need to build the diagram into the working window (You can take a copy of a block by right-clicking and dragging it);
- Make connections between the blocks using the mouse;
- Change the parameters of any block by double clicking on this block, a window will open.
- When you have finished the diagram, save the model to a file: File > Save or the icon registration and give a name to the model (*.mdl).

5.4 - Masks and subsystems

5.4.1 – Subsystems

To group multiple blocks into a single block (subsystem), select the blocks you want contained in the subsystem, then select Create subsystem from the Edit menu (or right mouse button)

5.4.2 - Masking of subsystems

To mask a subsystem file:

- Open the subsystem file to be masked.
- In the Simulink toolstrip, on the Subsystem tab, click Create System Mask. Alternatively, right-click anywhere on the canvas and select Mask and click Create System Mask. The Mask Editor dialog opens.
- Add mask parameters and click OK.

Simulink invokes the **callback** functions to disable or hide the controls whenever you open the dialog boxes. Disable a Button on a Dialog Box.

5.5- Study of some simulation examples

The `simple` model consists of three blocks: Step, Transfer Function, and Scope. The Step is a **Source** block from which a step input signal originates. This signal is transferred through the

line in the direction indicated by the arrow to the *Transfer Function* **Continuous** block. The *Transfer Function* block modifies its input signal and outputs a new signal on a line to the *Scope*. The *Scope* is a **Sink** block used to display a signal much like an oscilloscope.

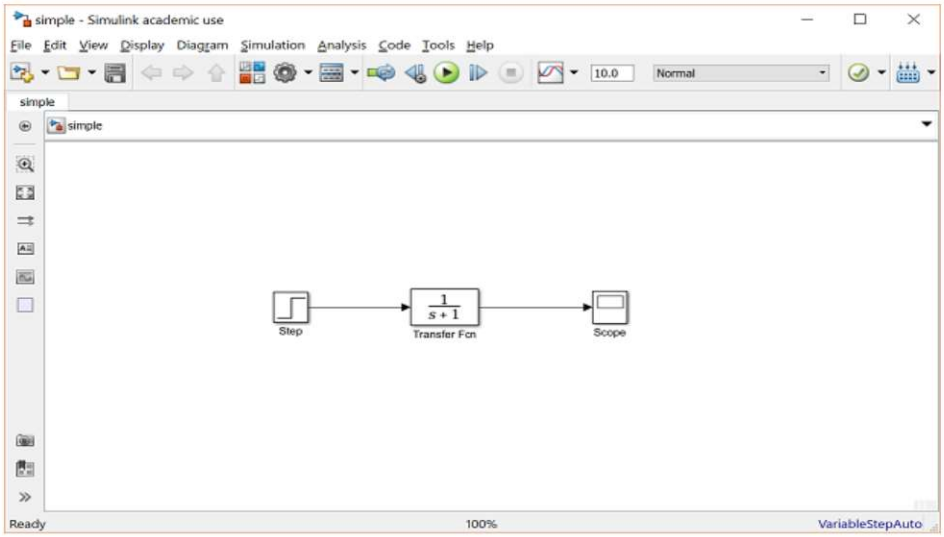
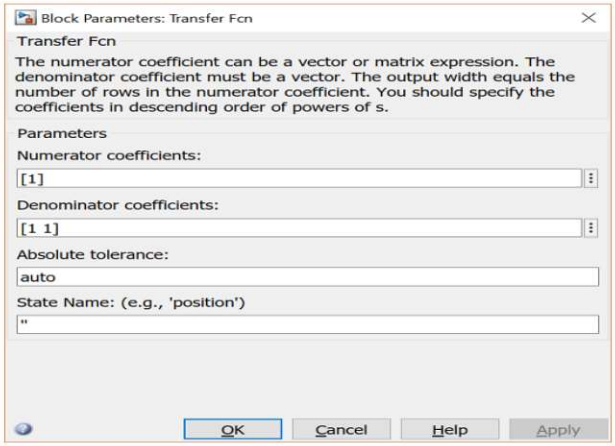


Figure 5.2 *Simple simulation example*

There are many more types of blocks available in Simulink, some of which will be discussed later. Right now, we will examine just the three we have used in the `simple` model.

5.5.1- Modifying Blocks

A block can be modified by double-clicking on it. For example, if you double-click on the *Transfer Function* block in the `Simple` model, you will see the following dialog box. This dialog box contains fields for the numerator and the denominator of the block's transfer function. By entering a vector containing the coefficients of the desired numerator or denominator polynomial, the desired transfer function can be entered. For example, to change the denominator to: s^2+2s+4

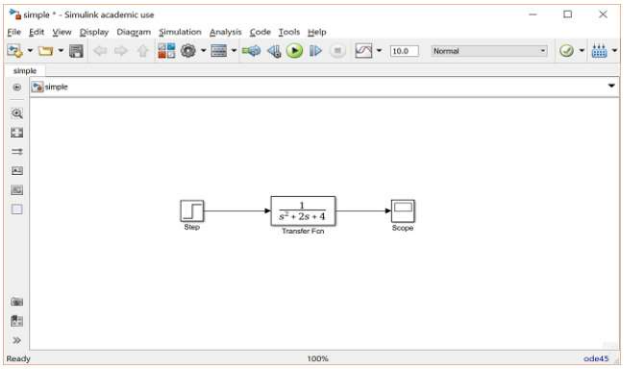


enter the following into the denominator field

```
[1 2 4]
```

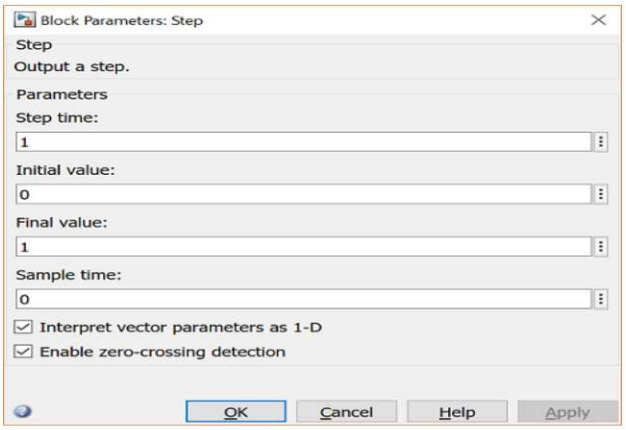
and hit the close button, the model window will

change to the following, which reflects the change in the denominator of the transfer function.



The *Step* block can also be double-clicked, bringing up the following dialog box.

The default parameters in this dialog box generate a step function occurring at time = 1 sec, from an initial level of zero to a level of 1 (in other words, a unit step at t = 1). Each of these parameters can be changed. Close this dialog before continuing.



When a simulation is performed, the signal which feeds into the scope will be displayed in this window. Detailed operation of the scope will not be covered in this tutorial.

5.5.2- Running Simulations

Before running a simulation of this system, first open the scope window by double-clicking on the scope block. Then, to start the simulation, either select **Run** from the **Simulation** menu, click the **Play** button at the top of the screen, or hit **Ctrl-T**.

The simulation should run very quickly and the scope window will appear as shown below.

5.5.3- Simulation parameters

- Open the configuration settings by selecting Configuration Parameters in the Simulation menu.
- Set simulation times using the Start time fields and Stop time in the Solver section of the dialog box Configuration Parameters.

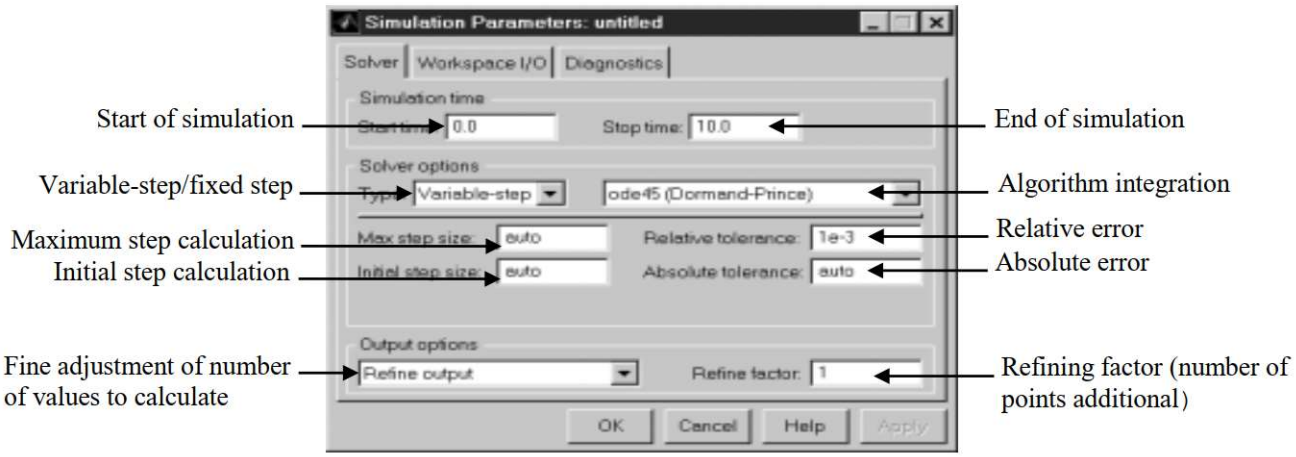


Figure 5.3 *Configuration parameters*