

Simulink Tutorial

In this tutorial we will show some of the most important functions in Simulink and how they work. Each function will be described with a note in a context of an example. It is strongly advised that the reader tries this examples on his/her own time for better understanding and repeatability.

To open Simulink, launch Matlab and click on 8th icon on a menu toolbar:

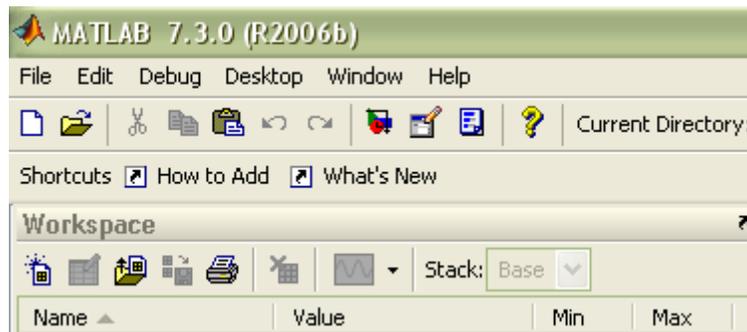


Figure 1: Front Panel of Matlab Screen

After loading Simulink, a new window will open up (Figure 2). Simulink contains a number of blocksets that one can use for a specific application. The most commonly used blockset is Simulink itself, which contains most of the basic function blocks, such as transfer function, summation junction, state-space function block, gain, step function, e.t.c. One should be able to use these function blocks to design and simulate a control system, analyze its stability and robustness. In this tutorial we will use Simulink and Simulink Extras blocksets in order to design and simulate a control system. Simulink Extras contains a functional block for a PID controller, which we choose for our design. Design in Simulink is done by means of drag-and-drop. Each of the functional blocks can be dragged and dropped in a design window. After all of the required blocks have been created, they are connected with virtual wires completing the system.

Let's start by creating a design window. Click on the first icon in Simulink window toolbar. New window should appear, but it is untitled. Save it and rename it according to your project, make sure the name is meaningful.

In order to create a closed loop control system we need the following elements: Reference signal, summation junction, controller, plant and a scope for displaying and analyzing results. In this tutorial we choose a plant to be a second order motion system. The specifications of the design are to obtain a 3 ft displacement in 1 second or better.

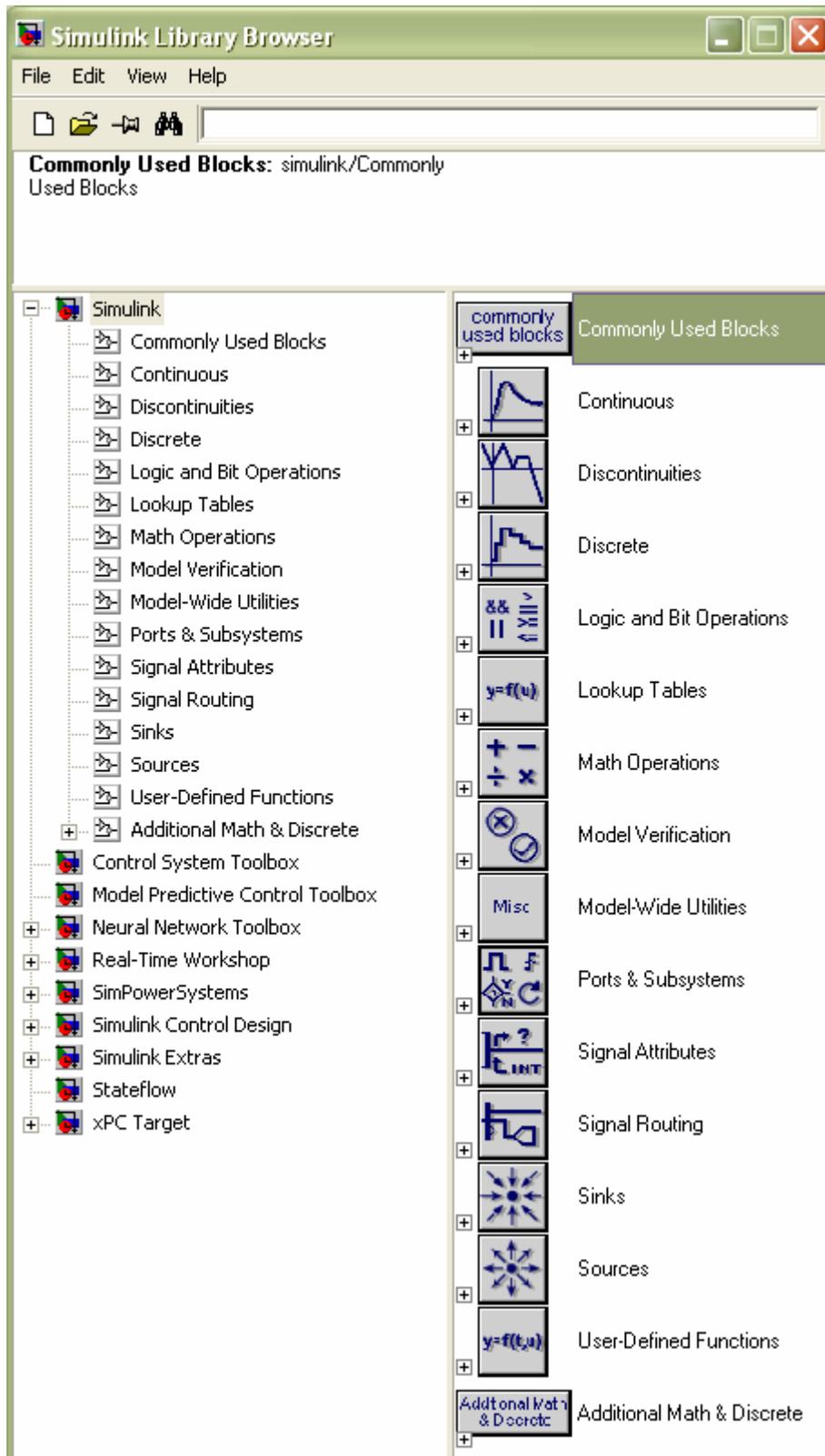


Figure 2: Simulink Window

We will use step function as our reference signal, which can be found under Sources in Simulink blockset. Left click on the step function icon and drag it to your design window:

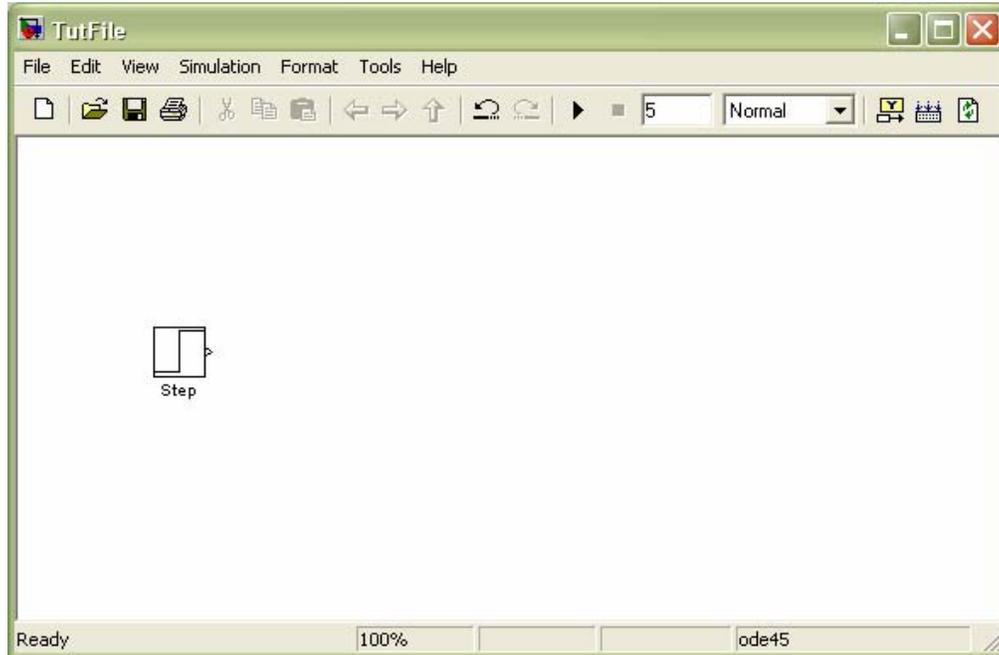


Figure 3: Design window with reference signal

Summation junction can be found under Commonly Used Blocks in Simulink blockset. Drag the Sum icon and drop it in a design window:

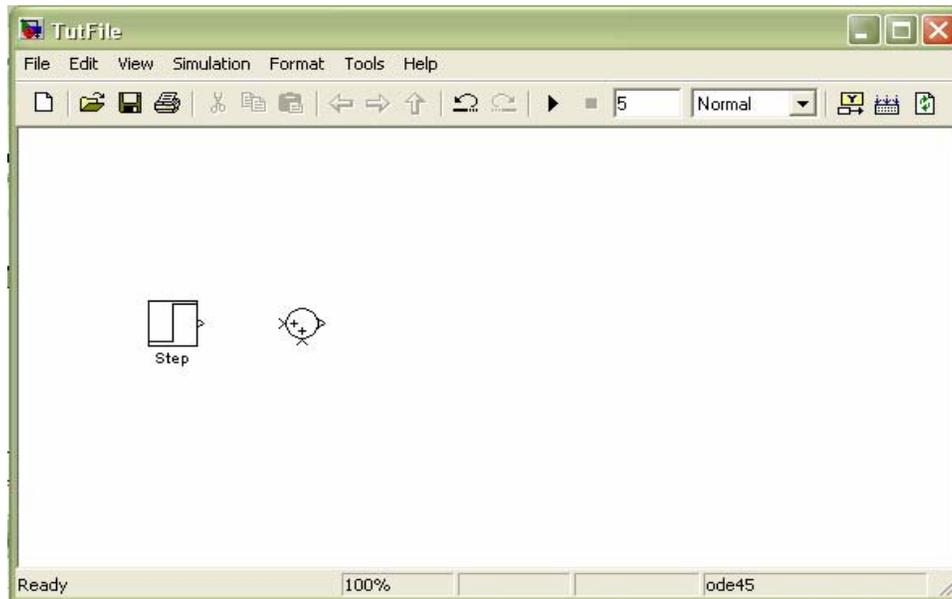


Figure 4: Addition of Summation Junction

PID controller is located under Simulink Extras/Additional Linear. Drag PID icon and drop in a design window:

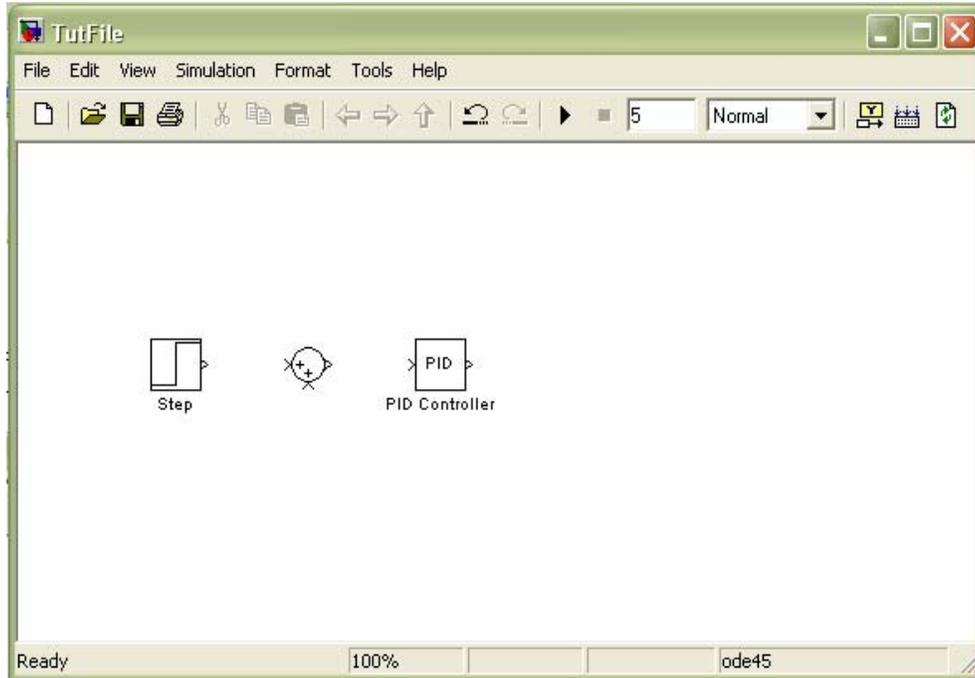


Figure 5: Addition of PID controller

The plant is generally represented as a transfer function. Therefore, we will use transfer function block, which can be found under Continuous in Simulink blockset, as our plant. Drag transfer function icon and drop it in a design window:

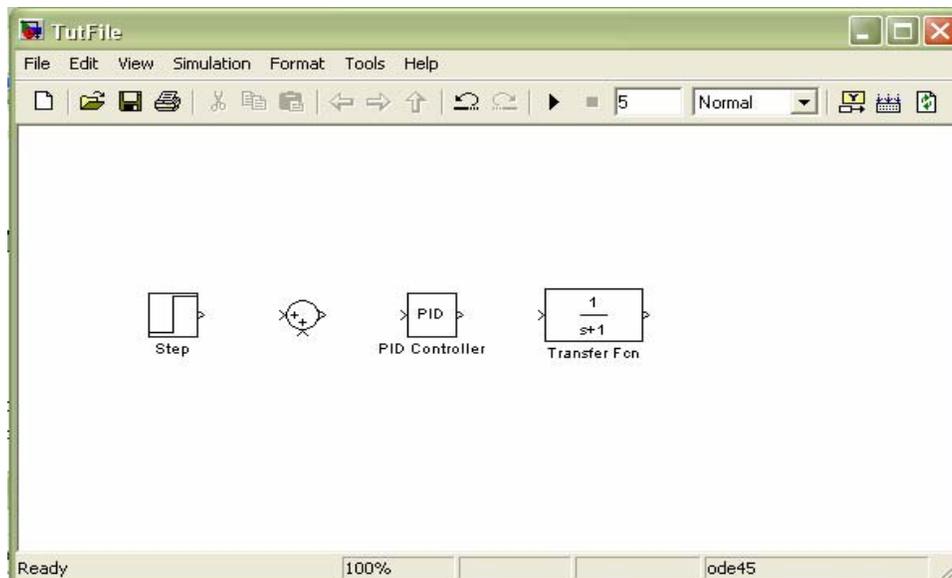


Figure 6: Addition of a plant

The Scope can be found under Sinks in Simulink blockset. Drag and drop the scope icon in design window. Since in the end of the design we would like to compare the input and output of our system, it would be convenient if we could display them on the same screen. Add Mux, which can be found under Commonly Used Blocks, to the design window in front of a scope. In this example, Mux has two inputs (reference signal and output) and one output (to the Scope).

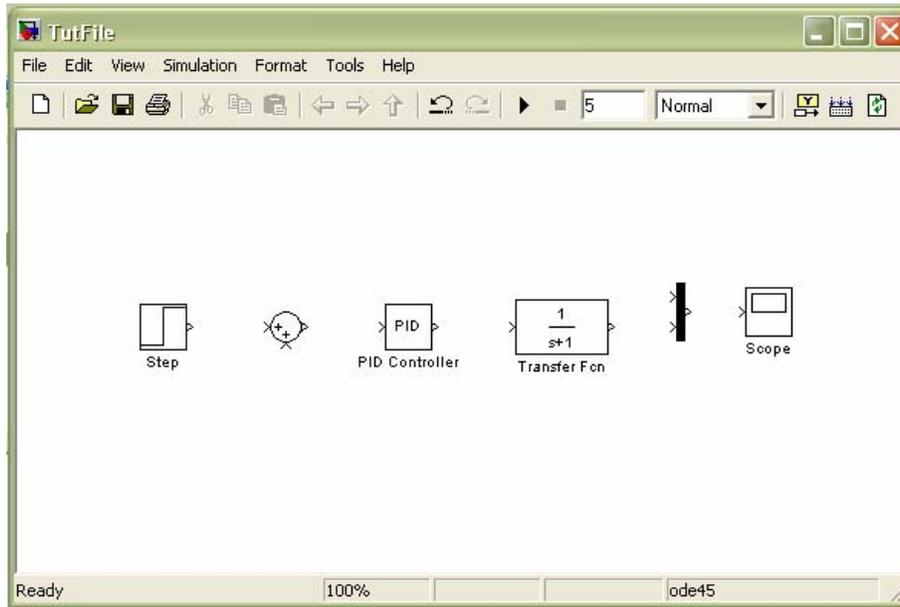


Figure 7: Addition of a Mux and a Scope

Now that the system is put together, we need to wire it up. Place the mouse to the output of a step function block, left click, hold and drag the wire to the input of the summation junction. Do the same for the rest of the system:

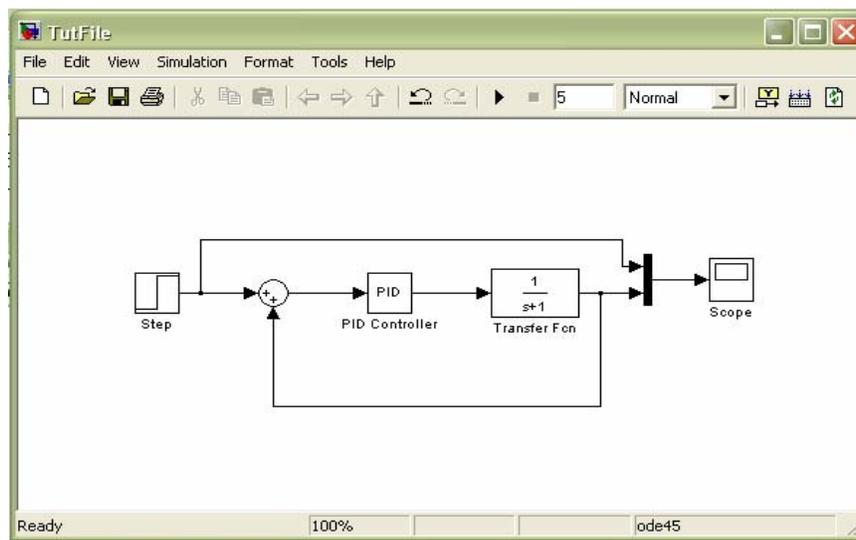


Figure 8: Complete Closed Loop Control System

Next step is to set all of the blocks parameters.

1. Double-click on the Step Block Diagram and set Step Time to 0, Initial Value to 0 and Final Value to required value, which is 3.

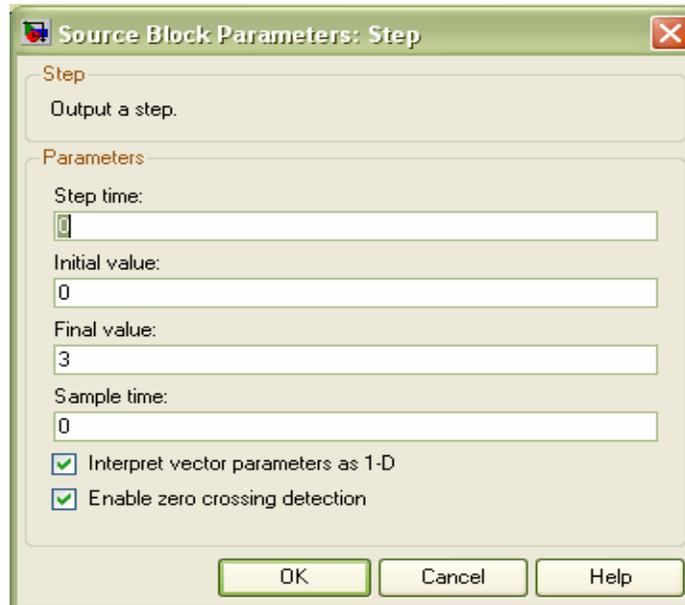


Figure 9: Setting Parameters

2. Double-click on Summation Junction icon and change second plus sign to minus.
3. We choose our plant to have numerator of 5 and denominator of: s^2+5s+6 . Double-click on the Transfer Function Icon and change the coefficients of the numerator to [5] and coefficients of the denominator to [1 5 6]. The system should look like in Figure 10:

In order to correctly set a scope, double-click on it icon, click on scope parameters and under scope parameters click on data history. Always uncheck all of the options in order to obtain correct results.

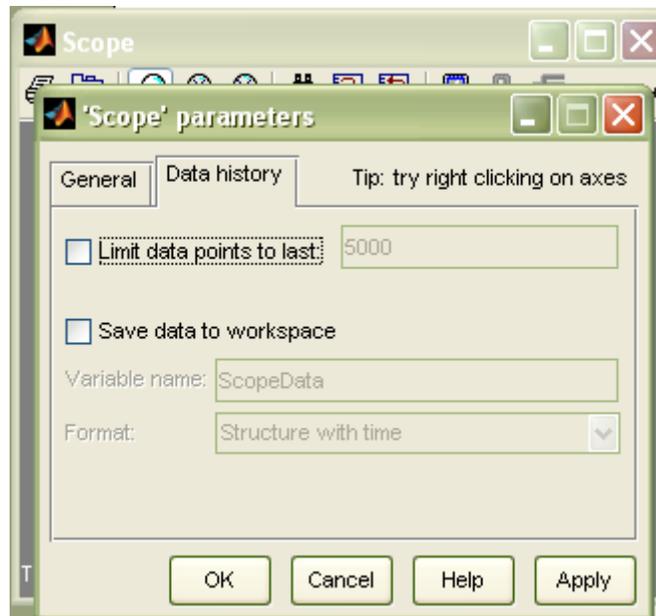


Figure 11: Setting up a scope

You can notice that the last block that needs to be set up is a PID controller. If you double-click on PID icon, new window will show 3 parameters: Proportional, Integral and Derivative. It is up to a designer to set these parameters in a way that the desired results can be obtained. Also, designer should be varying the simulation time which is set to 10 seconds by default.

Set the Proportional, Integral and Derivative gains to 20, 6 and 3, respectively. Set the simulation time to 5 seconds and click on Run icon on the Design Window tool bar (next to simulation time window):



Once the simulations are done, double-click on the scope icon and observe the results. Note that the output reaches the desired value of 3 within 1 second. If the display does not show all of the details or not refreshed, click on Autoscale button to rescale and refresh the result.

Using this tutorial, one can quickly start using basic functions of Simulink. Since the Simulink contains a large variety of functions and possibilities, it is impossible to put everything in one document. Therefore it is strongly advised to build up upon the basic knowledge of simulink in order to be able to design and simulate a lot more complicated systems. Also, finding out shortcuts and better work flow management will make your job a lot simpler.

Good Luck!