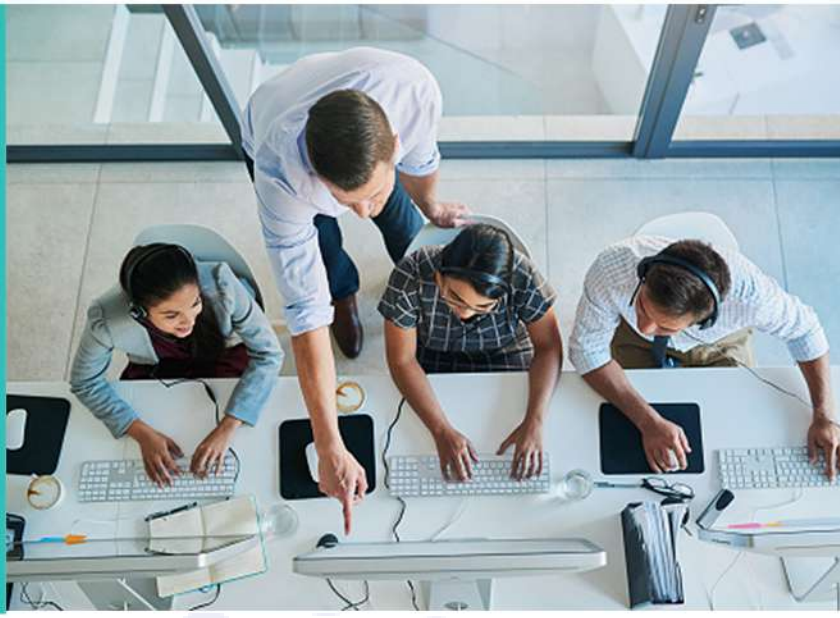
Andhra Pradesh State Skill Development Corporation

Skill AP
APSSDC



SciLab

Xcos Introduction

Xcos Introduction

INTRODUCTION

Numerical simulation is nowadays essential in the system design process. Complex phenomena simulation (physical, mechanical, electronics, etc.) allows the study of their behavior and results without having to conduct costly real experiments. Widely used in the world of industry, the future generation of engineers and scientists are trained since secondary school to the concepts of modeling and simulation.

Xcos is a Scilab tool dedicated to the modeling and simulation of hybrid dynamic systems including both continuous and discrete models. It also allows simulating systems governed by explicit equations (causal simulation) and implicit equations (acausal simulation). Xcos includes a graphical editor which allows to easily represent models as block diagrams by connecting the blocks to each other. Each block represents a predefined basic function or a user---defined one.

XCOS-MODELLING ENVIRONMENT

In this section, we describe the Xcos modeling environment. Xcos provides an integrated modeling environment where we can create, edit, and simulate dynamic models. After launching Scilab, the environment by default consists of the console, files and variables browsers, and command history.

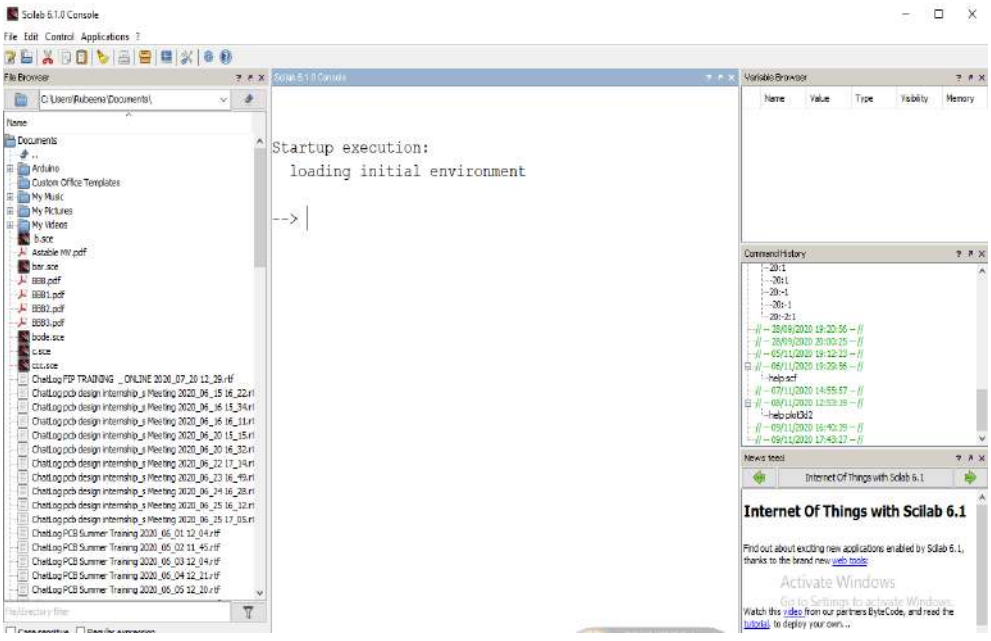


Fig.Console window

In the console after the prompt " --> ", just type a command and press the Enter key to obtain the corresponding result.

Xcos can be launched:

- From the toolbar, via the  icon, or
- From the menu bar, go to the Applications and click on Xcos, or
- From the console, in typing: -->xcos

Xcos opens by default with two windows:

- A palette browser which provides a set of predefined blocks,
- An editing window is a working space to design diagrams.

To design a diagram, just select blocks in the palette browser and position them in the editing window (click, drag, and drop). Blocks are then connected to each other using their different ports (input, output, event) in order to simulate the created model.

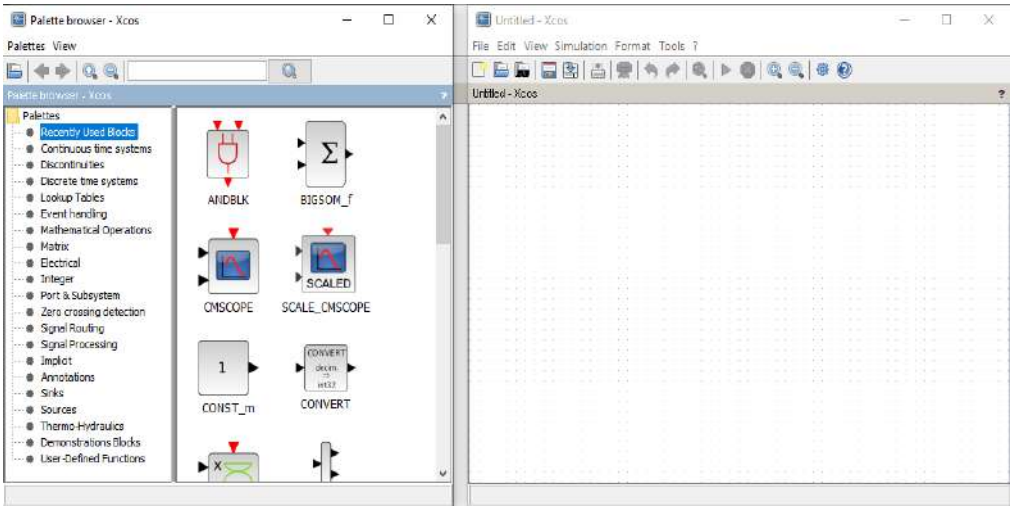


Fig.Palette browser window and Untitled Xcos window

PALETTE BROWSER

The palette browser window has a number of facilities to search a block. If a user is not sure which palette contains a particular block, the search facility can be very helpful in locating the block based on its partial name. For example, suppose we want to list all the blocks that contain the word sum in their names. For this, we type sum in the search box and press search.

The palette browser lists all the blocks related to sum, as shown in the figure.

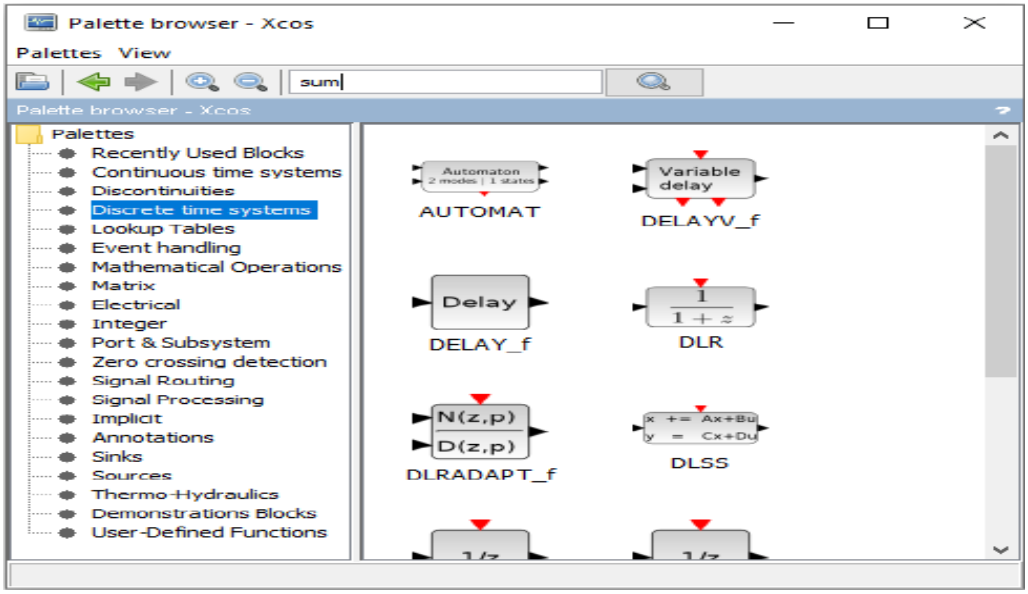


Fig.Finding blocks with the help of search command

SIMULATION OF FIRST ORDER TRANSFER FUNCTION

Xcos has a very powerful and easy to use graphical editor for creating models using blocks.

We will now simulate a first-order system with a step input.

To begin, I will select a transfer function block from the Continuous-time systems palette.

Drag this block to the Untitled-Xcos window.

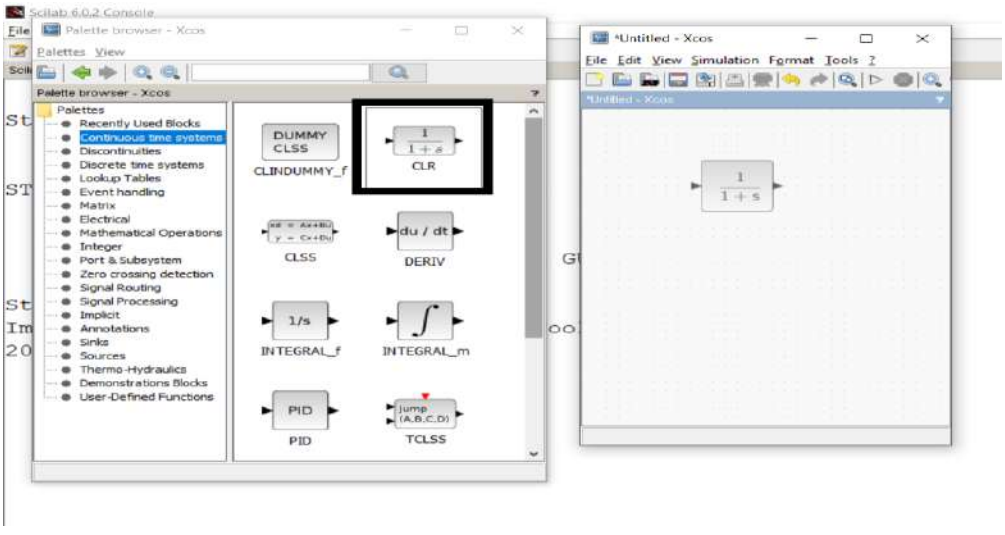


Fig.Selecting transfer function block

Select the required source in the Sources palette.

I will scroll down and use the STEP FUNCTION block.

I will drag and place it before the transfer function block.

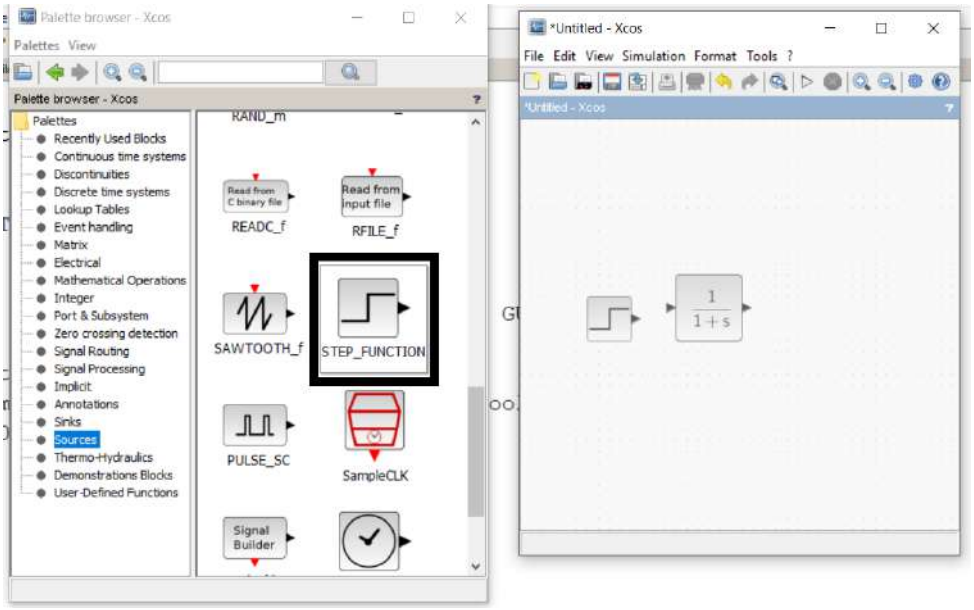


Fig.Selecting step input block

Similarly, the output is displayed using the CSCOPE block, which is available in the Sinks palette.The CSCOPE block is placed after the transfer function block.

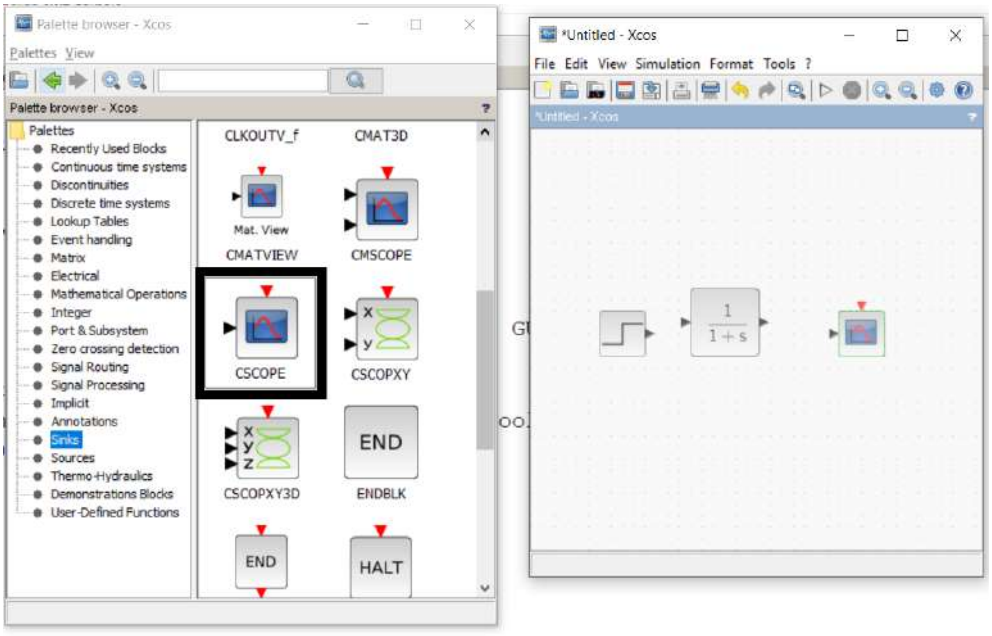


Fig.Selecting cscope block

The red input port in CSCOPE denotes that this block is an “event-driven” block.

It needs an event input for execution.

An event generator block is available in the Event handling palette.

The name of this block is Clock underscore c.

Drag and place this block above the CSCOPE block.

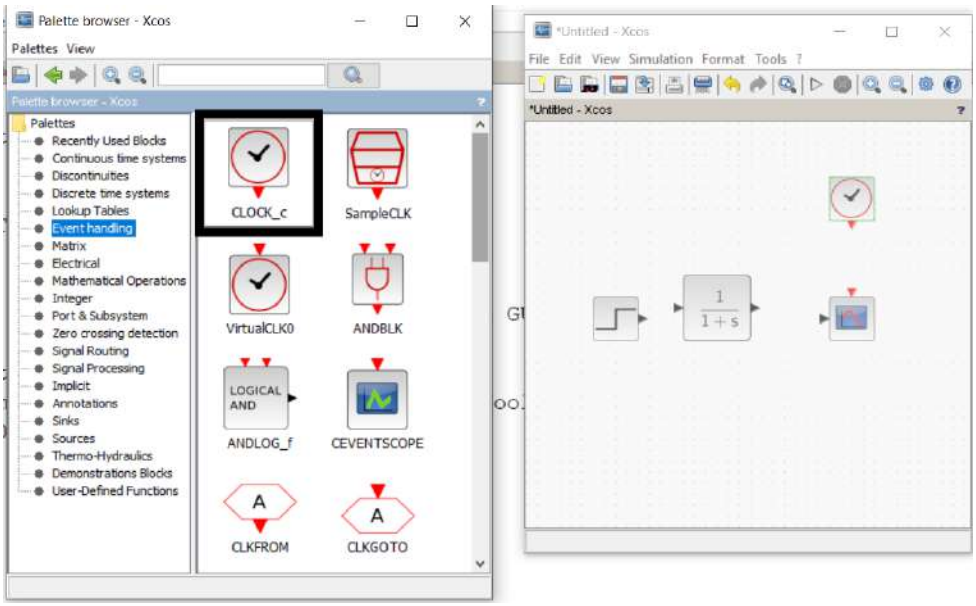


Fig.Selecting clock block

We have collected all the required blocks to do the simulation.

Let us now connect the blocks together.

Select the output port of the step function block and connect it to the input port of the transfer function block.

Notice that the selected input port gets highlighted in green color.

Similarly, connect the remaining blocks as shown.

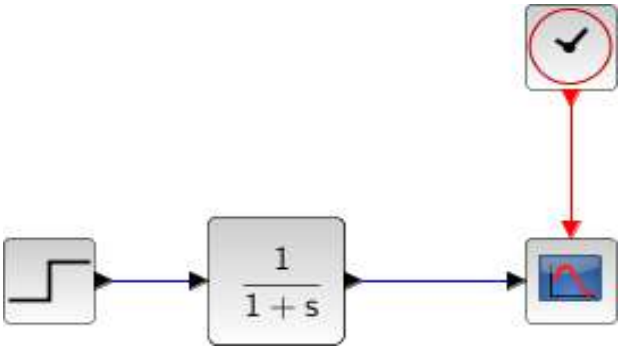


Fig.first order transfer function

Now we will set the parameters of each block.

First, go to the step block and double-click on it.

A pop-up window appears, asking for the value of Step Time, Initial Value, and Final Value.

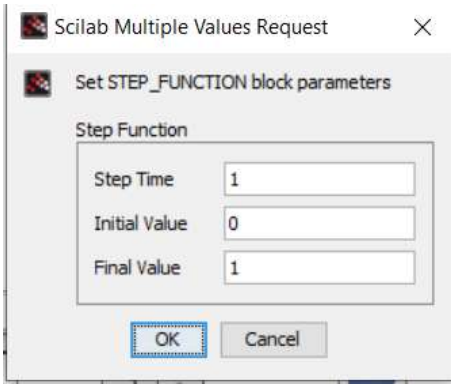


Fig. setting block parameters

Step Time is the time at which the step change will occur.

We will keep it as 1, which is the default value.

Initial Value is the initial output value of the step function.

We will keep it as 0, which is the default value.

The final Value is the output of the step function after Step Time has lapsed.

We will change it to 2.

Click on OK.

Follow a similar procedure to configure any other block.

For the transfer function block, the following configuration is required.

The numerator value in the Laplace domain has to be entered.

We will keep it as 1, which is the default value.

The denominator value in the Laplace domain has to enter.

We will change it to 2 asteric s plus 1.

Click on OK.

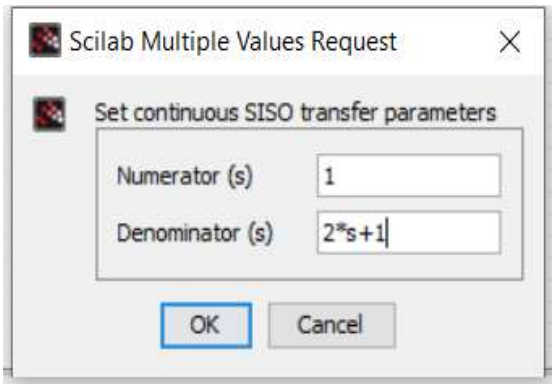


Fig.set transfer function parameters

Double click on the CSCOPE block, to configure the following parameters.

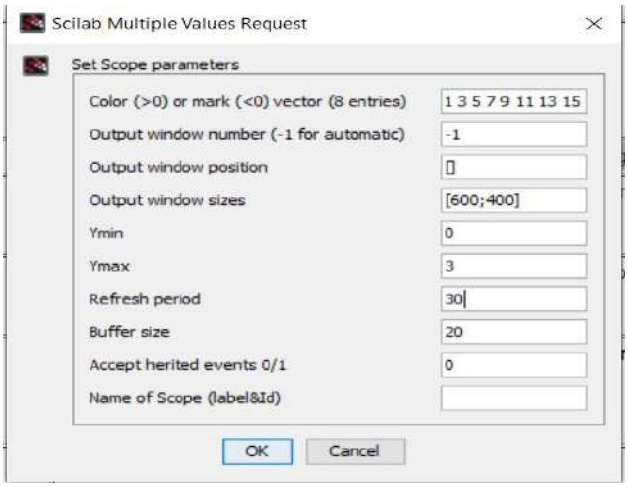


Fig.set scope parameters

The value of Ymin and Ymax should be set, depending on the range of the value, of the variable to be plotted.

Set the value of Ymin to 0.

Since I have given step input as 2, to display the output on the graph, I should change the Ymax to any higher value.

We will change it to 3.

Make a mental note of the default value of the refresh period.

The default value is 30.

We will keep the other parameters unchanged.

Click on OK.

Now go to the Simulation in the menu bar and click on Setup.

A pop-up window appears.

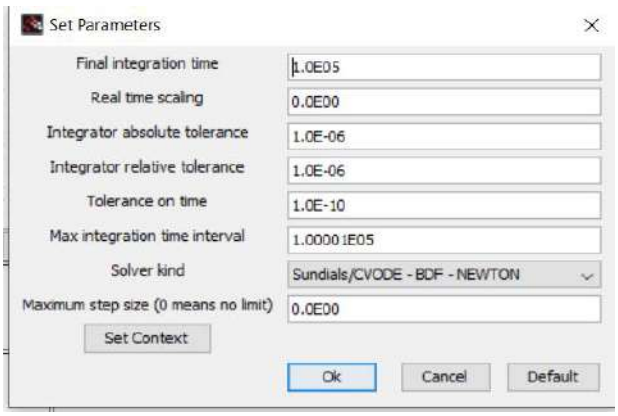


Fig. change the final integration time

We will change the value of Final integration time.

The final integration time decides how long the simulation will run.

Recall the value of refresh period of the CSCOPE block.

I will keep the value of Final integration time equal to the value of refresh period of CSCOPE block.

So, I will change it to 30.

Keep the other parameters unchanged.

Click on OK.

Now press Control + S to save the file, with a suitable file name.

I will save it as firstorder.xcos

To start the simulation, click on the Start button, available on the menu bar of Xcos window.

A Graphic window will open, showing the step response of the first-order transfer function.

We can save this plot as an image file.

To save the graph, go to the File menu and click on “Export to” option.

I will name it as firstorder.png and click on OK

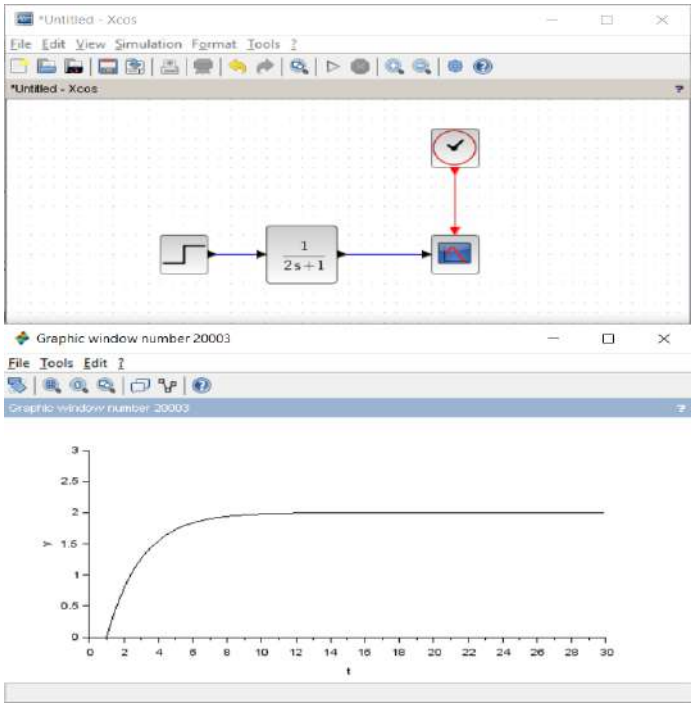


Fig.simulate the system and set the output

There are certain parameters that we chose to keep as the default value.

These can be changed.

For example, In the Clock underscore c block, the period, which means the sampling period and initial time can be set.

In the CSCOPE block, the output window number, position, size, buffer size, graph color etc can also be set.