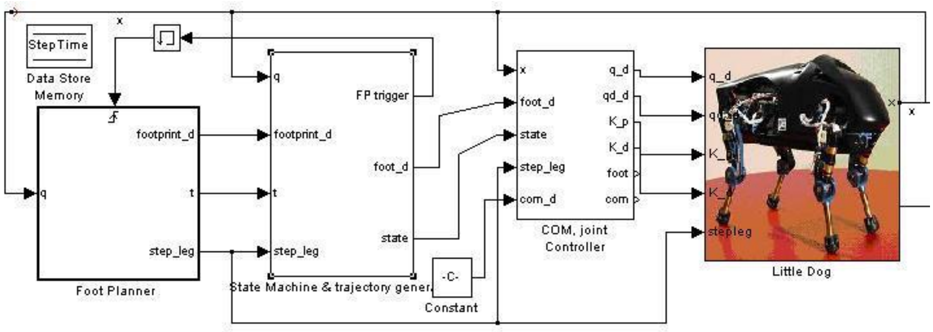
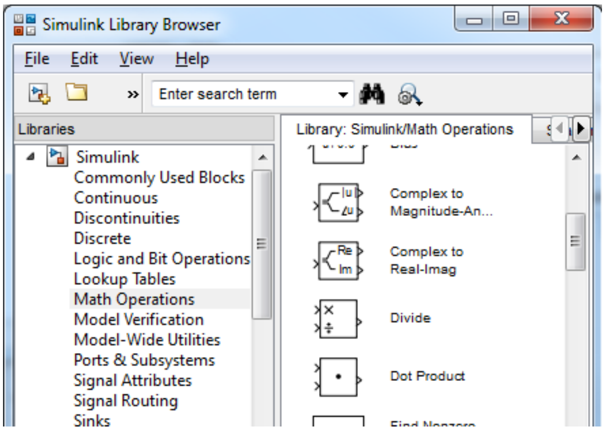


Exercise 10 – Simulink.

In this exercise we are going to have a look at an introduction to Simulink implementing a very basic model followed by modeling a mass-spring-damper mechanism, (something you will cover in detail in Vibration and Control in your second year).

Firstly launch MATLAB and then launch Simulink by typing simulink in the command line or press the simulink button in MATLAB .

The Simulink Library Browser will open. This is where all the Simulink blocks are found that can be placed in the models. The blocks we will use in these exercises can be found in the Continuous, Math Operations, Sinks and Sources libraries. Browse through these libraries to see the available function blocks.

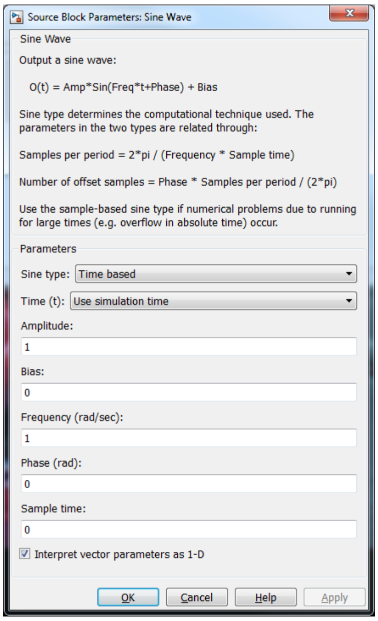


We are going to create a new simulation. First off we'll do a simple model that will model a sine wave and add some random noise.

1. Open a new model by selecting the new Model button . This will open a new window into which we can start to build our simulation.

i. We need to add a few blocks to our model. Firstly we'll add a sine wave block. This is found in the Sources library. In the Simulink Library Browser, click on Sources and then scroll down until you can find the Sine Wave block. Click and drag the block onto your model window.

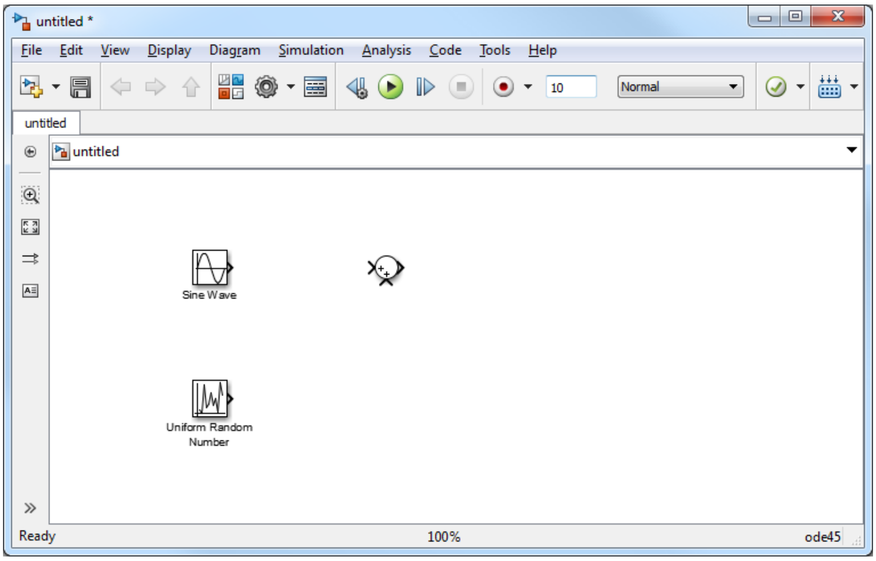
ii. You can change the parameters associated with the Sine Wave by double clicking on the function block.



Change the amplitude to 10 and the frequency to 2*pi. Select OK.

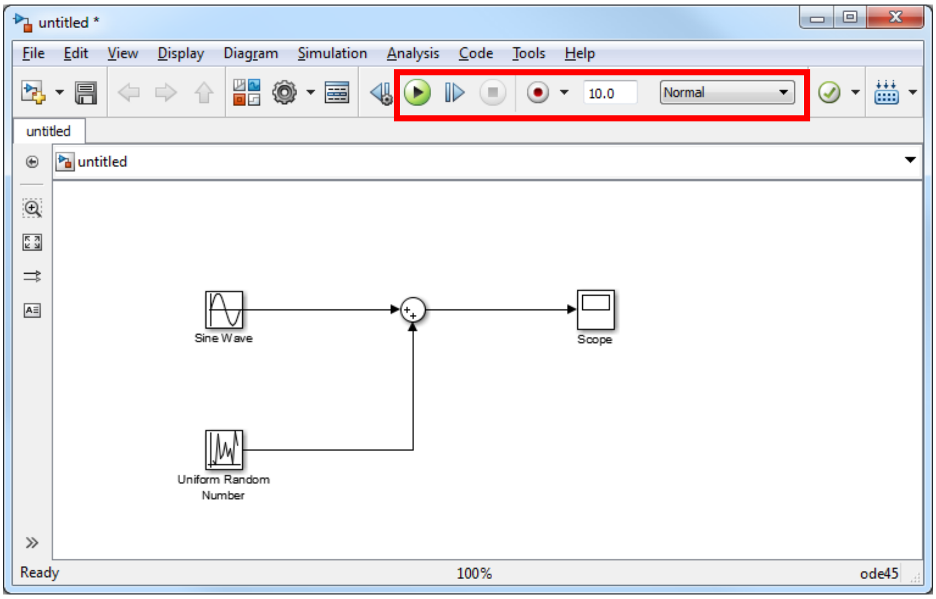
iii. We are going to add some noise to this signal using a further Source block. Find the Uniform Random Number block and place this on the model window. Again double click on Uniform Random Number and change the minimum to -2 and maximum to 2 and the sample time to 0.01.

iv. Next we are going to sum the sine wave and the random number. To do this we can add a sum block. This can be found in the Commonly Used Blocks or Math Operations libraries. Your model should look like this:



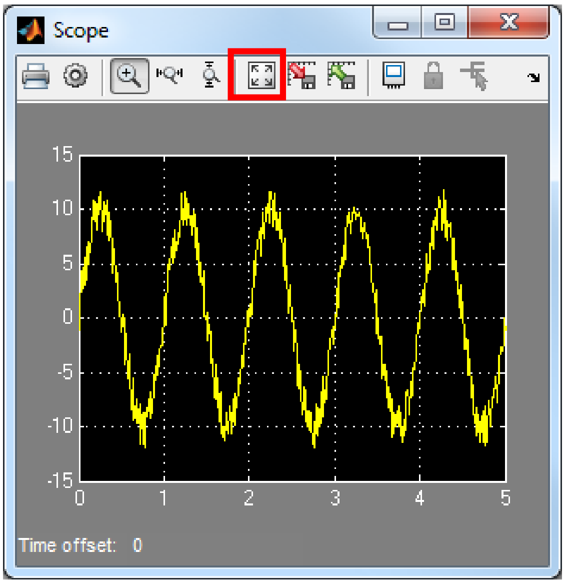
v. Now we need to connect the block to show where the data is flowing. Use the mouse and left click and hold on the output of the Sine Wave block and drag across to the input of the sum block. You should see them connected. Alternatively we could left click on one block, then press and hold ctrl key and left click the block we want to connect it to. Do this to connect the Uniform Random Number block to the sum block.

vi. Now we want to view the result. In the Library Browser go to Sinks and select Scope. Drag it into your model window and wire up to the output of the sum block. Your finished model window should look like this:



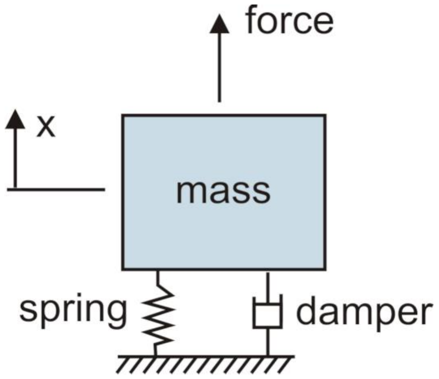
vii. We are now going to run the simulation. The area highlighted above shows some of the controls to run the simulation. The default time the simulation will run for is 10 seconds, change it to 5 seconds, then press the play button to run the simulation.

viii. Once it's ran, double click on the scope block to open a plot figure showing the value wired to scope during the 5 seconds. In the figure window right click and select Autoscale or click the Autoscale button.



ix. You've now created your first Simulink model, save it as simple_model.mdl.

2. Now we are going to look at a more in depth example. Simulink is designed to solve differential equations and so solving problems including these are made relatively straight forward. We are going to look at a simple mass-spring-damper system below:



The system is described using the following equation:

$$F = m\ddot{x} + c\dot{x} + kx$$

where m is the mass in kg, c is the damping coefficient in Ns/m, k is the spring stiffness in N/m and F is the external force in N. The equation describes the motion of the mass attached to ground via a spring and damper in parallel when subjected to an external force. It can be rearranged to give us the acceleration:

$$\ddot{x} = \frac{1}{m}(F - c\dot{x} - kx)$$

As it can be seen, it is a second order differential equation. We are going to model this in Simulink to allow us to see how the system responds to the external force, F being applied.

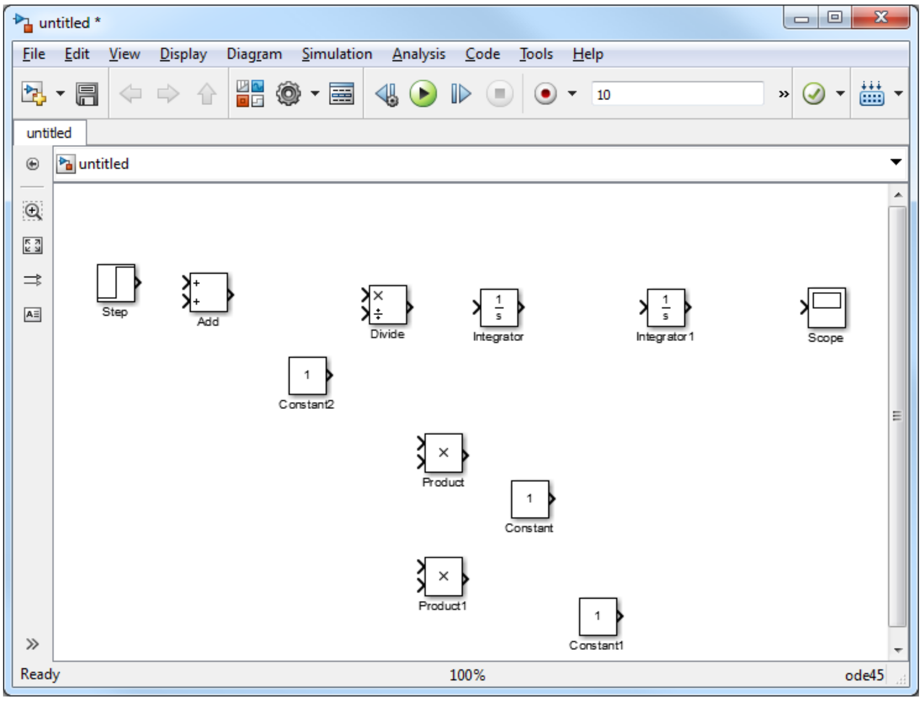
Follow the instructions below:

i. Open a new model window and save it as mass_spring_damper.mdl

ii. Place the following blocks on the model window:

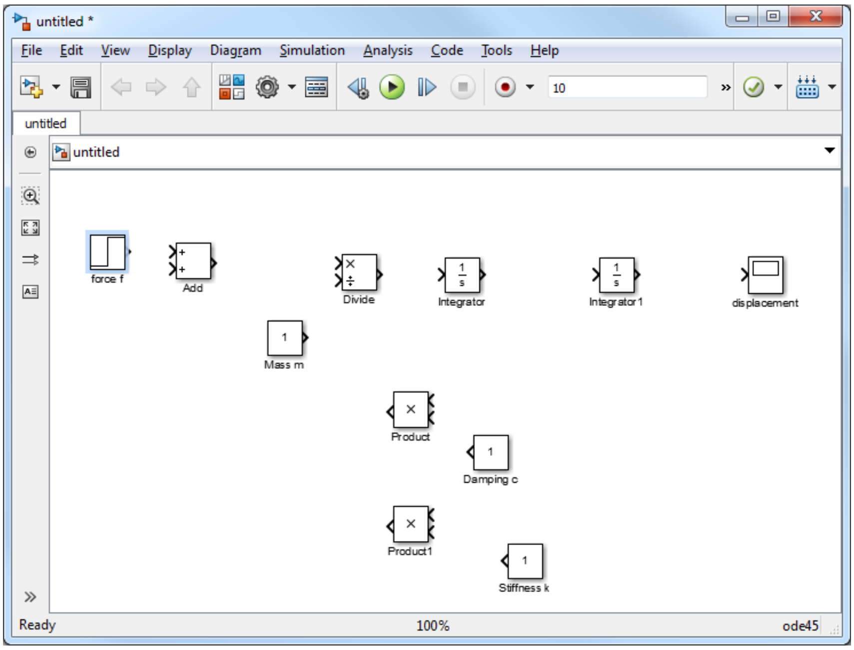
Block name	**Library**	**Number required**
Add	Math operations	1
Product	Math operations	2
Divide	Math operations	1
Integrator	Continuous	2
Step	Sources	1
Constant	Sources	3
Scope	Sinks	1

Rearrange the blocks to look like the following:

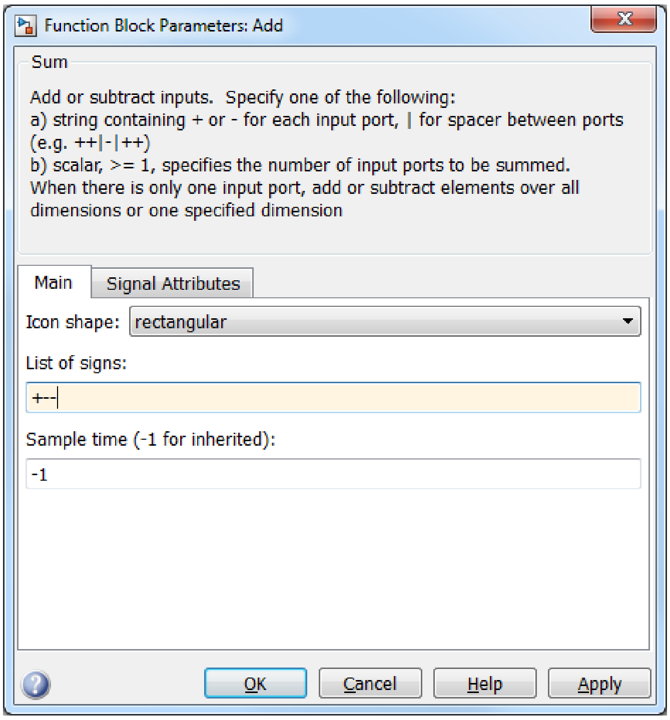


iii. In order for the wires to run the correct way, we need to change the direction of four of the blocks. Right click on one of the product blocks and select Rotate and Flip -> Flip Block. Do the same for the other product block and two of the constant blocks.

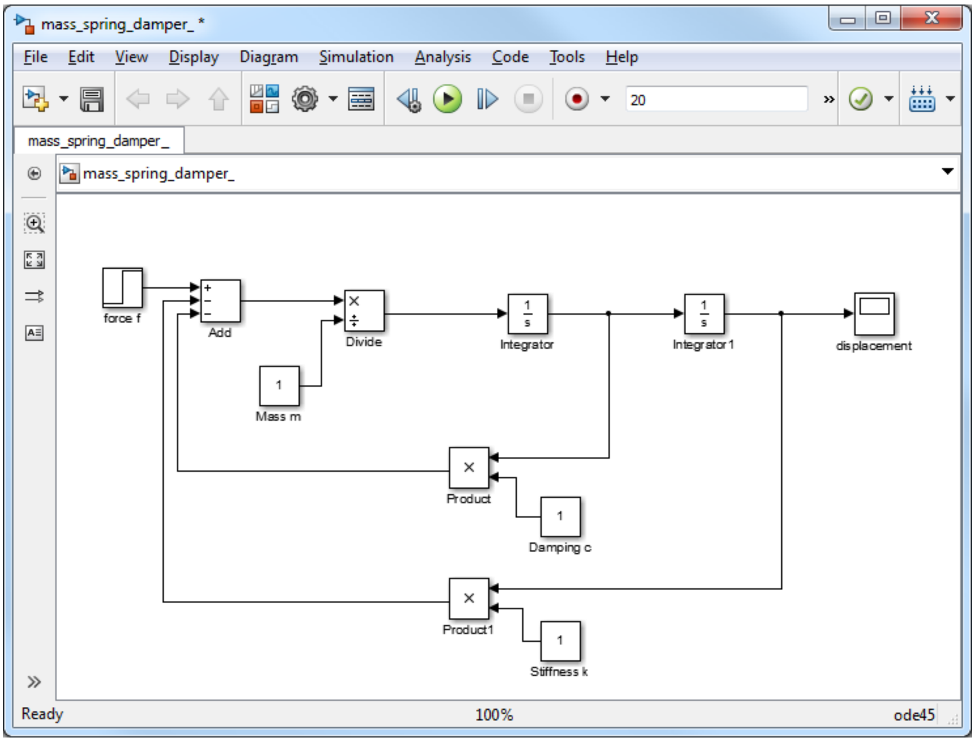
iv. Rename one constant block 'stiffness k', another 'damping c' and the third 'mass m'. You can do this by left clicking on the label. Rename Scope 'displacement' and Step as 'force f'. (note that the constant and product block could be replaced by a gain block) Your block diagram should look as below:



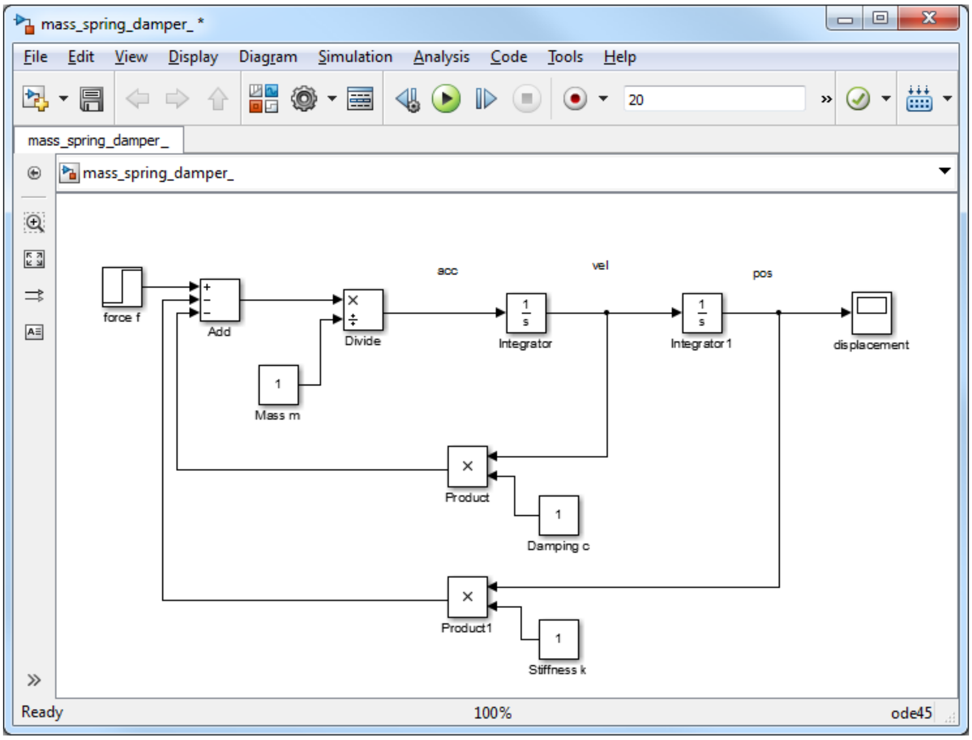
v. We need to alter the add block to include more terminals. If we double click the add block the Function Block Parameters window will appear. Change the List of signs to the following: +-- and press OK.



vi. We are now going to wire up the diagram. Add wires so that your model looks like the image below:



We can add labels to the model by double clicking on blank space, do this to add labels to the wires. Add labels ‘acc’, ‘vel’ and ‘pos’ as below:



Hopefully you can see how this relates to the second order differential equation, the value in the connector labeled 'acc' should be equal to:

$$\frac{1}{m}(F - c\dot{x} - kx).$$

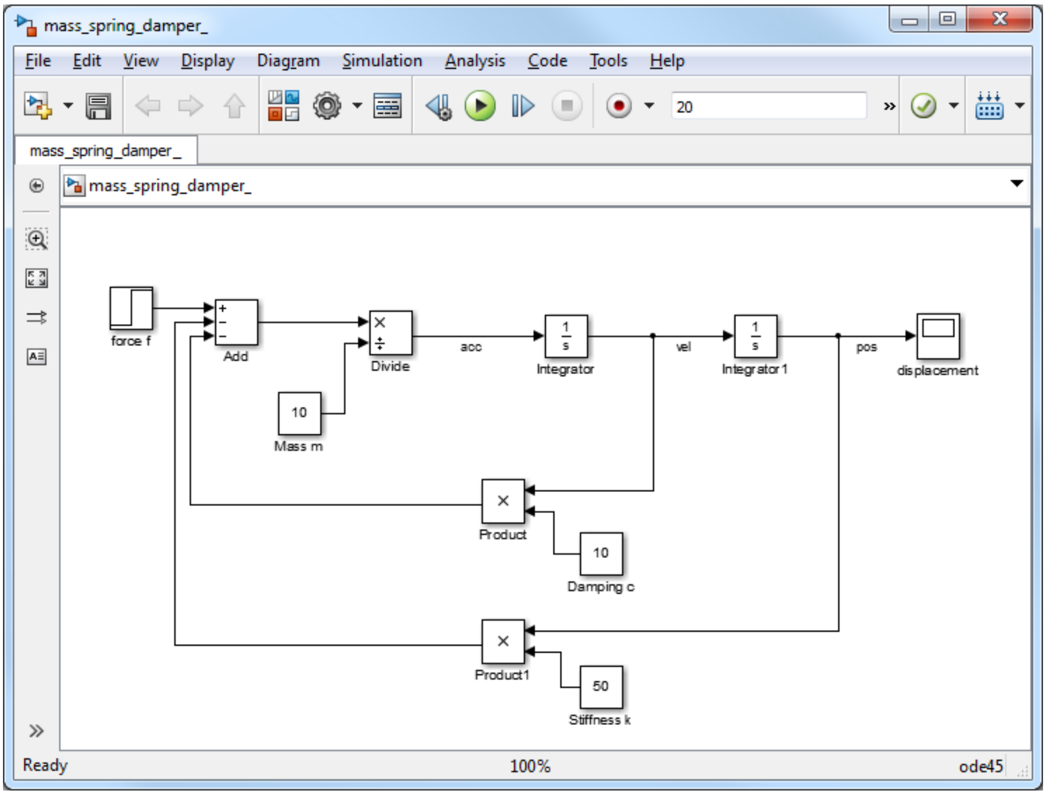
Simulink uses numerical solvers to find the solution, it is possible to specify which particular solver is used but this is beyond the scope of this module.

vii. Prior to running the code, we need to set the parameters used for F, m, c and k. We are going to use the following:

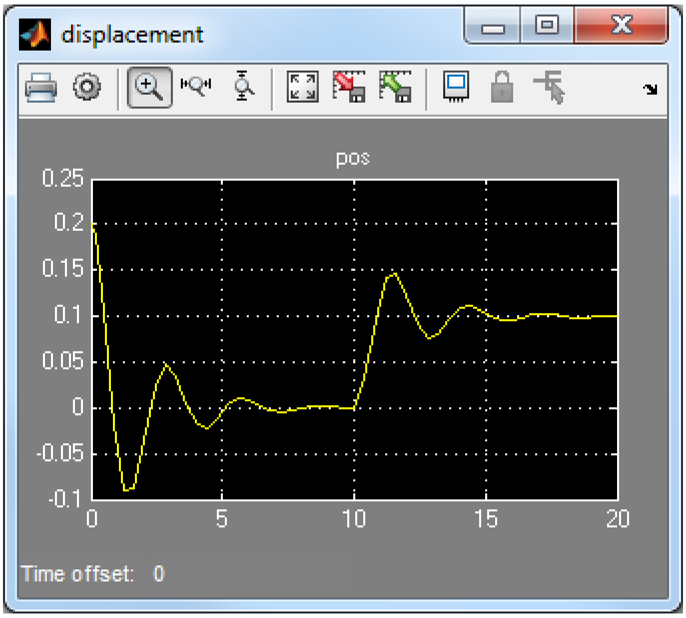
Constant	Block	Value	Units
m	mass m	10	kg
c	damping c	10	Ns/m
k	stiffness k	50	N/m
f	force input f	5 at step time 10 sec	N

To change the values in the three constants, double left click to open the block parameters and change the Constant Value to the above.
To change the step input, double click to open the block parameters and change Step time to 10 and Final Value to 5, initial value should be zero.. Remember to close all the block parameter windows by clicking OK.

viii. Finally we want to set an initial condition for the second integrator block. This will give us an initial displacement output. Double click on the right integrator block to open the Function Block Parameters. Set the Initial Condition value to 0.2. Press OK. The model should now look like this:



ix. We are now ready to run our simulation. Double click the displacement scope block to open the plot window. Now change the simulation time in the model menu bar to 20.0. Press the run button. Press the binocular icon on the displacement figure and you should see the plot below:



The plot shows the displacement of the mass when subjected to an initial displacement and then an external force. Try changing the stiffness and damping coefficients and see the effect it has on the response. You'll cover more of the theory about this in MECH2620 – Vibration and Control.

End of exercise 10