Lab 6 – MATLAB EXPLORATION

Problem 1

I want you guys to figure it out.

Problem 2 Hints

Commands to be used for following questions are given in bold, underline – To find more details use Matlab command window – type "help (command you need without these braces)" For example – **help tf**

Part a – **tf** command to get output needed

Part b – **expr or tf** to compute, **ezplot** to plot necessary graph

Part c - **tf** to compute , **bodemag** to plot

Part d – all plots related command – **pzmap , impulse , step**

Part e - **dsolve**

Analytical study of integrator

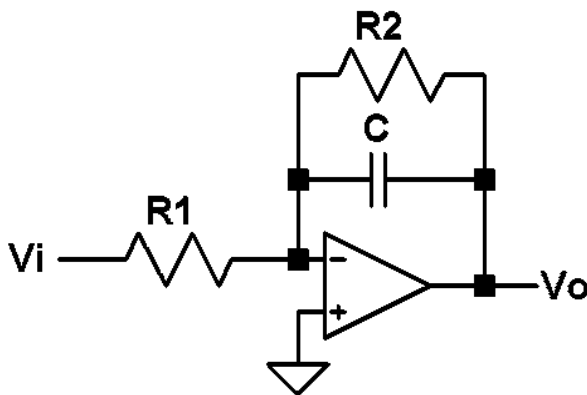You have to find Transfer function of circuit below

Figure 1. Integrator with gain

This is called a Integrator of First order since we have only one complex component in the circuit. For transfer function – Read through the below notes.

The Integrator

The fundamental integrator circuit is constructed by placing a capacitor C, in the feedback loop of an inverting amplifier.
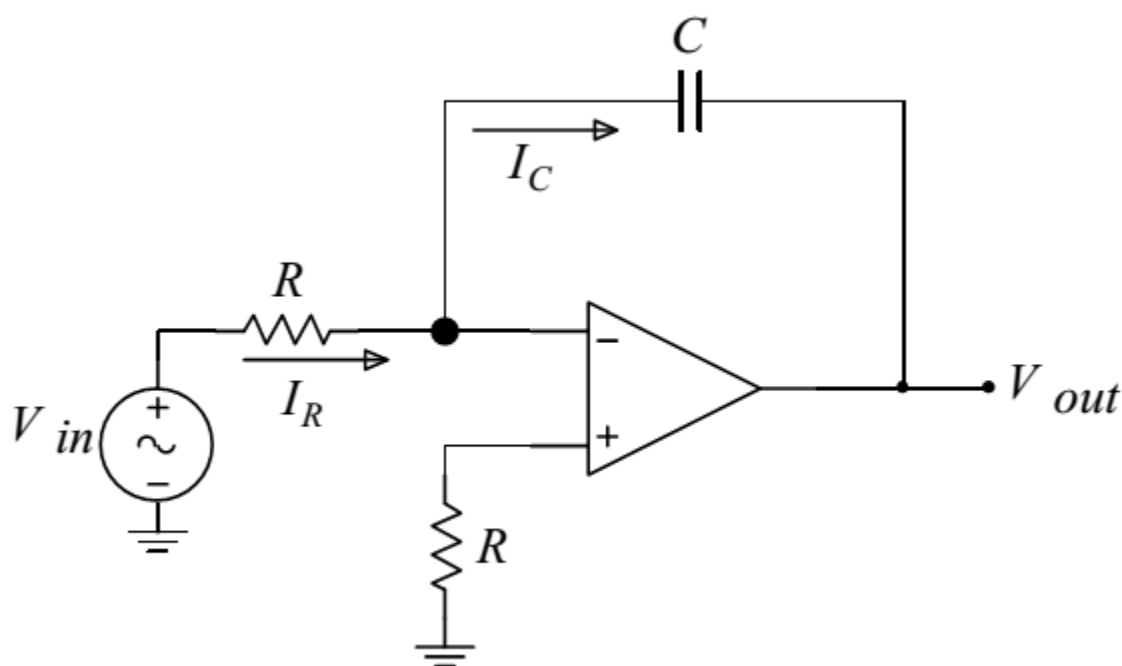

Figure 2. The integrator circuit without gain

Assuming an ideal op-amp, current conservation at the indicated node gives

$$I_R = I_C$$

$$\frac{V_{in}}{R} = -C\frac{dV_{out}}{dt}$$

Rearranging Equation and integrating from 0 to t, we obtain

$$\int dV_{out} = -\int \frac{V_{in}(\tau)}{RC} d\tau \quad \Rightarrow \quad V_{out}(t) = -\frac{1}{RC} \int_0^t V_{in}(\tau) d\tau + V_{out}(0)$$

The output voltage is thus the integral of the input. The voltage $V_{out}(0)$ is the constant of integration and corresponds to the capacitor voltage at time $t = 0$.

The frequency domain analysis is obtained by expressing the impedance of the feedback components in the complex plane. The transfer function may thus be written as

$$\frac{V_{out}}{V_{in}} = -\frac{Z_C}{Z_R} = \frac{\frac{1}{j\omega C}}{R} = \frac{j}{\omega RC}$$

For $\omega=0$ the gain is infinite and it falls at a rate of 20dB per decade of frequency change. The infinite gain for dc signals represents a practical problem for the circuit configuration of Figure 27. Since the equivalent circuit of a capacitor for $\omega=0$ is an open circuit, the feedback path is open. This lack of feedback results in a drift (cumulative summing) of the output voltage due to the presence of small dc offset voltages at the input. This problem may be overcome by connecting a resistor, R_F, in parallel with the feedback capacitor C as shown.
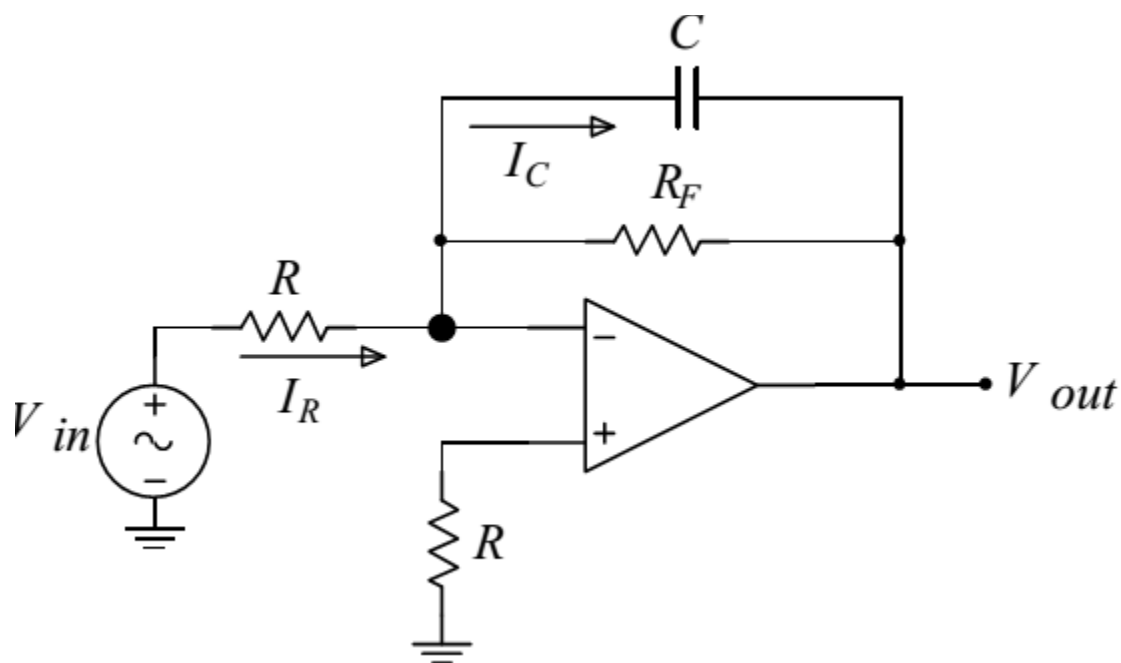
Figure 3: Integrator with gain

The feedback path consists of the capacitor C in parallel with the resistor RF. The equivalent impedance of the feedback path is=

$$Z_F = \frac{R_F Z_C}{R_F + Z_C} = \frac{\dfrac{R_F}{j\omega C}}{R_F + \dfrac{1}{j\omega C}} = \frac{R_F}{1 + j\omega R_F C}$$

The transfer function $\dfrac{V_{out}}{V_{in}} = -\dfrac{Z_F(\omega)}{Z_1(\omega)}$ becomes

$$\frac{V_{out}}{V_{in}} = -\frac{Z_F(\omega)}{Z_1(\omega)} = -\frac{R_F}{R_1}\frac{1}{1 + j\omega R_F C} = -\frac{R_F}{R_1}\frac{1}{1 + \dfrac{j\omega}{\omega_H}}$$
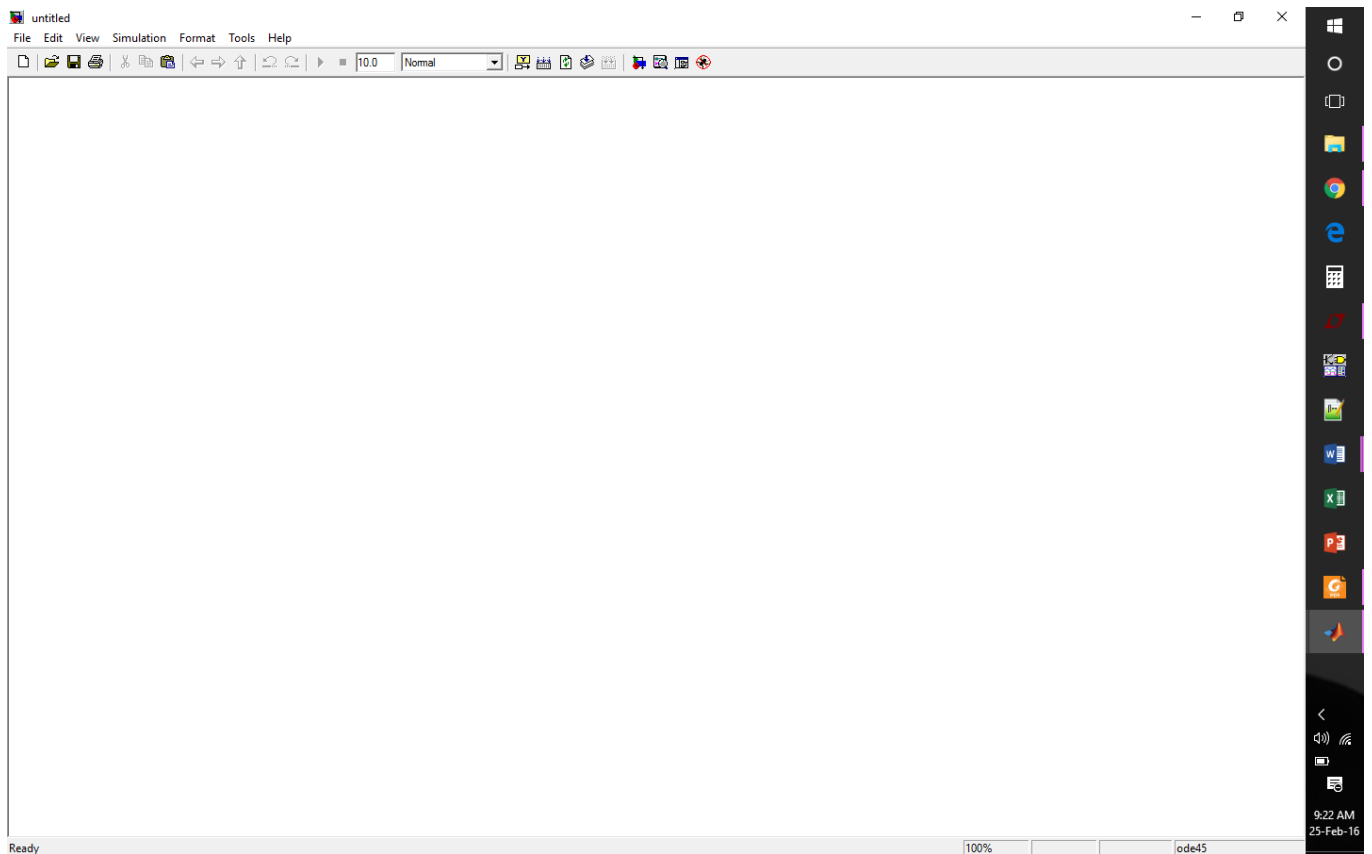
Now this final equation is of the form

$$\textbf{H(s) = Gain factor / 1 + sRC}$$

Hints are also given in the Lab manual

Now once you find the Transfer function – You have to model it using the blocks

Getting started with Simulink

Click **File ➔ New ➔ Model ➔** This brings up new blank window like this one

Here you have to add models to simulate your system at hand.

Now for adding block – Click on this icon in Tool bar – It is called library browser. It contains all the block necessary to model your system
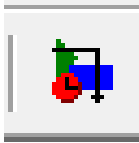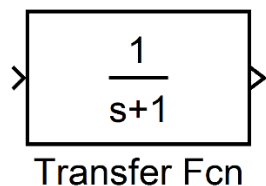
Use **library search box.** Some important block that have to be used are:

   i)   **Add Signal sources** – Square and Sine wave sources

Sine Wave

Step
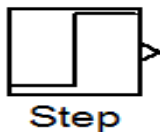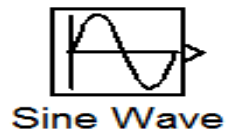
    ii)   Add **Transfer function model** – To model your transfer function

$$\frac{1}{s+1}$$

Transfer Fcn

    iii)   Add Scope to see the outputs – By default scope will have only scope channel you can add more by double clicking on the scope **after adding the scope in your model window**. Then click on the Parameter to change Number of axes. As shown below
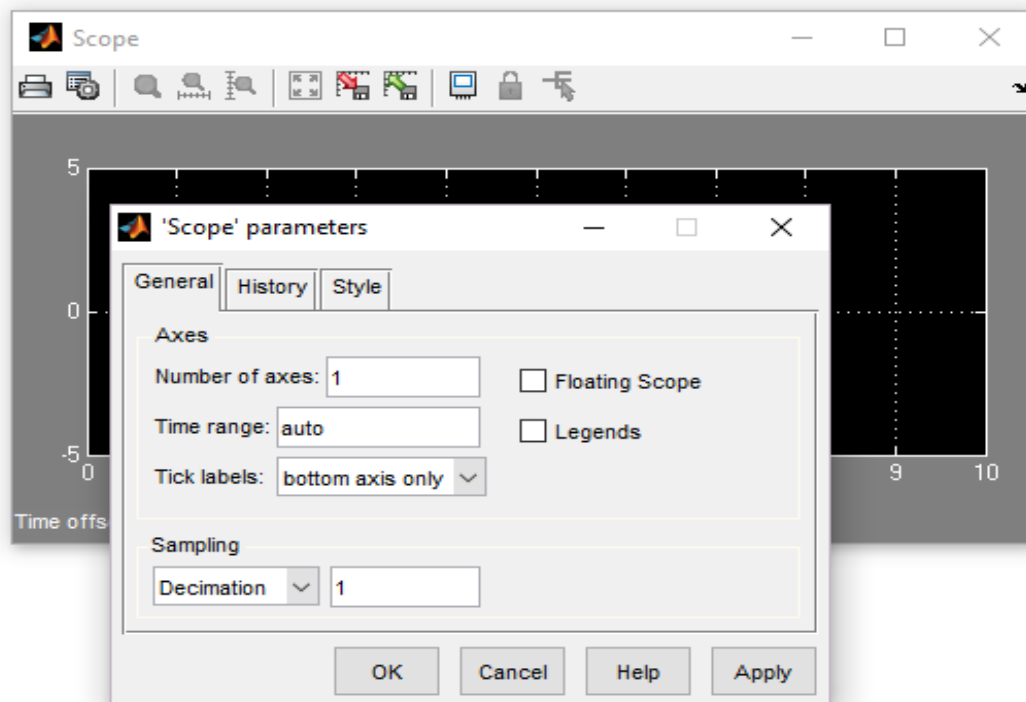
iv)   Search for **powergui** in **Library search box** then **add** it your model schematic.
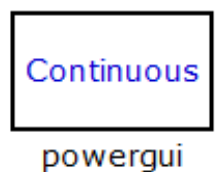
As of now you are done with placing with all the blocks. All you need to do is to connect them all logically and run the simulation. For more details see lab manual.

References

- *http://ocw.mit.edu/courses/electrical-engineering-and-computer-science/23_op_amps2.pdf*
- *Matlab Tutorials*