# EXPERIMENT 2

BASIC EXPERIMENTS USING LTSPICE

OBJECTIVE

To become familiar with LTSpice and performing basic experiments using it. We will be using the following components in this experiment:
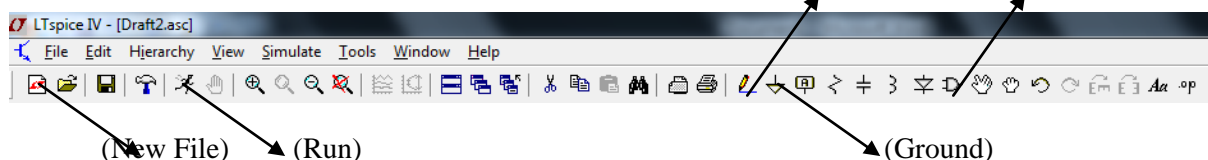
a) Voltage Source
b) Current Source
c) Resistor
d) Voltage controlled voltage source (VCVS)
e) Voltage controlled current source (VCIS)
f) Current controlled current source (ICIS)
g) Current controlled current source (ICVS)
h) Voltage controlled switch
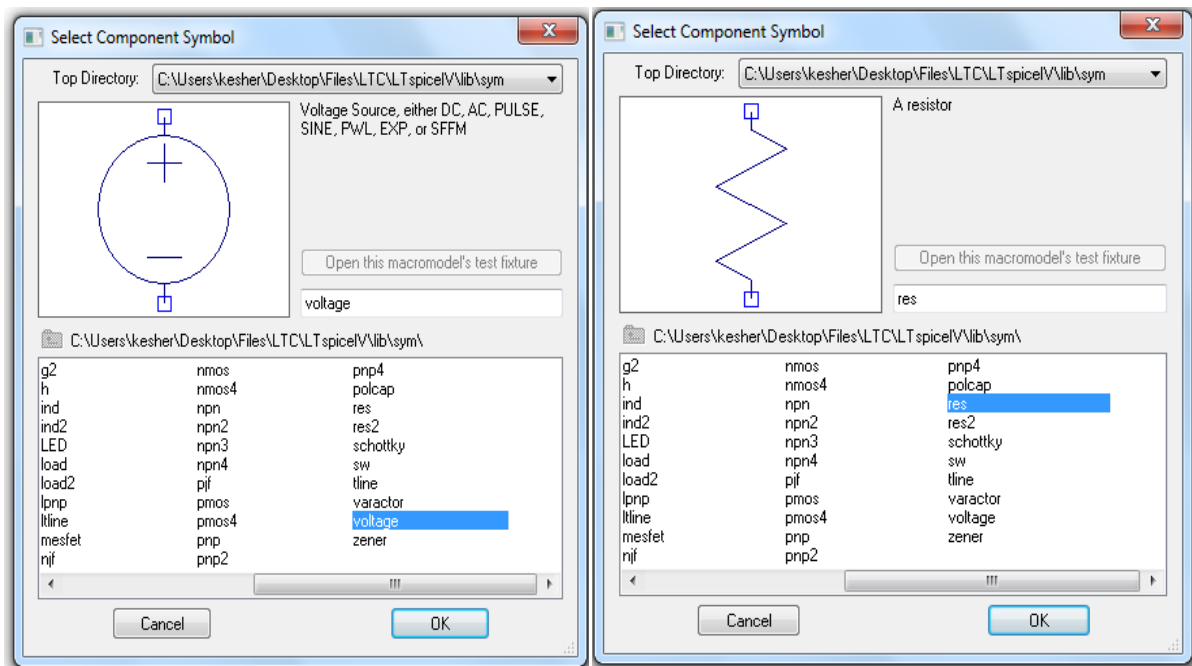i) Current controlled switch

TASKS

1. To verify Ohm's law using LTSpice.
   a) Create a new file in LTSpice and implement the following design.
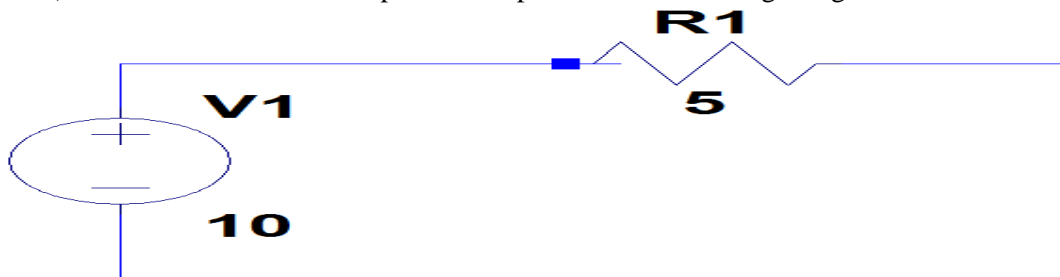
   b) The voltage source and resistor components are taken from the component tab in the toolbar.   (Wire)   (Component tab)

   (New File)   (Run)   (Ground)

Connect the negative terminal of voltage source,V1 to ground. Use wire to connect the components.

c) The value of voltage and resistance can be given to the components by right clicking on it. Give the values to the voltage source and resistor as prescribed in the design.

d) **.op command- Finding the DC operating point: -** DC solution is performed in order to find the operating point of the circuit. Use .op if you wish this operating point to be found. The results will appear in a dialog box.

   In the "Simulate" tab, go to the "edit simulation command" option. Go to the "Dc op pnt" tab and click ok. Place the .op command any place in the design. Now, hit the run tab. All the nodal voltages and current through all the devices will appear in the pop up window.

e) **.DC -- Perform a DC Source Sweep Analysis:-** This performs a DC analysis while sweeping the DC value of a source. It is useful for computing the DC transfer function of an amplifier or plotting the characteristic curves of a transistor for model verification.
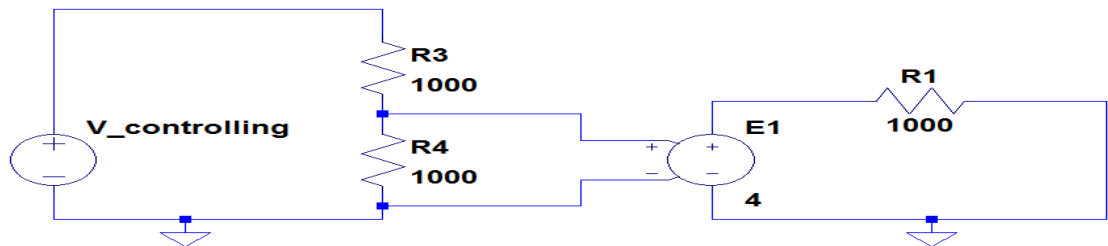   In the "edit simulation command" option (in the "simulate" tab), go to the DC Sweep tab. Fill the entries as shown below.
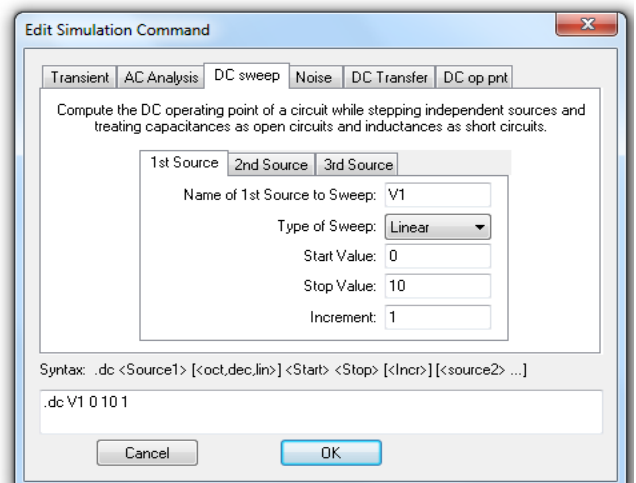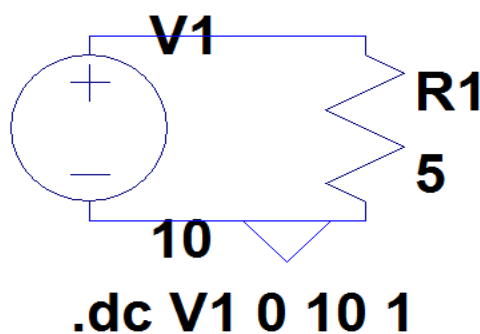
f) Hit the run tab and Observe the output waveform by putting the voltage marker on R1.

2. To implement controlled voltage and current sources in LTSpice.
   a) Select the Voltage controlled voltage source (VCVS) component (component name "e") from the component tab.
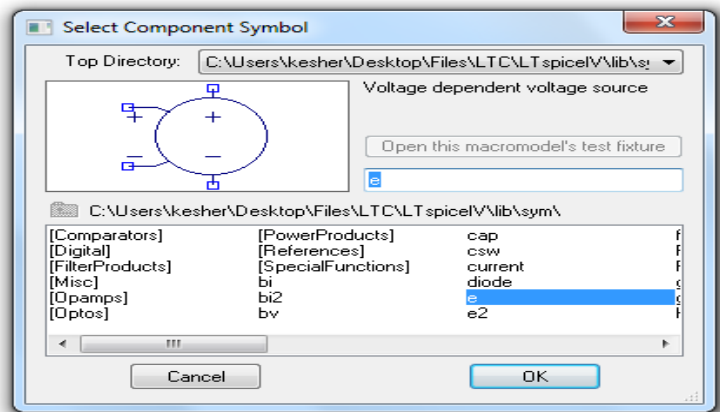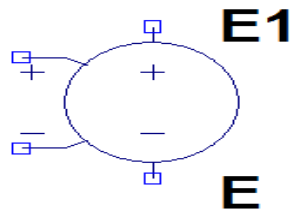
   b) Make the following design in LTSpice.

   .dc V_controlling 0 5 0.01

In this design, V_controlling is a dc voltage source, whose values are swept from 0 to 5V with a step of 0.01V. Right click on the voltage source V_controlling and configure it for the before said parameters (similar to configuring V1 in task 1.(e)).

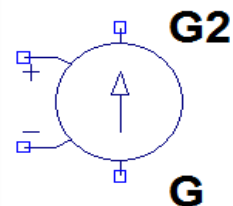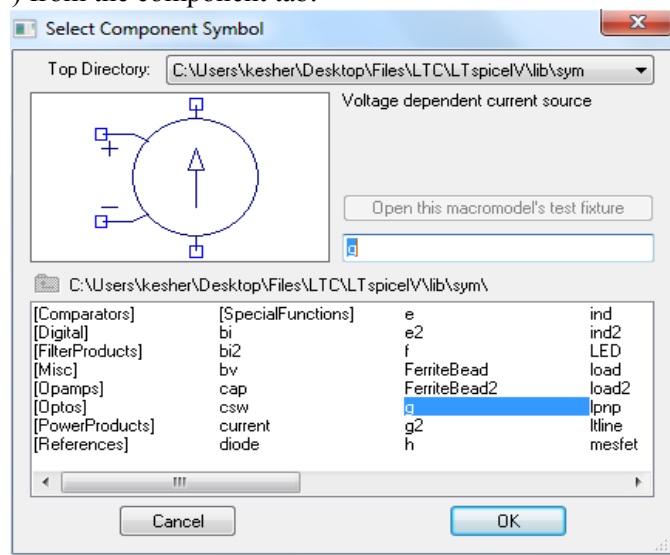Right click in the device E1 and configure it as shown below.
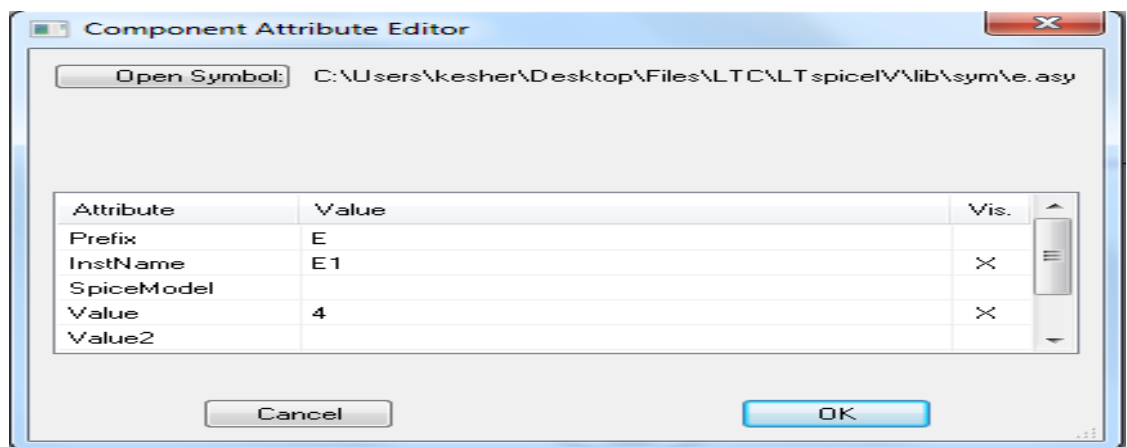
The number in the value field specifies the gain factor of VCVS.

c) In the "edit simulation command" option (in the "simulate" tab), go to the DC Sweep tab and do the appropriate setting.

d) Hit the run tab and observe the output voltage waveform by putting the voltage marker on R1 and controlling voltage waveform by putting a voltage marker on R4.

e) Click on the VCVS and do the following setting in the Value field.
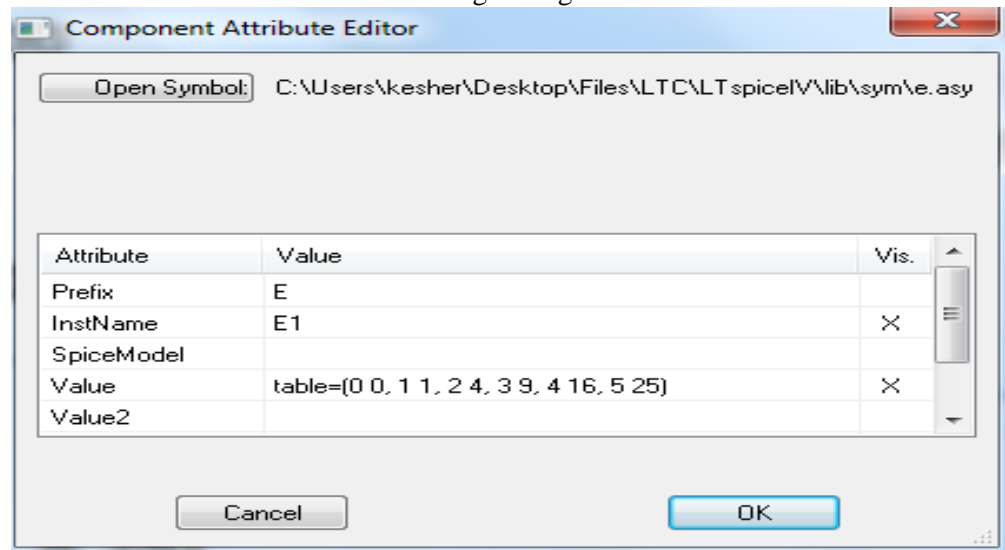
A look-up table is used to specify the transfer function. The table is a list of pairs of numbers. The second value of the pair is the output voltage when the control voltage is equal to the first value of that pair. The output is linearly interpolated when the control voltage is between specified points. If the control voltage is beyond the range of the look-up table, the output voltage is extrapolated as a constant voltage of the last point of the look-up table.

f) Hit the run tab and observe the output voltage waveform by putting the voltage marker on R1 and controlling voltage waveform by putting a voltage marker on R4.

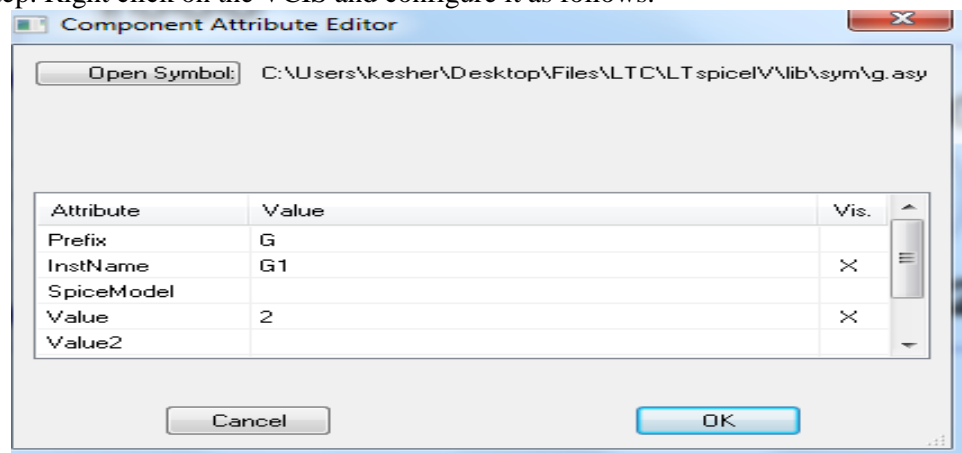g) Select the Voltage controlled current source (VCIS) component (component name "g") from the component tab.
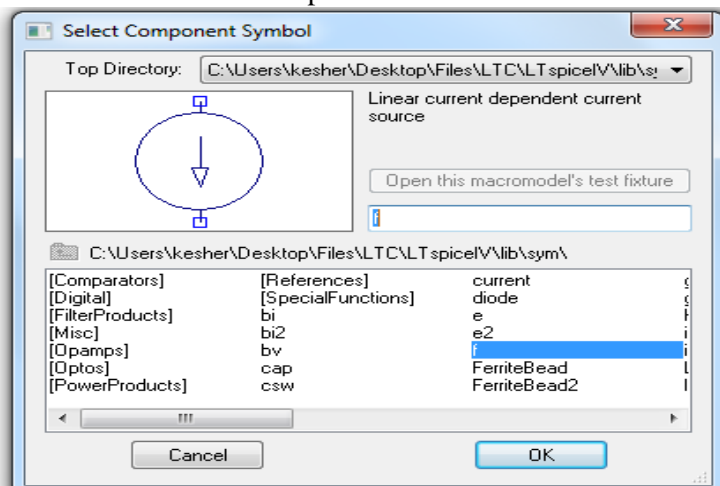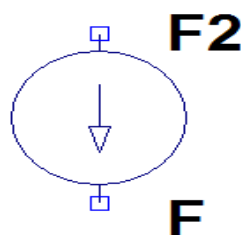

h) Make the following design in LTSpice.
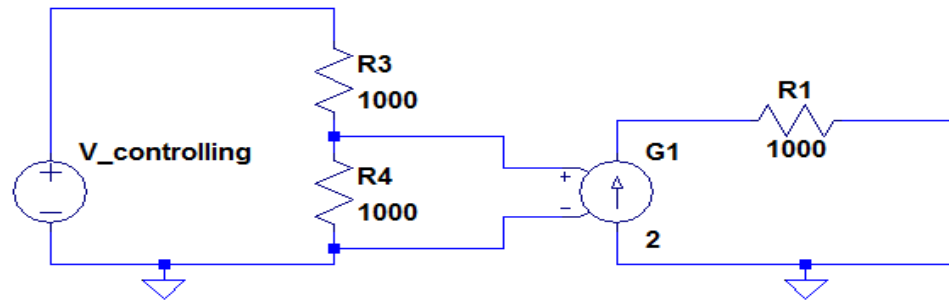
.dc V_controlling 0 5 0.01

Right click on the voltage source V_controlling1 and configure it similar to task 2.(b) step. Right click on the VCIS and configure it as follows.
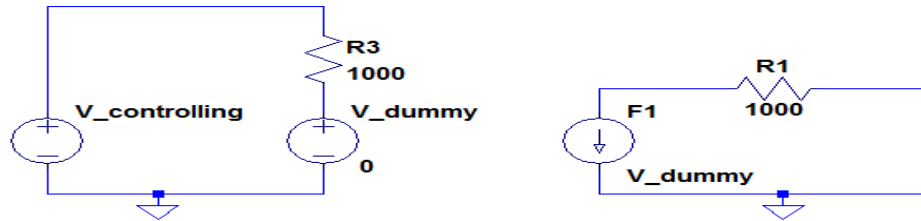

The number in the value field specifies the gain.

i) In the "edit simulation command" option (in the "simulate" tab), go to the DC Sweep tab and do the settings as required.

j) Hit the run tab and observe the output voltage waveform by putting the current marker in the middle of G1 and voltage across R4 by putting a voltage marker on R4.

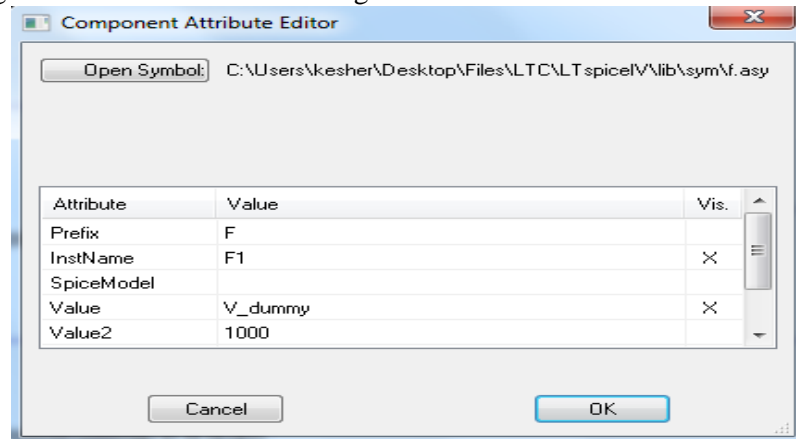k) Select the Current controlled current source (ICIS) component (component name "f") from the component tab.

l) Make the following design in LTSpice.

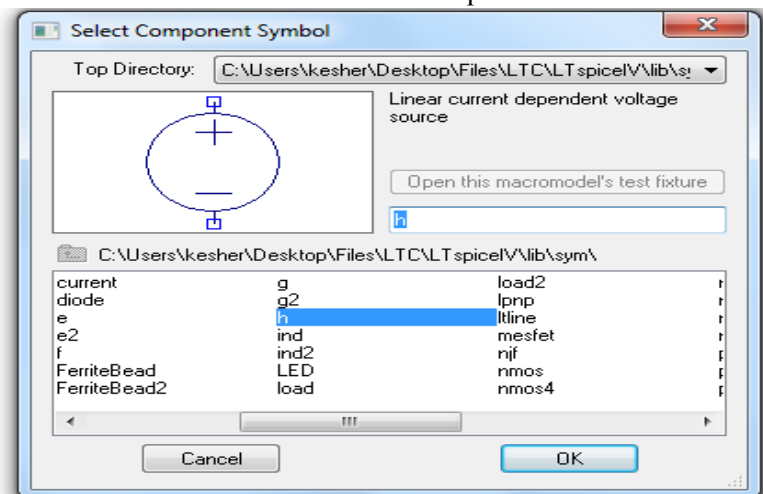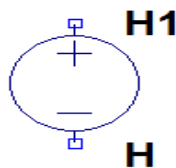.dc V_controlling 0 5 0.01

Right click on the voltage source V_controlling and configure it similar to task 2.(b) step. Right click on the ICIS and configure it as follows.
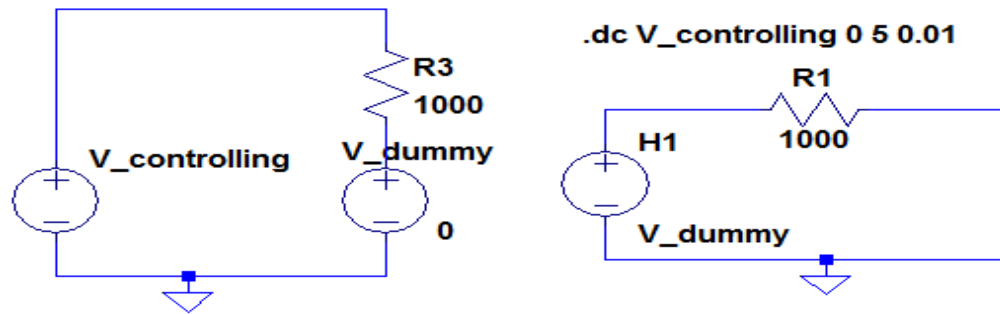

The number in the value2 field specifies the gain. The current through ICIS is proportional to the current through the voltage source named in the value field.
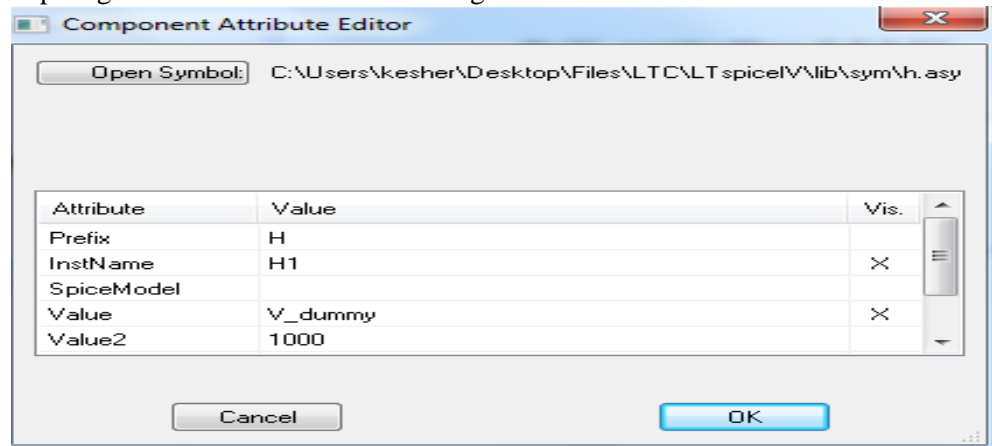
m) In the "edit simulation command" option (in the "simulate" tab), go to the DC Sweep tab and do the settings as required.

n) Hit the run tab and observe the output voltage waveform by putting the current marker in the middle of F1 and current through R3 by putting a current marker.

o) Select the Current controlled voltage source (ICVS) component (component name "h") from the component tab.

p) Make the following design in LTSpice.

Right click on the voltage source V_controlling and configure it similar to task 2.(b) step. Right click on the ICVS and configure it as follows.


The number in the value2 field specifies the gain. The voltage through ICVS is proportional to the current through the voltage source named in the value field.

q) In the "edit simulation command" option (in the "simulate" tab), go to the DC Sweep tab and do the settings as required.

r) Hit the run tab and observe the output voltage waveform by putting the voltage marker on H1 and current through R3 by putting a current marker.

3. To implement controlled switch in LTSpice.
   a) Select the Current controlled switch component (component name "csw") from the component tab.

   b) Make the following design in LTSpice.

R1
10
V_main
10
W1
MySwitch

R2
1000
V_dummy
0
V1
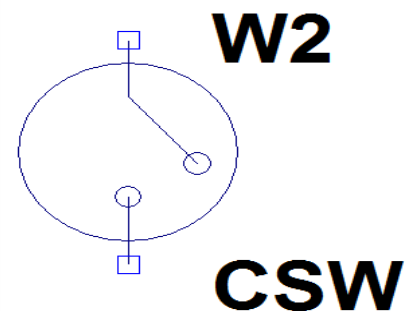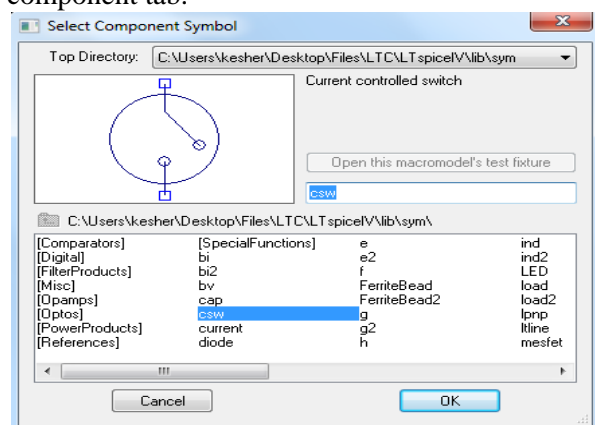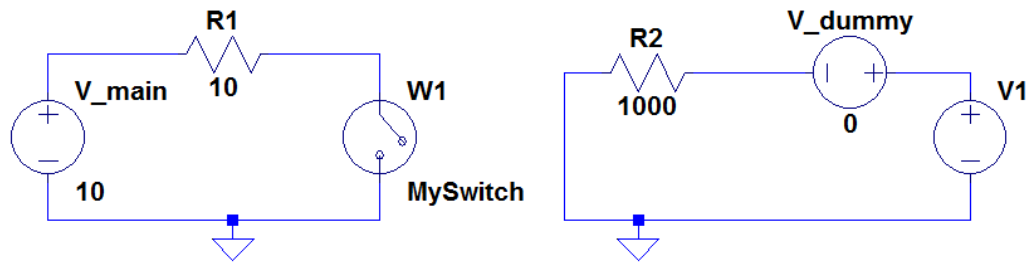
.tran 4

PWL(0 0 1 0 1.0000001 5 2 5 2.0000001 0)

.model MySwitch CSW(Ron=0.01 Roff=1e6 It=1e-3)

Take the voltage source from the component tab. Instead of giving the dc value, hit the advanced button. The voltage source V1 is configured as piece wise linear (PWL). For times before t1, the voltage is v1. For times between t1 and t2, the voltage varies linearly between v1 and v2. Configure the PWL voltage source as follows:
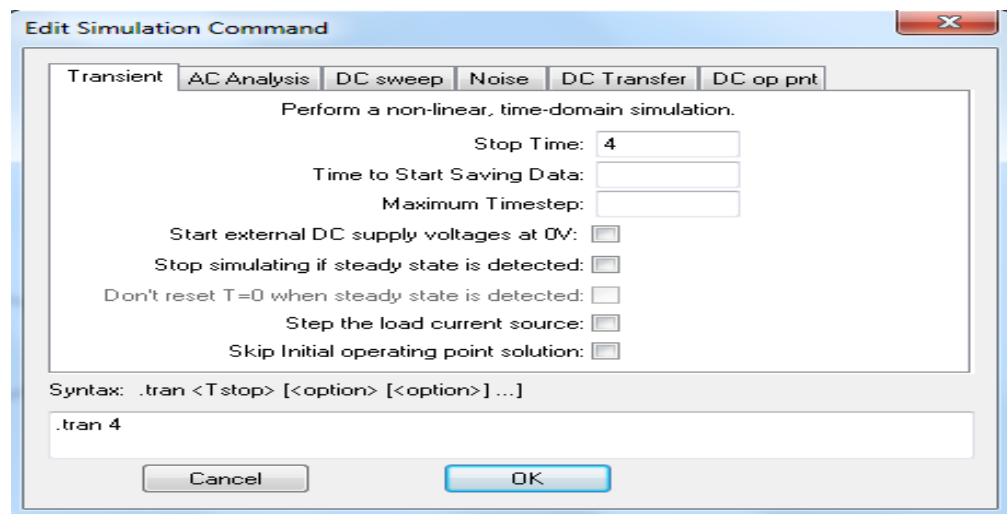

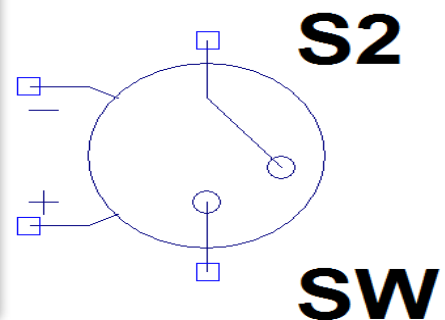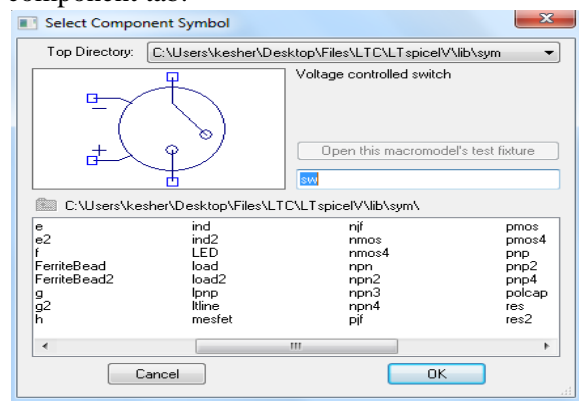
.

Right-click on the switch and configure it as follows:

The current through the V_dummy voltage source controls the switch's impedance. A model card is required to define the behavior of the current controlled switch. The model card (.model directive) has the following parameters:

.model MySwitch CSW(Ron=.1 Roff=1Meg It=0). Here Ron and Roff are the open and close resistance of the switch. It is the threshold current, if a current more than or equal to flows through the V_dummy voltage source then switch is considered closed otherwise open.
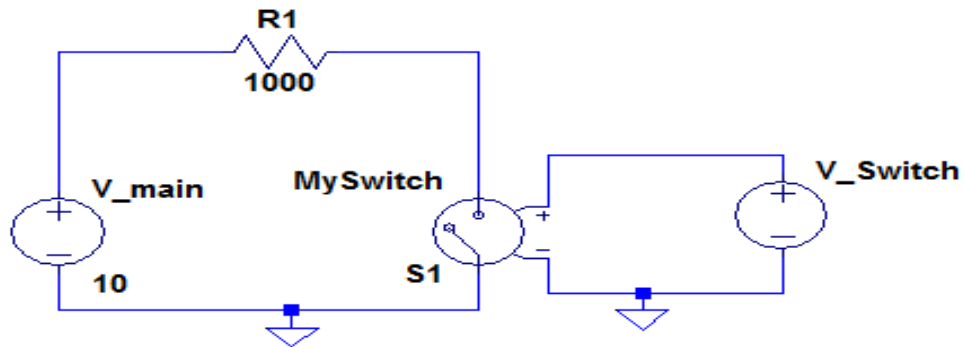
c) In the "edit simulation tab" do the following setting.

d) Observe the voltage at current control switch W1 and voltage source V1 using the voltage marker.
e) Select the voltage controlled switch component (component name "sw") from the component tab.

f) Make the following design in LTSpice.

R1
1000

V_main

MySwitch

V_Switch

S1

10

.tran 4
.model MySwitch SW(Ron=0.001 Roff=10e6 Vt=1)
PWL(0 0 1 0 1.000001 5 2 5 2.000001 0)

The voltage source V_Switch is configured as piece wise linear (PWL). Configure the PWL voltage source as was described in task3 part (b).

Right-click on the switch and configure it as follows:
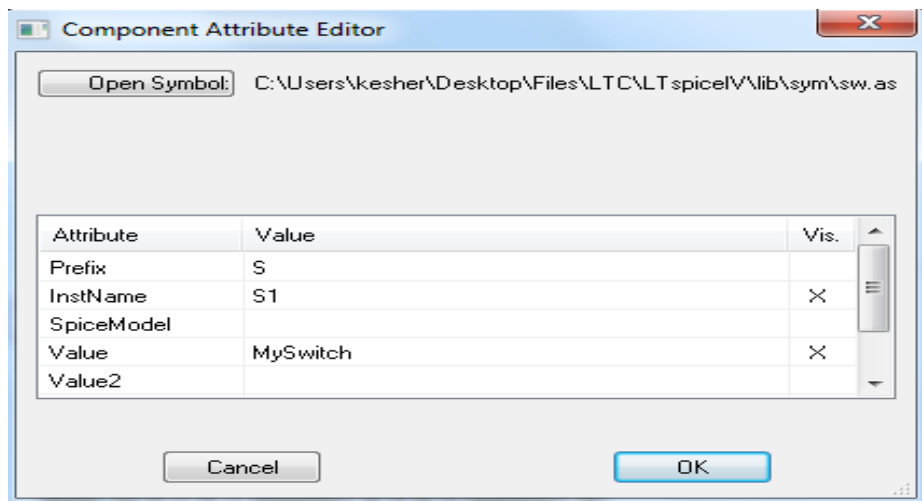
The voltage V_Switch controls the switch's impedance between nodes n1 and n2. A model card is required to define the behavior of the switch. The model card (.model directive) has the following parameters:
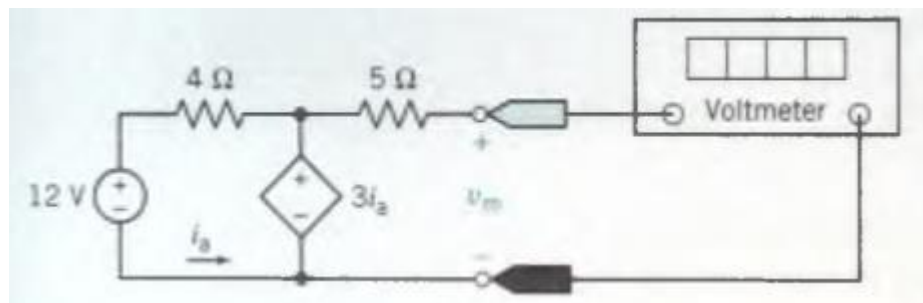
.model MySwitch CSW(Ron=.1 Roff=1Meg Vt=1). Here Ron and Roff are the open and close resistance of the switch. The switch is always completely on or off depending upon whether the input voltage (V_Switch) is above the threshold.

g)   In the "edit simulation tab" tran setting similar to task 3 part (c).
h)   Observe the voltage at voltage control switch S1 and voltage source V_Switch using the voltage marker.

4. Implement the following circuit in LTSpice and answer the respective questions.

a) What is the value of current "i" at t=4s?
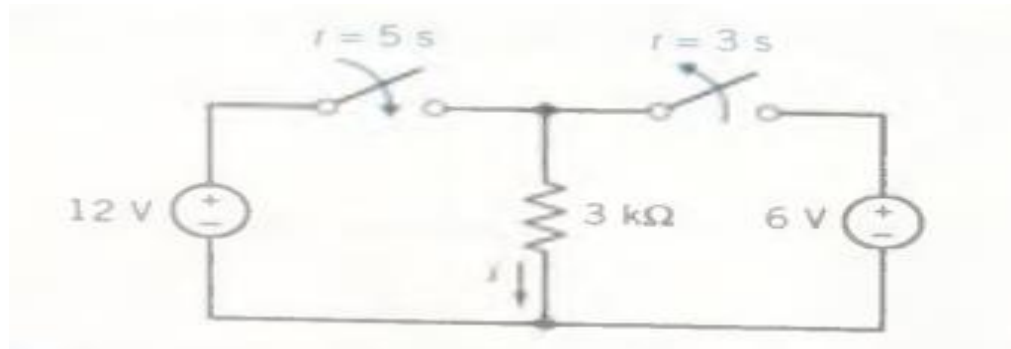
Hint: use current/voltage controlled switches and configure them so that one closes at 5s and other opens at 3s. Do the transient analysis.

b) Determine the value of voltage (in volts) measured by the voltmeter.

END