A Brief Tutorial on LTspice

Engineers use software tools for the design and analysis of circuits. The software tools help to simulate the behavior of electronic circuits in different operating conditions, and support the design of circuits before actually building them using hardware.

SPICE (Simulation Program for Integrated Circuits Emphasis) is widely used circuit design and analysis software in industry. SPICE was developed in the late 1970's by University of California at Berkeley to design and analyze ICs. In the original SPICE, a circuit (all its components and their inter connection) is described using a text file called **netlist**.

Currently there are several derivatives of the original SPICE developed by different companies. Some of the popular derivatives of SPICE are Pspice (from OrCAD/ Cadence), Hspice (from Avanti/Synopsys), LTspice (from Linear Technology) and others.

In the course B EE 215, we will introduce the basic functions of LTspice by simulating circuits that we build in the experiments. LTspice is a powerful and fast software and has a schematic capture, which is a GUI (graphical user interface) that is used to enter the circuit for simulation using graphical symbols of components and wires.

Installing LTSpice:

LTspice is freely available software and can be downloaded from the website of Linear Technology. To download LTspice

- visit http://www.linear.com/designtools/software/ltspice.jsp
- Select the **Register** on **No Thanks** option
- Run the ***LTspiceIV.exe*** file on your computer
- Install LTSpice

Simulation of circuits using LTspice:

Simulation of circuits using LTspice has two steps:

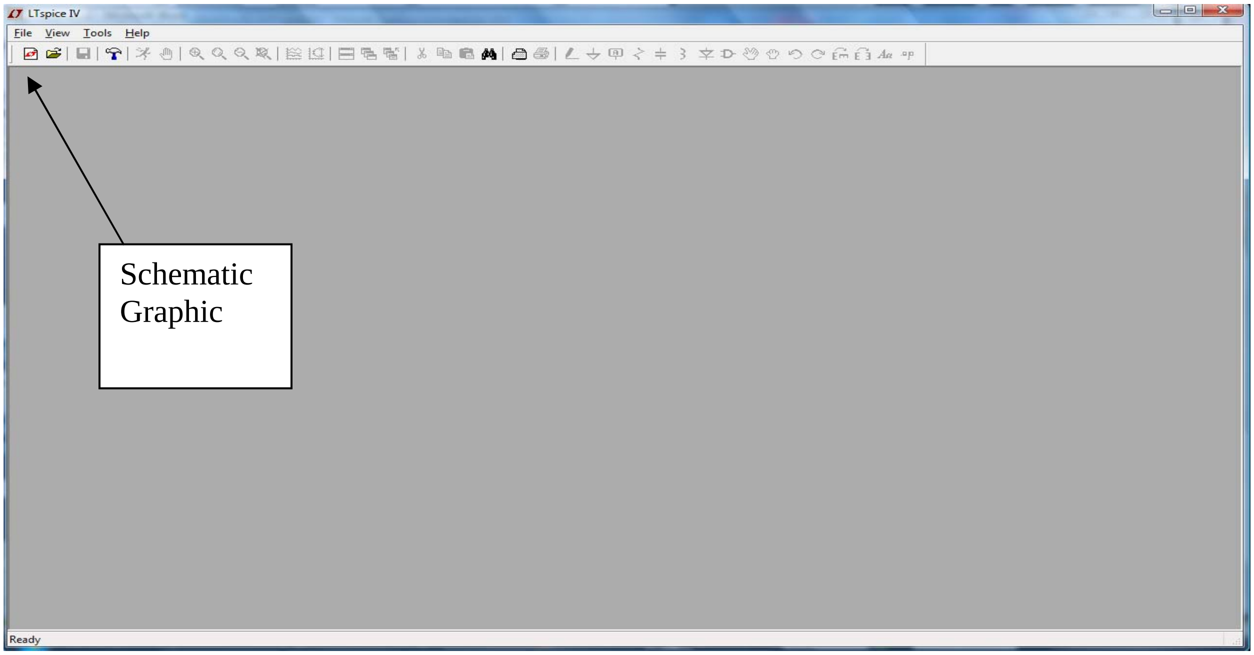
1. Drawing (editing) or entering the circuit using the schematic capture
2. Defining the desired type of simulation and running it.

I. Entering Circuit Using the Schematic Capture

a.) Starting Schematic Capture

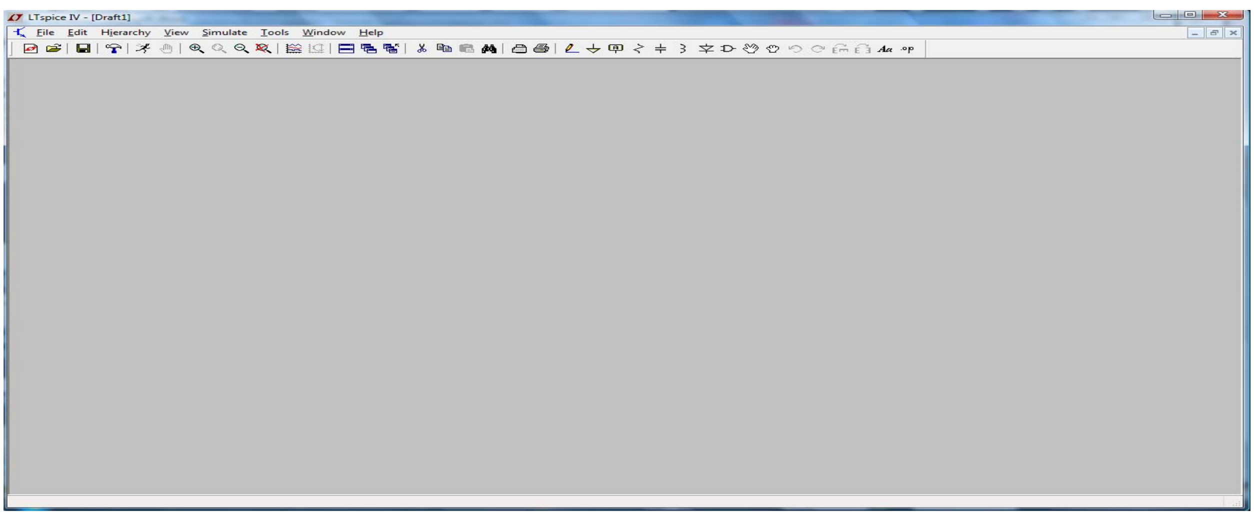
- First run **LTspice IV** from the **start** menu of windows.

The following window appears



- Press the **New Schematic** button to launch it or
- Select **File** followed by **New Schematics** from the main menu.

The following window appears



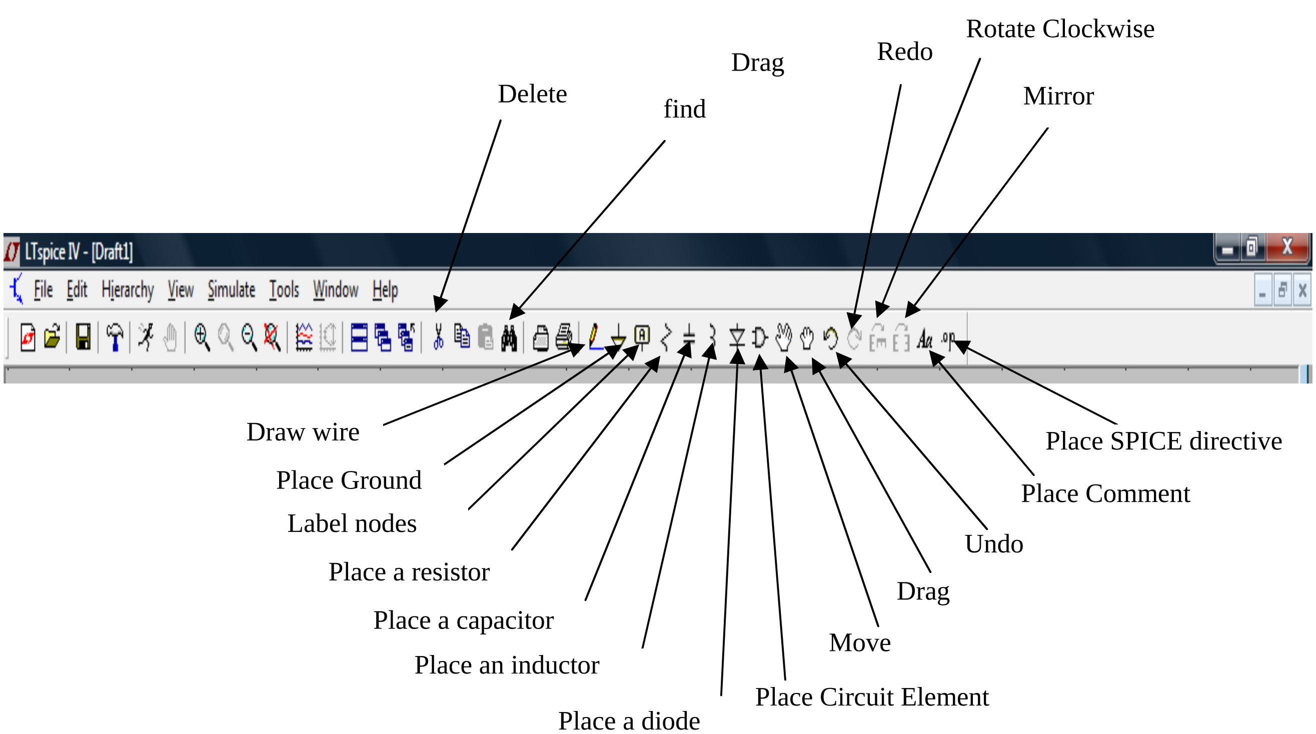
Drawing a circuit is easier when the grid is active. To activate the grid, click on **View** and select **Show Grid**.

b) Placing circuit elements:

There are buttons on the main menu for basic circuit elements including resistor, capacitor, inductor, diode, wire as well as ground (GND). To place one of these circuit elements**:**

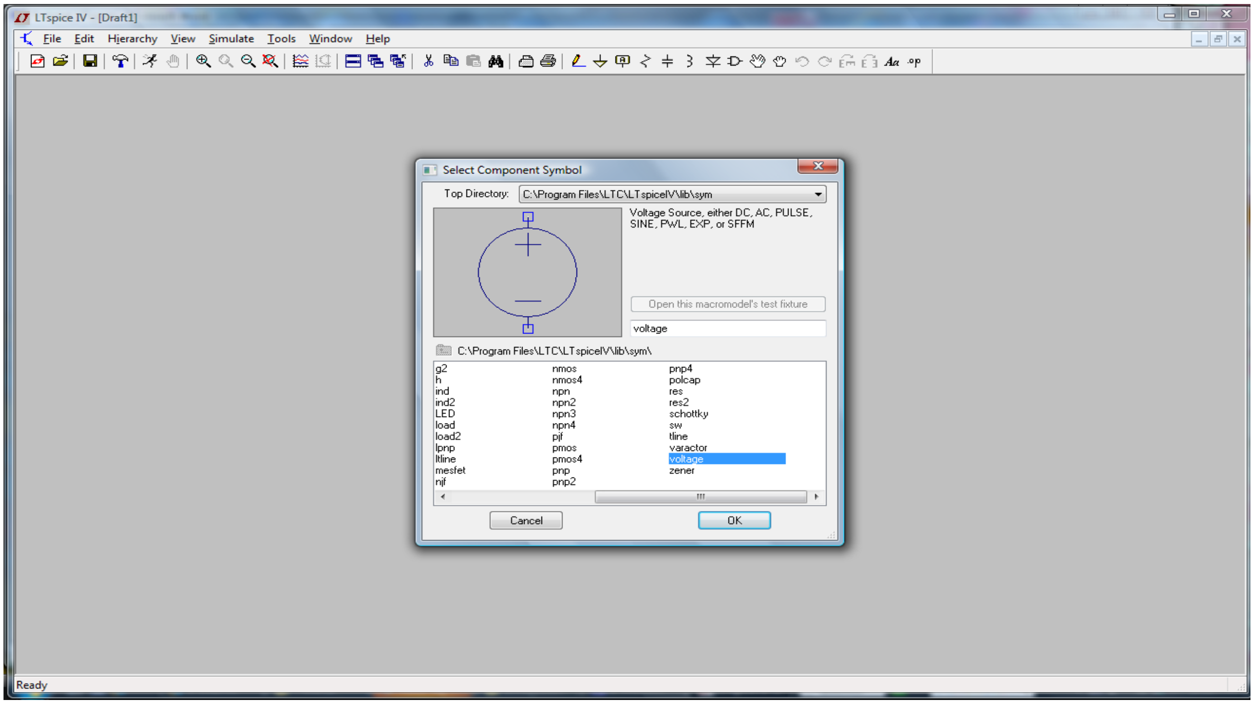
- Click on the corresponding button
- Move the mouse to a location where the element is to be located
- Click the left key of the mouse to place the element
- You can move the mouse to a different location to place another element of the same type in another location or press the right key of the mouse to deselect the circuit element type
- continue with placing another type of circuit element.

Note: another type of selecting elements is using the keyboard: ‘r’ for resistor, ‘c’ for capacitor, ‘l’ for inductor and‘d’ for diode.



To place circuit elements beyond those shown on the main menu, we click the button **Place Circuit Element** and we select the desired element from the list of elements that appears in a window that looks like the figure shown below.

For example, if you want to place a voltage source
- Click the **Place Circuit Element** button
- Select **Voltage** from the element list and double click it.
- Move the mouse to the position where the voltage source is desired to be placed
- Press the left key of the mouse to place the voltage source and select the right mouse to deactivate the voltage source selection.



c) Wiring the elements

To wire the circuit elements, select the **Draw Wire** (pencil tool) button. You can draw horizontal and vertical lines using the pencil tool.

- Mark the beginning of a wire by pressing the left key of the mouse.
- Move the mouse to stretch the line horizontally or vertically.
- If change of direction of the wiring is desired, press the left key of the mouse and move the mouse in the desired direction.
- Press the right key of the mouse to stop wiring
- Press the ESC key to deselect the pencil tool.

d) Editing the elements

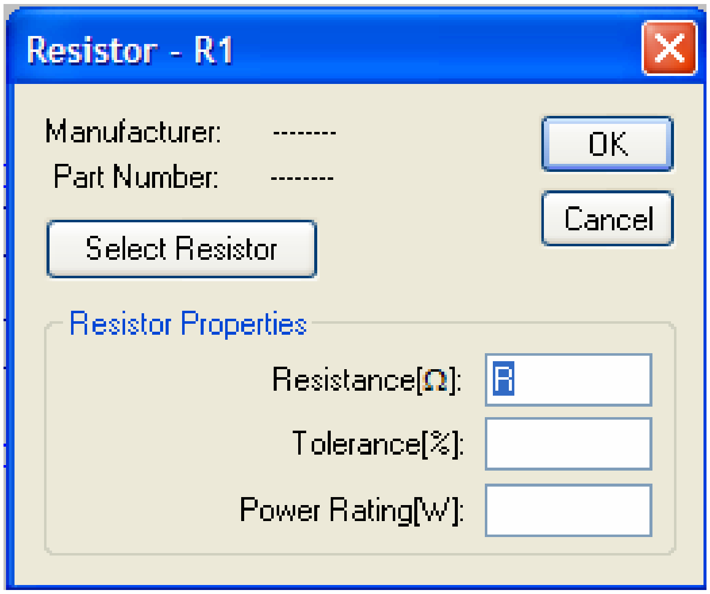
<u>Delete an element</u>: Press the **Delete** button and click the scissors on the element. Press the right mouse key or ESC to quit deleting.

<u>Move an element</u>: Press the **Move** button (or F7) and click the left mouse key over the element to be moved. Move the mouse to the desired position and click the left mouse key to place the element.

<u>Rotate an element</u>: Press the **Rotate** button to rotate an element 90^0 in clockwise direction. The element can be rotated before it is placed in a position. If an element is already positioned, Move the element to get the Rotate button activated.

<u>Add Ground</u>: Press the Ground button to add a button to a circuit. ***Note that any circuit should have a Ground (a reference of zero voltage) for the simulation of a circuit to work properly***.

Modifying an element: The value of a component can be changed or modified on a window that emerges after clicking the right mouse key over the component. For example, after right clicking the mouse over a resistor, the following window is displayed.



To change the value of the resistor enter the value in ohms in the entry for resistance or press the **Select Resistor** button and choose the value from the list that appears.

II. Defining and Running Simulations

LTspice can be used to run several types of simulations. These types of simulation include:

- DC operating point
- Transient Analysis
- AC Analysis
- DC Sweep
- Noise
- DC transfer

In this course we will be limited to the DC operating point and transient analysis. In this first tutorial we will explain the DC operating point. The other types of simulation will be explained in subsequent tutorials.

a) **DC operating point**: This type of simulation gives the value of voltages and currents through nodes and components for a fixed input operating condition. The simulation results are given in text format.

To run a dc operating point simulation**:**

- Select **Simulate** button from the main menu.
- From the pull-up menu of **Simulate**, select **Edit Simulation Cmd**
- Select **DC op pnt** button
- Select OK