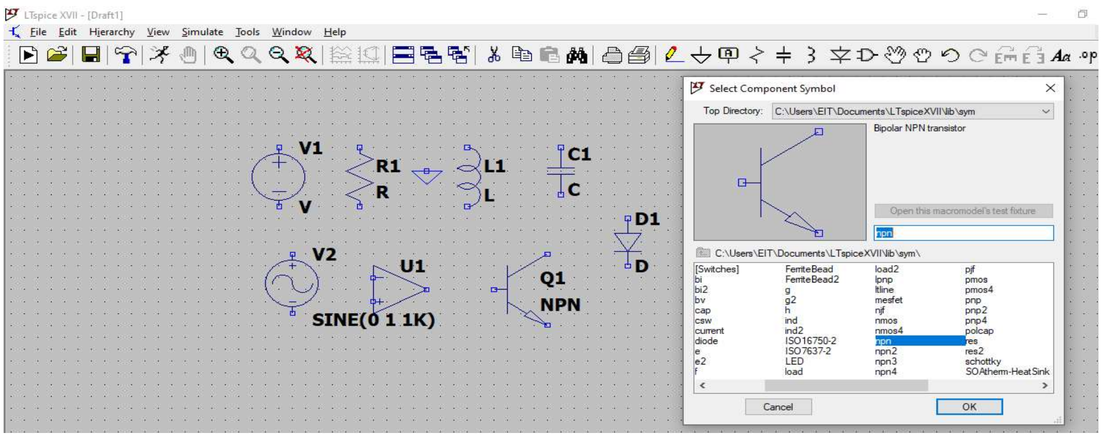
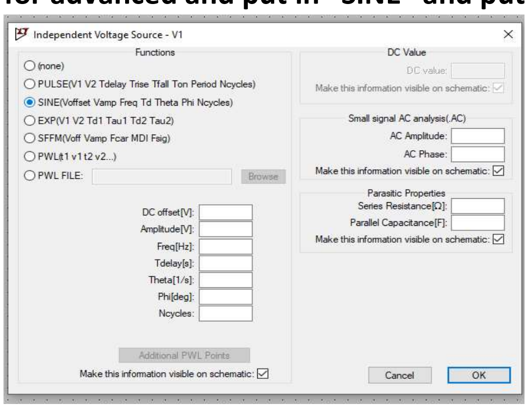
Simulation through LT-Spice

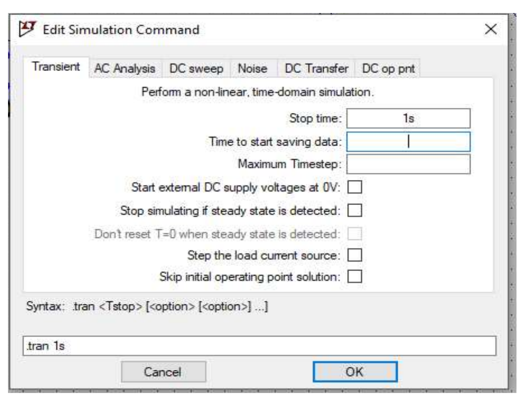
First download and install ; https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html#

1. **Open the program and press the File>New Schematic> blank new screen pop out.**
2. **Drag the Resistor, Inductor, Capacitor, Ground and right click to put the values**
3. **Open the Components icon to create DC voltage, Transistor, Ac Signal, Op-amp etc.**
4. **Scissor Icon is used to cut the components, Other icons are for dragging, coping components.**
5. **Similarly, pen icon is to wire the different components**



6. **Right click in DC-Voltage supply and give any required voltages and for AC signal you have to either call AC signal from Components Icon or right click to DC-Voltage and go for advanced and put in "SINE" and put DC=0, Amp and frequency as you desired.**





7. **To simulate Click the "Running Man" icon before this go to simulation and edit "simulation cmd" as shown above the figure. You can also go to the AC analysis and draw the bode plot of magnitude and phase diagram through "octave and decade function". Rest you can see prerecorded simulation and "YouTube LT-spice clips".**