## **Simulation through LT-Spice**

First download and install; <u>https://www.analog.com/en/design-center/design-tools-and-</u> 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.

9 Independent Voltage Source - V1	×	Figure 2 Edit Simulation Command	×
Functions (none) PULSE(V1 V2 Tdelay Trise Tfall Ton Period Novcles)	DC Value DC value:	Transient AC Analysis DC sweep Noise DC Transfer DC op pnt	
SINE(Voffset Vamp Freq Td Theta Phi Ncycles)	Make this mornalion visible on schematic.	Perform a non-linear, time-domain simulation.	
O EXP(V1 V2 Td1 Tau1 Td2 Tau2)	Small signal AC analysis(.AC)	Stop time: 1s	- T
SFFM(Voff Vamp Fcar MDI Fsig)     PWL#1v1t2v2	AC Phase:	Time to start saving data:	1
O PWL FILE: Browse	Make this information visible on schematic:	Maximum Timestep:	
	Parastic Properties	Start external DC supply voltages at 0V:	_
DC offset[V]:	Parallel Capacitance[F]:	Stop simulating if steady state is detected:	
Freq[Hz]:	Make this information visible on schematic:	Don't reset T=0 when steady state is detected:	
Tdelay[s]:		Step the load current source:	
Phi[deg]:		Skip initial operating point solution:	
Ncycles:		Syntax: tran <tstop> [<option> [<option>]]</option></option></tstop>	
Additional PWL Points		Itran 1s	
Make this information visible on schematic:	Cancel OK	Cancel OK	

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".