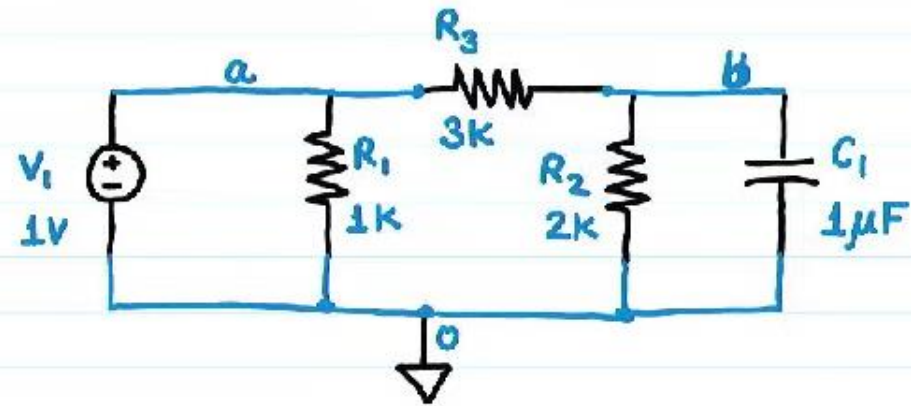


Circuit Simulations using SPICE :

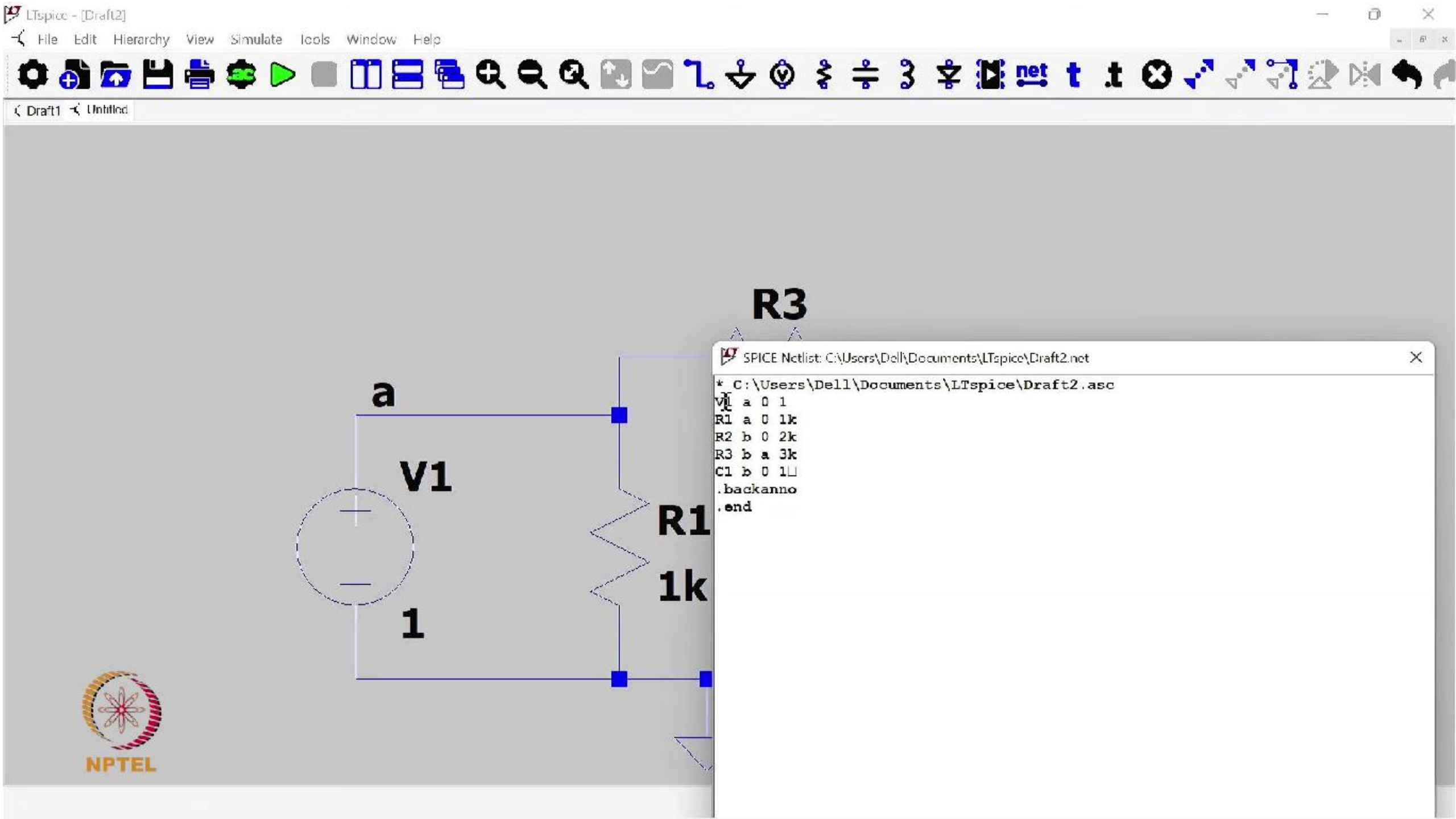
- Simulation Program with Integrated Circuit Emphasis .
- A freeware SPICE circuit simulator is " LTSPICE " developed at Linear Technologies .
- Written to support circuit designers .
- It includes an extensive library of Linear Technology devices .

Schematic Capture vs Netlist :



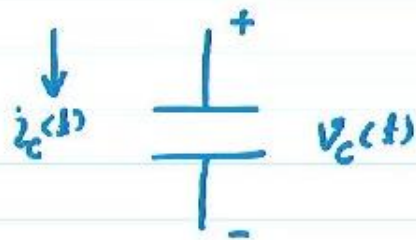
V1	a	o	1
R1	a	o	1k
R2	b	o	2k
R3	b	a	3k
C1	b	o	1μ

Schematic

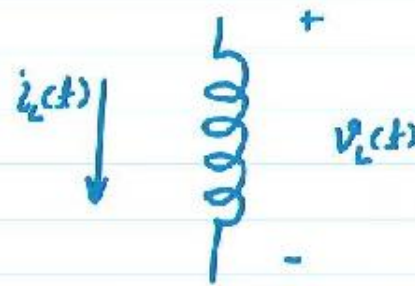


DC operating point of SPICE Simulation :

- Usually a dc solution of the circuit, given each component has its own current and voltage characteristics.

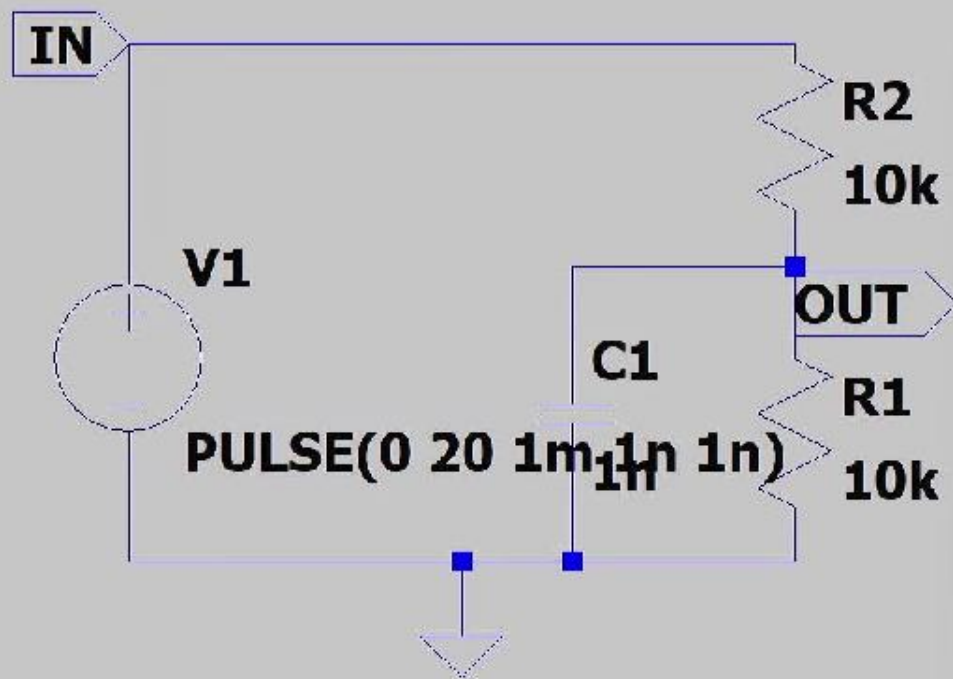


impedance $Z_c = \frac{1}{j\omega C} = \frac{1}{sC}$



impedance $Z_L = j\omega L$

- Output data as a list of node voltages and branch currents at steady state.



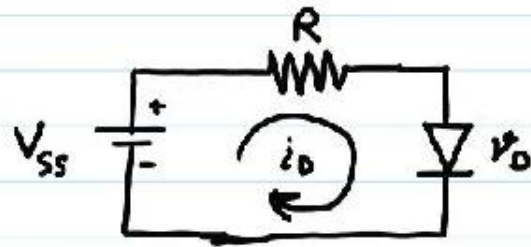
.ic V(OUT) = 3V
.op

C:\Users\Del\Documents\LTspice\Draft3.asc

--- Operating Point ---

V(out):	3	voltage
V(in):	0	voltage
I(C1):	3e-21	device_current
I(R1):	0.0003	device_current
I(R2):	-0.0003	device_current
I(V1):	0.0003	device_current

Load line and Q-point :



$$V_{SS} = Ri_D + v_D$$

