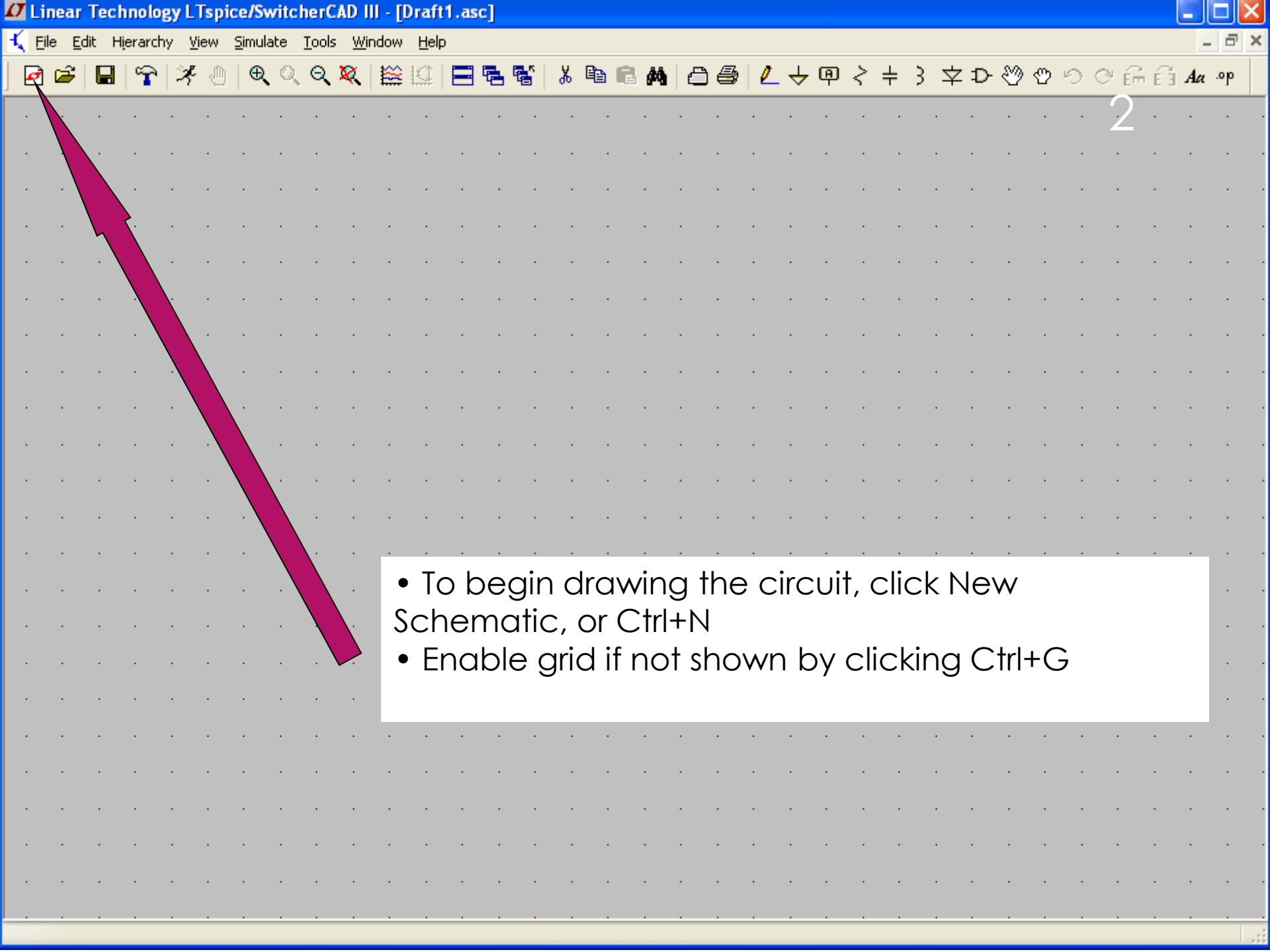
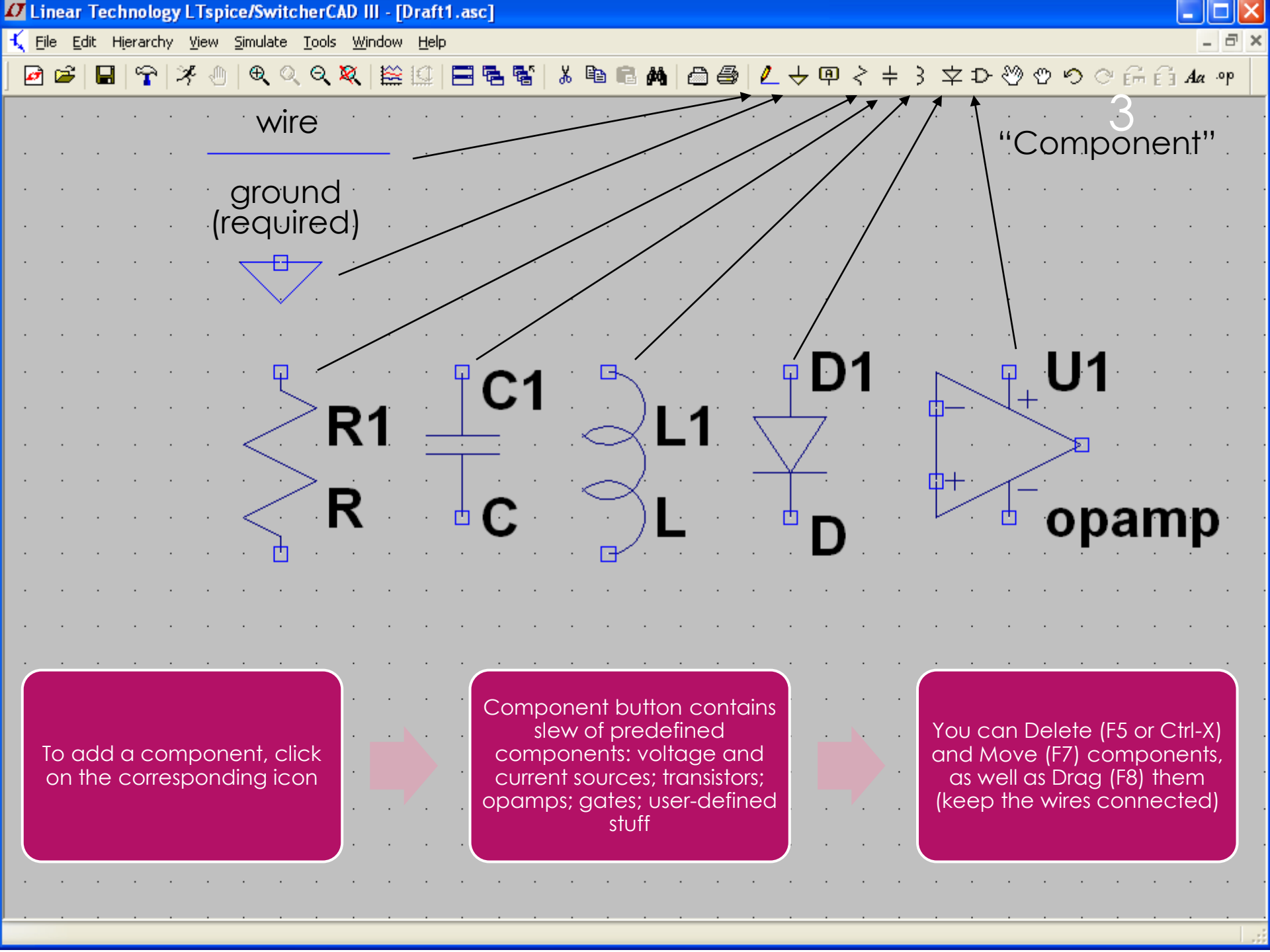
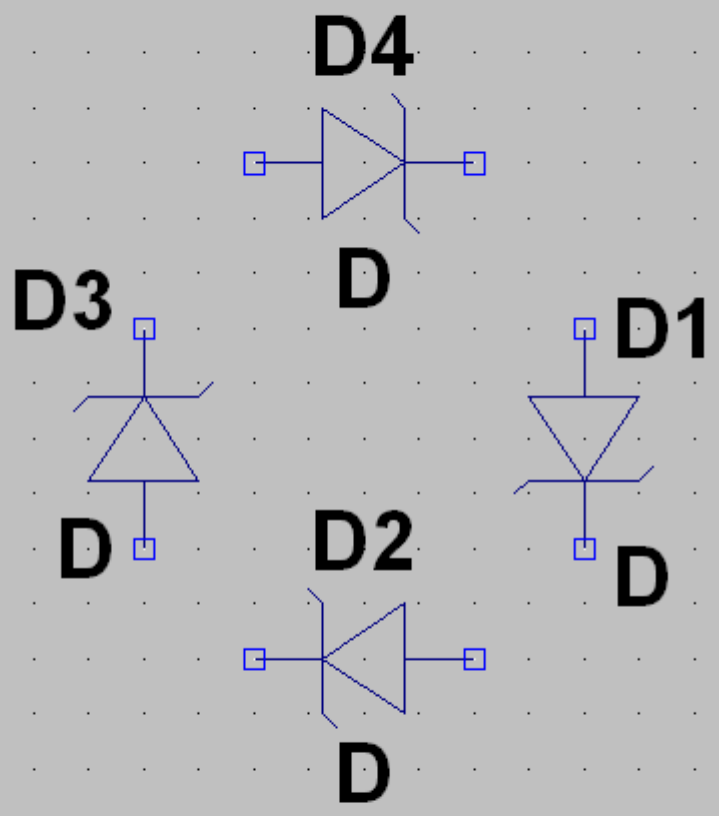
Introduction To LTSpice


Linear Technology LTspice/SwitcherCAD III - [Draft1.asc]
File Edit Hierarchy View Simulate Tools Window Help
2
• To begin drawing the circuit, click New Schematic, or Ctrl+N
• Enable grid if not shown by clicking Ctrl+G


Linear Technology LTspice/SwitcherCAD III - [Draft1.asc]
File Edit Hierarchy View Simulate Tools Window Help
wire
ground
(required)
3
"Component"
R1
R
C1
C
L1
L
D1
D
U1
opamp
To add a component, click on the corresponding icon
Component button contains slew of predefined components: voltage and current sources; transistors; opamps; gates; user-defined stuff
You can Delete (F5 or Ctrl-X) and Move (F7) components, as well as Drag (F8) them (keep the wires connected)



To rotate the component prior to placing press Ctrl-R

To mirror the component prior to placing press Ctrl-E

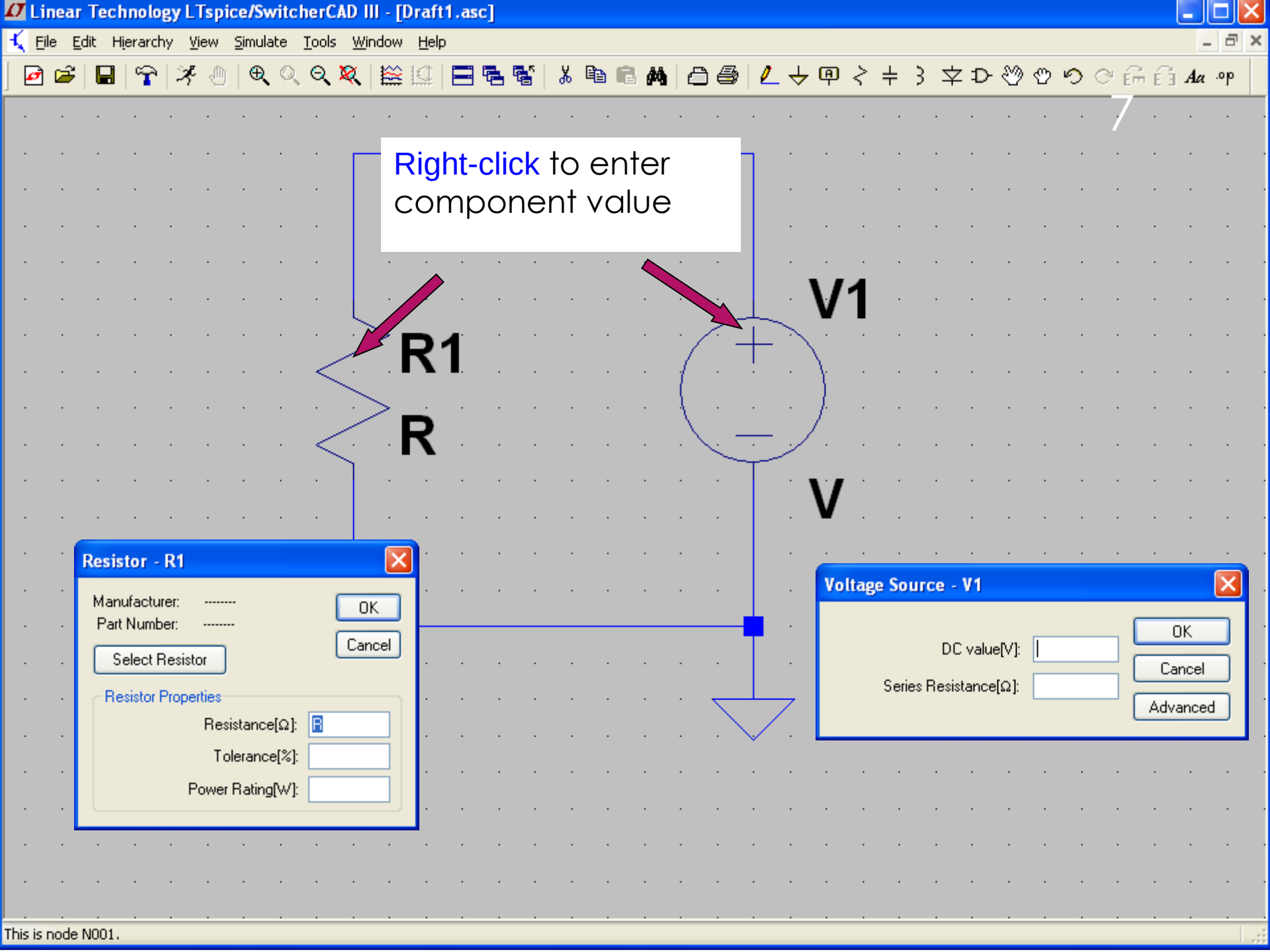
Toolbar Summary

Place Diode
Place Inductor
Place Capacitor
Place Resistor
Label Node
Place Ground
Draw Wire
Place Circuit Element
Simulate
Zoom In
Pan
Zoom Out
Auto Scale
Delete
Copy
Paste
Find
Move
Drag
Undo
Redo
Rotate
Mirror
Place Comment
Place SPICE Directive

Prefixes are *case insensitive*: T = t, G = g, and so on

- T = terra = 10^{12}
- G = giga = 10^{9}
- MEG = meg = 10^{6}
- K = kilo = 10^{3}
- M = milli = 10^{-3}
- U = micro = 10^{-6}
- N = nano = 10^{-9}
- P = pico = 10^{-12}
- F = femto = 10^{-15}

No need to enter units, they are assumed (e.g. "1M" is 1mV if entered for voltage, 1ms if entered for time, etc.)

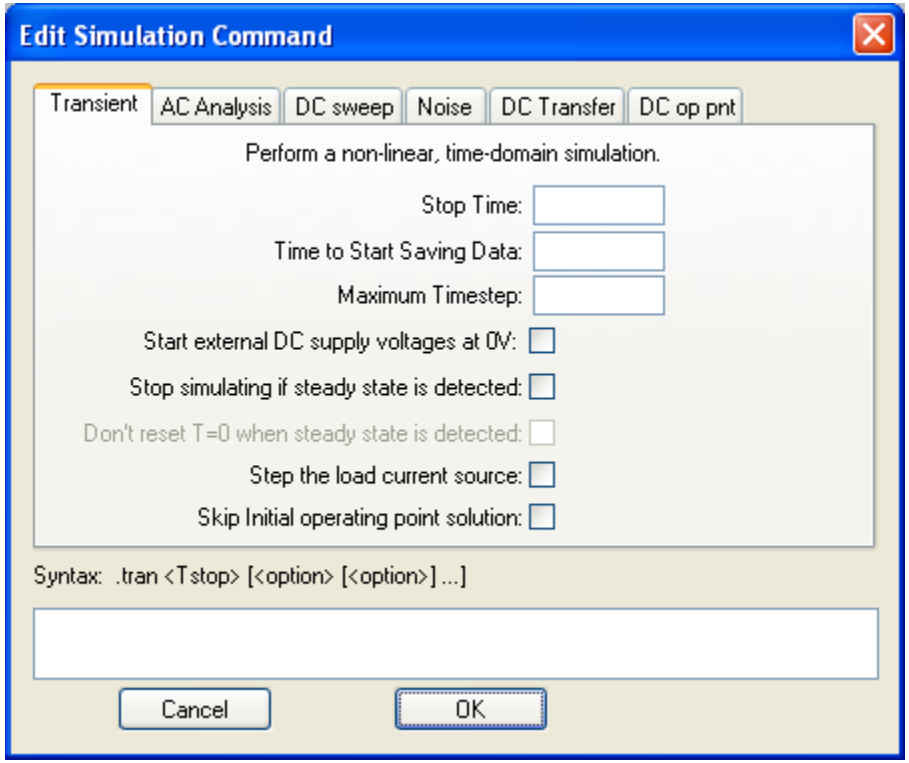

Linear Technology LTspice/SwitcherCAD III - [Draft1.asc]
File Edit Hierarchy View Simulate Tools Window Help
Right-click to enter component value
R1
R
V1
V
Resistor - R1
Manufacturer:
Part Number:
Select Resistor
OK
Cancel
Resistor Properties
Resistance[Ω]:
Tolerance[%]:
Power Rating[W]:
Voltage Source - V1
DC value[V]:
Series Resistance[Ω]:
OK
Cancel
Advanced
This is node N001.

Simulation Types

Your choices:

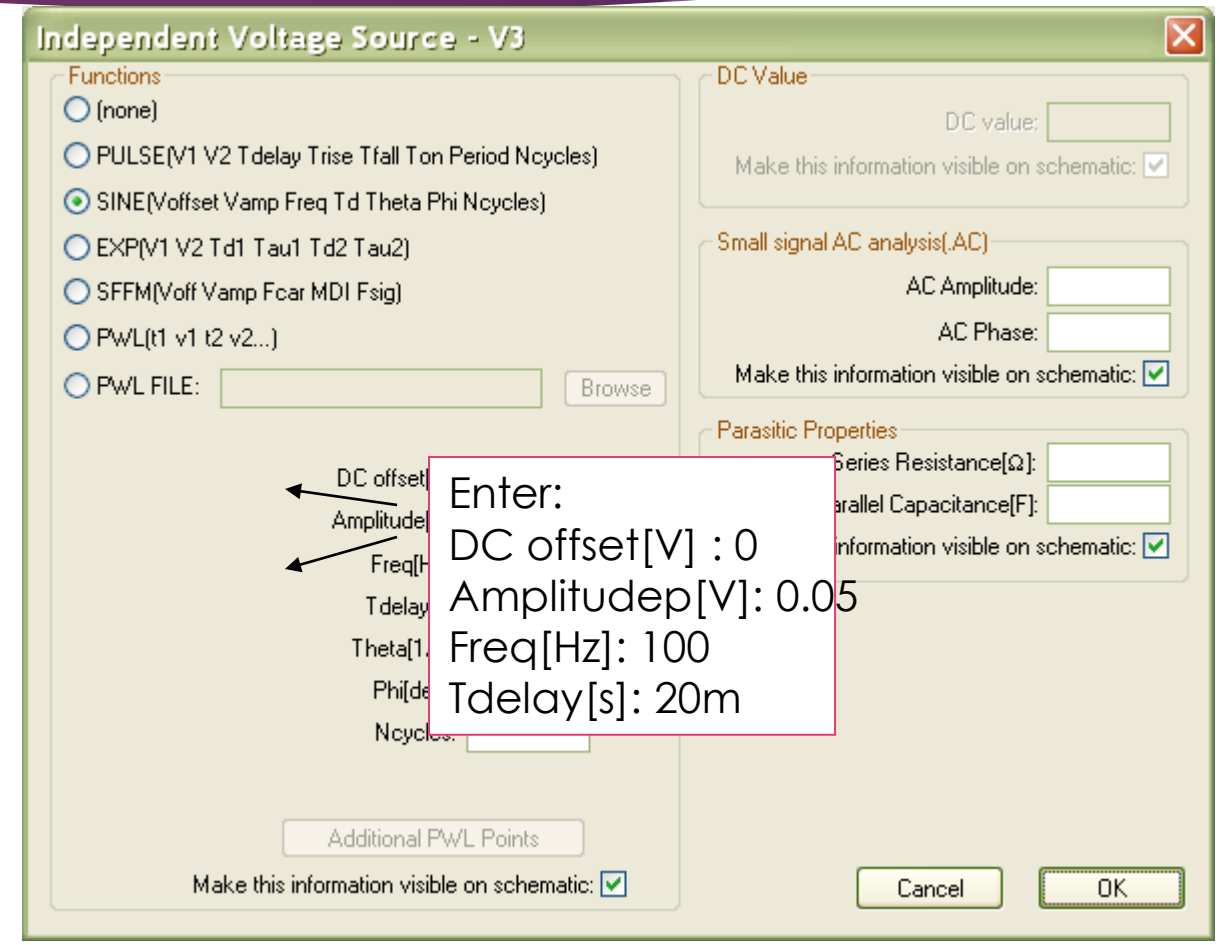
- Transient analysis
- AC small signal anlysis
- DC sweep
- Noise analysis
- DC transfer function
- DC operating point (Q-pt)

Highlighted is what you will be using in this course



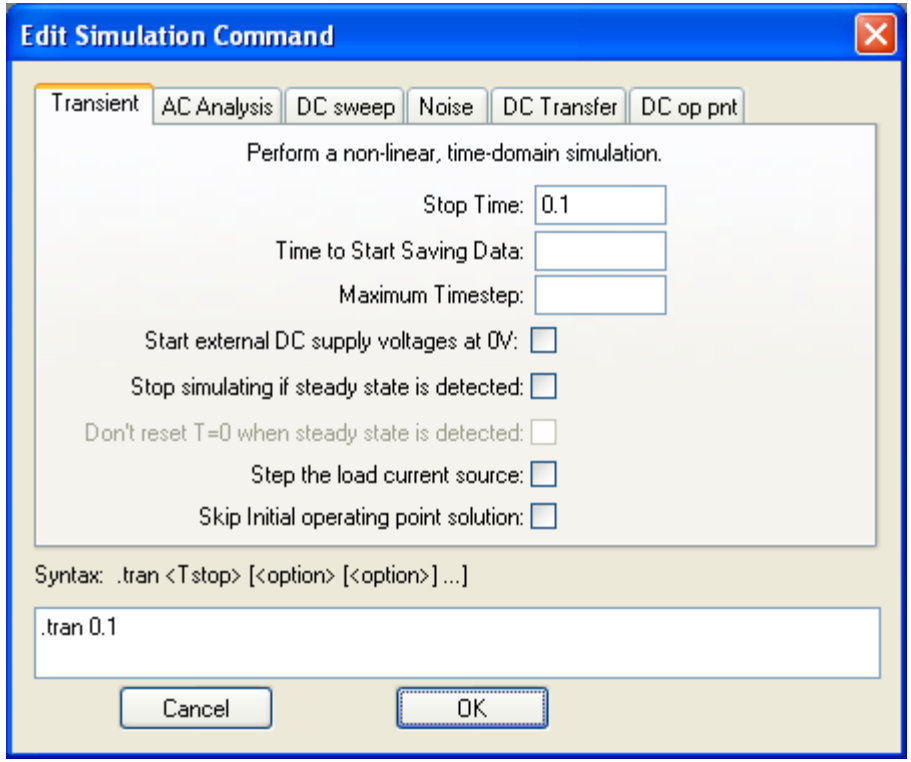
Assigning Voltage from Voltage Source

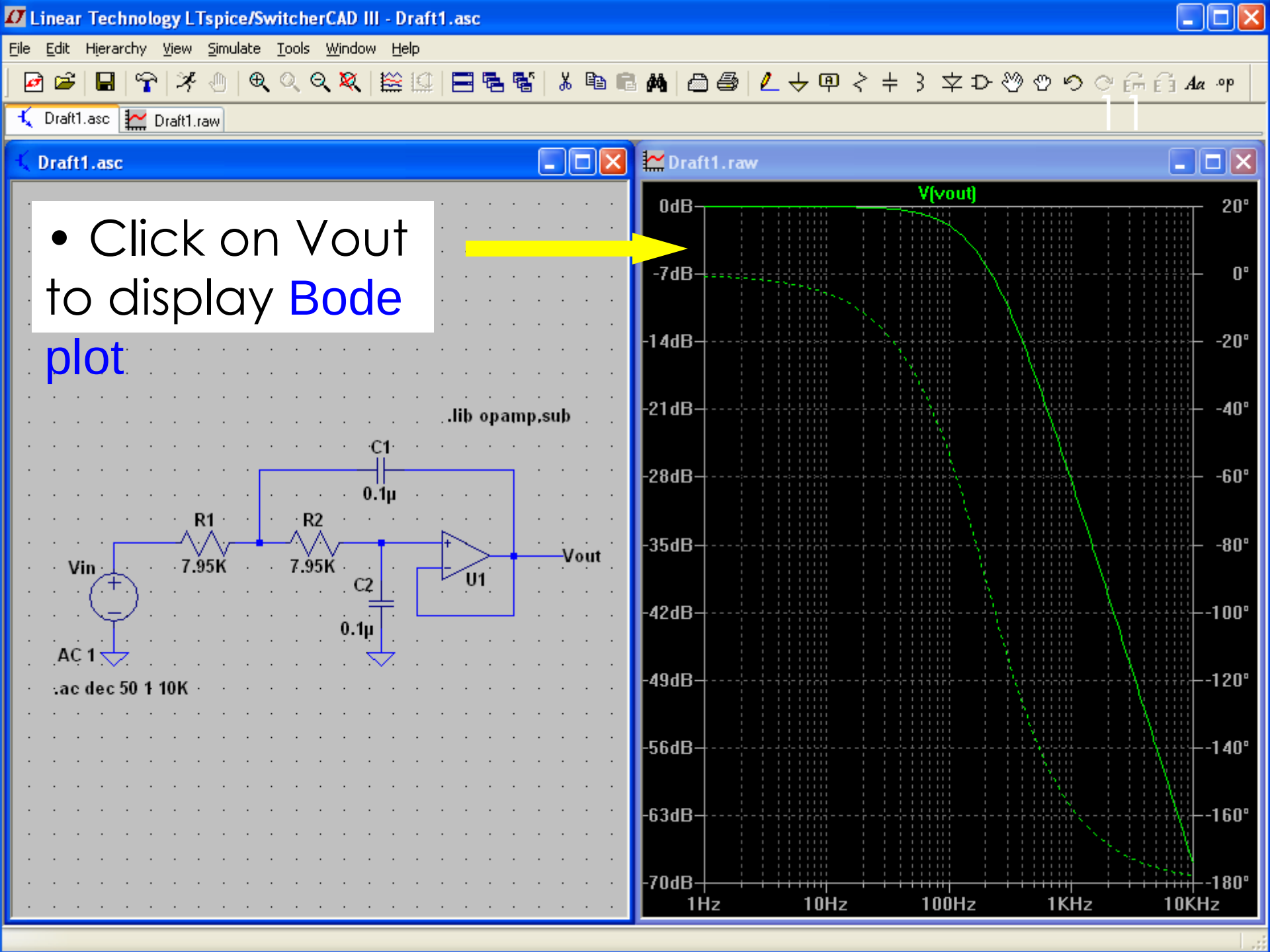
- Right-click on Vin, then click Advanced. You will see a window like this. Choose SINE Function.



Time Transient Analysis

- Choose Simulate → Edit Simulation Cmd
- Indicate Stop Time of 0.1 s
- Click OK and place SPICE directive somewhere on your circuit
- Ready to go!




Linear Technology LTspice/SwitcherCAD III - Draft1.asc
File Edit Hierarchy View Simulate Tools Window Help
Draft1.asc
Draft1.raw
Draft1.asc
• Click on Vout to display Bode plot
.lib opamp.sub
C1
0.1µ
R1
7.95K
R2
7.95K
Vin
AC 1
C2
0.1µ
U1
Vout
.ac dec 50 1 10K
Draft1.raw
V(vout)
0dB
-7dB
-14dB
-21dB
-28dB
-35dB
-42dB
-49dB
-56dB
-63dB
-70dB
20°
0°
-20°
-40°
-60°
-80°
-100°
-120°
-140°
-160°
-180°
1Hz
10Hz
100Hz
1KHz
10KHz

“

THANK YOU

”