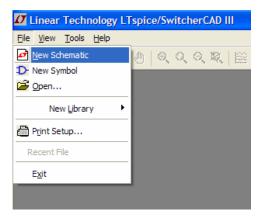
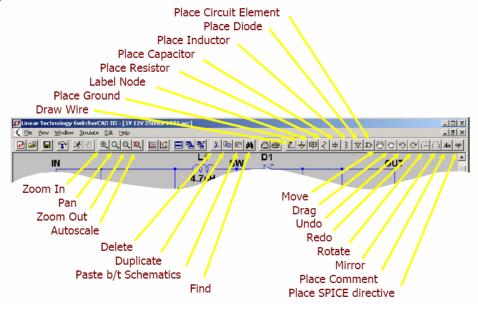
# **LTSpice Guide**

Click on the "SwCAD III" shortcut created by the software installation.

Select "File" and "New Schematic".



### Add a component



- Add a resistor Press "R" or click the resistor button to insert a resistor. If you want to rotate the resistor before placing, press "ctrl+R" or click the rotate button. Left click where you want to place the resistor. Press "esc" to quit adding resistors.
- Add an inductor Press "L" or click the inductor button to insert an inductor. If you want to rotate the inductor before placing, press "ctrl+R" or click the rotate button. Left click where you want to place the inductor. Press "esc" to quit adding inductors.

- Add a capacitor Press "C" or click the capacitor button to insert a capacitor. If you want to rotate the capacitor before placing, press "ctrl+R" or click the rotate button. Left click where you want to place the capacitor. Press "esc" to quit adding capacitors.
- Add other components Press "F2" or click the component button to insert a component. A "Select Component Symbol" window will open. Take a minute to browse through the list of available components. Note that sources are labeled as "current" and "voltage".

Select Comp	onent Symbol		
Top Directory:	C:\Program Files\LTC\	.SwCADIII\lib\sym	*
		Open this macromodel's tes	t fixture
💼 C:\Program F	iles/LTC/SwCADIII/lib/	sym\	
[Comparators] [Digital] [FilterProducts] [Misc] [Opamps] [Optos] [PowerProducts] [References] [SpecialFunctions bi bi2	bv cap CNSW csw current diode e e2 ] f FerriteBead FerriteBead2	FerriteBead_Z(I) g g2 h ind ind2 LED load load2 Ipnp Itline	mesfet nif nmos npnos4 npn2 npn3 npn4 pif pmos pmos4
<	IIII	]	>
	ancel	ОК	.::

Select the component that you want to add and click "OK". If you want to rotate the component before placing, press "ctrl+R" or click the rotate button. Left click where you want to place the component. Press "esc" to quit adding this component.

#### **Modify Component Values**

To modify the value of an added component, right click on the component. Enter the value of the component in the default units (ohms, farads, henrys, etc.).

Resistor - R1	×
Manufacturer: Part Number:	OK
Select Resistor	Cancel
Resistor Properties	
Resistance[Ω]:	B
Tolerance[%]:	
Power Rating[W]:	

The value can also be edited by right clicking on the value of the component shown on the schematic instead of the component itself.

Enter new Value for R1	
Justification	OK
Left Vertical Text	Cancel

For a DC voltage source, enter the voltage and series resistance. For all other voltage sources, click "Advanced". Select the type of voltage source and enter the required parameters.

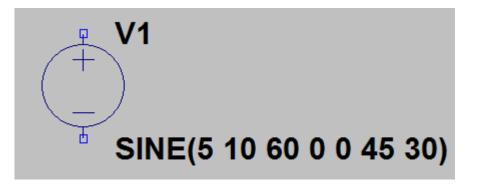
Voltage Source - V1	X
DC value[V]: Series Resistance[Ω]:	OK Cancel Advanced

Independent Voltage Source - V1	×
<ul> <li>Functions</li> <li>(none)</li> <li>PULSE(V1 V2 T delay Trise T fall T on Period Ncycles)</li> <li>SINE(Voffset Vamp Freq T d Theta Phi Ncycles)</li> </ul>	DC Value DC value: Make this information visible on schematic: 🗹
<ul> <li>EXP(V1 V2 Td1 Tau1 Td2 Tau2)</li> <li>SFFM(Voff Vamp Fcar MDI Fsig)</li> <li>PWL(t1 v1 t2 v2)</li> </ul>	Small signal AC analysis(.AC) AC Amplitude: AC Phase: Make this information visible on schematic: 🗸
Param1: Param2:	Parasitic Properties Series Resistance[Ω]:
Param3: Param4: Param5:	Parallel Capacitance[F]: Make this information visible on schematic: 🗹
Param6: Param7:	
Param8:	OK Cancel

For example, simulate  $v(t) = 5 + 10 \sin(2\pi 60 + 45^\circ)V$  for 500 ms (30 cycles) as a transient analysis.

Functions       DC Value         ○ (none)       PULSE(V1 V2 Tdelay Trise Tfall Ton Period Ncycles)         ③ SINE(Voffset Vamp Freq Td Theta Phi Ncycles)       Make this information visible on schematic: ✓         ③ SINE(Voffset Vamp Freq Td Theta Phi Ncycles)       Small signal AC analysis(AC)         ③ SFFM(Voff Vamp Fcar MDI Fsig)       AC Amplitude:         ④ PWL(t1 v1 t2 v2)       AC Phase:         DC offset(V):       5         Amplitude[V]:       10         Freq(H2):       60         Tdelay(s):       0         Phi(deg):       45         Ncycles:       30         Additional PWL Points       OK	Independent Voltage Source - V1		
	Functions (none) PULSE(V1 V2 Tdelay Trise Tfall Ton F SINE(Voffset Vamp Freq Td Theta Phi EXP(V1 V2 Td1 Tau1 Td2 Tau2) SFFM(Voff Vamp Fcar MDI Fsig) PWL(t1 v1 t2 v2) DC offset[V]: Amplitude[V]: Freq[Hz]: Tdelay[s]: Theta[1/s]: Phi[deg]: Ncycles:	Ncycles) 5 10 60 0 0 45 30 1 Points	DC value: Make this information visible on schematic: ✓ Small signal AC analysis(.AC) AC Amplitude: AC Phase: Make this information visible on schematic: ✓ Parasitic Properties Series Resistance[Ω]: Parallel Capacitance[F]: Make this information visible on schematic: ✓

This information is shown on the schematic view.



To edit the input information, you can right click on the component as above or right click on the value of the SINE information.

	Enter new Value for V1	
<u> </u>	Justification	ОК
(+)	Left  Vertical Text	Cancel
$\left( -\right)$	SINE(5 10 60 0 0 45 30)	
👘 SI	NE(5 10 60 0 0 4	45 30)

To simulate this same function using an ac analysis, enter the magnitude and phase offset in the "Small Signal AC Analysis" fields.

Independent Voltage Source - V1	×
Functions	DC Value
💿 (none)	DC value:
O PULSE(V1 V2 T delay Trise T fall T on Period N cycles)	Make this information visible on schematic: 🗹
SINE(Voffset Vamp Freq Td Theta Phi Ncycles)	
EXP(V1 V2 Td1 Tau1 Td2 Tau2)	Small signal AC analysis(.AC)
◯ SFFM(Voff Vamp Fcar MDI Fsig)	AC Amplitude: 10
○ PWL(t1 v1 t2 v2)	AC Phase: 45
	Make this information visible on schematic: 🗹
Param1:	Parasitic Properties
Param2:	Series Resistance[Ω]:
Param3:	Parallel Capacitance[F]:
Param4:	Make this information visible on schematic: 🔽
Param5:	
Param6:	
Param7:	
Param8:	
Additional PWL Points	ОК
Make this information visible on schematic: 🗹	Cancel

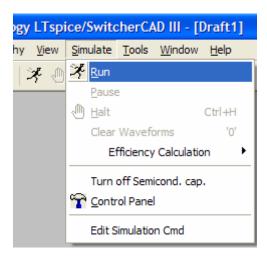
- <u>Delete a component</u> To delete a component, press the "delete" button and click the scissors on a component to delete. Press "esc" to quit deleting components.
- <u>Move a component</u> To move a component, press "F7" or click the move button and left click on the component you want to move. Move the component

to the new location, and left click to place the component. Press "esc" to quit moving components.

- <u>Add a wire</u> To add a wire, press "F3" or click the add wire button. Left click on the starting location, move to the ending location and left click again. If the wire is not straight, click where a 90 degree change in direction is desired. Press "esc" to quit adding wires.
- <u>Add ground</u> Press "G" or click the ground button and click on the schematic to add a ground. Press "esc" to quit adding grounds.
- **Label a net** Press "F4" or click the add net button. Type the name of the net and click "OK". Click to add the net to the schematic.

Net Name	
4D O G	iND(global node 0)
	ом
ABC	
Port	:Type: None 💌
(Port type is the	s only visible if drawn at end of a wire.)
Cancel	
Vout	
R	R1C1
<5	50 100µ

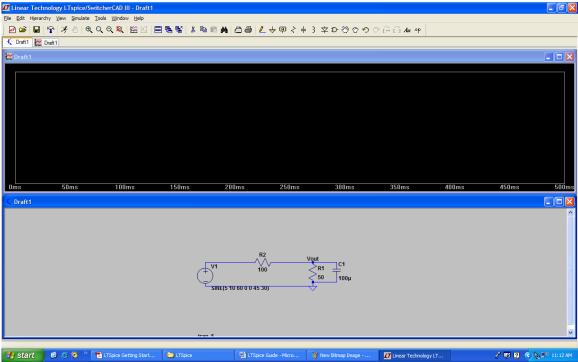
<u>Simulate</u> – To run a circuit, select "Simulate" from the file menu and "Run" or click the run button.



## **Transient Analysis**

Edit Simulation Command
Transient ACAnalysis DC sweep Noise DC Transfer DC op pnt
Perform a non-linear, time-domain simulation.
Stop Time:
Time to Start Saving Data:
Maximum Timestep:
Start external DC supply voltages at 0V: 📃
Stop simulating if steady state is detected: 📃
Don't reset T=0 when steady state is detected:
Step the load current source:
Skip Initial operating point solution:
Syntax: .tran <tstop> [<option> [<option>]]</option></option></tstop>
Cancel OK

Select the appropriate tab for simulation and enter the simulation options and click "OK". For the AC source shown above, a Transient analysis would be desired to see the outputs for the desired 500ms. At a minimum, a stop time 0.5 seconds should be entered because the source was set to 30 cycles.



<u>Measurements</u> – NOTE: When making any measurements, a single click will add the measurement to the plot, while a double click will erase all existing measurements and plot the selected measurement by itself.

**Measure voltage** – To measure a voltage on a node, move the mouse over the node to be measured and click. Note: The voltage probe cursor will appear when the mouse cursor is above an appropriate node.



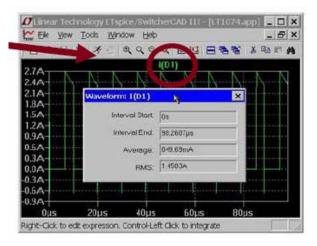
The above method will measure voltage with respect to ground. To measure voltage across a component, click on the positive node to be measured, drag the mouse to the negative node and release the mouse.



**Measure current** – To measure the current through a component, move the mouse cursor over the component and click. Note: The current probe will appear when the mouse cursor is above an appropriate component.



Average and RMS Voltage/Current – After selecting a voltage or current, hold the "ctrl" button and click on the name of the measurement in the measurement window. A new window will open displaying the average and RMS values.



- **Instantaneous Power** To measure the instantaneous power dissipated or supplied by a component, hold the "alt" key and click on the component to be measured.
- Average Power After selecting the instantaneous power of a component, hold the "ctrl" button and click on the name of the measurement in the measurement window. A new window will open displaying the average power.
- **Zoom** To zoom in on a measurement, left click and drag the mouse over the area. To zoom back to the normal area, right click and select zoom to fit or press "ctrl" "E". NOTE: When selecting an area, the size of the area can be viewed in the lower left hand corner.

dx = 14.6321ms(68.3429Hz) dy = 1.9W

#### **AC Analysis**

To perform an AC analysis, click on the AC Analysis tab and enter the type of sweep, number of analysis points, and the stop and start frequencies.

For the voltage source above, we can start by looking only at the response at the frequency of the source. The AC analysis is set to 1 point with a start and stop frequency of 60Hz.

E	dit Simul	ation Comm	and					×
[	Transient	AC Analysis	DC sweep	Noise	DC Transfer	DC op pnt		_
	Compute	the small signa	AC behavio	r of the ci point.	ircuit linearized	about its DC	operating	
			Type of	Sweep:	Octave	*		
		Number	of points per	octave:	1			
			Start Free	quency:	60			
			Stop Free	quency:	60			
1	Syntax: .ac	<oct, dec,="" lin=""></oct,>	<npoints> &lt;</npoints>	StartFreq	> <endfreq></endfreq>			
	.ac oct 1 60	0 60						
l		Cancel	(	OK				

When running the AC analysis, a window will open showing the magnitudes and phases of the voltages and currents in the circuit.

frequency:	60	Hz		
V(a):	mag:	220 phase:	0°	voltage
V(b):	mag:	220 phase:	-120°	voltage
V(c):	mag:	220 phase:	120°	voltage
V(n001):	mag:	98.3618 phase:	63.4308°	voltage
V(n002):	mag:	98.3618 phase:	-56.5692°	voltage
V(n004):	mag:	98.3618 phase:	-176.569°	voltage
V(n003):	mag:	2.5619e-014 phase:	-33.6901°	voltage
I(L3):	mag:	19.6767 phase:	-86.5577°	device_current
I(L2):	mag:	19.6767 phase:	33.4423°	device_current
I(L1):	mag:	19.6767 phase:	153.442°	device_current
I(R3):	mag:	19.6767 phase:	-86.5577°	device_current
I(R2):	mag:	19.6767 phase:	33.4423°	device_current
I(R1):	mag:	19.6767 phase:		device_current
I(V3):	mag:	19.6767 phase:	-86.5577°	device_current
I(V2):	mag:	19.6767 phase:		device_current
I(V1):	mag:	19.6767 phase:	153.442°	device_current

To view the circuit for multiple frequencies, increase the number of points and set the start and stop frequencies to view the desired frequencies. The voltages, currents, and power can be measured as described above.