LTspice Tutorial

By Dubari Borah

Under the supervision of Dr. Dimitra Psychogiou and Dr. Zoya Popovic

ECE Department, University of Colorado, Boulder

Get started with LTspice

• Download and Install LTspice

Go to <u>http://www.linear.com/designtools/software/</u> and you can follow the link <u>https://www.youtube.com/watch?v=OTp9PP_qDcs</u> to download and install the video.

Drafting a Design using the Schematic editor

1. Create a new Schematic

⇒Select 'File' and 'New Schematic'.



2. Add a Component

> To add a resistor, press 'R' or click the resistor button.

Press 'L' or 'C' to insert an inductor or a capacitor respectively. Otherwise, click the respective buttons.



➢ For any other components, press F2 or click the component button. A 'Select Component Symbol' window will open and then from there, browse to get the desired component.

Select Compon	ent Symbol		
Top Directory: C	NProgram Files/LTC	.SwCADIII\lib\sym	~
C:\Program File	s\LTC\SwCADIII\lib\	Open this macromodel's to	est fixture
[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 lpnp Itline	mesfet njf nmos npns4 npn2 npn3 npn4 pif pmos pmos4
<)	>
Lan	cel		J .::

3. Modify Component Values

 \geq Right click on the component and enter the value in default unit (Ω , H, F etc.)



➢ For a DC Voltage source, enter the voltage and series resistance.



For all other voltage sources, click 'Advanced' and select the type of voltage source and enter the required parameters.

Independent Voltage Source - V1	2
Functions	DC Value
💿 (none)	DC value:
O PULSE(V1 V2 Tdelay Trise Tfall Ton Period Ncycles)	Make this information visible on schematic: 🗹
 SINE(Volfset Vamp Freq Td Theta Phi Ncycles) 	
O EXP(V1 V2 Td1 Tau1 Td2 Tau2)	Small signal AC analysis(.AC)
SFFM(Voff Vamp Fcar MDI Fsig)	AC Amplitude:
O PwL(t1 v1 t2 v2)	AC Phase:
	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:	Creat
Make this information visible on schematic: 🗹	Cancel

Example 1: simulate v(t)=5 + $10sin(2 * \pi * 60 + 45^{\circ})V$ for 100ms as a transient analysis.

Independent Voltage Source - V1	×
Functions	DC Value
	DC value:
O PULSE(V1 V2 Tdelay Trise Tfall Ton Period Ncycles)	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:
O PWL(t1 v1 t2 v2)	AC Phase:
O PWL FILE: Browse	Make this information visible on schematic: \bigtriangledown
	Parasitic Properties Series Resistance[Ω]:
DC offset[V]: 5	Parallel Capacitance[E]:
Amplitude[V]: 10	Hales this information with the on a character [7]
Freq[Hz]: 60	Make this information visible on schematic:
Tdelay[s]: 0	
Theta[1/s]: 0	
Phi(deg): 45	
Ncycles: 6	
Additional PWL Points Make this information visible on schematic: 🔽	Cancel OK

• This information will be seen on the schematic view as follows and to edit the input information, you can right click on the component as before.



• Now, this source is connected to a 50Ω load to see the waveform as shown below.



4. Delete a Component

≻Click the scissors in the toolbar and press 'esc' to quit deleting.

5. Move an element

➢ To move a component, 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.

6. Add a wire

Click the add wire button. Left click on the starting location, move to the ending location and left click again. Press 'esc' to quit adding wires.

7. Add ground

Press 'G' or click the ground button and click on the schematic to add a ground. Press 'esc' to quit adding grounds.

8.Label a net

Press 'F4' or click the add net button. Type the net name and click 'OK'. Click to add the net to the schematic.





Run and Probe a Circuit

8.Simulate

➢To run a circuit, select 'Simulate' from the file menu and 'Run' or click the run

button.



9. Analysis

Select the appropriate tab for simulation and enter the simulation options and

click 'OK'.



10.Measurement

When making any measurement, 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.

➢ To measure a voltage on a node with respect to ground, move the mouse over the node to be measured and click. The voltmeter probe cursor will appear when the mouse cursor is above an appropriate node.

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.

➢ To measure the current through a component, move the mouse cursor over the component and click. The clamp ammeter will appear when the mouse cursor is above an appropriate component.



Example 2: Basic DC analysis in LTspice



- 1. All the components are added as mentioned before.
- 2. The values of V1, R1,R2,R3 and R4 are modified to give the desired values.
- 3. The net names 'V1' and 'V2' are used as shown in the figure below to measure voltages at those points.
- 4. Select 'Simulate' from the menu and click the 'Edit Simulation Cmd' button to see the different kind of analysis that you can perform here. These are Transient, AC Analysis, DC sweep, Noise, DC Transfer and DC op pnt.

ITspice IV - [parametersweep]		Transmission in the other distances
🔨 <u>F</u> ile <u>E</u> dit H <u>i</u> erarchy <u>V</u> iew	<u>S</u> imulate <u>T</u> ools <u>W</u> indow <u>H</u> elp	_
] 🖻 🛎 🖬 😤 🛪 🕚	≪7 <u>R</u> un	s 🗈 🖻 🖻 🛤 🛛 🗁 🕼 🖊 🔶 🗭 🤇 🕹
🕻 parametersweep 🔛 paramete	Pause	
	Clear Waveforms '0' Efficiency Calculation	
	Tontrol Panel	-
	Edit Simulation Cmd	

5.From these, select 'DC op pnt' and press 'OK' as shown below.

or Edit S	Sim	ulation Com	mand					\times
Transie	ent	AC Analysis	DC sweep	Noise	DC Transfer	DC op pnt		
	Co	mpute the DC	operating poi inducta	int treatin ances as	g capacitance short circuits.	s as open cir	cuits and	
Syntax:	.op	L						_
.op								
L		Cancel	[OK				

This will give voltages at different nodes and currents flowing through different elements in tabular form.

(Operating Point ·		
7(2001) •	10	voltage	
7(n002) •	5	voltage	
7(n003) •	5 2381	voltage	
(B4) ·	0 0047619	device current	
(B3) :	0.005	device current	
(R2):	0.0047619	device current	
(R1):	0.005	device current	
(V1):	-0.0097619	device current	

6.Next, we will do a 'Transient' analysis. For that, again select the 'Simulate' button from the menu and click the 'Edit Simulation Cmd' button. Select 'Transient' from the window, set the 'Stop Time' as 100µs and click 'OK'.

🗸 Edit Sim	ulation Com	mand				>
Transiant	101.1.1	50		DOT (DO	
Transient	AU Analysis	DC sweep	Noise	DC Transfer	DC op pnt	
	Perf	orm a non-lin	ear, time	-domain simulat	ion.	
			Stop T	ime: 100u		
	T	ime to Start S	Saving D	ata:		
		Maximu	um Times	tep:		
	Start external D)C supply vol	tages at	0V: 🗌		
SI	op simulating i	f steady state	e is detec	ted: 🗌		
Don't r	eset T=0 wher	n steady state	e is detec	ted:		
	Ste	p the load cu	irrent sou	rce: 🗌		
	Skip Initia	al operating p	oint solu	tion: 🗌		
Syntax: .tra	n <tstop> [<o;< td=""><td>otion> (<optio< td=""><td>n>]]</td><td></td><td></td><td></td></optio<></td></o;<></tstop>	otion> (<optio< td=""><td>n>]]</td><td></td><td></td><td></td></optio<>	n>]]			
.tran 100u						
	Cancel]	OK			

Now click the 'Run' button to see the following panel.



7. To measure voltage at any point of the circuit, take the cursor there and the cursor should change to a voltmeter probe as shown below.

To measure current through a component in the circuit, take the cursor there and it should change to a clamp ammeter as shown below.

In the following plot, the voltages at points V1 and V2 as well as current through R2 are shown.

🔛 Wheatsto	ne bridge.r	aw								,
5.24	٧	′(v1)			V[v2]			I(R2)	5.8mA
5.22V-										-5.6mA
5.20V-										-5.4mA
5.16V-										-5.2mA
5.14V-										-5.0mA
5.12V-										-4.8mA
5.08V-										- 4.6MA
5.06V-										-4.4mA
5.04V-										-4.0mA
5.02V-										- 3.8mA
4.98V										
Oµs	10µs	20µs	30µs	40µs	50µs	60µs	70µs	80µs	90µs	100µs

Example 3: Half-Wave Rectification



AC Analysis

Example 4-Design a maximally flat low pass filter with a cutoff frequency of 2GHz and plot the response for f=0 to 5GHz.

Let's design a 2nd order low pass filter as shown below which has a cutoff frequency of 2GHz. For this, all the required components are added in the schematic as it was done in the previous example. Then, change the element values to those obtained in calculation.



 To assign a value to the voltage source, right click on the voltage source and then select 'Advanced' as we will do AC analysis. Assign values for AC Amplitude and Series Resistance. You can also assign values for 'AC Phase' and 'Parallel Capacitance' if necessary.

🖍 Voltage Source - V1	×
DC value[V]: Series Resistance[Ω]:	OK Cancel Advanced
Independent Voltage Source - V1	×
Functions	DC Value
(none)	DC value:
O PULSE(V1 V2 Tdelay Trise Tfall Ton Period Ncycles)	Make this information visible on schematic: \bigtriangledown
O SINE(Voffset Vamp Freq Td Theta Phi Ncycles)	
O EXP(V1 V2 Td1 Tau1 Td2 Tau2)	Small signal AL analysis(.AL)
O SFFM(Voff Vamp Fcar MDI Fsig)	AC Amplitude: 2
	Make this information visible on schematic:
O PWL FILE: Browse	
	Parasitic Properties Series Resistance[Ω]: 50
	Parallel Capacitance[F]:
	Make this information visible on schematic: 🗹
Additional PWL Points	Cancel

- Click on the 'Simulate' button and select ' Edit Simulation Cmd' as in the previous example.
- Select the appropriate tab (in this case 'AC Analysis') for simulation, enter the simulation options and press 'OK'.

σ Edit Simulation Command	×
Transient ACAnalysis DC sweep Noise DC Transfer DC op pnt	
Compute the small signal AC behavior of the circuit linearized about its DC operating point.	
Type of Sweep: Decade 🗸	
Number of points per decade: 20	
Start Frequency: 1e9	
Stop Frequency: 5e9	
Syntax: .ac <oct, dec,="" lin=""> <npoints> <startfreq> <endfreq></endfreq></startfreq></npoints></oct,>	_
.ac dec 20 1e9 5e9	
Cancel OK	

• Then, click the 'Run' button and place the voltage probe at the output which will give an output as shown below. To modify the settings for all the axes, right click on the graph and go to 'Manual Limits'. It will give an window as shown in the next slide where you can change the units, range etc.





• To plot the input impedance of the filter, right click on the plot and select 'Add Trace' as shown below.

Draft1.raw						
				${\cal J}$ Add Traces to Plot		×
				Available data:	Only list traces matching	OK Cancel
				V(n001) V(n002) I(C1) I(L1) I(R1) I(V1) frequency		
1.0GHz 1.4GHz	1.8GHz	2.2GHz	2.6GHz			Iz
🔨 Draft1.asc						
	V1	C1	L1 3.979n <	Expression(s) to add:		
	AC 2.1	3.183p	<	^{>} 50		

In 'Expressions to add', write V(n001)/I(V1) to measure the input impedance. If you want to measure the dissipated power or loss by the load R1, write the expression V(n002)*I(R1).



Comparison between 2nd order LPF and 5th order LPF

