LT spice starting on a Mac Minhyea Lee

1. Download .dmg file and install as instructed in Handout.

2. Starting Ltspice and open New Schematic – this will be your .**asc file** to submit for assignment

Ś	LTspice	File Edit	Window H	lelp		
• 😳		New Open Open Red) 第0 cent	New Schematic D- New Symbol	ЖN	
		Close Save Save As Revert to	፠W ፠S Saved			
•		Page Set Print	up 企業P 業P			
• 30		5				
	11	200	2			
) • 🅎		-	Ser 2		-	

3. It will open a gray area for drawing circuits. Note that there are only 3 buttons on the menu. Dotted gray background can be selected from **View> grid**

and the second s		the second states and
	Untitled	
*	P	Q (normal)
· · · · · · · · · · · · · · · · · · ·		
	this monu approars by "righ	ot click"
	this menu apprears by "righ	
		\mathbf{X}
	And	
	View Fedit	
	😳 😳 🕹 🕹 🕹 🕹 🕹 🕹 🕹 🕹 🕹 🕹 🕹 🕹 🕹	
· · · · · · · · · · · · · · · · · · ·	Draw A Hierarchy	

4. On Mac, LTspice uses right-click a lot! So this is a good time to check your computer how it was set .



** Go to your **System Preference > mouse** or **> trackpad** and check your setting for **Secondary click is right-click for trackpad**.



5. Once you figure out your right click, here we can start with drawing wire by clicking **Draft> Wires:**



6. Here I draw a box for closed circuit. Now add resistance. Again go to **Draft>Component** type Resistor and then click OK



Then place it where you'd like it to be. If you'd like to rotate, press **Ctrl-r** until the correct direction. You can escape the component cursor mode by pressing **esc** key.

7. **IMPORTANT**: LTspice circuit will not work WITHOUT ground!! Add wire and right click at the end of wire . Select **Label Net then choose GND**, then the wire tap will change to

0 🔴 🌑	Untitled	
* • •	Enter Net Name	Q (normal)
	GND(global node 0)	
	Port Type: (none)	· · · · · · · · · · · · · · · · · · ·
	Cancel OK	
	R1:	
	<u> </u>	
· · · · · · · · · · · · · · · · · · ·		Highlight Net
	wire and	Label Net
	t click on the tip to	Mark as Reference
see t	his:menu:::::::::::::::::::::::::::::::::::	Place .op Data Label

8. Now add voltage source – either DC or AC Pulse – Right click and **Draft>Component** and start typing. Then highlight the one you are looking for (here voltage source) from menu and click **OK.** You can specify the characteristics of Voltage source by **right-click.** It opens up a property menu for Voltage source (See Step 10).



9. Add resistance values : **Right click** on the resistor (cursor changes from arrow to hand). Enter the number. "k" represent 103, "MEG", 106, etc. Please google the numeric format.

O O Untitled								
	Edit Resistor R2	Q (normal)						
	Pick a standard percent value							
	Cancel OK							
· · · · · · · · · · · · · · · · · · ·	••••••	Vout						
$(\overset{+}{})$								
V V								
	· · · · · · · · · · · · · · · · · · ·							
	· · · · · · · · · · · · · · · · · · ·							

10. Specify Voltage source: Similarly **right click** on the voltage source – either DC (simply enter the value) or AC , at least give DC offset (0V), amplitude (1V) and Freq (1000 Hz)

000	Untitled	000	Untitled		
×	Edit Voltage Source V1		Edit Voltage Source V1		
	style: DC Value		style: SINE(Voff <u>Vamp Freq Td θ φ <ncycles>)</ncycles></u>		
	DC Value[V]:		DC offset[V]: 0	::::::	
			Amplitude[V]: 1		
	just enter number e.g. 5 (V) 🔡		Freq[Hz]: 1000	:::::	
::::::					
				:::::	
	::		e.g. DC offset 0 V, et/si		
			1V amplitude (NOT NCycles:		
:::::	::			:::::	
			peak2peak) and f=		
				:::::	
	Make this information visible on the schematic 🗹		1000 Hz Make this information visible on the schematic		
	Small Signal Parameters(.AC)				
::::::	AC Amplitude:		Small Signal Parameters(.AC)		
			AC Amplitude:		
:::::	AC Phase[°]:		AC Phase[°]:	:::::	
	Make this information visible on the schematic		Make this information visible on the schematic		
THE .	Parasitic Impedances		Make this information visible on the schematic		
143	Series Resistance[Ω]:	THE A	Parasitic Impedances	and the second	
	Parallel Capacitance[F]:	123	Series Resistance[Ω]:		
	Make this information visible on the schematic		Parallel Capacitance[F]:	Sector S	
	Cancel		Make this information visible on the schematic	The second	
			Cancel		

11. Finally, Each node to be measured have to be specify. Here that would be **Vin** and **Vout** . In order to do this, place the cursor for the location for Vin (or Vout) would be measured (indicated as arrow) and right click. The select **Label Net** and on third item, add



** Now your circuit is completed and need to measure What Vin and Vout looks like

12. Right-Click anywhere and choose Draft>SPICE directive .



13. Edit Box appears. Don't know what to type? Again **Right-click** and choose **Analysis Cmd**



14. We will mostly use **transient** and AC analysis. **Transient** is for time-domain (i.e. x-axis is time as oscilloscope screen).

Transient	AC Analysis	DC Sweep	Noise	DC Transfer	DC Bias Point		
	Perform a n	on-linear, time	-domain s	simlulation.			
		St	op Time:				
	Time to Start S	Saving Wavefo	rm Data:				
	Ma	aximum Times	tep Size:				
	Start external	supply voltag	es at OV:				
Stop the sim	Stop the simulation once steady state is detected ¹ :						
Don't res	et T=0 when ste	eady state is d	etected ¹ :				
Skip the so	lution of the inti	al operating b	ias point:				
1] Only appilcable for S	MPS simulation si	nce the steady s	state detect	tion is written into	the error amp model.		
							
		Car	ncel			0	

15. Specify **Stop time** -- consider your frequency. 5 periods. e.g. for 1kHz, it would be 5 ms. For now, just specify the **Stop Time** and leave others blank (which is always good thing to do, if you don't know what to do with it.) Then click **ok** 16. Now your circuit simulation is ready to run by clicking (obvioualy) run button

	Draft3.asc
* • *	Q (normal)
· · · · · · · · · · · · · · · · · · ·	2 · · · · · · · · · · · · · · · · · · ·
· · · · · · · · · · · · · · · · · · ·	۸.,
	Ĵk [*] · · · · · · · · · · · · · · · · · · ·
	· · · · · · · · · · · · · · · · · · ·
••••••	
SINE(0 1 1000)	
	·····
	· · · · · · · · · · · · · · · · · · ·
	· · · · · · · · · · · · · · · · · · ·
	•••••••••••••••••••••••••••••••••••••••
tran 5n	ns
	·····
This is	your analysis-time domai
The second s	

17. Then graph screen appears Note that x-axis is already marked up to 5 ms



**You can right-click and specify property – like showing grid and data points.



18. Bring the cursor near Vin. The arrow will change to voltage probe and clicking while "Voltage probe" shape cursor, it will plot the Vin.

	Draft3.asc				
* 1	Click to plot voltage: V(Vin)	Q (n			
	R2				
5	1.0k				
· · · · · · · · · · · · · · · · · · ·	Vout				
		•••••			
SINE(0 1 1000)				
	.tran.5ms.				

19. Then click Vout with the voltage probe. It will overplot . (Below green is Vin and blue is Vout)



** If you click while holding down Shift-Ctrl, the voltage probe will change to the **current probe cursor** and clicking will make current vs time plot appear (on current values showns on right y-axis).

Ok .. this is just a bare-bone instruction and now go and play around !!! END