

LTS defense Tutorial

1. Navigate to

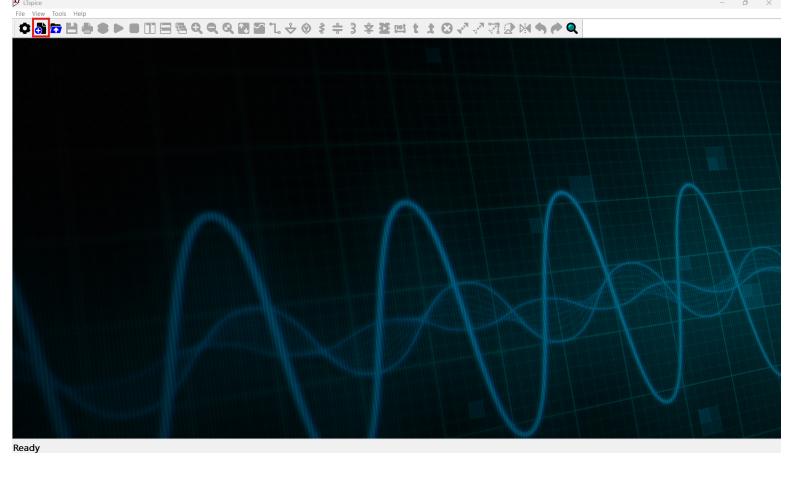
<https://www.analog.com/>

The screenshot shows the Analog Devices website with the URL <https://www.analog.com/> in the address bar. The page title is "LTS defense Tutorial". The main content area features a sidebar with categories: Amplifier & Linear, Clock & Timing, Data Converter, EE-Sim, LTS defense (highlighted in blue), Power Management, RF & Synthesis, and Cybersecurity. Below the sidebar, a section titled "LTS defense" contains a heading "Fast • Free • Unlimited". It describes LTS defense as a powerful, fast, and free SPICE simulator software with enhancements and models for improving the simulation of analog circuits. A call-to-action button "Download LTS defense" is present, along with a note about download availability for various operating systems. To the right, there is a large "LTS defense 24" logo.

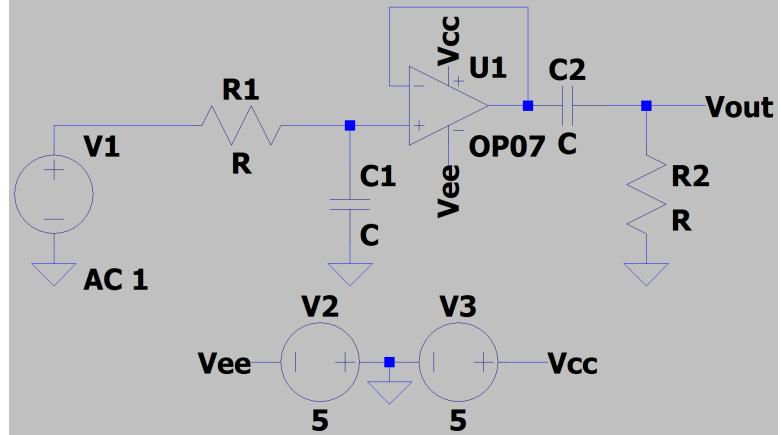
2. Install LTS defense software for your operating system

This screenshot shows the same website page as above, but the "LTS defense" section is expanded. It includes a detailed description of LTS defense's capabilities and its graphical schematic capture interface. Below this, a "Download LTS defense" button is highlighted with a red box. Further down, there are sections for "Get Support" and "Find Demo Circuits". The "Get Support" section includes links to forums and support pages, while the "Find Demo Circuits" section links to demonstration circuit libraries. The "Find Demo Circuits" link is also highlighted with a red box.

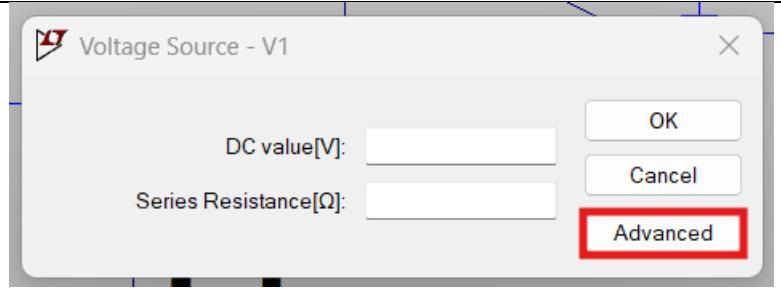
3. Create a new schematic



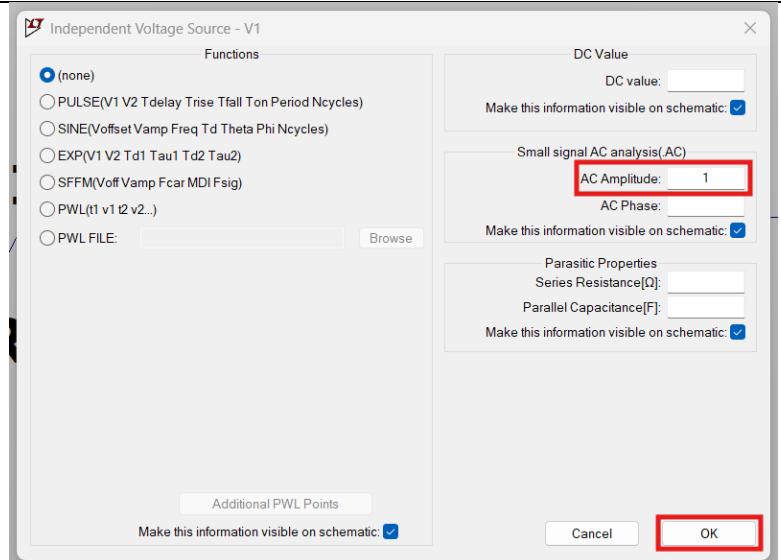
4. Construct the following circuit using the controls listed at the end of the LTSpice tutorial



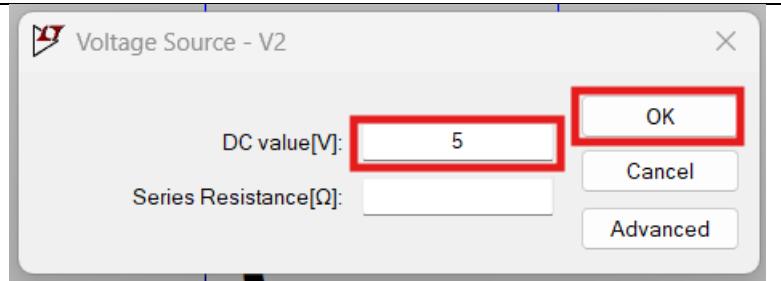
5. Right click on the voltage source V1 (as depicted in the schematic in step 4) and click Advanced



6. In the “Small Signal AC Analysis” section, set the AC Amplitude to “1” and then click Ok



7. Set the other 2 voltage sources at 5V and click Ok



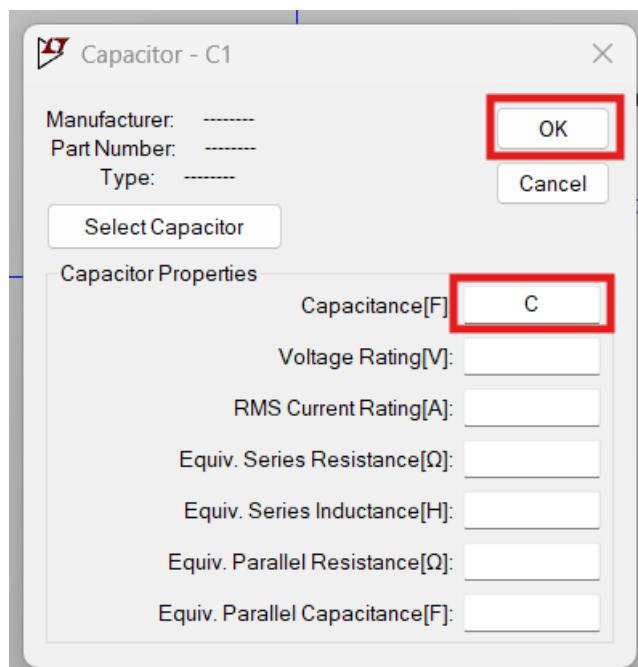
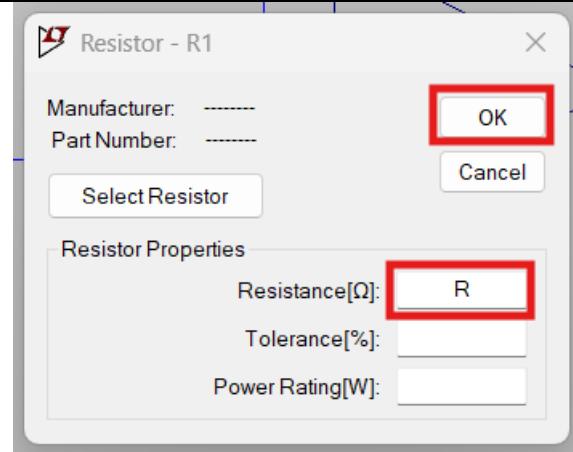
8. The cutoff frequency for a high and low pass filter is defined in the following equation where R is resistance measured in ohms and C is capacitance measured in farads. Use it to determine appropriate capacitor and resistor values for the $f_c=5\text{kHz}$ cutoff

$$f_c = \frac{1}{2\pi RC}$$

9. Enter these resistor and capacitor values into R1 and C1 from the schematic in step (4) by right clicking on the components, entering the value into the appropriate box and clicking Ok

Note: You can use the following postfixes for your numbers according to their order of magnitude

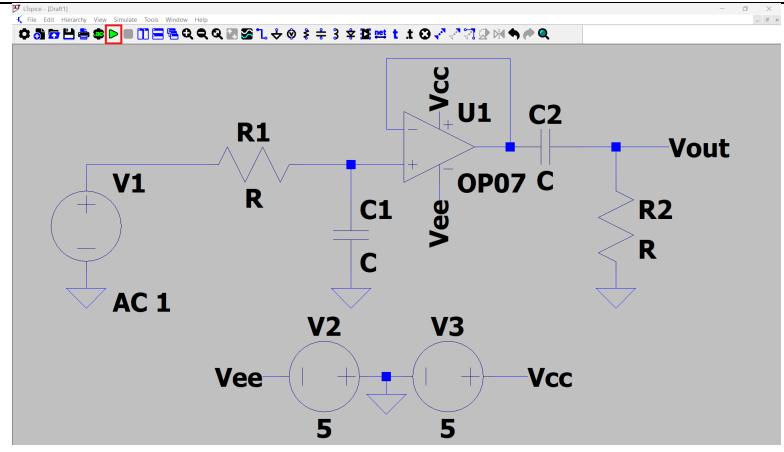
- Mega-: "Meg" (10^6)
- Kilo-: "k" (10^3)
- Milli-: "m" (10^{-3})
- Micro-: "u" (10^{-6})
- Nano-: "n" (10^{-9})
- Pico-: "p" (10^{-12})
- Femto-: "f" (10^{-15})



10. Use the equation from step (7) to determine appropriate capacitor and resistor values for a $f_c=250\text{Hz}$ cutoff

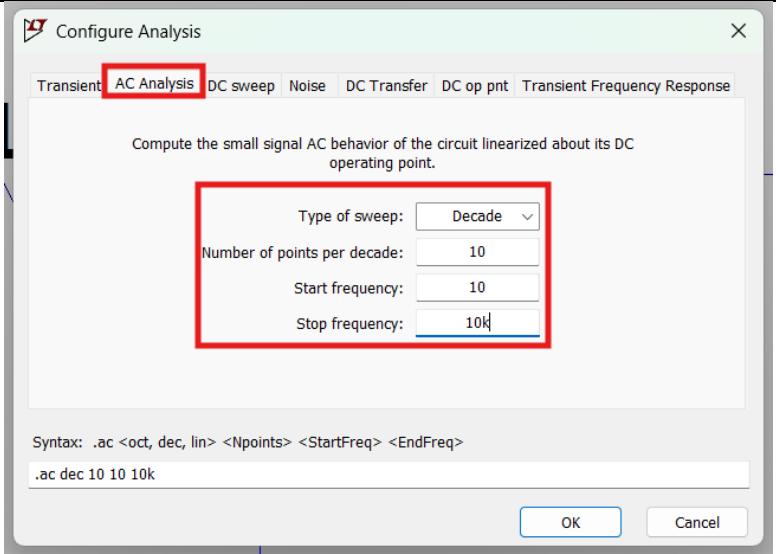
11. Enter the values calculated in step (9) into components R2 and C2 from the schematic in step (4)

12. Click the simulate button in the toolbar

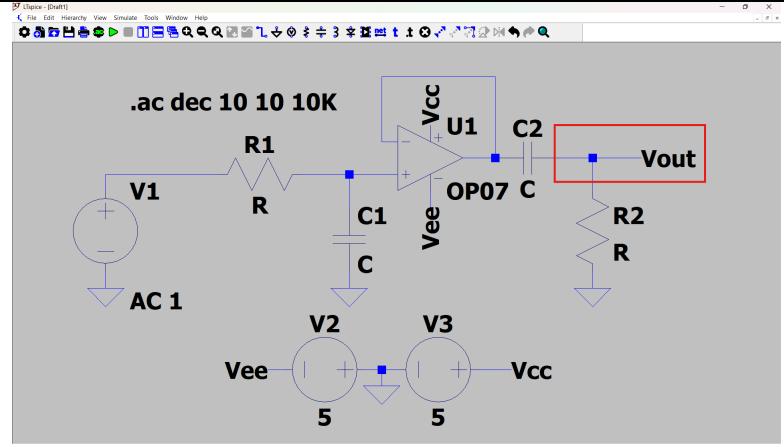


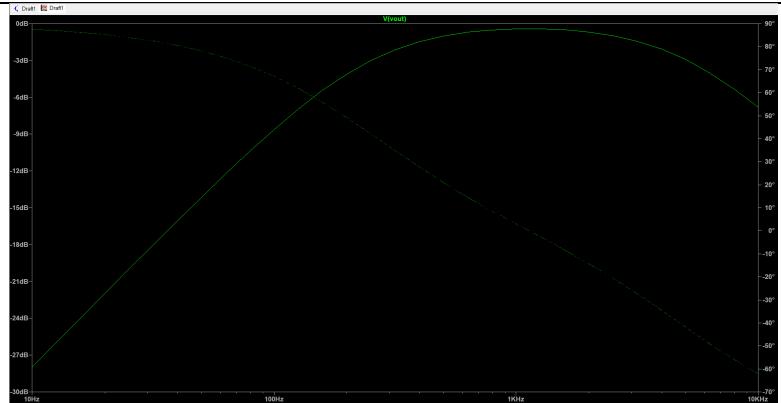
13. Click on “AC Analysis” and adjust the simulation parameters to match the ones depicted in the image on the right.

Note: If you are on a mac, press the “s” key and type the following into the spice directive: “.ac dec 10 10 10k”

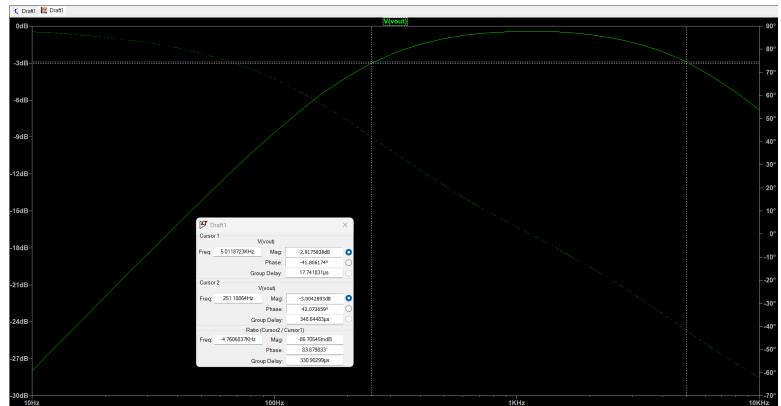
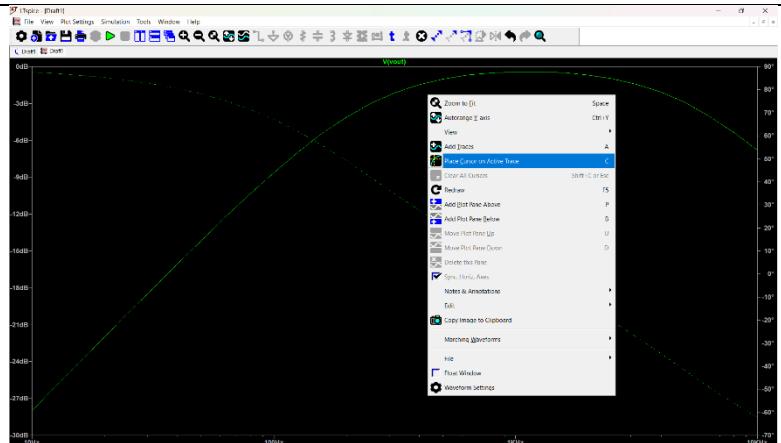


14. After clicking Ok you should get the following graph. You may need to click on the “Vout” net in the schematic.





15. Add 2 cursors by right clicking on the graph and selecting Place Cursor on Active Trace. You can adjust the x-position of the cursors using the arrow keys



16. Ensure that the graph shows a -3db attenuation at 250Hz and 5kHz. If this is not the case, recheck your calculations for the resistor and capacitor values. If you still can't figure out the issue, reference the LTSpice video tutorial.

[LTSpice Tutorial Video](#)

SCHEME AND WAVEFORM EDITING SHORTCUTS

Place Components*		
[W]	wire	[F3]
[G]	ground	[G]
[Alt][G]	com	
[V]	voltage	[V]
[R]	resistor	[R]
[C]	capacitor	[C]
[L]	inductor	[L]
[D]	diode	[D]
[P]	component	[F2]
[N]	label net	[F4]
[T]	text/comment	[T]
.	spice directive right-click text field to open "Help me Edit" dialog	[S]
[B]	bus tap	[B]
left-click	toggle directive/comment	

*Press [Esc] or right-click to exit mode.

Schematic Options		
hold [Ctrl]	place angled wires	hold [Alt]
hold [Ctrl]	draw shapes off grid	hold [Alt]
[Ctrl][Alt][H]	rsr←-1 show hidden text, e.g. parallel or series resistance	
[Ctrl][U]	show/hide unconn pin marks	
[Ctrl][A]	show/hide text anchor marks	
most options available in Settings		

most options available in Settings

Probe Schematic		
click	Probe Wire plot voltage Probe Component plot current	click
[Alt] click	Probe Wire plot current Probe Component plot instantaneous power	[Alt] click
drag net-to-net	Probe Wire plot differential voltage	drag net-to-net

Probes available after simulation is run.

Schemes, Waveforms, Symbols		
[Ctrl][X] or [Delete] or backspace	delete	[F5]
[Ctrl][C]	copy/duplicate*	[F6]
[M]	move* select components to move	[F7]
[S]	stretch* select anchor points to move	[F8]
[Ctrl][R]	rotate	[Alt][R]
[Ctrl][E]	mirror	[Alt][E]
[Z]	Schematic zoom area (drag over area) Waveform zoom area is default mode	Zoom in and out with scroll wheel or use pinch on track pad
[Space]	zoom out	
[Ctrl][G]	toggle grid	
[Ctrl][Z]	undo	[F9] or [Alt][F9]
[Ctrl][Shift][Z]	redo	[Alt][F9] or [Alt][Shift][F9]

Choose mode first, then select component or waveform title.

*Press [Esc] or right-click to exit mode.

Edit Directives & Component Parameters		
right-click >	.t	[C]
	edit directive with help	edit limited parameters
[Ctrl]	edit directive directly	edit all parameters
Text preceded by an underscore, e.g. "_FAULT" is displayed with an overbar, "FAULT".		
Simulator		
[A]	configure analysis	
[Alt][R]	run/pause	
[Alt][S]	stop	
[Ctrl][L]	view SPICE log	[Alt][L]
[O]	reset sim waveform T = 0	

Schematics can be edited even as a simulation runs.
Edits affect subsequent simulations.

Waveform Viewing		
click or [C]	add cursor and see measure	click
[L]	label current cursor position	
[Alt][C] or [Esc]	clear all cursors	close measure dialog
[Alt] click	highlight corresponding net in schematic	[Alt] click
[Ctrl] click	integrate	[Ctrl] click
drag	move trace (to another pane)	drag
drag, hold [Ctrl]	copy trace (to another pane)	
[A]	add trace	
[P]	add pane above	
[B]	add pane below	
[U]	move active pane up	
[D]	move active pane down	
[D]	select steps	
[Q]	recenter	

Mouse actions are on waveform trace label.

Waveform Pan & Cursor		
[Up]	No Cursors pan ~25%	
[Left][Right]	Cursor Present snap cursor to next time data point	
[Up][Down]	Cursor Present cycle cursors through traces at current time data point	
[Home] + [Left][Right]	Cursor Present snap cursor to next data point No Cursors pan ~50%	
[Ctrl] or [Home] + [Left][Right]	Cursor Present bump cursor 10 data points	
[Ctrl][Home] + [Left][Right]	Cursor Present bump cursor 100 data points	
[Ctrl]	pan with mouse	
[Ctrl][Home]	pan left and right with mouse	
[Ctrl][Alt]	pan up and down with mouse	
Click in waveform pane to apply keyboard functions to active frame.		
Analog Devices LTspice®24 Fast • Free • Unlimited		

SPICE QUICK REFERENCE

SPICE Analysis (requires exactly one*)

ac
or
A

.ac	perform small signal AC analysis
.dc	perform DC source sweep analysis
.fra	perform a specialized transient simulation to analyze the frequency response of a feedback loop.
.noise	perform noise analysis
.op	find the DC operating point
.tf	find the DC small-signal transfer function
.tran	perform nonlinear transient analysis

* Simulation requires exactly one active spice analysis directive.

Tip: Open Configure Analysis to activate one directive and comment the others.

SPICE Directives

.backanno	annotate subcircuit pin names on port currents; automatically added by netlister
.end	end of netlist; required; added by netlister
.ends	end of subcircuit definition; use with .subckt
.four	compute fourier component
.func	user defined functions
.global	declare global nodes
.ic	set initial conditions
.include	include text from file
.lib	include library
.loadbias*	load a nodeset
.loadstate**	load a previously solved DC solution
.machine	arbitrary state machine
.measure	evaluate user-defined electrical quantities
.model	define a SPICE model
.net	compute network parameters in .AC analysis
.nodeset	supply hints for initial DC solution
.options	set simulator options
.param	user-defined parameters
.save	limit the quantity of saved data
.savebias*	save a nodeset to file
.savestate**	save comprehensive snapshot of state at time in a proprietary file format
.step	parameter sweeps
.subckt	define a subcircuit
.temp	temperature sweeps
.wave	write selected nodes to a .WAV file

* superceded by .savestate/.loadstate, **versions 24.1 and later

Spice Lines

Leading Character	Type of Line
*	comment
A	special function device
B	arbitrary behavioral source
C	capacitor
D	diode
E	voltage dependent voltage source
F	current dependent current source
G	voltage dependent current source
H	current dependent voltage source
I	independent current source
J	JFET transistor
K	mutual inductance
L	inductor
M	MOSFET transistor
O	lossy transmission line
Q	bipolar transistor
R	resistor
S	voltage controlled switch
T	lossless transmission line
U	uniform RC-line
V	independent voltage source
W	current controlled switch
X	subcircuit invocation
Z	MESFET or IGBT transistor
@	frequency response analyzer
&	frequency response analysis probe
.	simulation directive; for example: .options reltol=1e-4
+	continuation of the previous line



LTspice® 24
Fast • Free • Unlimited

NUMBERS

Constants

LTspice	Means
e	Euler's number
pi	π
k	Boltzmann constant
q	charge constant
true	1
false	0

Used in waveform math

DRAWING

editor >						
t						
arrow						
line						
rectangle						
ellipse						
arc						

not all options available in all modes

COMMAND LINE FLAGS

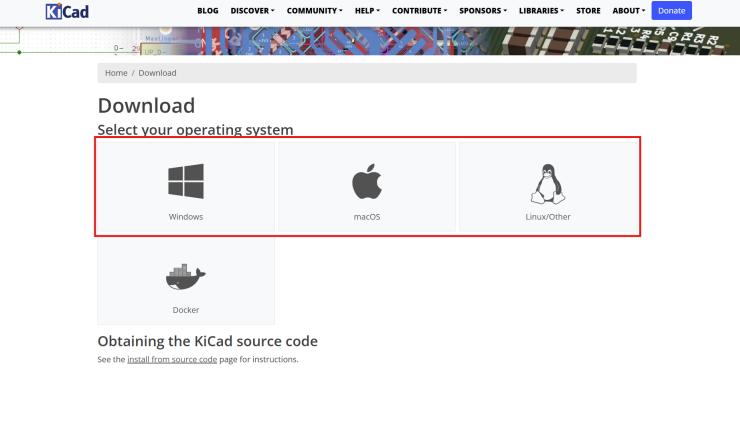
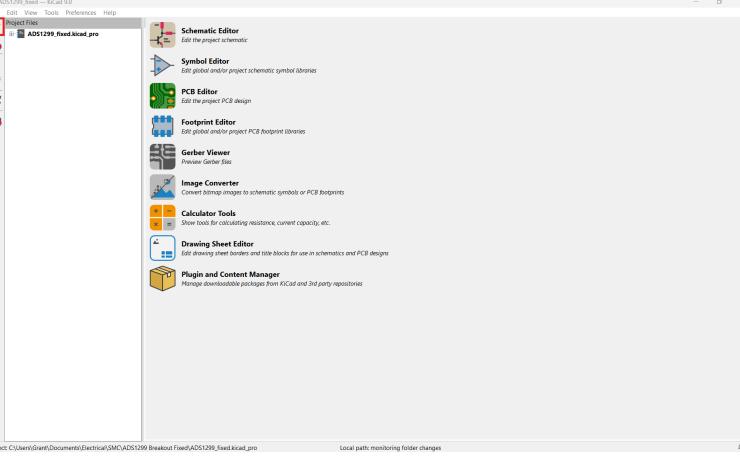
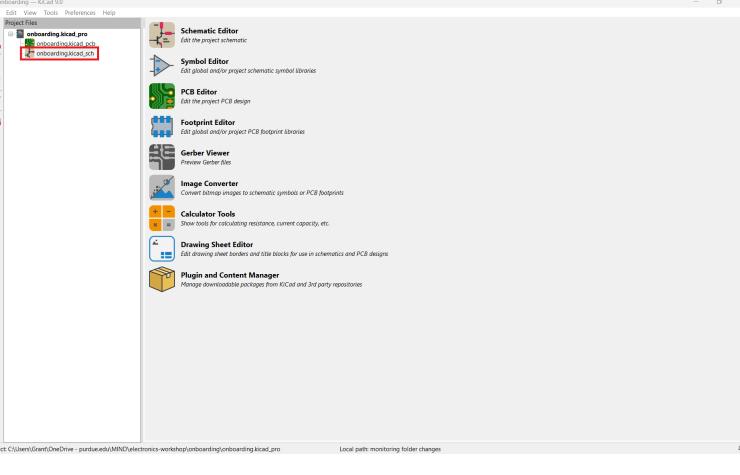
-alt	set solver to Alternate
-ascii	use ASCII .raw files, degrading performance
-b <command>	batch mode of -run -netlist, or -sync, eg. ...-b -run
-big or -max	start LTspice as a maximized window
-ini <path>	use non-default .ini file
-l<path>	path to insert in the symbol and file search paths; no space after l (cap "l"); eg. -lC:\Users\...
-norm	set solver to Normal
-run	open the schematic and simulate
-encrypt	encrypt a model library
-FastAccess	convert a binary .raw file to Fast Access format
-FixUpSchematicFonts	convert the font size field of very old user-authored schematic/symbol text to the modern default
-FixUpSymbolFonts	convert the font size field of very old user-authored schematic/symbol text to the modern default
-netlist	batch conversion of a schematic to a netlist
-PCBnetlist	convert schematic to a PCB format netlist
-sync	update component libraries
-uninstall	uninstall LTspice

Syntax: LTspice.exe -l<path> <schematic.asc> -b -run -ini <path>

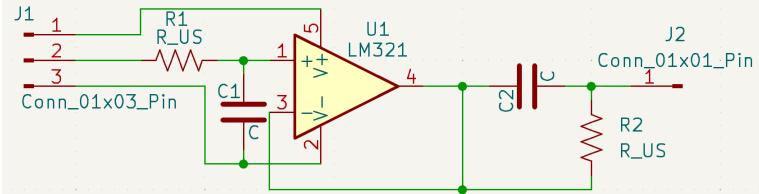
Path required for files not in same directory as LTspice.exe.

Can be stated as a full file path or defined using l<path>.

KiCad Tutorial

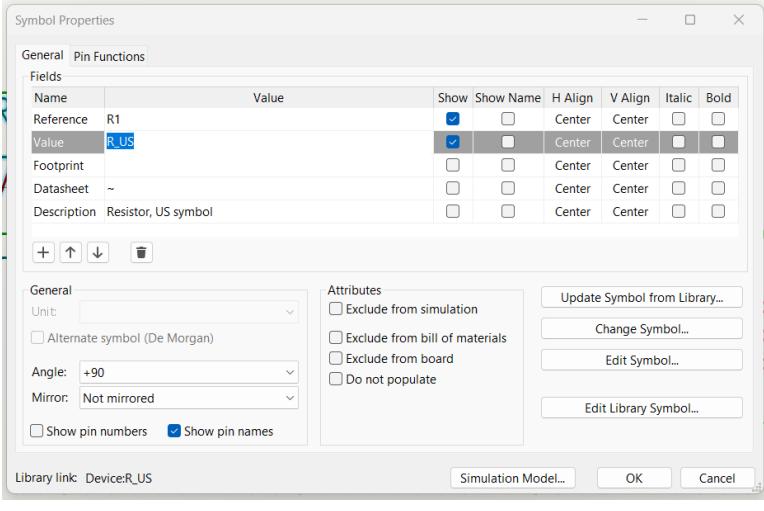
<p>1. Navigate to https://www.kicad.org/ and install the KiCad software for your operating system keeping all the installation defaults</p>	
2. Start a new project	
3. Double click on the schematic file	

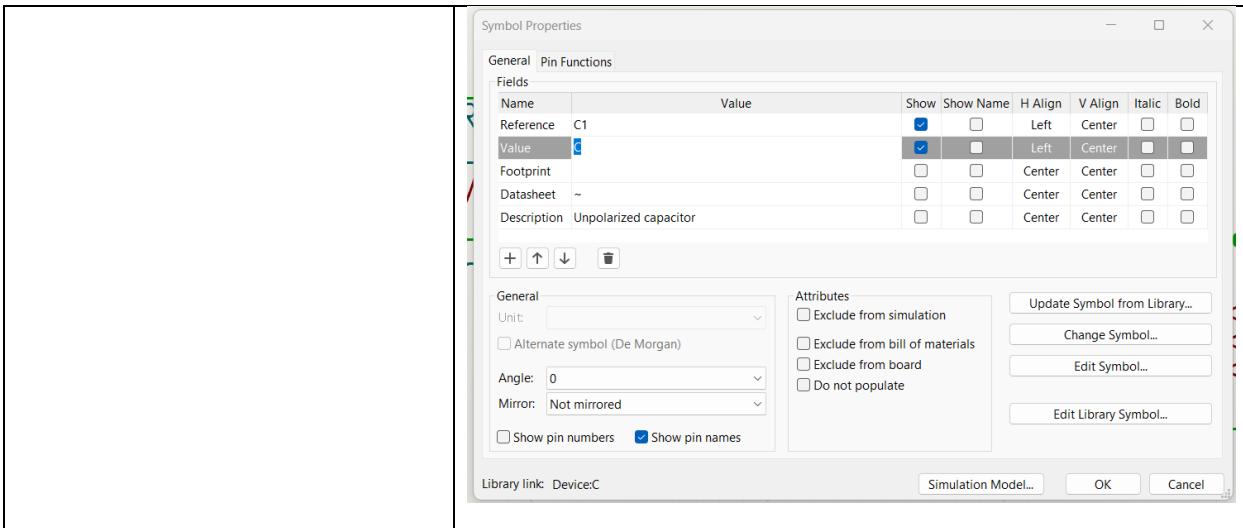
4. Construct the following circuit using the controls listed below



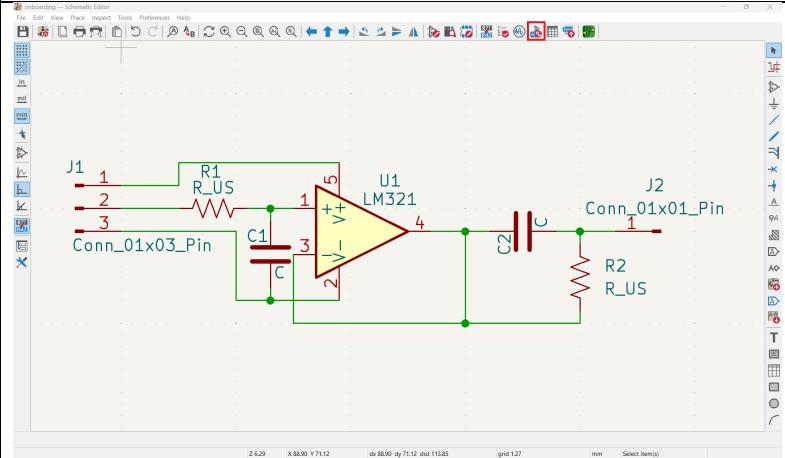
- “w” will create a wire, the terminals of the components can also be clicked to connect components
- “r” will rotate the selected component
- “Ctrl+d” will duplicate a selected component
- Right click dragging will move the screen
- “esc” will stop whatever operation you are performing
- “a” will open a menu with all the parts
- Clicking on a component/wire and pressing “del” will delete the component/wire
- For resistors use the symbol “R_US”
- For capacitors use the symbol “C”
- For the 1x3 connector use the symbol “Conn_01x03_Pin”
- For the 1x1 connector use the symbol “Conn_01x01_Pin”
- For the op amp use the symbol “LM321”

5. Enter the resistor and capacitor values you found in the LTSpice tutorial, into the KiCad schematic by double clicking on the components

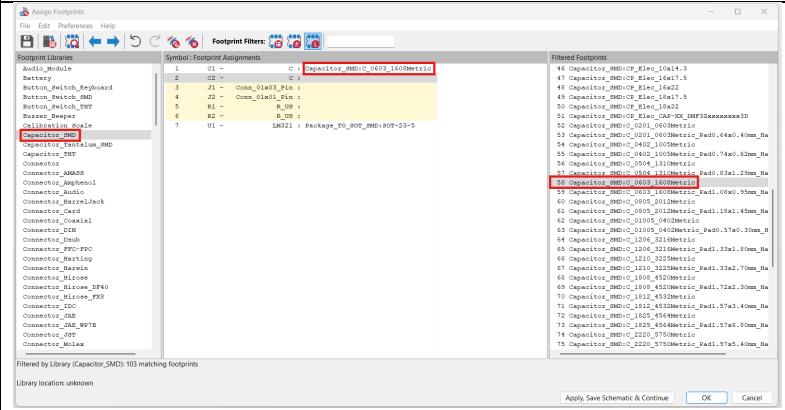




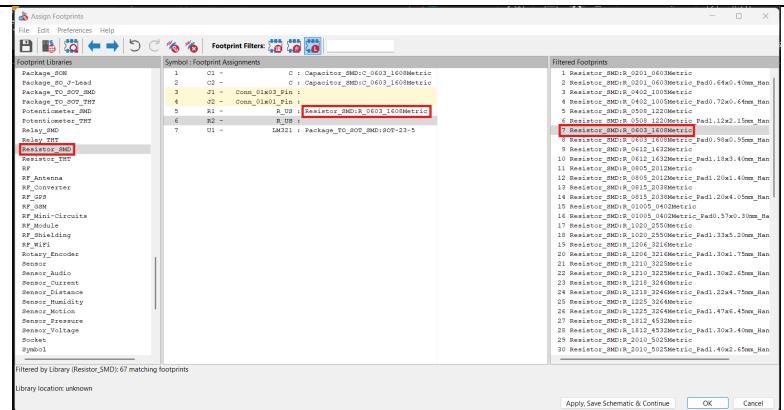
6. Click the Assign Footprints button in the toolbar



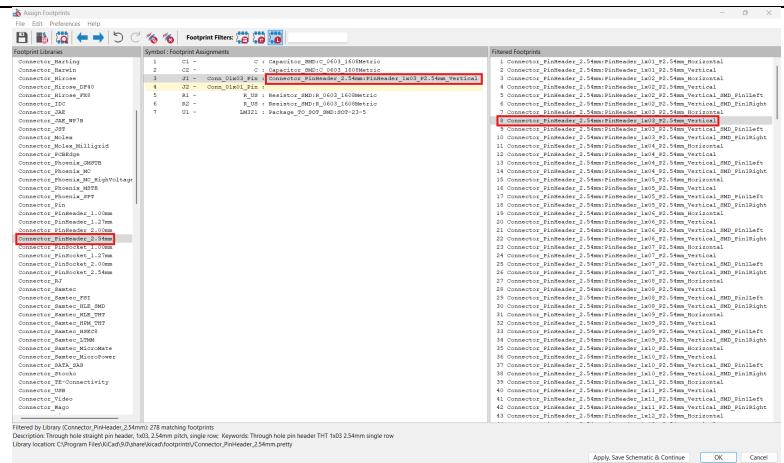
7. Click on capacitors in the middle panel (you may have to do each separately), navigate to “Capacitor_SMD” in the left panel and click it. Navigate to number 58, “C_0603_1608Metric,” in the right panel and double click it. The footprint should populate in the middle panel after the component value.



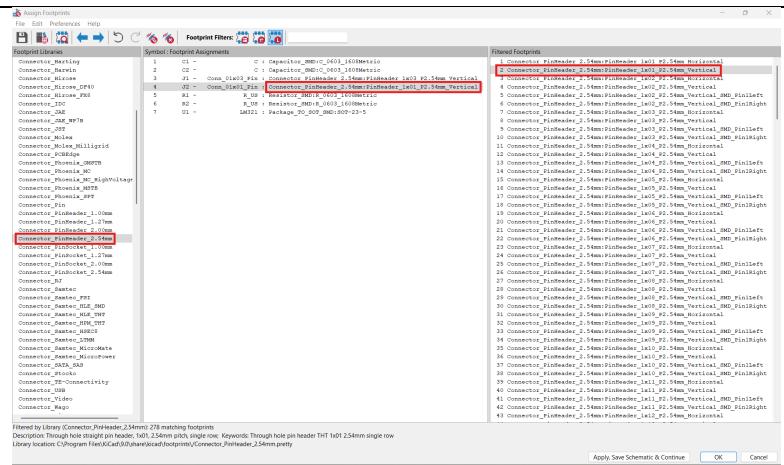
8. Click on the resistors in the middle panel (you may have to do each separately), navigate to “Resistors_SMD” in the left panel and click it. Navigate to number 7, “R_0603_1608Metric,” in the right panel and double click it. The footprint should populate in the middle panel after the component value.



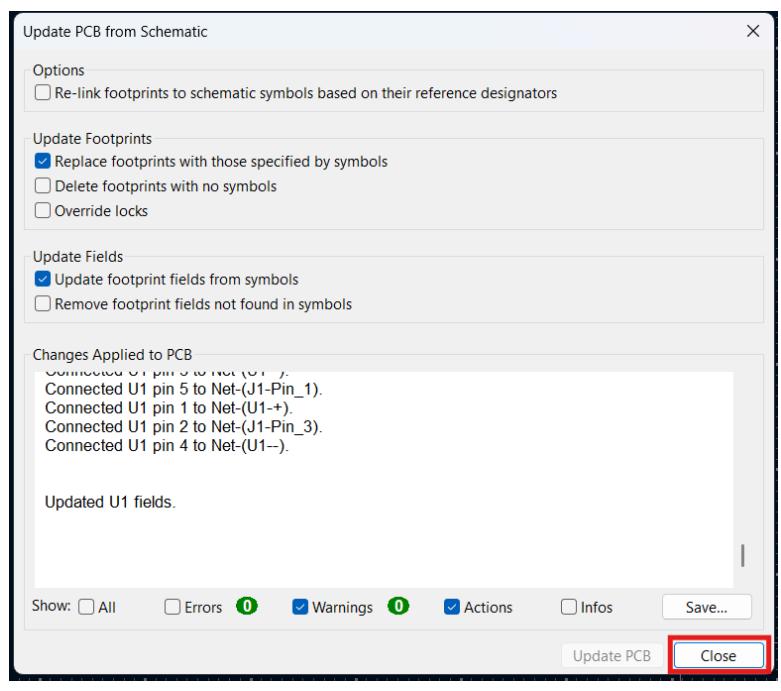
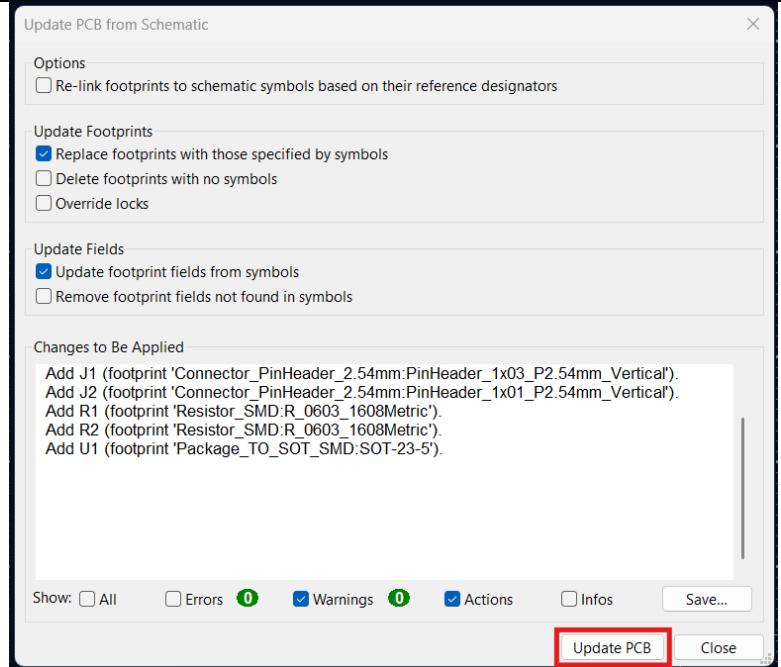
9. Click on the 1x3 connector in the middle panel, navigate to “Connector_PinHeader_2.54mm,” in the left panel and click it. Navigate to the number 8, “PinHeader_1x03_P2.54mm_Vertical,” in the right panel and double click it. The footprint should populate in the middle panel after the component name.



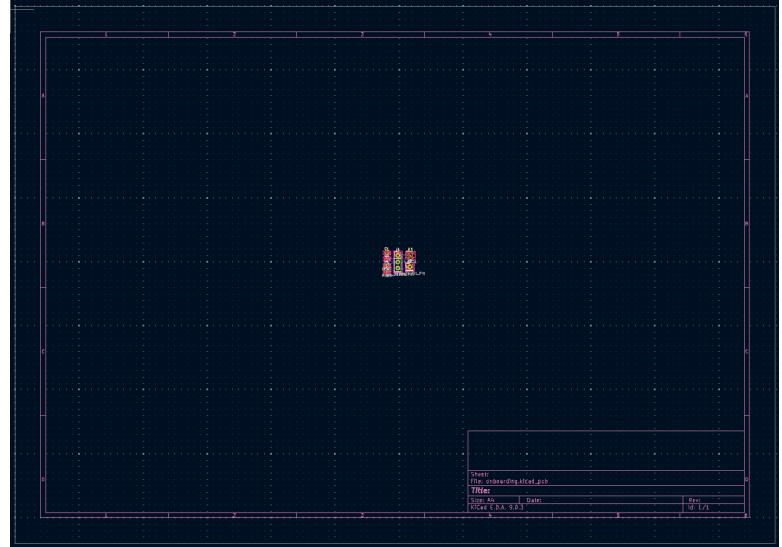
10. Click on the 1x3 connector in the middle panel, navigate to “Connector_PinHeader_2.54mm,” in the left panel and click it. Navigate to the number 8, “PinHeader_1x01_P2.54mm_Vertical,” in the right panel and double click it. The footprint should populate in the middle panel after the component name. Press Ok when finished.



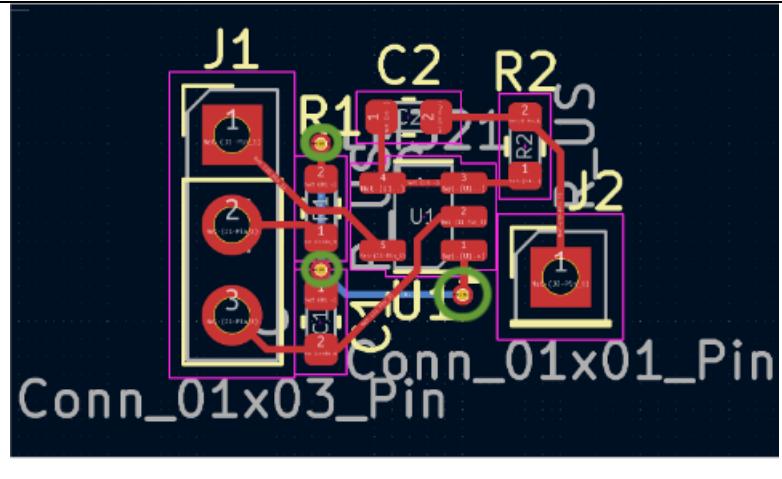
11. Press F8 to create your PCB and click “Update PCB” then click “Close”



12. Left click to place the group of components in the middle of the screen.

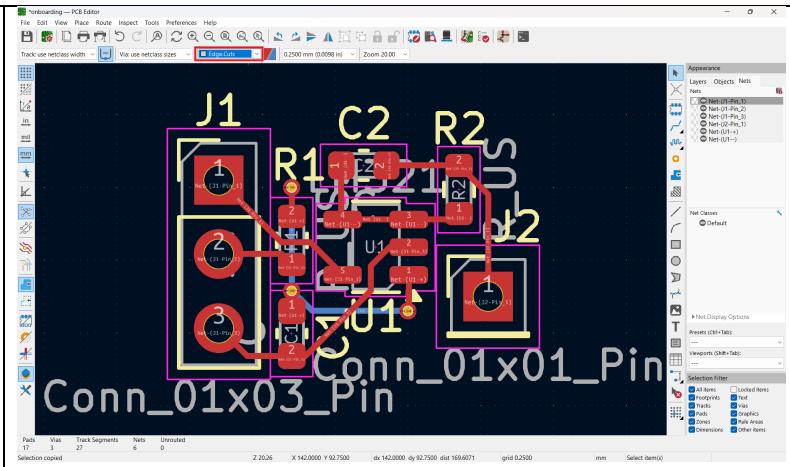


13. Arrange the components in the compact way possible using the following controls. One arrangement example is depicted in the picture on the right

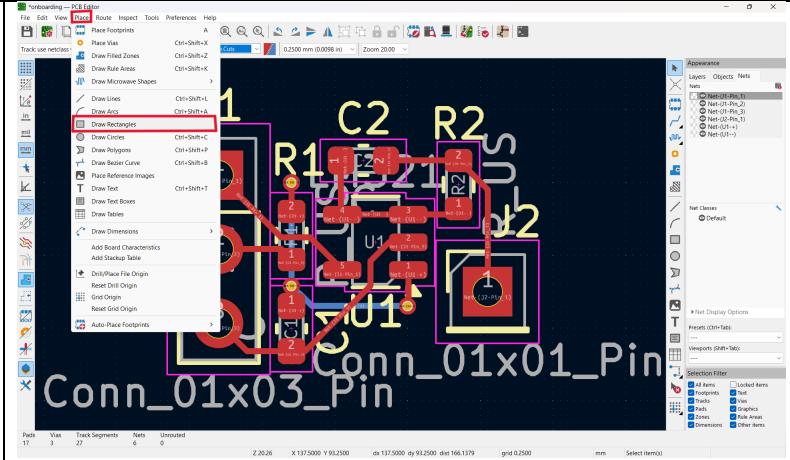


- “x” will create traces (wires) between components
- “v” will create a via which switches the side the trace is on, allowing them to cross
 - The vias are circled in green in the above image
- “r” will rotate the selected component
- Right click dragging will move the screen
- “esc” will stop whatever operation you are performing
- Clicking on a /trace and pressing “del” will delete the trace
- Do not delete components
- Traces of the same color/level can’t cross
- The pink borders of different components can’t cross

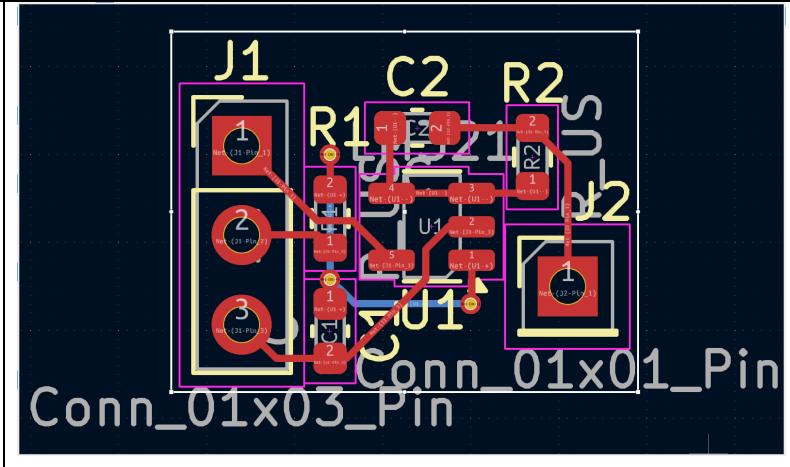
14. Select “Edge Cuts” from the layer drop down menu



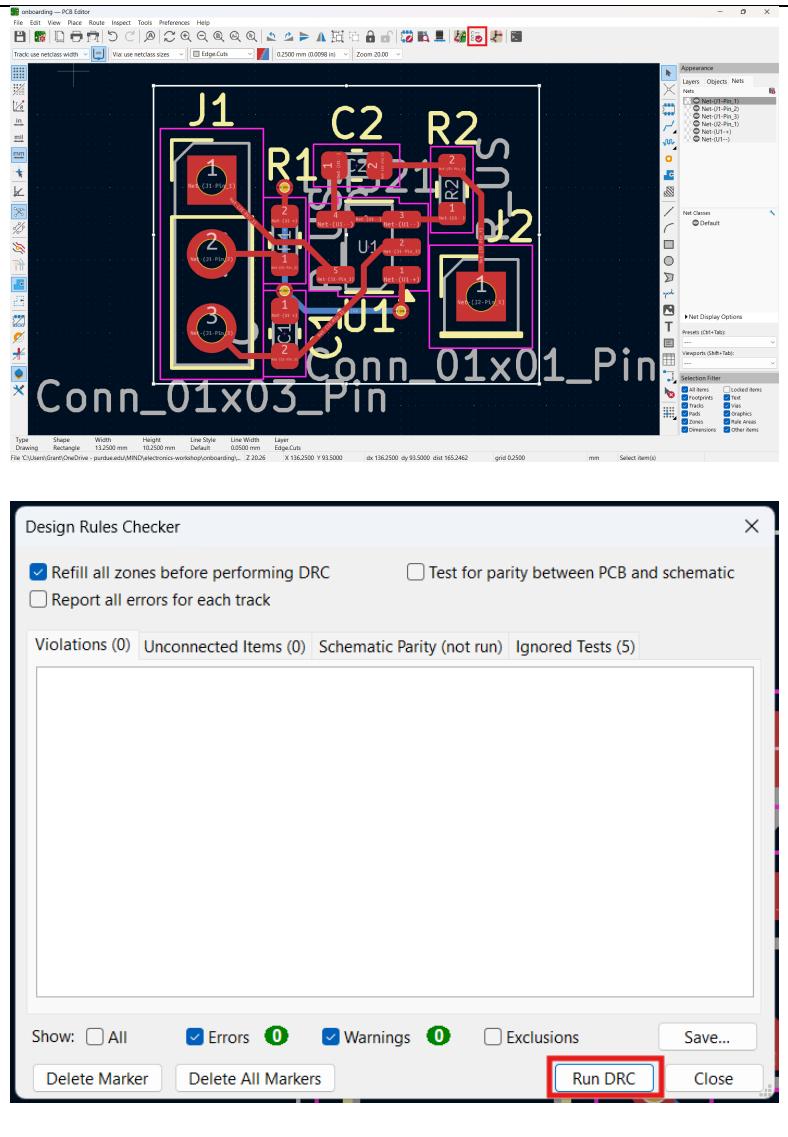
15. Go to “Place->Draw Rectangle”



16. Draw the smallest possible rectangle around your components that still surrounds all footprints and yellow labels

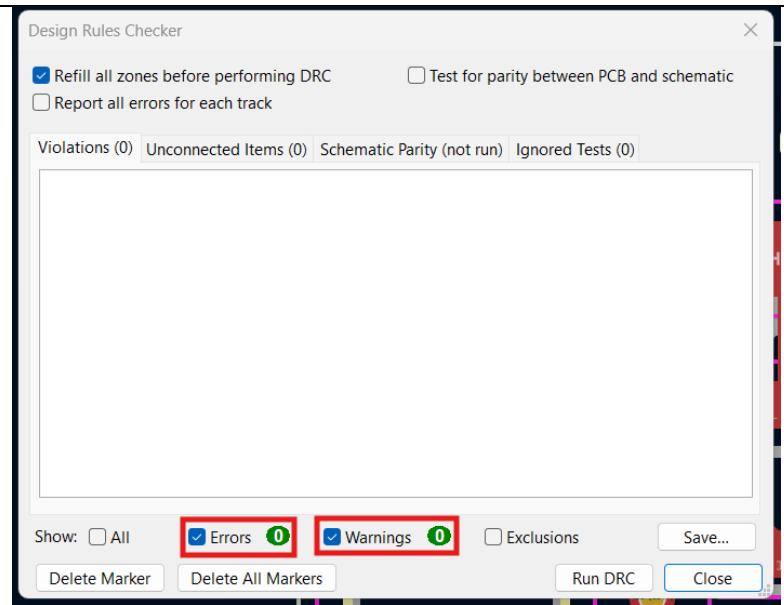


17. Click “Design Rules Checker” then “Run DRC”

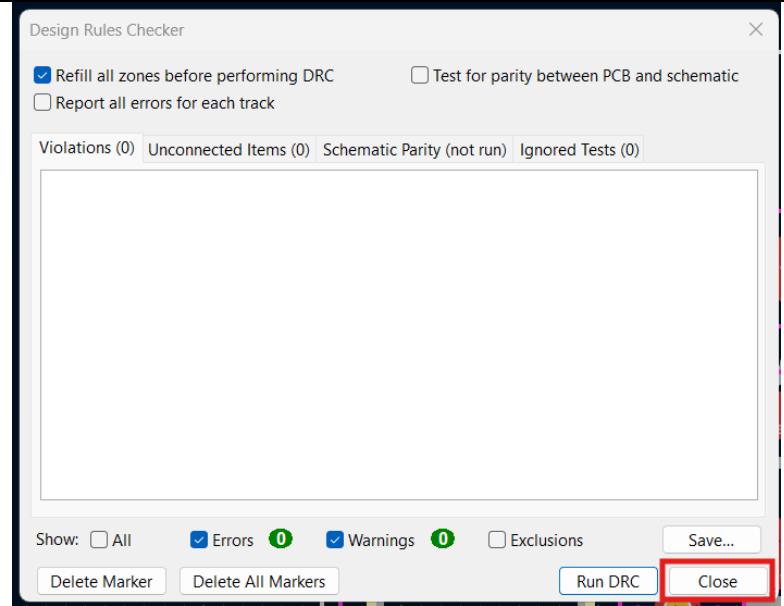


18. Verify that you get 0 errors and 0 warnings. If not, recheck your schematic and PCB layout. If it still isn't working, reference the video tutorial.

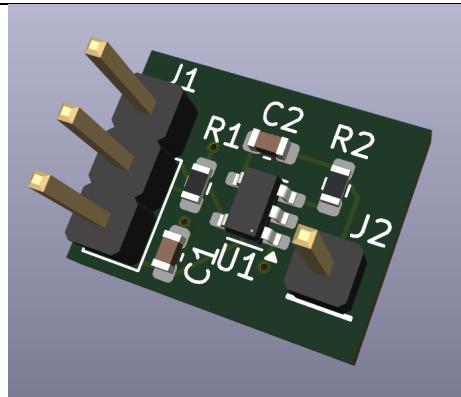
<https://youtu.be/eTiLhKjf7fY>



19. Once there are no errors or warnings, click "Close"



20. Press "Alt+3" to see a 3D model of your PCB



Theory (optional)

Low & high pass filter: <https://www.youtube.com/watch?v=lagfhNjMuQM>

Band pass filter: https://www.youtube.com/watch?v=ENy_zg9dX5c