

LTS defense Tutorial

1. Navigate to

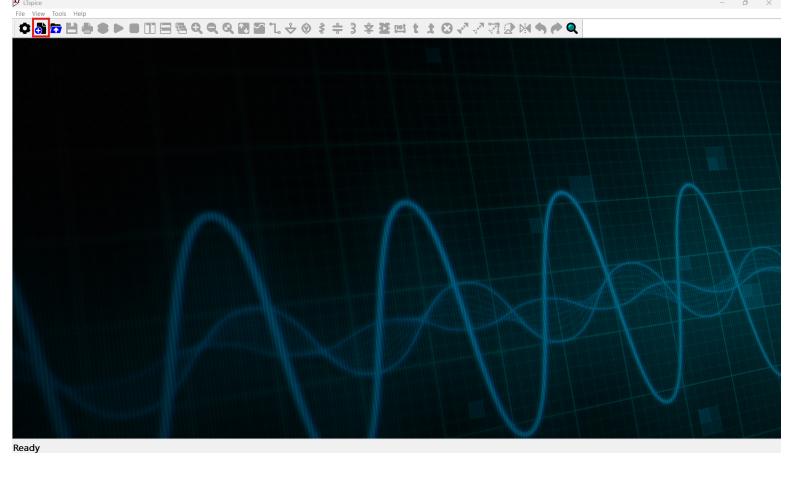
<https://www.analog.com/>

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. The main content area has a heading "LTS defense" and a sub-section "Fast • Free • Unlimited". It describes LTS defense as a powerful, fast, and free SPICE simulator software. A call-to-action button "Download LTS defense" is present, along with download links for Windows 10 64-bit and forward (Version 24.1.9) and Mac OS 10.9 (Version 17.2.4). A sidebar on the right contains the Analog Devices logo and a "Feedback" link.

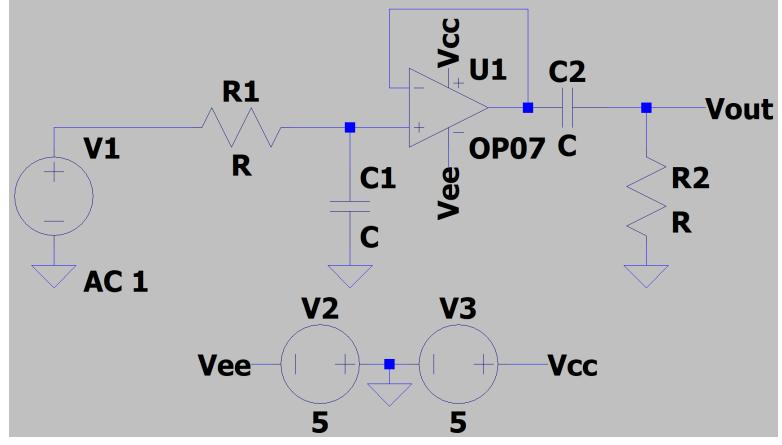
2. Install LTS defense software for your operating system

This screenshot shows the same website section as above, but the "LTS defense" category is now highlighted in blue. The main content area includes a detailed description of LTS defense, download links for Windows 10 64-bit and forward (Version 24.1.9) and Mac OS 10.9 (Version 17.2.4), and a "Feedback" link. The sidebar remains the same.

3. Create a new schematic

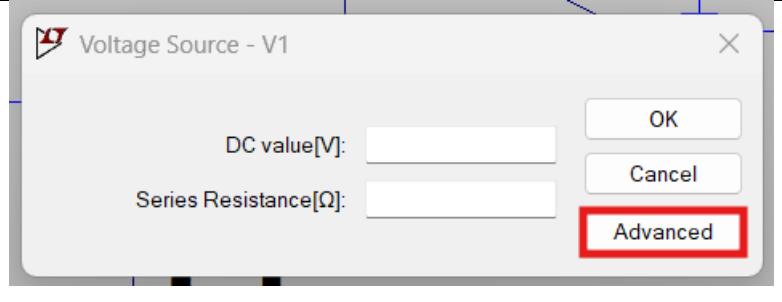


4. Construct the following circuit using the controls listed below if on a Windows computer. If on a mac, refer to the shortcuts listed at the end of the LTSpice tutorial

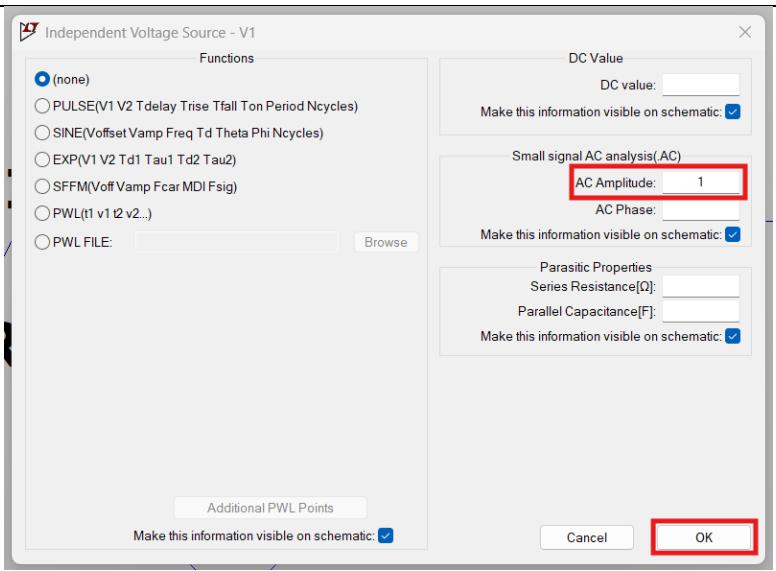


- “v” creates a voltage source
- “r” creates a resistor
- “c” creates a capacitor
- “g” creates a ground reference point
- “n” creates a net which can connect parts of your circuit with labels rather than wires
- “w” creates a wire
- “Ctrl+r” will rotate a selected component
- “esc” will stop whatever operation you are performing
- “del” activates the deletion tool
- “Ctrl+c” activates the duplicate tool
- “p” opens the components menu
 - Use the component labeled “OP07”
- “s” activates the selection tool
- Scrolling controls zoom
- Left click dragging moves the screen

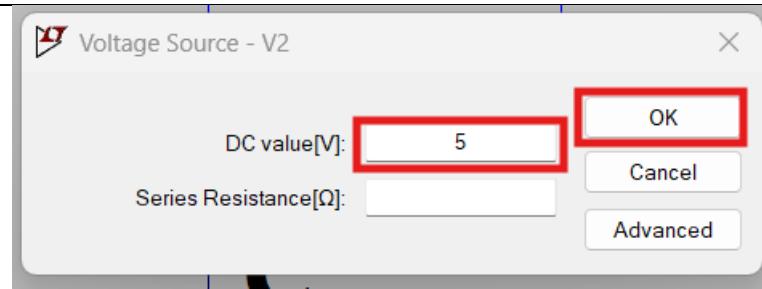
5. Right click on the voltage source V1 (as depicted in the schematic in step 4) and click Advanced



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

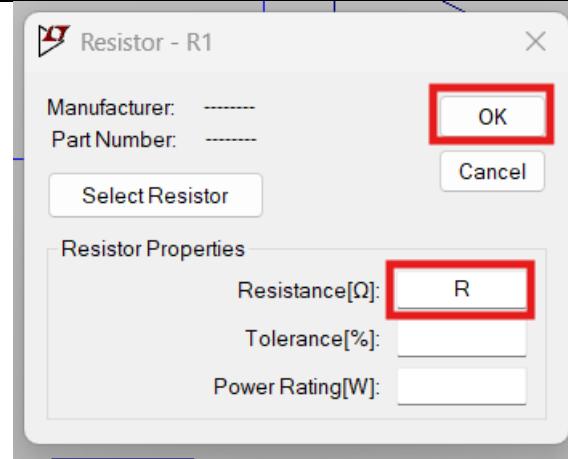


8. The cutoff frequency for a high and low pass filter is defined in the following equation. Use it to determine appropriate capacitor and resistor values for the 5kHz cutoff

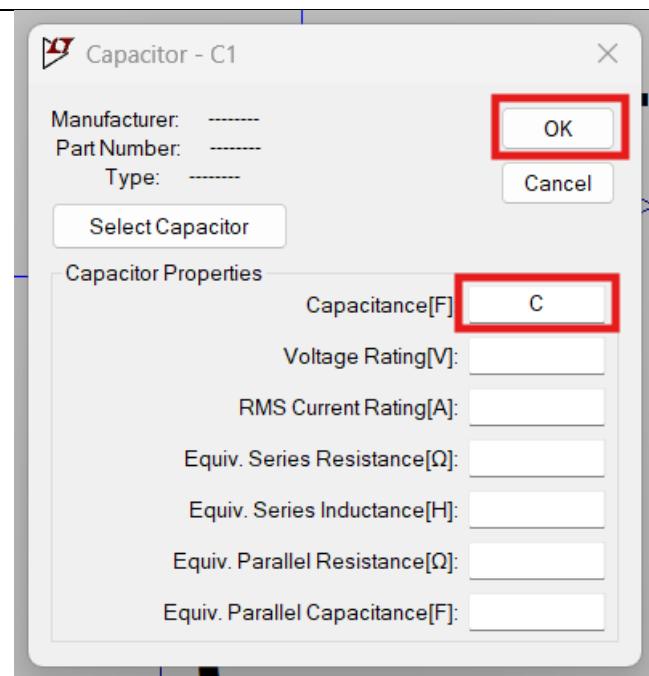
$$f_c = \frac{1}{2\pi RC}$$

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



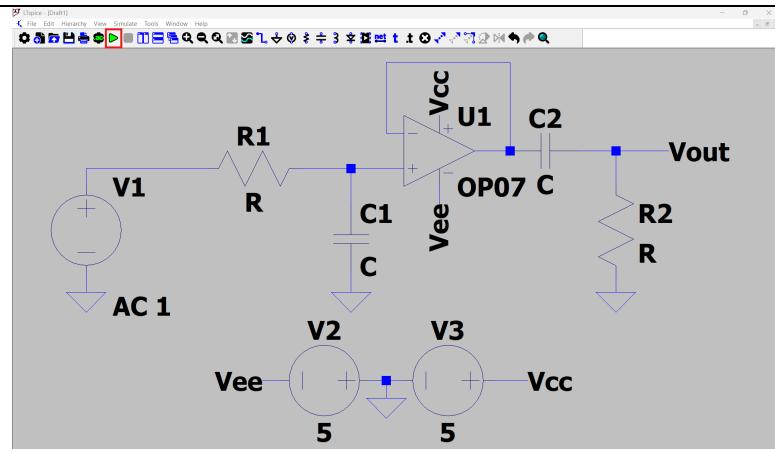
- Mega-: “Meg” (10^6)
- Kilo-: “k” (10^3)
- Milli-: “m” (10^{-3})
- Micro-: “u” (10^{-6})
- Nano-: “n” (10^{-9})
- Pico-: “p” (10^{-12})
- Femto-: “f” (10^{-15})



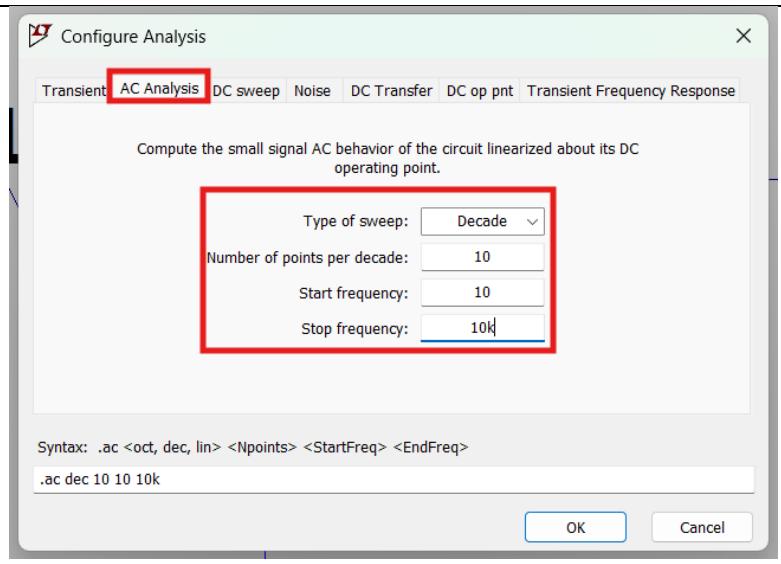
10. Use the equation from step (7) to determine appropriate capacitor and resistor values for a 250Hz cutoff

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

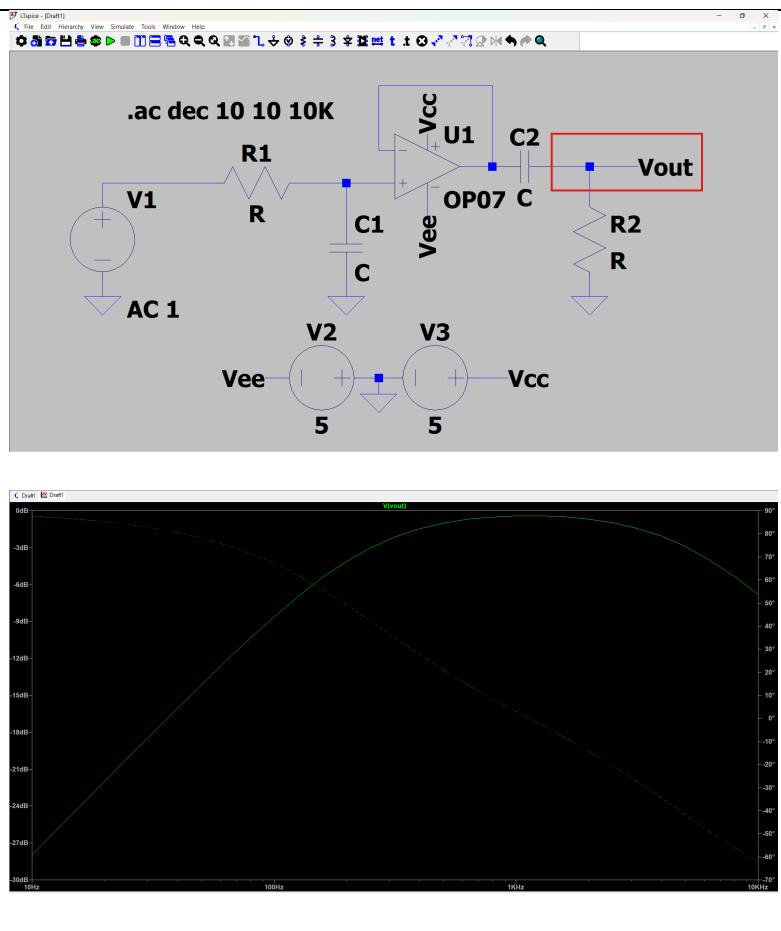
12. Click the simulate button in the toolbar



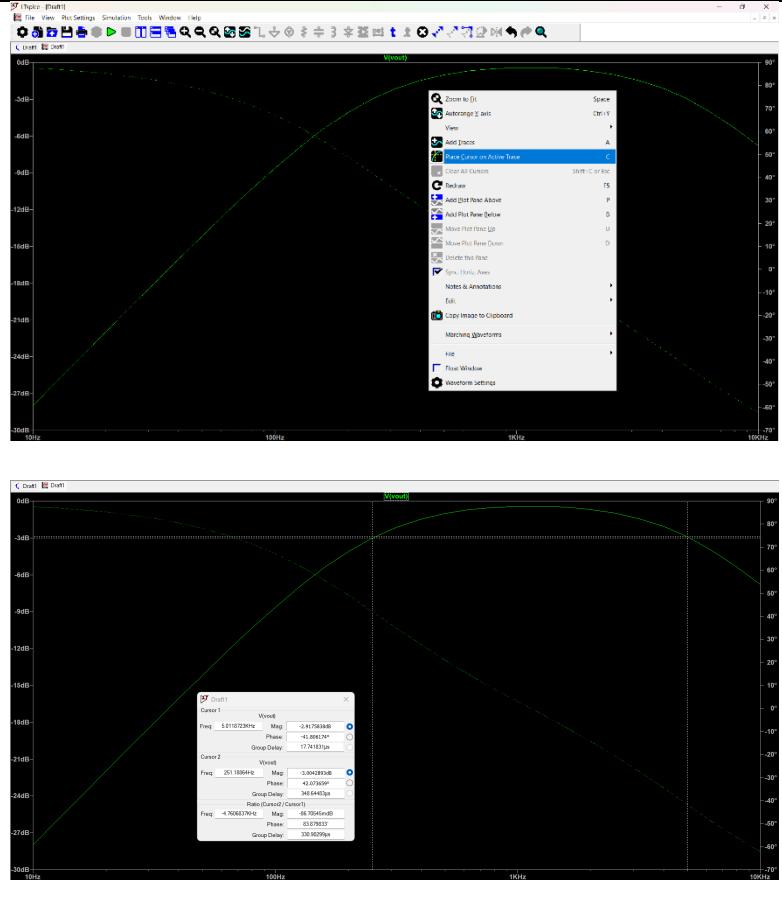
13. Click on “AC Analysis” and adjust the simulation parameters to match the ones depicted in the image on the right.



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



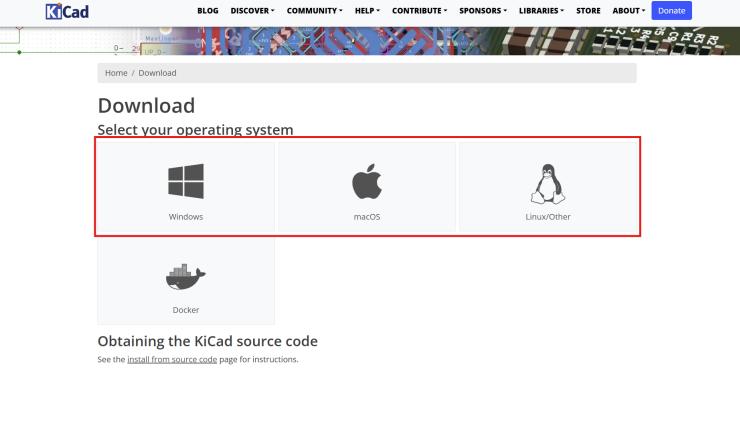
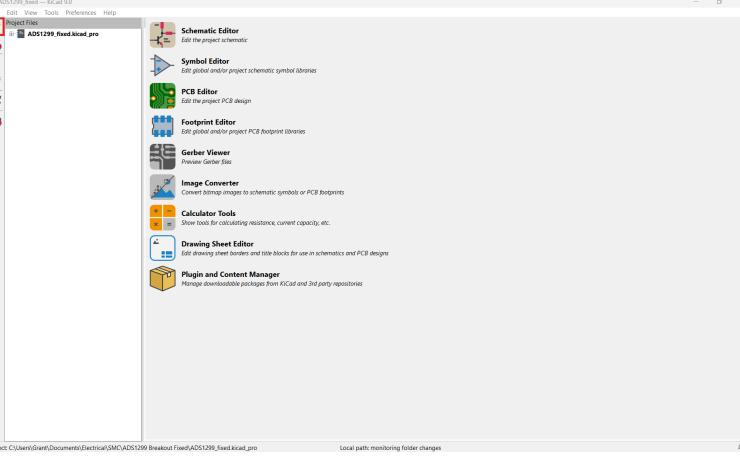
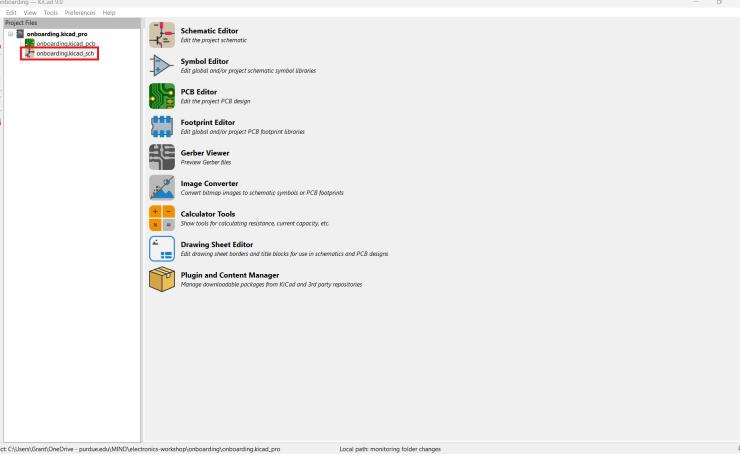
15. Add 2 cursors by right clicking on the graph and selecting Place Cursor on Active Trace. You can adjust the x-position of the cursors using the arrow keys



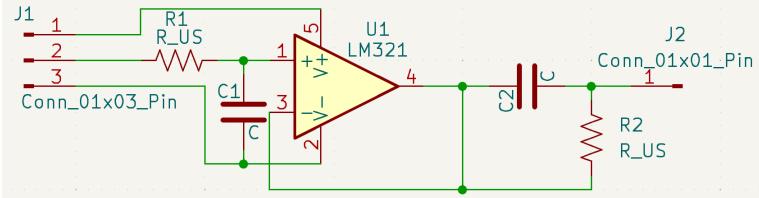
16. Ensure that the graph shows a -3db attenuation at 250Hz and 5kHz. If this is not the case, recheck your calculations for the resistor and capacitor values. If you still can't figure out the issue, reference the LTSpice video tutorial.

[LTSpice Tutorial Video](#)

KiCad Tutorial

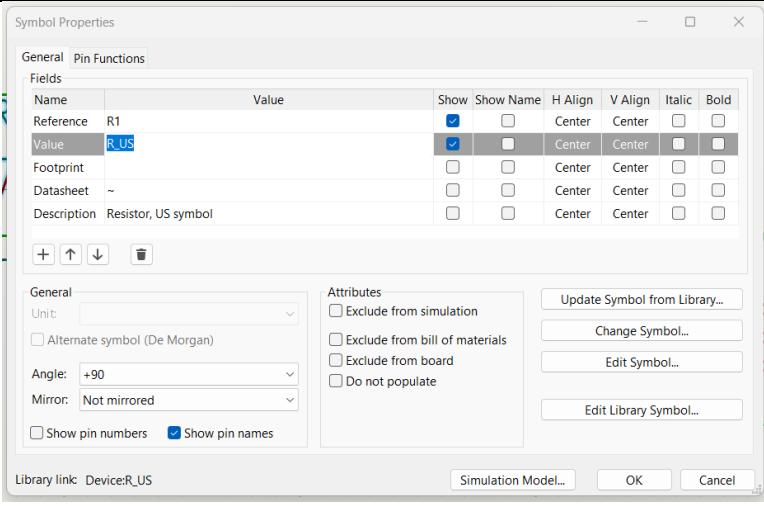
<p>1. Navigate to https://www.kicad.org/ and install the KiCad software for your operating system keeping all the installation defaults</p>	
2. Start a new project	
3. Double click on the schematic file	

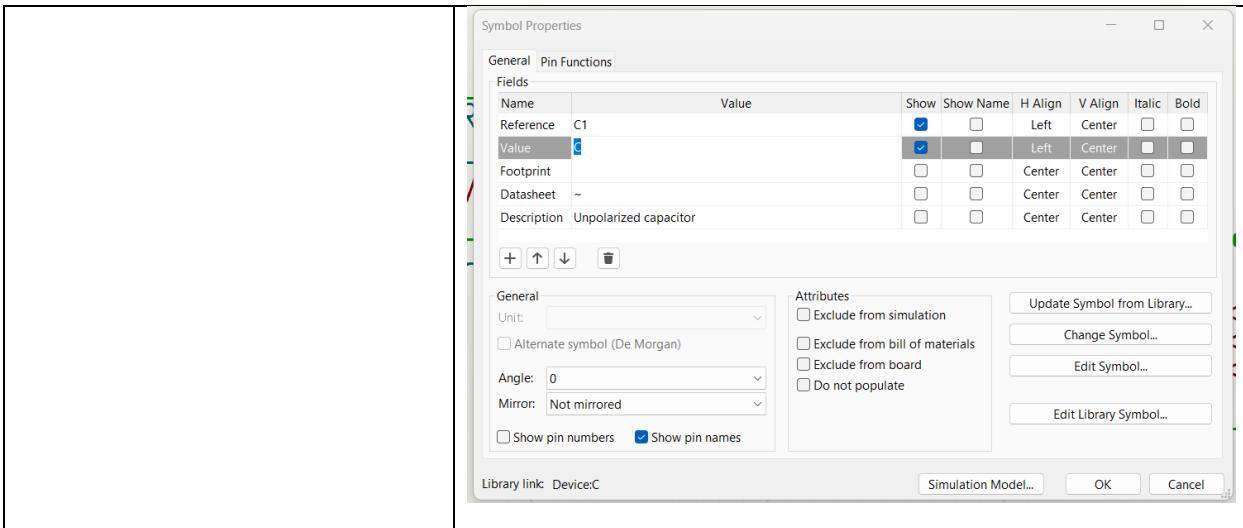
4. Construct the following circuit using the controls listed below



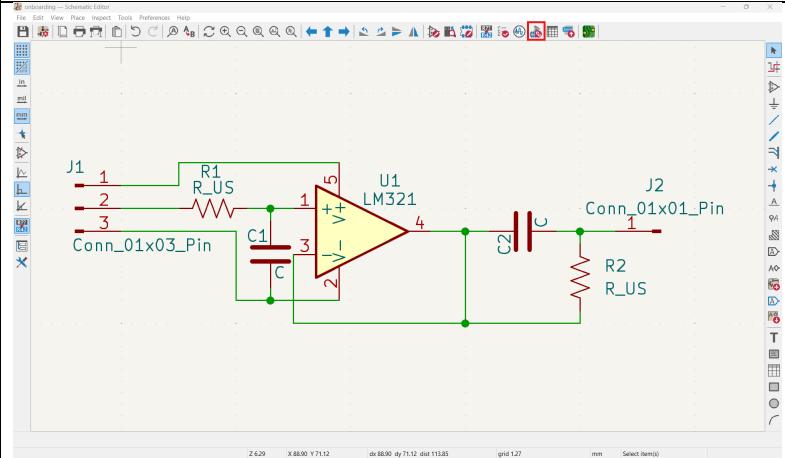
- “w” will create a wire, the terminals of the components can also be clicked to connect components
- “r” will rotate the selected component
- “Ctrl+d” will duplicate a selected component
- Right click dragging will move the screen
- “esc” will stop whatever operation you are performing
- “a” will open 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

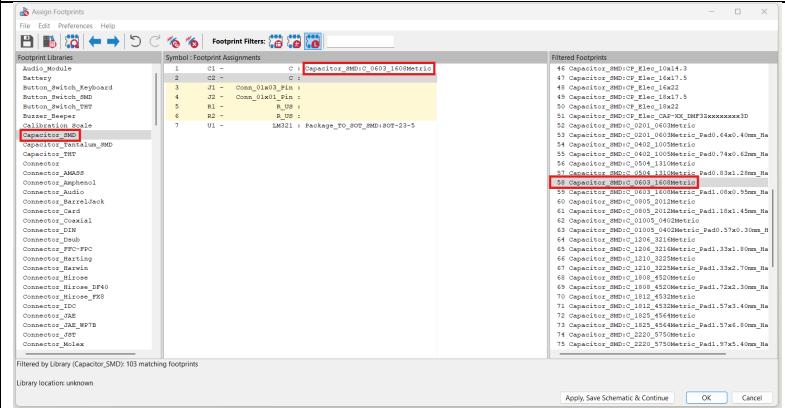




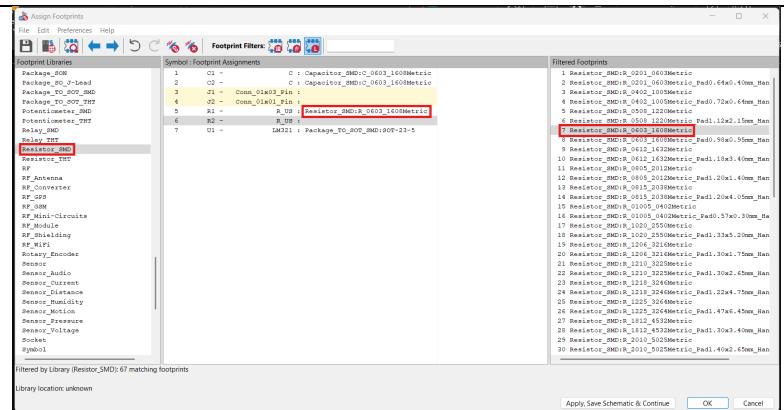
6. Click the Assign Footprints button in the toolbar



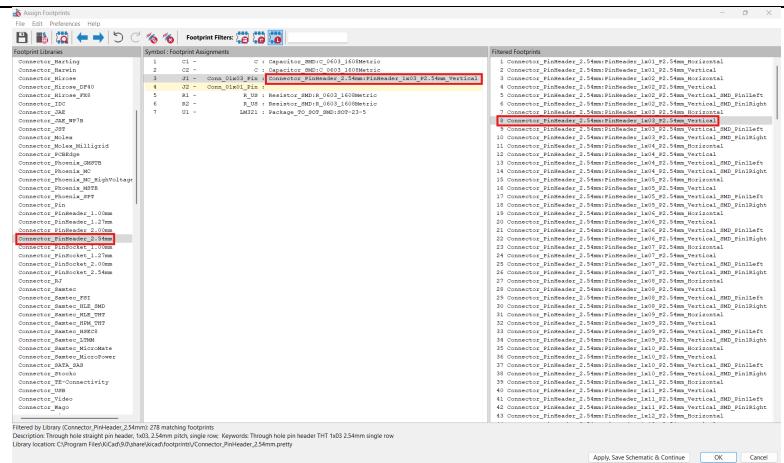
7. Click on capacitors in the middle panel (you may have to do each separately), navigate to “Capacitor_SMD” in the left panel and click it. Navigate to number 58, “C_0603_1608Metric,” in the right panel and double click it. The footprint should populate in the middle panel after the component value.



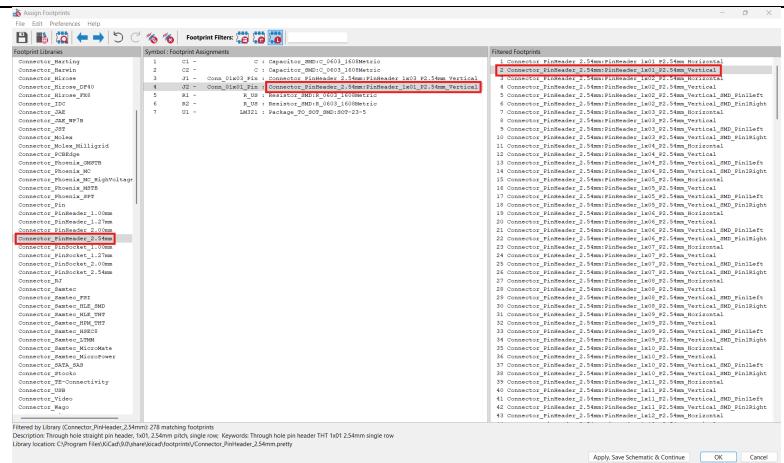
8. Click on the resistors in the middle panel (you may have to do each separately), navigate to “Resistors_SMD” in the left panel and click it. Navigate to number 7, “R_0603_1608Metric,” in the right panel and double click it. The footprint should populate in the middle panel after the component value.



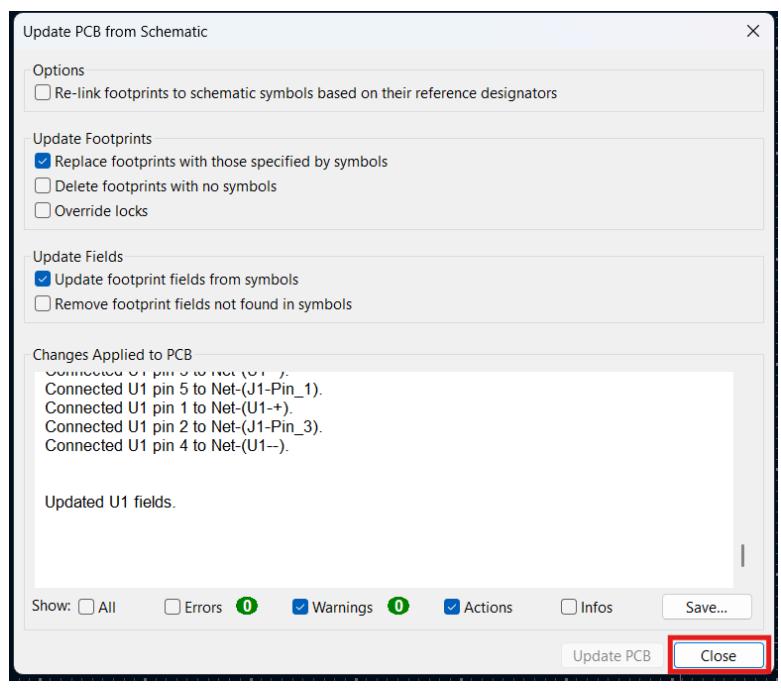
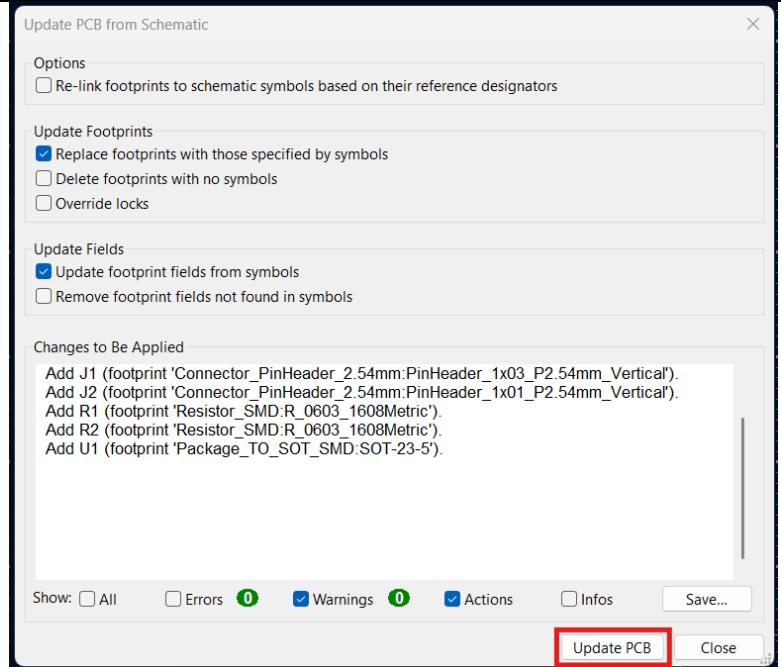
9. Click on the 1x3 connector in the middle panel, navigate to “Connector_PinHeader_2.54mm,” in the left panel and click it. Navigate to the number 8, “PinHeader_1x03_P2.54mm_Vertical,” in the right panel and double click it. The footprint should populate in the middle panel after the component name.



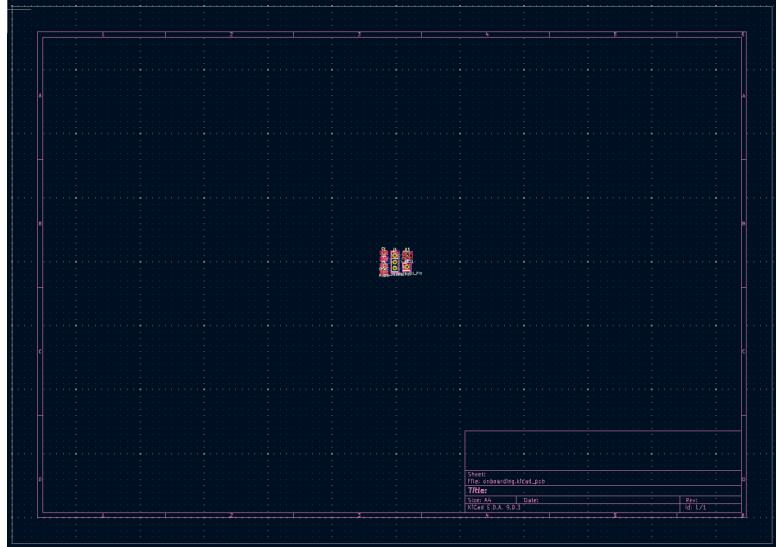
10. Click on the 1x3 connector in the middle panel, navigate to “Connector_PinHeader_2.54mm,” in the left panel and click it. Navigate to the number 8, “PinHeader_1x01_P2.54mm_Vertical,” in the right panel and double click it. The footprint should populate in the middle panel after the component name.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



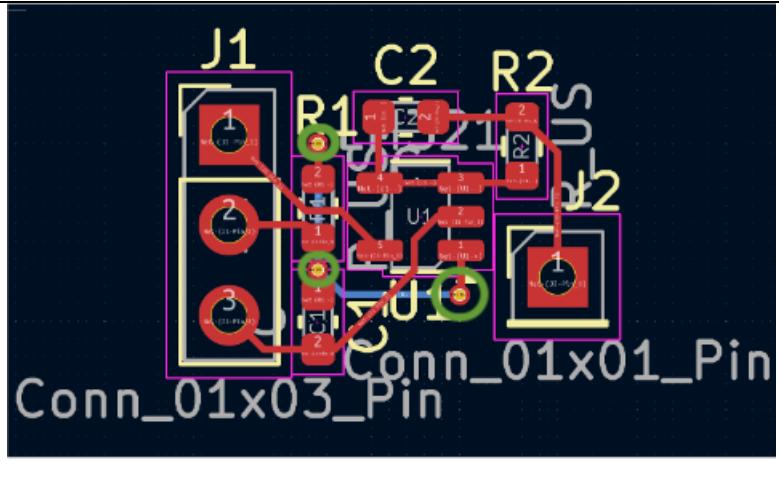
11. Press F8 to create your PCB and click “Update PCB” then click “Close”



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

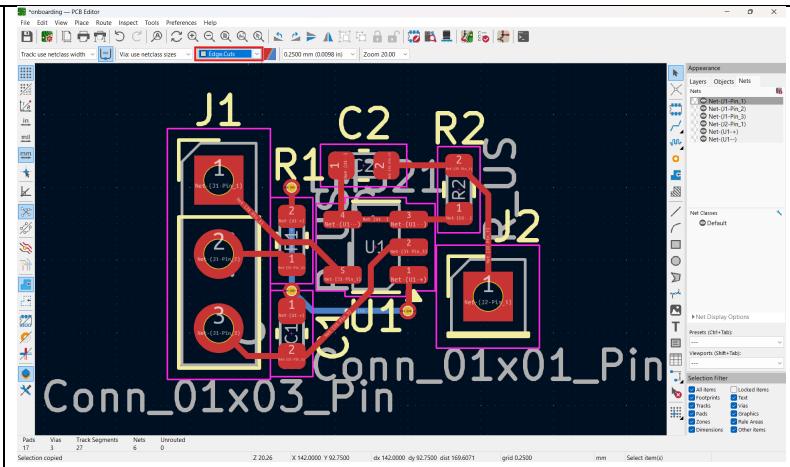


13. Arrange the components in the compact way possible using the following controls. One arrangement example is depicted in the picture on the right

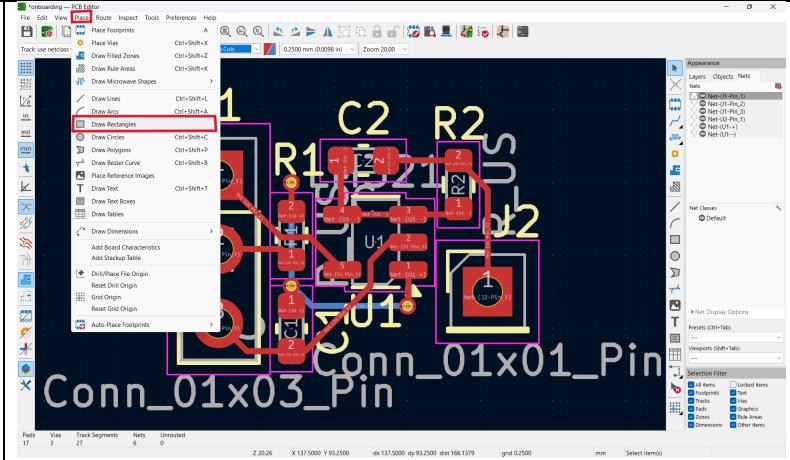


- “x” will create traces (wires) between components
- “v” will create a via which switches the side the trace is on, allowing them to cross
 - The vias are circled in green in the above image
- “r” will rotate the selected component
- Right click dragging will move the screen
- “esc” will stop whatever 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

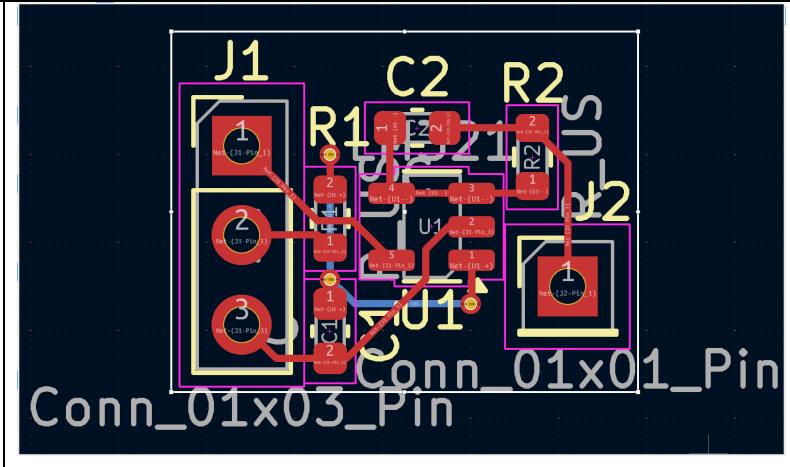
14. Select “Edge Cuts” from the layer drop down menu



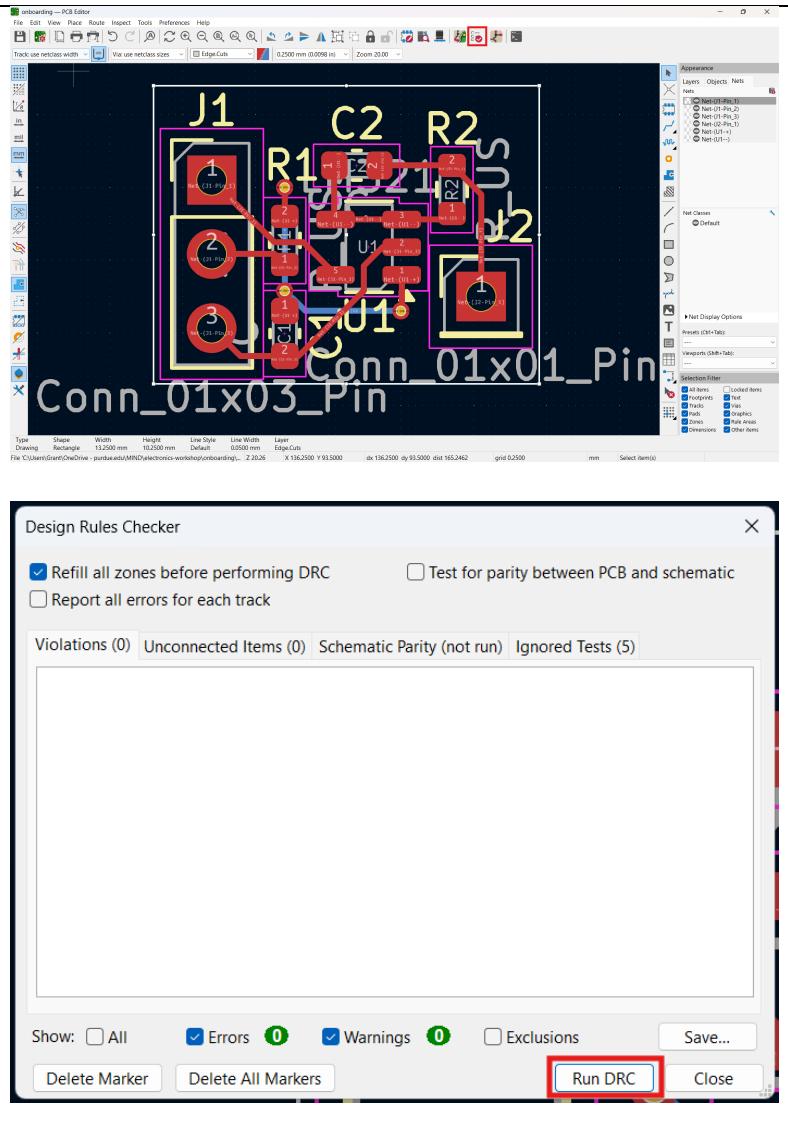
15. Go to “Place->Draw Rectangle”



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

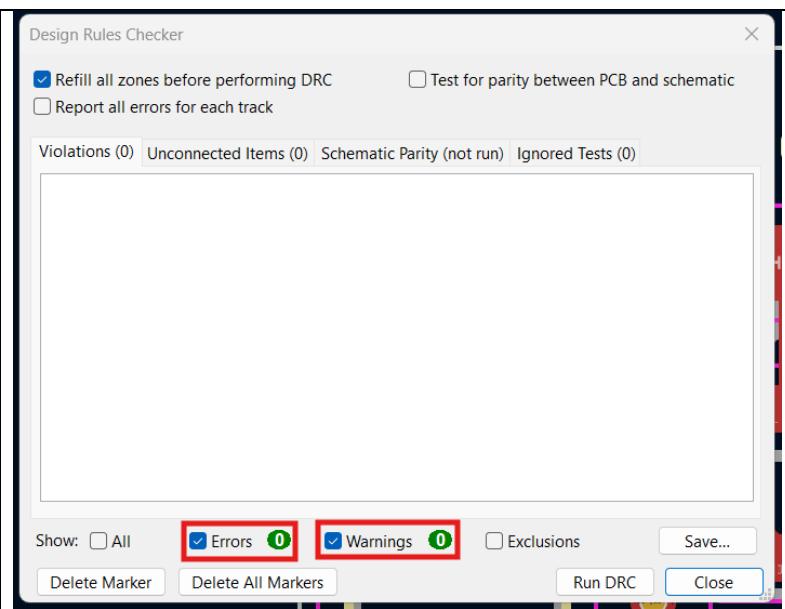


17. Click “Design Rules Checker” then “Run DRC”

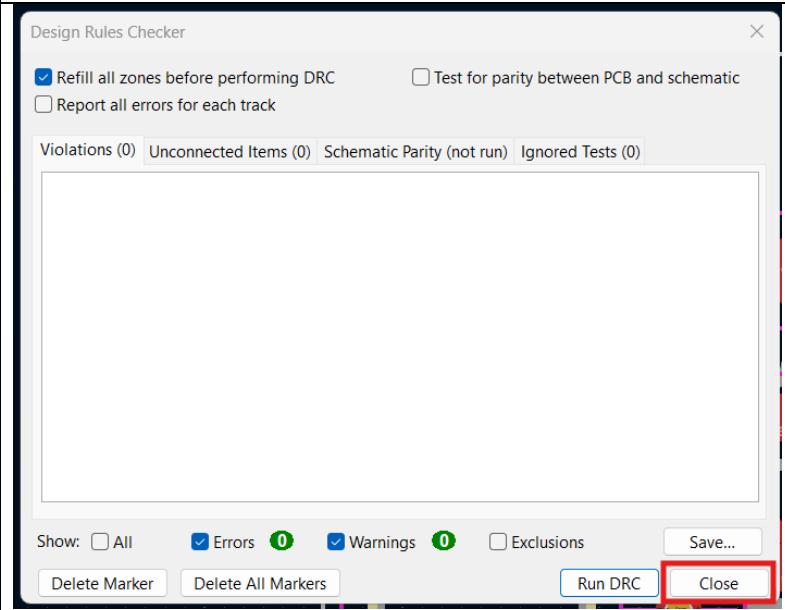


18. Verify that you get 0 errors and 0 warnings. If not, recheck your schematic and PCB layout. If it still isn't working, reference the video tutorial.

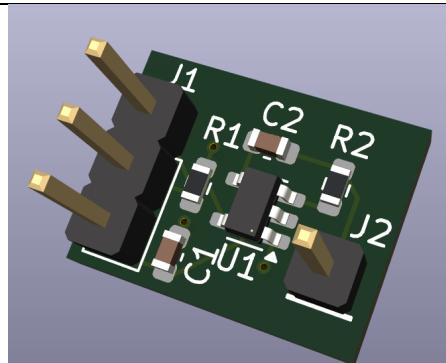
<https://youtu.be/eTiLhKjf7fY>



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

LTSPICE SHORTCUTS ON A MAC

11/5/2013 REV 3

a	DRAW CIRCLE
b	BUS TERMINATION
g	GROUND
l	DRAW LINE
s	ADD SPICE DIRECTIVE (right click for HELP ME EDIT)
t	ADD TEXT COMMENT
w	DRAW BOX
⌘ H	HIDE LTSPICE
⌘ L	SPICE LOG
⌘ N	NEW SCHEMATIC
⌘ O	OPEN
⌘ Q	QUIT LTSPICE
⌘ S	SAVE
⌘ Z	UNDO
⇧ ⌘ Z	REDO
⌘ M	MINIMIZE
⊜ ⌘ M	MINIMIZE ALL
⌘ W	CLOSE
⊜ ⌘ W	CLOSE ALL
⌘ P	PRINT
⇧ ⌘ P	page seupt
F2	COMPONENT
F3	WIRE
F4	NET NAME
F5	DELETE
F6	DUPLICATE
F7	MOVE (CNTRL-R to rotate, CNTRL-E to mirror)
F8	DRAG (CNTRL-R to rotate, CNTRL-E to mirror)
F9	UNDO
⇧ F9	REDO
SPACE BAR	ZOOM TO FIT
2 FINGER PINCH	ZOOM IN
2 FINGER SPREAD	ZOOM OUT

Here are the modifier key symbols you may see in OS X menus:

⌘	COMMAND
⊜	ALT OR OPTION
⇧	SHIFT