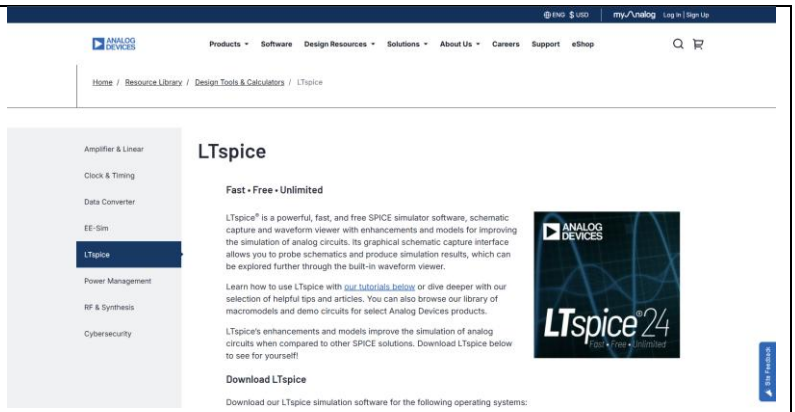
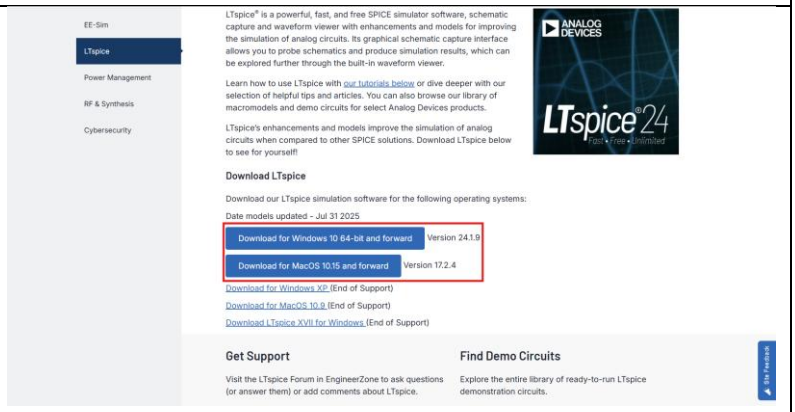
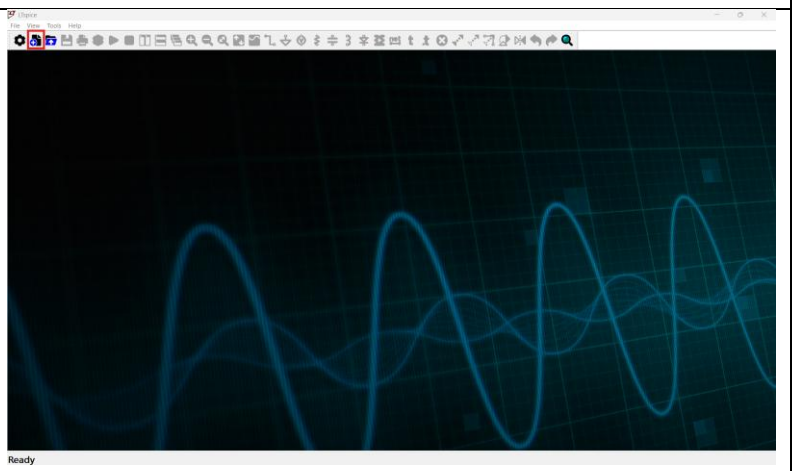
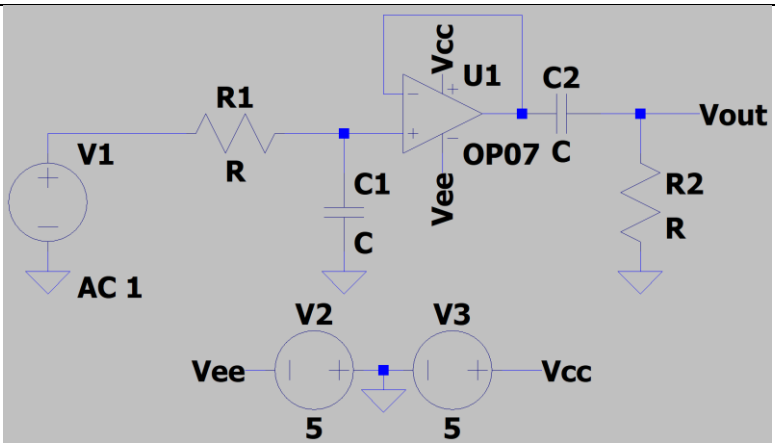
LTSpice Tutorial

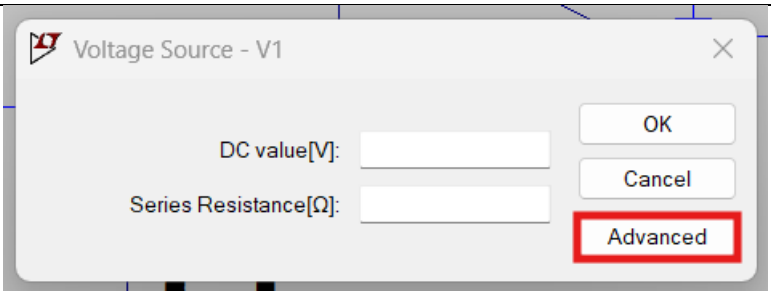
1. Navigate to https://www.analog.com/	
2. Install LTSpice software for your operating system	
3. Create a new schematic	

4. Construct the following circuit using the controls listed below

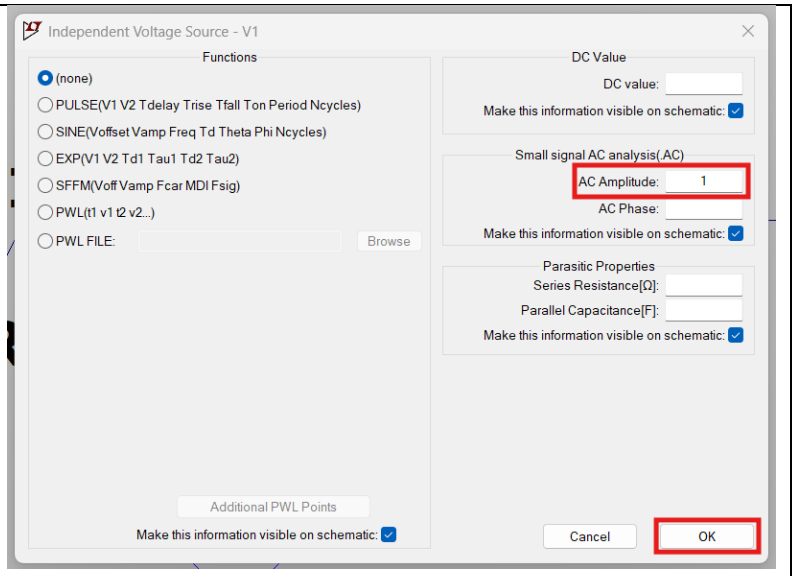


- "v" creates a voltage source
- "r" creates a resistor
- "c" creates a capacitor
- "g" creates a ground reference point
- "n" creates a net which can connect parts of your circuit with labels rather than wires
- "w" creates a wire
- "Ctrl+r" will rotate a selected component
- "esc" will stop whatever operation you are preforming
- "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

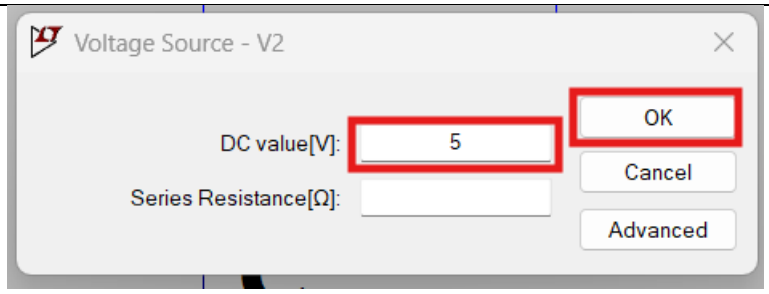
5. Right click on the voltage source V1 (as depicted in the schematic in step 4) and click Advanced



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

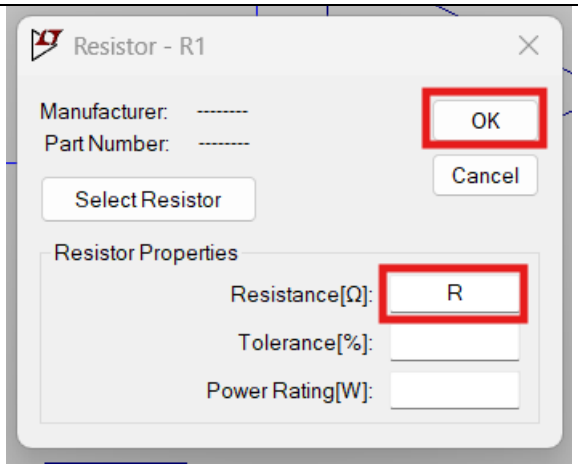


8. The cutoff frequency for a high and low pass filter is defined in the following equation. Use it to determine appropriate capacitor and resistor values for the 5kHz cutoff

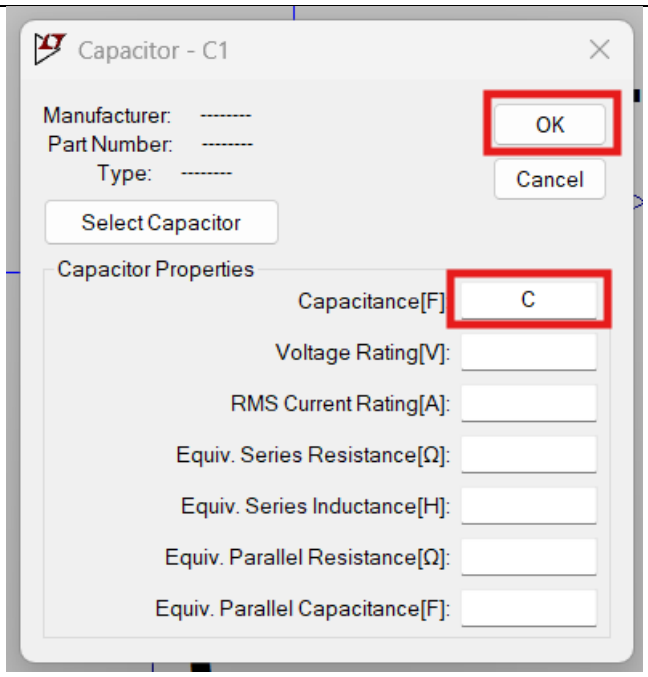
$$f_c = \frac{1}{2\pi RC}$$

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



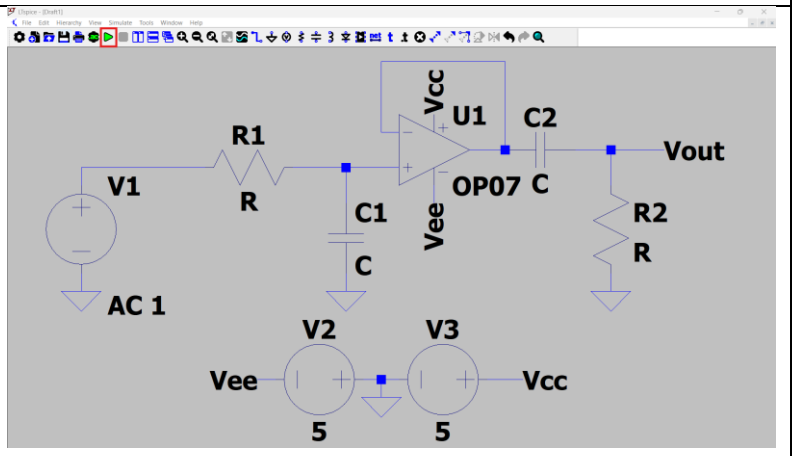
- Mega-: “Meg” (10^{6})
- Kilo-: “k” (10^{3})
- Milli-: “m” (10^{-3})
- Micro-: “u” (10^{-6})
- Nano-: “n” (10^{-9})
- Femto-: “f” (10^{-12})



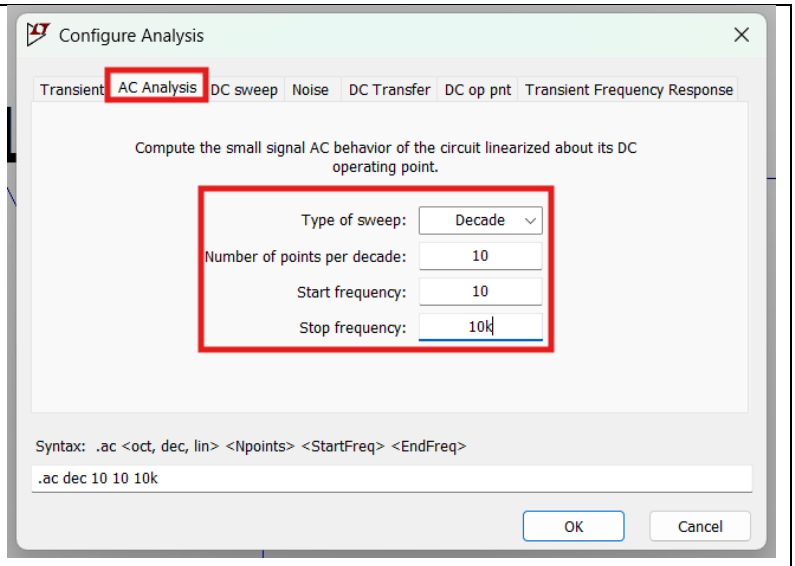
10. Use the equation from step (7) to determine appropriate capacitor and resistor values for a 250Hz cutoff

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

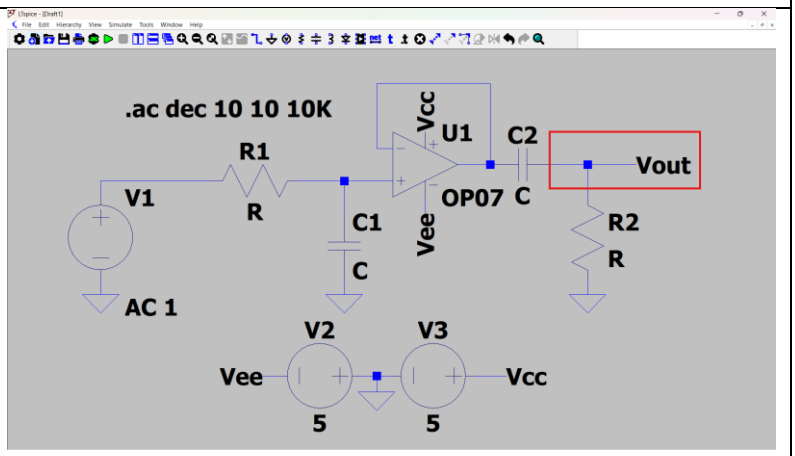
12. Click the simulate button in the toolbar

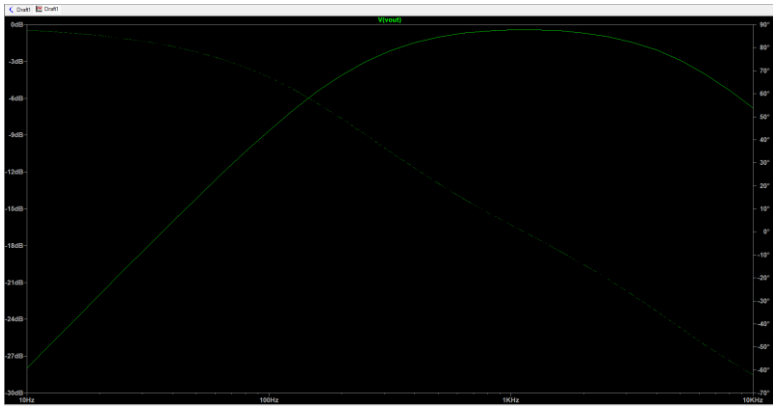


13. Click on “AC Analysis” and adjust the simulation parameters to match the ones depicted in the image on the right.

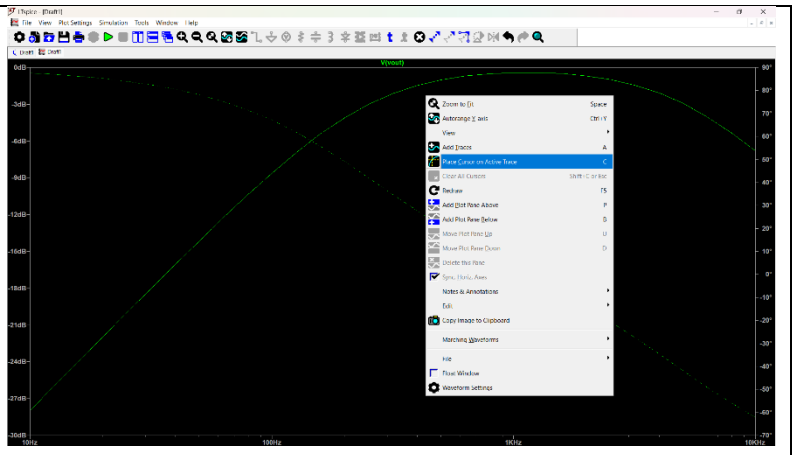


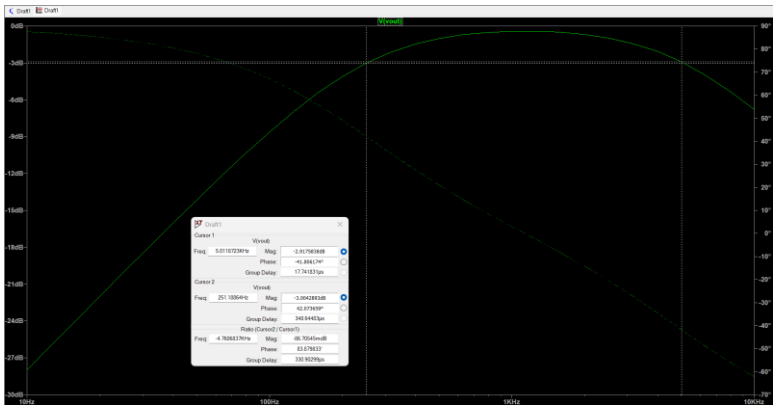
14. After clicking <u>Ok</u> you should get the following graph. You may need to click on the “Vout” net in the schematic.





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

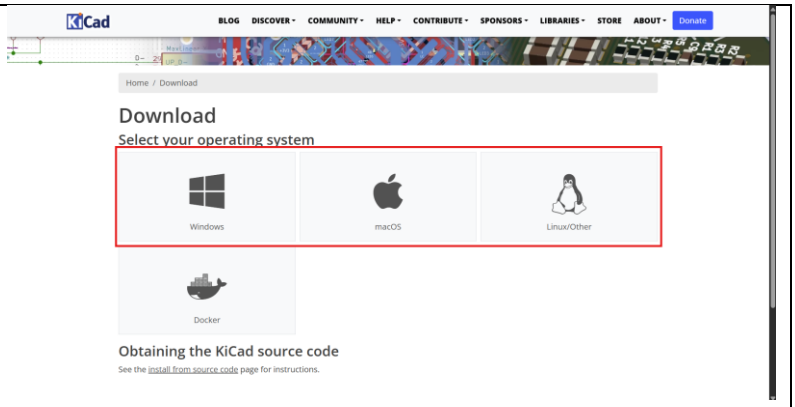
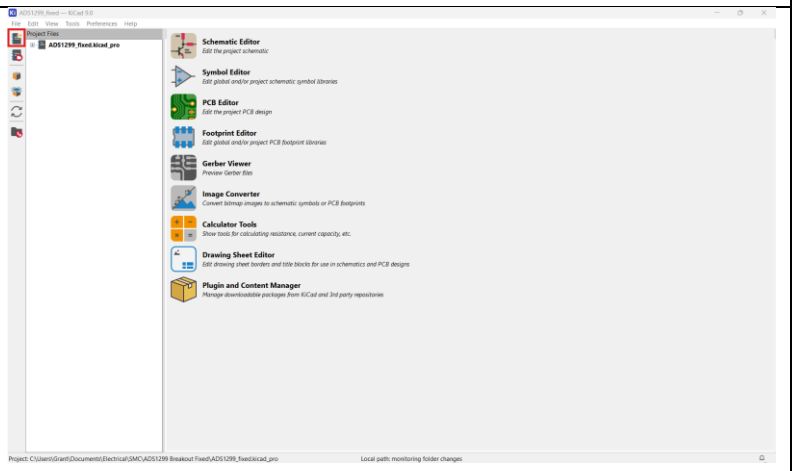
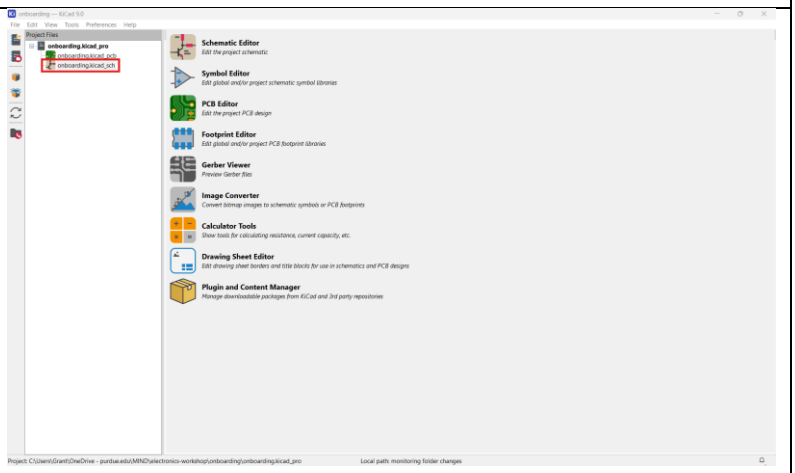




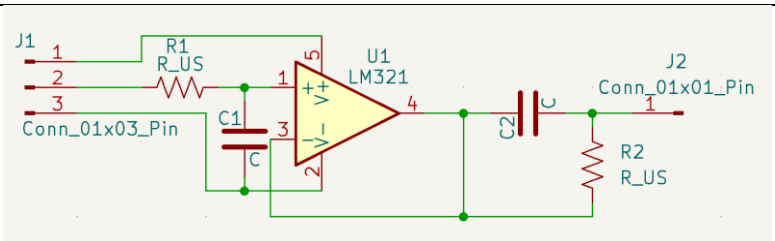
16. Ensure that the graph shows a -3db attenuation at 250Hz and 5kHz. If this is not the case, recheck your calculations for the resistor and capacitor values. If you still can't figure out the issue, reference the LTSpice video tutorial.

LTSpice Tutorial Video

KiCad Tutorial

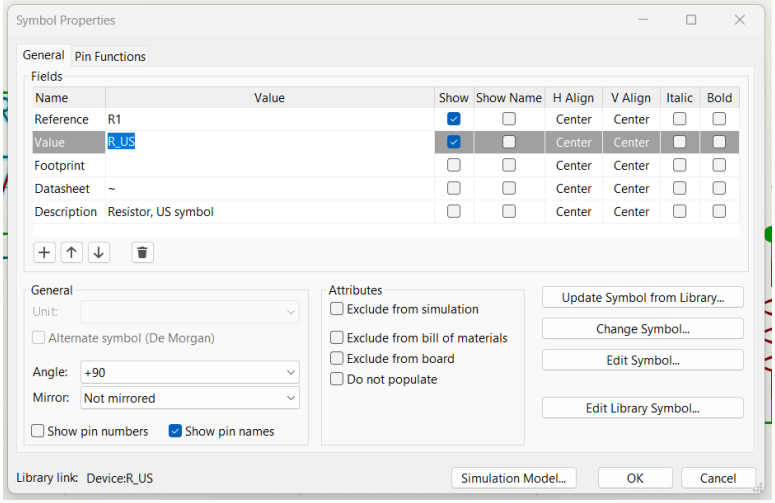
1. Navigate to https://www.kicad.org/ and install the KiCad software for your operating system keeping all the installation defaults	
2. Start a new project	
3. Double click on the schematic file	

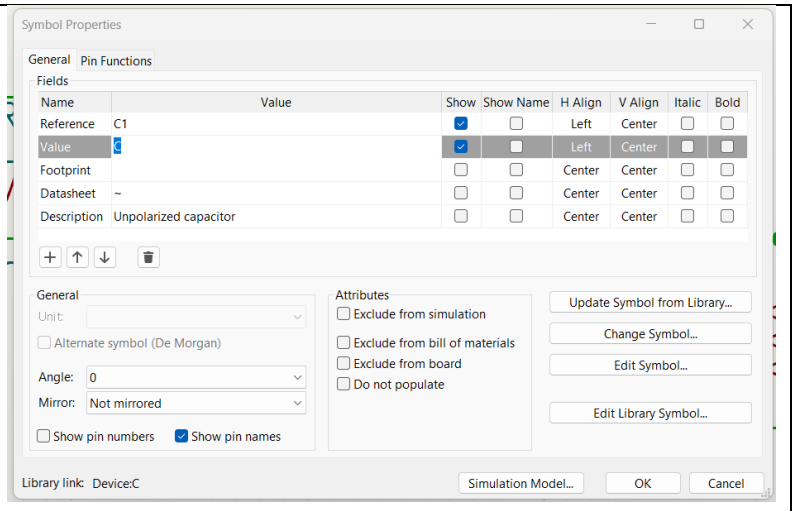
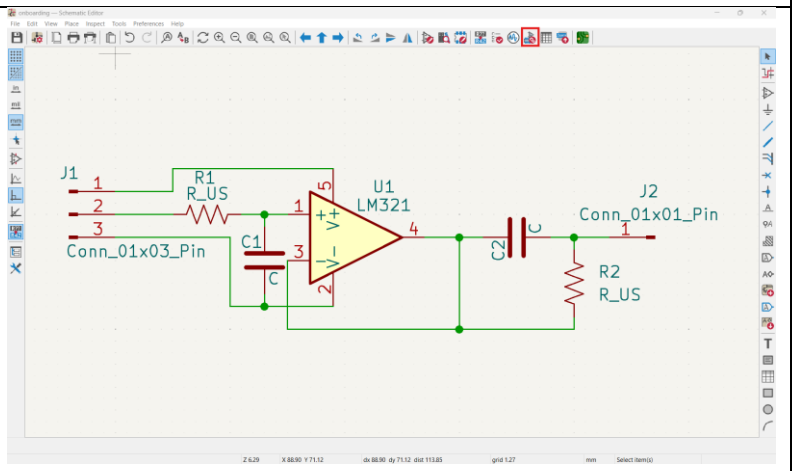
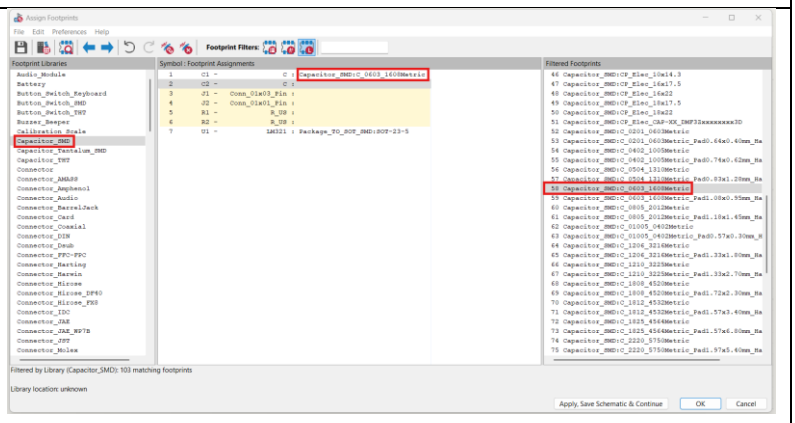
4. Construct the following circuit using the controls listed below

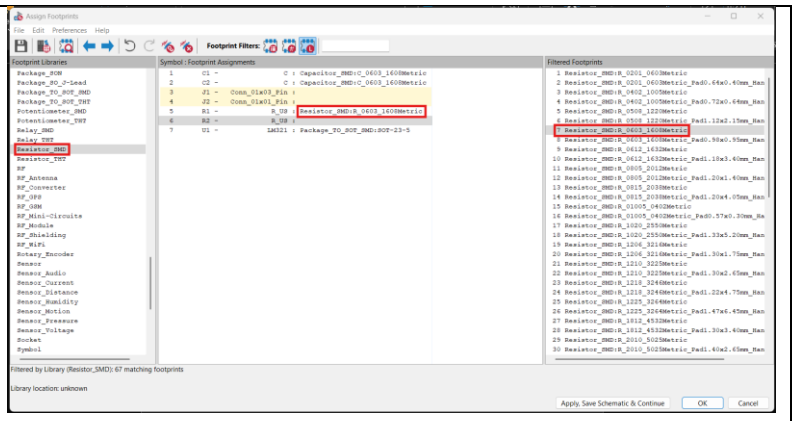
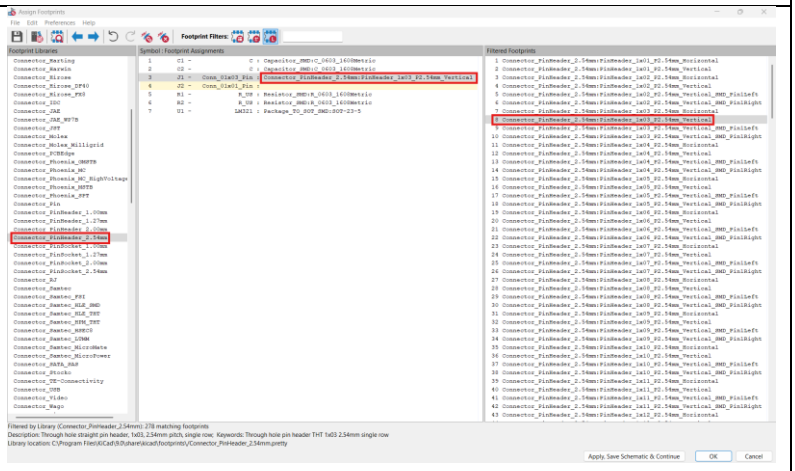
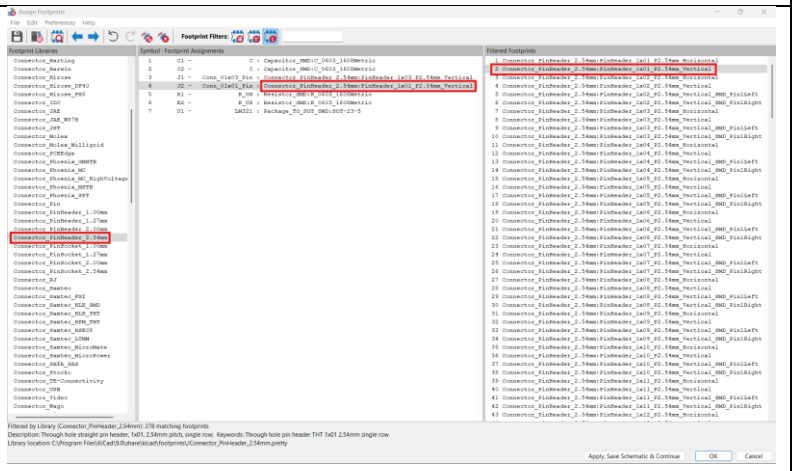


- "w" will create a wire, the terminals of the components can also be clicked to connect components
- "r" will rotate the selected component
- "Ctrl+d" will duplicate a selected component
- Right click dragging will move the screen
- "esc" will stop whatever operation you are preforming
- "a" will open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

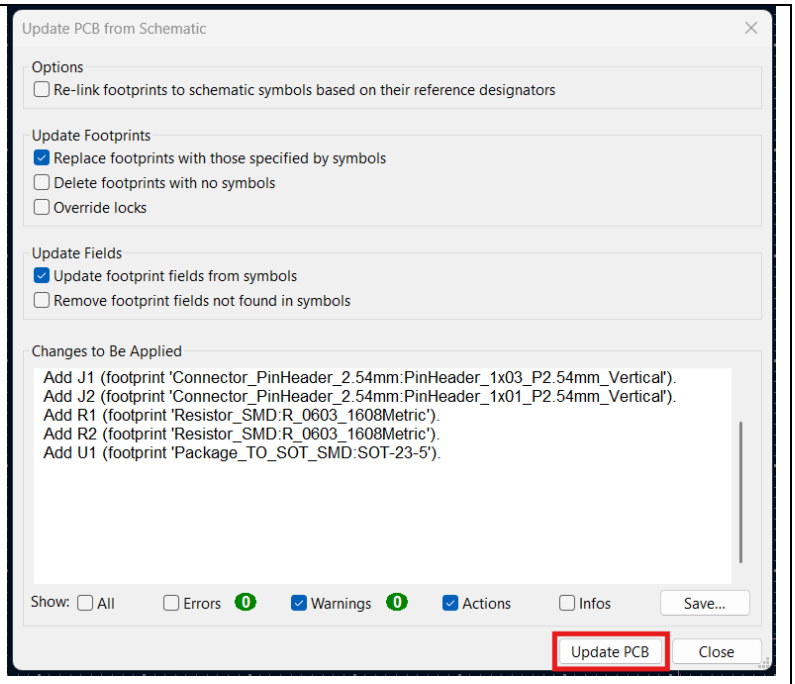
5. Enter the resistor and capacitor values you found in the LTSpice tutorial, into the KiCad schematic by double clicking on the components

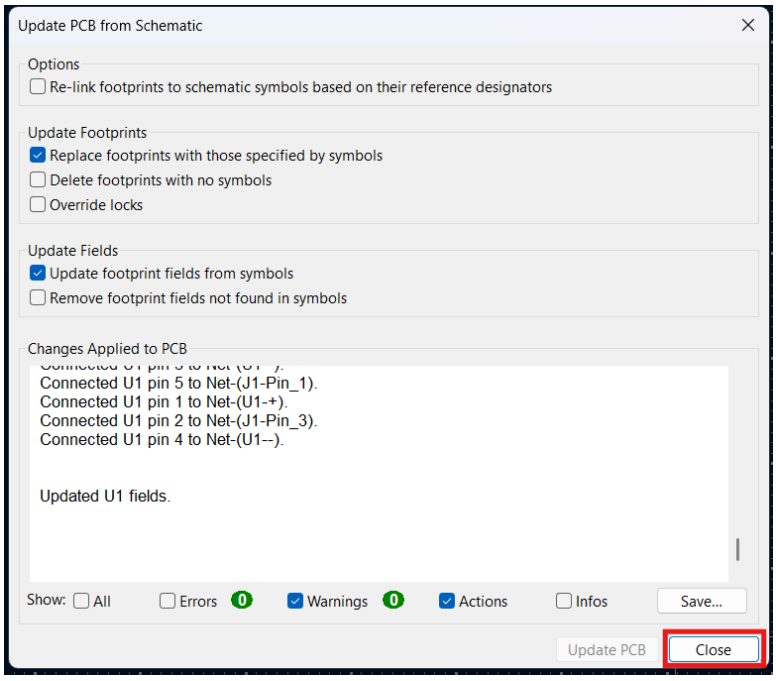


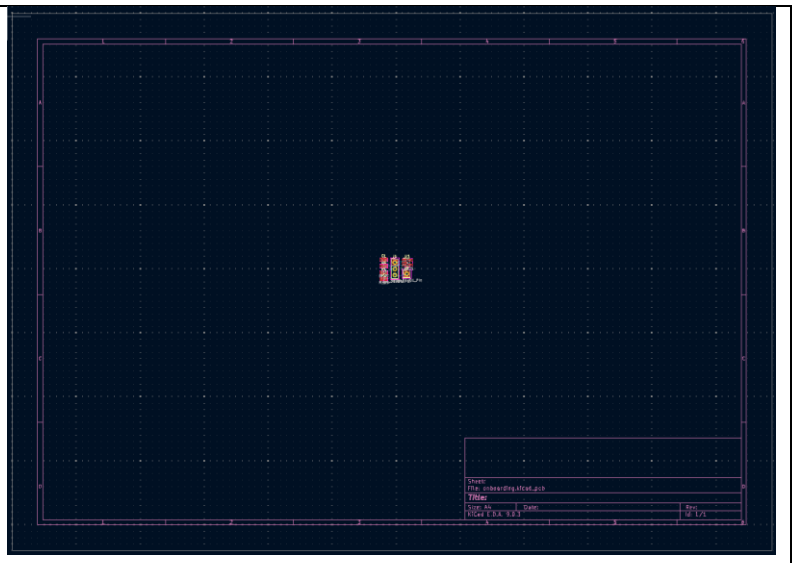
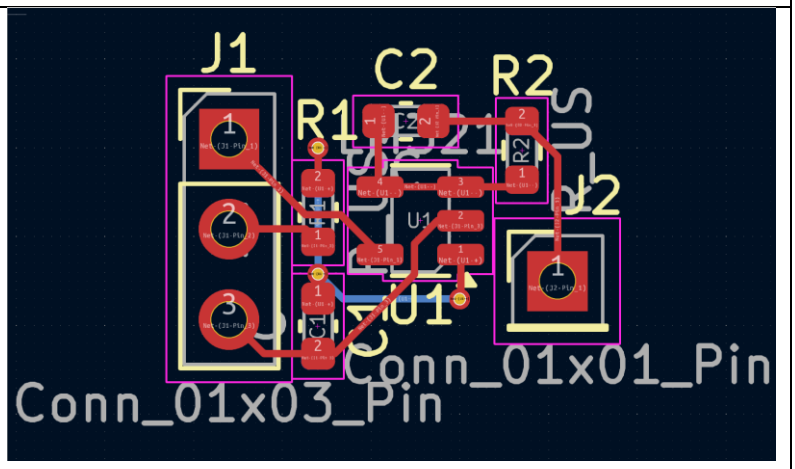
	
6. Click the <u>Assign Footprints</u> button in the toolbar	
7. Click on capacitors in the middle panel (you may have to do each separately), navigate to “Capacitor_SMD” in the left panel and click it. Navigate to number 58, “C_0603_1608Metric,” in the right panel and double click it. The footprint should 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.	

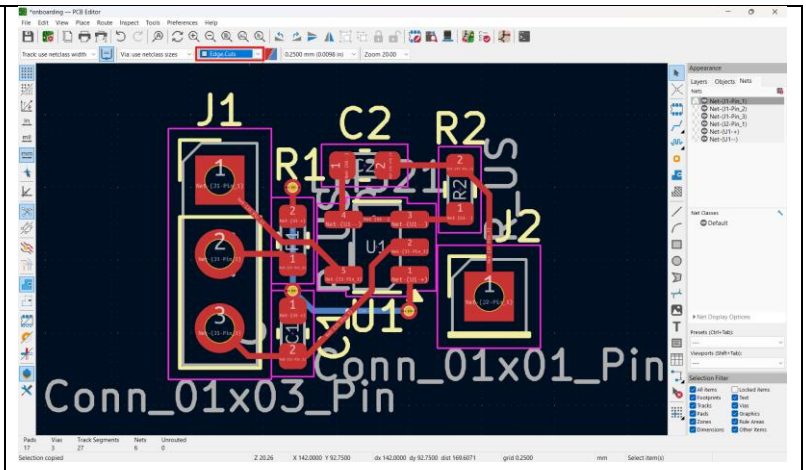
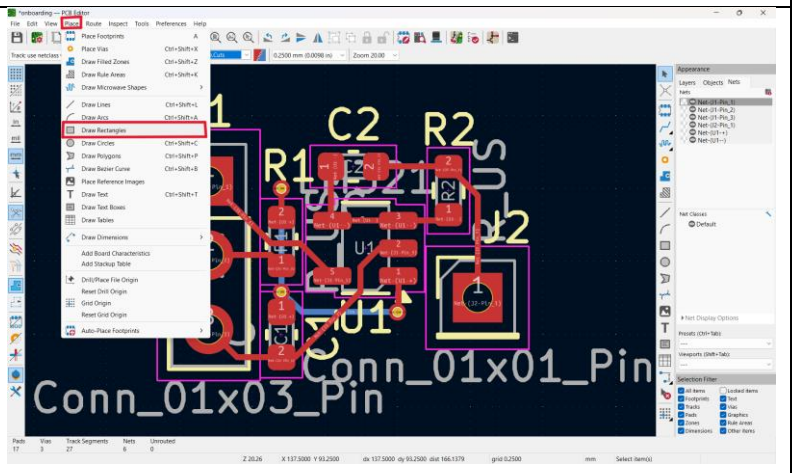
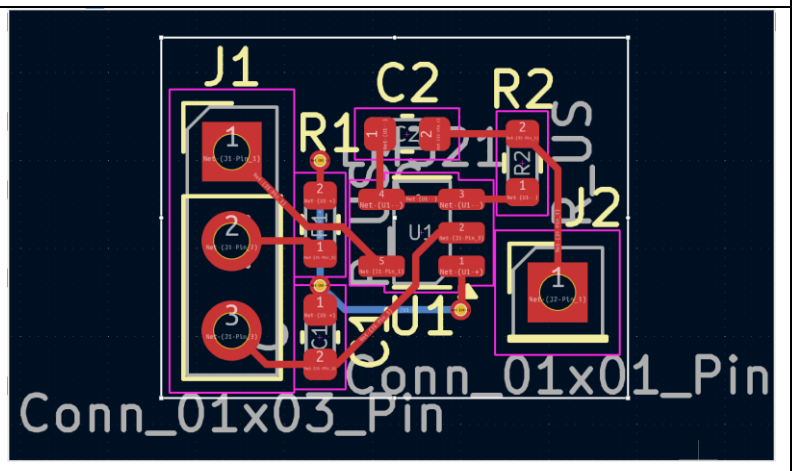
11. Press *F8* to create your PCB and click “Update PCB” then click “Close”



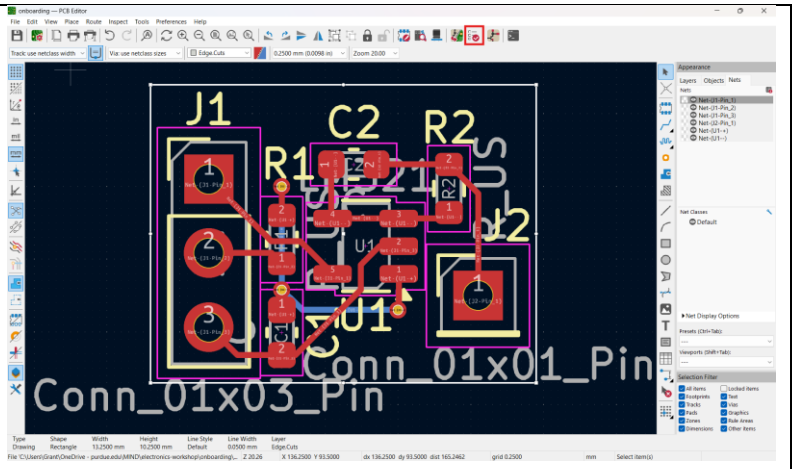


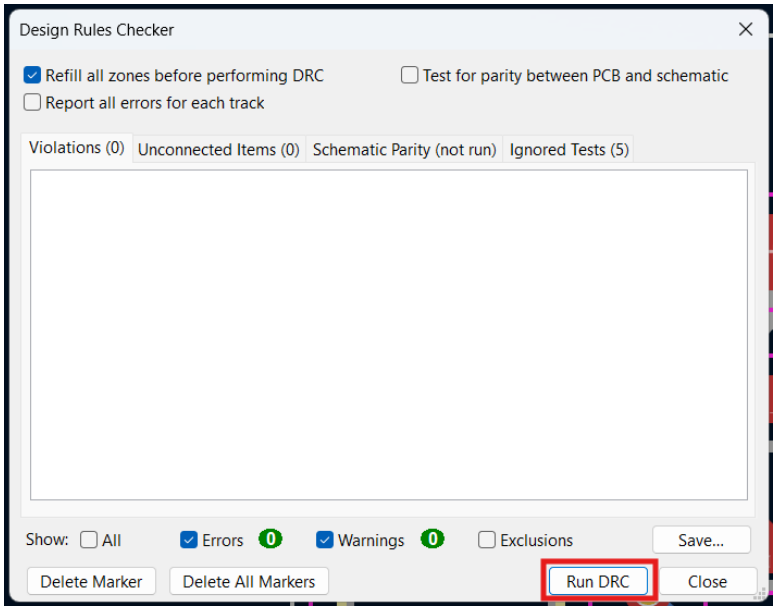
12. Left click to place the group of components in the middle of the screen.	
13. Arrange the components in the compact way possible using the following controls. One arrangement example is depicted in the picture on the right	

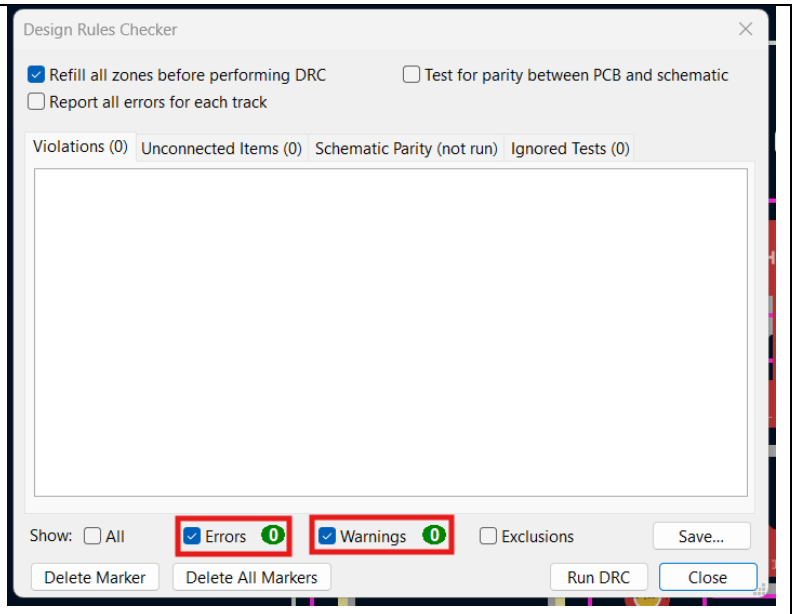
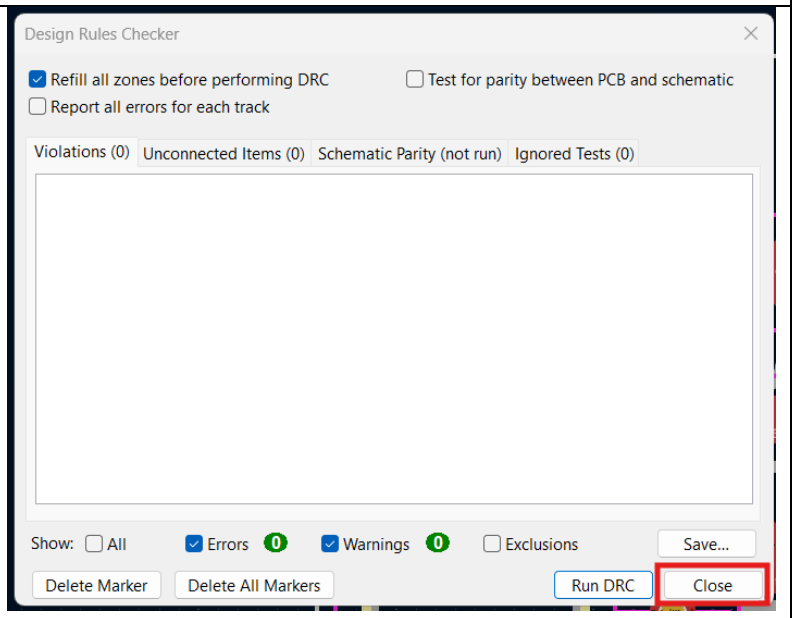
- “x” will create traces (wires) between components
- “v” will create a via which switches the side the trace is on, allowing them to cross
- “r” will rotate the selected component
- Right click dragging will move the screen
- “esc” will stop whatever operation you are preforming
- Clicking on a /trace 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