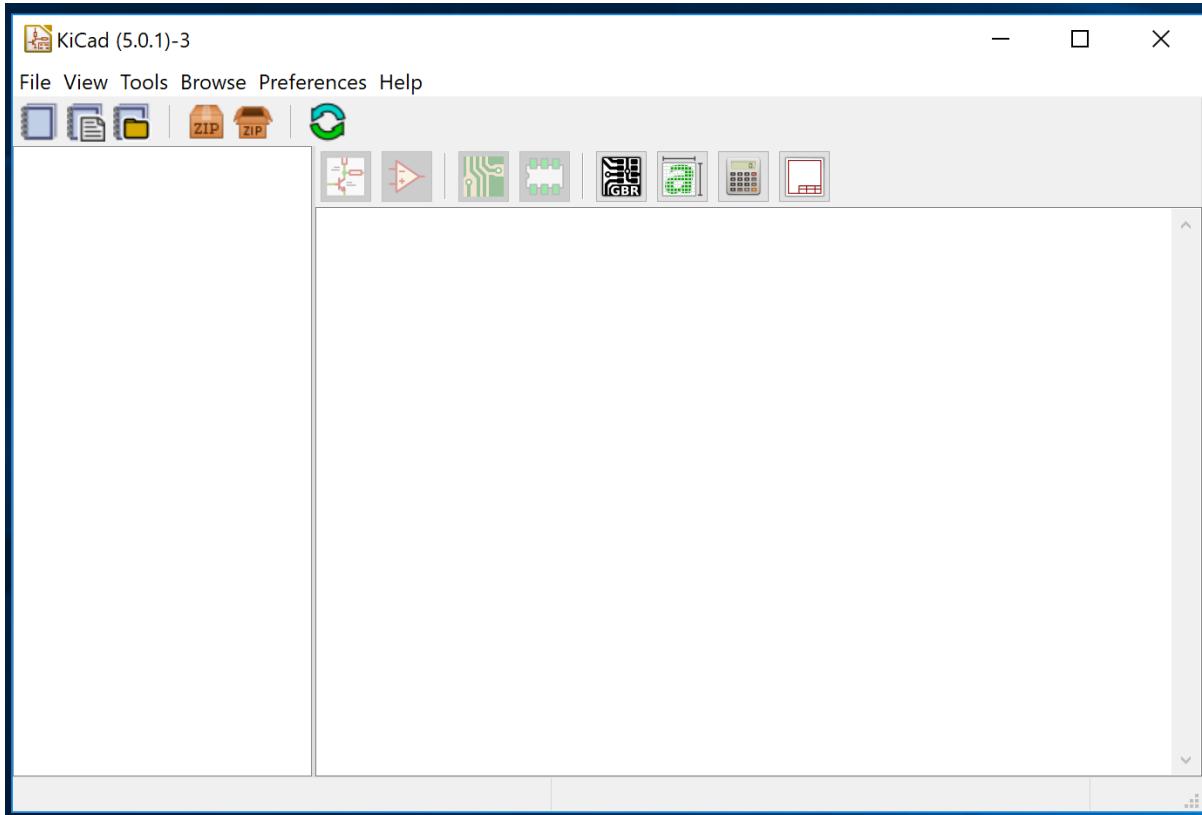
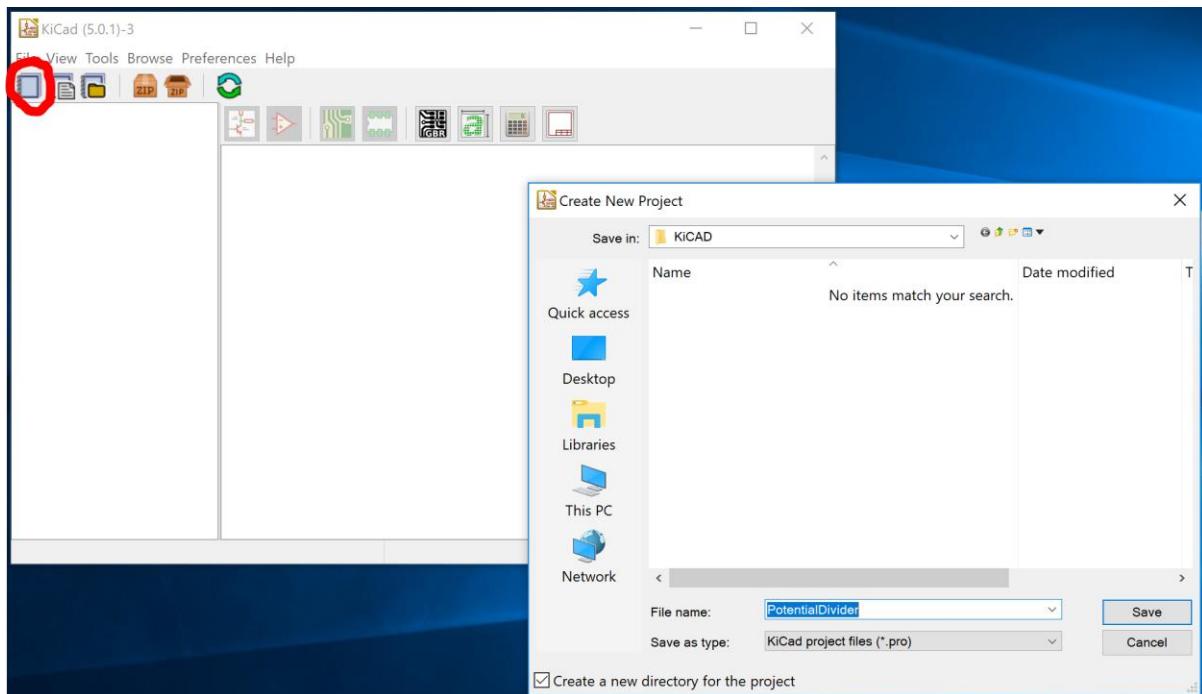


# KiCAD PCB Exercise

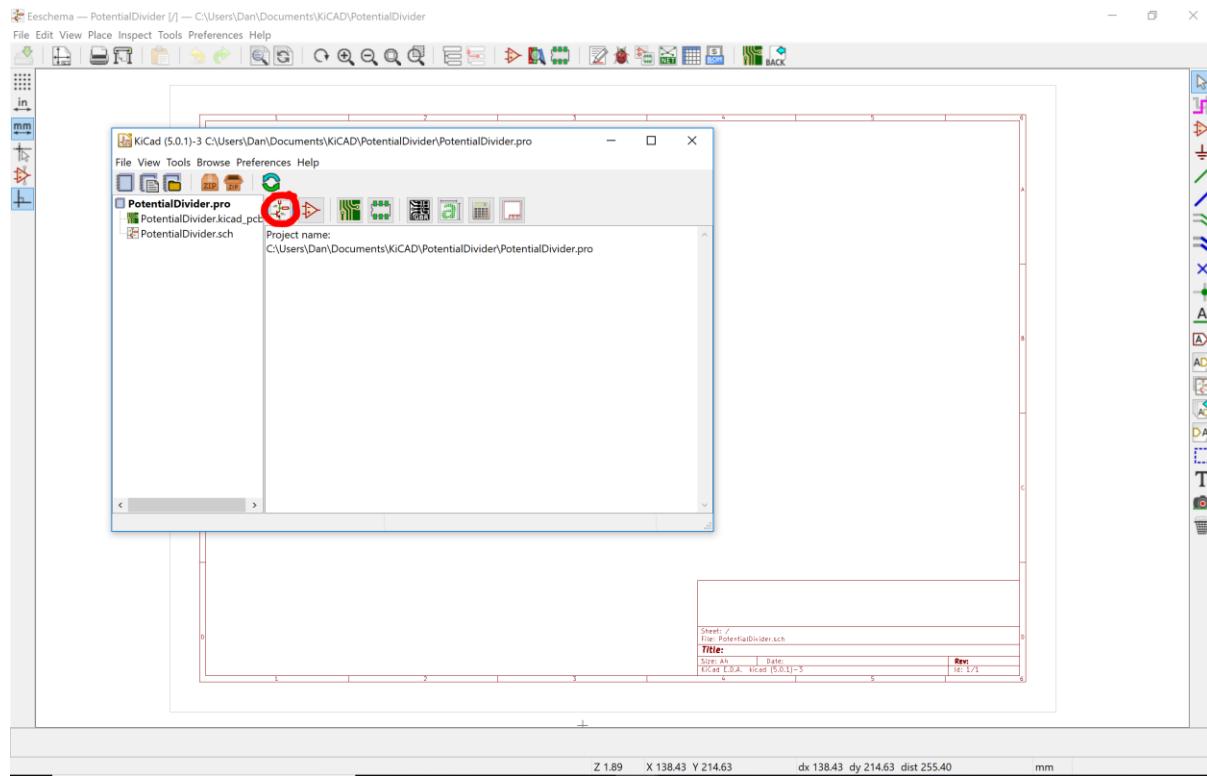
## 1. Launch KiCAD



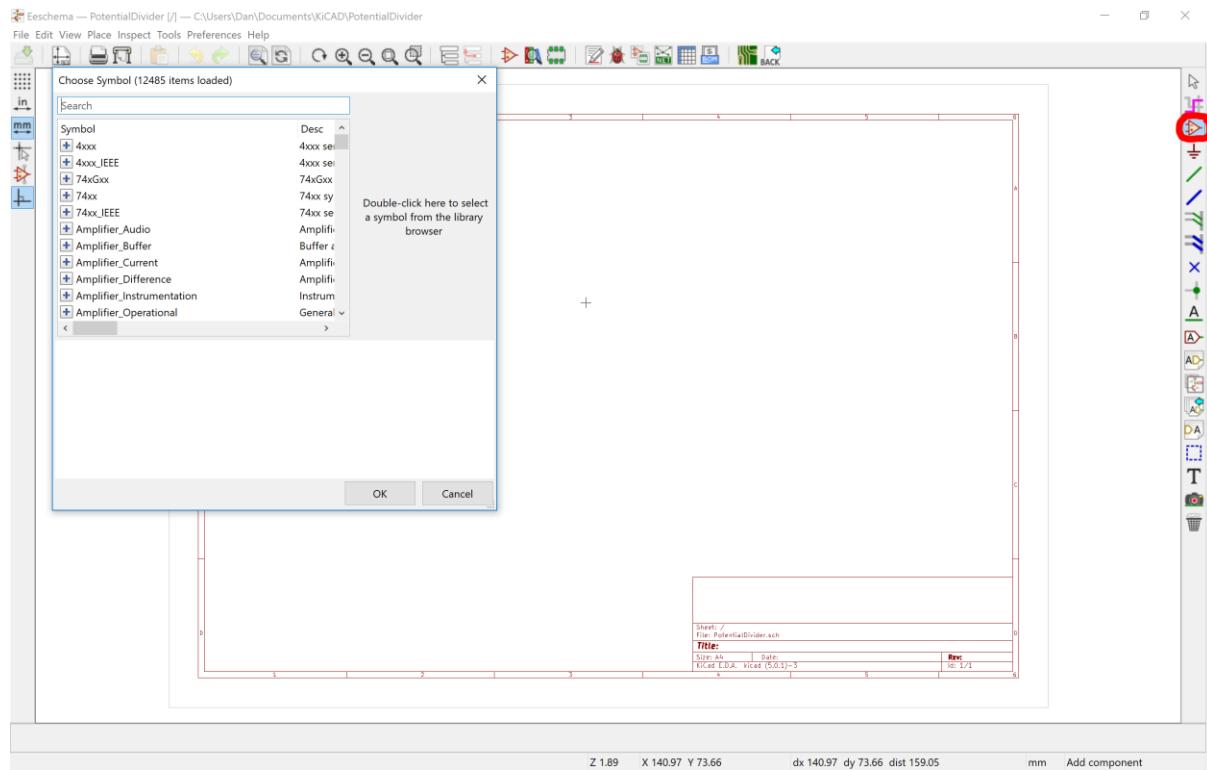
## 2. Create a new project



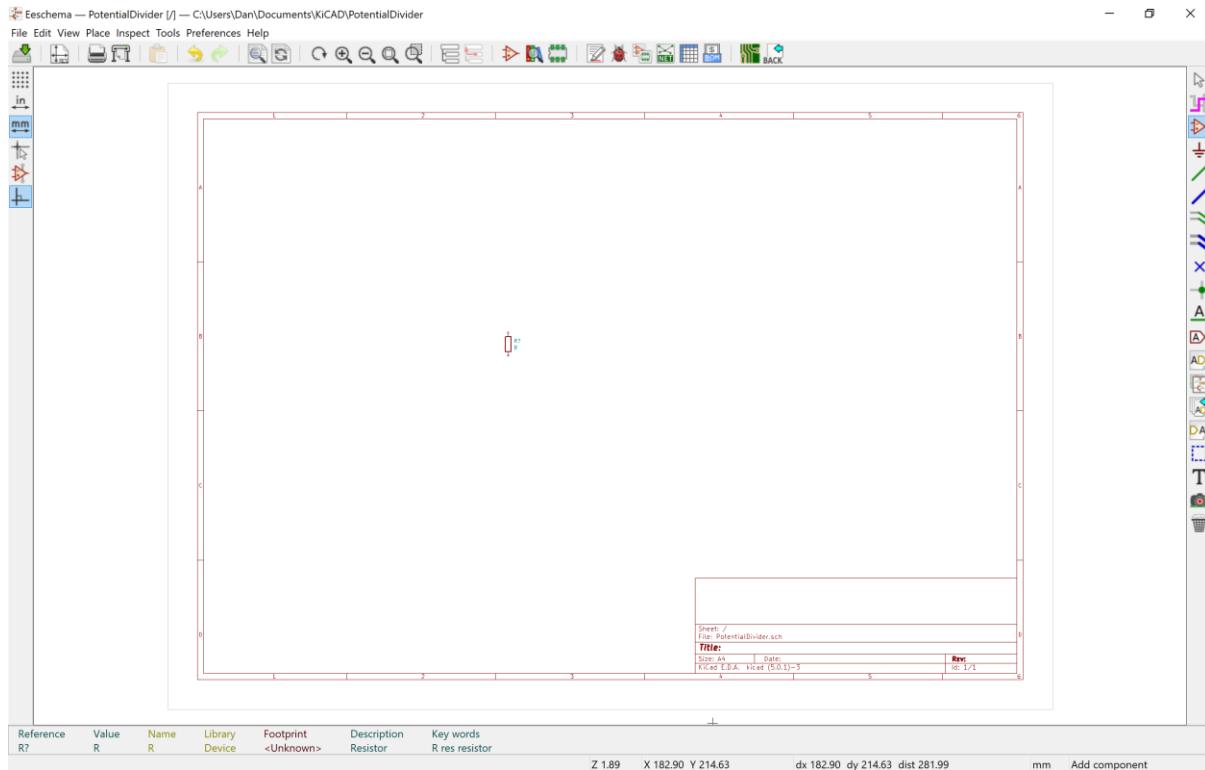
### 3. Open the schematic editor



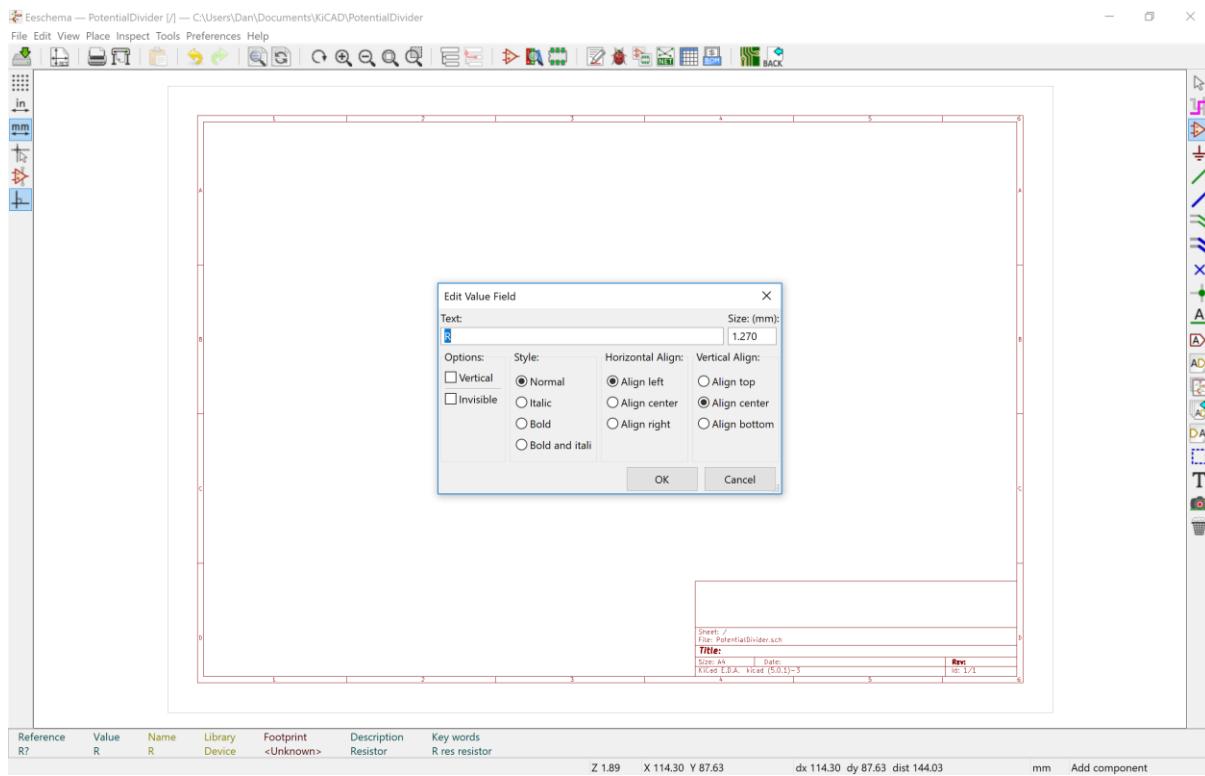
### 4. Click the component place tool, then click anywhere on the schematic to open the browser



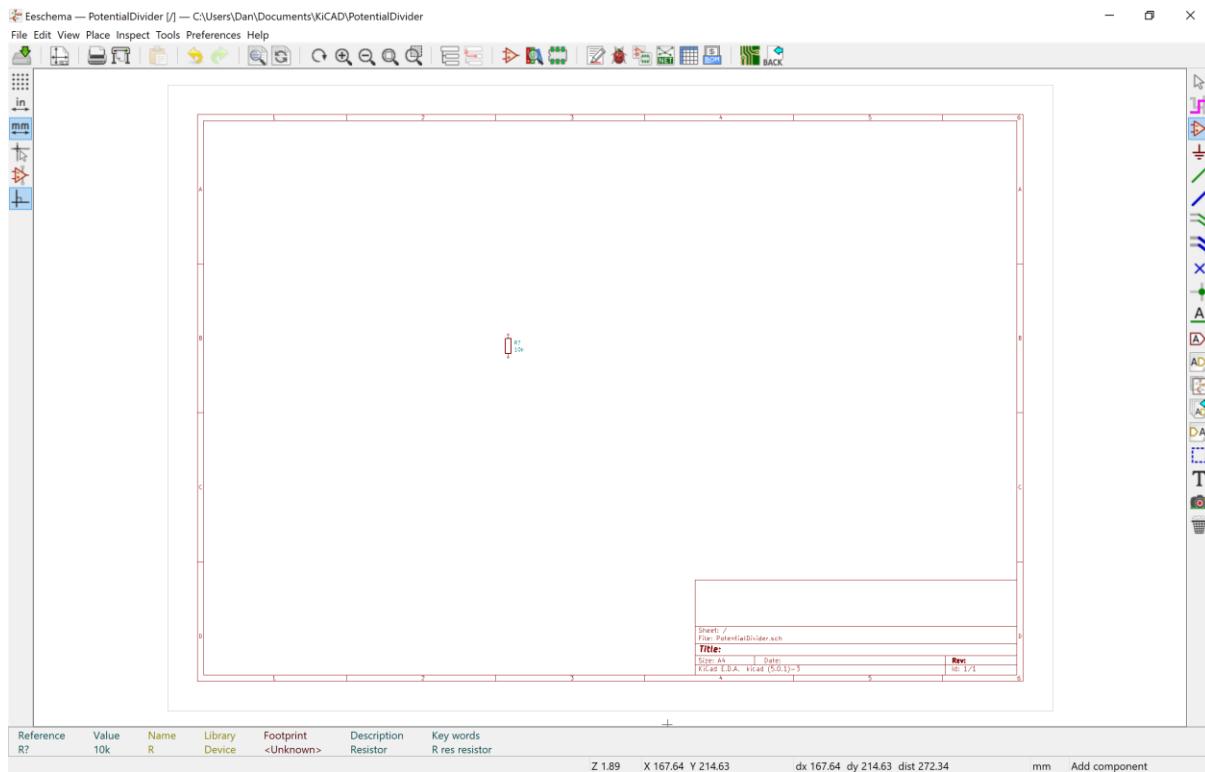
5. Search R to find the resistor symbol, click OK and place one in the schematic



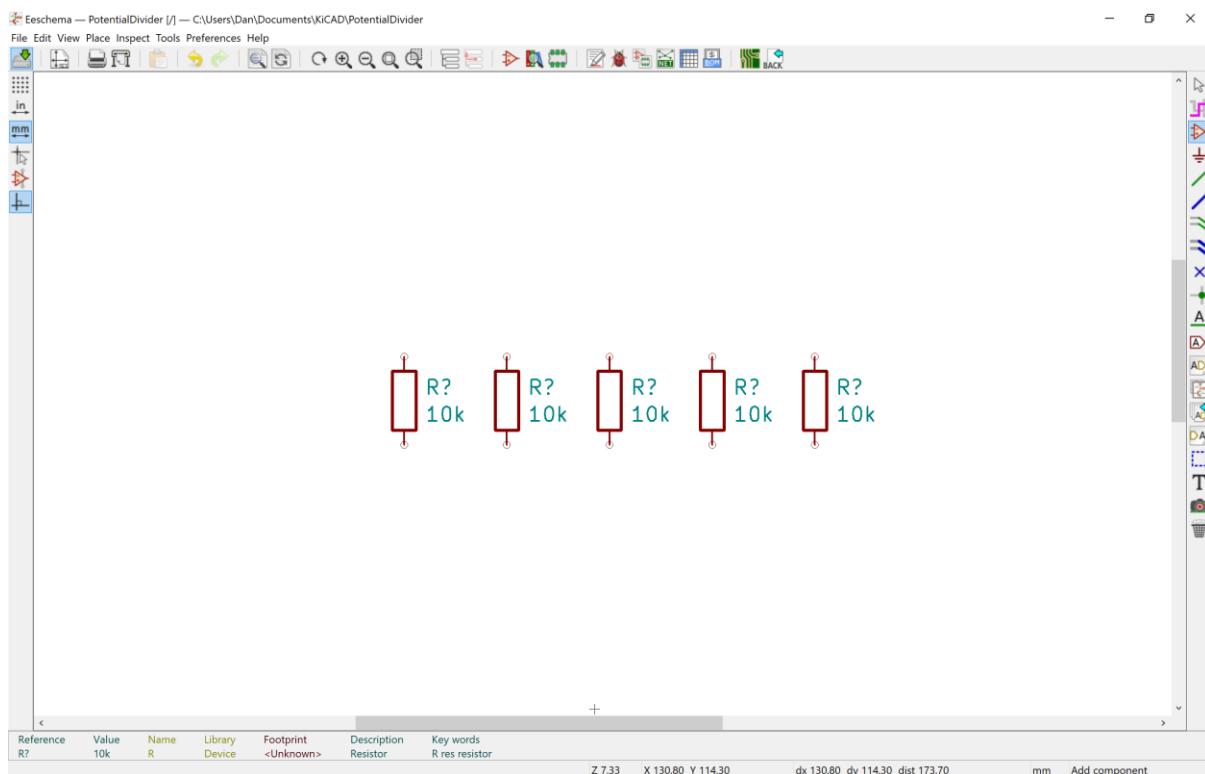
6. Use the mouse wheel to zoom in, hover over the resistor and press 'v' to set its value



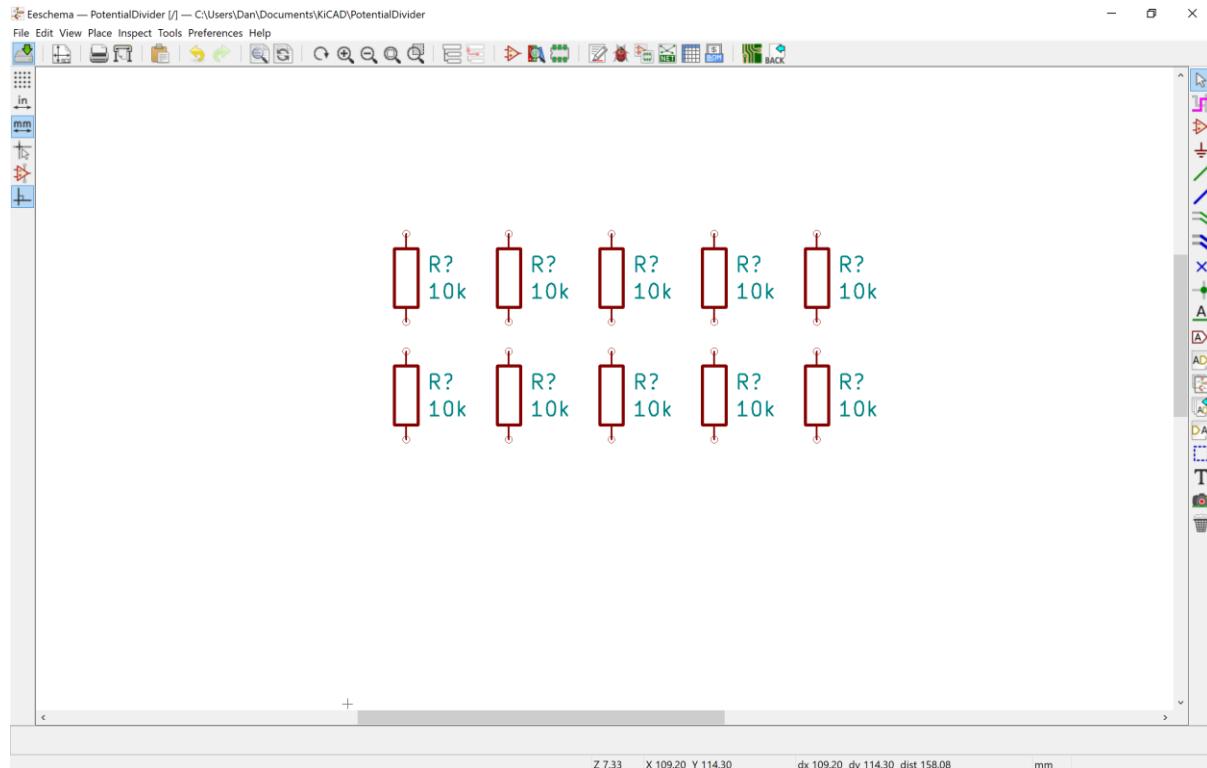
## 7. Set the value to '10k' then click OK (or press enter)



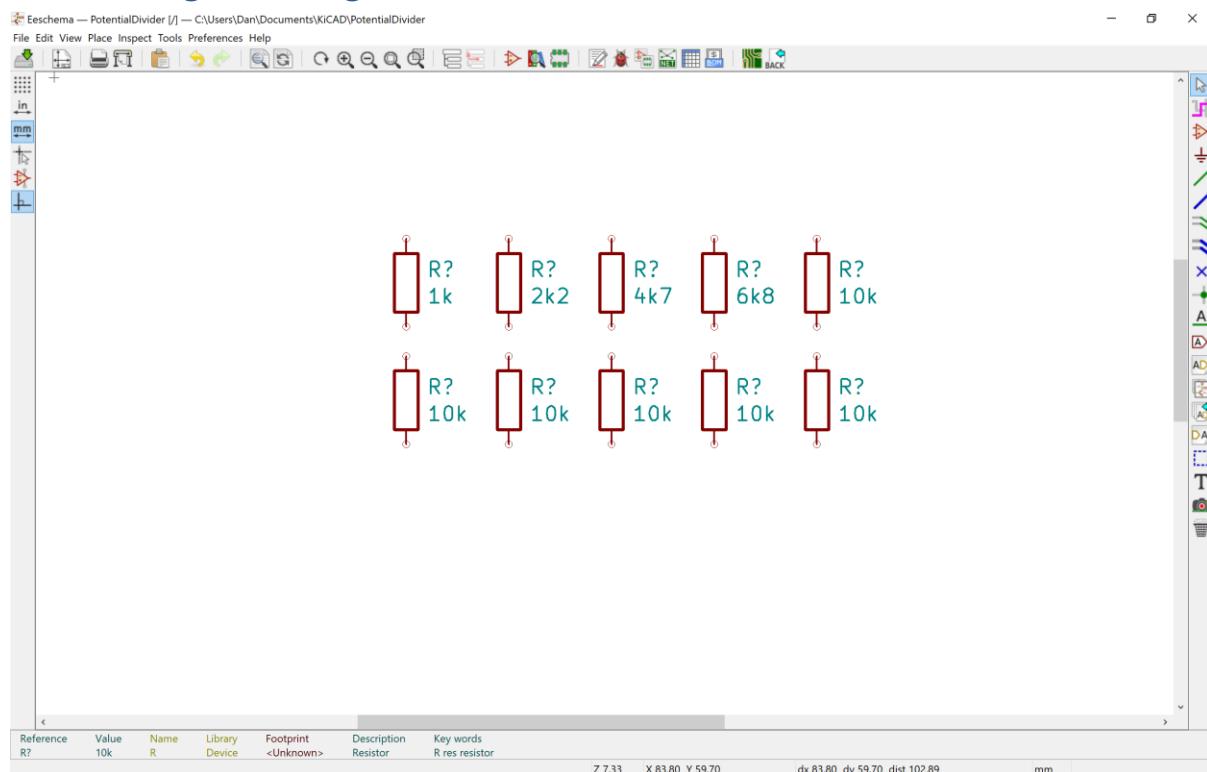
## 8. Hover over the resistor and press 'c' to copy it, move the copy into position and click the left mouse button to place. Repeat for until you have 5 resistors. If mis-placed, hover over and press 'm' to move the resistor or 'del' to delete the resistor.



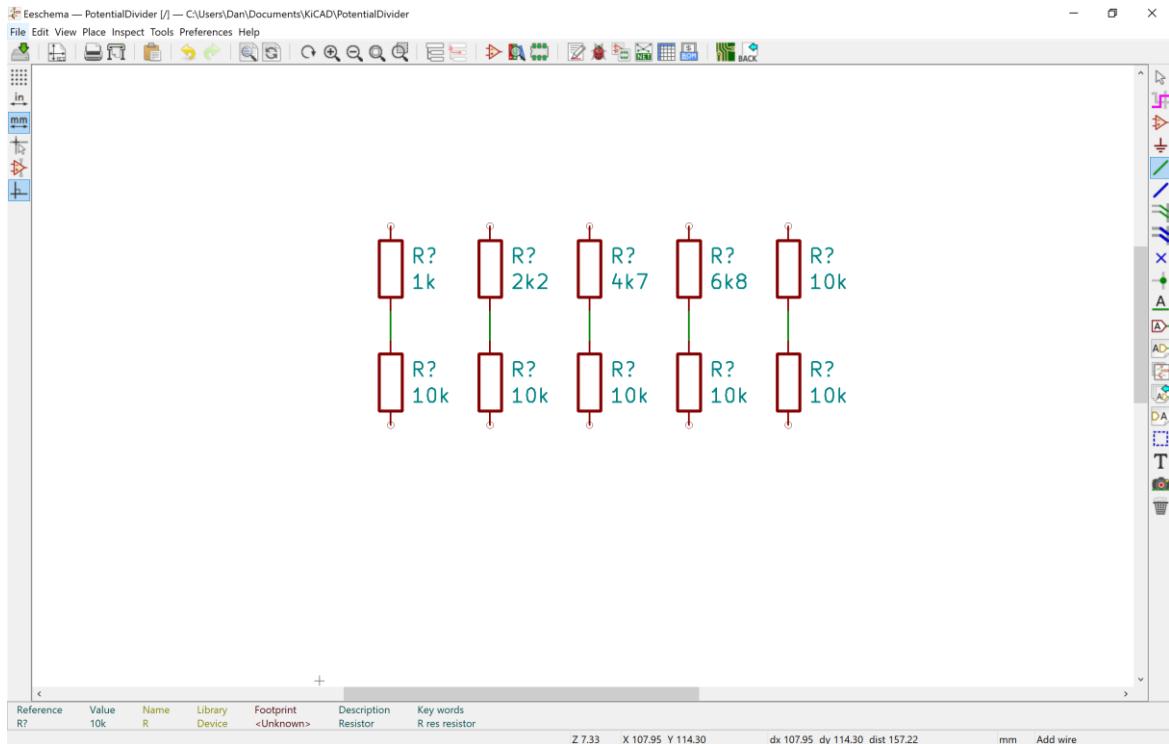
9. Switch to the selection tool. Click and drag to select all the resistors. Right click and select copy block. Right click in an empty space and select paste to copy the resistors.



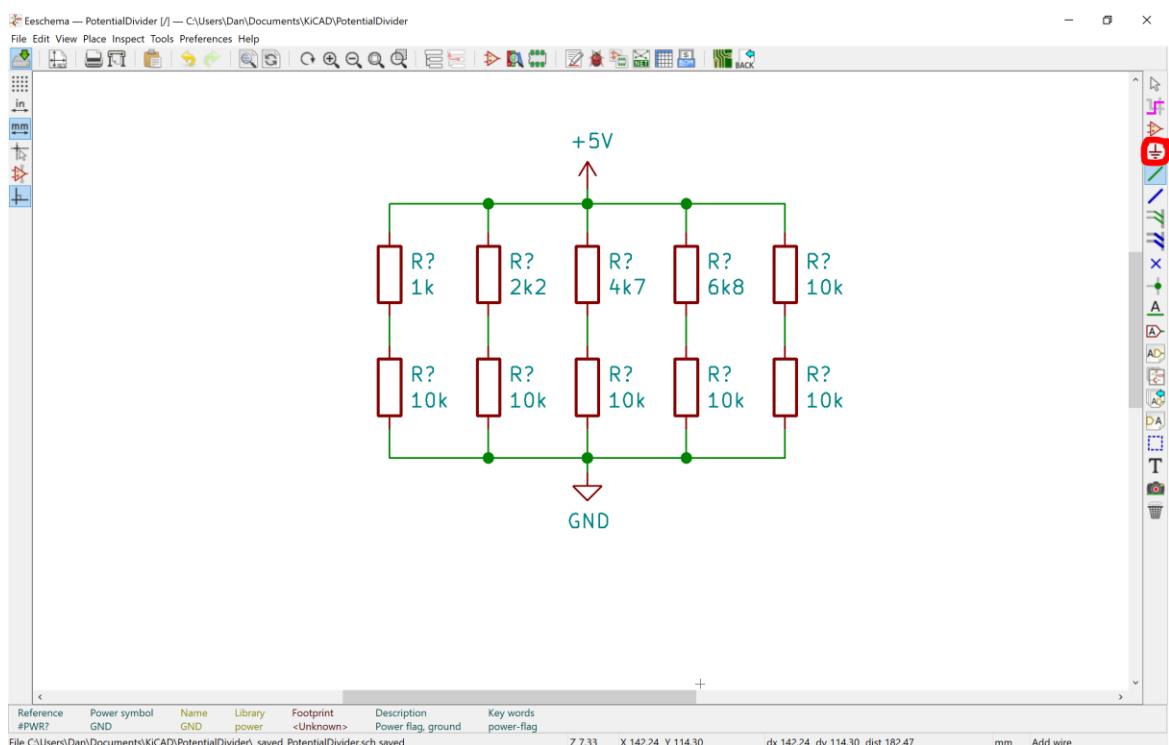
10. Using 'v' change the value of each resistor as shown



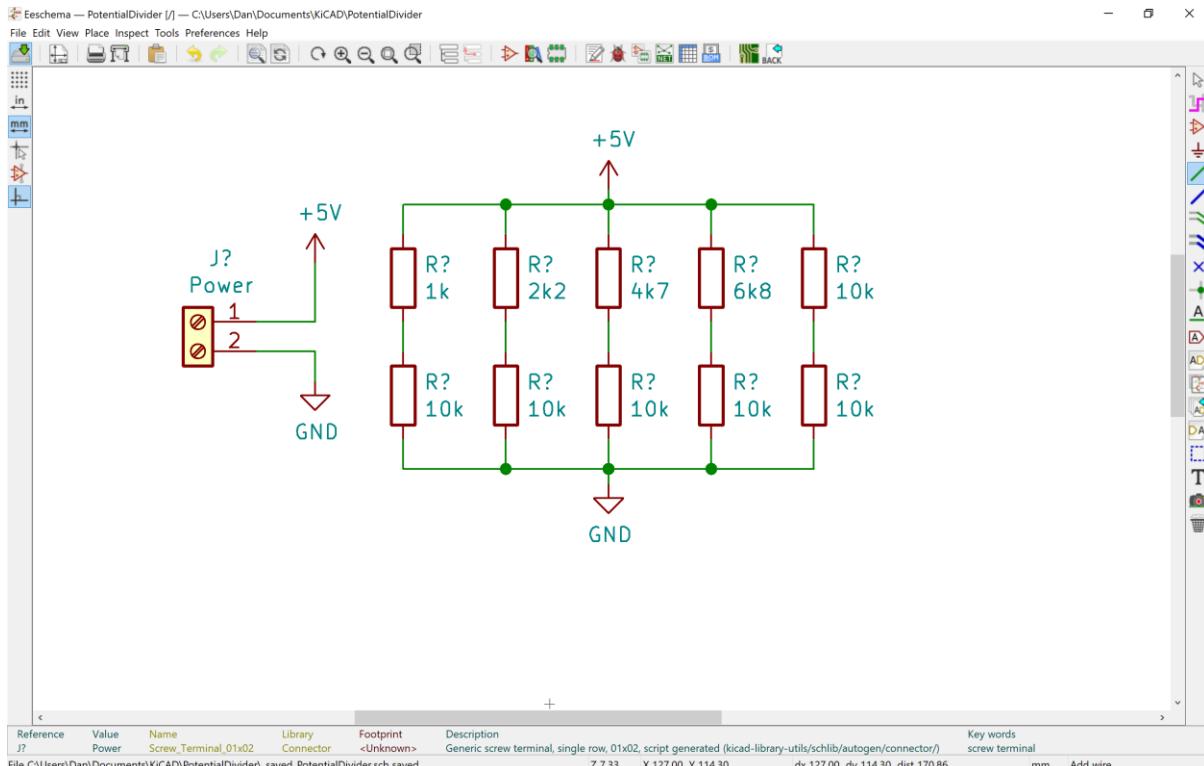
11. In turn, hover over the bottom connection of each top resistor, press 'w' to start drawing a wire, hover over the top connection of the resistor below and press 'w' again to end the wire.



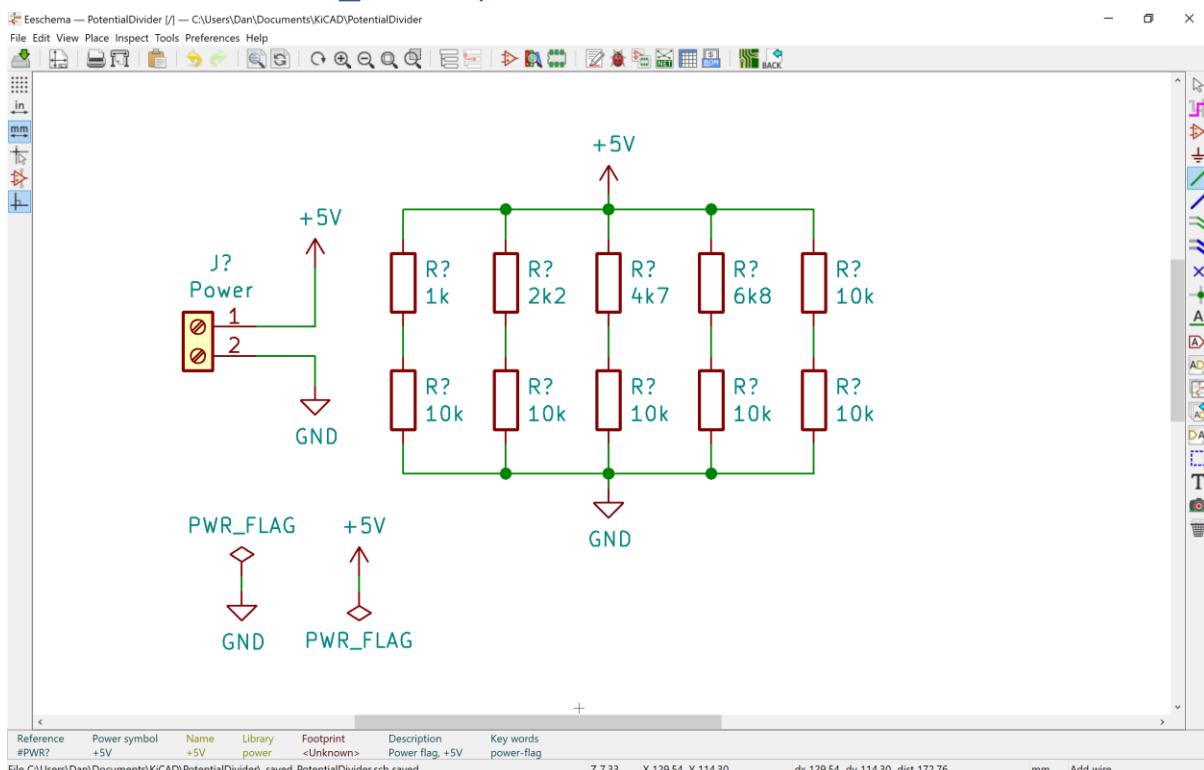
12. Use the symbol selector to place '+5V' and GND symbols below the resistor blocks. Join with wires as shown. You can press 'w' to make a corner in the wire.



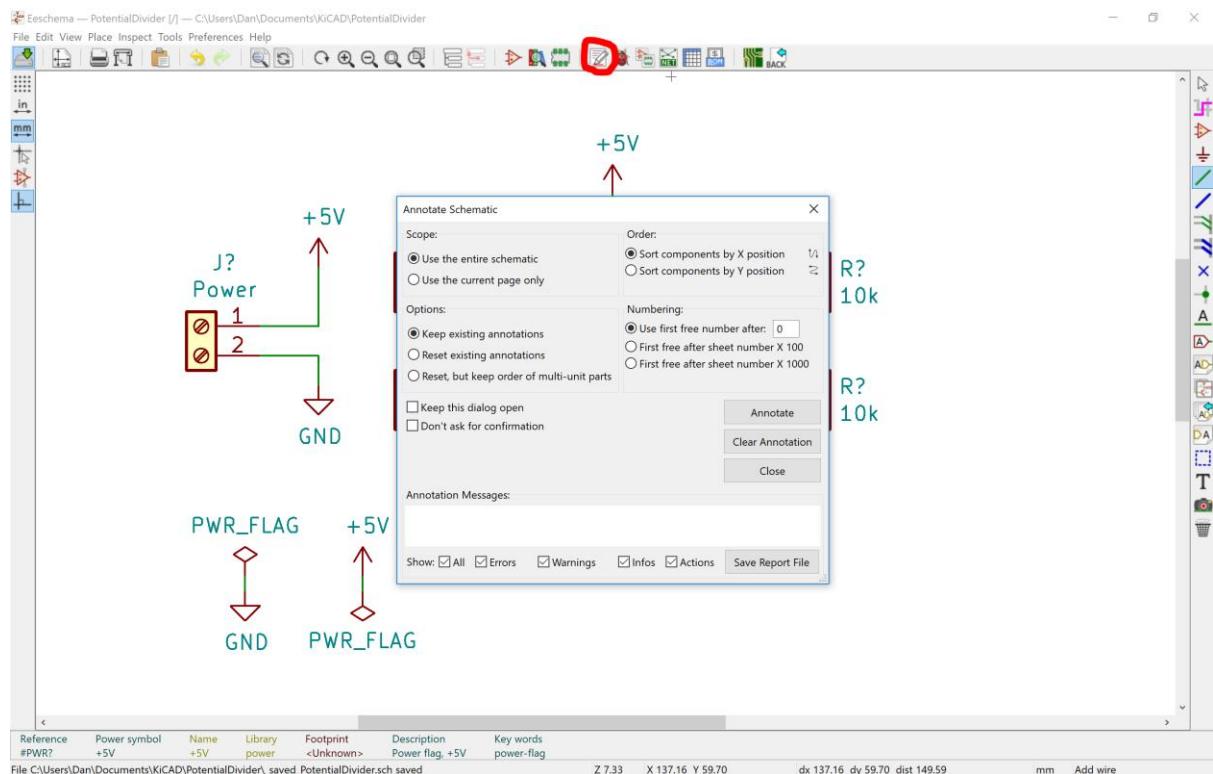
13. Place a ‘screw terminal’ on the schematic and connect to +5V and GND. Press ‘y’ to flip the terminal in the y-axis. Change the value of the screw terminal to rename it.



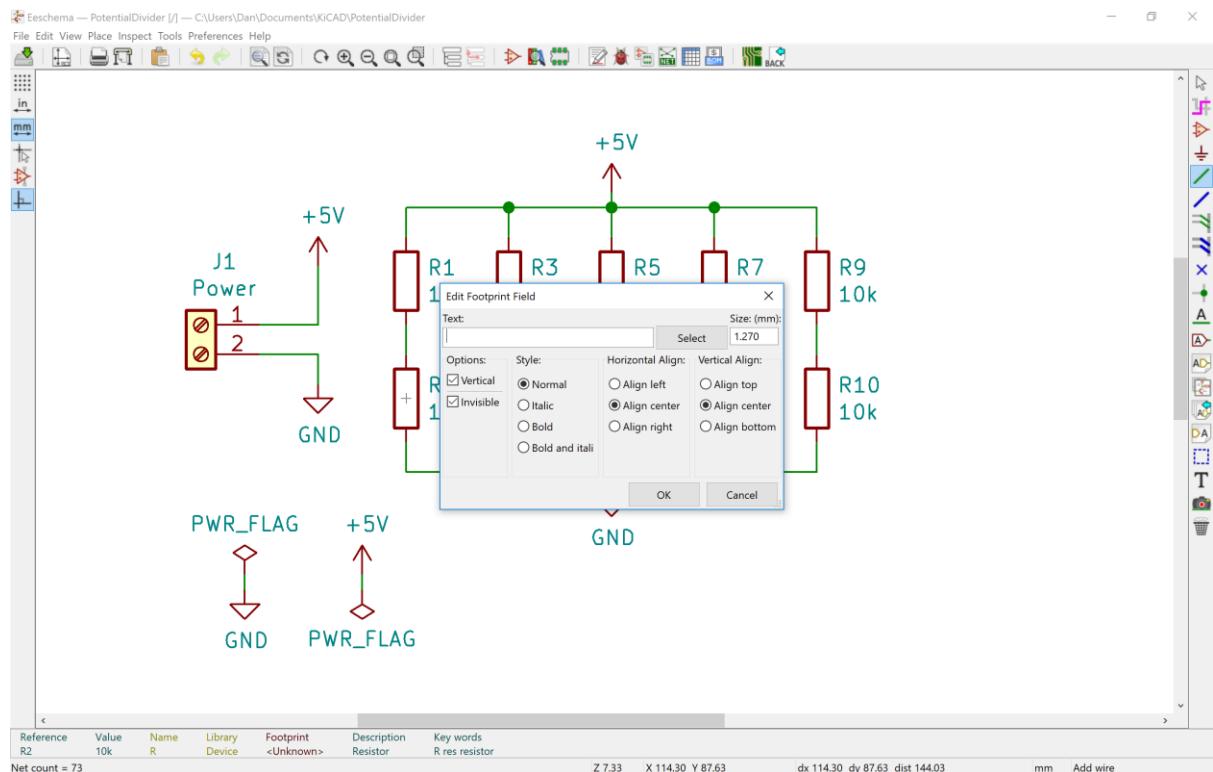
14. Add two PWR\_FLAG symbols and connect to +5V and GND



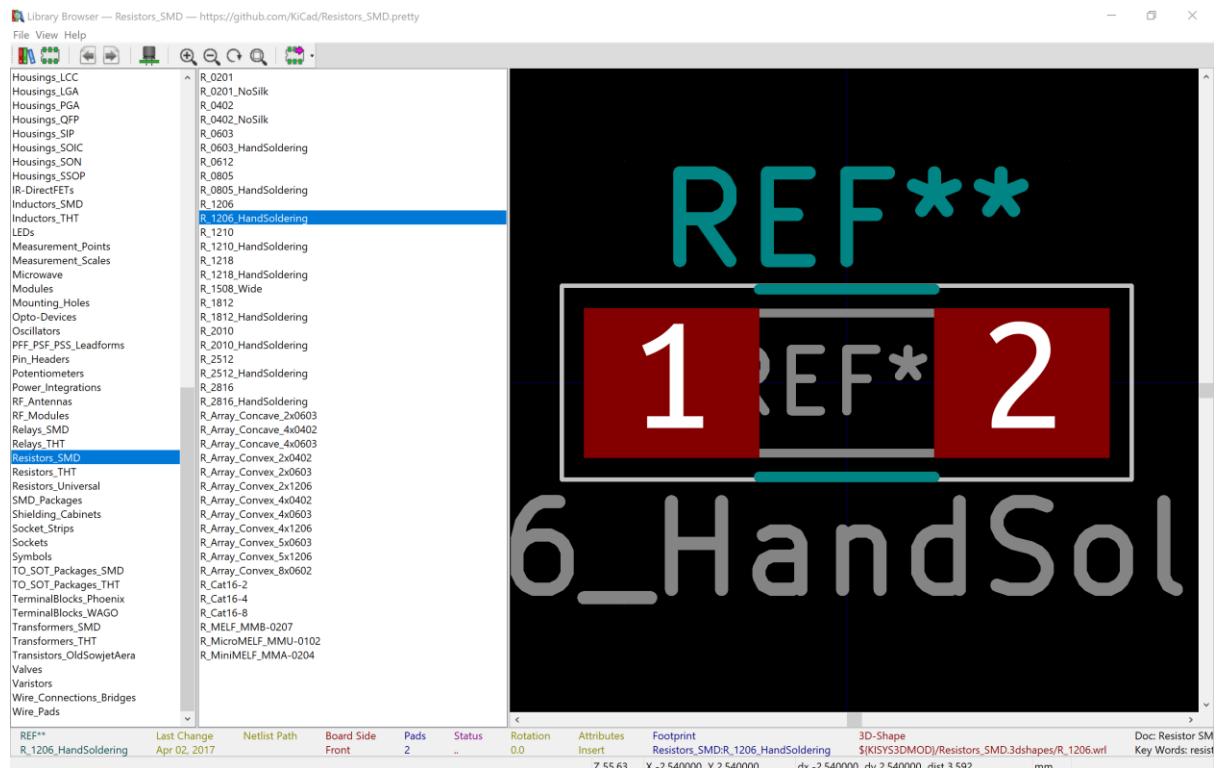
## 15. Annotate the schematic



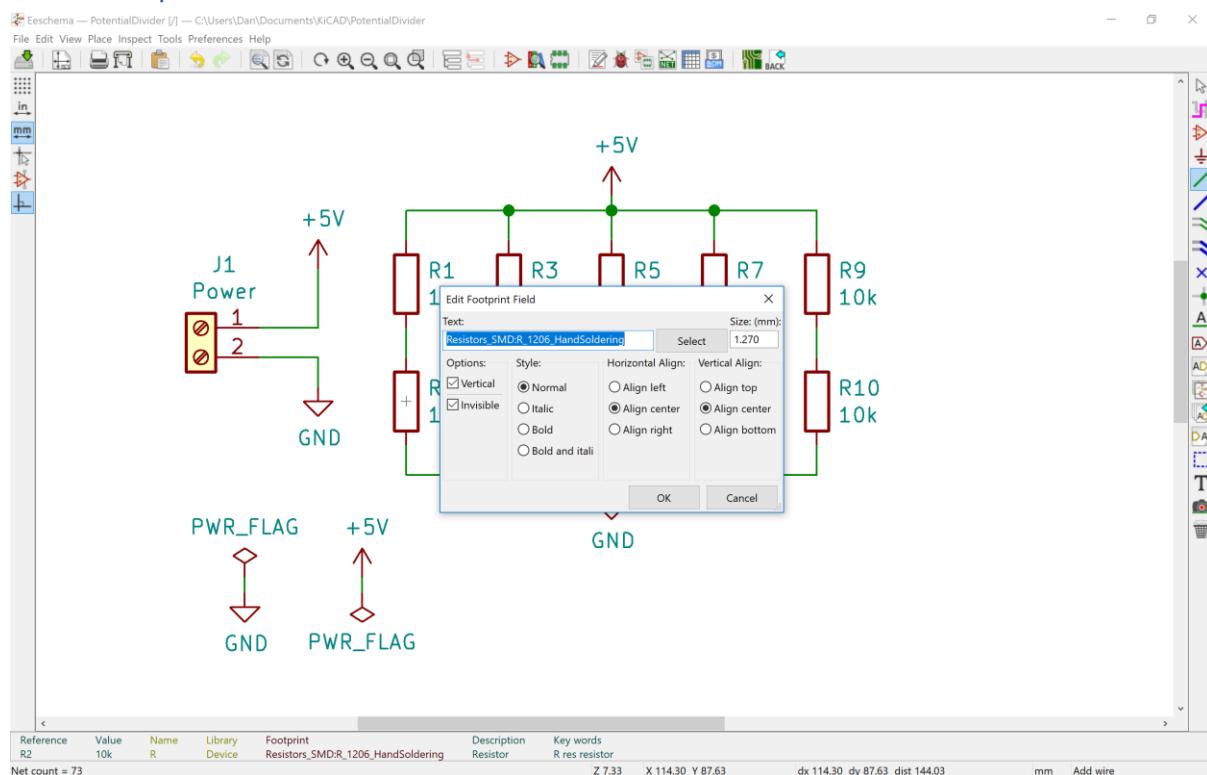
## 16. Hover over one of the bottom resistors and press 'f' to open the footprint window



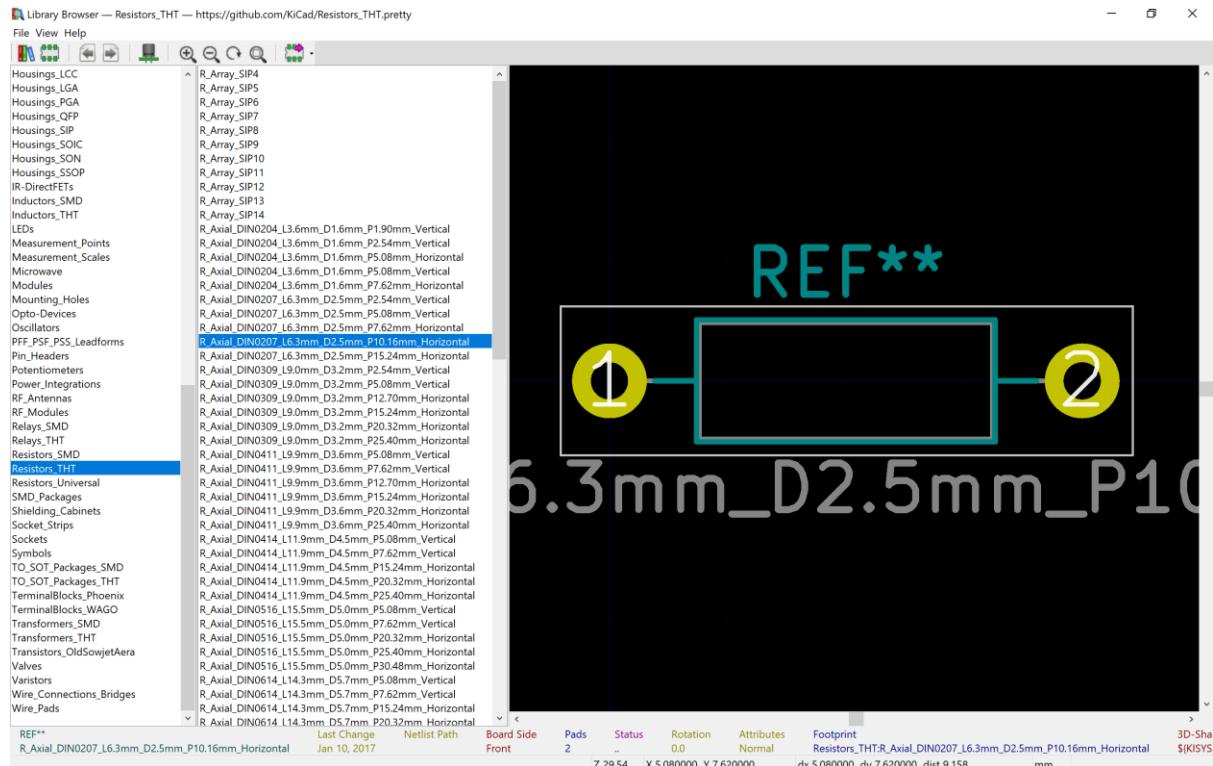
17. Click select, wait for the selection window to open, browse to 'Resistors\_SMD' and double click 'R\_1206\_HandSoldering'.



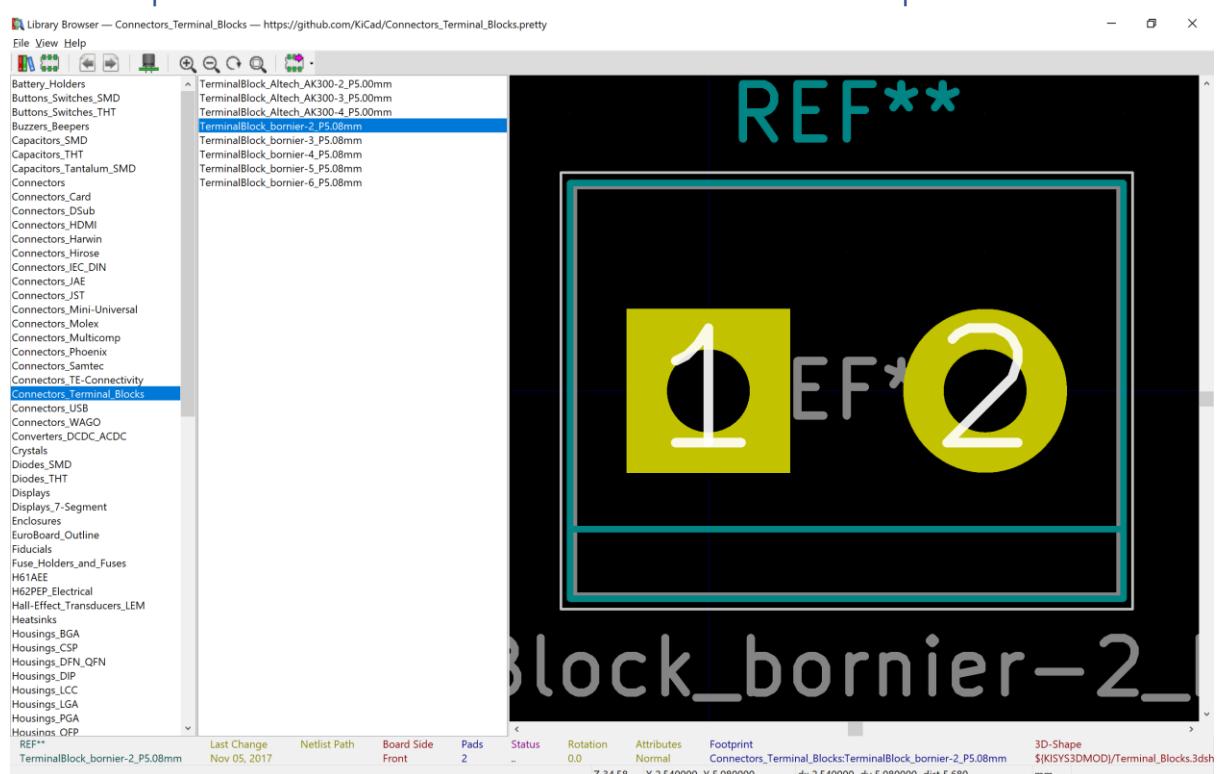
18. Select the footprint text and copy it with Ctrl+C. Paste it into the footprint field for each of the bottom resistors.



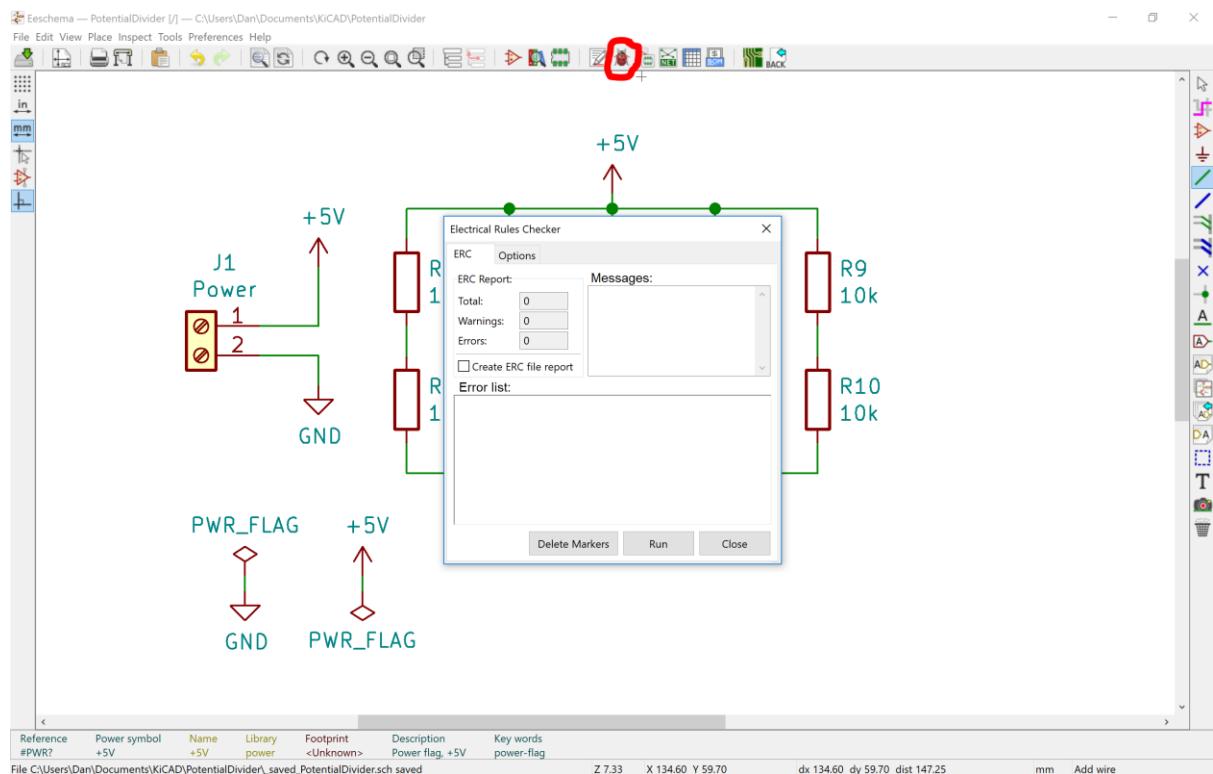
19. Repeat the steps to set the top resistor footprint but use 'Resistors\_THT' and 'R\_Axial\_DIN0207\_L6.3mm\_D2.5mm\_P10.16mm\_Horizontal'



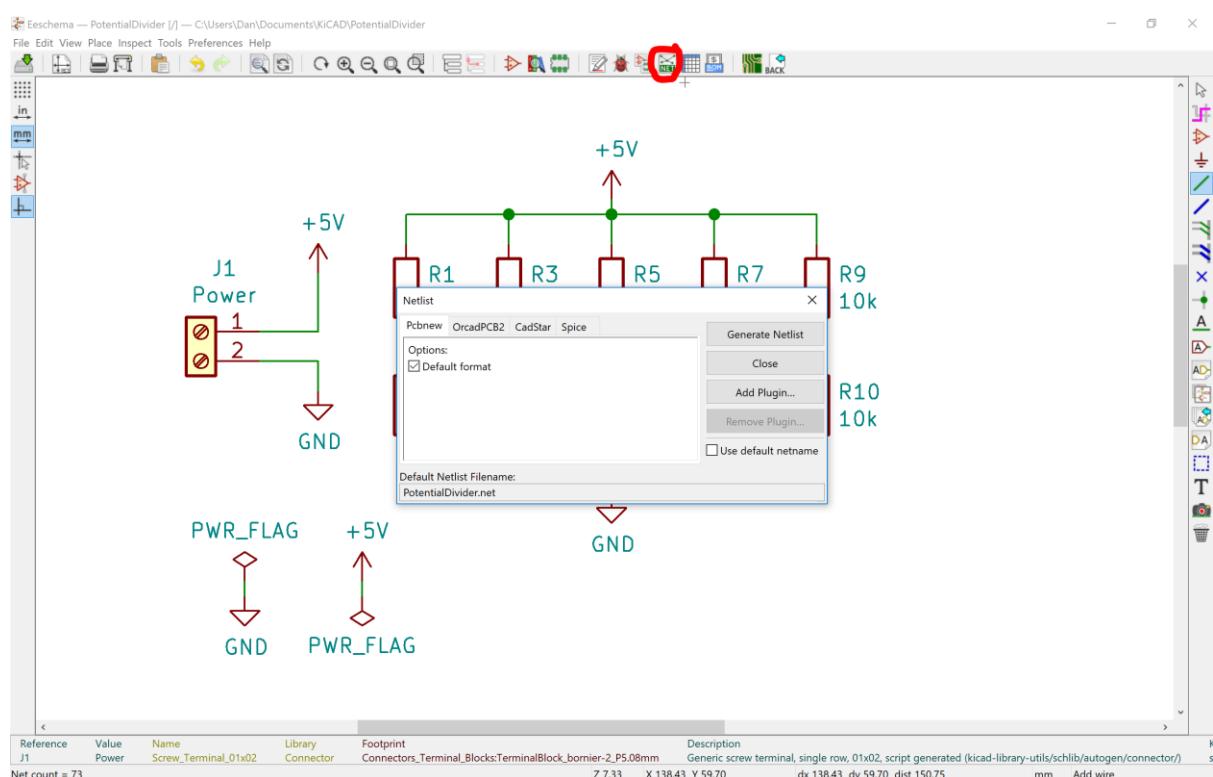
20. Repeat for the screw terminal and set the footprint as shown.



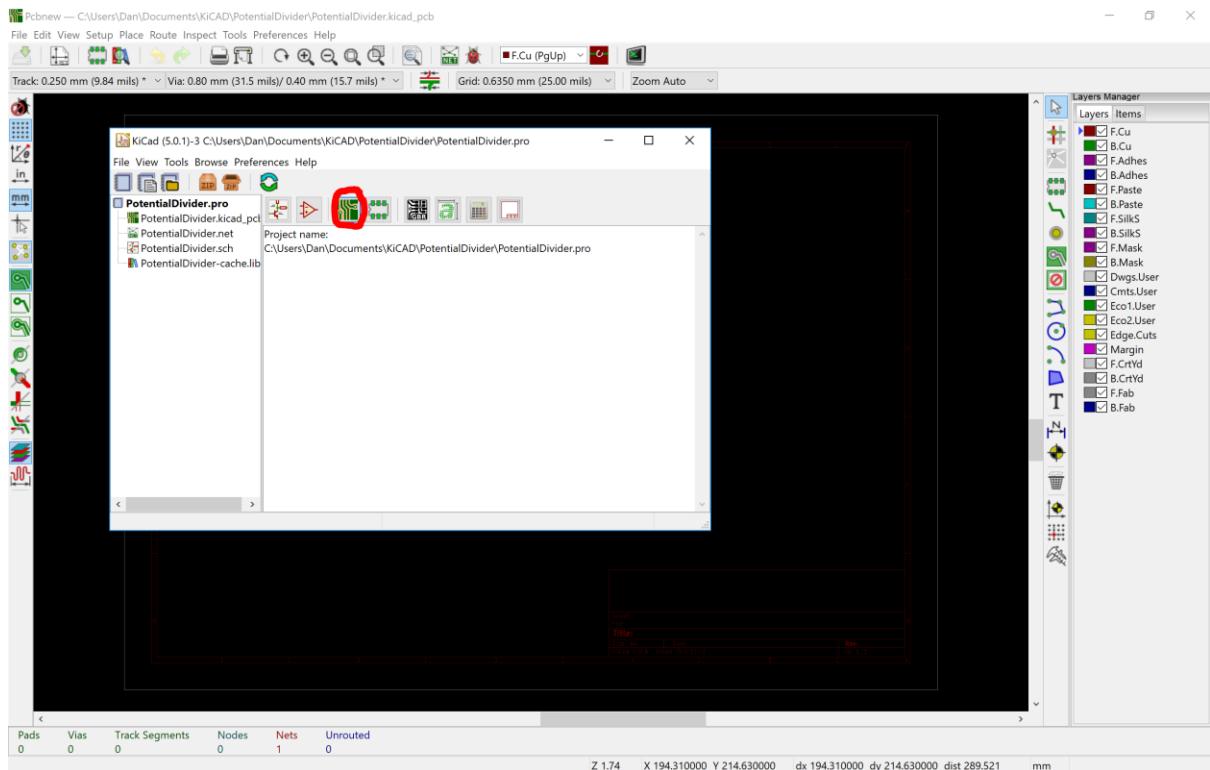
## 21. Run the DRC and check there are no errors



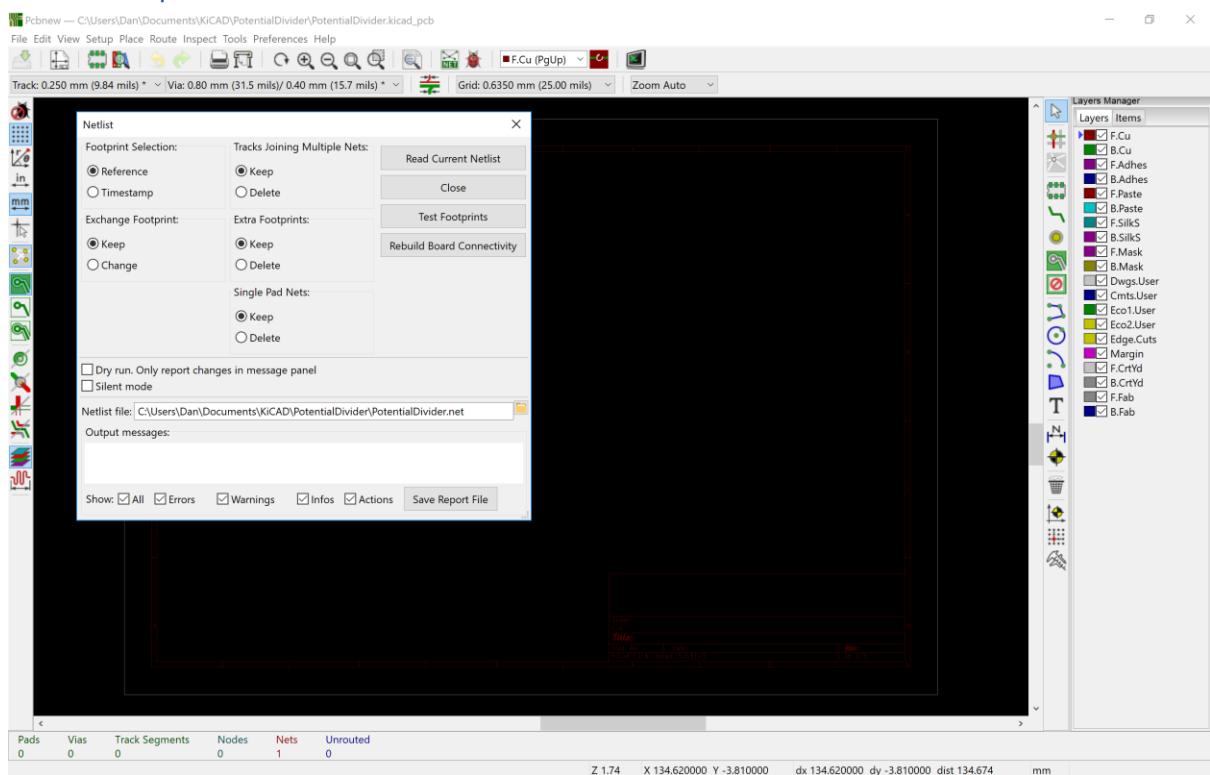
## 22. Generate the netlist for the PCB



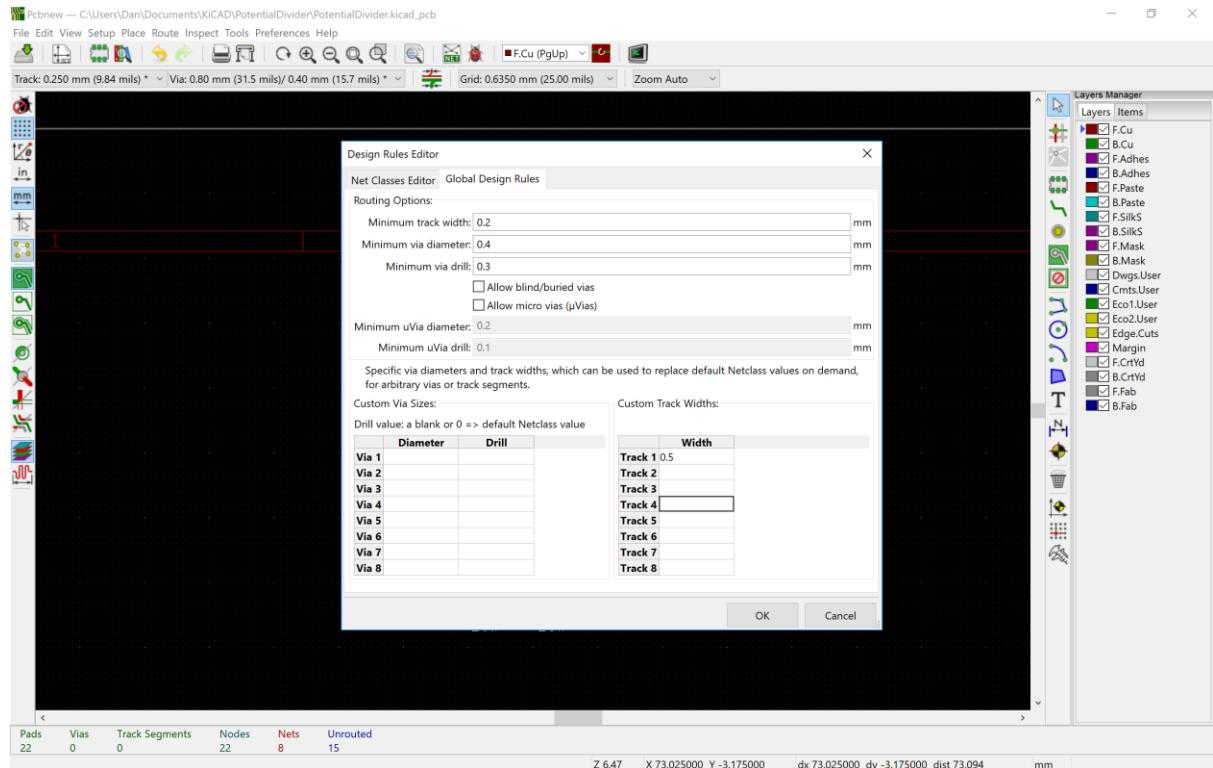
## 23. Save and close the schematic editor. Open the PCB editor.



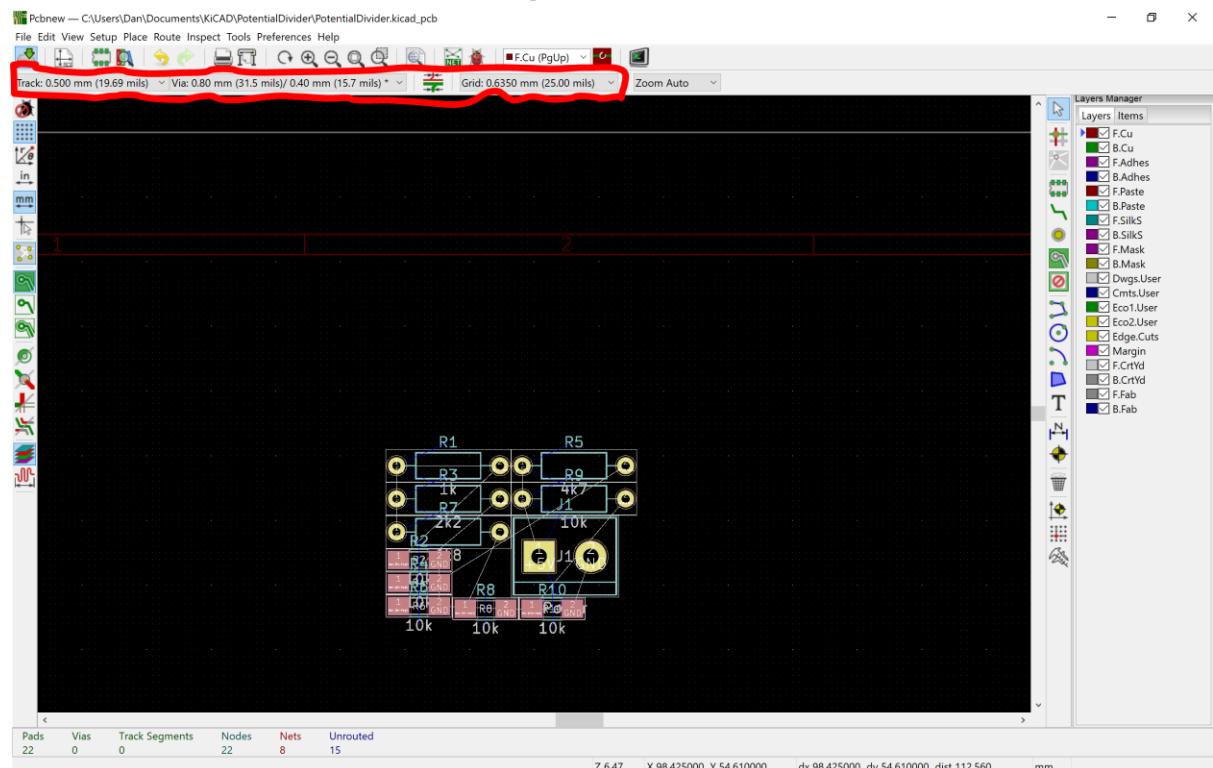
## 24. Import the netlist into the PCB and click to place the components.



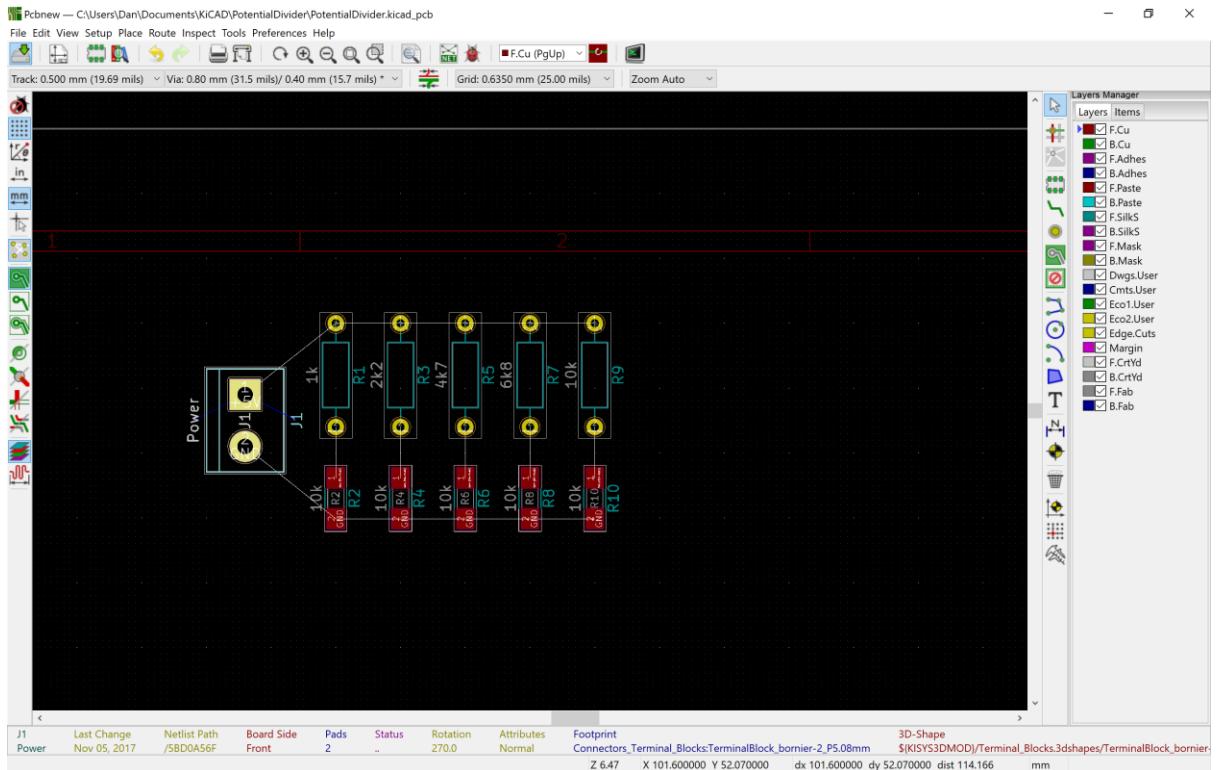
25. Save the PCB (do this often). Open the design rules editor (setup → Design Rules) and add 0.5mm as a track option



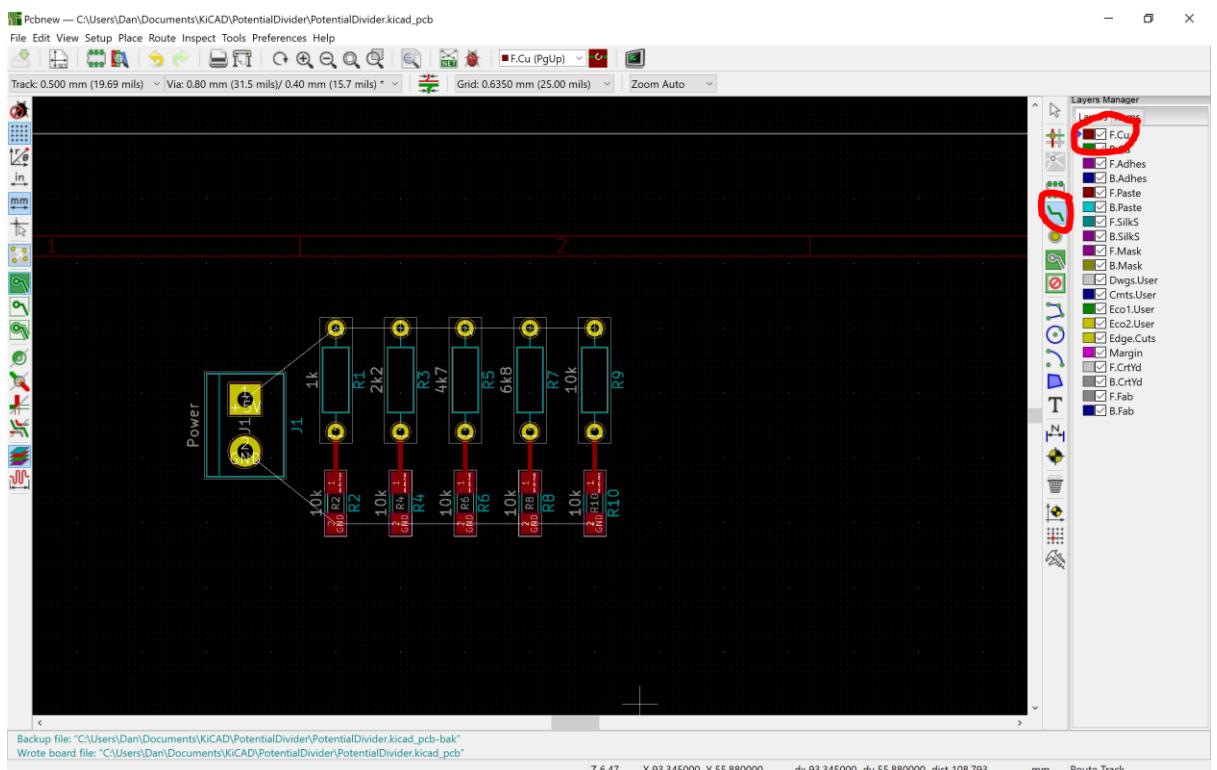
26. Set the track width, and grid as shown



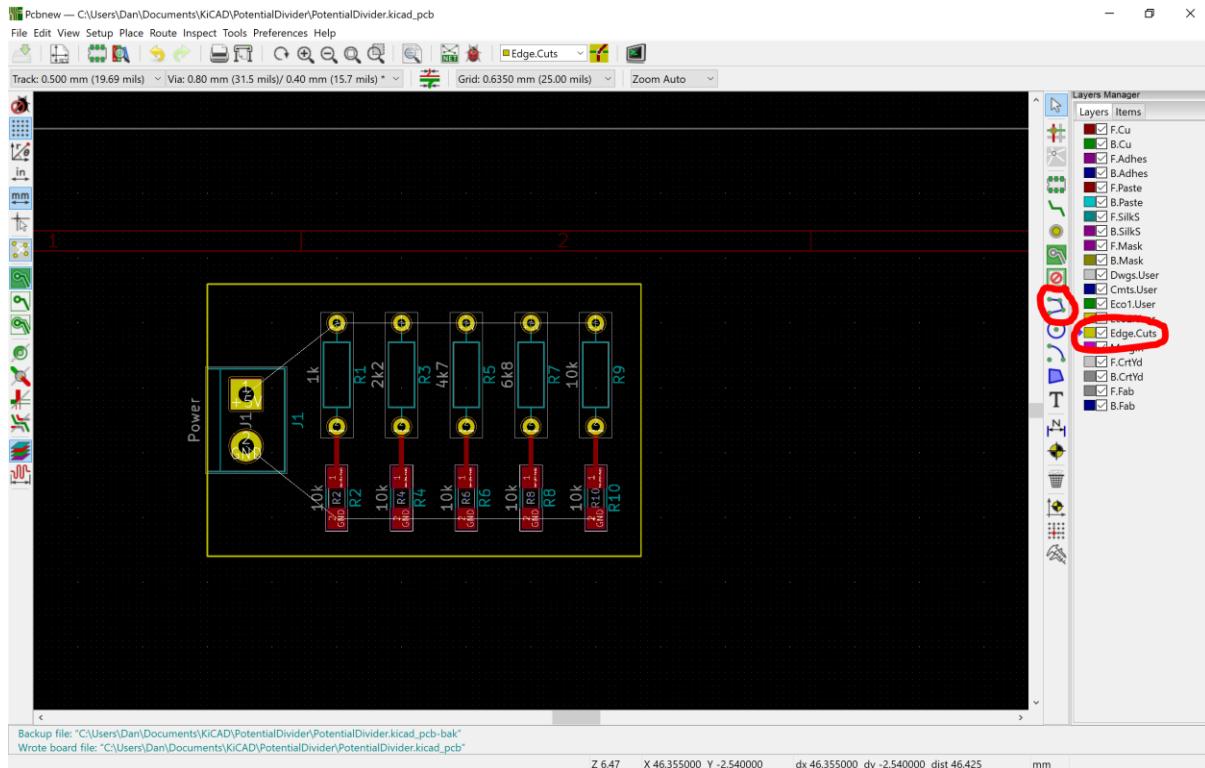
27. Select each component in turn by clicking on it. Press 'm' to move the component and 'r' to rotate by 90deg while moving. Arrange the components as shown.



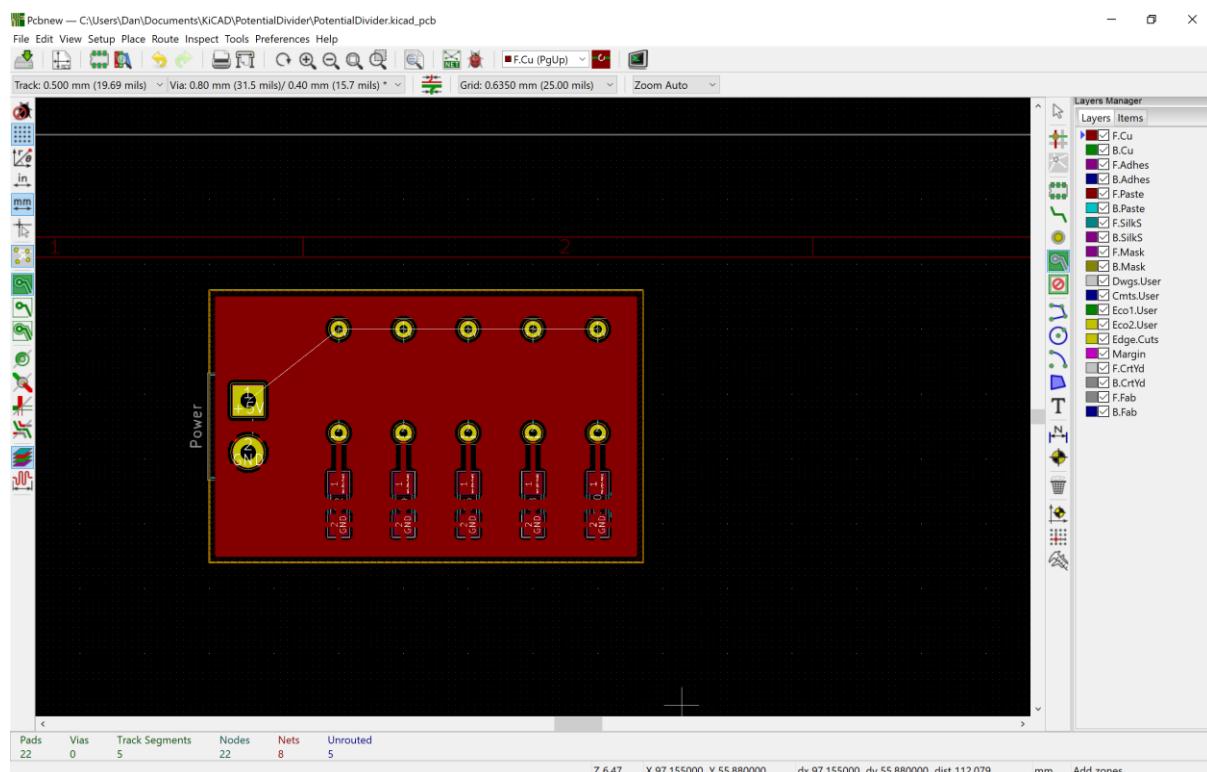
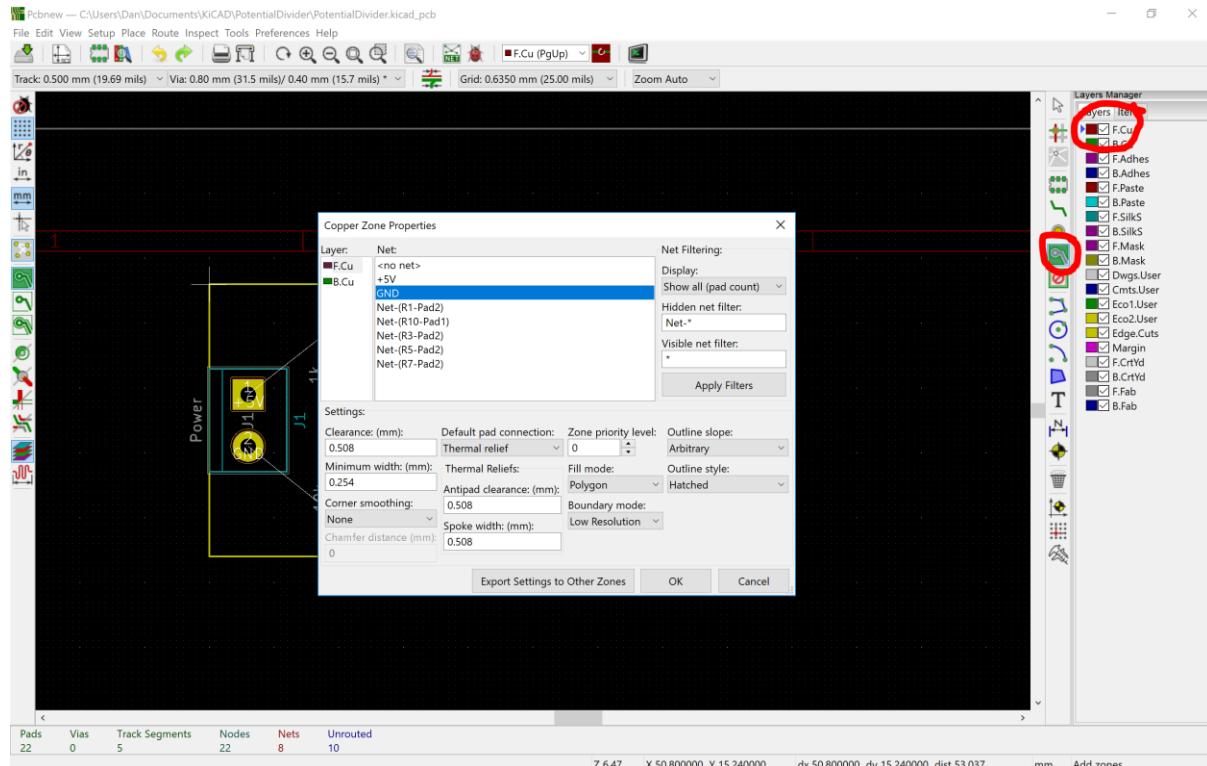
28. Make sure you are on the F.Cu layer, select the track tool and connect the resistors as shown



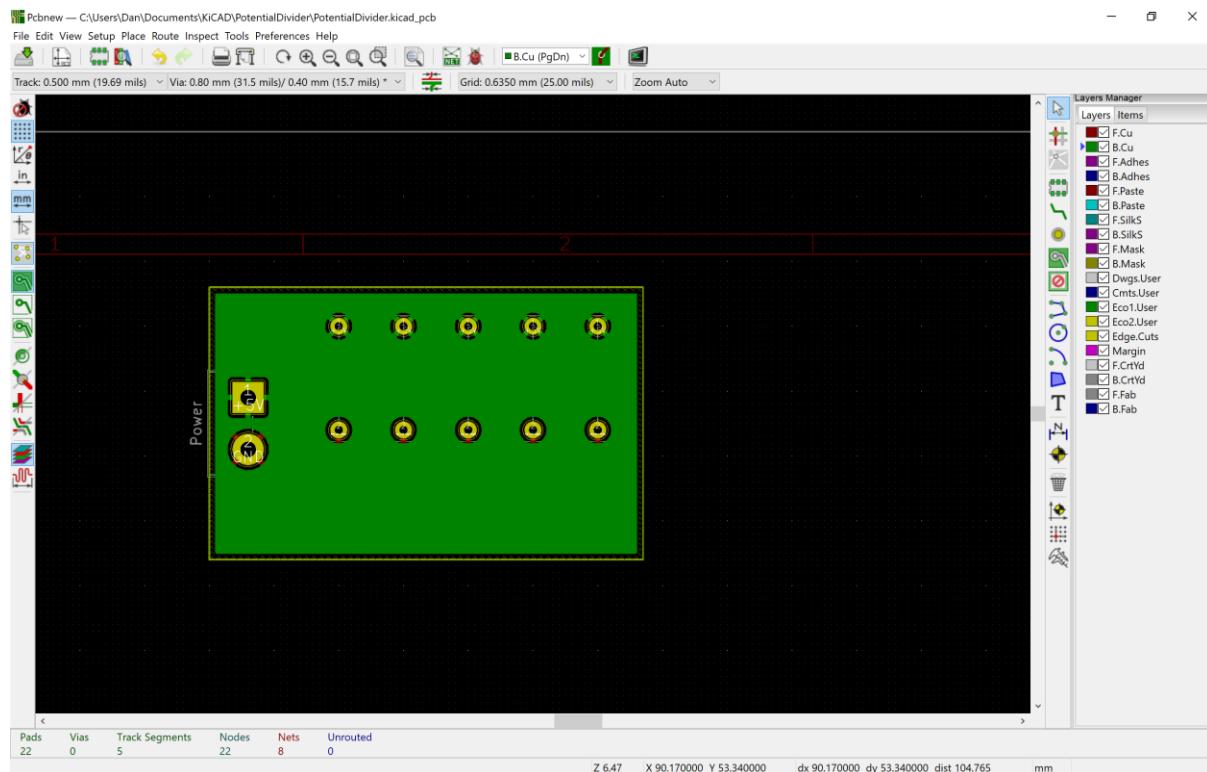
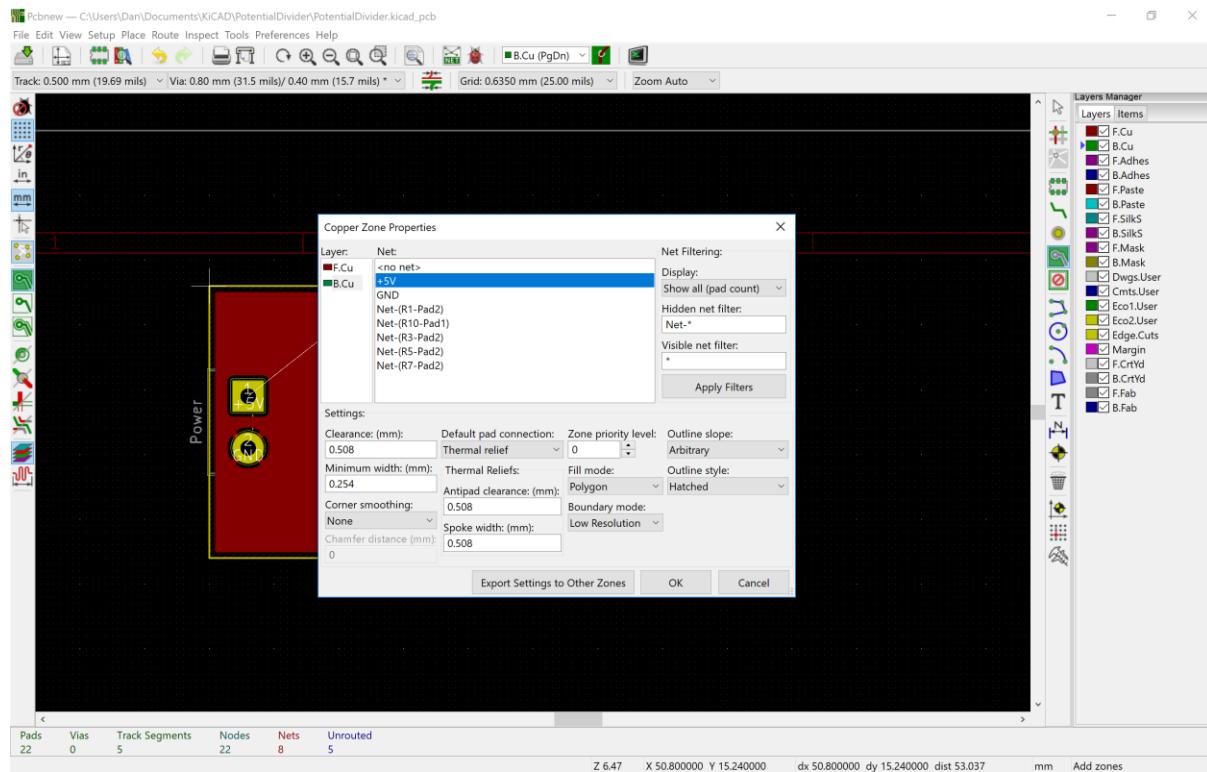
29. Switch to the Edge.Cuts layer and select the line tool. Draw a box around your circuit – click to place corners. Double click to stop drawing



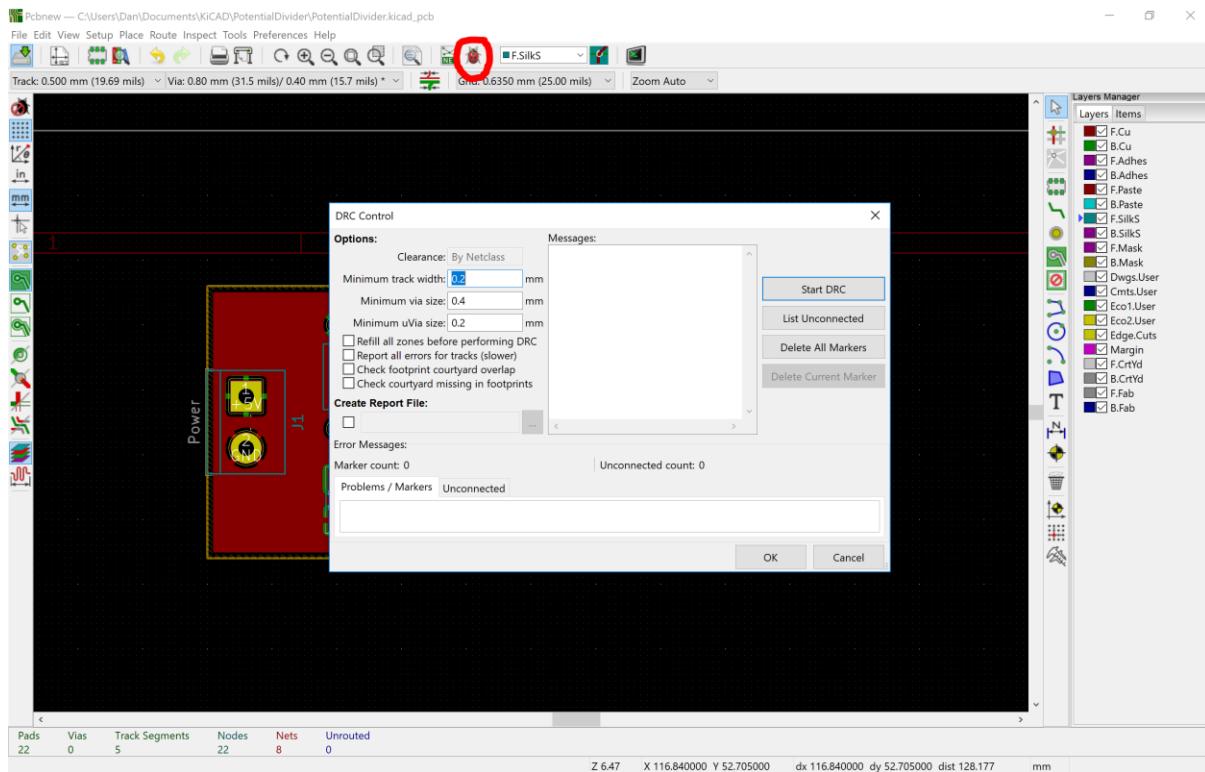
30. Switch to the F.Cu layer and select the pour tool. Draw a region following the edge.cuts lines. Select the net to GND on the pop-up as shown



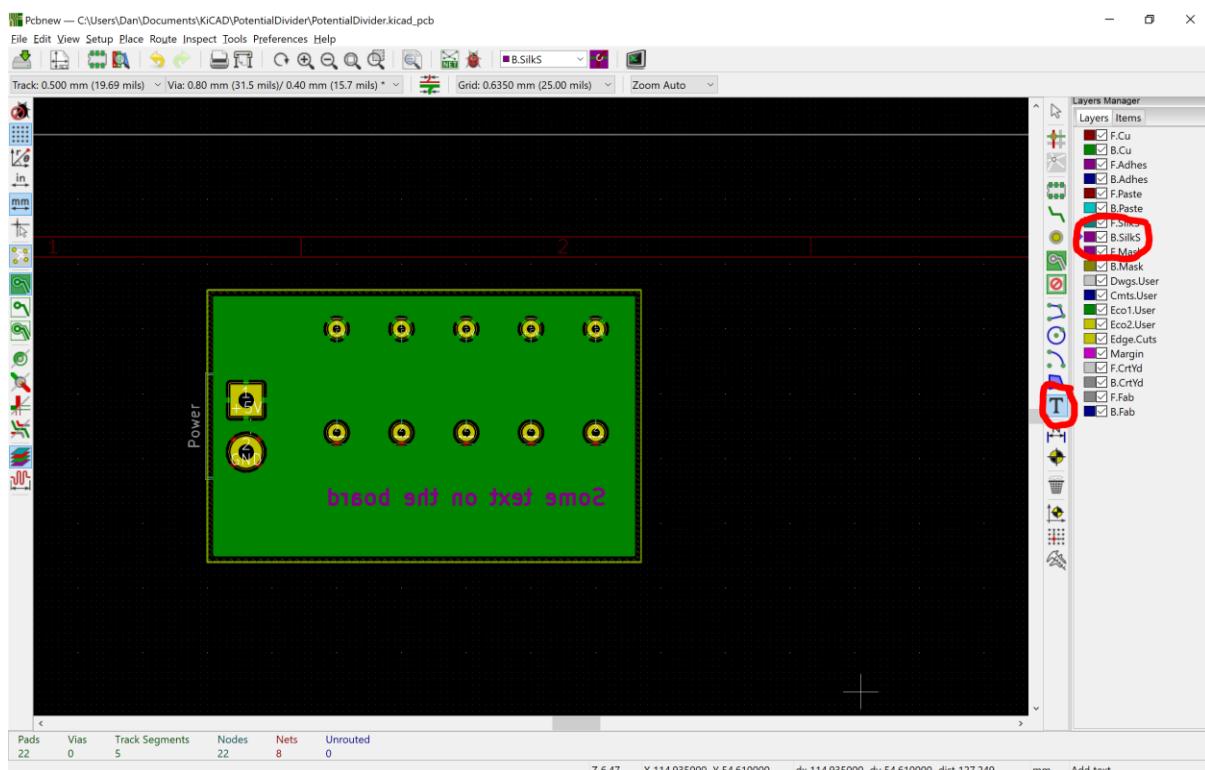
### 31. Switch to B.Cu and repeat but set to the 5V net



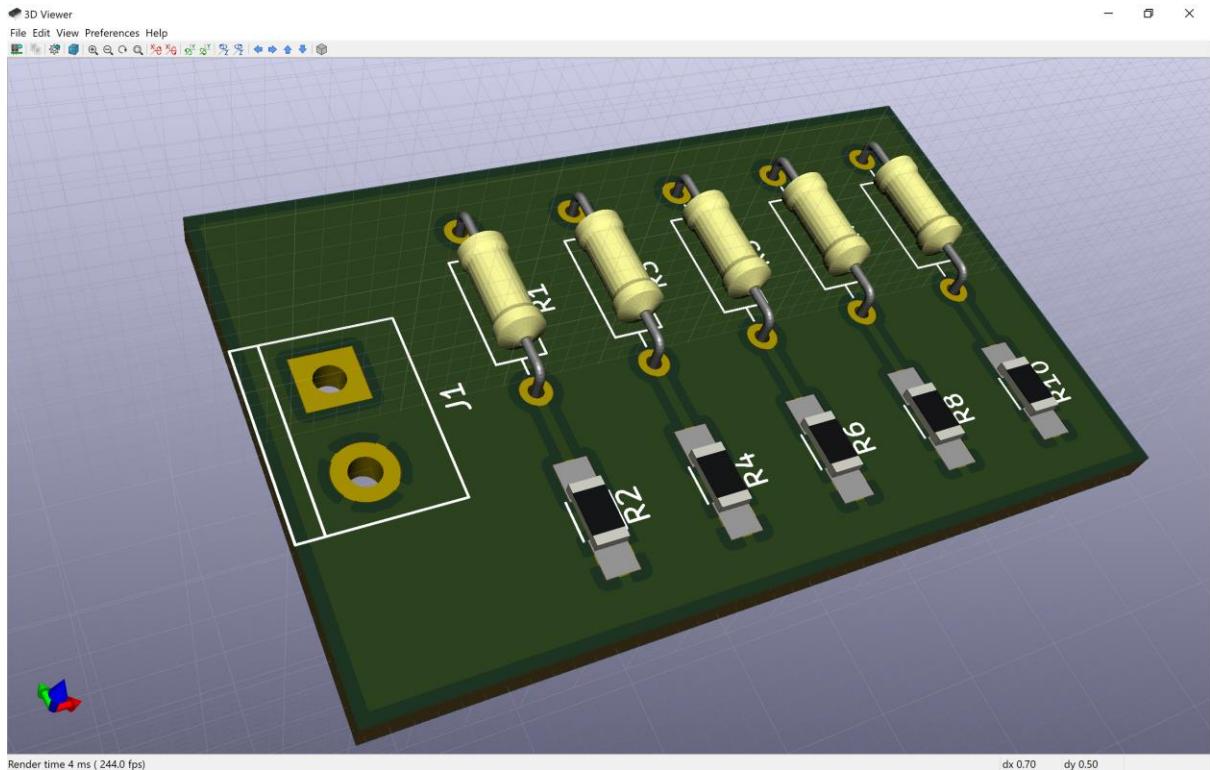
### 32. Run the DRC tool to check your board



### 33. Select B.SilkS, Click the text tool and place some text on the board



34. Press Alt+3 or View→3D Viewer, to see a 3D rendering of your board.



#### Additional tasks

- Add mounting holes
- Round the corners
- Label the terminal block connections on the silkscreen
-