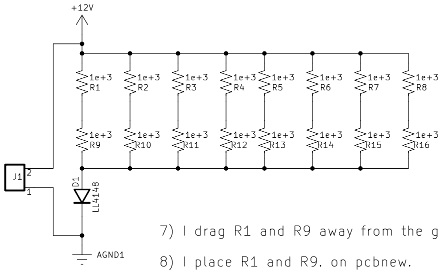

1) I draw schematic diagram. I started with one resistor and duplicated so that all would have same pin orientation.

2) I set footprints for all components.

3) I let KiCad annotate the schematic.

4) I opened my pcbnew file and drew a 75 x 40 mm rectangle in edge cuts layer.

5) I updated the pcbnew file from the schematic.

6) My pin numbers are invisible but I check on pcbnew to see that R1 pin 1 is connected to R9 pin 2.

7) I drag R1 and R9 away from the group and delete the other resistors from pcbnew.

8) I place R1 and R9. on pcbnew.

9) I duplicate R1 and R9 on pcbnew to make 4 resistors. I duplicate the 4 to make 8. Then I duplicate the 8 to make 16.

10) I manually re-assigned the ref designators for R2-R8 and R10 - R16. The rats nest looks correct!

11) I saved both files.

12) I updated pcb from schematic, after checking "Re-link footprints to schematic symbols based on their reference designations."

13) There was some added work to manually re-assign the reference designations on pcbnew, but I think this was less work than manually placing all of the resistors.

This is a stupid heater for layout test of what I would call a "component array"