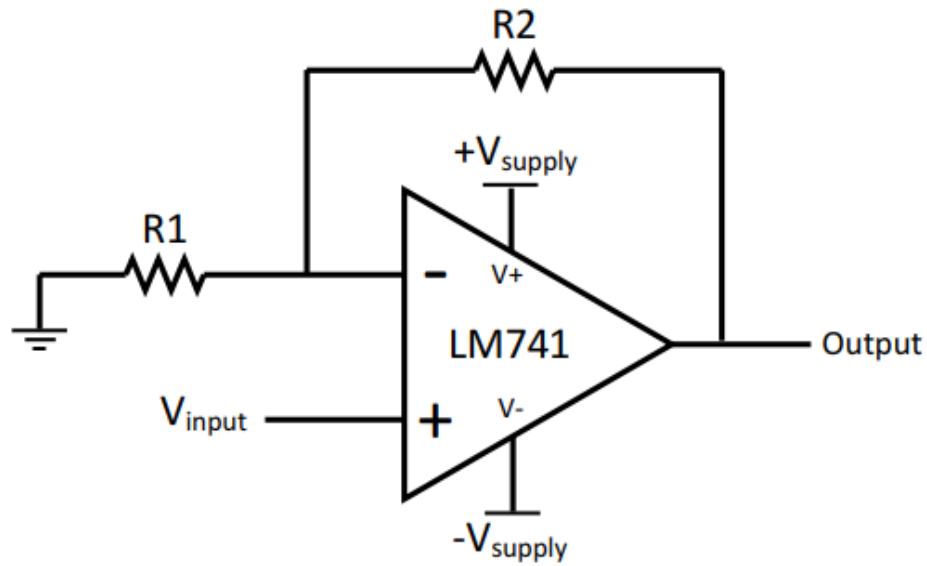




Logbook amp1

L4 – Workshop Skills/Learning in the Digital Era

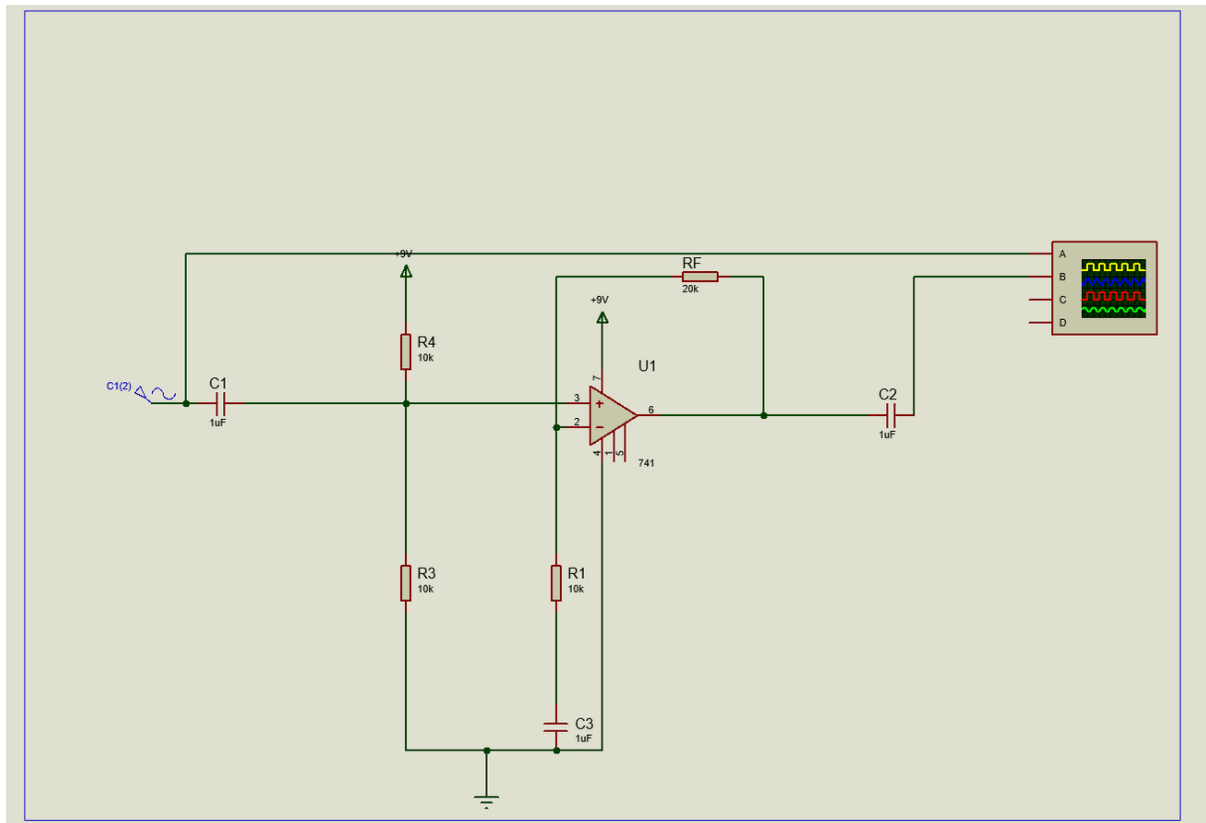
BENG2–PT Embedded Electronic Systems

Samuel Langdown
2214267

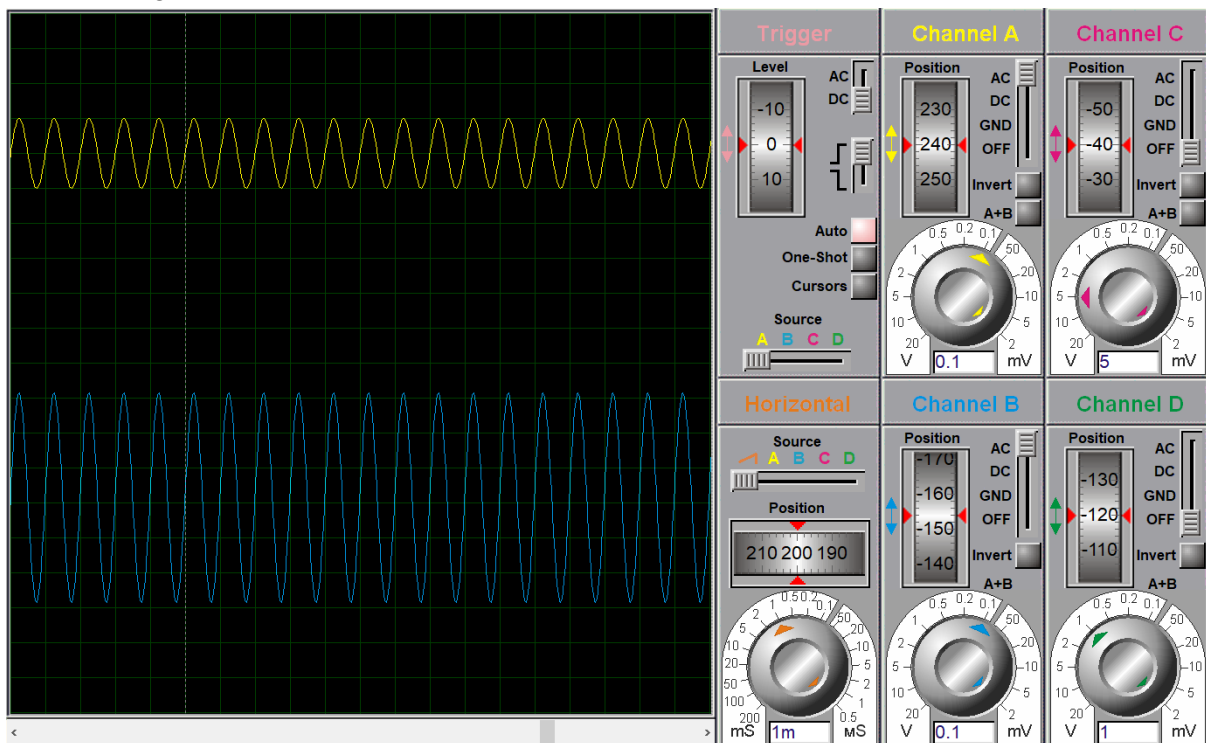
Table of contents

<u>Proteus Schematic</u>	<u>2</u>
<u>Simulation results</u>	<u>3</u>
<u>Easy-PC Schematic</u>	<u>4</u>
<u>PCB</u>	<u>5</u>
<u>PCB Results</u>	<u>10</u>
<u>Breadboard</u>	<u>12</u>
<u>Breadboard Results</u>	<u>13</u>

Proteus Schematic

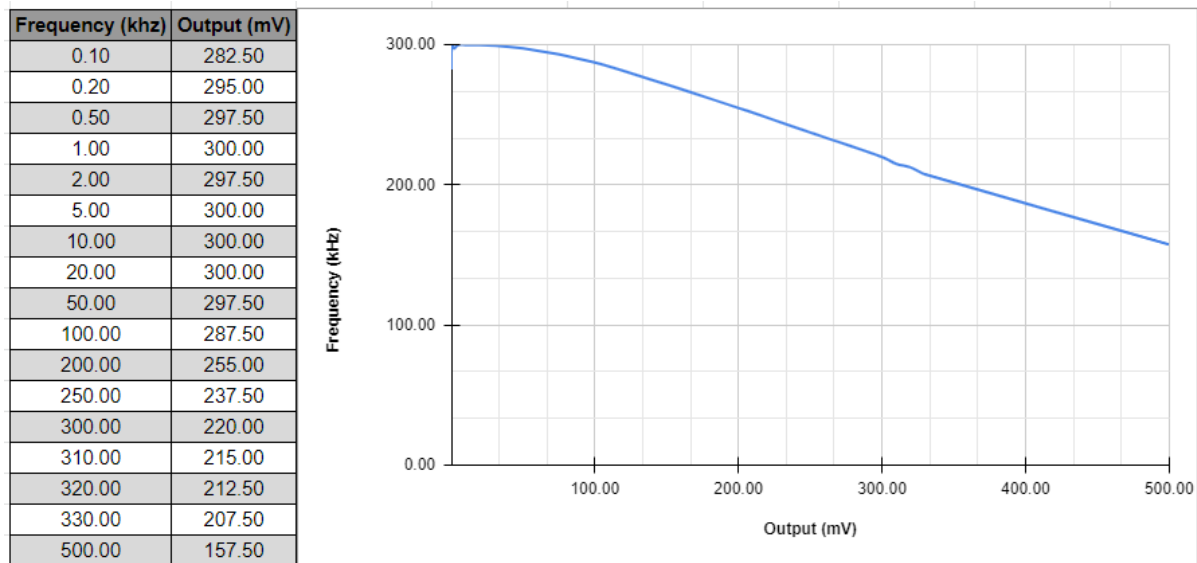


I created the circuit in proteus by adding each component individually and then connected all the wires together.



Once my circuit was complete I viewed the oscilloscope to see the waves of the components.

Simulation results



I created a table for different frequencies and then used the oscilloscope to read off what the output would be for each. I then recorded my findings and compared them to each other.

Electrical characteristics

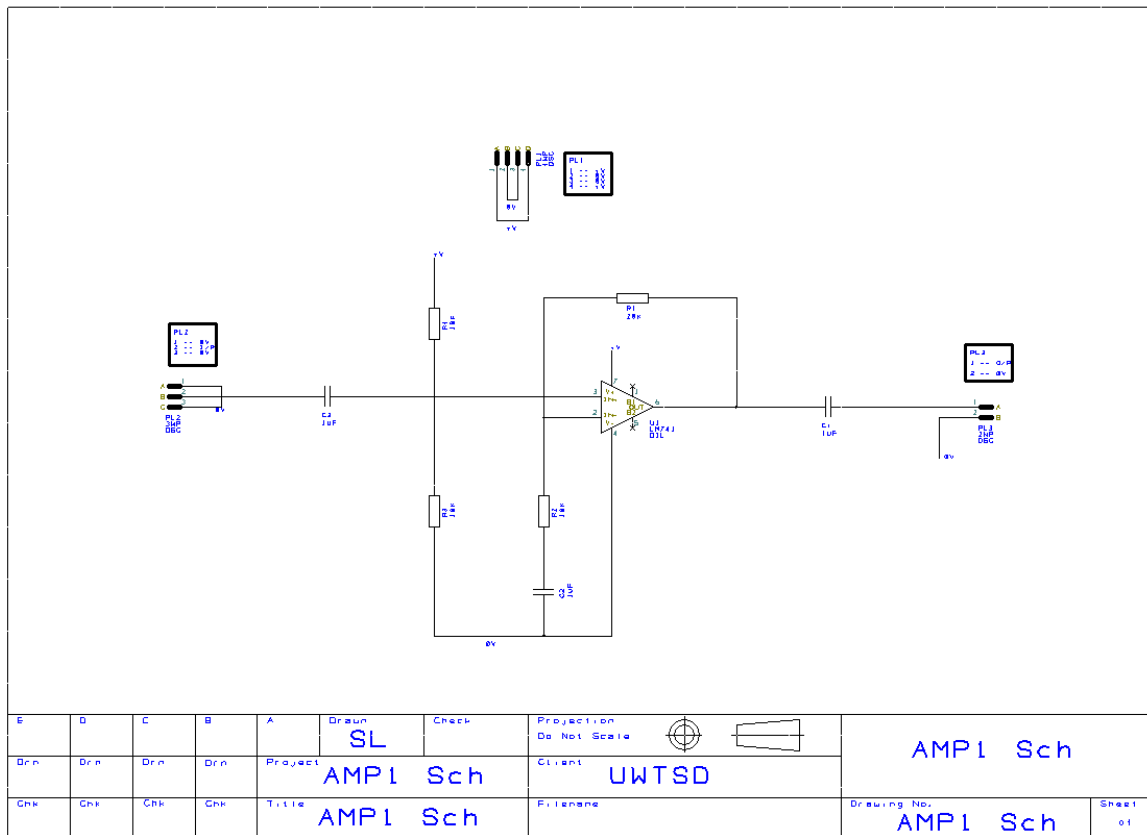
UA741

**Table 3. Electrical characteristics at $V_{CC} = \pm 15\text{ V}$, $T_{amb} = 25^\circ\text{C}$
(unless otherwise specified) (continued)**

Symbol	Parameter	Min.	Typ.	Max.	Unit
GBP	Gain bandwidth product $V_i = 10\text{ mV}$, $R_L = 2\text{ k}\Omega$, $C_L = 100\text{ pF}$, $f = 100\text{ kHz}$	0.7	1		MHz
THD	Total harmonic distortion $f = 1\text{ kHz}$, $A_v = 20\text{ dB}$, $R_L = 2\text{ k}\Omega$, $V_o = 2\text{ V}_{pp}$, $C_L = 100\text{ pF}$, $T_{amb} = +25^\circ\text{C}$		0.06		%
e_n	Equivalent input noise voltage $f = 1\text{ kHz}$, $R_s = 100\text{ }\Omega$		23		$\frac{\text{nV}}{\sqrt{\text{Hz}}}$
ϕ_m	Phase margin		50		Degree

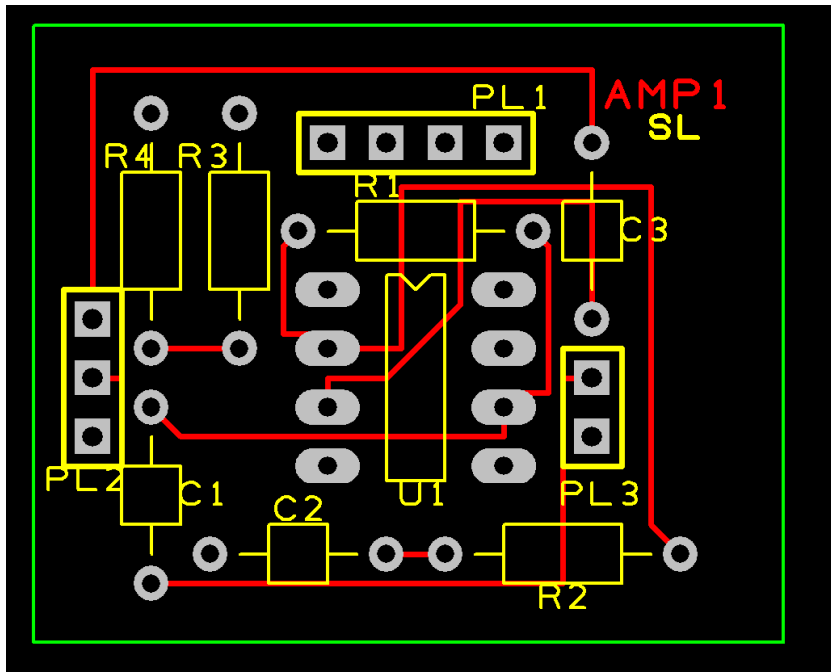
Firstly, I created the circuit for the op amp on proteus 8. I then viewed the oscilloscope so I could record the data for different frequencies in a table. I then worked out the gbp and compared it against the manufacturer's claims.

Easy-PC Schematic

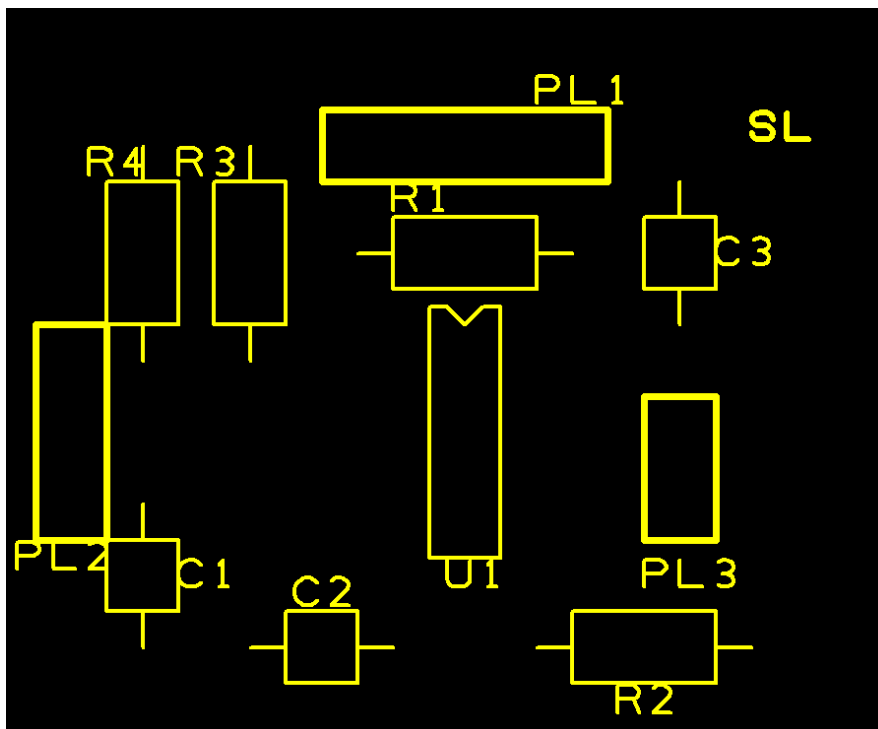


Created a schematic on easy-pc using the circuit from proteus as a guide. To make this into a pcb i used the “translate to pb” button to make a clean slate that i could position all my components on to.

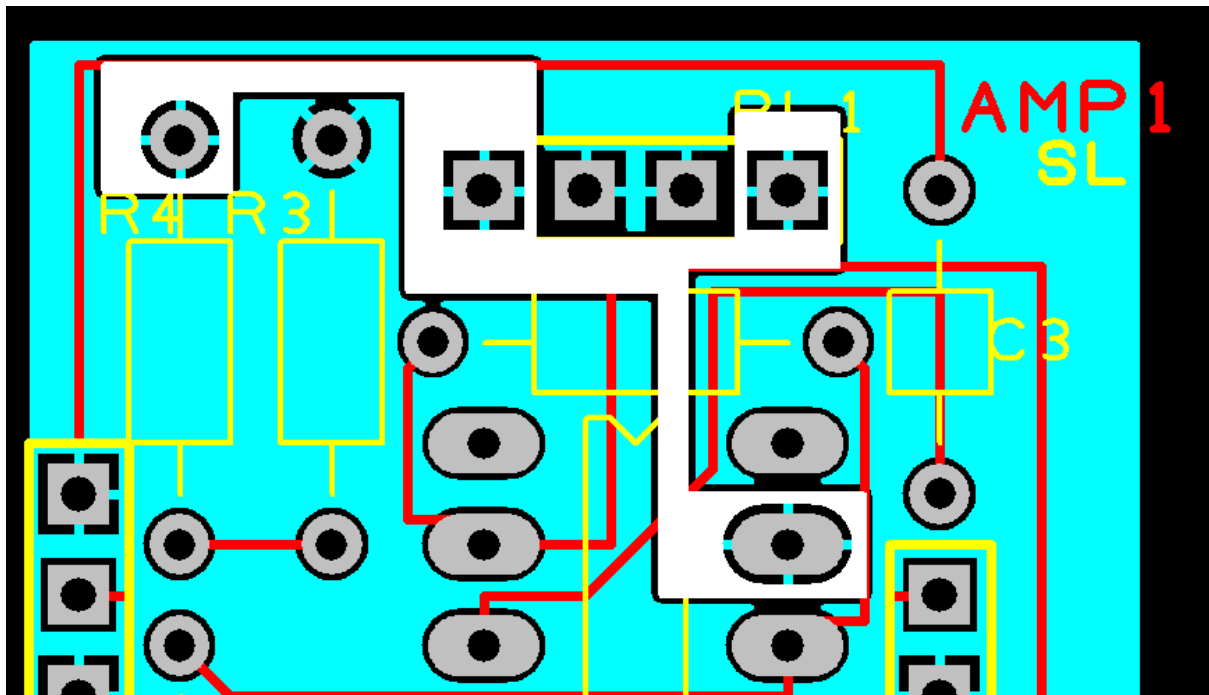
PCB



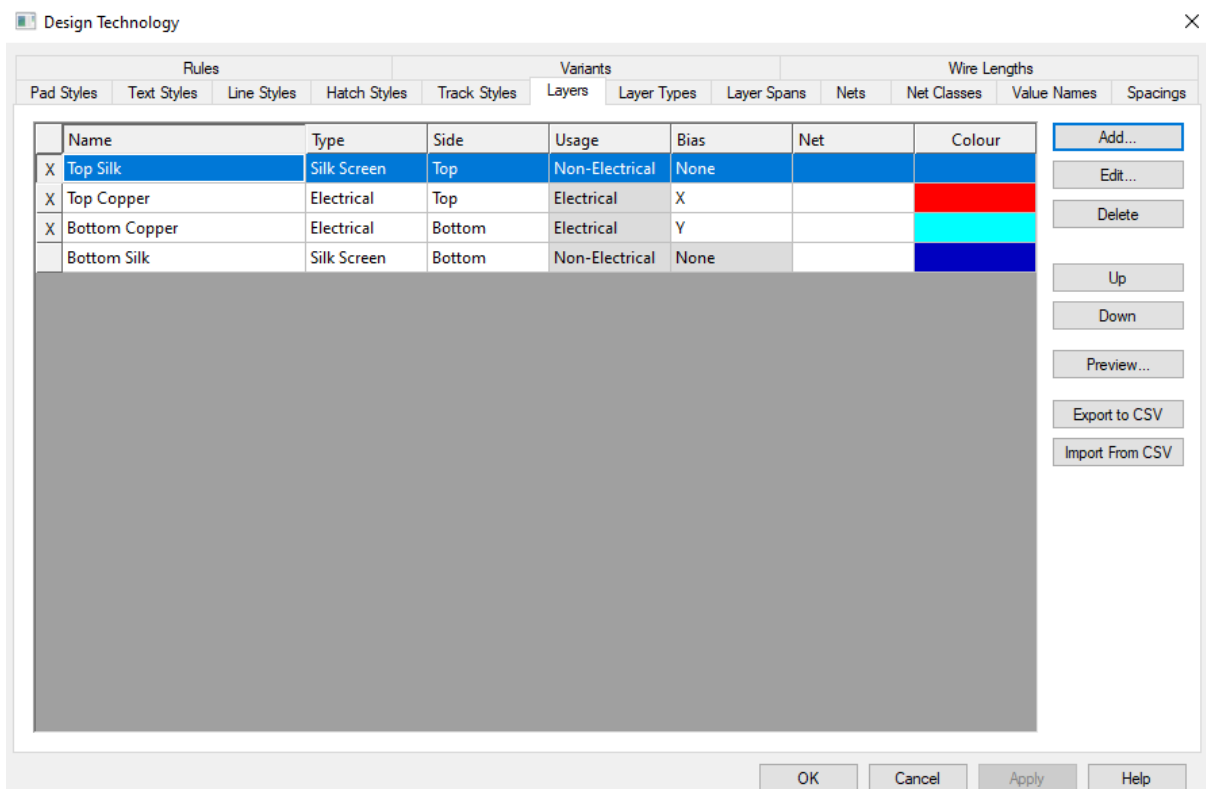
Had to link all the components together without touching the metal contacts so I had to carefully think about which route to take for each component.



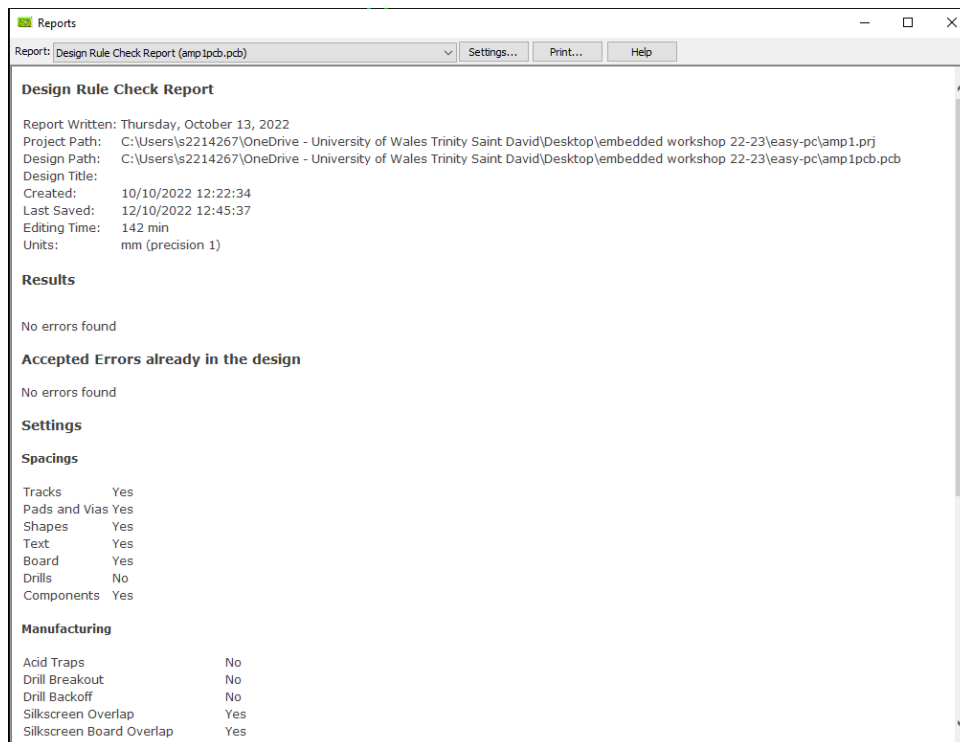
This is the original position of my components. I have placed them in this way because I needed everything to be as close as possible to each other to increase the integrity of the board since a smaller board is physically stronger.



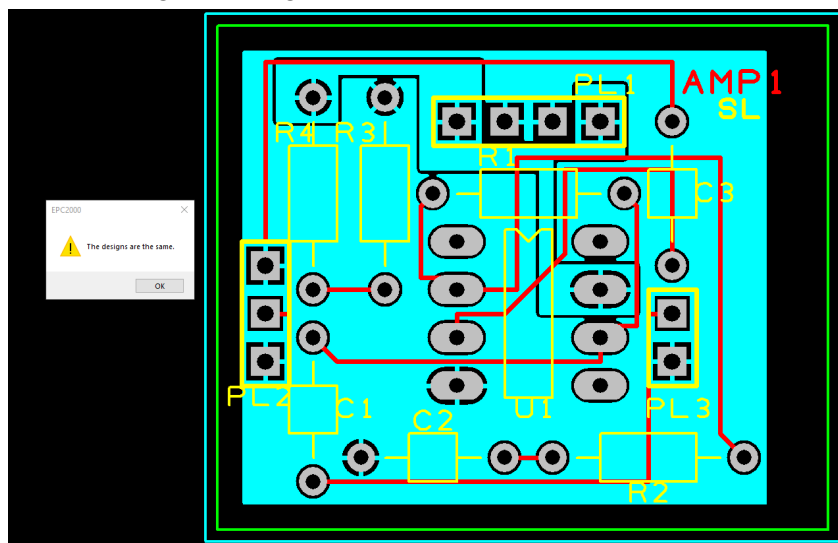
highlighted in white is a specifically poured copper of +V charge specifically encasing all the +V components. Everything else is in the light blue 0V charged copper.



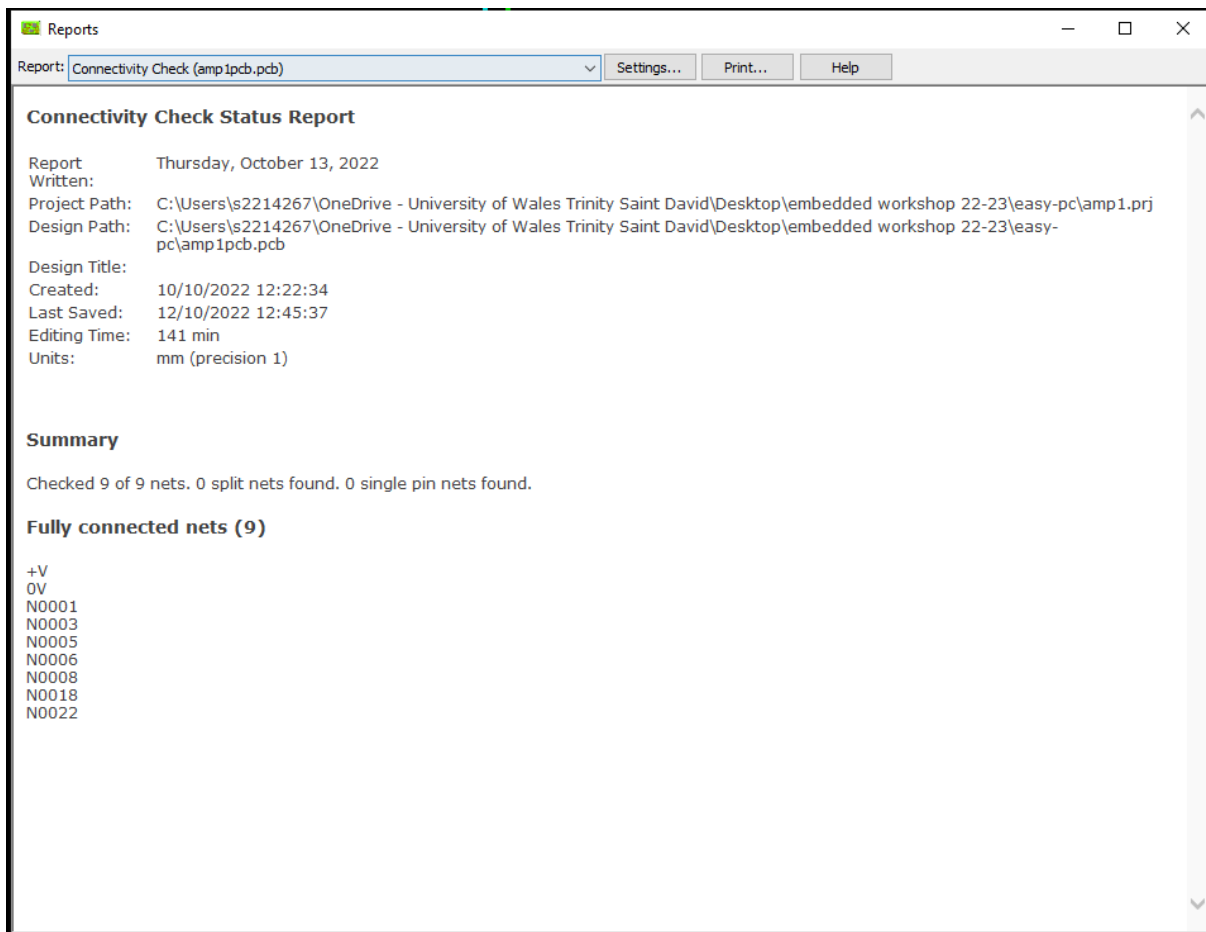
I used the design technology tool to remove unwanted layers so I could do the copper pour.



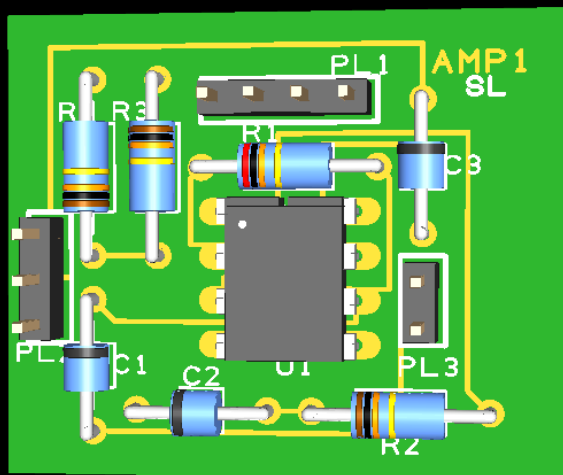
Created the pcb making it as space efficient as i could and ran the checks that all had 0 errors. Originally there were 14 errors however they were all an easy fix. I had to move the names closer to components and rapport the copper. That left me with 0 errors remaining after passing the design rule check report.

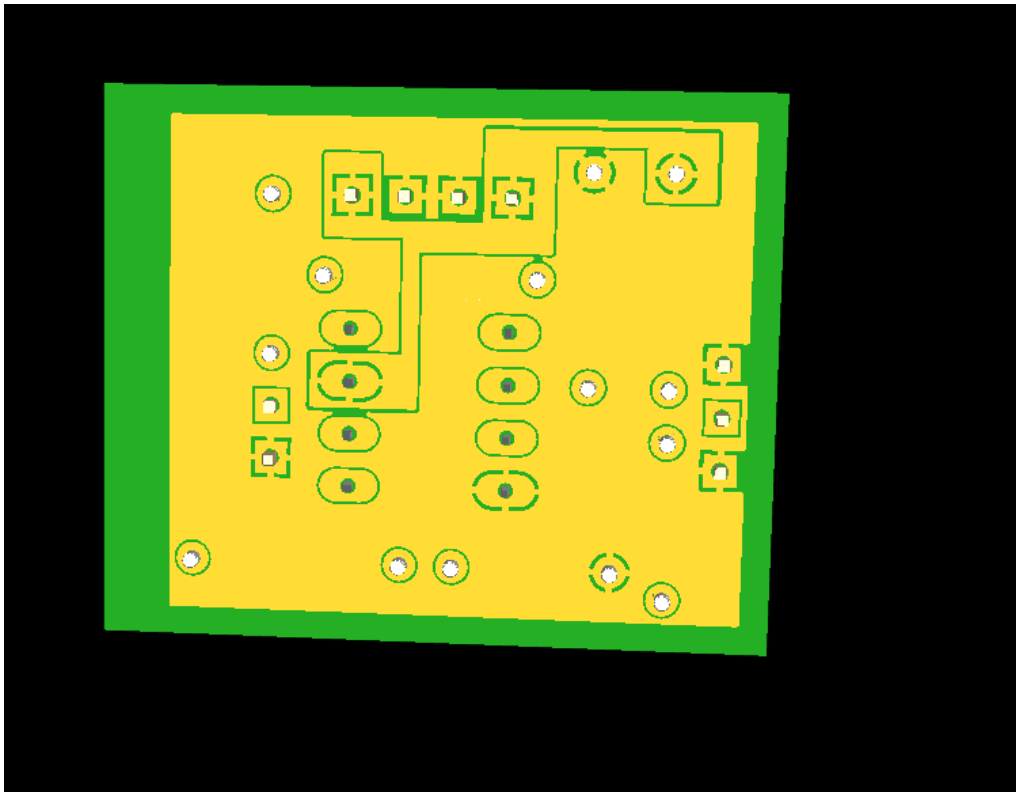


When I did the copper pour I made sure everything was included inside it by having no gaps.



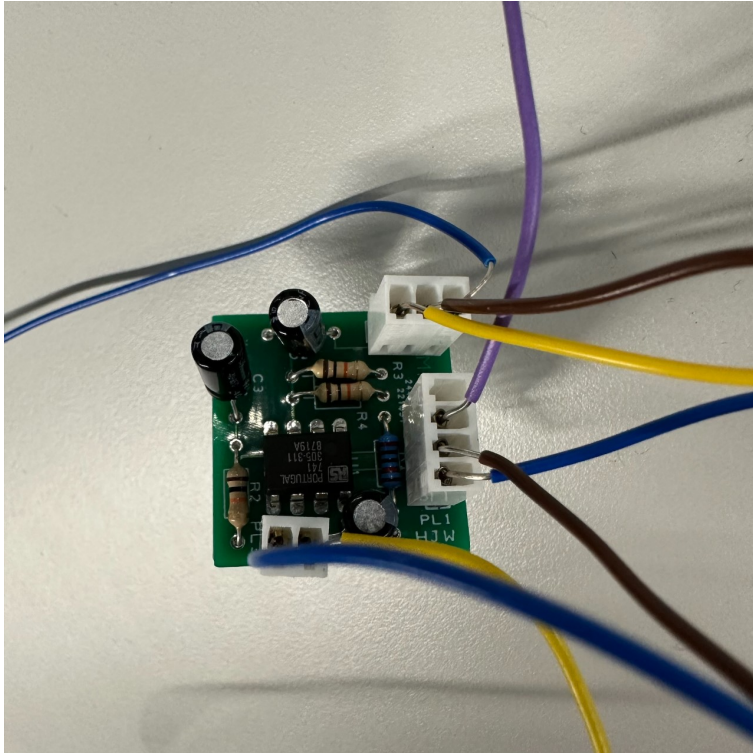
I optimised nets by pressing the “optimise nets” tool button, then manually connected the remaining ones to where they are meant to go. I had to make sure I didn't cross over any existing routes so it made it quite challenging.



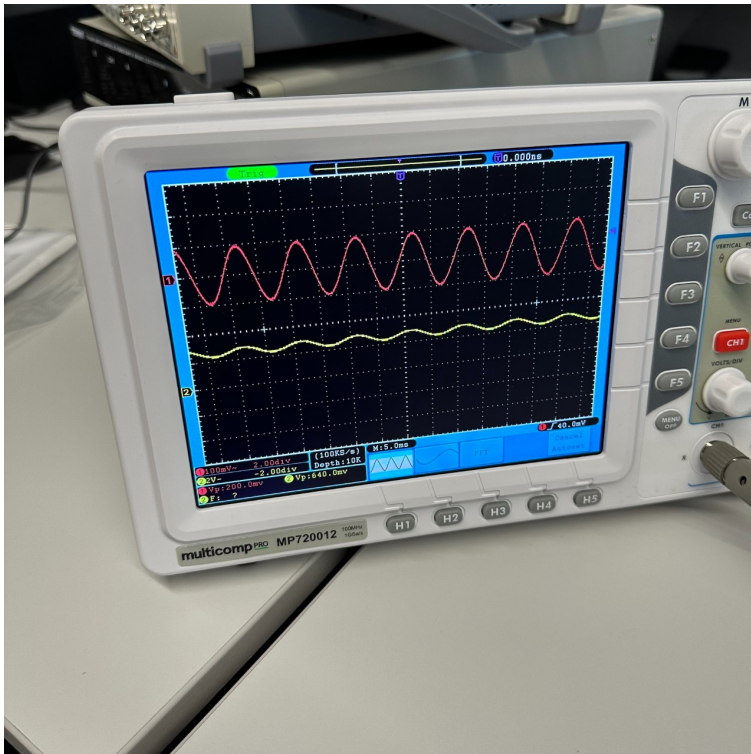


I viewed it in 3d to see how it would look in person and to visualise how everything fits together.

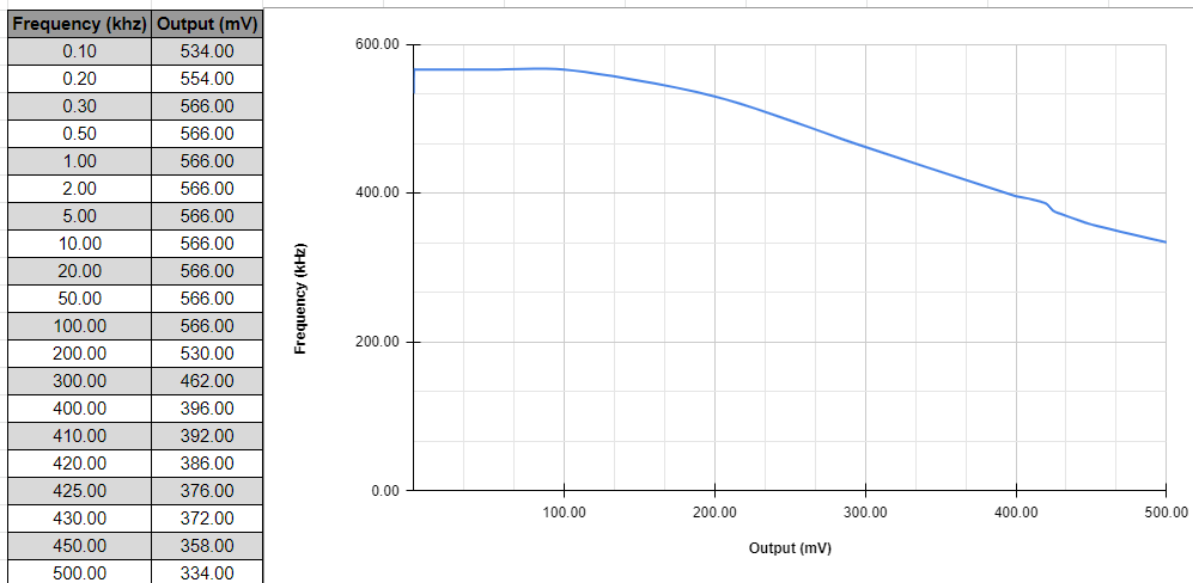
PCB Results



I connected the wires to the pcb via the pins that i soldered onto the board.

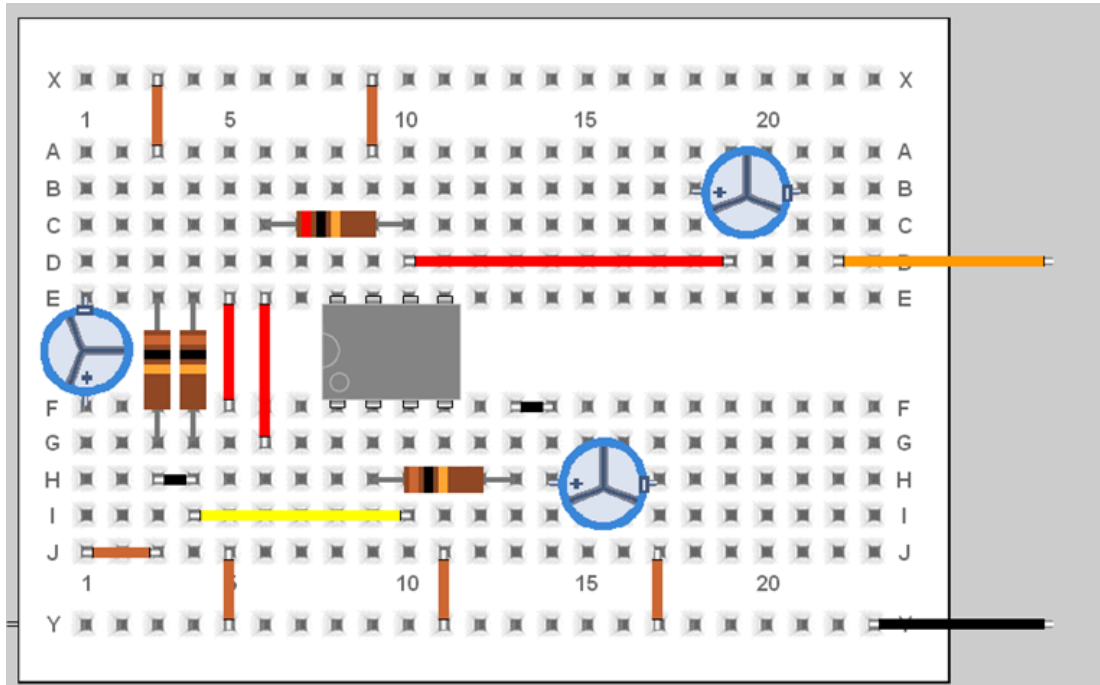


I then connected the board to the oscilloscope to see the waves and take measurements.

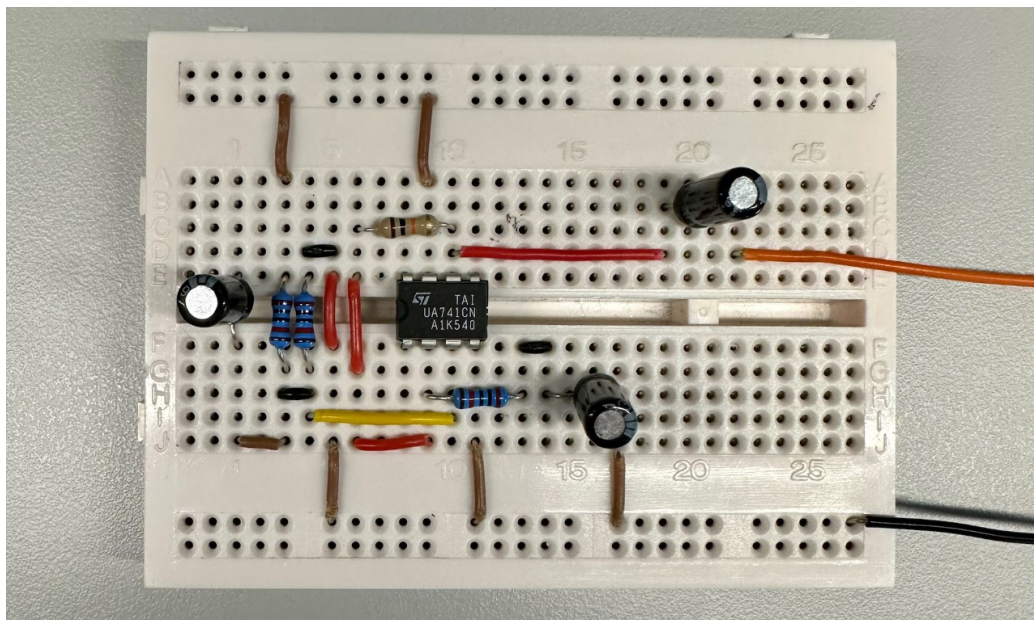


I created a table and a graph of the results for the PCB and compared it to the breadboard and simulation.

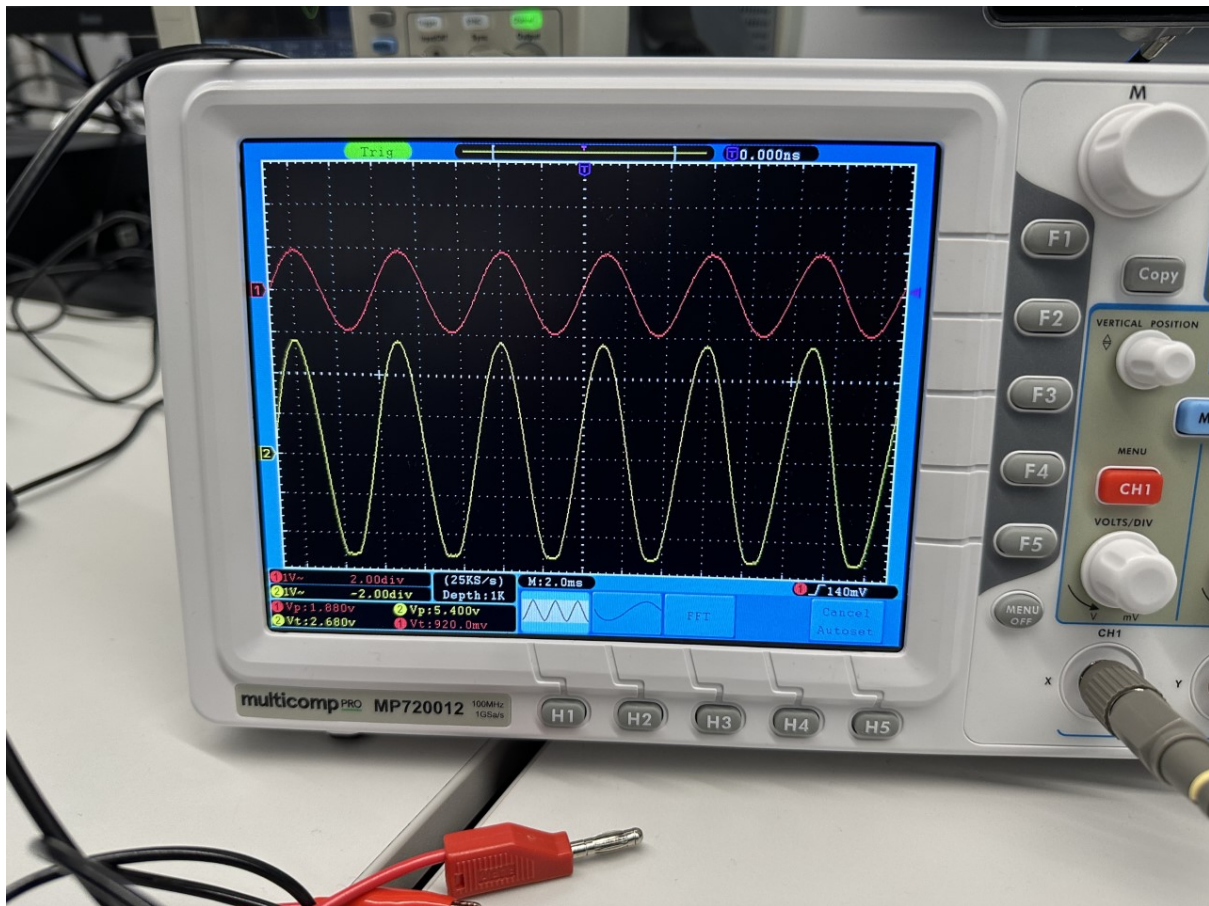
Breadboard



I created an online version to map out where everything goes then I created it using a real breadboard.

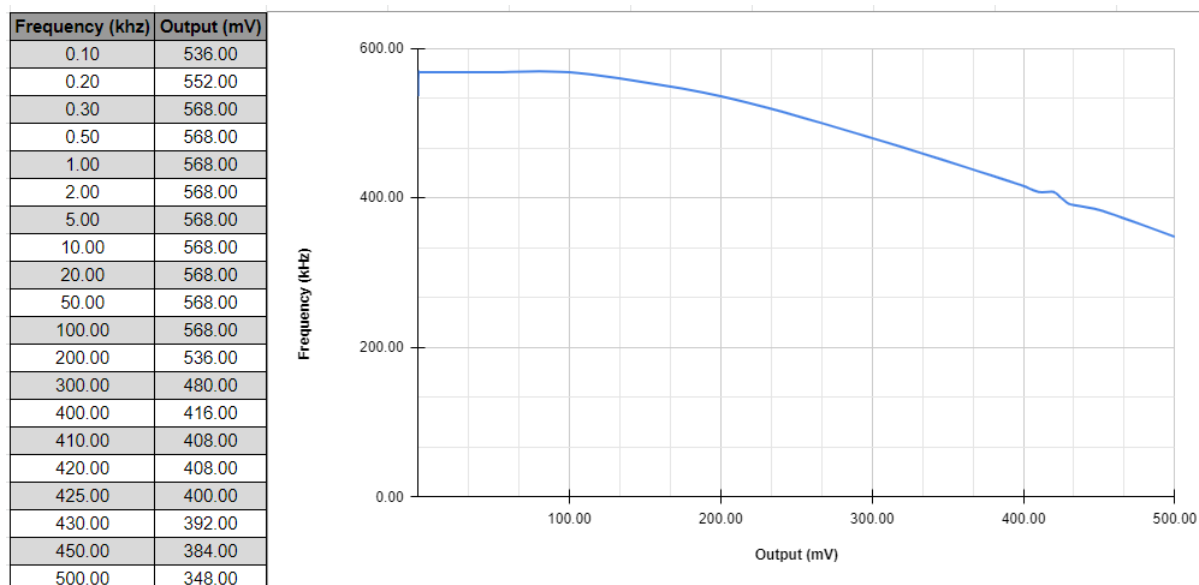


I created a breadboard for the circuit. It started as an online render that I had to convert into a physical form using wires and components. I made it as neat as possible to make it easier to fix if anything went wrong.

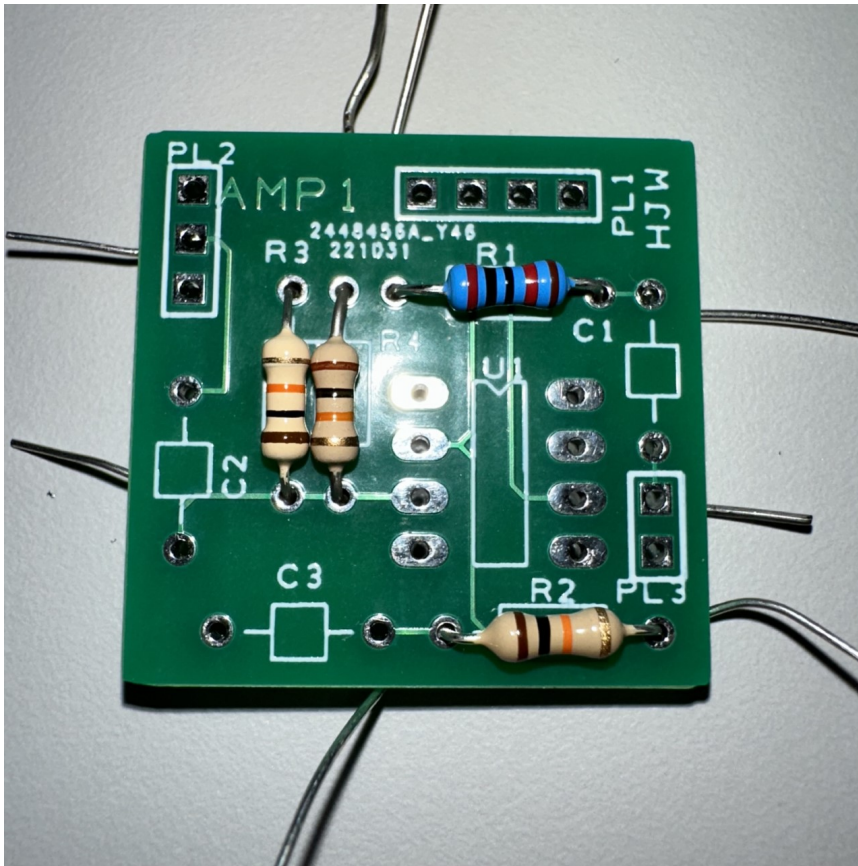


The signal generator confirms a gain of 3 which is what the intention was.

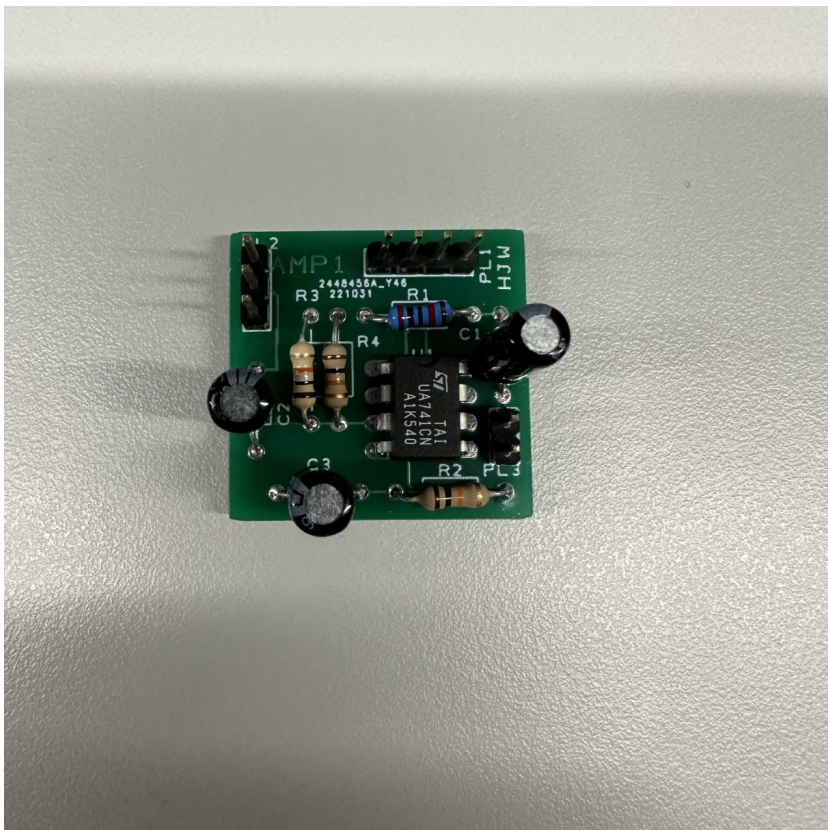
Breadboard Results

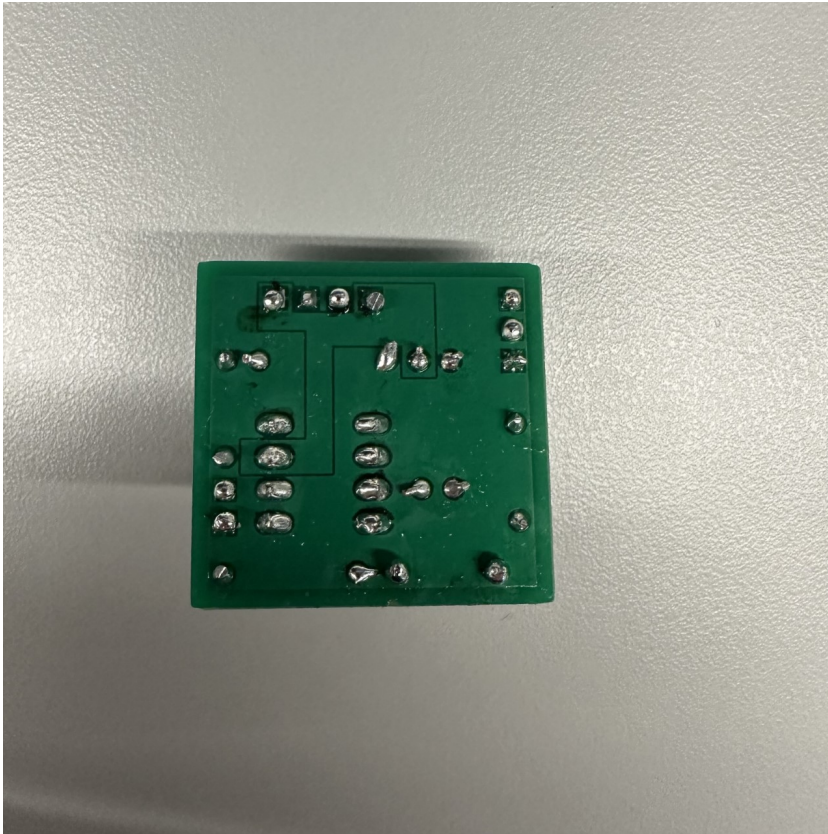


I made a table and plotted results for different frequencies then put them into a graph.



Firstly, I installed the resistors as they are shorter so it is easier to solder into the pcb.





I then soldered the resistors onto the board followed by the capacitors and the chip.