ELTN 115 Paul Sandeen
Spring 2020 10 June 2020

PCB Design Project: Display Clock

INTRODUCTION

Electronics 115 is a course in Printed Circuit Board (PCB) design and fabrication offered at Pasadena City College, a two-year community college located in the city of Pasadena, north of Los Angeles California. PCBs are widely used and are found in nearly all electronic devices to provide a physical "platform" to hold discrete electronic components and to provide an electrically conductive path between them. The course teaches students practical, hands-on techniques for the production of PCBs. As part of the course, students were given an electronic circuit to capture and design a PCB layout using a computer-aided design (CAD) program known as Eagle by the company AutoDesk. After the board design was complete, the design files were sent to a PCB prototype production service provider, PCBWay (pcbway.com) with their factory located in Shenzhen, China.

PROJECT DESCRIPTION

The PCB will act as a display clock to show the current time. Light-emitting diodes (LEDs) are located at the traditional hour positions on a clock face (1 o'clock, 2 o'clock, etc.) to indicate hour, while minutes are displayed as numbers on two 7-segment displays. Time is kept by using the 60 Hz AC signal supplied by the utility power grid.

Time is measured by a special type of computer known as a microcontroller. Unlike a traditional desktop or laptop PC that has discrete components for CPU, memory and local storage, a microcontroller places all of these into a single integrated circuit chip package. The microcontroller selected for the project is the Atmel ATMEGA328P, the same chip used in the popular Arduino Uno maker platform.

The clock has an alarm function, allowing the user to set a time for an alarm to sound. The clock also plays the "Westminster Quarters", a series of melodies played on the quarter hour. A switch on the PCB allows the user to disable the alarm and the playing of the Westminster Quarters.

EAGLE CAD DESIGN PROCESS

The Eagle CAD package provides a two-stage process for PCB design. First, the schematic is laid-out in the Schematic Editor window. Electronic parts are selected from a large component library and placed onto the Schematic Editor window. With the parts in place, they are connected using Nets (Networks). With all the parts in place and connected, an ERC (Electrical Rule Check) is performed to make sure all the required connections are made and no connections are misting.

The second step is the layout of the PCB. The window switches from Schematic Editor to Board Editor, with an outline of the PCB visible and all components located outside and to the left of the board. At this point the connections between components are called “Air Wires” – representations of how the components are connected, but not the final electrical traces that will be on the PCB. The shape and dimension of the PCB can be altered (for this project, the shape was changed from rectangular to round) and the parts are moved from outside the PCB area to inside. With all the components placed, the traces can be routed using the Autorouter; when the Autorouting is complete, a DRC (Design Rule Check) is performed to verify trace widths and component spacing.

Designing a PCB layout is part art and part science. One should design a PCB that looks organized and consistent, but there are basic formal design rules to observe:

1. The “flow” of the components on the board should locate inputs on the left side of the board, and outputs on the right. Inputs could mean incoming AC signals, high voltage signals, or signals that need to be processed. Outputs could mean outgoing DC signals, lower voltage signals, or signals that have been processed by the board and are ready to send to another board or system. Following the “flow” of the board helps field technicians troubleshoot the board by taking a step-by-step, input-to-output approach.
2. Electronic parts that handle high voltage should be located away from parts that accept lower voltage in order to avoid cross-talk where high voltages interfere with lower voltage signals.
3. Heat can damage electronic parts. Parts that operate at high temperatures should be isolated and kept as far as possible from other parts to prevent high temperature parts from causing damage to the other parts. Heat sinks should be attached to parts that generate heat to aid in heat dissipation.

4. Design for Manufacturing (DFM): While designing the PCB, keep in mind that the board must be able to be manufactured. Allow spacing between components to allow soldering them to the board. Be consistent with the orientation of diodes, capacitors, and LEDs. Avoid part overlap. Remember that heat sinks or other mounting hardware may be required for certain parts, so provide adequate spacing.
5. Make copious use of text: Eagle CAD provides a silkscreen text layer for both the top and bottom sides of a PCB. Important parts should be labeled, especially parts the user may need to adjust like buttons or knobs, pinhead connectors, or screw terminals.

PROJECT HARDWARE

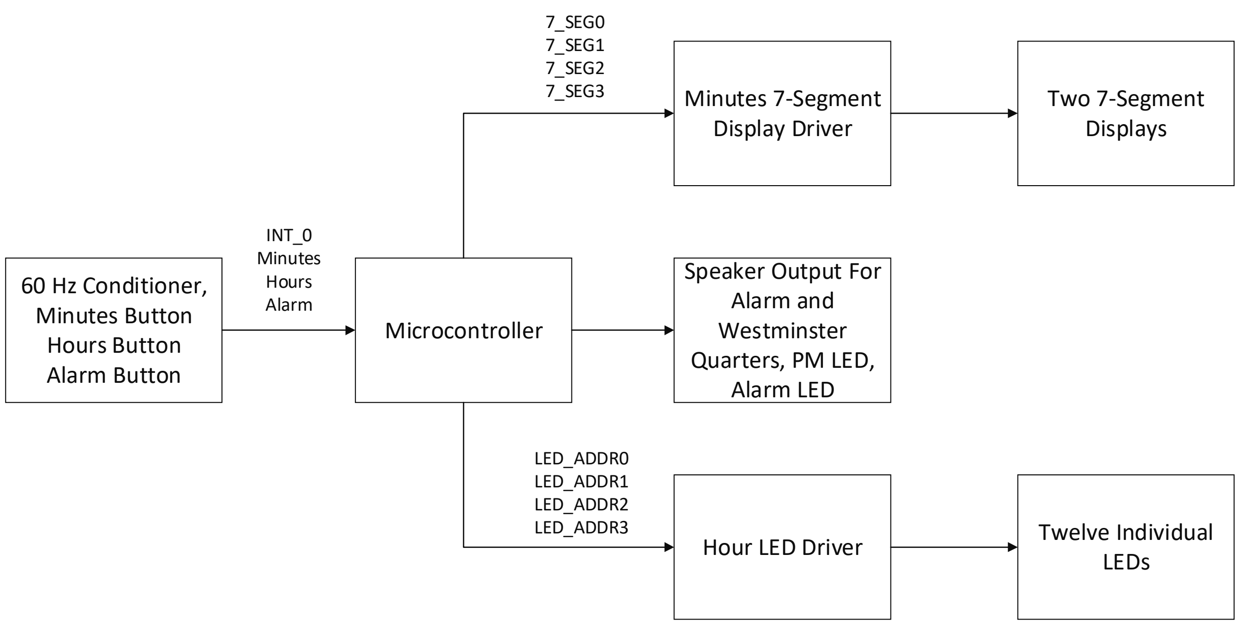

Figure 1 *– Block Diagram of the Major Subsystems of the Clock*

The clock's hardware can be broken up into sections containing the subsystems.

60 Hz Conditioner, Hours Button, Minutes Button, Alarm Button: The 120 VAC power coming from the utility grid is stepped down using a wall-mount transformer to 12 VAC. The connections from the transformer are fed into an optocoupler (OK1, H11L1M) that electrically isolates the incoming AC signal and uses a Schmitt trigger to send out a signal 60 times per second to Digital Pin 2 on the Microcontroller. There is also a bridge rectifier (B2, B-DIL) that changes the AC signal to a pulsating DC signal. A 470 uF capacitor smooths the pulsating DC to a more steady DC signal with some ripple variations, and the 7805 voltage regulator regulates the voltage to a steady 5 VDC.

There are three inputs buttons for setting the clock time Hours and Minutes and to set the Alarm to adjust the Alarm time Hours and Minutes.

Microcontroller Circuit: The Atmel ATMEGA328P is the DIP microcontroller that processes the 60Hz signal from the optocoupler, sends the signal to the Hour LEDs and Minutes 7-segment displays, the PM LED, the Alarm LED, the Alarm sound output, and other headers for possible future use. There is also a 16 MHz crystal to provide timing for the internal operation of the microcontroller.

LED Driver: The twelve individual LEDs used to indicate the hour are powered by two 74LS138N decoders that take the signal LED_ADDR0 through LED_ADDR3 and decodes it to a 0 through 11 value that will illuminate one LED.

7-Segment Display Driver: Two 74HC4543 BCD to 7-segment latch decoder drivers illuminate the LEDs for two 7-segment displays. This particular IC is a latching decoder, which means that the Load (LD) pin (pin 1) must see a logic LOW to logic HIGH transition to take the BCD value presented at the input pins and decode it to power the LEDs of the 7-segment display. The LEDs would burn out due to the current coming out of this IC, so a 330 ohm resistor network is provided to limit the current.

Individual I/O: There are outputs for the PM LED (to indicate the time is in PM Hours), Alarm LED (to indicate the Alarm is active) and to play the Westminster Quarter melody on the quarter hour on a speaker.

SPECIAL DESIGN CONSIDERATIONS

The student was given Microsoft Word documents that showed a design for a particular subsystem of the clock. The student would then capture the schematic with Eagle Schematic Editor, and once completed, proceed to Eagle Board Editor to perform the PCB layout. During the design, there were several considerations that had to be made.

The width of board traces is important as it relates to their current-carrying ability, known as *ampacity*. Power and ground traces had to be wider than signal traces. Power traces were set to 20 mils, while signal traces were set to 15 mils. Also, the spacing between the traces was important, as good spacing prevents signal crosstalk, and was set to 10 mils.

Figure 2 *– The Net Classes Rules Tab*

The settings for Net classes in Eagle Schematic View is found under:
Edit > Net classes … menu item; in the Net classes dialog box, select the Rules tab, or by typing Class in the editor window.

Once the PCB layout is complete, the traces must be routed between components. While this can be done manually, the purpose of using CAD software is to help the designer with tedious tasks, so an Autorouter function is present in the Eagle Board Editor. The Autorouter produces multiple board route patterns with alternative routes that differ from each other in the number of vias (through-hole connections that pass from the top side of the board to the bottom) produced. Ideally, the number of vias should be minimized since each via requires a drilling operation, which can add to the price to manufacture a board.

A challenge with routing a board is that if a component must be moved to a different location, all traces on the board must be undone (called a Ripup operation) and the board re-routed. The routing of a board is a computationally-intensive task, and can take several minutes even on modern PC hardware.

PCBs must be attached to an enclosure through mounting holes. The initial board layout did not include mounting holes, and adding mounting holes required a significant re-design of the board. Likewise, since the hole must accommodate mounting hardware, there must be space around the hole for a washer, lock nut, or screw head, and this requires a restriction called tRestrict (top of the board, layer 41) and a bRestrict (bottom of the board, layer 42) to prevent traces from being located too close to the hole.

There are several user-configurable features on the clock PCB, including a switch to turn the Alarm on or off, buttons to adjust the clock Hours and Minutes, and to adjust the Time Hours and Minutes. To aid the user, these buttons are labeled using the silkscreen on the top (tPlace, layer 21) and the bottom (bPlace, layer 22). The goal is to make the board self-documenting in order to make regular use and field service simple.

<u>ASSEMBLY PROCESS</u>

Once the PCB design was finalized, the CAD/CAM files were sent to the board manufacturer for production. When the board was delivered, the student inserted the components onto the board and hand-soldered them, a process that requires manual dexterity and good hand-eye coordination. Since this was the student's first PCB design, great care was given to testing to make sure components were labeled correctly, as were their mounting orientations.

After each project photo below are design comments that provide suggestions to improve the design or point out errors in the design that should be corrected.

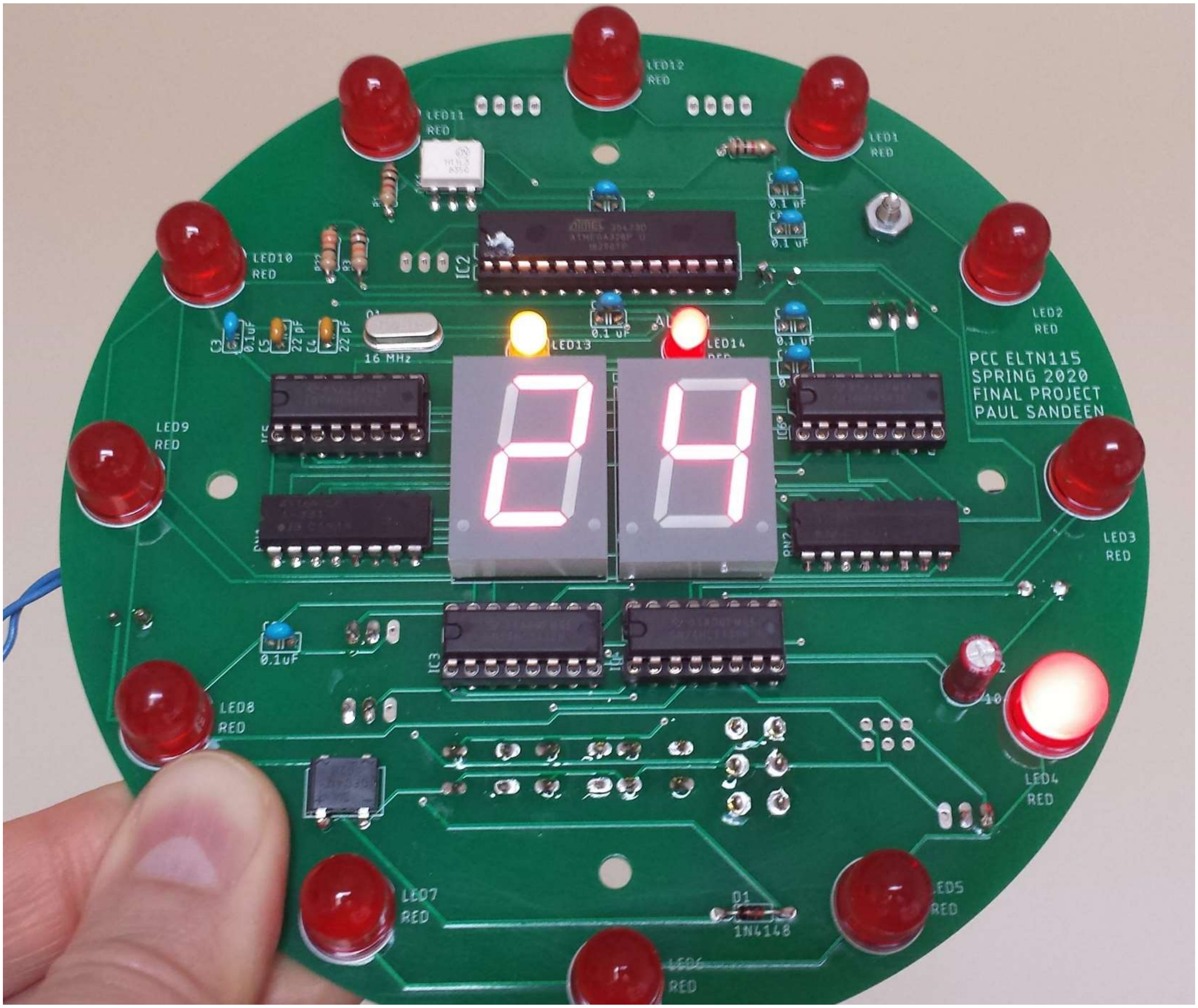

Figure 3 *– Top of Assembled and Running Clock PCB Displaying 4:24 PM with the Alarm Setting Active*

Design Comments: The tall 10 uF capacitor is too close to the 4 o'clock LED and should be located on the bottom side of the board since it is used for the speaker. The large LEDs and 7-segment displays are very bright in a dark room and need a dimming feature.

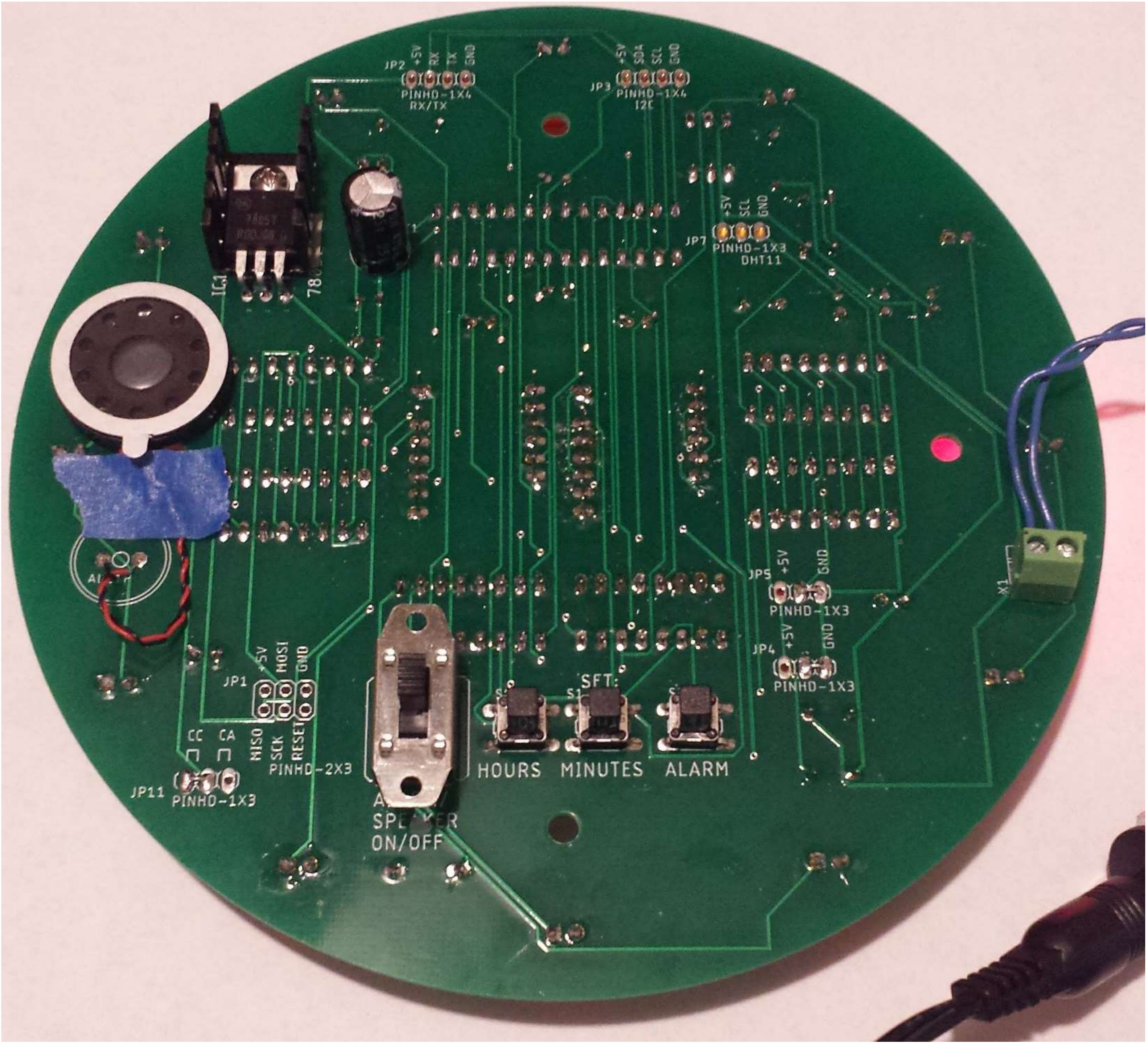

Figure 4 *– Bottom of Assembled and Running Clock PCB*

Design Comments: The power connector is facing up toward the top of the board and should be located out toward the right side. The 470 uF capacitor could be moved further from the heatsink of the 7805 voltage regulator to prevent it from picking up heat. The heatsink on the 7805 puts out a lot of heat and replacing the 12VAC wall transformer with a lower voltage (7VAC or 9VAC) might help.

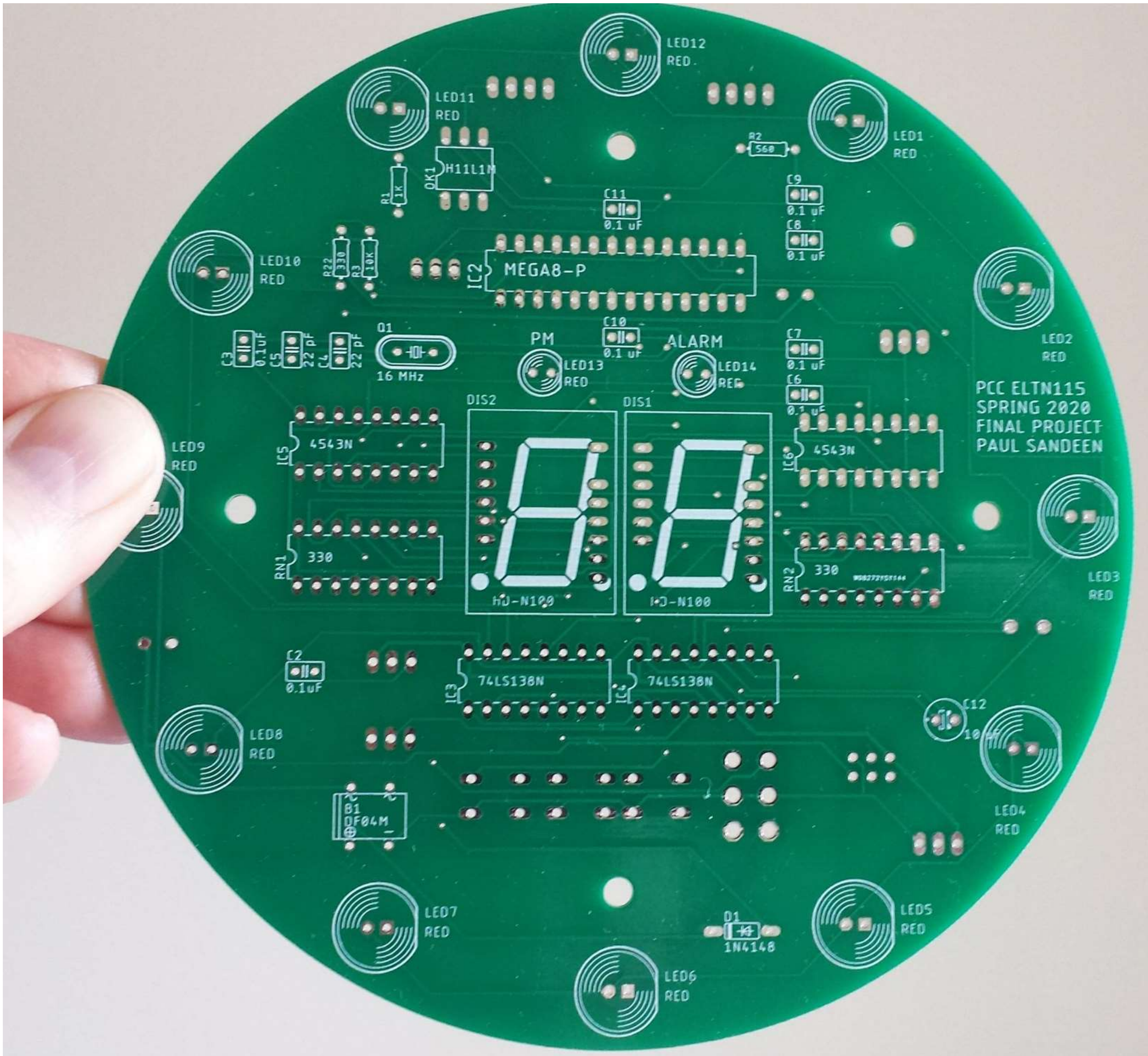

***Figure 5** – Top of Pre-assembled PCB*

Design Comments: Silkscreen errors were found on capacitor C12 which overlaps the footprint for LED 4. The power components (the DF04M bridge rectifier and the H11L1M optocoupler) should be located closer together. It would be preferred if the through-holes for resistors and capacitors had pads like the ICs and diode D1 (1N4148).

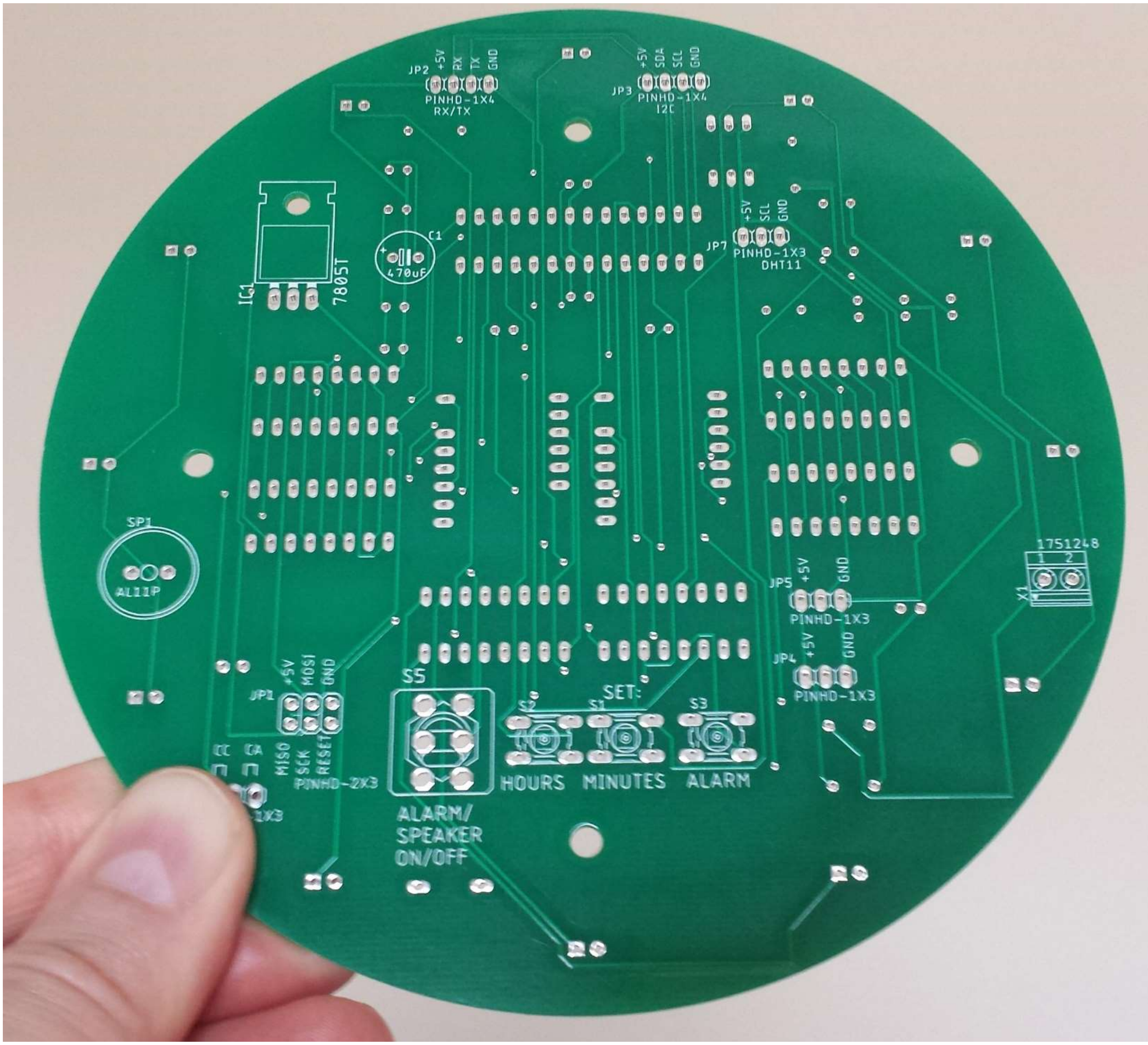

Figure 6 *– Bottom of Pre-assembled PCB*

Design Comments: The silkscreen labels for 5V and Ground on jumpers JP4 and JP5 are incorrect and show reversed polarity (5V should be labeled GND and GND should be labeled 5V). For jumper J11, the Common Cathode (CC) should be labeled Common Anode (CA) and Common Anode (CA) should be labeled Common Cathode (CC).

SUMMARY

PCB design requires a diverse skill set: a knowledge of electronics, considerations on how parts physically fit together in a confined area, and heat dissipation issues. Making the board aesthetically pleasing or at least well laid-out and organized is important. Also, a knowledge of the manufacturing process is import. While soldering the board, it became clear that some parts were too close together to be soldered easily (the large 7-segment displays can "get in the way" of soldering other components).

The most useful feature to improve the board is the ability to dim the LEDs and 7-segment displays. In a dark room (like a home office) the lights were simply too bright.

It would also be helpful to enable the alarm function without also enabling the Westminster Quarters, which get annoying in a very short period of time.

The big lesson aside from the others listed is to double-check silkscreen text used to label configurable parts. It is not clear how inverting 5V and GND occurred on jumpers JP4 and JP5, but both of these jumpers are on the bottom of the board, and it is possible that when moving the jumpers from the top to the bottom of the board that the student forgot the pin orientation and labeled them backwards.

<u>APPENDIX</u>

Bill of Material (BOM)

Item	Part	Value	Device	Package	Description
1	B1	DF04M	B-DIL	B-DIL	RECTIFIER
2	C1	470uF	CPOL-USE3.5-8	E3,5-8	POLARIZED CAPACITOR, American symbol
3	C2	0.1uF	C-US025-025X050	C025-025X050	CAPACITOR, American symbol
4	C3	0.1uF	C-US025-025X050	C025-025X050	CAPACITOR, American symbol
5	C4	22 pF	C-US025-025X050	C025-025X050	CAPACITOR, American symbol
6	C5	22 pF	C-US025-025X050	C025-025X050	CAPACITOR, American symbol
7	C6	0.1 uF	C-US025-025X050	C025-025X050	CAPACITOR, American symbol
8	C7	0.1 uF	C-US025-025X050	C025-025X050	CAPACITOR, American symbol
9	C8	0.1 uF	C-US025-025X050	C025-025X050	CAPACITOR, American symbol
10	C9	0.1 uF	C-US025-025X050	C025-025X050	CAPACITOR, American symbol
11	C10	0.1 uF	C-US025-025X050	C025-025X050	CAPACITOR, American symbol
12	C11	0.1 uF	C-US025-025X050	C025-025X050	CAPACITOR, American symbol
13	C12	10 uF	CPOL-USE2,5-6E	E2,5-6E	POLARIZED CAPACITOR, American symbol
14	D1	1N4148	1N4148DO35-7	DO35-7	DIODE
15	DIS1	HD-N100	HD-N100	HDSP-Q	LED DISPLAY
16	DIS2	HD-N100	HD-N100	HDSP-Q	LED DISPLAY
17	IC1	7805T	7805T	TO220H	Positive VOLTAGE REGULATOR
18	IC2	MEGA8-P	MEGA8-P	DIL28-3	MICROCONTROLLER
19	IC3	74LS138N	74LS138N	DIL16	3-line to 8-line DECODER/DEMULTIPLEXER
20	IC4	74LS138N	74LS138N	DIL16	3-line to 8-line DECODER/DEMULTIPLEXER
21	IC5	4543N	4543N	DIL16	BCD to 7-segment LCD DECODER
22	IC6	4543N	4543N	DIL16	BCD to 7-segment LCD DECODER
23	JP1	PINHD-2X3	PINHD-2X3	2X03	PIN HEADER
24	JP2	PINHD-1X4	PINHD-1X4	1X04	PIN HEADER
25	JP3	PINHD-1X4	PINHD-1X4	1X04	PIN HEADER
26	JP4	PINHD-1X3	PINHD-1X3	1X03	PIN HEADER
27	JP5	PINHD-1X3	PINHD-1X3	1X03	PIN HEADER
28	JP7	PINHD-1X3	PINHD-1X3	1X03	PIN HEADER
29	JP11	PINHD-1X3	PINHD-1X3	1X03	PIN HEADER
30	LED1	RED	LED10MM	LED10MM	LED
31	LED2	RED	LED10MM	LED10MM	LED
32	LED3	RED	LED10MM	LED10MM	LED
33	LED4	RED	LED10MM	LED10MM	LED

34	LED5	RED	LED10MM	LED10MM	LED
35	LED6	RED	LED10MM	LED10MM	LED
36	LED7	RED	LED10MM	LED10MM	LED
37	LED8	RED	LED10MM	LED10MM	LED
38	LED9	RED	LED10MM	LED10MM	LED
39	LED10	RED	LED10MM	LED10MM	LED
40	LED11	RED	LED10MM	LED10MM	LED
41	LED12	RED	LED10MM	LED10MM	LED
42	LED13	RED	LED5MM	LED5MM	LED
43	LED14	RED	LED5MM	LED5MM	LED
44	OK1	H11L1M	H11L1M	DIL06	6-Pin DIP Optocoupler
45	Q1	16 MHz	CRYSTALHC49S	HC49/S	CRYSTAL
46	R1	1K	R-US_0204/7	0204/7	RESISTOR, American symbol
47	R2	560	R-US_0204/7	0204/7	RESISTOR, American symbol
48	R3	10K	R-US_0204/7	0204/7	RESISTOR, American symbol
49	R22	330	R-US_0204/7	0204/7	RESISTOR, American symbol
50	RN1	330	8R-N	DIL16	DIL RESISTOR
51	RN2	330	8R-N	DIL16	DIL RESISTOR
52	S1	N/A	10-XX	B3F-10XX	OMRON SWITCH
53	S2	N/A	10-XX	B3F-10XX	OMRON SWITCH
54	S3	N/A	10-XX	B3F-10XX	OMRON SWITCH
55	S5	N/A	M9040P2	M9040P2	TOGGLE SWITCH
56	SP1	AL11P	AL11P	AL11P	SPEAKER Source: Buerklin
57	X1	1751248	1751248	1751248	MKDS 1/ 2-3,5 Printklemme

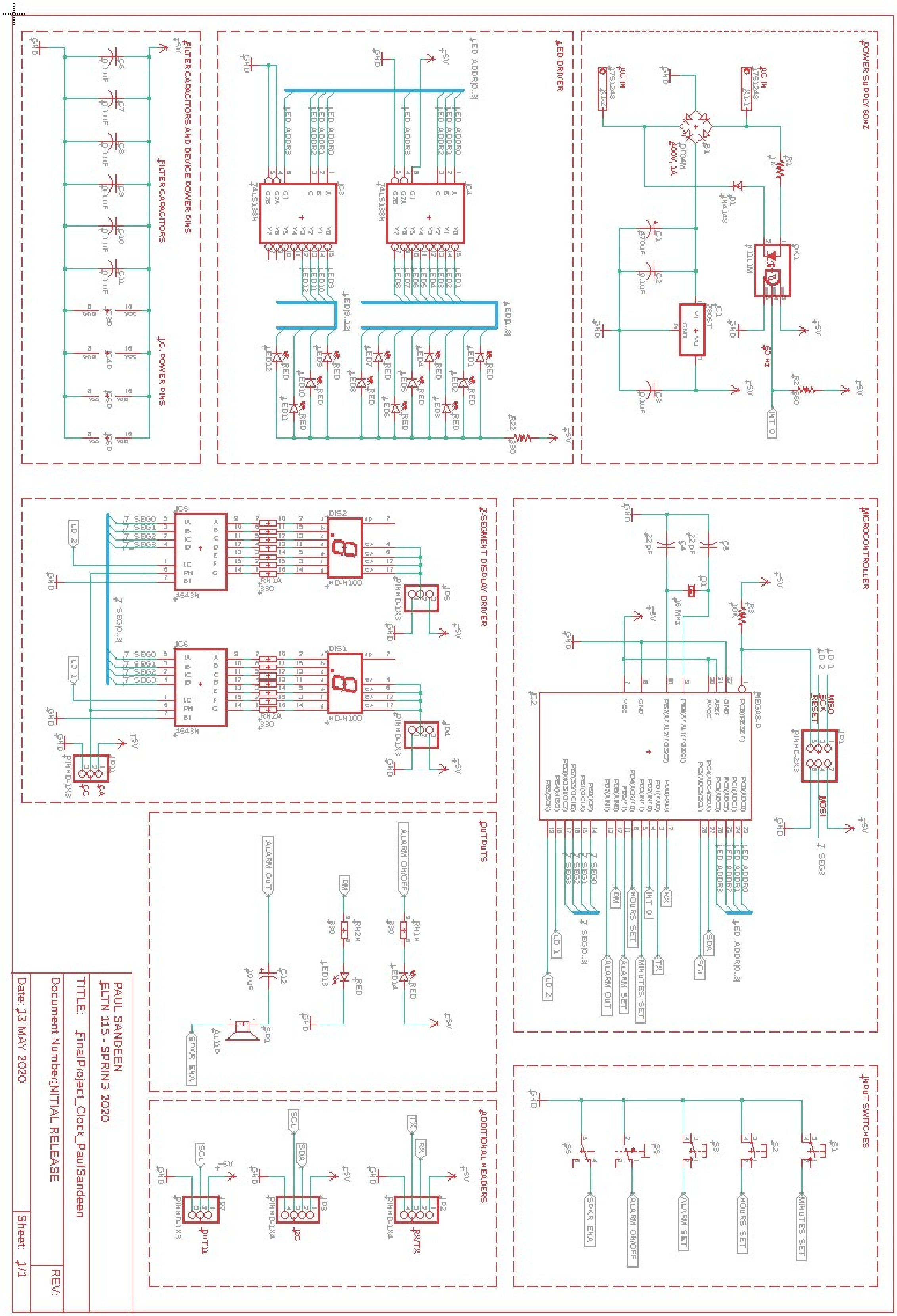

***Figure 7** – Clock PCB Design Schematic*

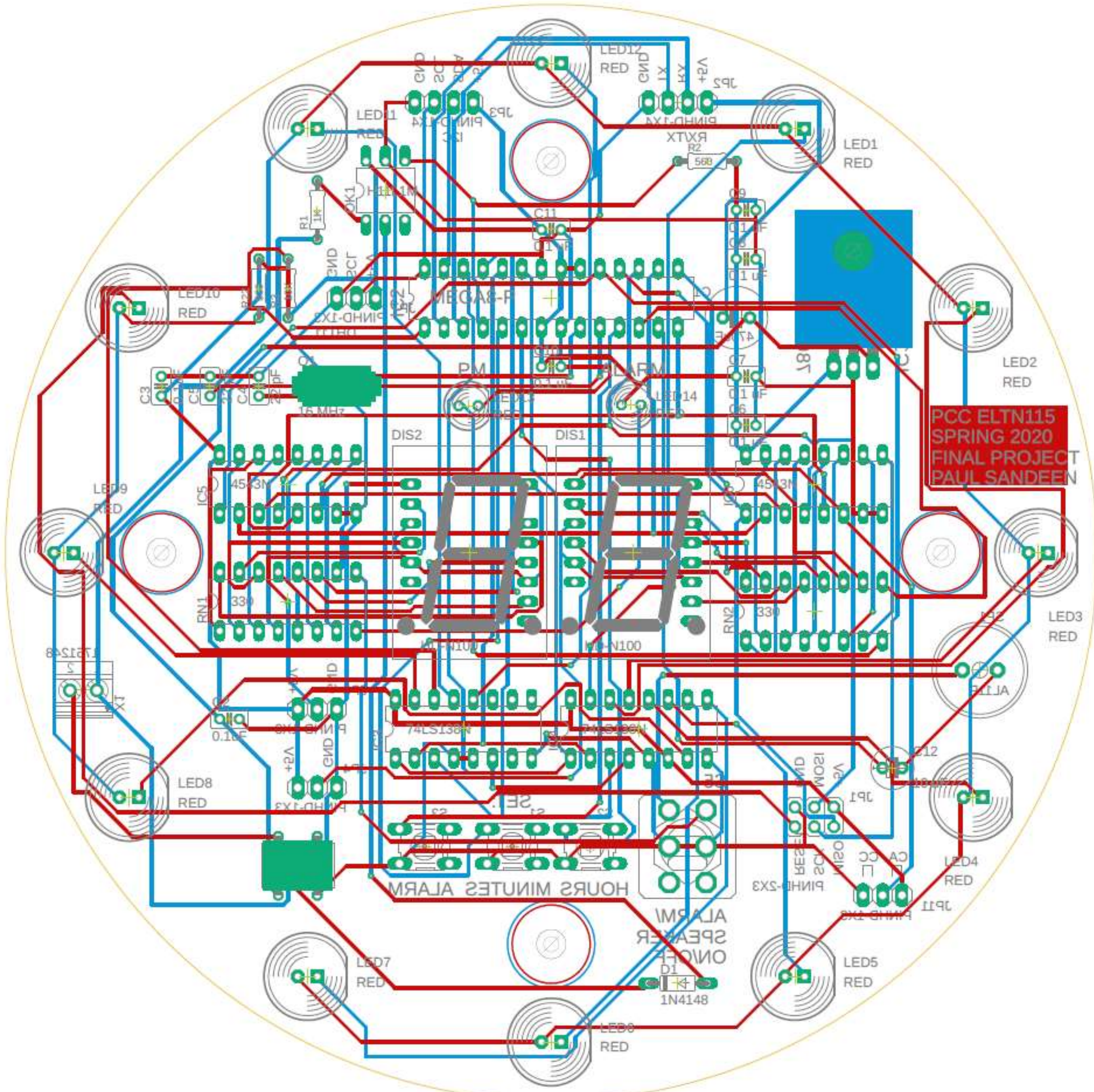

***Figure 8** – Clock PCB Design Layout*