

C

Creating another Printed Circuit Board

In this chapter, we will learn the following to World Class standards:

- **Starting with a Finished Schematic**
- **Creating the Layers for the Printed Circuit Board**
- **Drawing the Components and Pads**
- **Drawing the Electrical Traces**
- **Produce a Silkscreen Artwork**
- **Produce the Pads and Trace Artwork**
- **Produce a Drill Pattern**
- **Revise the Schematic to Agree with the PCB**

Starting with a Finished Schematic

Now that the Printed Circuit Board (PCB) schematic shown in Figure C.1 is completed, we can begin to lay out the board. If we did not draw the schematic ourselves after finishing chapter 6, we might want to accomplish that task before moving towards making the circuit board. In addition, the chapter on the transmitter and receiver does contain the datasheets that will be essential in making a working product. When we wrap up making the fiberglass card that contains the electrical circuits, we will return to this schematic and change the reference designation, since the location of the device on the electronic assembly determines the component's number.

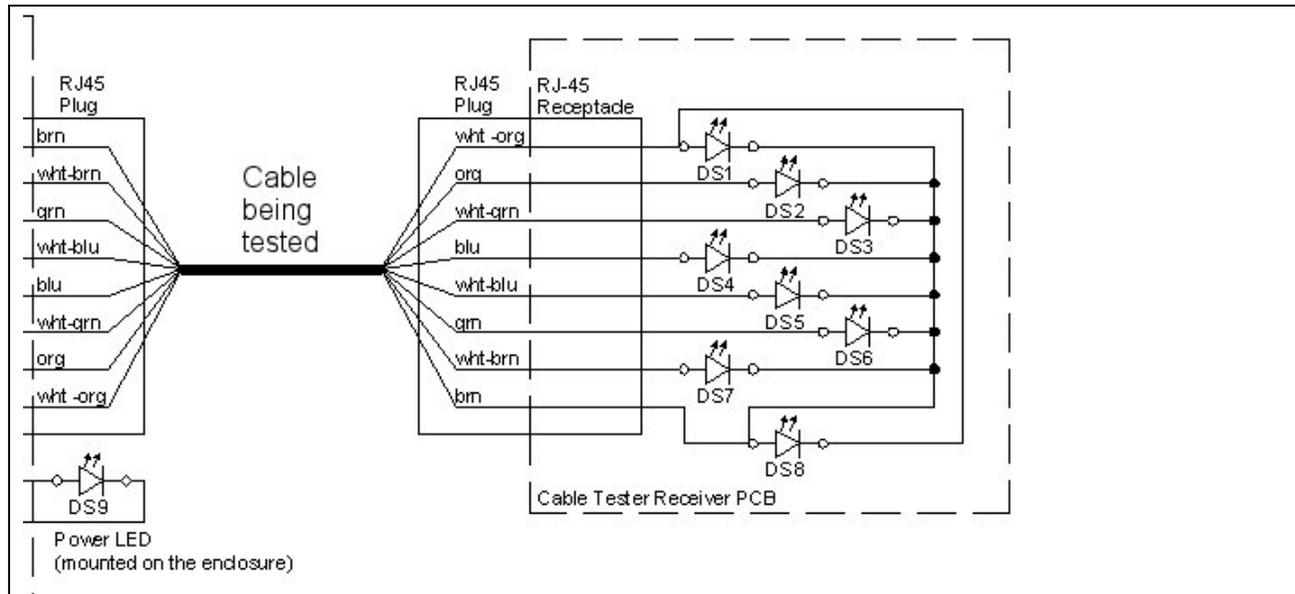


Figure C.1 – Receiver Printed Circuit Board

We begin the process by turning on the grid feature in our CAD program. We will fabricate the board layout in AutoCAD, since we only design a few circuit boards each year. If we were making a more complex PC boards or we were producing multiple units each month, we would use a software application designed specifically for that purpose. Many printed circuit board software packages have the ability to test the circuit, so as we do more circuit cards, we will want to learn how to use the applications.

Creating the Layers for the Printed Circuit Board

We will need to provide a silkscreen, a drill pattern and a solder mask to the manufacturer of the board, so we will make layers called drill_pattern, pad, pcb, silkscreen and track. The drill pattern will mark the point and the size of the hole in the printed circuit board. The layers pad and track can be turned on together to show the electrical circuit, while the silkscreen will help in the layout and troubleshooting of the board. This fiberglass card will just have traces on the backside, but boards that are more complex will have tracks on both sides or can be multiple layer boards.

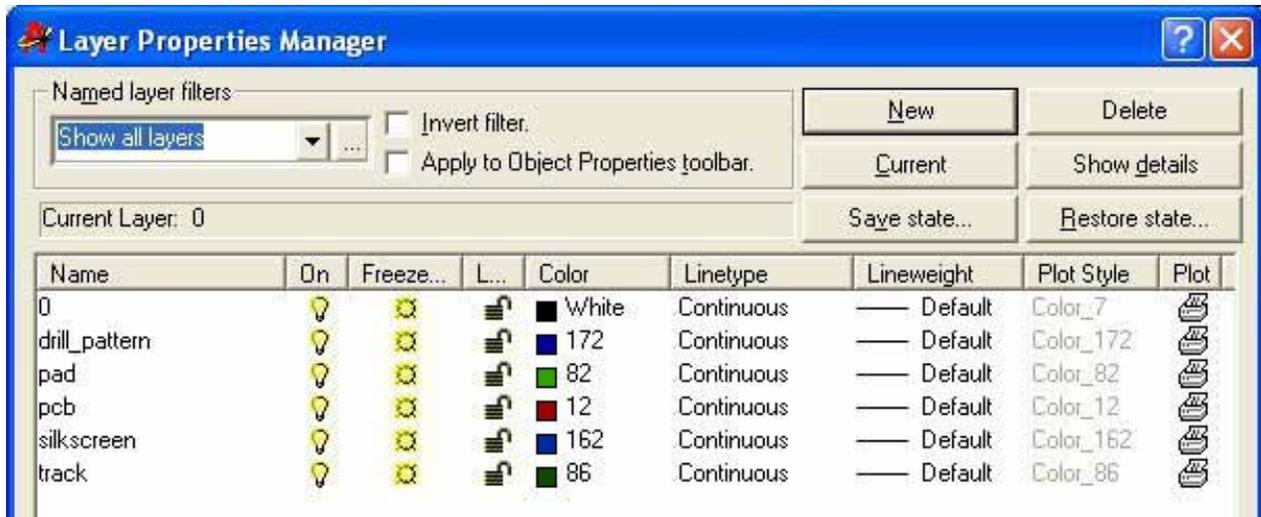


Figure C.2 – Creating Layers for a Printed Circuit Board

Drawing the Components and Pads

We will set the grid in our CAD program to 0.25 inches and the snap to 0.010 inches. We have previously used the grid and snap to make the schematics and wiring diagrams with the electrical symbols library. Printed Circuit Boards are easier to layout when the grid and snap are set and the traces connect effortlessly to the electrical components.

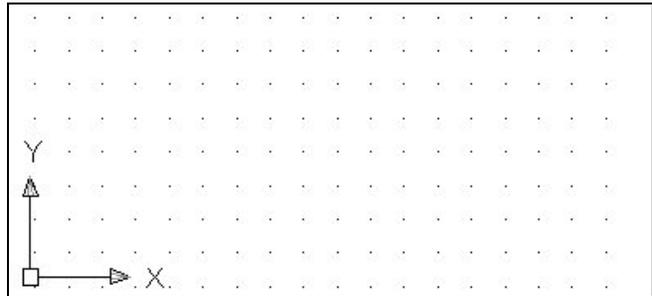


Figure C.3 – Setting the Grid and Snap

On the pad layer, we will draw two circles on a snap point, one at 0.035 diameter and the second at 0.055 diameter. Then we will solid hatch between the two circles to create the pad. We will erase the two original circles. On the drill pattern layer, we will draw a centerline and place the letter “A” in the upper right hand corner as shown in Figure C.4.

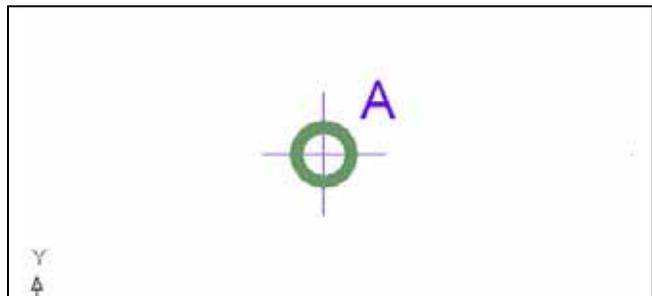


Figure C.4 – Drawing a 0.035 Hole and Pad

We will refer to the datasheets that we utilized in the previous chapter to create the pattern for the integrated circuits. We will use the pad and drill location to show where the integrated circuit's pins will go through the board.

In figure C.5, we place the same RJ45 Receptacle on the receiver card as we put on the transmitter

board. In chapter 6, we can see that pin number 1 is on the very bottom of the graphic and pin number 2 is just to the left and above the first pin. Pin number 3 is directly above pin 1 and the sequence continues all the way to eight.



Figure C.5 – Placing the RJ45 Receptacle on the Board

Next, we draw the Light Emitting Diode (LED) to the left of the RJ45 connector, locating the pads 0.100 apart as shown in chapter 6. We denote the anode with a positive sign (+) and the cathode lead of the LED is on the side of the part with the flat end. The part outline and the positive sign are drawn on the silkscreen layer.

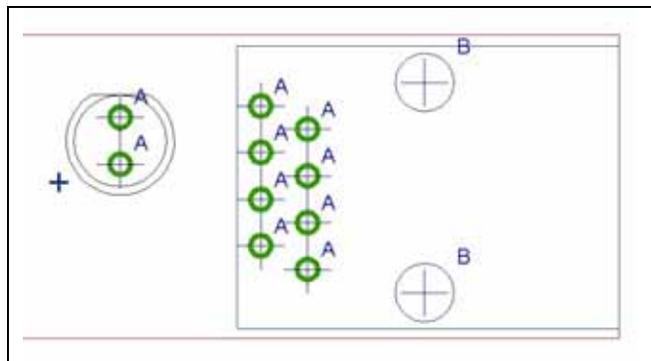


Figure C.6 – Adding the LED

We will array all of the drawing entities representing the LED to the left. According to the schematic, we need eight LEDs. We will place them 0.25 inches apart.

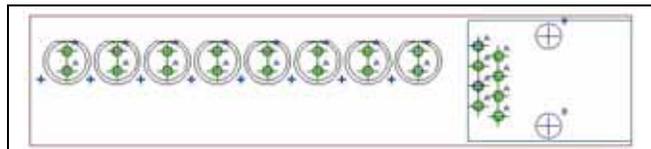


Figure C.7 – Arraying the LEDs

Drawing the Electrical Traces

Now the challenge begins when we start to draw the electrical traces. Our traces on this board are 0.025 thick and we will use the Polyline tool to make the connection. We sat down with a pencil and through trial and error connected the pads. This takes about one hour for simple boards, so again we avoid buying and learning another software package.

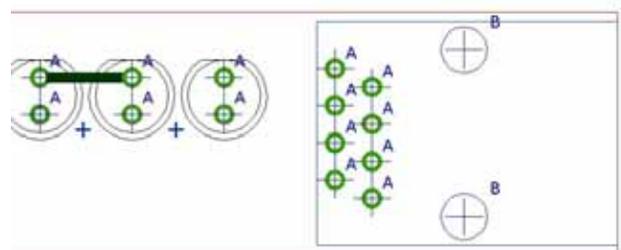


Figure C.8 – Drawing a Trace

We decide to designate the LED on the far left as DS1, so the LED on the far right is DS8. On the schematic, the cathodes of the first seven LEDs are connected. In figure C.8, we draw a 0.025 track from DS7 to DS6. In figure C.9, we connect all of the cathodes for the first seven LEDs. We highlight the schematic in yellow to account for every track drawn on the fiberglass card.

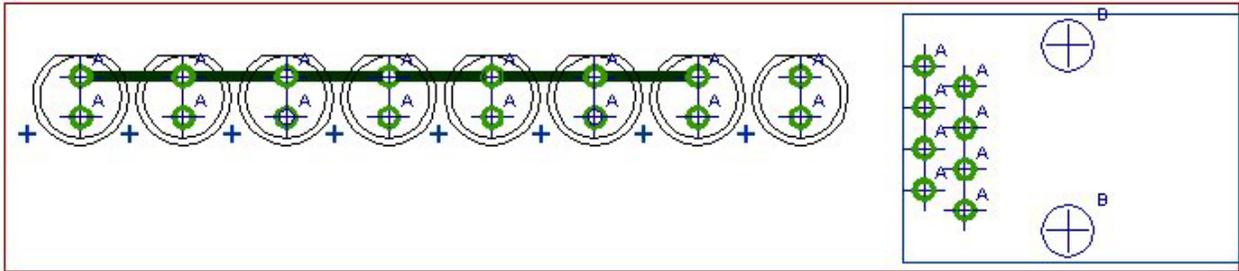


Figure C.8 – Drawing Multiple Tracks

Next, we connect the cathode of DS8 to the anode of DS1. Then we attach the cathode of DS7 to the anode of DS8 as shown in figure C.9. In this simple board, we only have eight traces left to draw from the pins on the RJ45 connector to the eight LEDs.

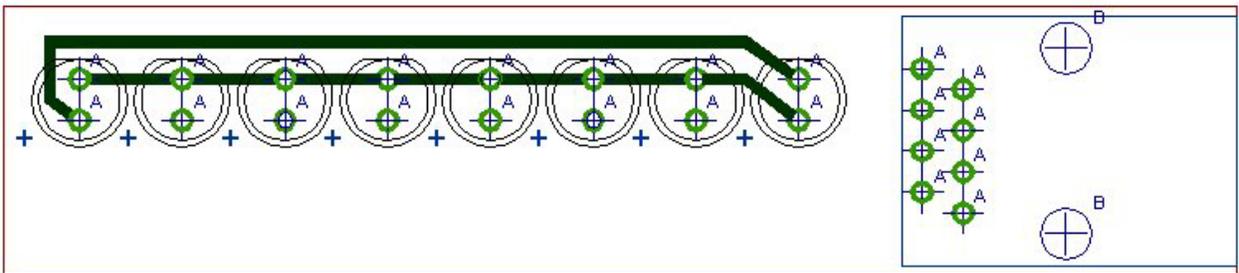


Figure C.9 – Adding Two More Tracks

When we pass between two pads, we will truncate the pad. We reduce the width of the electrical trace to 0.015 to reduce the possibility that the two electrical wires will touch causing a short circuit. The Polyline command in AutoCAD has the ability to start the line with a 0.025 width and end the line with a 0.015 thickness. We can draw the next segment with a 0.015 thickness and then increase the thickness from 0.015 to 0.025 on the last segment.

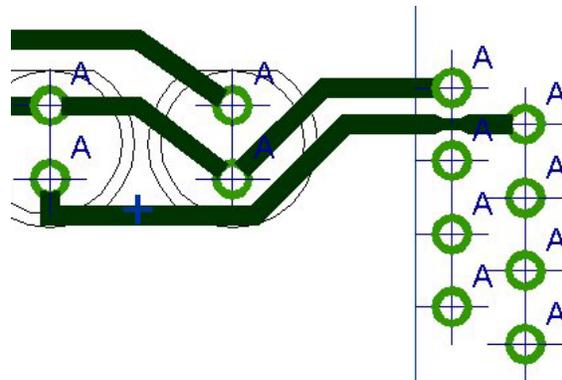


Figure C.10 – Truncated Traces

In figure C.10, we draw a track from the cathode of DS8 to pin 8 of the RJ45 receptacle. We can see the second track with the truncated section that parallels the first trace on its way to the cathode of DS2. Our strategy is to bend the track at 45°, so we can efficiently use the space on the card.

We need to go ahead and complete the connection of pads for this board, and each time we draw a track in our CAD program, we will highlight the wire on the schematic. When every pad is connected and each line in the wiring diagram is highlighted, the printed circuit board will appear as shown in Figure C.11.

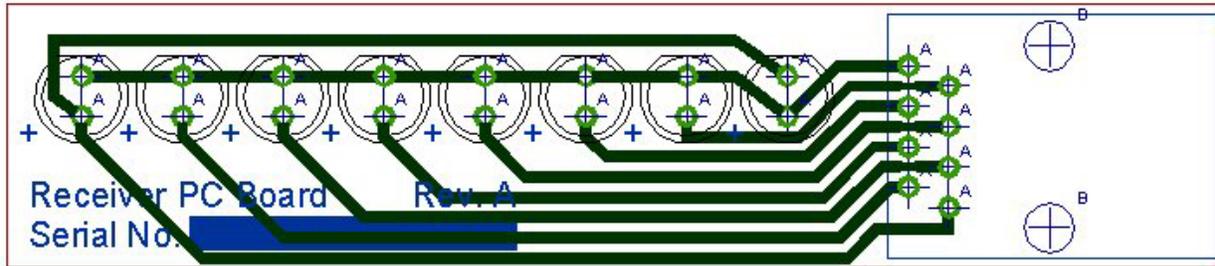


Figure C.11 – The Completed Traces

The finished silkscreen will appear as shown in Figure C.12. We can add the company name, product part number, and a solid white area to write the board's serial number with a pen or stamp.

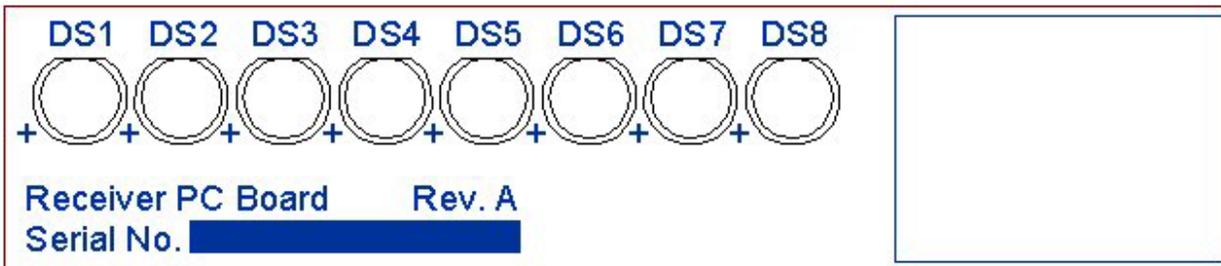


Figure C.12 – The Silkscreen

We mirror the solder mask and this is what the backside of printed circuit board will look like.

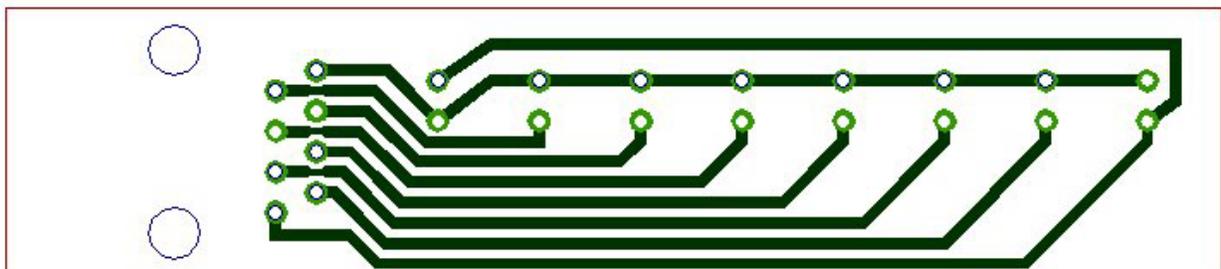
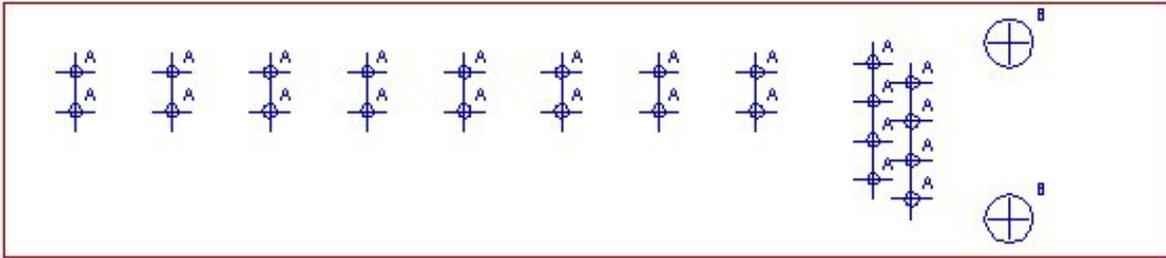


Figure C.13 – The Silkscreen

There are 24 holes marked with an A and 2 holes labeled with a B. For accuracy, we can transmit the CAD file, so the manufacturer can create a computer file to control the location of the drilled holes.



Note:

#65 Drill (0.035) thru, 24 places marked "A"

0.125 Drill thru, 2 places marked "B"

Figure C.14 – The Drill Pattern

We will want to order three or more sample boards for the prototype. We will test the first units to give us the confidence that the cable tester will function properly.

Redraw the Schematic to Agree with the PCB

After revising the prototype and finishing the project, we need to rework the schematic. We place the component identifiers such as DS1 to DS8 on the electrical drawing in chapter 6 not knowing the actual placement of the part on the printed circuit board. Once the PCB is completely done, we always go back the initial drawing and adjust the component names.

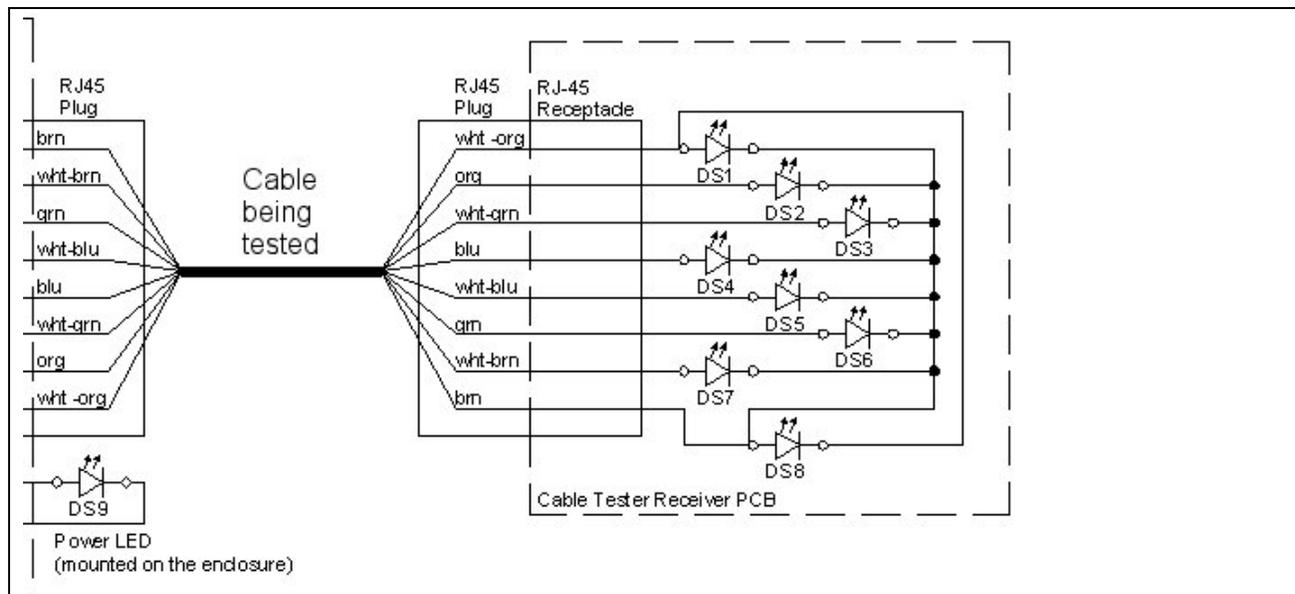


Figure C.15 – The Revised Schematic

*** World Class CAD Challenge 13-9 * - The project engineer wants to you to create a printed circuit board art of the cable tester in your CAD program. Use should make a silkscreen, drill pattern and electrical trace layer in the next 120 minutes.**

Continue this drill four times using some other ideas, such as the Receiver PCB completing the drawing under 120 minutes to maintain your World Class ranking.