

# Chapter 7

## Creating a Printed Circuit Board

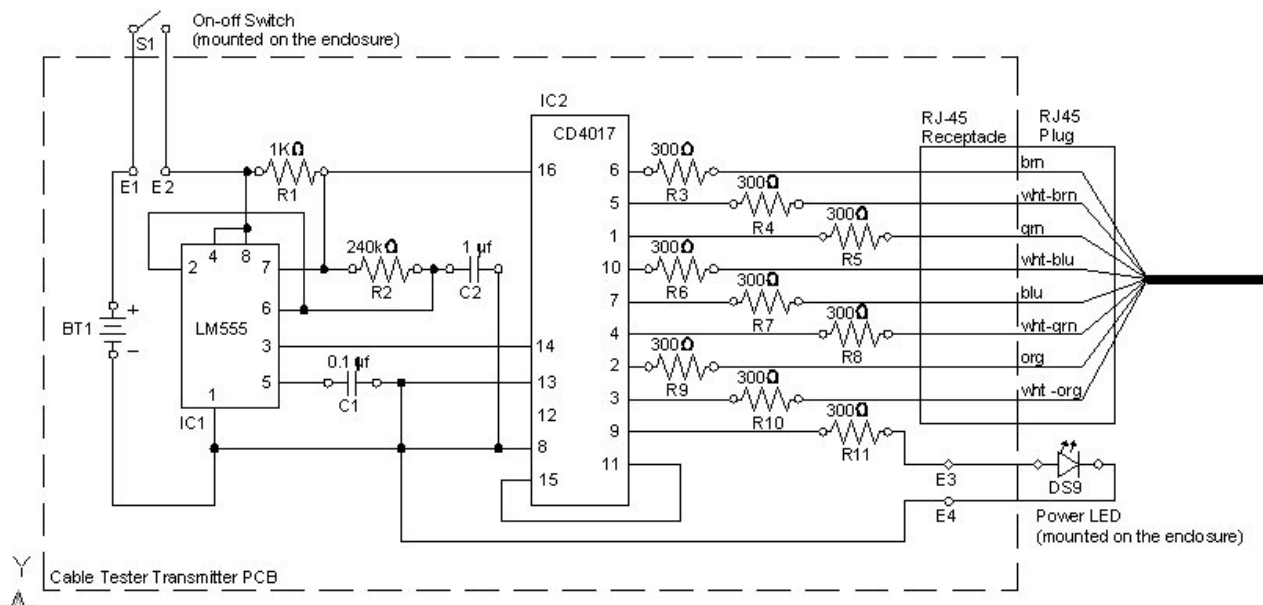
---

**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 7.1 is completed, we can begin to lay out the PCB. 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 previous chapter does contain the datasheets that will be essential in making a working product. When we wrap up making the board, 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 7.1 – Transmitter 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. With training, we could use software such as ORCAD, Proteus or Altium to create the board layout, the drill pattern, the artwork for the board's silkscreen and the trace pattern.

## 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 board will just have traces on the back side, but more

complex boards will have tracks on both sides or can be multiple layer boards.

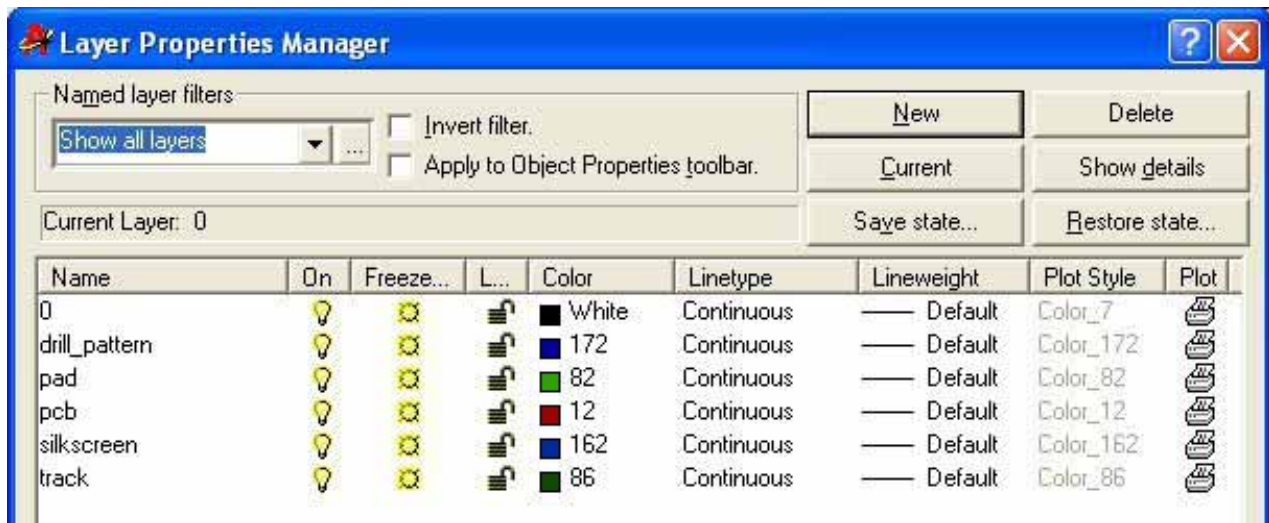


Figure 7.2 – Creating Layers for a Printed Circuit Board

## 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 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.

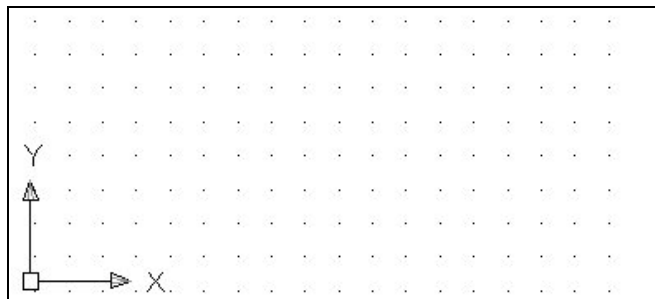


Figure 7.3 – Setting the Grid and Snap

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 7.4.

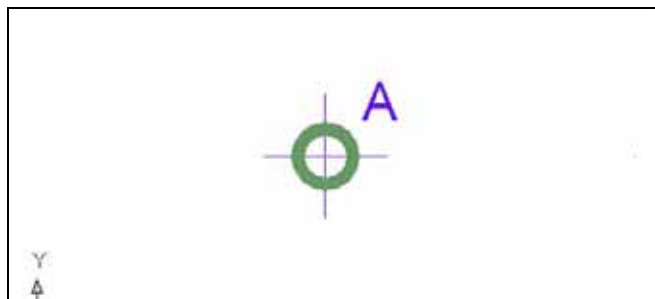


Figure 7.4 – Drawing a 0.035 Hole and Pad

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.

We need to draw the CD4017 chip with the pads and drill pattern 0.300 in width and with the vertically spaced holes 0.100 apart. We can use the Computer Aided Design program's Array tool to make the two columns and eight rows. On the silkscreen layer, we draw the outside of the chip including a "1" which will show where pin number one is located. We will turn off the drill pattern layer off when we draw the electrical traces on the track layer. Next, we will place the resistors to the left of the 4017 chip, directly before the RJ45 connector.

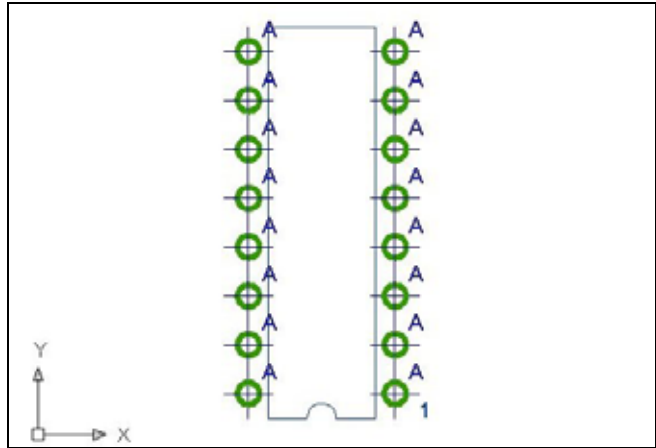


Figure 7.5 – Arraying the Hole and Pad

A quarter watt resistor's body measures 0.235 inches long and 0.091 inches in diameter. The resistor's lead diameter is 0.025 diameter and the leads are 1.5 inch long.

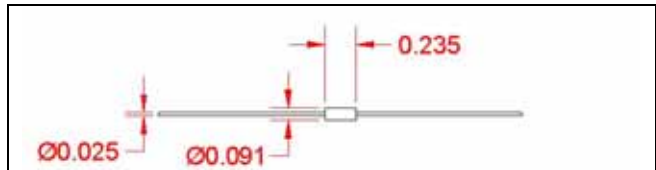


Figure 7.6 – Inserting a 1/4 Watt Resistor

We will draw the resistor body to the closest 0.010 inch, so the body length is 0.024 and the width is 0.080. We placed two 0.035 drilled holes 0.4 inches apart and centered across the resistor body. When the assembly technician builds the Transmitter PCB, we will bend the leads at 90° and the leads will be 0.4 inches apart.

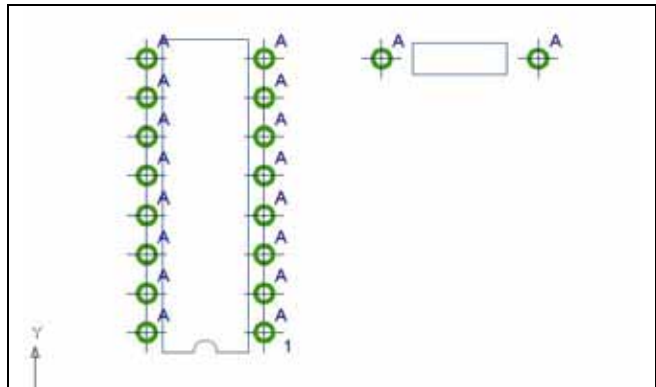


Figure 7.7 – Inserting a 1/4 Watt Resistor

We will need 8 resistors, one for each LED and on the Transmitter PCB and 1 resistor for the 555 timer. We will space the 1/4-watt resistors at 0.1 inches apart, which is the same spacing as the pins on the integrated circuit. We can move groups around on the graphical display when we lay out the electrical traces, so we do not worry now about how close the collection of resistors are to the 4017 chip.

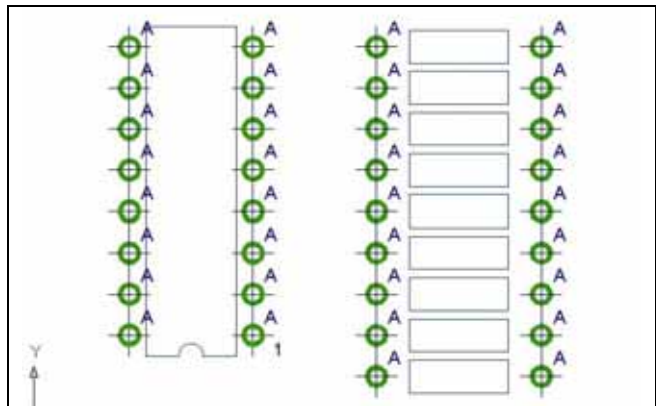


Figure 7.8 – Arraying the Resistor

Next, we draw the RJ45 receptacle with the outline of the connector on the silkscreen layer, the holes on the drill\_pattern layer and the pads on the pad layer. We get the dimension of the RJ45 jack from the datasheet in chapter 6. We should label the 0.125 diameter holes “B” since they are different than the “A” size holes. The eight pin holes are the same size as the resistor and IC holes.

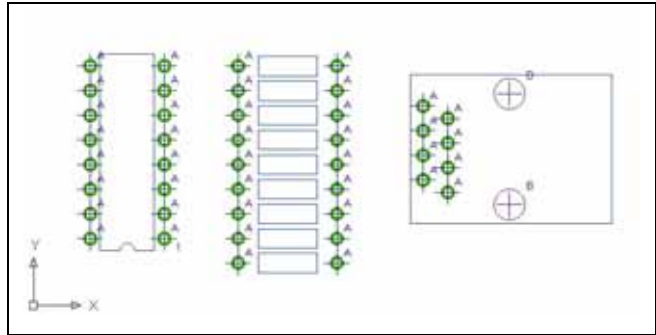


Figure 7.9 – Inserting the RJ-45 Jack

We then draw the eight pin 555 IC that is half the size of the 4017 chip. When we produce drawings of specialized components, we can make blocks or discreate files of the series of holes, pads and markings, so that can use the information again quickly on other projects, The power of printed circuit board software is their library.

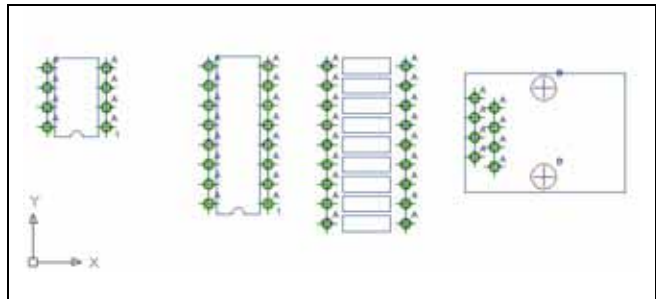


Figure 7.10 – Drawing the 555 Timer

Now, we place the position of the test points, E1 to E7, the last two resistors and the two capacitors. Test points will have wire soldered to them, so we can connect to the power switch, the battery, the “On LED” and to make jumps off the board by wire. A printed board is referred to as art in the industry, since we have to use our own judgment as to the location of components.

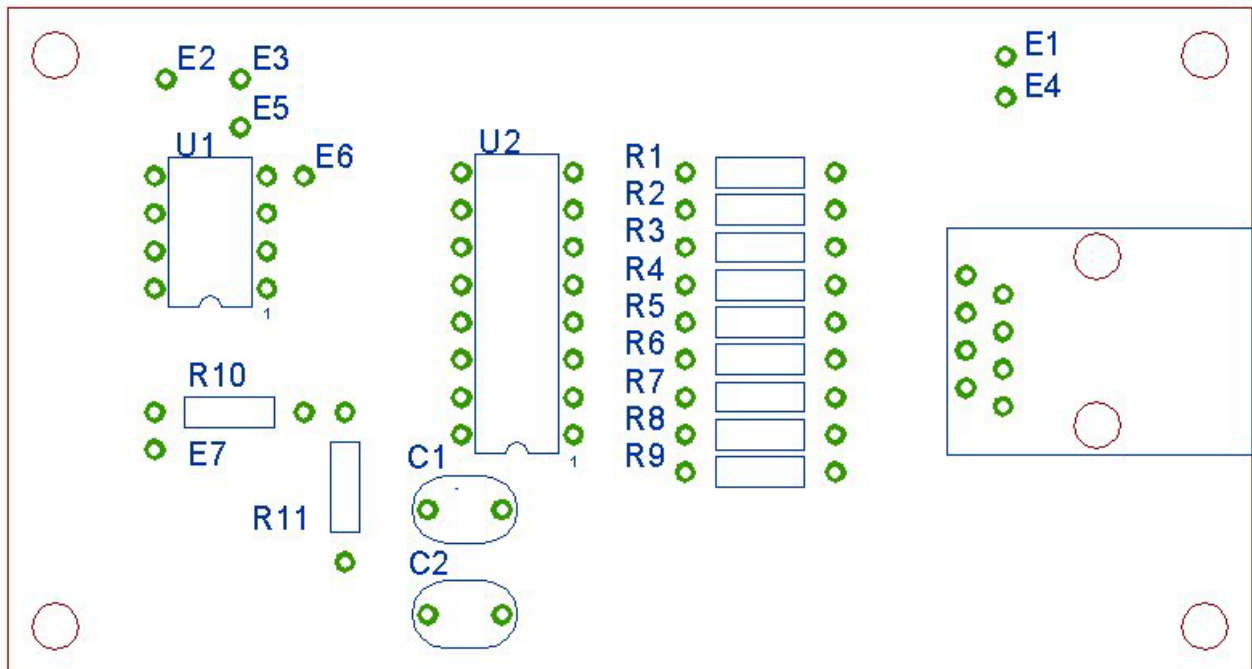


Figure 7.11 – Adding Two Resistors, Two Capacitors and Four Test Points

Test points E5, E6 and E7 did not exist on the original schematic, but we see that we need these points when we lay out the board. We decide the place the 9V battery to the left of the board and opposite the RJ 45 jack. Our board is 1.8 by 3.4 inches and it will fit into a 2 by 5 inch plastic box easily.

## Drawing the Electrical Traces

Now the challenge begins when we start to draw the electrical tracks. 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. We utilize the Chamfer command to make 0.05 by 0.05 breaks in the traces. This feature can help reduce the distance between points and save space on the board.

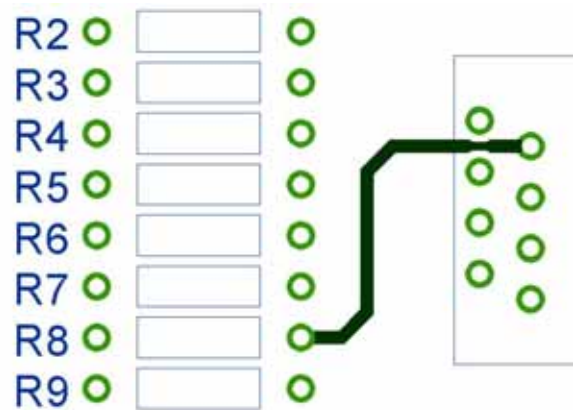


Figure 7.12 – Drawing a Trace

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.

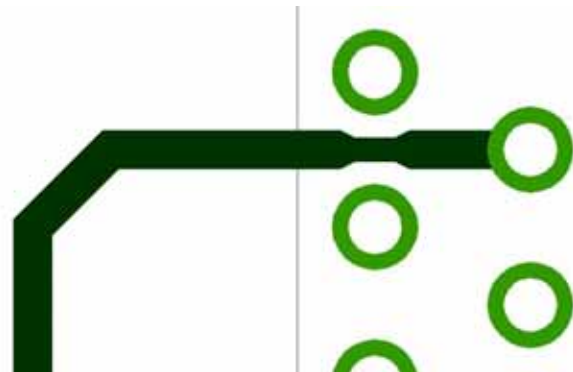


Figure 7.13 – Truncated Traces

We work in one area of the board at a time and we finish the traces from the bank of resistors to the E4 test point and the RJ45 receptacle. If we compare our artwork to any other designer's layout working on the same project, we will find differences. The traces on this board are actually on the backside of the printed circuit board and not touching the silkscreen on the front side of the board. We must miss pads and holes as we see in figure 7.14 and we bevel the edges of the traces as shown.

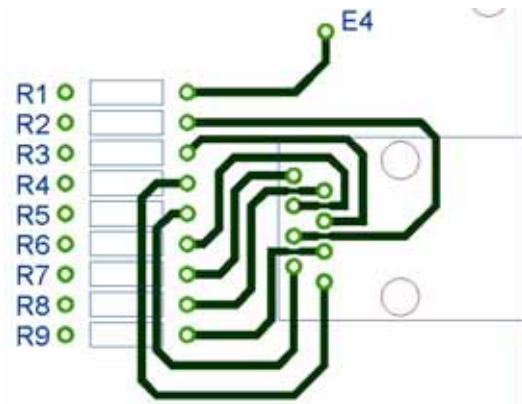
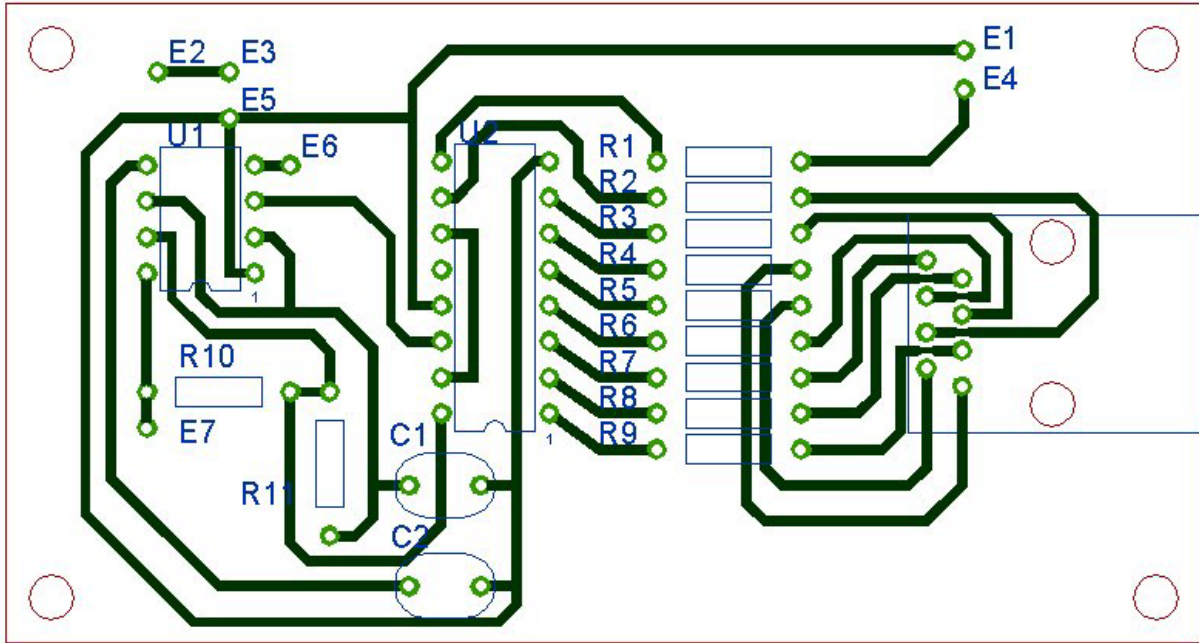


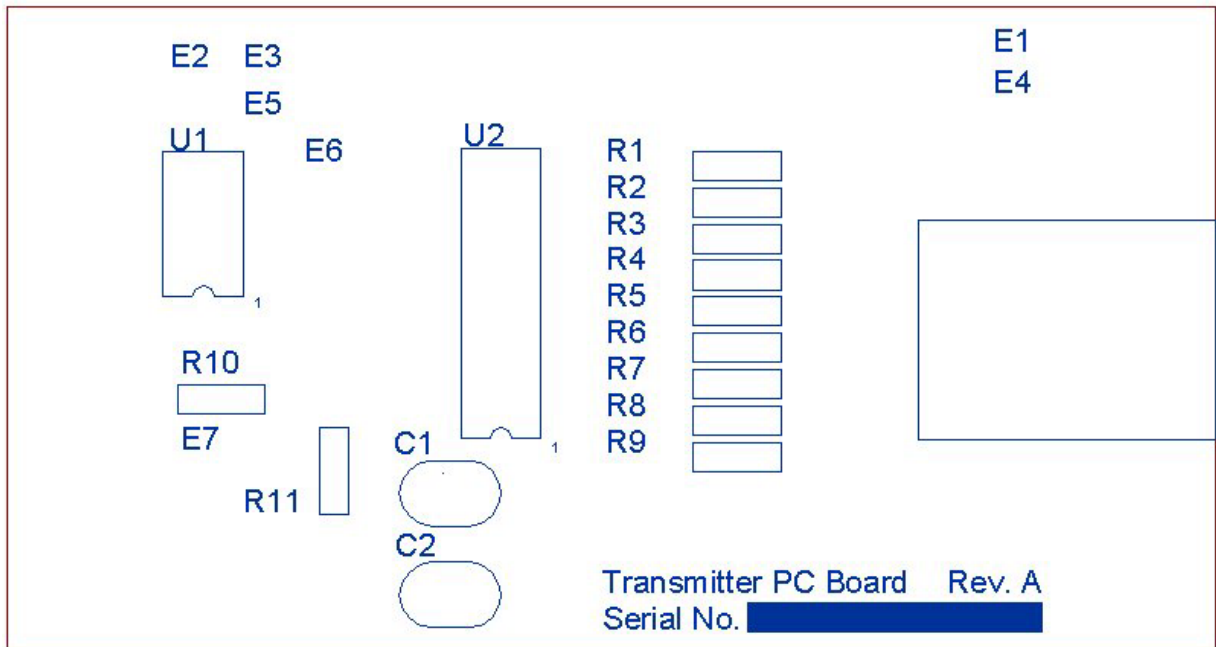
Figure 7.14 – Drawing a Group of Tracks

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 7.15.



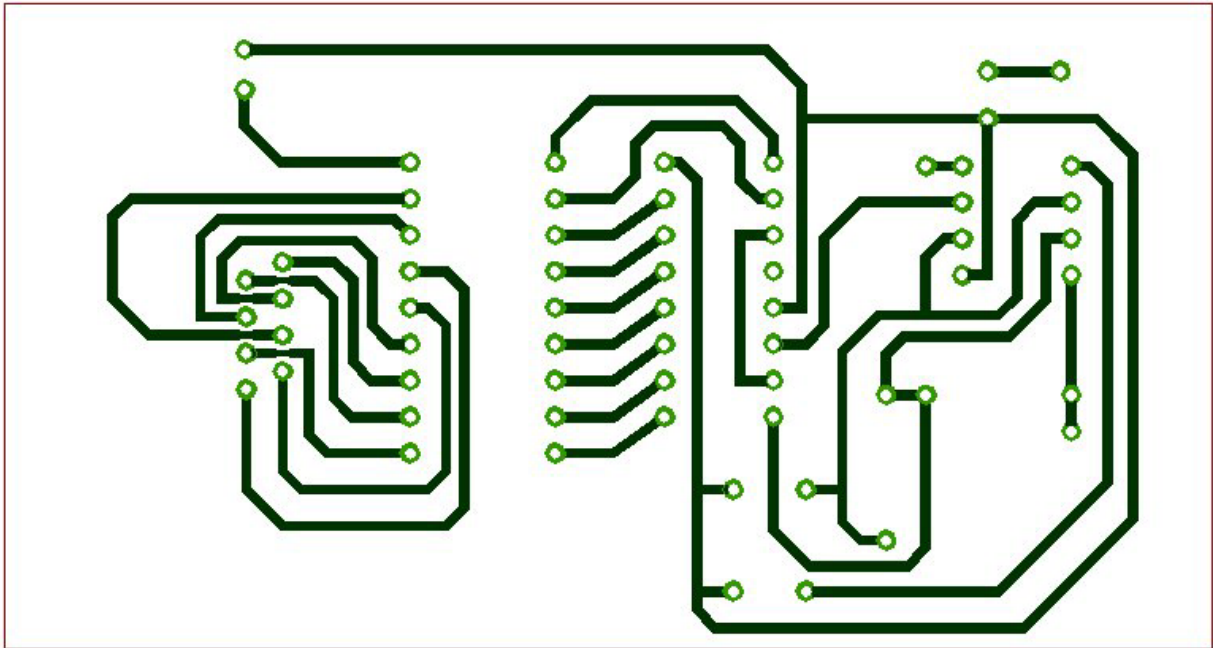
**Figure 7.15 – Truncated Traces**

The finished silkscreen will appear as shown in figure 7.16. We can add the company name, product part number, and a solid white area to write the boards serial number with a pen or stamp.



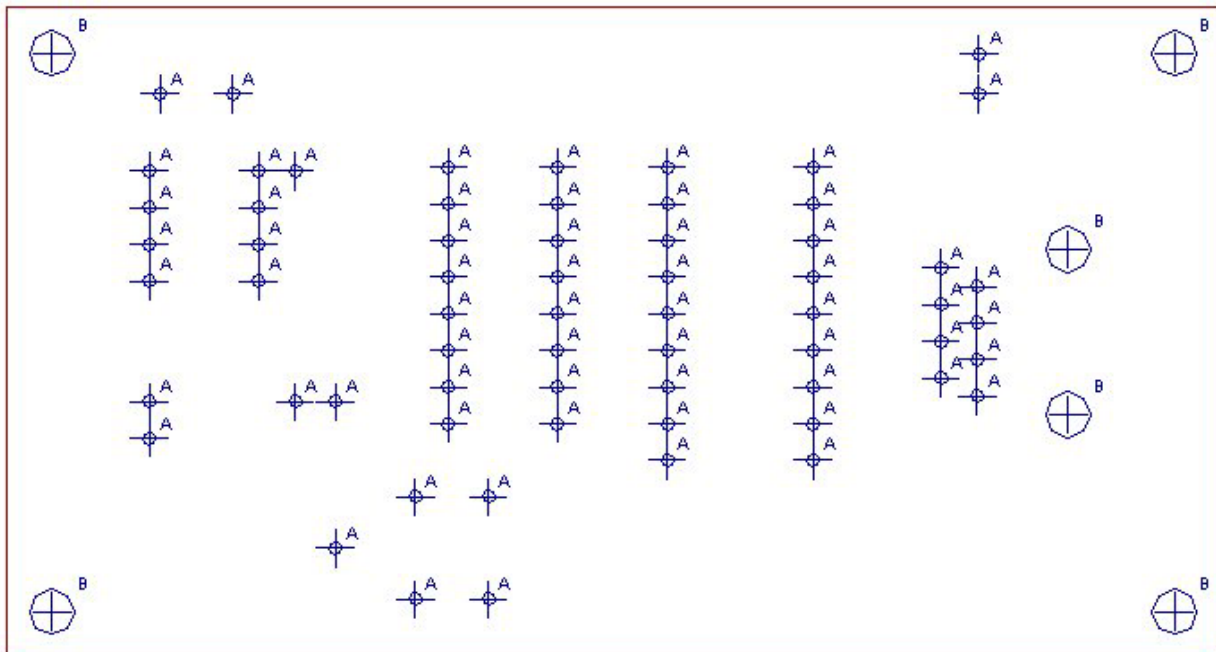
**Figure 7.16 – The Silkscreen**

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



**Figure 7.16 – The Silkscreen**

There are 64 holes marked with an A and 6 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.



Notes:  
#65 Drill (0.035) Thru, 64 places marked A  
0.125 Drill Thru, 6 places marked B

**Figure 7.17 – 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 U1, R1 and C1 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. We also insert the three new test points.

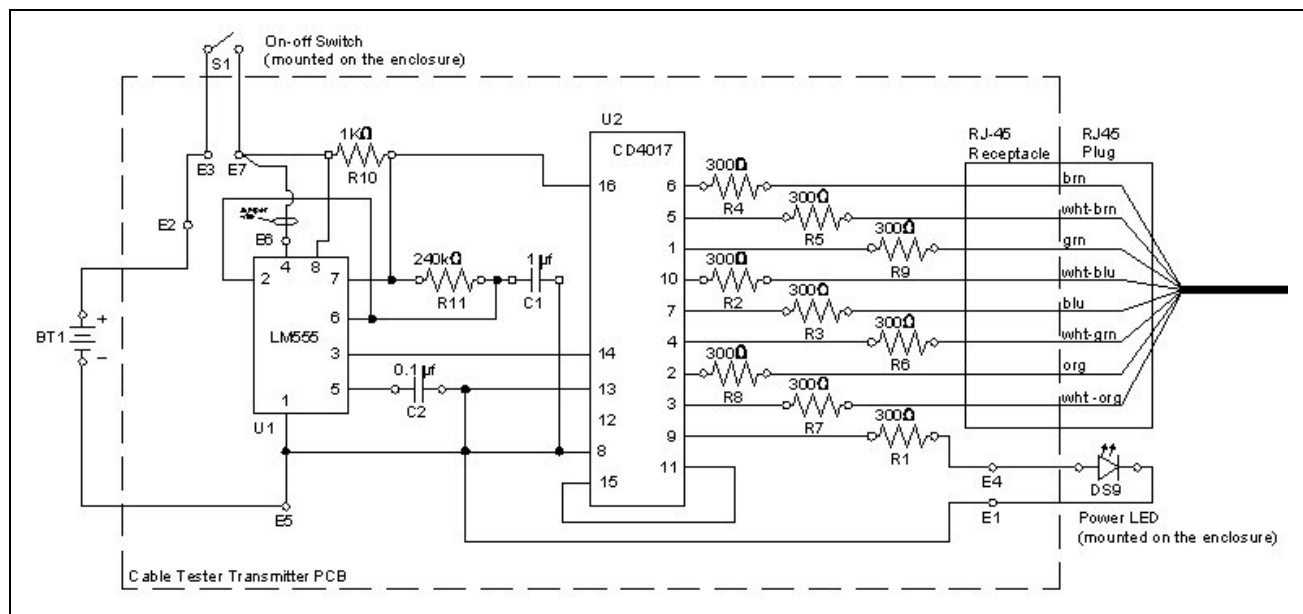


Figure 7.18 – The Revised Schematic

**\* World Class CAD Challenge 13-8 \* - 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.**