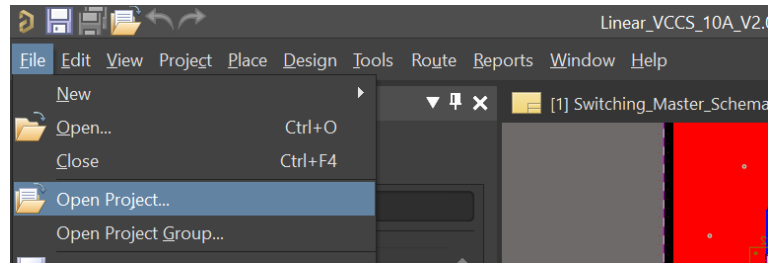
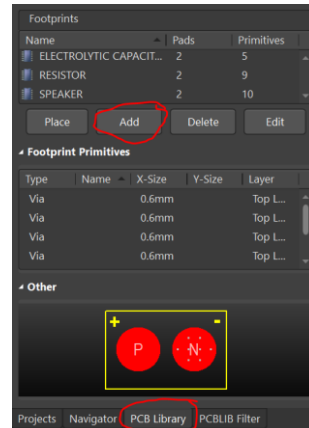


PCB Library

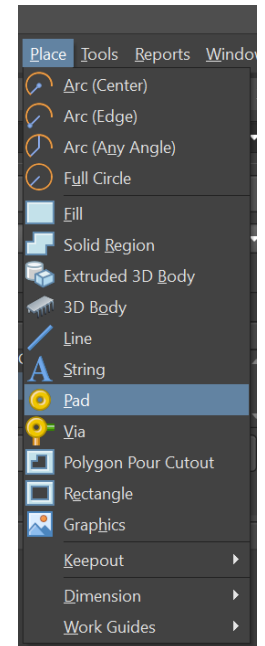
1. Open the program "Altium Designer"
2. Go to *File* → *Open Project* → *outreach keyboard project*



3. Open *PCBLibrary.PcbLib*
4. Double click on the new component and rename it "555Timer"

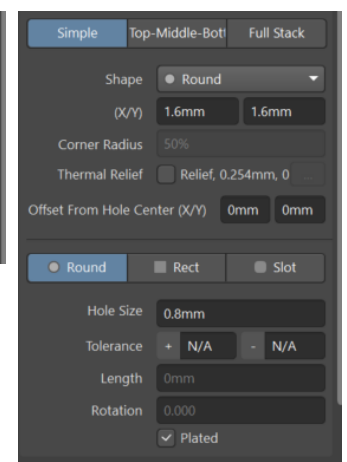
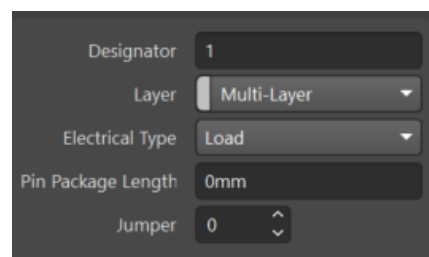


5. Click on the main editing window (big grey area) to put us into editing mode.

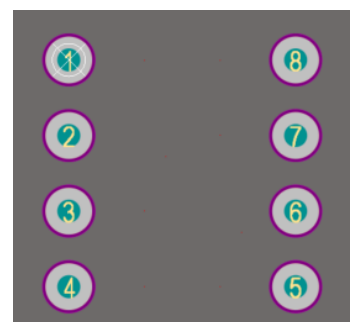
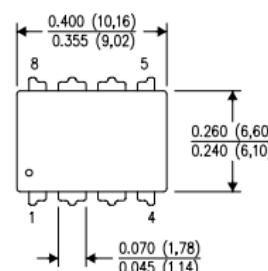
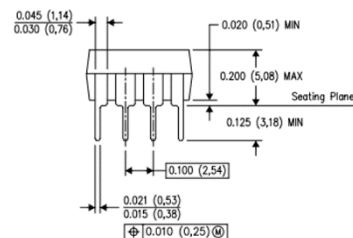


6. Press *G* and select 100 mil. This is how far apart the pins are on our component so we are setting the grid to this size, so everything is in the right place.
7. Now go to *Place* → *Pad* then press *TAB* to open the properties for the pad. You might notice that the menu has underlined letters, these are the hotkeys for that selection so instead of selecting place pad, we could press "pp" to achieve the same thing.

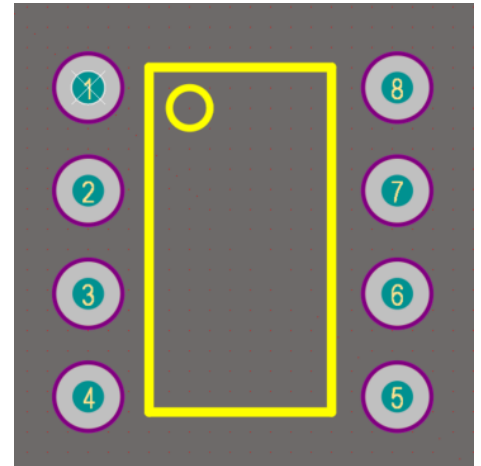
8. Now we need to make sure the pad is the right size for our component. Use a multilayer pad, which means there will be a hole for the pins, with a 0.8mm hole and 1.6mm pad. Keep the hole round but choose any shape of pad you want. Also make sure the designator is 1.



9. Now click to place the pads as shown. They should be spaced by one grid space vertically, and 4 horizontally. Sometimes we will make pin 1 a different shape to make it easy to find but we will also add another indicator later



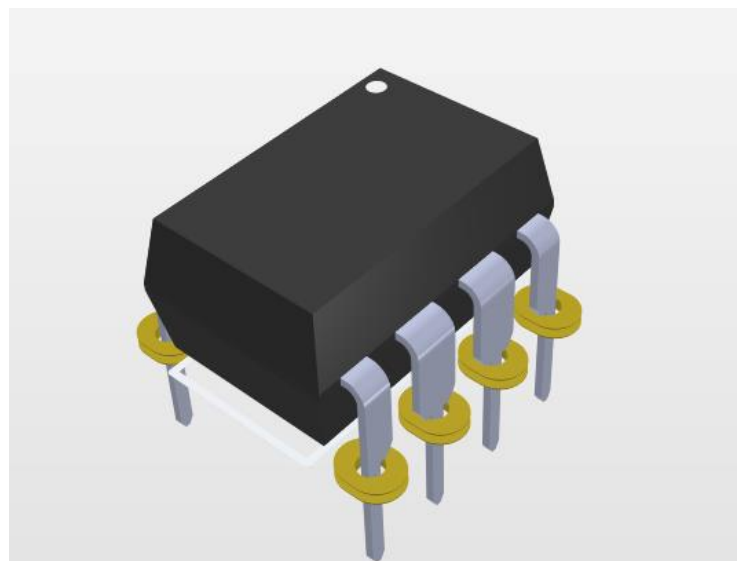
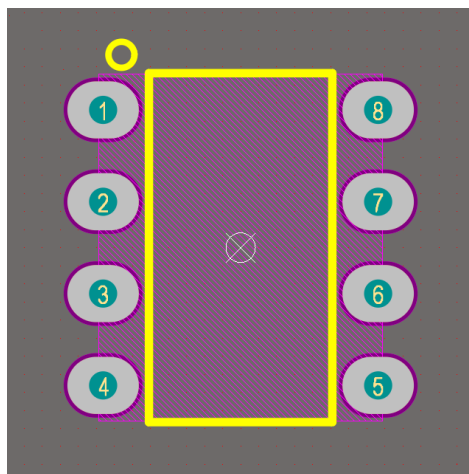
10. Now we're going to add some aesthetic bits. Move to the *Top Overlay* and use *Place* → *Rectangle* and *Place* → *Full Circle* to make your part look something like this. You may need to change your grid size to get it looking how you want. We put a circle near pin 1 to show which way the component goes. It can be inside the part to save space, or outside to make it easier to see.



11. Finally, we are going to add a 3D model. While this isn't necessary it helps us check what our design will look like and can sometimes make errors easier to spot. Use *Place* → *3D Body* and select the SA555PE4.STEP file. Click to place it.

12. Now Press 3 to go into 3D view mode. Click on the 3D model then *Properties* on the right-hand side and rotate it so it's up the right way. Press 2 to go back into 2D mode, drag the model so it's centred on your pads. Then go back and check in 3D to make sure everything looks right.

13. Press Save (or *Ctrl S*) and we have made a PCB Library Component.

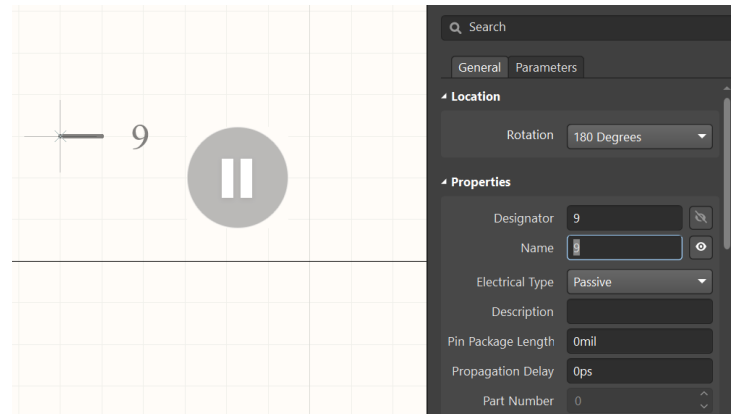


Schematic Library

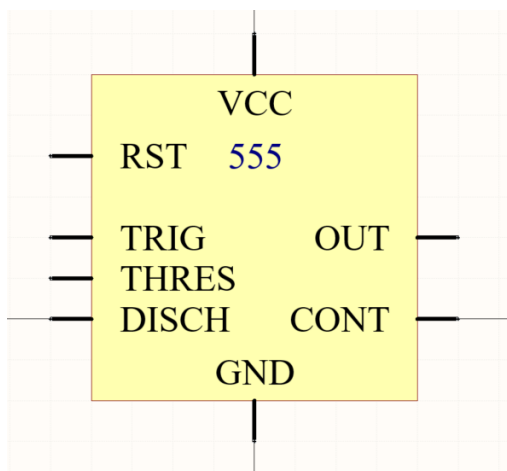
1. First Open the Schematic Library, and add a new part, just like with the PCB Library. Name This Part 555Timer too.

2. Open *Properties* and set the Designator to "U?" This is a universally used code for this type of part.

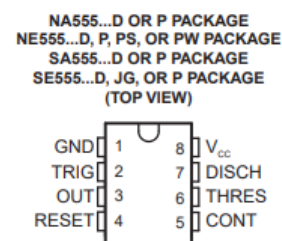
3. Now use *Place* → *Pin* to place a pin, press *TAB* to open up its properties. Here we have the Designator, which is what Pad in our PCB Library the pin represents and Name, which is what we want to refer to the pin as. Use the little eye to hide and show these. I show the name and hide the designator.



4. Let's place our pins. We can place them the same way they are physically on the part, however, normally we place them so power runs top to bottom and signals run left to right. This makes it easier for us to read the schematic. Knowing what is going to make a part easy to read is just practice. Mine is below as an example. The little cross on one side is where we will connect our wires to so this should point outwards from our part



6 Pin Configuration and Functions

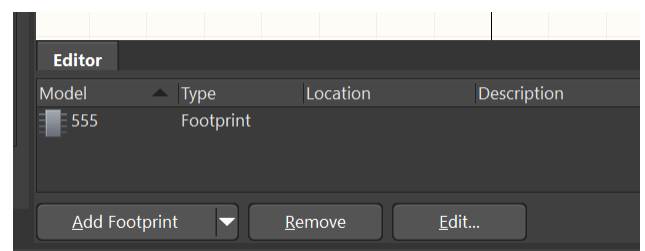


5. Now place a rectangle on the inside of all your pins. Then use *Edit* → *Move* → *Send to Back* to make the names show again.

6. Use *Place* → *Text String* to add a part name in blue, I added 555.

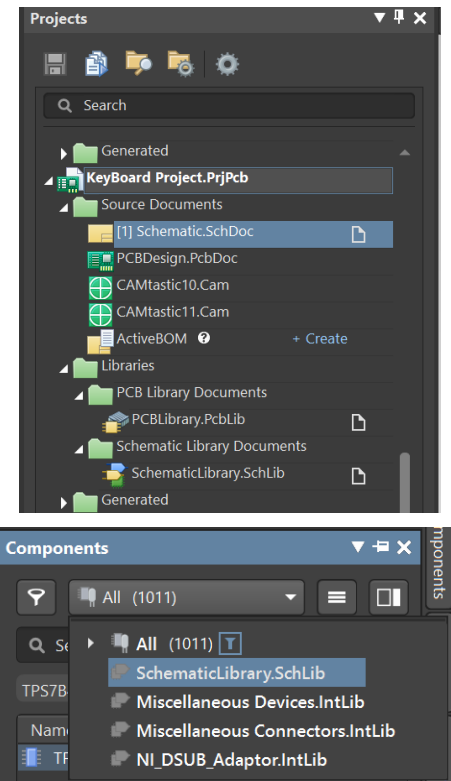
7. Finally, Down the bottom select *Add Footprint* → *Browse* and select the footprint we made earlier.

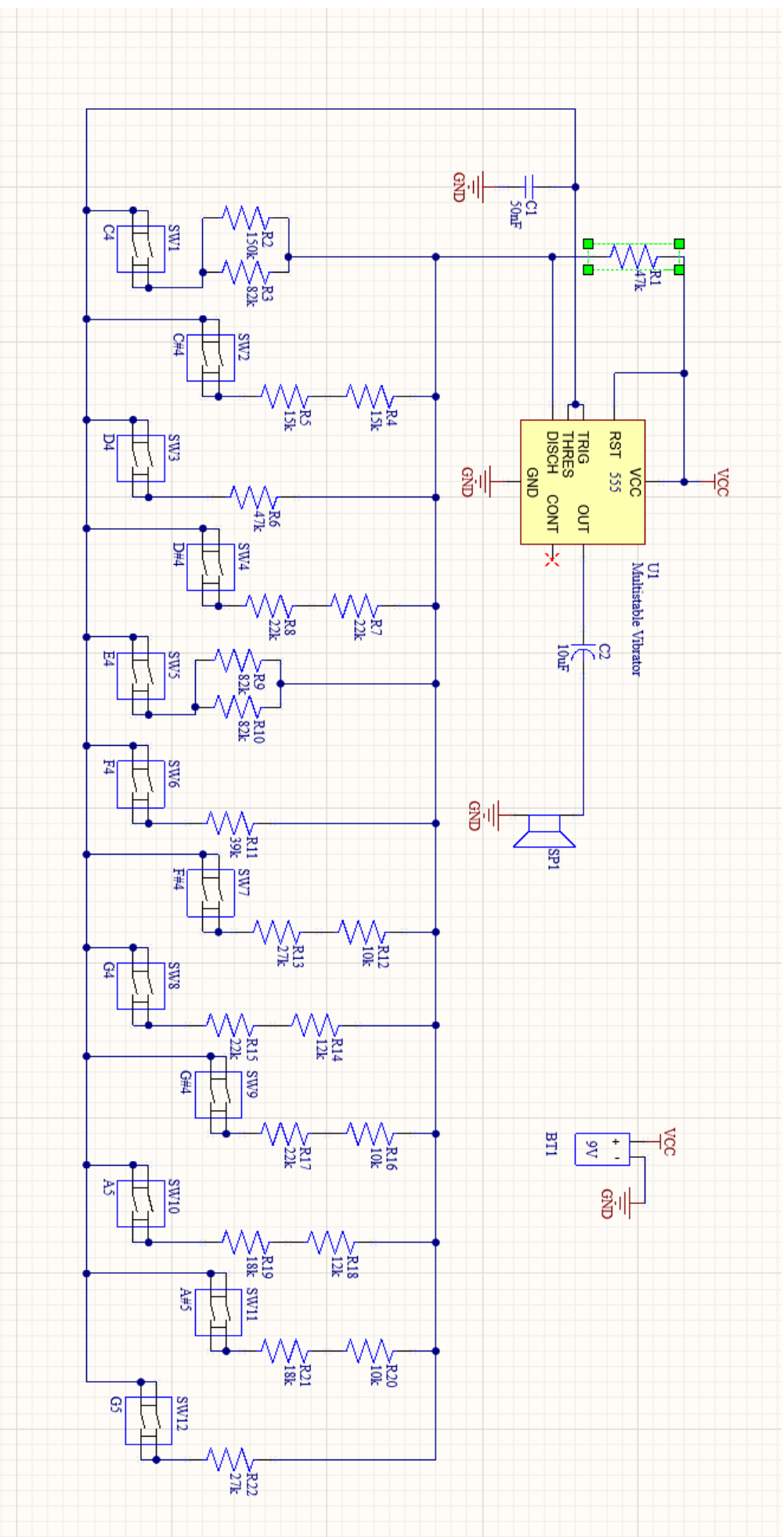
8. Save our component and our libraries are completed.



Schematic

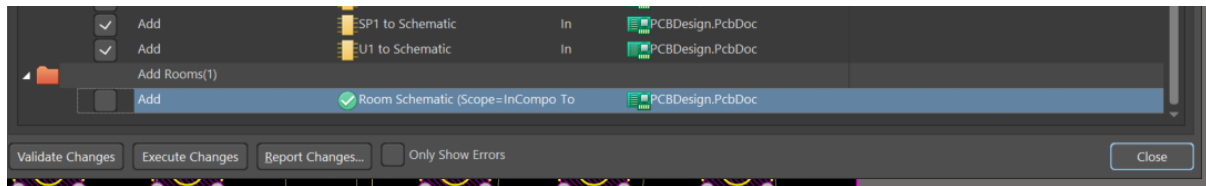
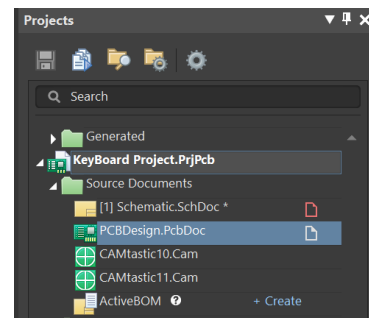
1. Open *Schematic.SchDoc* This is the document where we will show what is connected to what.
2. Using *Place* → *Part* select *SchematicLibrary.SchLib* to find our library of parts
3. Place the following Parts on the schematic:
 - a. Our 555 Timer
 - b. Electrolytic Capacitor
 - c. Ceramic Capacitor
 - d. 9v Battery
4. Move the parts around, using *SPACEBAR* to rotate them. Also open the properties of our new resistor and make the comment 47k so we know the right resistor to use when soldering the boards.
5. Now we are going to wire everything together. To do this use *Place* → *Wire*. You can use *SPACEBAR* to change which way it turns the corner, and *SHIFT + SPACEBAR* to change the corner style. If there is a junction (more than 2 connections) a little dot will show.
6. For our power we are going to use a port. These connect pins without using a wire. This helps our schematic not get too busy, so it is easier to read. Use *Place* → *Power Port* and press *TAB* to set the name and style. Everything with the same name will be connected. Make the positive side of the battery VCC and the negative GND
7. Because we don't want to connect anything to the CONT pin of our 555 we will use *Place* → *Directives* → *Generic No ERC* and put it on the CONT pin. When we check everything is correct this will tell the program we meant to leave it unconnected.
8. Finally, use *Tools* → *Annotation* → *Annotate Schematics Quietly* and save the file. This gives each component a unique number (i.e. R12) so that we can relate the component on the PCB to the schematic. Once you're done your schematic should look like this:



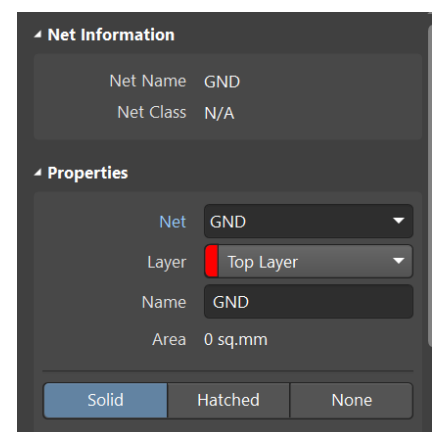


PCB Design

1. Open *PCBDesign.PcbDoc*
2. *Design* → *Import Changes From.....*
3. Scroll to the bottom, untick the room, then press *Execute Changes* then close the pop up. Now all our components will be on our PCB



4. Drag components around, get them all on the black PCB shape, using *SPACEBAR* to rotate parts, try to untangle the “spiders web” the best you can. Like in the PCB library we can use 2 and 3 to move between 2D and 3D view, and press *G* to change grid size. I recommend starting with a 1mm Grid initially, going smaller if you need to. Don’t worry if not everything can be untangled. You may be able to loop a connection around another one or move one to a different layer.
5. Make sure you are on either *Top Layer* or *Bottom Layer* (these are the copper on the top & bottom of the board) you can use *Place* → *Track* to connect the pins that need to be connected (shown by the spider’s web lines).
 - a. *SHIFT + SPACEBAR* will change how the track goes round a corner, and *SPACEBAR* will change which way it turns that corner. Most PCBs will use a 45 degree corner, but use whatever you want.
 - b. Try to keep as much as possible on one side of the PCB.
 - c. If you need to swap between layer part of the way along a track use *Place* → *Via* and click on your track. This will create a little connection between layers.
 - d. Don’t worry about connecting GND for the moment, we’ll use a trick for that later.
 - e. You can swap between step 4 and 5 as much as you need, most designs will take a few attempts to come up with something that works well.
6. Once everything but GND is connected, move to the top layer, the use *Place* → *Polygon Pour* then press *TAB*. Select Net to be GND, then press the pause button to resume. Draw a square around the PCB staying a little bit inside the outline then press *ESC* when done. This fills all of our spare area with copper, connecting all the GND pins. Often this will reduce noise on our circuit but it is also easier than routing ground, and we already paid for that copper, we may as well keep it.
7. Repeat step 6 on the bottom layer
8. On the Top Overlay & Bottom Overlay we can design our silkscreen.



- a. Make sure to use *Place* → *String* to put your name on the PCB
 - i. If you are on the bottom overlay you will need to find the “mirror” function otherwise the text will be backwards when it is made.
 - b. Other things you might add are:
 - i. A PCB name (i.e. Outreach Keyboard Project)
 - ii. The date it was designed
 - iii. A version number
 - iv. Directions for use
 - c. You can place an image using *Place* → *Graphics* too.
 - d. Keep checking how things look in 3D mode as you are doing this.
9. Once you are happy use *Tools* → *Design Rule Check* then press *Run Design Rule Check*. This will check if you forgot to do anything. Let us know if anything shows up here. Sometimes it will bring up errors that don't really matter too much.
10. Save the File and you have designed your own PCB.