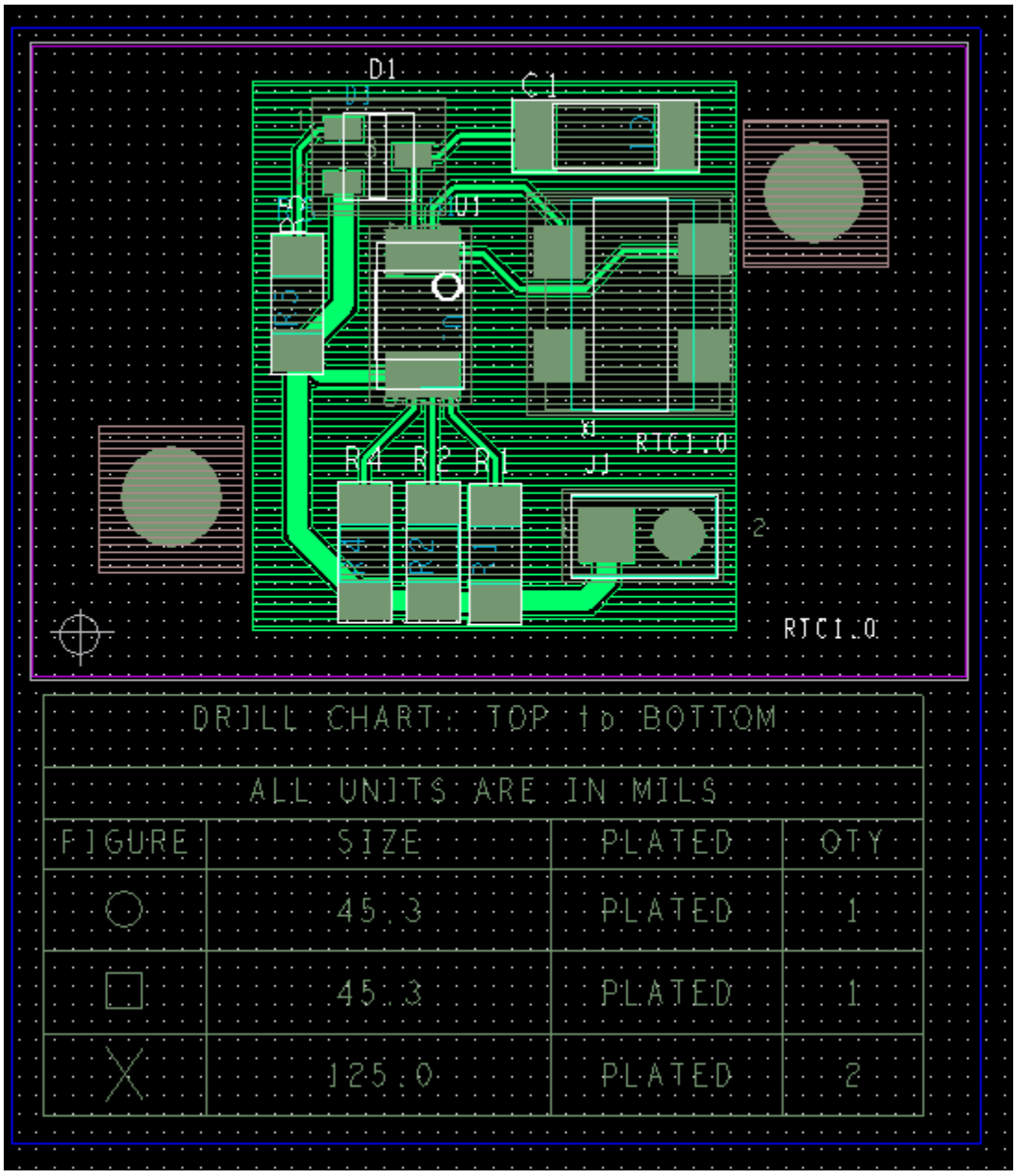
PCB Layout for RTC circuit

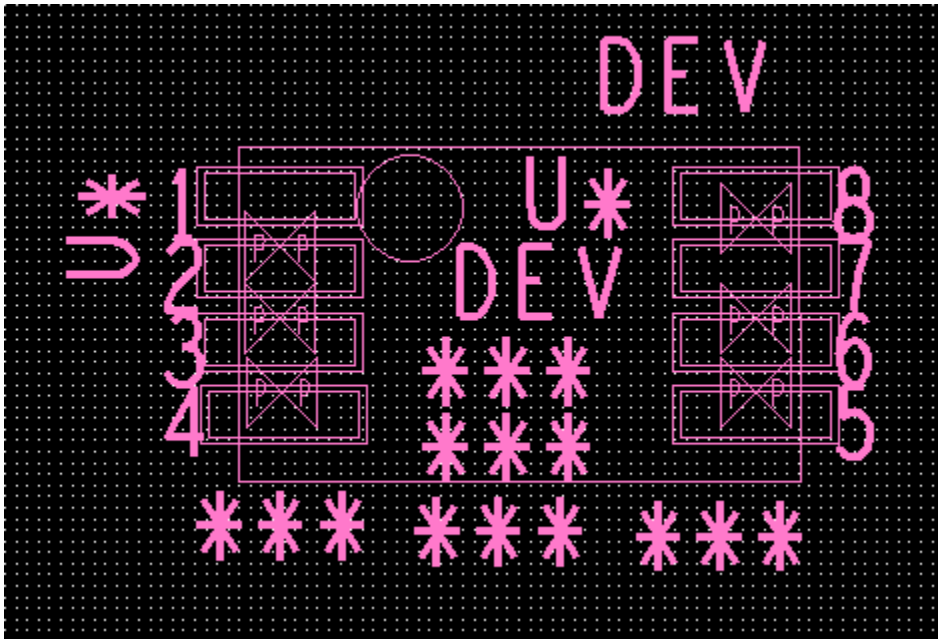
To learn the nuances of PCB layout the circuit of the RTC was used for PCB layout.



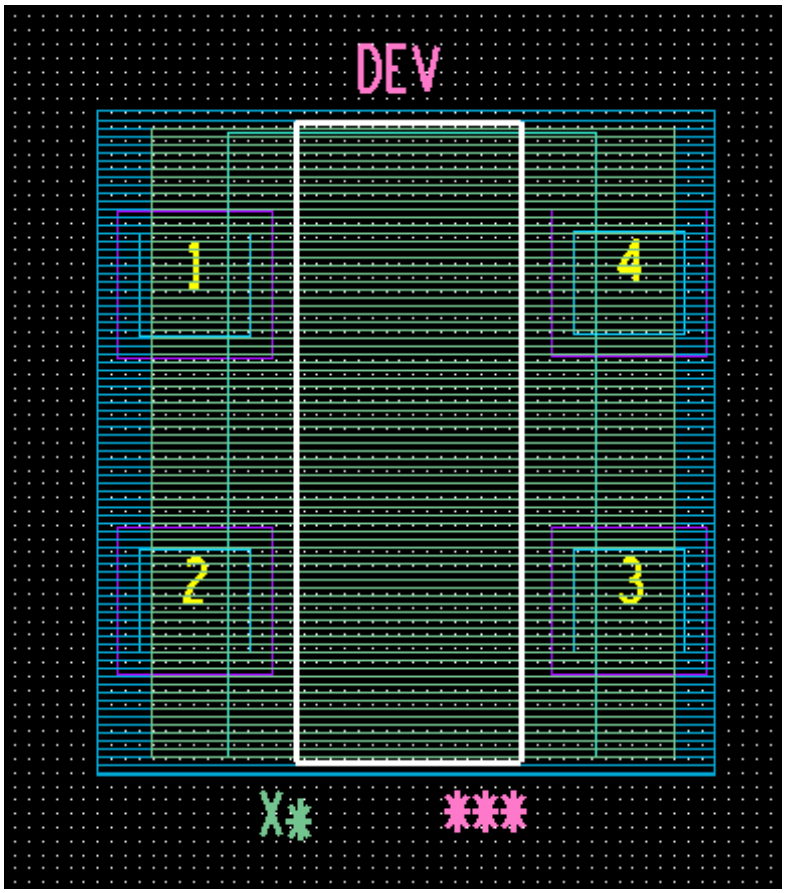
I used the allegro PCD designer for the task. This design includes all custom padstacks as in the datasheets of the respective components. First of all the parts library was built by me according to the specifications in the datasheet. The software IPC land pattern viewer proved very helpful to obtain dimensions of all packages of components.

Below are the footprints of individual parts in the design.

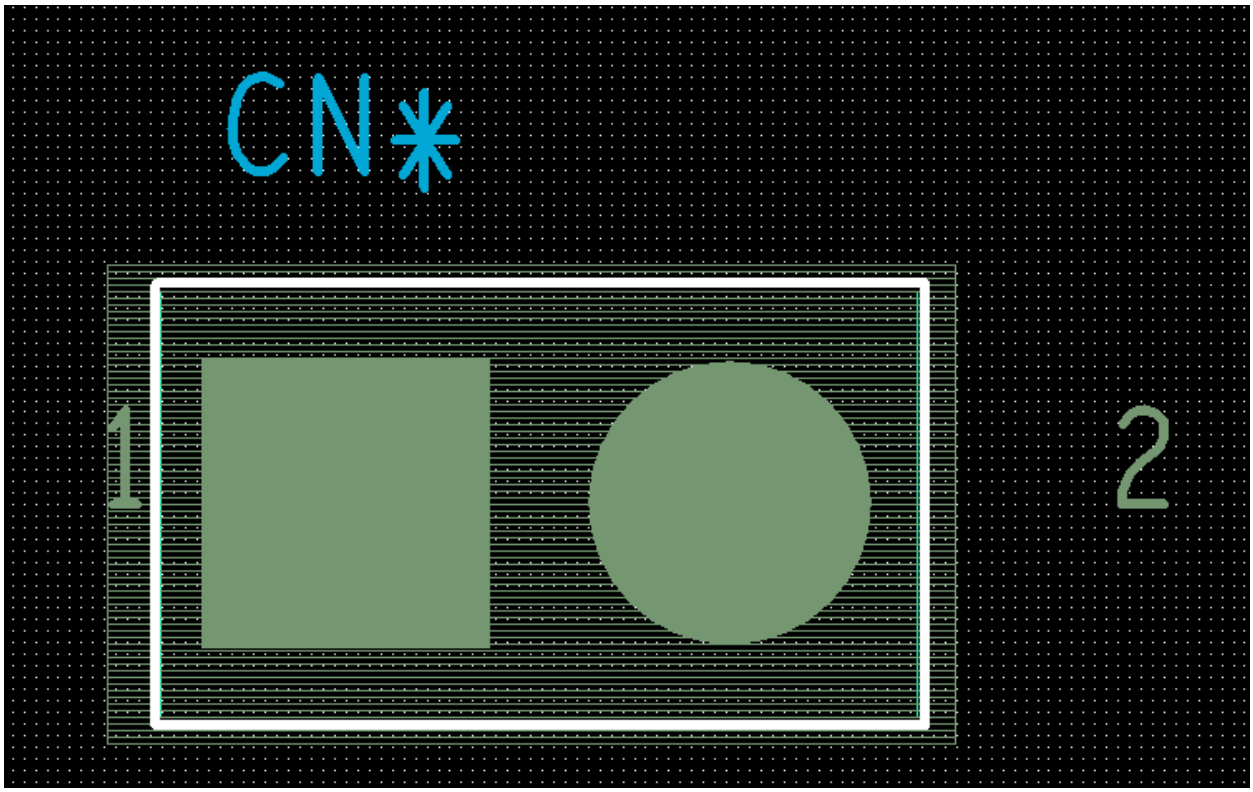
The RTC chip DS1339



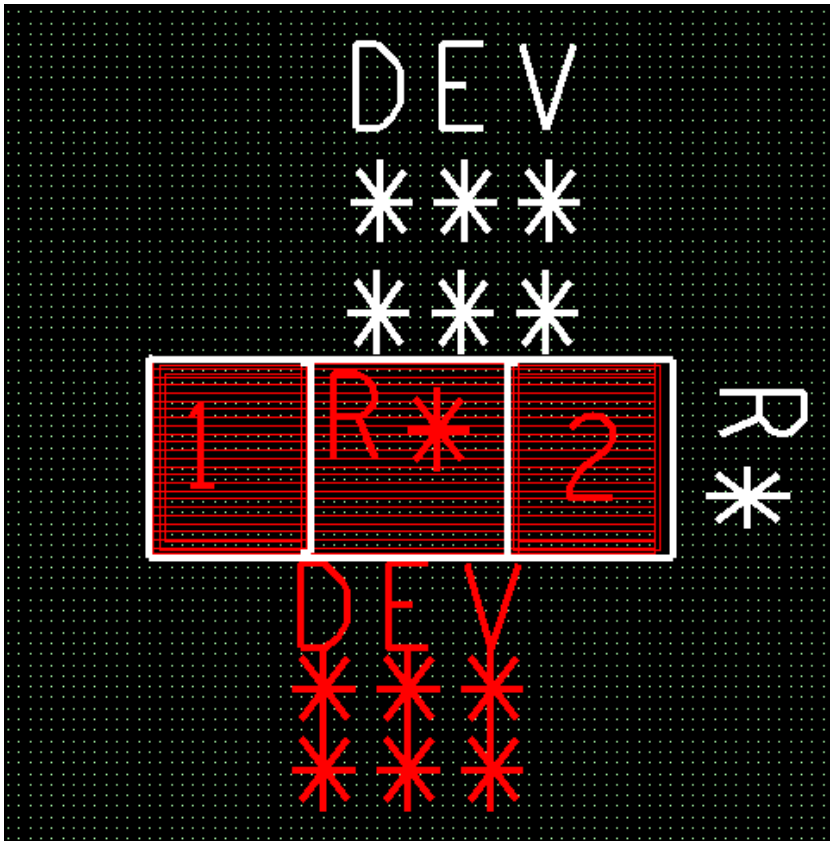
The SP-T3 SMT crystal footprint.



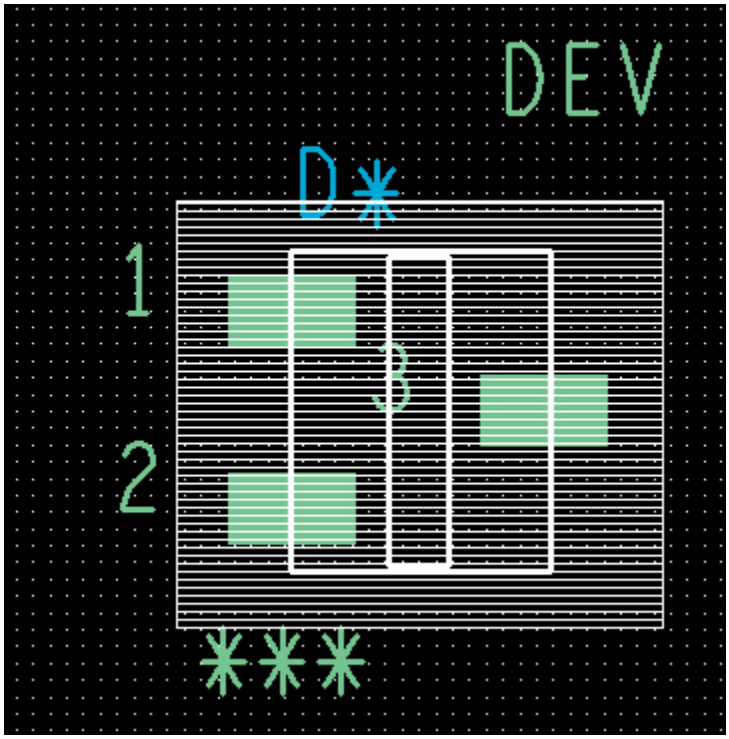
2.54mm 2 pin header footprint with custom pad stack as in respective datasheet.



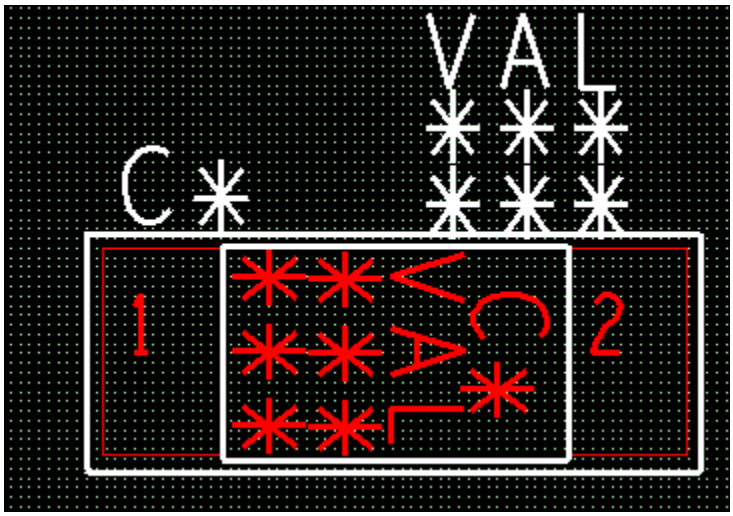
SMD resistor footprint



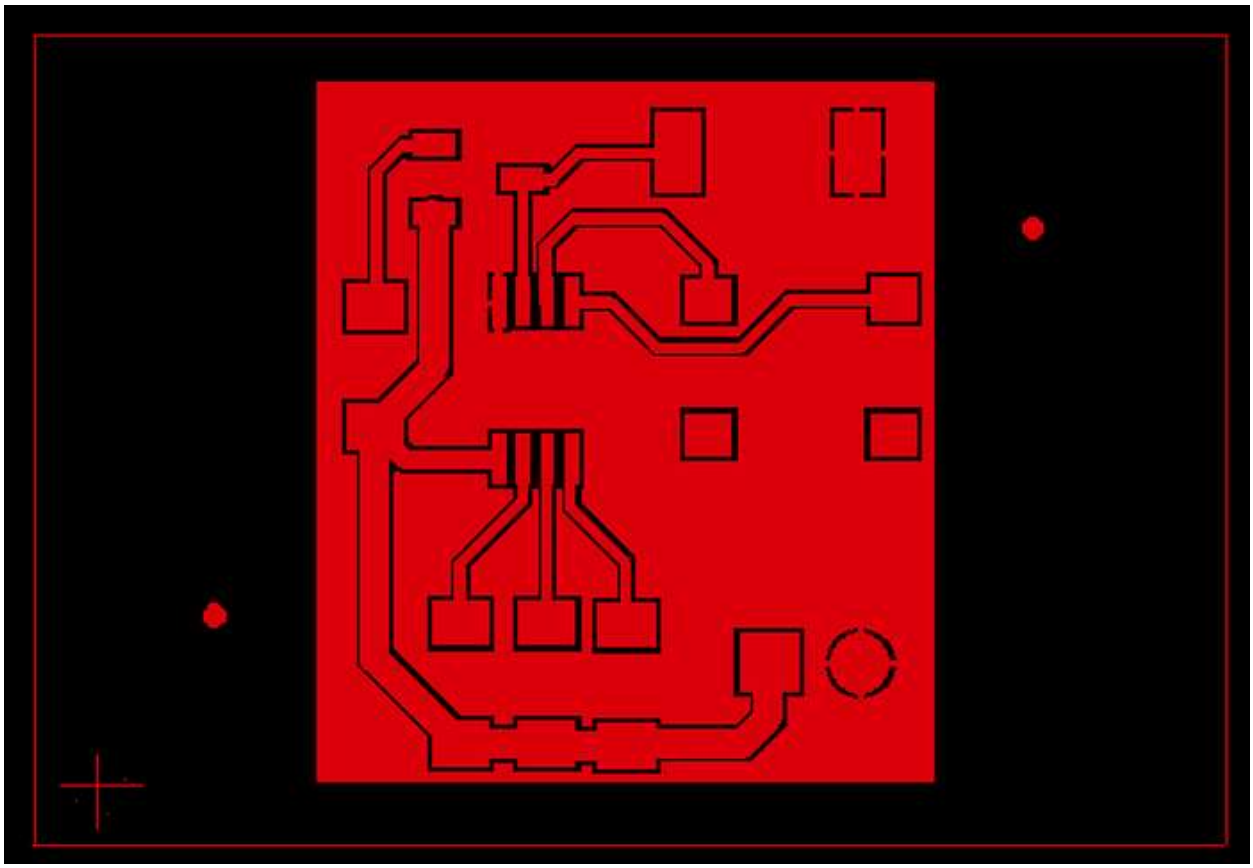
800mA Low dropout voltage regulator AMS1117



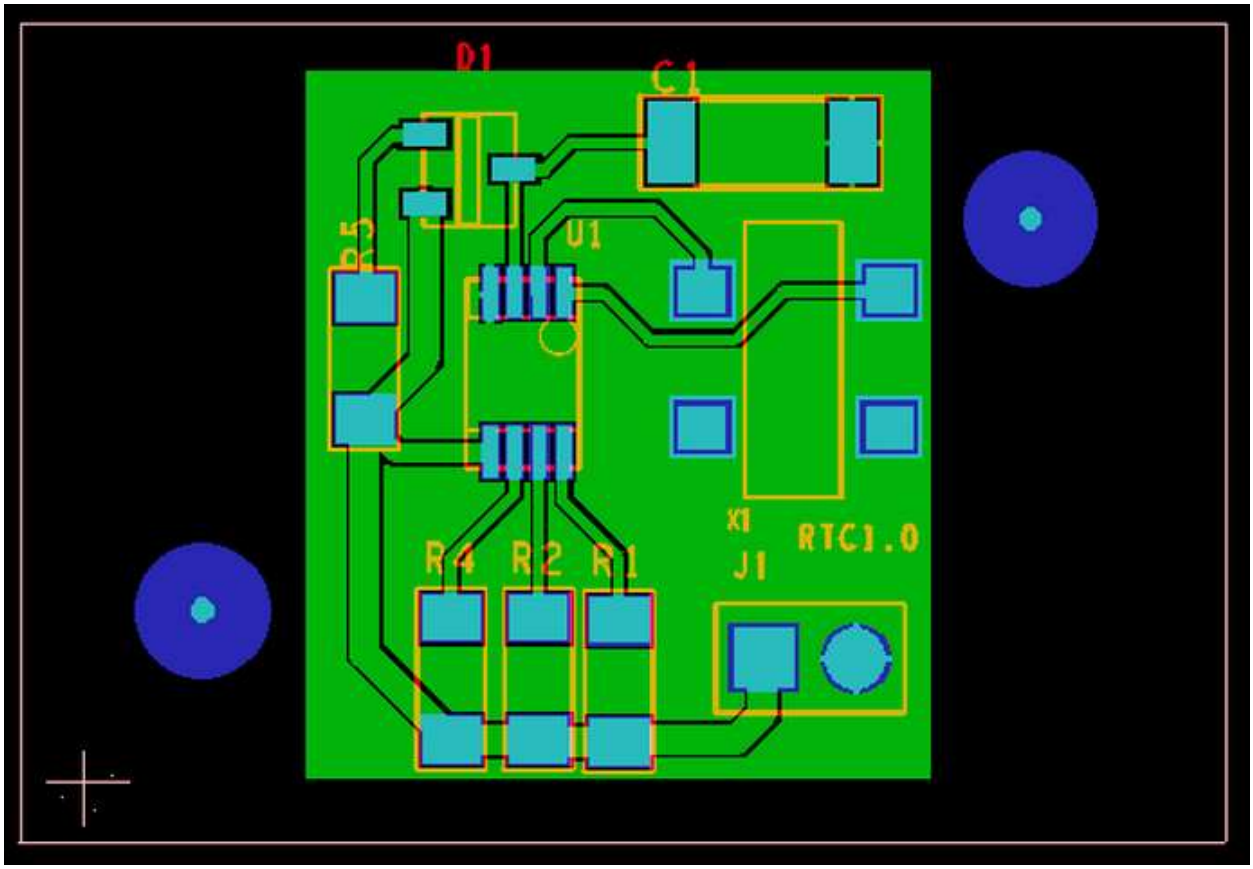
SMT capacitor footprint.



The following is a gerber out of the TOP etch layer in the board. The gerber out generated by the PCB Editor artwork generation tool. Each film has a marked reference point and an outline for reference. All the parts of the board not containing the connections are designated to be the ground plane to absord any inductance generated by the conducting lines. The angles of turning are 45degress only to avoid EMI at 90 degree turns.



The following gerber out is with the silkscreen and the soldermask layers added.



The following gerber out shows the Drill chart with 4 drill holes 3 for the 2.54 mm header and 2 drill holes for the board.