**SCOPE OF WORK:**

This re-spin represents the Prototype run 3 for the Power Factor Correction circuit of the electrosurgical generator project, T800. The learnings from Prototype run 2 shall be taken as inputs for the PCB redesign. This design will undergo peer review in order to improve design.

**PERIOD OF PERFORMANCE:**

All work must be scheduled to complete within this timeframe. The period of performance will start from the day when:

**PLACE OF PERFORMANCE:**

The design changes shall be made within the USMI R&D facility. The prototype vendor, which is currently Smart Prototyping (http://www.smart-prototyping.com/) will produce the prototpyes in Qtys of atleast 5 boards per job output. The production will take place as per specifications provided in the gerber files.

**PCB DESIGN CONSTRAINTS:**

**Phase I List of Changes: Post board review**

1. Add Pull Back on each layer.
2. Add new logo with barcode.
3. Add new footprint for standoff holes (Increase hole size to accomodate 6-32), Increase the solder paste thickness.
4. Change the footprint test point.
5. Add polarities to connectors. CN1, CN2, CN3.
6. Add power layer and ground layer.
7. Add silk screen over the vias (Force tenting on top and bottom layers).
8. Make all the pin1s square (Changes would be made within the PCB library and then imported throughout the project).
9. Add strings to label components on the PCBdoc itself.
10. Make C4 and C5 holes 0.3mm diameter.
11. Add digital inrush current limiting.
12. Make the inductor through holes bigger by 0.3mm diameter.
13. Add 0.5mm diameter to D5 solder paste.
14. Buy better heat sink which is easy to attach.
15. Change the mirrored component markings on bottom layer.
16. Change the component marking font size (Using PCB list feature)
17. Change the Q1 footprint for TO-247-N to TO-247-3
18. Rectify issue with R12
19. Rectify issue with U1
20. Reduce jumper boundary by 1mm on each side
21. Make testpoint markings on top layer instead of the bottom layer.
22. Move D2 and D3 closer to each other.
23. Changing the fuse footprint

https://ssl.gstatic.com/ui/v1/icons/mail/images/cleardot.gif

**Phase 2: List of Changes, Post Peer Review (Reviewer: Dr. Taisen Zhuang)**

1. Redesign and reroute the small signal section (the controller and the gate driver sections).
2. Add holes and cutouts for inductor zip tie.
3. Add heatsink within the schematic and PCB footprint.