**PCB Documentation Check List**

PCB: SM1000-D \_\_\_\_\_\_\_\_\_\_ ASM: \_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_

VARIANT: None\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_

DATE: 01/23/15\_\_\_\_\_\_\_\_\_\_\_ By: RGB\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_

1. [ ] Update revision levels and titles on schematic and PCB.
2. [ ] Run ERC on schematic & fix any errors.
3. [ ] Run DRC on pcb & fix any errors.
4. [ ] Run *drillcfg.* Generates *filename.drl*
5. [ ] Run Tools/drill legend from menu and generate drill table(drill legend) on PCB.
6. [ ] Run *BOM-AM-19* on schematic and check for undefined parts.
7. [ ] Generate *BOM-filename.csv*  & BOM-filename.xls files. Use parts data base & update/add new parts.
8. [ ] Generate BOM-filename.pdf file and move to MFG folder.
9. [ ] Generate *LOC-filename.csv & LOC-filename.xls* files.
10. [ ] Generate LOC-filename.pdf file and move to MFG folder
11. [ ] From Print menu, generate *Schematic* SCH-filename.PDF file and move to MFG folder.
12. [ ] Generate DRT-filename.pdf (DRILL TABLE) and copy to MGF folder.
13. [ ] Run *CAM* (Exelon & board*)* jobs on board.
14. [ ] Move *Gerber* files to GERBER folder
15. [ ] Use *Viewmate* to check layout & *produce GBR-filename.PDF* file of all Gerber layers & move to MFG folder.
16. [ ] Generate assembly AST-filename.pdf and ASB-filename.pdf drawings and move to MFG folder.
17. [ ] Update readme.txt files(gerber & manufacturing) (see other cards) for Gerber & MFG folders.
18. [ ] Add dimensions to PCB, save as DIM-filename.pdf file and move to MFG folder
19. [ ] *Generate* GERBER-filename.zip and MFG-filename.zip in GERBER & MFG folders respectively.
20. [ ] Move .sch, .brd, .lbr, MFG.zip & GERBER.zip to RELEASE Folder.