

## PCB Documentation Check List

### PCB Documentation Check List

PCB: \_\_\_\_\_ ASM: \_\_\_\_\_

VARIANT: \_\_\_\_\_

DATE: \_\_\_\_\_ By: \_\_\_\_\_

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.