**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 *BOM-AM-19* on schematic and check for undefined parts.
5.  [ ]  Generate *BOM-filename.csv*  & BOM-filename.xls files. Use parts data base & update/add new parts.
6.  [ ]  Generate BOM-filename.pdf file and move to MFG folder.
7.  [ ]  Generate *LOC-filename.csv & LOC-filename.xls* files.
8.  [ ]  Generate LOC-filename.pdf file and move to MFG folder
9.  [ ]  Generate  *Schematic* SCH-filename.PDF file and move to MFG folder.
10. [ ]  Run *drillcfg* (inches)
11. [ ]  Run *drilllegend.ulp* and generate drill table(drill legend) on PCB.
12. [ ]  Generate DRT-filename.pdf (DRILL TABLE) and copy to MGF folder.
13. [ ]  Run *CAM*  (board & *excelon)* jobson 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.