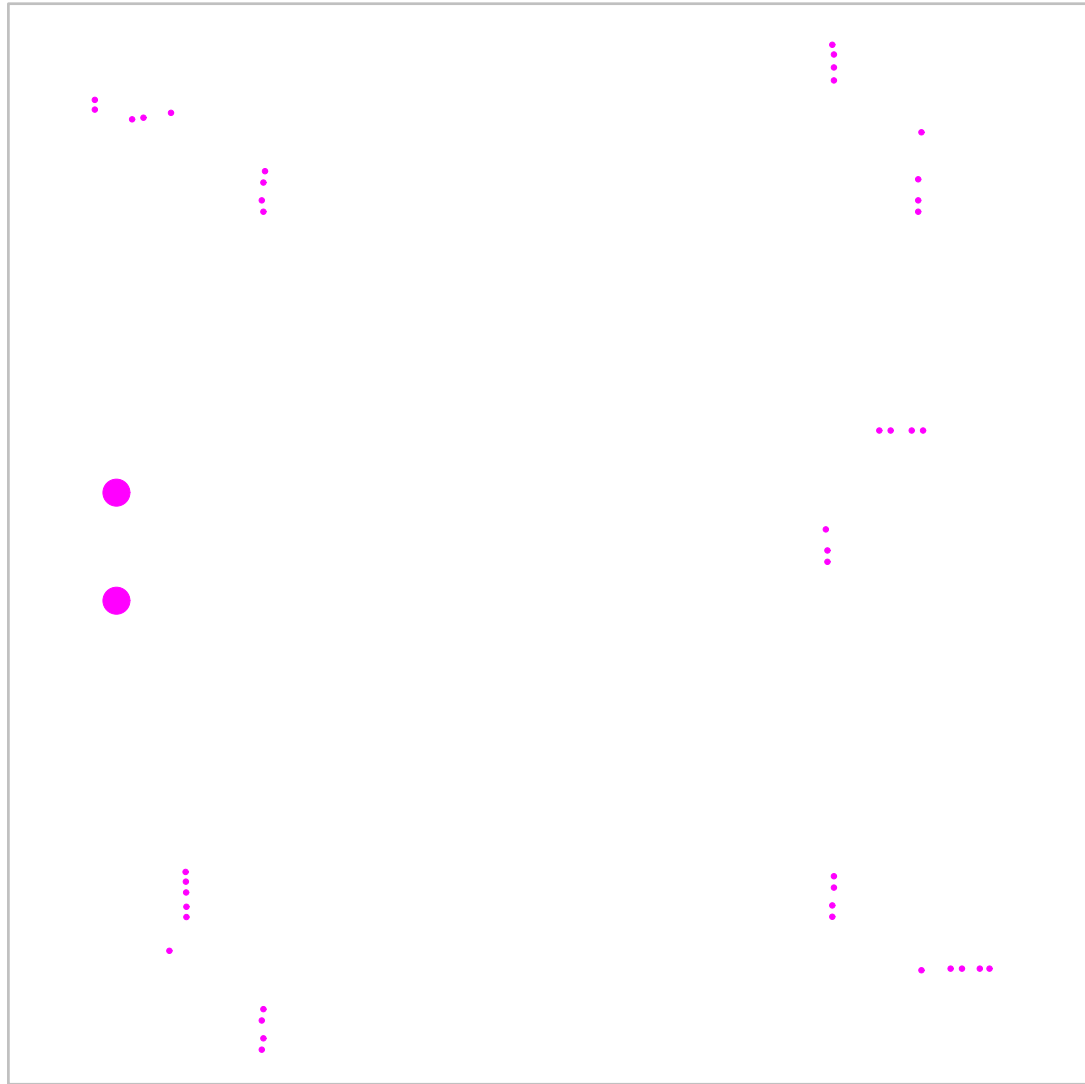
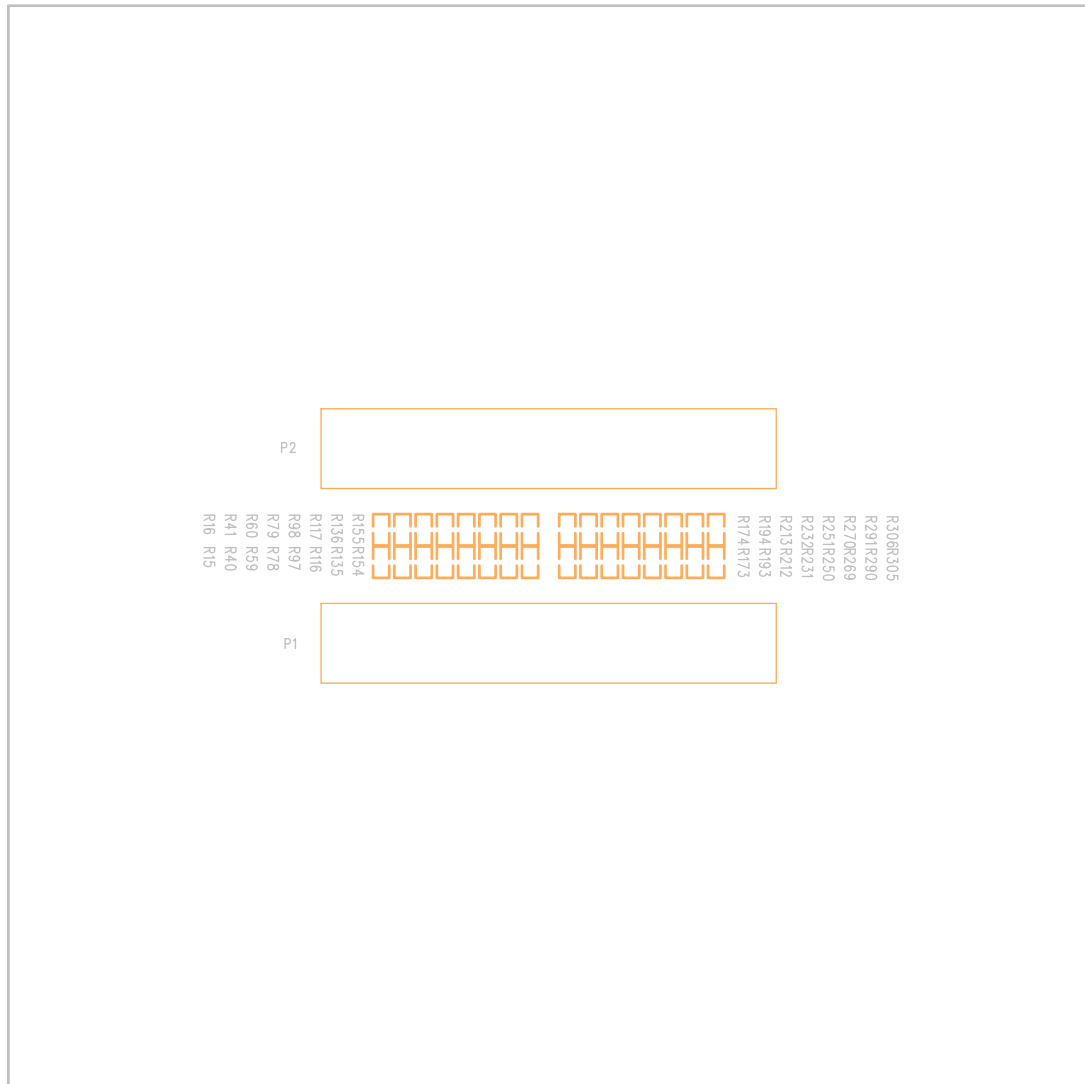


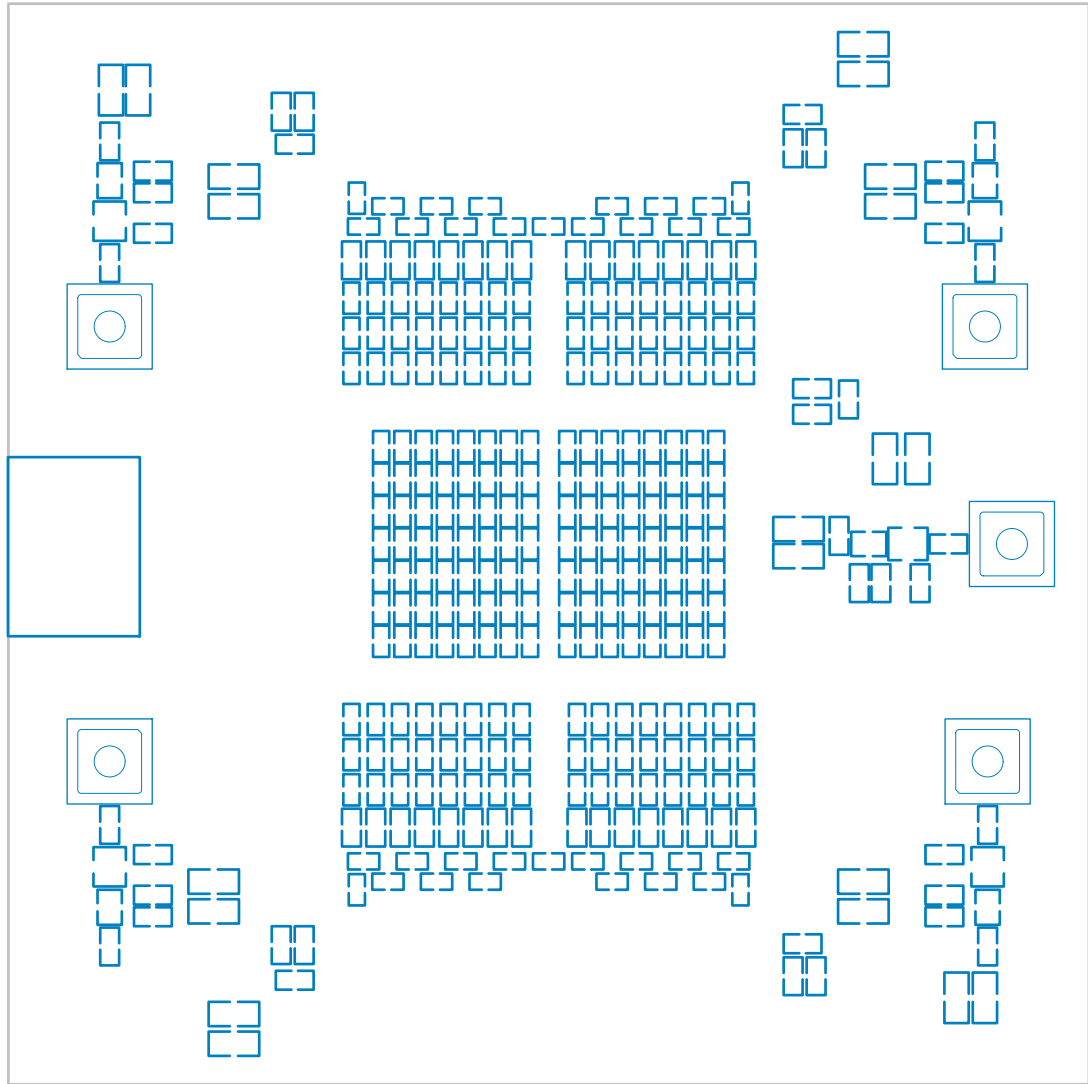
TOP



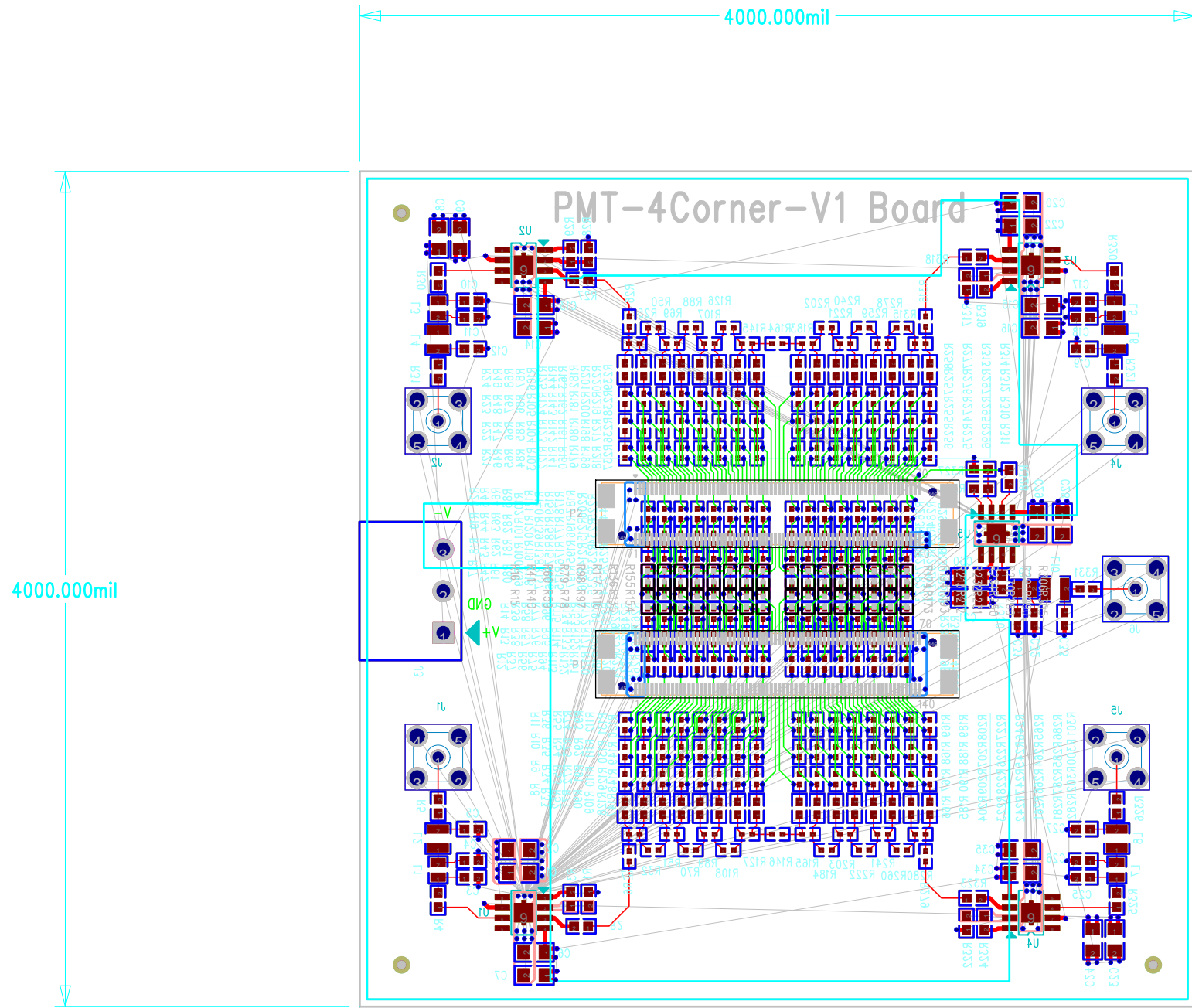
POWER_IN2



TOP
ASSYTOP

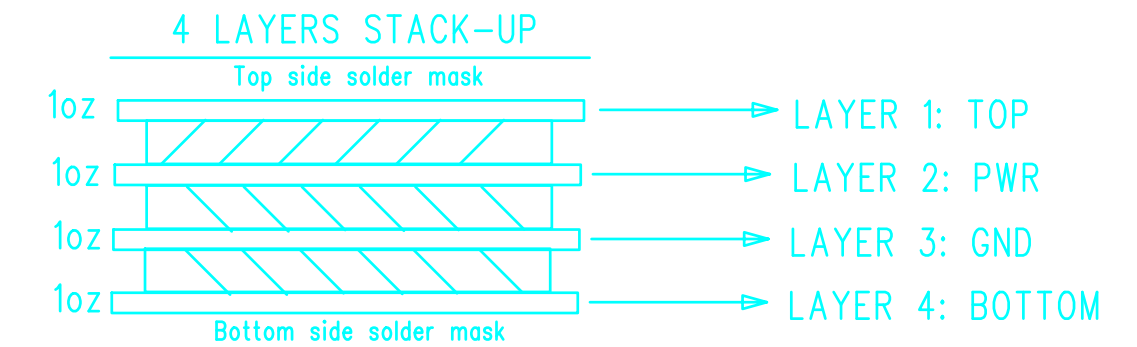


POWER_IN2
ASSYBOT



FAB NOTES:

1. Materials: 370HR or equivalent, see cross section for board thickness, copper weight and layer structures.
2. Components on the top layer and bottom layer .
3. Minimum signal trace width/spacing : 6/6 mils
4. Drilling
 - 4.1 minimum hole size is 10 mils drill
 - 4.2 all plated holes are to be plated thru and with min .001" thick
 - 4.3 all hole diameters are finished sizes
 - 4.4 all holes to be +/-0.003" from true position unless otherwise specified for PTH, or +/-0.002" if it's NPTH
 - 4.5 NC drill file has been supplied, see hole size table.
5. Front to back pattern shall register within .005"
6. Layer to layer(pattern to pattern) registration to be +/-0.005"
7. Soldermask: Both sides of board, Liquid Photo Imageable, soldermask over bare copper (SMOBC), green
8. Warpage and twist to meet IPC class II
9. Silkscreen using white non-conductive epoxy or equivalent
10. Finish:
 - 10.1 Edge to be free of loose sliver and rough edge
 - 10.2 Finished board shall not have nicks scratches, voids, exposed copper, poor plating or misdrilled holes
 - 10.3 Trace width tol=+/-0.001" of finished trace width
 - 10.4 Standard HASL or Silver is OK
11. Date code and UL rating shall be permanently marked or etched on PCB
12. Remove non-functional pad in inner layers and planes



Overall Thickness 62 mils +/- 10%

PASSPORT IN 2 TOP
 ASSYBOP