

1. Part number: 10020895-MOB2
2. Revision: Rev0, initial release
3. Contact information: Denis Klimov, MBARI, 7700 Sandholdt Rd, Moss Landing, CA 95039;
 - a. Phone: 831-775-1914,
 - b. After hours: 831-235-6515
 - c. Email: Klimov@mbari.org
4. Dimensions: 3.000"x 6.000" with 6 mounting holes - see drawing 2nd page.
5. Material: FR4 0.062" thick
6. Trace width/Space: 0.010"/0.007"
7. Smallest hole size: 0.014"
8. Copper Layers: 4 layers, Top, Ground, Power, Bottom
9. Plating: Hot Air Solder Level (HASL), Leadless
10. Copper weight: 1oz (outer), 0.5 oz (inner)
11. Solder Mask: Solder Mask Top (Green), Solder Mask Bottom (Green)
12. Legend: 1 layer Silk Screen Top (White).
13. Holes plating: all holes plated thru
14. Boards to be panelized 2-up (2 per panel), with standard rail (0.5"), scoring edge-to-edge

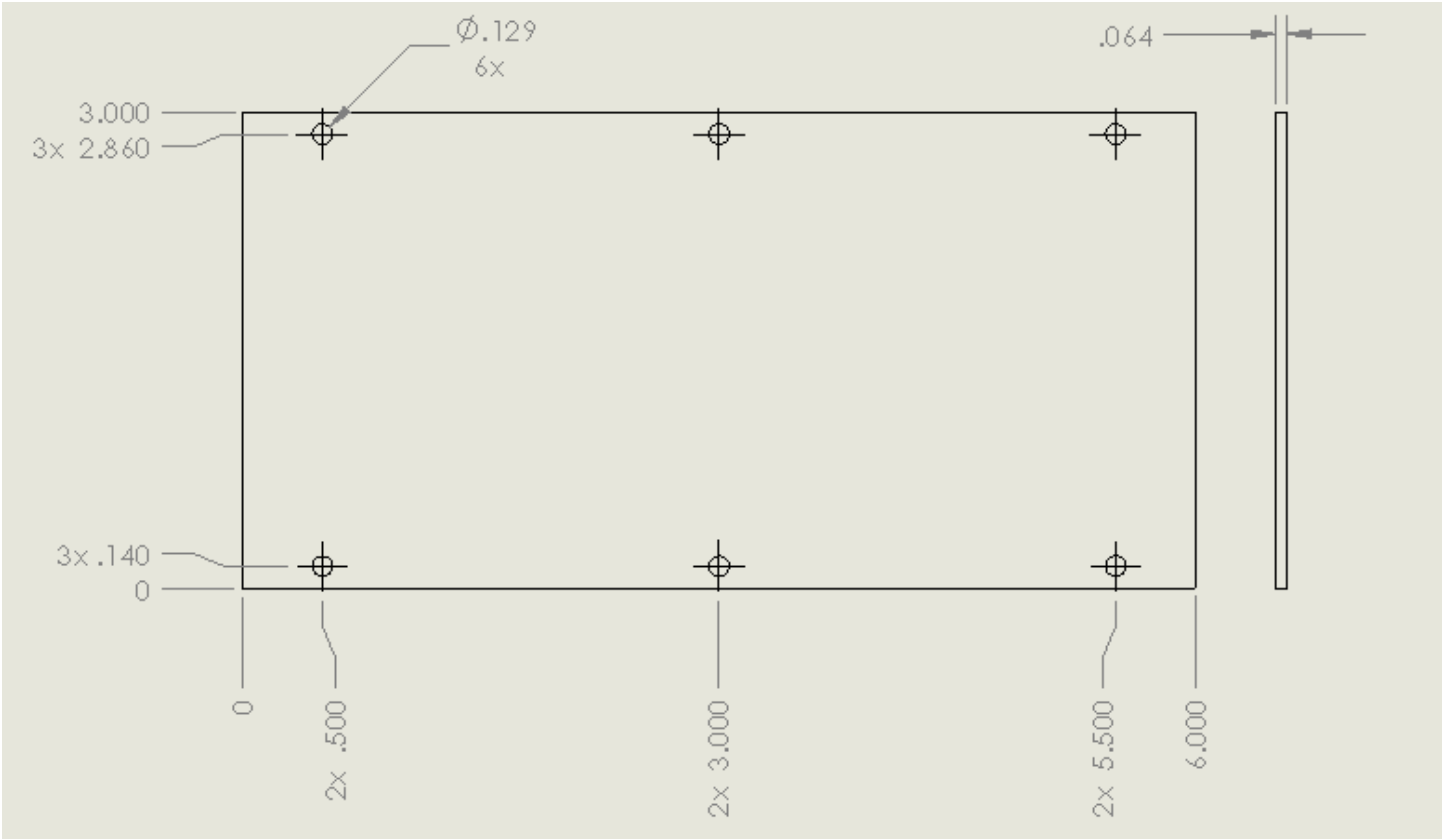
Table 1. Layers position in the board (also indicated in right upper part of the board)/ ORCAD CONVENTION

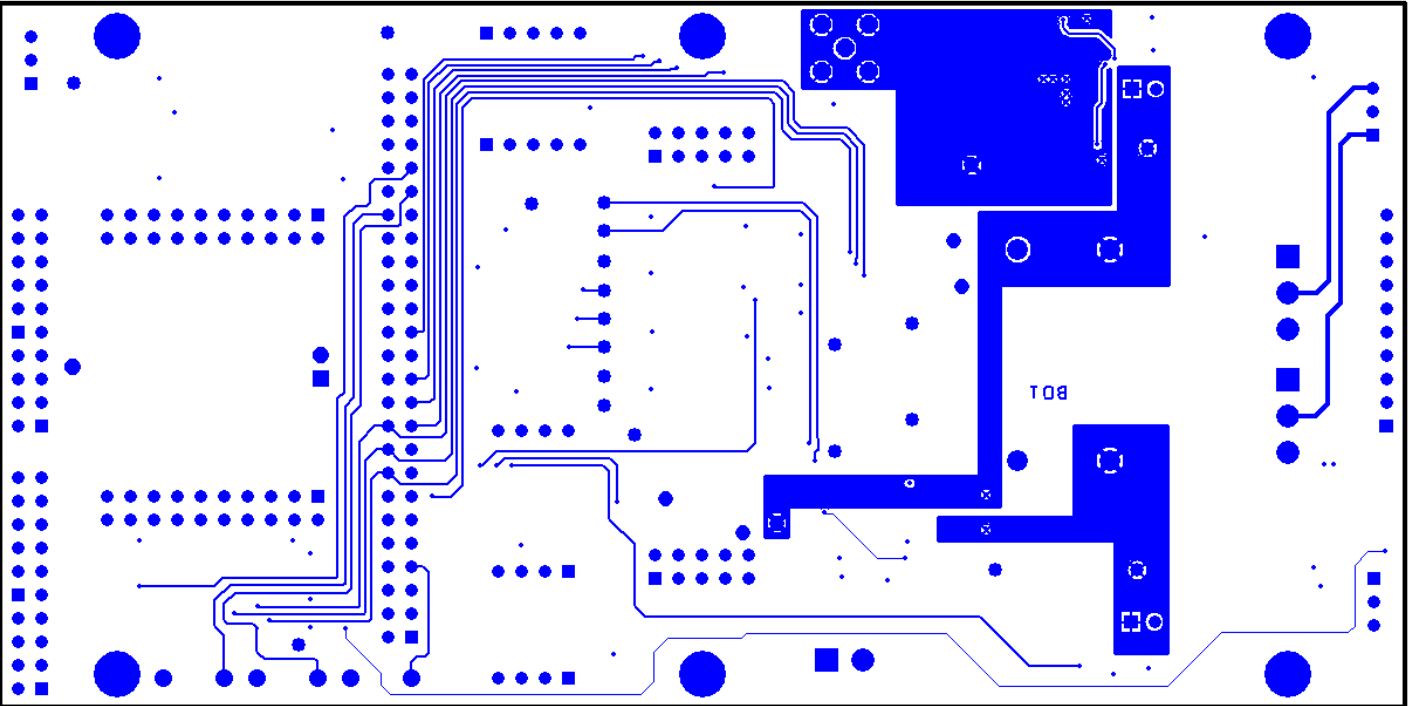
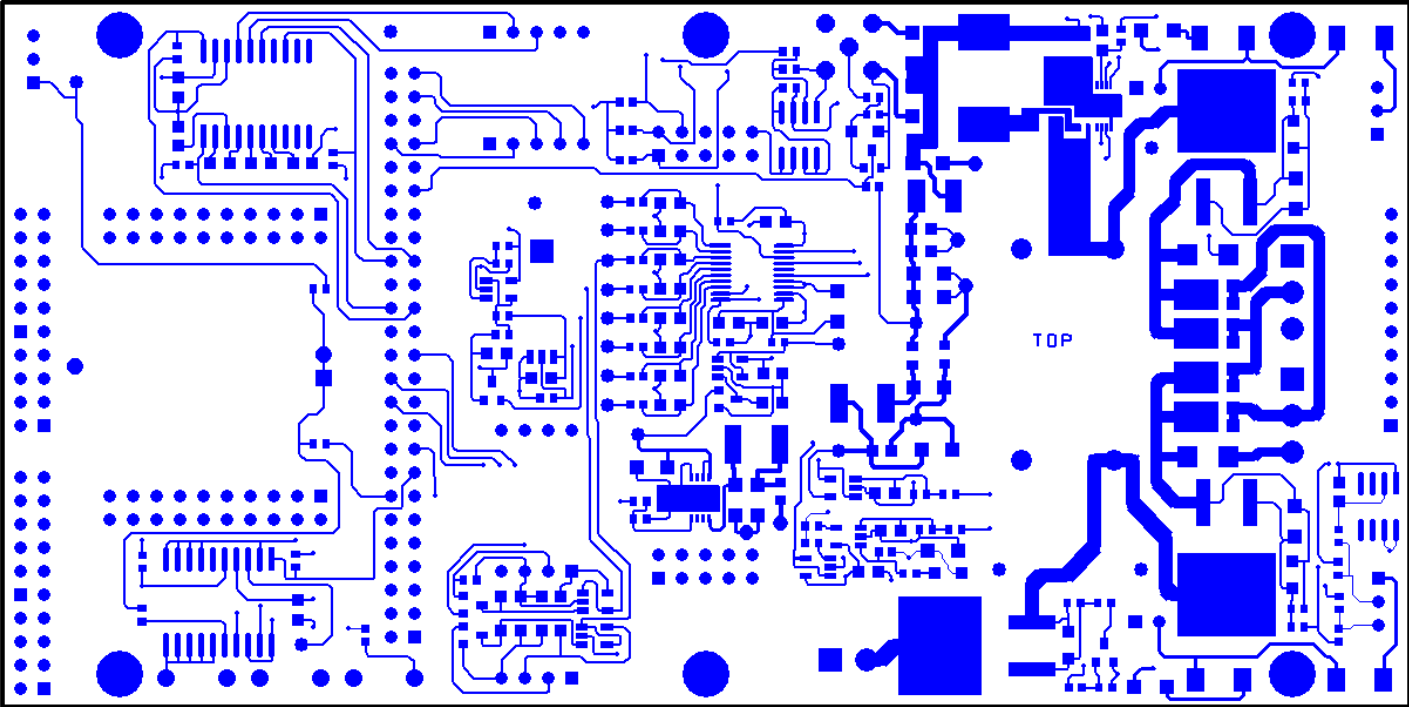
Layer number	Extended Gerber Filename	Layer position, type, name	Layer function
1	*.SST	Silk Screen Top	Annotation layer
2	*.SMT	Solder Mask Top	Solder mask
3	*.TOP	Top Layer	Routing layer
4	*.GND	Ground Layer	Plane Layer
5	*.PWR	Power layer	Plane Layer
6	*.BOT	Bottom Layer	Routing layer
7	*.SMB	Solder mask Bottom	Solder mask

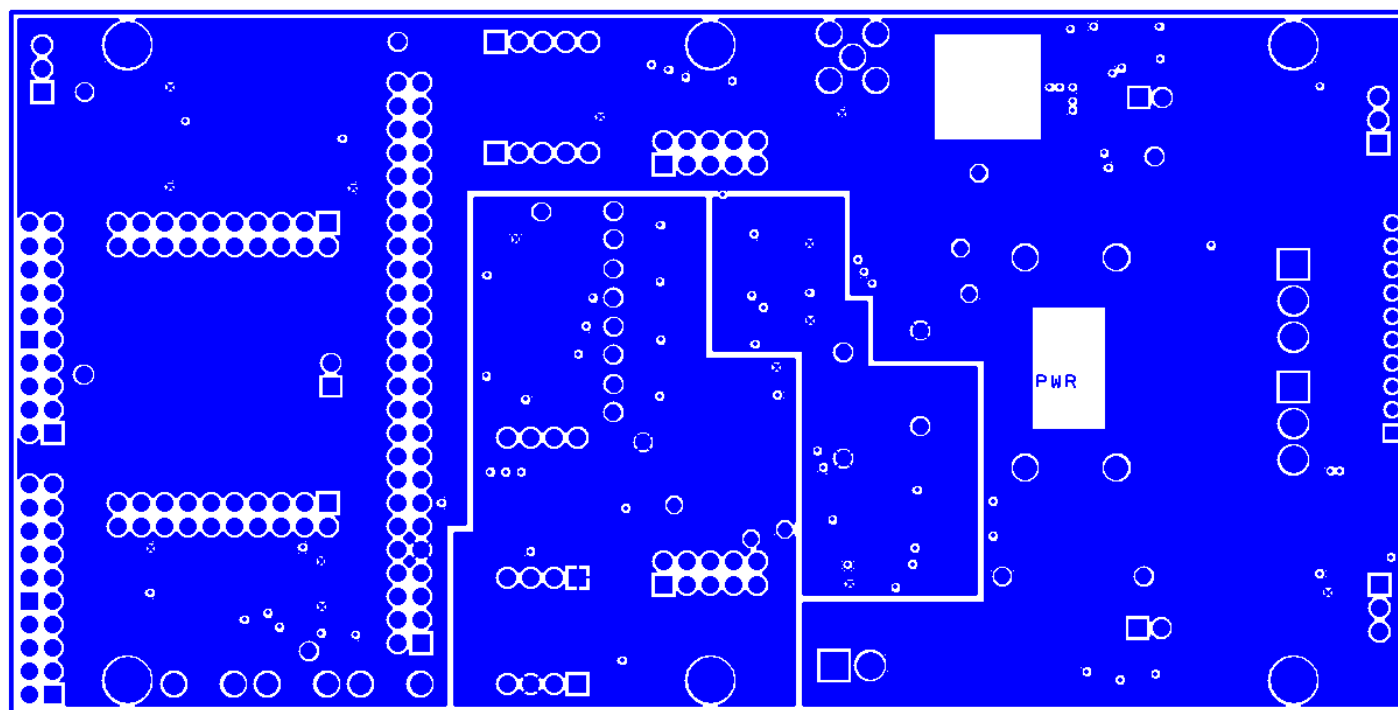
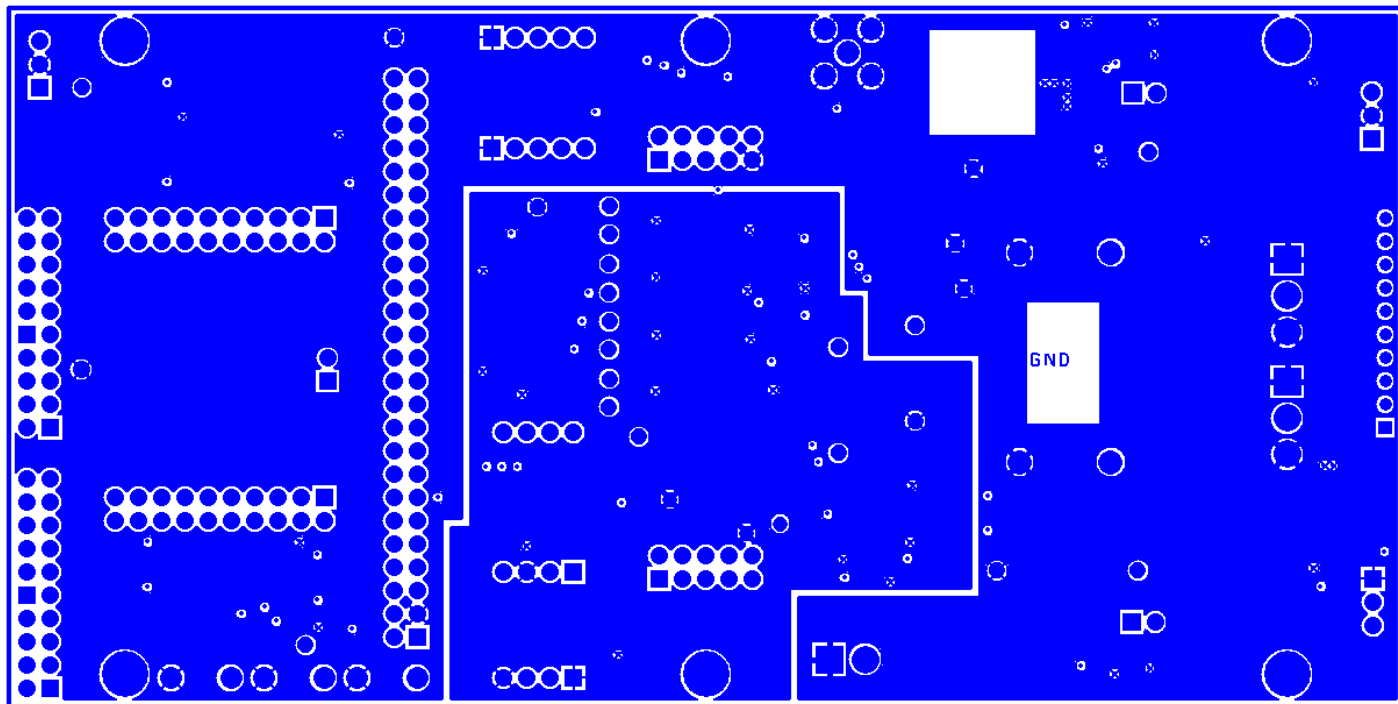
Table 2. Other files included (ORCAD 16 CONVENTION):

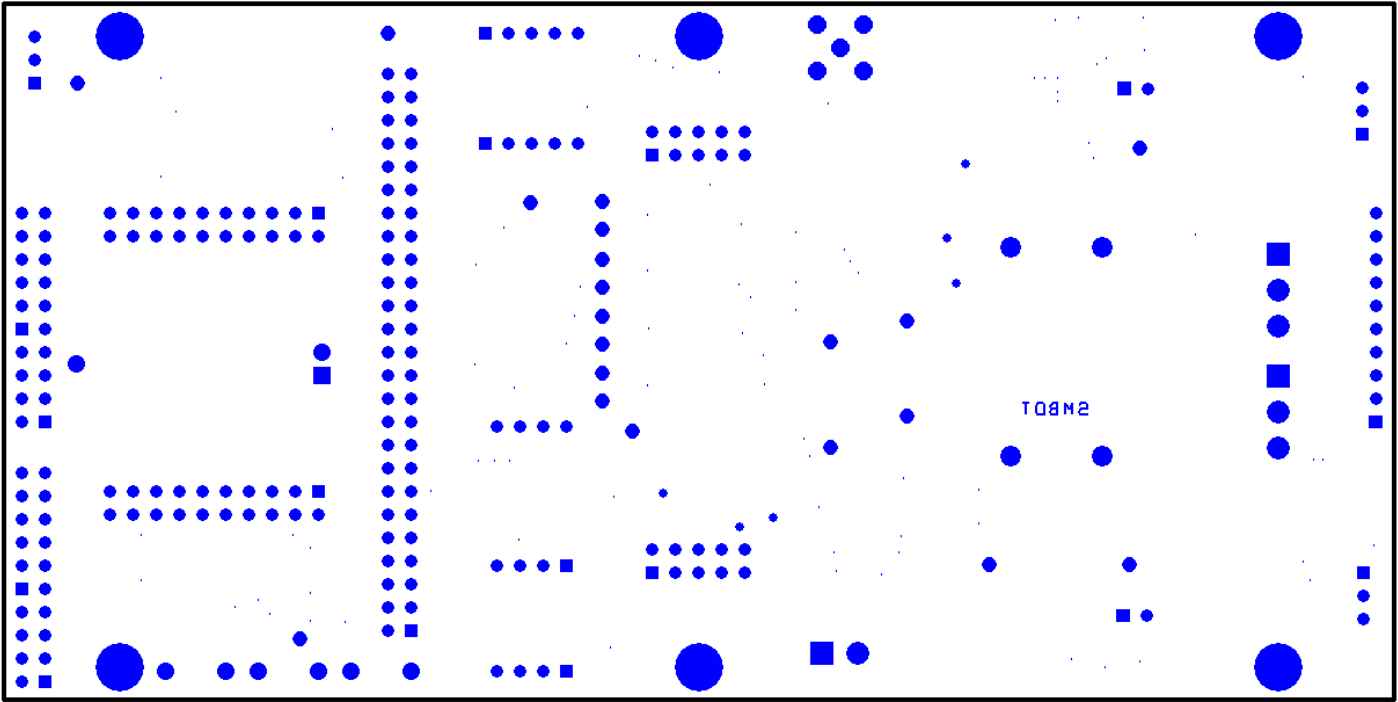
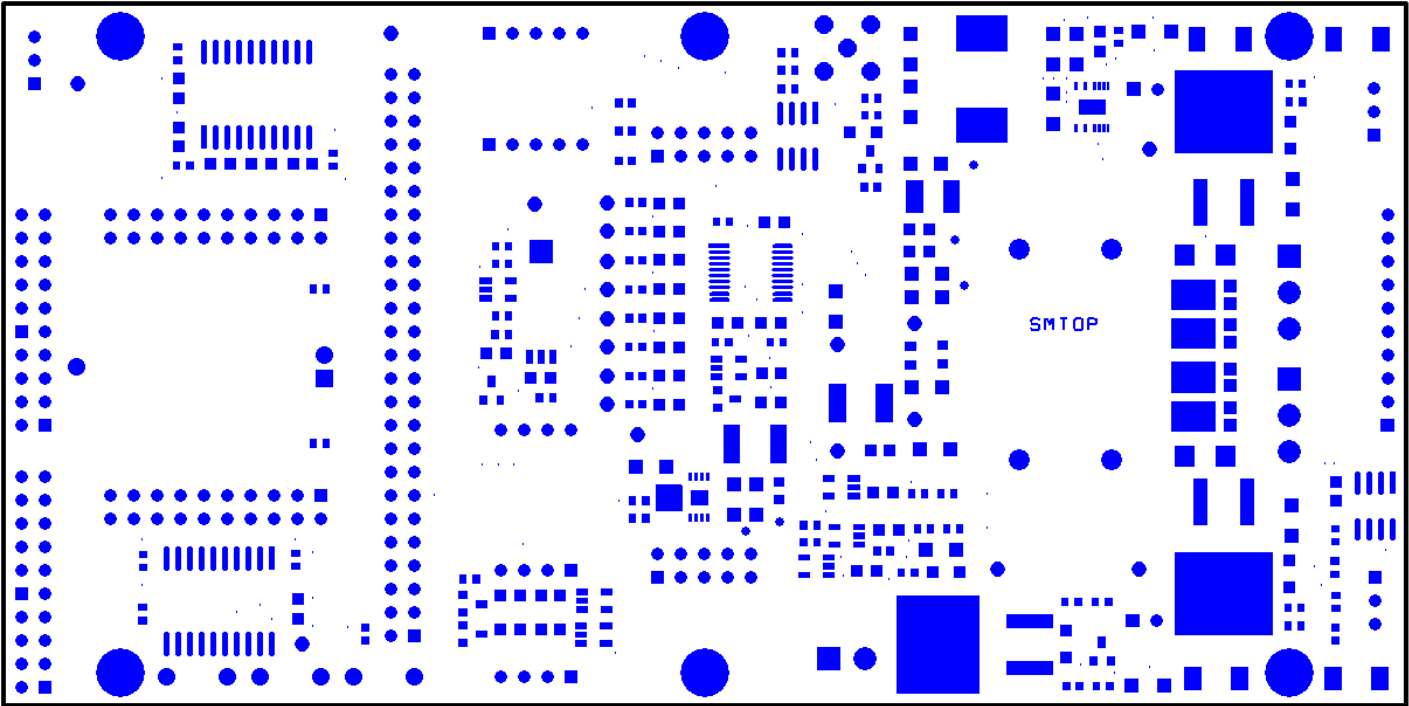
*.DRD	Drill file
*.FAB	Fabrication file with dimensions
THRUHOLE.tap	Drill file.
*.DTS	Drill Tape Summary Report
*.GTD	Gerber Design File
*.LIS	Post processor report
*.SPT	Solder Paste Top

Total: 15 files, including Read_me.doc file

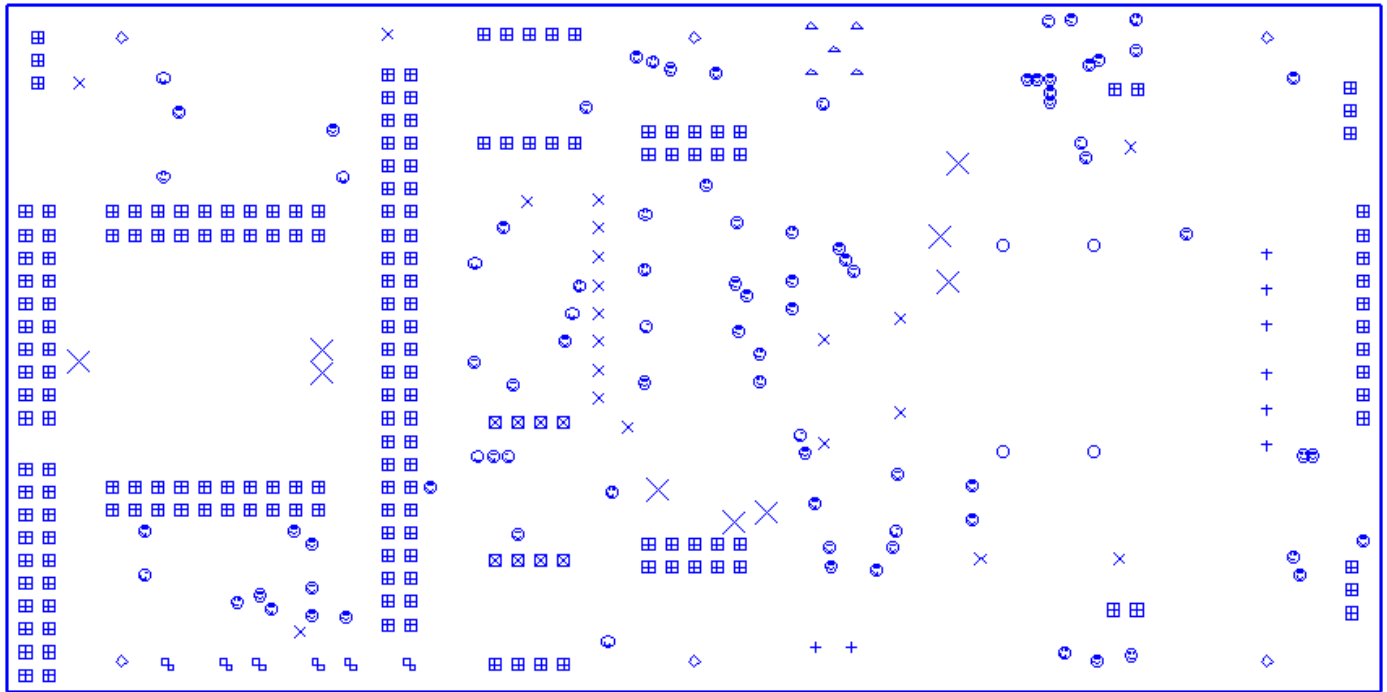








Solder paste layer



DRILL CHART				
SYM	DIAM	TOL	QTY	NOTE
⊙	0.014	+/-0.005	86	
⊠	0.034	+0.008/-0.002	8	
⊞	0.037	+0.008/-0.002	187	
×	0.038	+0.008/-0.002	20	
⊠	0.040	+0.008/-0.002	9	
⊠	0.051	+0.008/-0.002	6	
○	0.052	+0.008/-0.002	4	
△	0.060	+0.008/-0.002	5	
+	0.070	+0.008/-0.002	8	
◇	0.129	+/- .005	6	
TOTAL			339	

