

ASEK-20
85-0540-001-FAB
Originator: S. Upton

FAB Drawing

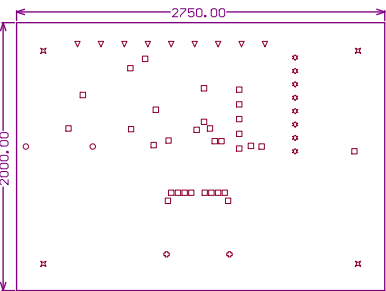
Rev 1
9/28/2012

1. Finished PCB is RoHS
2. Dimensions are in inches, unless otherwise noted.
3. Applicable Standards:
 - 3a. Manufacture in accordance to IPC-6011, IPC-6012.
 - 3b. UL Approved to a minimum category of 94V0.
4. Laminate:
 - 4a. Thickness: 0.062inch
 - 4b. Type: FR4
5. Copper:
 - 5a. Layer Count: 4
 - 5b. Exterior layers: 2oz min
 - 5c. Interior layers: 1oz min
 - 5d. Plated through holes: plate to 1mil min copper thickness
 - 5e. Trace separation: 10mil
 - 5f. Trace min width: 10mil
6. Surface Finish:
 - 6a. Immersion gold
7. Soldermask:
 - 7a. Top/Bottom soldermask required
 - 7b. Soldermask color shall be green.
8. Silkscreen:
 - 8a. Top/bottom silkscreen required.
 - 8b. Silkscreen color shall be white.
 - 8c. Min silkscreen line width: 8mil
 - 8d. Epoxy or acrylic ink allowed
9. Drill holes:
 - 9a. No blind or buried vias.
 - 9b. Hole sizes are specified after plating.
10. Mill separate or V-score according to mech1 (*.g1) layer.
11. Contact information:
 - 11a. Shawn Upton, supton@allegromicro.com, 603.626.2429
 - 11b. If fast turn board, 24hr contact info: N/A
12. Electrical test of PCB required.

Layer Stack Up Detail for: 85-0605-001 R1.PCBDOC

Layer Name	Gerber Document	Copper Thickness	Dielectric Height
Top Solder Mask	(.GTS)		0.4mil
TopLayer	(.GTL)	2.8mil	16.933mil
MidLayer1	(.G1)	2.8mil	16.933mil
MidLayer2	(.G2)	2.8mil	16.933mil
BottomLayer	(.GBL)	2.8mil	
Bottom Solder Mask	(.GBS)		0.4mil

Mechanical Layer 1 *.gml is Board Outline, slots and circular cutouts
Mechanical Layer 2 *.gm2 is footprint notes (not for fab house)
Mechanical Layer 3 *.gm3 is hole location guide
Mechanical Layer 4 *.gm4 is board outline dimensions
Mechanical Layer 5 *.gm5 is topside labels (if no silk)
Mechanical Layer 6 *.gm6 is bottomside labels (if no silk)
Mechanical Layer 7 *.gm7 is FAB drawing notes
Mechanical Layer 8 and up are not for usage by PCB board house
*.gts and *.gbs are top and bottom layer silkscreen (aka overlay)
*.gts and *.gbs are top and bottom soldermask
*.drl is NC Drill
*.apr is aperture file
*.gpb and *.gpt are pad master layers, and are not used (ignore if in zip file)
Keepout layer *.gko is for internal usage only, and is not to be used by board house



Symbol	Hit Count	Tool Size	Plated	Hole Type
□	32	15mil (0.381mm)	PTH	Round
◊	8	42mil (1.067mm)	PTH	Round
○	2	56mil (1.422mm)	PTH	Round
▽	9	62mil (1.575mm)	PTH	Round
⌘	4	125mil (3.175mm)	PTH	Round
◊	2	266mil (6.756mm)	PTH	Round
57 Total				

