

ASEK730, Board, Demo, SOIC8

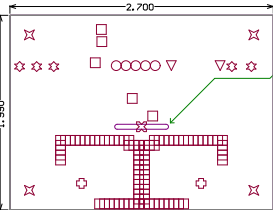
85-0739-003-FAB
Originator: S. Upton

Layer Stack Up Detail for: 85-0739-003-R1.PC8D0C

Layer Name	Gerber Document	Copper Thickness
TopLayer	(.GTL)	5.6mil
BottomLayer	(.GBL)	5.6mil

Symbol	Hit Count	Finished Hole Size	Plated	Hole Type
s	1	50.00mil (1.270mm)	NPTH	Slot
v	2	56.00mil (1.422mm)	PTH	Round
o	2	266.00mil (6.756mm)	PTH	Round
x	4	125.00mil (3.175mm)	PTH	Round
o	5	42.00mil (1.067mm)	PTH	Round
o	5	62.00mil (1.575mm)	PTH	Round
□	89	15.00mil (0.381mm)	PTH	Round
108 Total				

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 FRB drawing notes
Mechanical Layer 8 *.gm8 is top selective hard gold
Mechanical Layer 9 *.gm9 is bottom selective hard gold
Mechanical Layer 10 and up are not for usage by PCB board house
*.gto and *.gbo 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



50mil wide slot to be milled here
Feature is on mecl1 layer
Slot is to not be plated

FAB Drawing / FAB Notes and Requirements

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 for Class 2 applications.
3b. PCB shall meet acceptance criteria as required for Class 2 PCB as defined in IPC-600
3c. UL Approved to a minimum catagory of 94V0.
4. Laminate:
4a. Thickness: 0.062inch +/- 10%
4b. Type: high temp FR4
4c. this line left blank
4d. Core/prepreg thickness:
4d.1. See chart to left.
4d.2. Core/prepreg thicknesses to be roughly uniform thickness or +/-5mil from indicated
4d.3. No impedance controlled stackup required
5. Copper:
5a. Layer Count: 2
5b. Exterior layers: 4oz min
5c. Interior layers: N/A
5d. Plated through holes: plate to 1mil min copper thickness
5e. Trace separation: 10mil
5f. Trace min width: 10mil
5g. Line width reduction due to pinholes nicks or shrinking: 20% max.
5h. No impedance controlled route used; no trace width adjustment required.
6. this line left blank
7. Total board flatness shall not exceed 0.002inch per inch.
8. Artwork layer registration shall be within 0.003inch total.
9. Surface Finish:
9a. Immersion Gold
9b. No hard gold required, no *.gm8 nor *.gm9 layer provided
10. Soldermask:
10a. Top/Bottom soldermask required
10b. Soldermask color shall be green.
10c. Soldermask finish may be matte or glossy.
11. Silkscreen
11a. Top/bottom silkscreen required.
11b. Silkscreen color shall be white.
11c. Min silkscreen line width: 8mil
11d. Epoxy or acrylic ink allowed
12. Drill holes:
12a. No blind or buried vias.
12b. Hole sizes are specified after plating.
12c. No via in pad used. No via filling required.
13. Mill separate according to mechl (*.gml) layer.
14. Electrical testing required.
15. Contact information:
15a. Shaun Upton supton@allegromicro.com, 603.626.2429
15b. If fast turn board, 24hr contact info: N/A

Rev 1
10/13/2014

