PCB decription MSP-GSSB_V2

GomSpace A/S
Langagervej 6
9220 Aalborg East
Denmark
Phone: +45 71741741

Document revision v2

CHECKED IN

Date:	02-03-2020
Program & PCB Name:	MSP-GSSB V2
Part Number:	108153-1
Revision:	A
PCB Designer:	KAME
PCB Specifications:	
Base Specification:	IPC-6013-D class 3
Material:	DuPont´s Pyra-Lux LF IPC 4204A/11
Construction:	High Density Interconnect (1+n+1)
Layer count:	4
Stackup details:	See Stack-up.pdf document
Tolerances:	Thickness: +/- 10%, Outline: +/- 0,2mm, Cluster dimensions: +/- 0,2mm
Special requirements:	
☐ Notation Top	White low-outgassing epoxy
☑ Notation Bottom	White low-outgassing epoxy
☐ Nickel/Hard Gold edge plating:	See Gold plated mechanical layer in files included below.
☐ IPC 6012 type VII vias	Through going, plugging paste filled and copper cap plated vias
☑ Copper filled Microvias	Microvias in SMD soldering pads.
☐ Surface finish:	Choose a surface from the drop-down
☐ Nickel/Hard gold contacts:	See Gold plated mechanical layer in files included below.
☐ Panelization	See Cluster.pdf document
☐ Panel gerber data required	The stepped gerber files for milling the PCB must be provided
☐ Minimum isolation distance:	100um
☐ Countersunk holes	All 2.5 mm holes countersunk by 90 degrees to 5.5mm opening from the buttom layer.
☐ PCB manufacturer logo - **Not allowed**	unless otherwise agreed
☑ PCB Manufacturer unique serial number	See specified area in mechanical layer 7
☐ Stencil data required	Stencil data shall be based on compensated production files
☐ Electrically test to be done.	In accordance with IPC-9252, test level B
☐ Peelable Solder Mask:	
☐ Impedance controlled nets	

Files included in data package

	File Description	File Name	Format	Comments
☑	Read-Me File	ReadMe.pdf	ACROBAT	This Document
☑	Outline (Mechanical 4)	xyz.GM4	gerber RS274X	Board Outline
☑	Top Layer	xyz.GTL	gerber RS274X	Top layer
☑	mid1	xyz.G1	gerber RS274X	Mid layer 1
☑	mid2	xyz.G2	gerber RS274X	Mid layer 2
☐	mid3	xyz.G3	gerber RS274X	Mid layer 3
☐	mid4	xyz.G4	gerber RS274X	Mid layer 4
☐	mid5	xyz.G5	gerber RS274X	Mid layer 5
☐	mid6	xyz.G6	gerber RS274X	Mid layer 6
☐	mid7	xyz.G7	gerber RS274X	Mid layer 7
☐	mid8	xyz.G8	gerber RS274X	Mid layer 8
☐	mid9	xyz.G9	gerber RS274X	Mid layer 9
☐	mid10	xyz.G10	gerber RS274X	Mid layer 10
☑	Bottom	xyz.GBL	gerber RS274X	Bottom layer
☐	Top Paste	xyz.GTP	gerber RS274X	Top paste
☐	Bottom Paste	xyz.GBP	gerber RS274X	Bottom paste
☐	Top SolderMask	xyz.GTS	gerber RS274X	Top soldermask
☐	Bottom SolderMask	xyz.GBS	gerber RS274X	Bottom soldermask
☑	mechanical 7	xyz.GM7	gerber RS274X	Area for manufacturer serial number
☐	mechanical 5	xyz.GM5	gerber RS274X	Notation top - white low-outgassing epoxy
☑	mechanical 6	xyz.GM6	gerber RS274X	Notation bottom - white low-outgassing epoxy
☑	drill	See NC Drill Files folder	gerber RS274X	Drill files
☐	ODB++	See ODB++ Files folder	ODB++	Netlist etc.
☐	Panel	Cluster.pdf	PDF	Cluster panel drawing
☐	Stackup details	Stack-up.pdf	ACROBAT	

All files are in millimeters and showed from top view.
Format: 4:3
Any changes/production optimizations shall be approved by GomSpace.

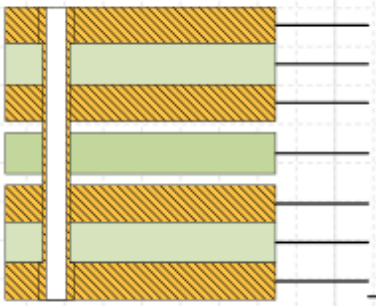
360 degree annularring verification by mandatory use of IPC type F or type R or similar registration test coupon is required.
The electrical registration test coupons shall be placed in min. all 4 corners of production panel, and be designed so breakout of innerlayer annularring for all drill sequences can be detected.

As delivery documentation Gomspace require the following items;
Boards from batch approved acc. to qty from PO
CoC with serial numbers noted of shipped boards
1 Thermal stress micro section from each production panel where boards originate from
1 test coupon with min section type A/B/D/E acc. to IPC-2221 per panel where boards originate from

Gomspace require PCB manufacturer to keep on storage following items for min. 10 years;
Rest of micro sections from batch
Rest of test coupons from batch
Production traveler and other production records concerning history of manufacturing.

PCB decription MSP-GSSB_V2

Layer Stack Legend



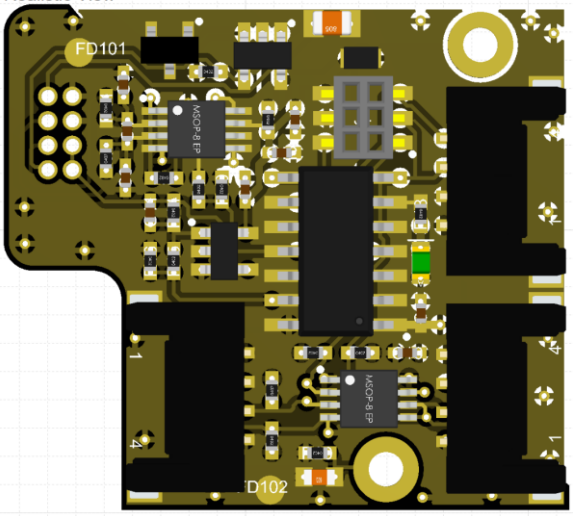
Material	Layer	Thickness	Dielectric Material	Type	Gerber
Copper	Top Layer	0.018mm		Signal	GTL
Core		*0.100mm*	*2 X Arlon 85N 106*	*Dielectric*	
Copper	Midlayer 1	0.035mm		Signal	G1
Prepreg		*0.101mm*	*Arlon 85N Laminat*	*Dielectric*	
Copper	Layer 2	0.035mm		Signal	G2
Core		*0.100mm*	*2 X Arlon 85N 106*	*Dielectric*	
Copper	Bottom Layer	0.018mm		Signal	GBL

Total thickness: 0.407mm

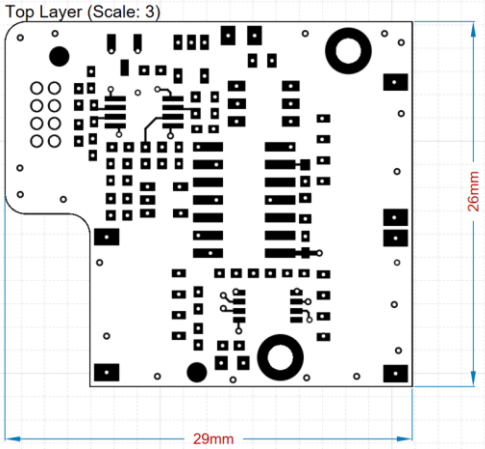
Total thickness 0.480 +/- 0.10mm. Material; Arlon 85N (Polyamide) -- IPC-4101/41. Copper foil thickness layer GTL and GBL (excluding copper plating) Cu plating in accordance with IPC-601X Hole Cu plating tables.

Note: Finished thickness may be less than specified (Depending on inner layer copper coverage), leading to a corresponding reduction of finished total thickness.

Realistic View



Top Layer (Scale: 3)



View from Top side (Scale 3)