DualCell_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:	Dualcell V2
	Part Number:	108150-01
	Revision:	A
	PCB Designer:	KAME
	PCB Specifications:	
	Base Specification:	IPC-6013-D class 3
	Material:	Glass/polyimide (GI) IPC-4101/40. Arlon 85N or Arlon 35N is preferable. Other vendors supplying IPC-4101/40 can be chosen. Full documentation on the material used shall be provided
	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:	Hot Air Solder Leveling
☐	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	DualCell_V2_PCB.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	DualCell_V2_PCB.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.

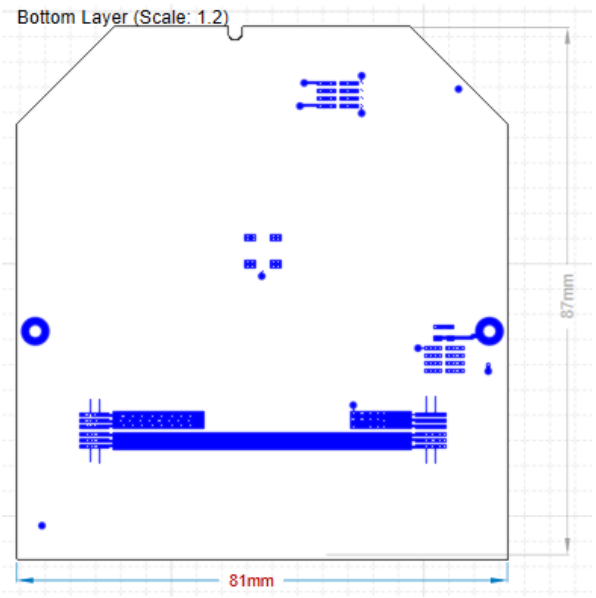
As delivery documentation Gomspace require the following items;
Boards from batch approved acc. to qty from PO
CoC with serial numbers noted of shipped boards
1 Thermal stress micro section from each production panel where boards originate from
1 test coupon with min section type A/B/D/E acc. to IPC-2221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

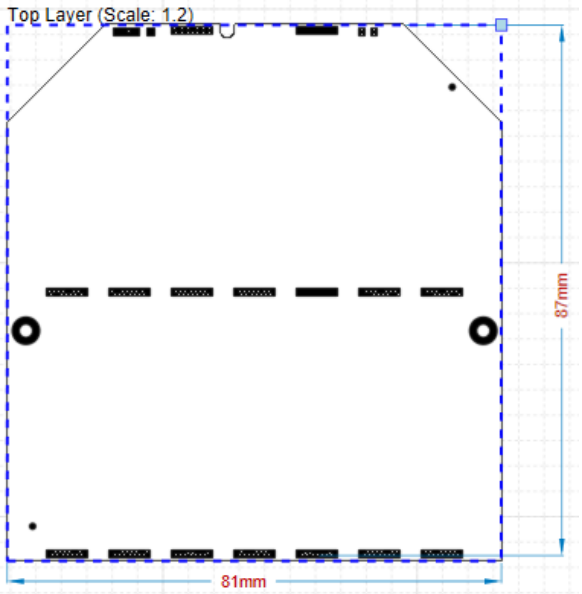
Gomspace require PCB manufacturer to keep on storage following items for min. 10 years;
Rest of micro sections from batch
Rest of test coupons from batch
Production traveler and other production records concerning history of manufacturing.

DualCell_V2

Stack up

PCB drawing





Filelist:

- Job description (this file)
- Gerber files
- NC Drill files
- ODB++ Files
- Cluster drawing