


Name	e1rxdqqw.zip	Id.	46627 - QED OK
Report Generated on	Jan 18, 2021 4:50:47 PM	Customer	InstantDFM
Board Id			

Single PCB View - Original



Summary - General - Original

PCB Size	4.8819 inch x 4.4488 inch	Surface Finish	unknown
PCB Thickness	62.00 mil	Max. Aspect Ratio on PTH	1.7
Customer Panel Size		Number of Nets	4
Copper Layers	2	Electrical Test	Single Sided
Solder Mask	Both	Drilled SMD Pads	No
Solder Mask Color	Black	SMD Pads Top	0
Legend	Both	SMD Pads Bottom	0
Legend Color	White	BGA Pads Top	0
Peeloff Mask	None	BGA Pads Bottom	0
Carbon Mask	None	Drill Hole Density	1 Holes/inch ²
Edge Connectors	No		

Summary - Copper Layer Minima - Original

Layer Type	Copper Width	Critical Copper Width	Trace Width	Critical Trace Width	Copper to Copper Clr.	Trace to Trace	Same Net Clr.	Ring	Plated	Copper to NPTH Clr.	Copper to Outline Clr.
	mil	mil	mil	mil	mil	mil	mil	mil	mil	mil	mil
Outer	>16.00	>16.00	>16.00	>16.00	1 9.84	2 9.84	>20.00	3 6.87	4 17.70	>64.00	5 3.94

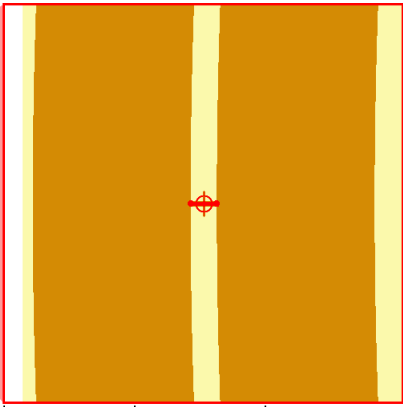
Name	e1rxdqqw.zip	Id.	46627 - QED OK
Report Generated on	Jan 18, 2021 4:50:47 PM	Customer	InstantDFM
Board Id			

Summary - Sequences - Original

Type	Sequences	Tools	Min. End Dia.	Max. End Dia.	Holes	Moves	Min. Ring on Outer	Min. Ring on Inner	Min. Hole to Copper Clr.
			mil	mil			mil	mil	mil
PTH	1	2	35.47	55.12	22	0	6.87		17.70
NPTH	1	1	129.96	129.96	5	0	>32.00		>64.00
Total	2	3	35.47	129.96	27	0	6.87		17.70

Summary Minimum Design Characteristics - Locations - Original

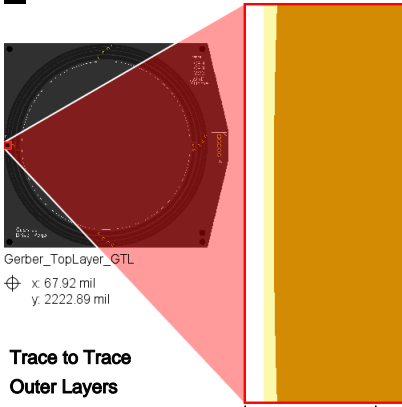
1



Gerber_TopLayer_GTL
 ⊕ x: 67.92 mil
 y: 2222.89 mil

Copper to Copper Clr.
Outer Layers
9.84 mil

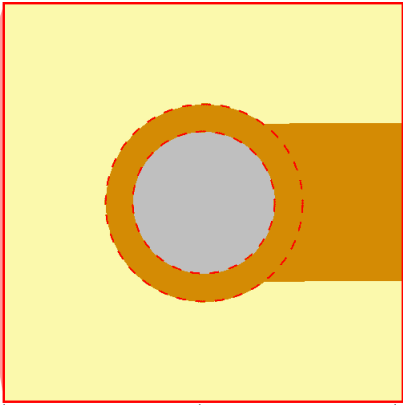
2



Gerber_TopLayer_GTL
 ⊕ x: 67.92 mil
 y: 2222.89 mil

Trace to Trace
Outer Layers
9.84 mil

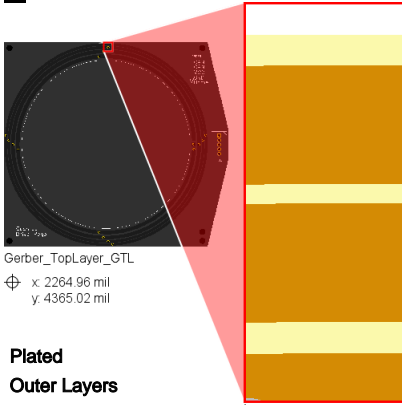
3



Gerber_BottomLayer_GBL
 ⊕ x: 4324.57 mil
 y: 2317.89 mil

Ring
Outer Layers
6.87 mil

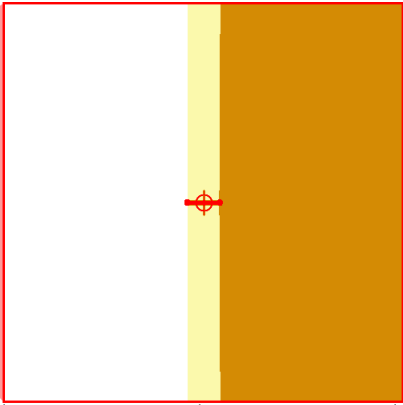
4



Gerber_TopLayer_GTL
 ⊕ x: 2264.96 mil
 y: 4385.02 mil

Plated
Outer Layers
17.70 mil

5



Gerber_TopLayer_GTL
 ⊕ x: 1.97 mil
 y: 2222.89 mil

Copper to Outline Clr.
Outer Layers
3.94 mil

Name	e1rxdqqw.zip	Id.	46627 - QED OK
Report Generated on	Jan 18, 2021 4:50:47 PM	Customer	InstantDFM
Board Id			

Stackup - Original



Pressing Stages	1
-----------------	---

Copper Layer Minima & Area - Original

File	Pos.	Copper Width	Critical Copper Width	Trace Width	Critical Trace Width	Copper to Copper Clr.	Same Net Clr.	Copper Area	
		mil	mil	mil	mil	mil	mil	inch ²	%
Gerber_TopLayer_GTL	1	>16.00	>16.00	>16.00	>16.00	9.84	>20.00	3.5582	17
Gerber_BottomLayer_GBL	2	>16.00	>16.00	>16.00	>16.00	21.26	>20.00	0.1284	1

Copper Layer Minima - Copper vs Drill - Original

File	Pos.	Ring					Copper vs Drill Clr.		Copper to Outline Clr.			
		Overall	Via	Laser Via	Comp.	Mech.	Plated	NPTH	Overall	Pad to Outline	Track to Outline	Region to Outline
		mil	mil	mil	mil	mil	mil	mil	mil	mil	mil	mil
Gerber_TopLayer_GTL	1	7.85			7.85		17.70	>64.00	3.94	17.26	3.94	>64.00
Gerber_BottomLayer_GBL	2	6.87			6.87		>32.00	>64.00	17.26	17.26	>64.00	>64.00

Drill Tools - Original

File	Tool Nr.	Span	Type	Method	Filled Via	Counter	Dia.	Tol. Min	Tol. Plus	Holes (in PCB)	Moves (in PCB)	Double Hits (in File)	Pre-drill Hits (in File)
							mil	mil	mil				
Gerber_Drill_NPTH_DRL	1	1-2	NPTH	unknown	unknown	unknown	129.96	0.00	0.00	5	0	0	0
Gerber_Drill_PTH_DRL	1	1-2	PTH	unknown	unknown	unknown	35.47	0.00	0.00	17	0	0	0
Gerber_Drill_PTH_DRL	2	1-2	PTH	unknown	unknown	unknown	55.12	0.00	0.00	5	0	0	0

Drill Tools - Drill vs Copper - Original

File	Tool Nr.	Span	Type	Method	Dia.	Min. Ring on Outer	Min. Ring on Inner	Min. Pad Size
					mil	mil	mil	mil
Gerber_Drill_NPTH_DRL	1	1-2	NPTH	unknown	129.96	>32.00		
Gerber_Drill_PTH_DRL	1	1-2	PTH	unknown	35.47	6.87		49.21
Gerber_Drill_PTH_DRL	2	1-2	PTH	unknown	55.12	11.81		78.74

Name	e1rxdqqw.zip	Id.	46627 - QED OK
Report Generated on	Jan 18, 2021 4:50:47 PM	Customer	InstantDFM
Board Id			

Sequences - Original

Span	Type	Tools	Min. End Dia.	Max. End Dia.	Holes	Min. Ring on Outer	Min. Ring on Inner	Min. Hole to Copper Clr.	Min. Hole to Outline Clr.	Min. Slot to Outline Clr.
			mil	mil		mil	mil	mil	mil	mil
1-2	PTH	2	35.47	55.12	22	6.87		17.70	24.12	disabled
1-2	NPTH	1	129.96	129.96	5	>32.00		>64.00	49.59	disabled
All	All	3	35.47	129.96	27	6.87		17.70	24.12	disabled

Rout Tools - Original

File	Tool Nr.	Type	Tool Dia.	End Dia.	Draw Length	Nibble Count
			mil	mil	mil	

Routed Holes - Original

File	Hole Nr.	Instances	X Size	Y Size	Draw Length	Nibble Count
			mil	mil	mil	

Solder Mask - Original

Side	Min. Mask to Mask Clr.	Min. Web	Min. Ring on Cu Defined Pads	Min. Ring on SM Defined Pads	Min. Mask to Copper Clr.	Fully Covered Via Holes	Partly Covered Via Holes	Half Mask Vias
	mil	mil	mil	mil	mil			
Top	>10.00	>10.00	3.94	>10.00	6.89	No	No	
Bottom	>10.00	>10.00	3.94	>10.00	>10.00	No	No	
Both	>10.00	>10.00	3.94	>10.00	6.89	No	No	No

Files - Original

Initial	Renamed	Format	Function	Position	Color
Gerber_TopSilkLayer.GTO	Gerber_TopSilkLayer_GTO	ger274x	silk	top	white
Gerber_TopSolderMaskLayer.GTS	Gerber_TopSolderMaskLayer_GTS	ger274x	mask	top	black
Gerber_TopLayer.GTL	Gerber_TopLayer_GTL	ger274x	outer	1	
Gerber_BottomLayer.GBL	Gerber_BottomLayer_GBL	ger274x	outer	2	
Gerber_BottomSolderMaskLayer.GBS	Gerber_BottomSolderMaskLayer_GBS	ger274x	mask	bottom	black
Gerber_BottomSilkLayer.GBO	Gerber_BottomSilkLayer_GBO	ger274x	silk	bottom	white
Gerber_Drill_NPTH.DRL	Gerber_Drill_NPTH_DRL	excellon2	nonplated	1-2	
Gerber_Drill_PTH.DRL	Gerber_Drill_PTH_DRL	excellon2	plated	1-2	
Gerber_BoardOutline.GKO	Gerber_BoardOutline_GKO	ger274x	cad_outline	none	
How-to-order-PCB.txt		text	document		

Comments - Original

--