DFM Report

Product Name:	
Prepared By:	
Report Rev:	**01**
Date:	
Page:	**23**

Table of Contents

Summary	**1**
PCB Image Overview	**2**
PCBA Image Overview(3D)	**3**
Package analysis(Assembly)	**4**
General assembly and process requirements(Assembly)	**10**
SMD land pattern standard(IPC7351)(Assembly)	**11**
Outer signal layer analysis(Fabrication)	**12**
Silkscreen layer analysis(Fabrication)	**15**
Solder mask layer analysis(Fabrication)	**16**
TP analysis(Test Point)	**21**
BOM Verification Report	**22**
No Library List	**23**

DFM Report

Generated by VayoPro-DFM Expert, Format v3.0

Basic Information

Customer Name	
Product Name	
Assembly P/N & Rev	
PCB P/N & Rev	

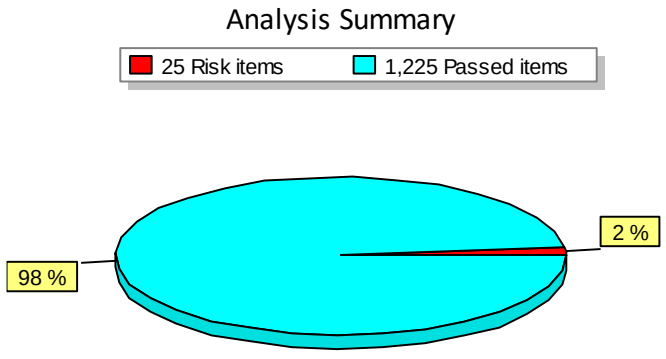
Basic Product Information

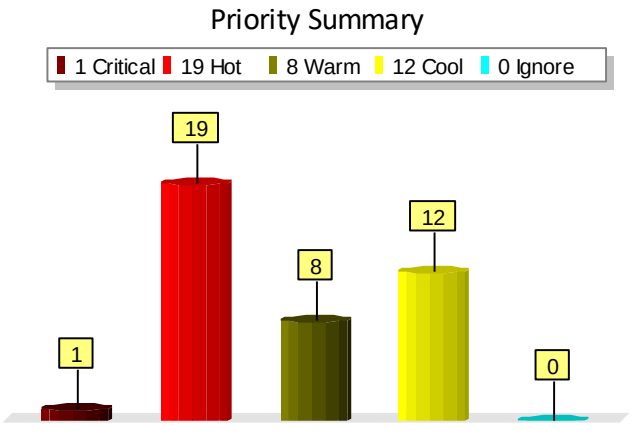
CAD Data Source	XY+GERBER
CAD File	PCB document, XY.txt
BOM File	BOM.xlsx
Matched Component (PN)	98.44%
Matched Component (RefDes)	99.58%
Size L x W(mm)	52.000 x 108.000
Thickness(mm)	1.600
Number of layers	4
Single or double reflow	Double Side
Through hole parts on 1 or both sides	None

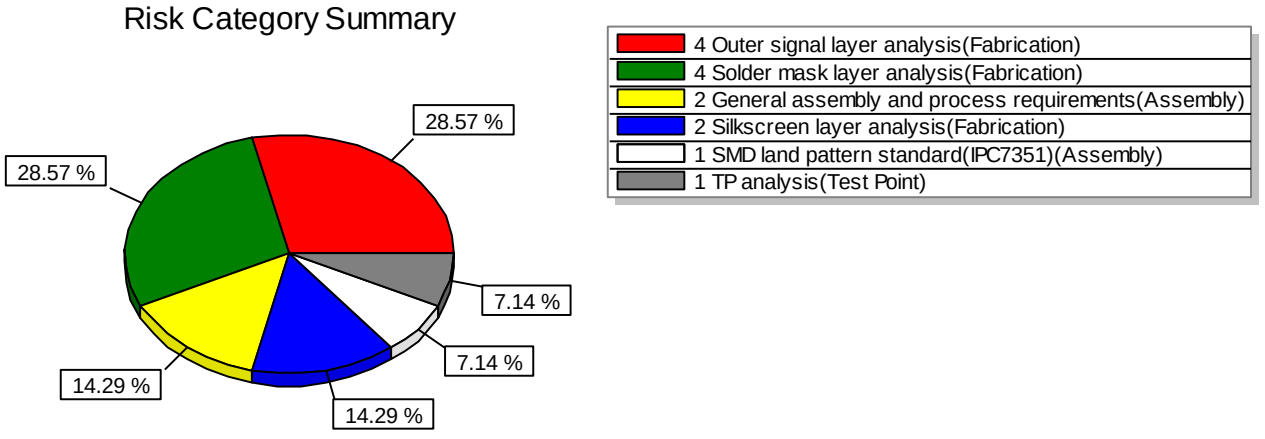
Assembly Process Flow

Printing->Mount->Reflow->De-panel

Analysis Summary





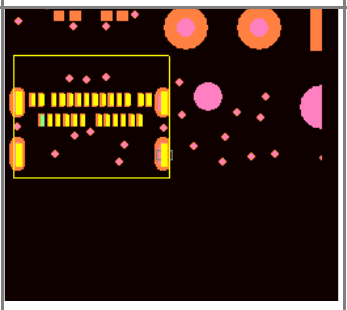


Report Items

Package analysis(Assembly)

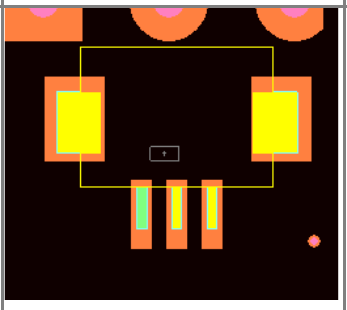
1) 124018612112A

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P/N: 124018612112A CAD Shape: CON-USBC-24pin	CON-USBC-24pin -3.862 -104.607				Location J1 from XY.txt is not match with the PCB Gerber.

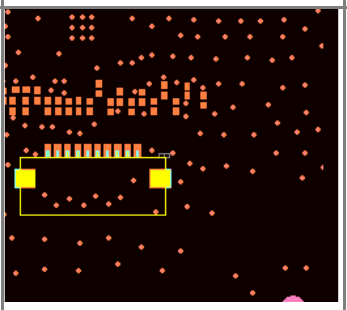
2) DF13A- 3P-1.25H(21)

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P/N: DF13A-3P-1.25H(21) CAD Shape: CON_BAT_3p_DF13A-3P-1.25H_HIROSE/53261-0	CON_BAT_3p_DF13A-3P-1.25H_HIROSE/53261-0 2.217 -91.388				Location J2 from XY.txt is not match with the PCB Gerber.

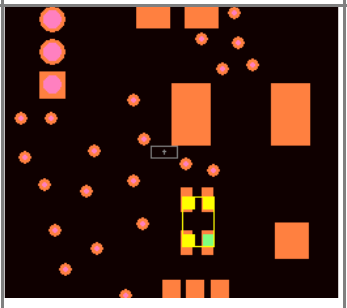
3) 686110148922

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P/N: 686110148922 CAD Shape: CON-10pts-1mm-SMT	CON-10pts-1mm-SMT -0.840 -30.993				Location J8 from XY.txt is not match with the PCB Gerber.

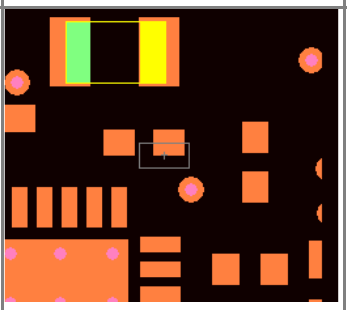
4) 744231091

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P/N: 744231091 CAD Shape: WE-CNSW_08058	WE-CNSW_08058 -22.471 -87.756				Location L4 from XY.txt is not match with the PCB Gerber.

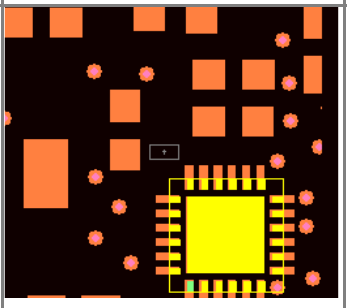
5) MLZ2012N150LT

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P/N: MLZ2012N150LT CAD Shape: 0805L_IPC	0805L_IPC -11.574 -42.188				Location L27 from XY.txt is not match with the PCB Gerber.

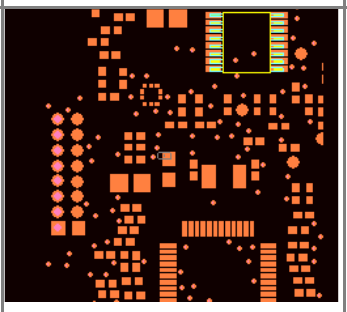
6) STUSB4500LQTR

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P/N: STUSB4500LQTR CAD Shape: STUSB4500LQTR-QFN-24	STUSB4500LQTR-QFN-24 -13.352 -85.520				Location U1 from XY.txt is not match with the PCB Gerber.

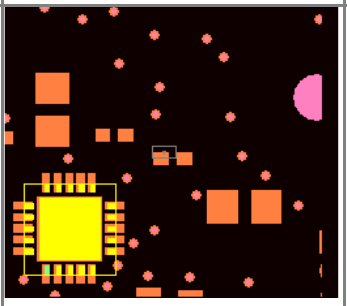
7) FT230XS-U

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P/N: FT230XS-U CAD Shape: FT230XS-U-S SOP16N	FT230XS-U-S SOP16N -18.508 -64.896				Location U2 from XY.txt is not match with the PCB Gerber.

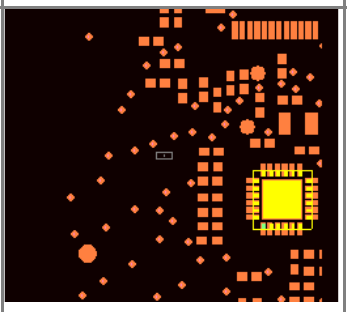
8) MCP73871-2CCI/ML

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P/N: MCP73871-2CCI/ML CAD Shape: MCP73871-Q FN20	MCP73871-Q FN20 3.106 -75.386				Location U3 from XY.txt is not match with the PCB Gerber.

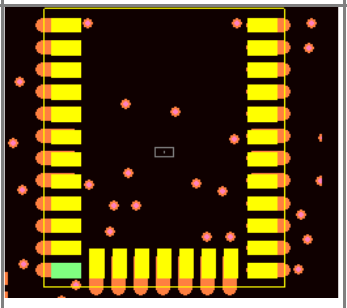
9) SX1262IMLTRT

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P/N: SX1262IMLTRT CAD Shape: QFN24_SX1262	QFN24_SX1262 -21.752 -42.133				Location U7 from XY.txt is not match with the PCB Gerber.

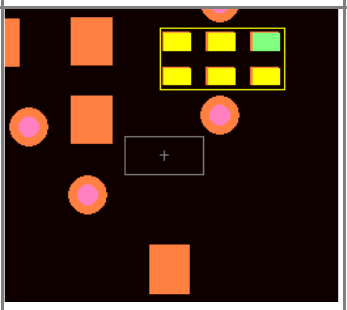
10) BGX220P22HNA21R

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P/N: BGX220P22HNA21R CAD Shape: BGX220P22HNA21R	BGX220P22HNA21R 4.376 -61.797				Location U6 from XY.txt is not match with the PCB Gerber.

11) HHM22152A2

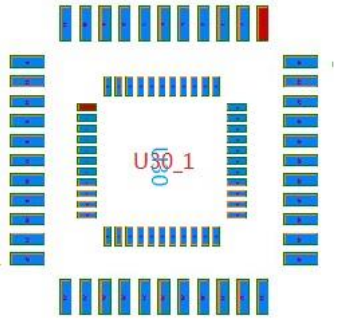
POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN

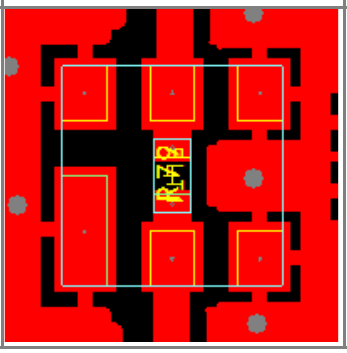
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P/N: HHM22152A2 CAD Shape: HHM22152A2	HHM22152A2 -11.990 -4.718				Location U9 from XY.txt is not match with the PCB Gerber.

General assembly and process requirements (Assembly)

1) SameCoordinator_SMD(InBOM)

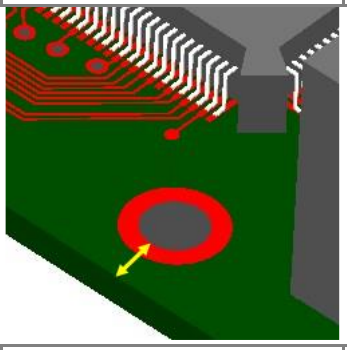
Check inbom SMD components with the same coordinator

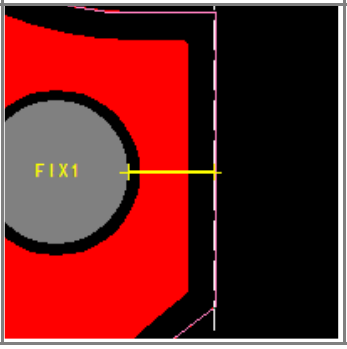
POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Different components with same coordinate, please confirm if it will cause interferes issue		

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Bottom Component FLT1 -15.384 -23.290	Bottom Component R79 -15.384 -23.290		0.000		FLT1 and R79 are designed at same location and both in the BOM, it might cause assembly issue.

2) ScrewHoleToProfile

Check distance of screw hole to PCB profile

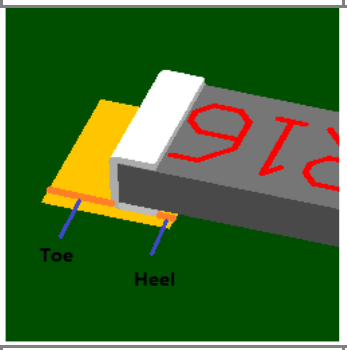
POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Too close distance may increase risk of PCB board damage during PCB fabrication.		

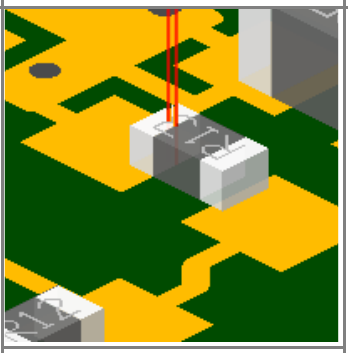
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P01x259PCB-B.TRN FIX1 16.033 -17.996	Profile LINE 17.660 -17.996	=>118mil(3mm)	1.627		

SMD land pattern standard(IPC7351)(Assembly)

1) Rectangular-End_CR_Heel(<0603)

Check solder joint heel size

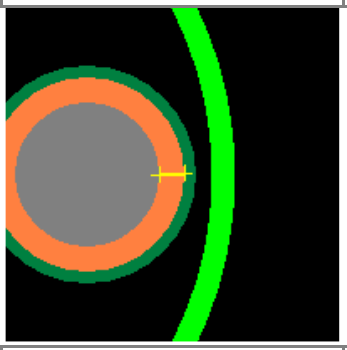
POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Cannot meet IPC-7351 Generic Requirements for Surface Mount Design and Land Pattern Standard		

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Top Component R13.2 -10.664 -82.828	Top Pads -10.579 -82.828	0mm	-0.084		Resistor-0402 from BOM, but land pattern is 0603. This issue exist for all locations with same part number. PartNumber:xx DESC:Resistor - CMS Type 0402 - Reflow Shape:0603R_HR MPN:xx MFG:OPEN

Outer signal layer analysis(Fabrication)

1) PadAnnularRing

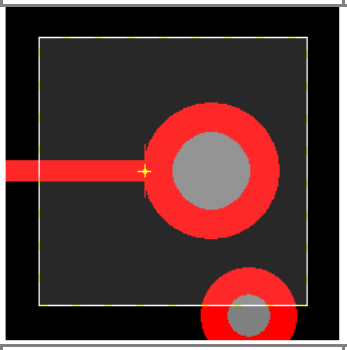
Check annular ring's width of VIA pad

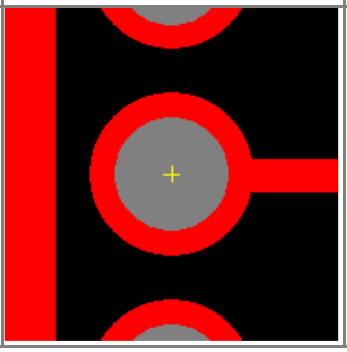
POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Cannot meet PCB fabricator's process capability		

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	BOT HHM22152A2 -11.990 -4.718	HHM22152A2 -11.990 -4.718				Location U9 from XY.txt is not match with the PCB Gerber.

2) NoTeardrop(PTH)

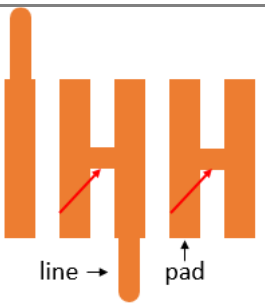
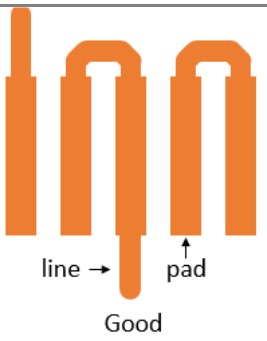
Check PTH without teardrop

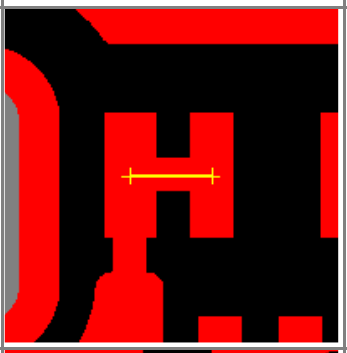
POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Teardrop design can enhance the bond strength of the pad and reduce the scrap caused by rework		

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P01x259PCB-B.CU2 J6 -27.181 -62.708	P01x259PCB-B.CU2 -27.181 -62.708		0.276		All the PTH pads are designed without teardrop, please confirm.

3) ICPadDirectConnection

Check pads of SMD component connected directly

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Pad design may cause issues for optical inspection (AOI). Please check Recommend design to the right.		

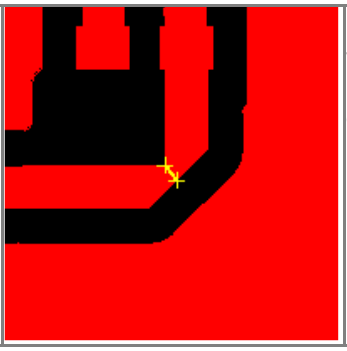
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P01x259PCB-B.CU1 J1 -11.361 -101.231	P01x259PCB-B.CU1 -11.861 -101.231				
2	P01x259PCB-B.CU1 J1 -5.261 -101.231	P01x259PCB-B.CU1 -4.761 -101.231				

4) CircuitBottleneck

Check circuit bottleneck width

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Cannot meet PCB fabricator's process capability		

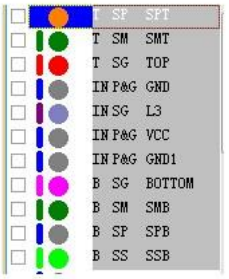
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment

1	P01x259PCB-B.CU2 -13.698 -1.206	P01x259PCB-B.CU2 -13.550 -1.032		0.228mm		Irregular trace design. Trace is wide at both ends but narrow in the middle. May affects electrical performance.

Silkscreen layer analysis(Fabrication)

1) MissingTopSilkscreenLayer

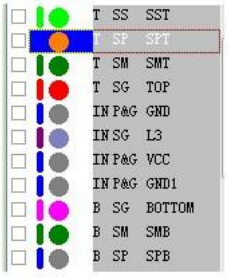
Check PCB missing top silkscreen layer

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Report missing top silkscreen layer		

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Missing Layer 0.000 0.000	0.000 0.000				Please ignore if it is a silk less design.

2) MissingBotSilkscreenLayer

Check PCB missing bottom silkscreen layer

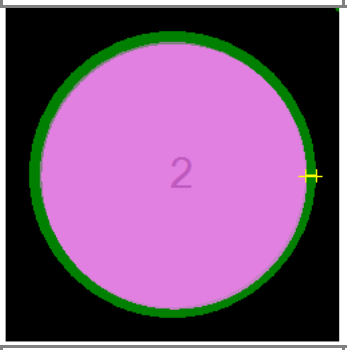
POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Report missing bottom silkscreen layer		

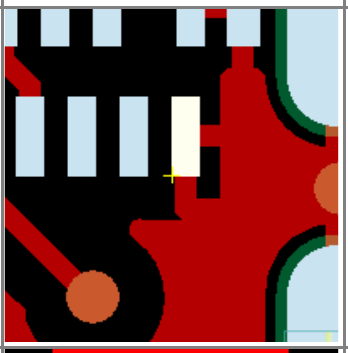
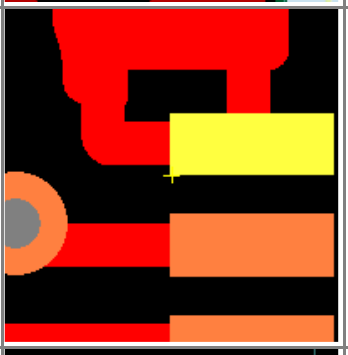
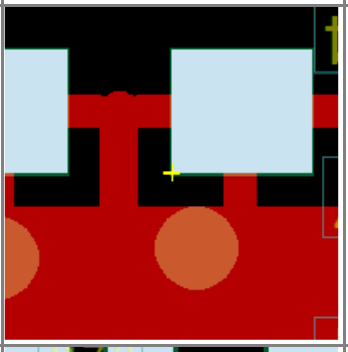
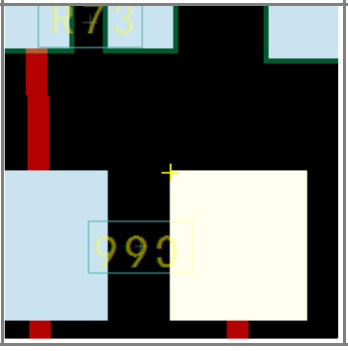
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Missing Layer 0.000 0.000	0.000 0.000				Please ignore if it is a silk less design.

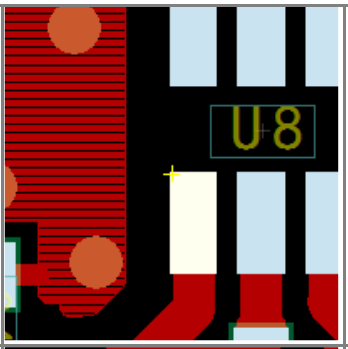
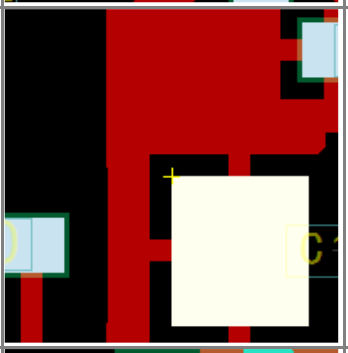
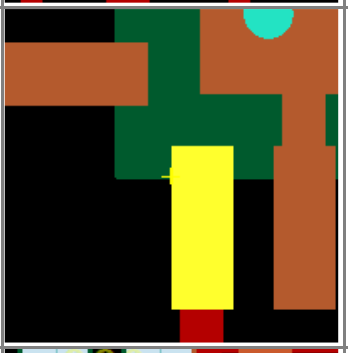
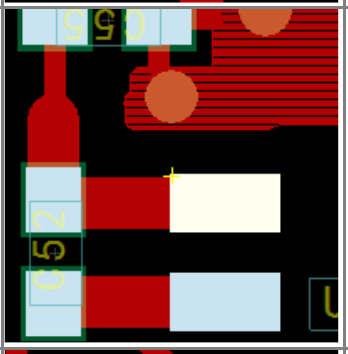
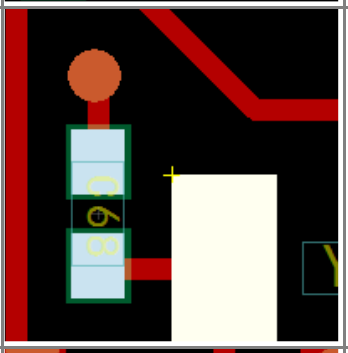
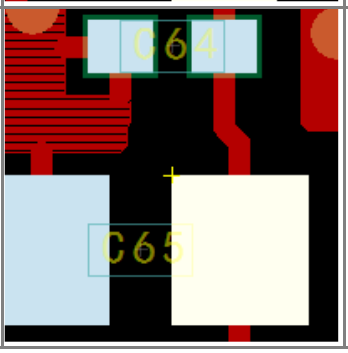
Solder mask layer analysis(Fabrication)

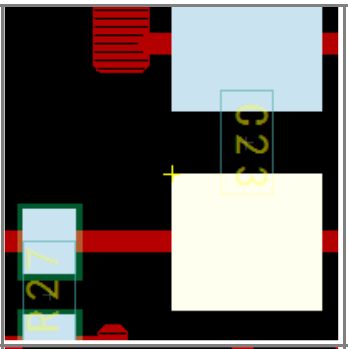
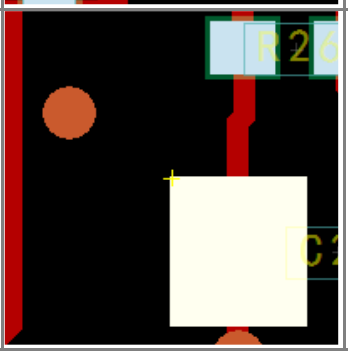
1) PadAnnularRing(SMD)

Check annular ring's width of SMD pad

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Annular ring cannot meet specification.		

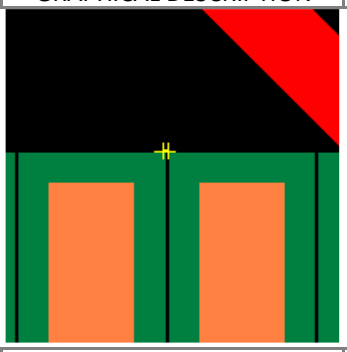
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Top Pads J1.14 -5.438 -102.862	P01x259PCB-B.VE1 J1.14 -5.438 -102.862	=>3mil(0.075 mm)	0.000		Solder mask open area=pad area. May cause issues in PCB fabrication process.
2	Top Pads U3.17 -3.386 -77.908	P01x259PCB-B.VE1 U3.17 -3.386 -77.908	=>3mil(0.075 mm)	0.000		Solder mask open area=pad area. May cause issues in PCB fabrication process.
3	Bottom Pads Y1.4 -7.131 -40.477	P01x259PCB-B.VE2 Y1.4 -7.131 -40.462	=>3mil(0.075 mm)	0.015		Solder mask open area=pad area. May cause issues in PCB fabrication process.
4	Bottom Pads C66.1 -2.135 -71.424	P01x259PCB-B.VE2 C66.1 -2.135 -71.424	=>3mil(0.075 mm)	0.000		Solder mask open area=pad area. May cause issues in PCB fabrication process.

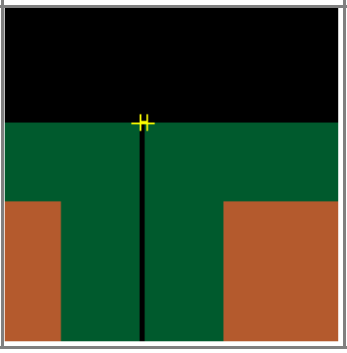
5	Bottom Pads U8.1 -14.323 -14.083	P01x259PCB-B.VE2 U8.1 -14.323 -14.083	=>3mil(0.075 mm)	0.000		Solder mask open area=pad area. May cause issues in PCB fabrication process.
6	Bottom Pads C12.1 -19.928 -76.808	P01x259PCB-B.VE2 C12.1 -19.928 -76.808	=>3mil(0.075 mm)	0.000		Solder mask open area=pad area. May cause issues in PCB fabrication process.
7	Bottom Pads U7.1 -15.102 -37.378	P01x259PCB-B.VE2 U7.1 -15.102 -37.378	=>3mil(0.075 mm)	0.000		Solder mask open area=pad area. May cause issues in PCB fabrication process.
8	Bottom Pads U10.1 -3.671 -7.007	P01x259PCB-B.VE2 U10.1 -3.671 -7.007	=>3mil(0.075 mm)	0.000		Solder mask open area=pad area. May cause issues in PCB fabrication process.
9	Bottom Pads Y2.1 -15.369 -64.030	P01x259PCB-B.VE2 Y2.1 -15.369 -64.030	=>3mil(0.075 mm)	0.000		Solder mask open area=pad area. May cause issues in PCB fabrication process.
10	Bottom Pads C65.2 -20.949 -63.270	P01x259PCB-B.VE2 C65.2 -20.949 -63.270	=>3mil(0.075 mm)	0.000		Solder mask open area=pad area. May cause issues in PCB fabrication process.

11	Bottom Pads C23.2 3.345 -47.478	P01x259PCB-B.VE2 C23.2 3.345 -47.478	=>3mil(0.075 mm)	0.000		Solder mask open area=pad area. May cause issues in PCB fabrication process.
12	Bottom Pads C22.1 -0.758 -53.745	P01x259PCB-B.VE2 C22.1 -0.758 -53.745	=>3mil(0.075 mm)	0.000		Solder mask open area=pad area. May cause issues in PCB fabrication process.

2) SMToSM

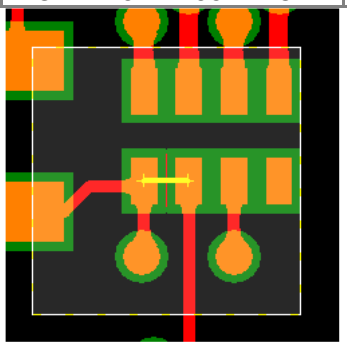
Check distance between solder mask pads

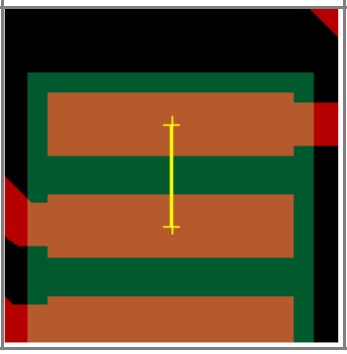
POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Cannot meet PCB fabricator's process capability		

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P01x259PCB-B.VE1 D3.2 -20.730 -95.309	P01x259PCB-B.VE1 D3.3 -20.720 -95.309	=>3mil(0.075 mm)	0.010		Distance too small between solder mask opening.

3) NoSMBridge

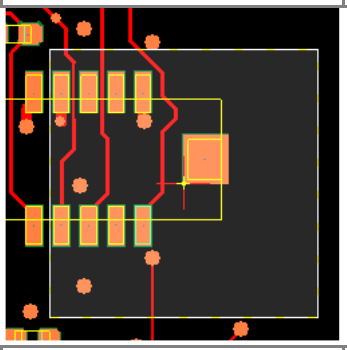
Check no solder mask bridge between exposed copper

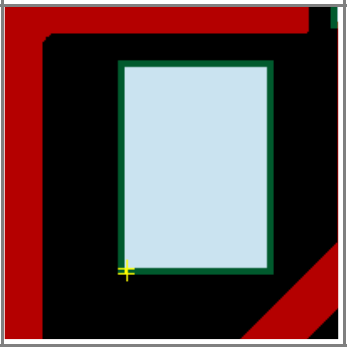
POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
No solder mask bridge will cause short issue. For fine pitch pads it is recommended to use SMD (solder mask definedpad) pad.		

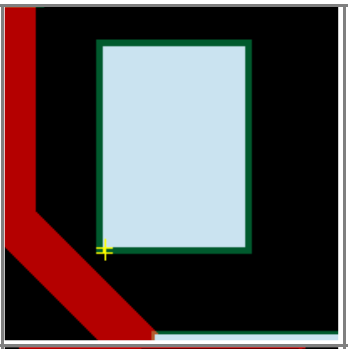
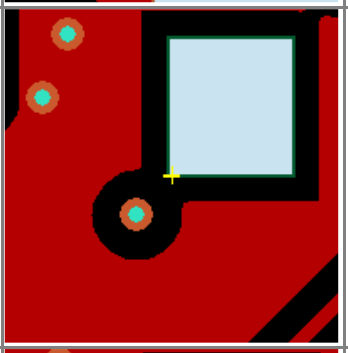
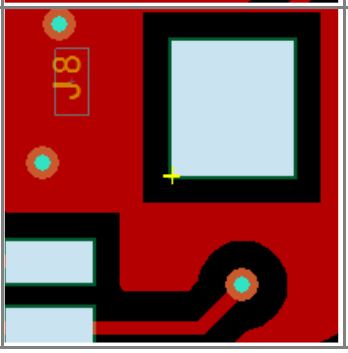
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P01x259PCB-B.VE2 U11.25 -18.188 -56.986	P01x259PCB-B.CU2 -18.188 -57.486				

4) NonSolderMaskDefinedPad

Check non solder mask defined pad for the mechanical pads

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Recommended to use SMD (Solder Mask Defined) pad type for mechanical PCB pads to increase mechanical withstand.		

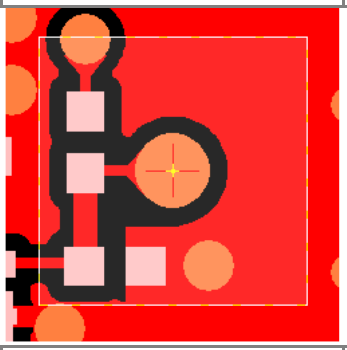
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P01x259PCB-B.VE1 J2.5 -3.031 -94.187	Top Pads J2.5 -3.031 -94.087				Recommend solder mask opening be smaller than pad to increase mechanical adhesion.

2	P01x259PCB-B.VE1 J2.4 4.368 -94.187	Top Pads J2.4 4.368 -94.087				Recommend solder mask opening be smaller than pad to increase mechanical adhesion.
3	P01x259PCB-B.VE1 J8.11 0.660 -46.042	Top Pads J8.11 0.660 -45.992				Recommend solder mask opening be smaller than pad to increase mechanical adhesion.
4	P01x259PCB-B.VE1 J8.12 0.660 -32.442	Top Pads J8.12 0.660 -32.392				Recommend solder mask opening be smaller than pad to increase mechanical adhesion.

TP analysis(Test Point)

1) TPMissingSP

Check test point missing solder paste pad

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
TP missing solder paste will cause TP is oxidation		

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	P01x259PCB-B.CU2 PTST12 -3.098 -74.980	P01x259PCB-B.BR2 -3.098 -74.980				Missing solder paste pad for all test point pads. Please ignore if it's not OSP PCB.

Appendix1: BOM Verification Report

BOM Verification Report

BOM & CAD Verification Report

Unloaded parts (in CAD not in BOM)

FIX0	FIX1	FIX2	FIX3	FIX4	FIX5	J12	J13
J14	J15	J16	J3	J4	J5	J6	J7
LOC0	LOC1	M2	M3	PTST1	PTST10	PTST12	PTST13
PTST14	PTST15	PTST16	PTST17	PTST19	PTST2	PTST20	PTST21
PTST23	PTST24	PTST27	PTST28	PTST29	PTST3	PTST31	PTST33
PTST34	PTST4	PTST5	PTST6	PTST9	PTSTHF1		

Appendix2: No Library List

Note: The device in the following list does not use the real component library. This analysis is the CAD package used.

No Library List

#	PN	Description	MPN	Manufacture
1	CL0331-0472-2-10	Embase Male CMS U.FL-R-SMT	CL0331-0472-2-	HIROSE

No Library RefDes List

#	RefDes	PN	Package
1	J10	CL0331-0472-2-10	Embase_coax_Murata