DFM Report

Product Name:	Tucson
Prepared By:	Benjamin Vo
Report Rev:	V15
Date:	2020-5-16
Page:	33

Table of Contents

Summary	1
PCB Image Overview	2
PCBA Image Overview(3D)	3
Safe distance analysis by PKG_TYPE(Assembly)	4
Fiducial mark analysis (Assembly)	6
General assembly and process requirements (Assembly)	7
SMD land pattern standard (IPC7351) (Assembly)	16
Drill analysis (Fabrication)	23
Profile analysis (Fabrication)	25
Outer signal layer analysis (Fabrication)	26
Silkscreen layer analysis (Fabrication)	28
Solder mask layer analysis (Fabrication)	29
Components Information Summary	31
BOM Verification Report	32
No Library List	33

DFM Report

Generated by VayoPro-DFM Expert, Format v3.0

Basic Information

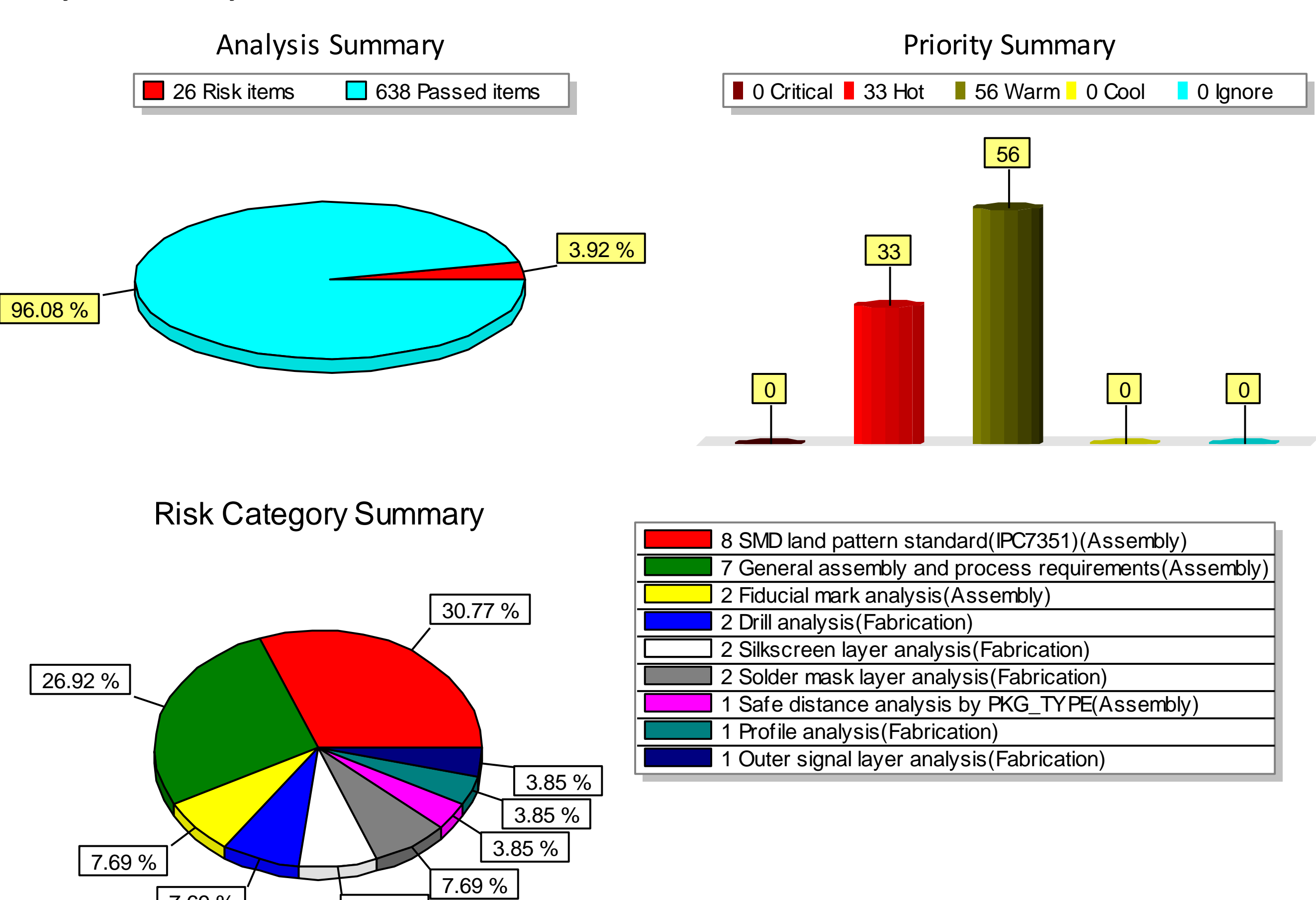
Customer Name	SOLECTRON
Product Name	Tucson
Assembly P/N & Rev	BT3217
PCB P/N & Rev	A9876

Basic Product Information

CAD Data Source	Gerber + Centroid XY
CAD File	BT3217.txt
BOM File	BT3217.xlsx
Matched Component (PN)	91.11%
Matched Component (RefDes)	95.06%
Size L x W(mm)	78.672 x 52.456
Thickness (mm)	1.600
Number of layers	2
Single or double reflow	Double Side
Through hole parts on 1 or both sides	None

Assembly Process Flow

Analysis Summary



PCB Overview

Overview of Top Image

Overview of Bottom Image

PCBA Overview(3D)

Overview of Top Image

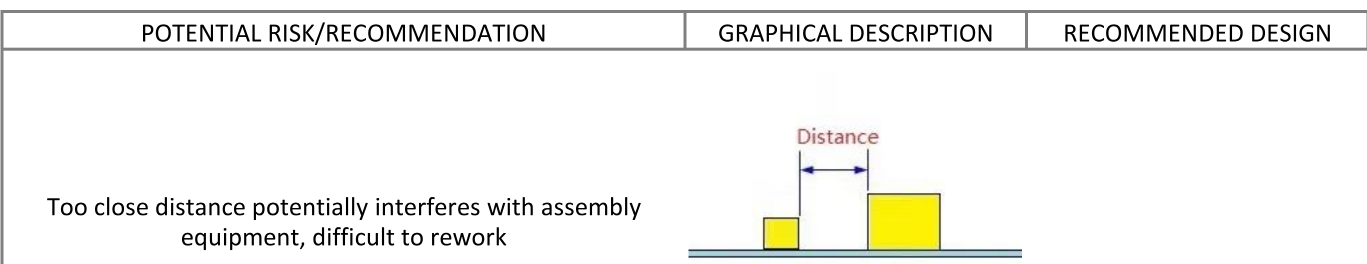
Overview of Bottom Image

Report Items

Safe distance analysis by PKG_TYPE(Assembly)

1) BodyToBody(SMD)

Check distance of component body to body (SMD)

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Too close distance potentially interferes with assembly equipment, difficult to rework		

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Top Component 2007 27.050 22.000	Top Component 2008 27.050 22.000	=>14mil(0.35 mm)	0.000		The position 2007 & 2008 are in same location, one of them may be a wrong location, please check in the data "3112 293 39383_A_125_08-Jan-2020.txt ". Notice: both position 2007 & 2008 are in the BOM.
2	Top Component 2019 55.750 30.600	Top Component 7705 55.750 29.800	=>14mil(0.35 mm)	0.000		The position 2019 & 7705 are too close, 2019 is a wrong position because there is no footprint for the part. Notice: both position 2019 & 7705 are in the BOM.
3	Top Component 6004 54.585 17.165	Top Component 7707 54.550 17.165	=>14mil(0.35 mm)	0.035		The position 7707 & 6004 are too close, it will cause assembly issue. The root cause may caused by the 6004 and it is not match with the land pattern, the part size has a little big.

Fiducial mark analysis (Assembly)

1) NoFidPad(Bottom)

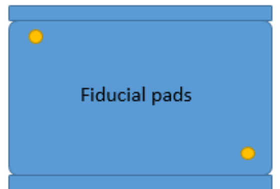
No fiducial mark on bottom side

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
A minimum of two global fiducial marks is required for correction of offsets (x and y position) and rotational offset (theta position).These should be located diagonally opposite and as far apart as possible on the circuit or panel.		

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Bottom Pads 0.000 0.000	Bottom Pads 0.000 0.000		0.000		There are no standard fiducial marks on bottom side. It is recommended to design two marks on the diagonal of the PCB.

2) NoFidPad (Top)

Top side no fiducial mark

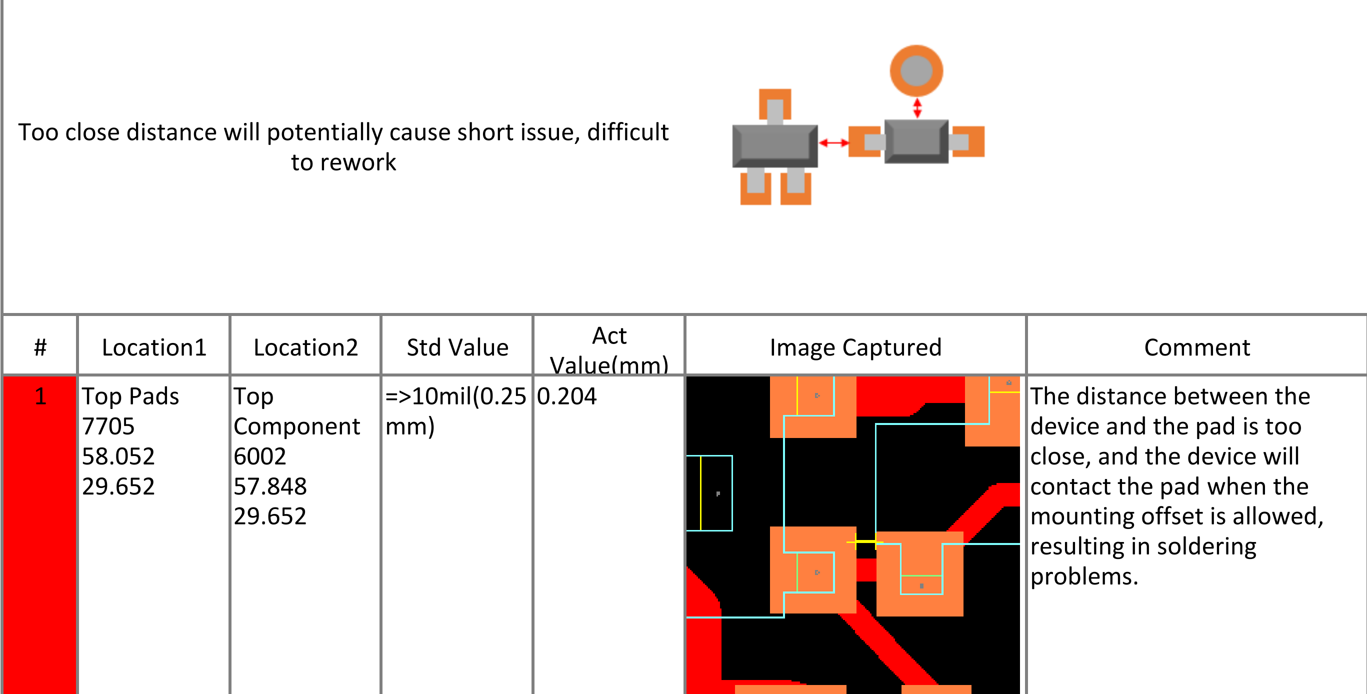
POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
A minimum of two global fiducial marks is required for correction of offsets (x and y position) and rotational offset(theta position).These should be located diagonally opposite and as far apart as possible on the circuit or panel.		

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Top Pads 0.000 0.000	Top Pads 0.000 0.000		0.000		There are no standard fiducial marks on top side. It is recommended to design two marks on the diagonal of the PCB.

General assembly and process requirements (Assembly)

1) BodyToFootprint (SMD)

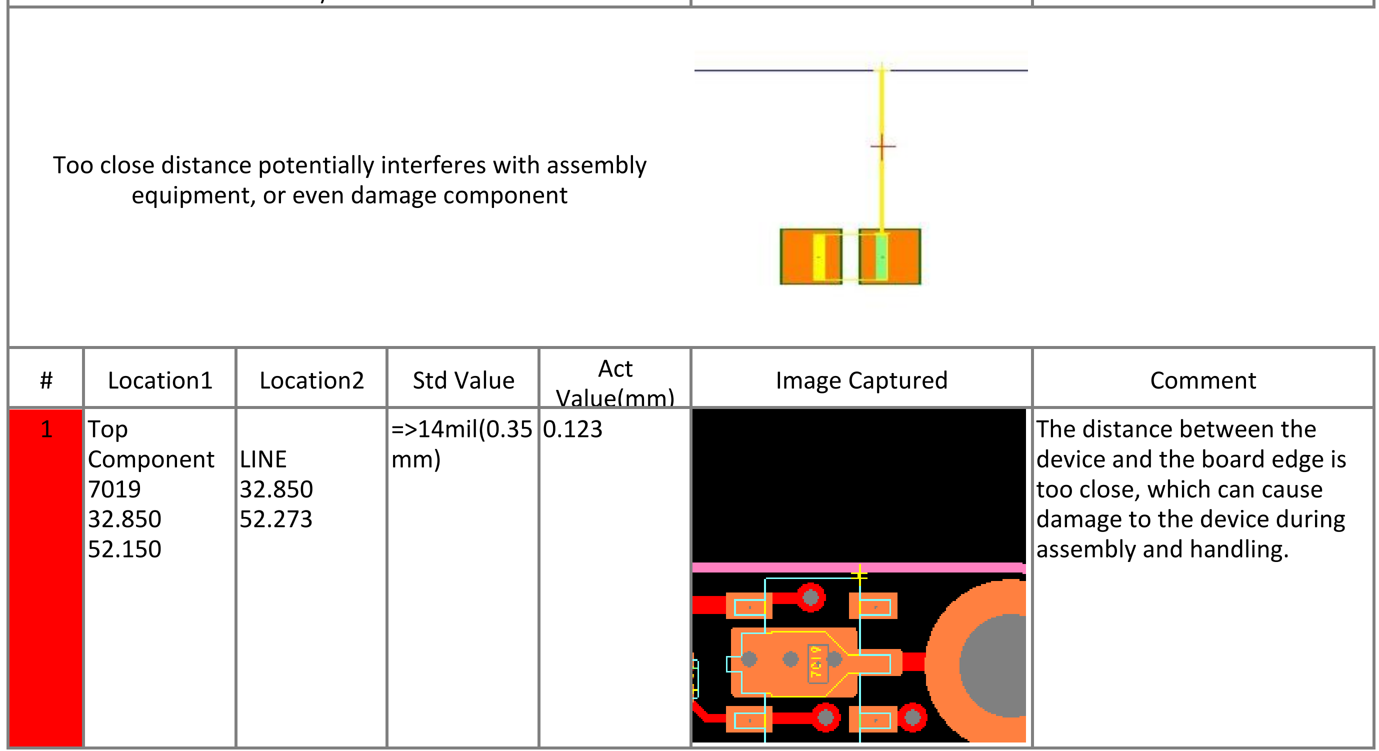
Check distance of component to other footprints



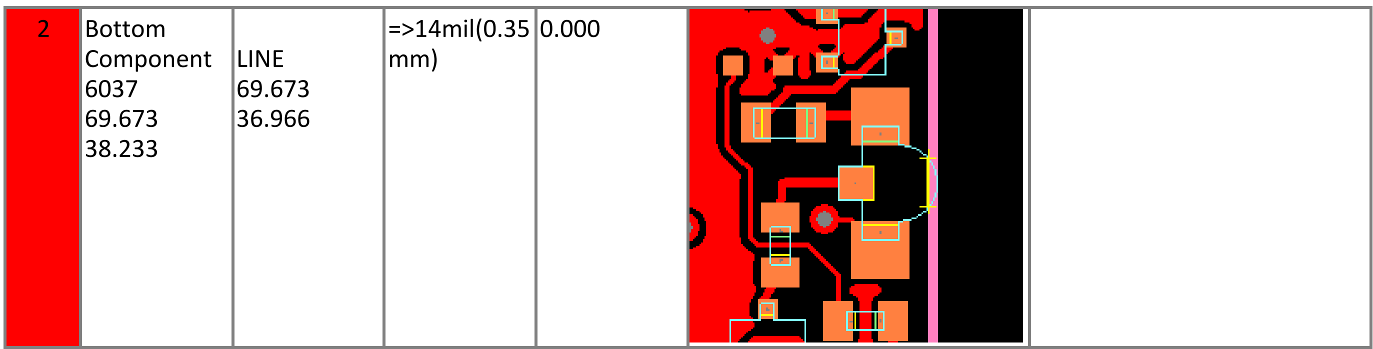
POTENTIAL RISK/RECOMMENDATION					GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Too close distance will potentially cause short issue, difficult to rework						
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Top Pads 7705 58.052 29.652	Top Component 6002 57.848 29.652	=>10mil(0.25 mm)	0.204		The distance between the device and the pad is too close, and the device will contact the pad when the mounting offset is allowed, resulting in soldering problems.

2) BodyToProfile (SMD)

Check distance of SMD component body to PCB profile



POTENTIAL RISK/RECOMMENDATION					GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Too close distance potentially interferes with assembly equipment, or even damage component						
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Top Component 7019 32.850 52.150	LINE 32.850 52.273	=>14mil(0.35 mm)	0.123		The distance between the device and the board edge is too close, which can cause damage to the device during assembly and handling.



#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
2	Bottom Component 6037 69.673 38.233	LINE 69.673 36.966	=>14mil(0.35 mm)	0.000		

3) SMDCompToNPTH

Check distance of SMD component to NPTH

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Too close distance potentially interferes with assembly, damage component in assembly process		

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Top Component 6006 56.350 25.682	Xdrl001.drl PAD 56.350 26.118	=>40mil(1mm)	0.000		6006 device touch to the NPTH hole. This NPTH is 1003's mounting hole. The component and the NPTH hole are too close, which will cause assembly problems.
2	Top Component 5001 26.218 29.261	Xdrl001.drl PAD 26.075 28.615	=>40mil(1mm)	0.662		5001 device close to the NPTH hole. The NPTH is usually the purpose of the device mounting hole or the tooling hole. The component and the NPTH hole are too close, which will cause assembly problems.

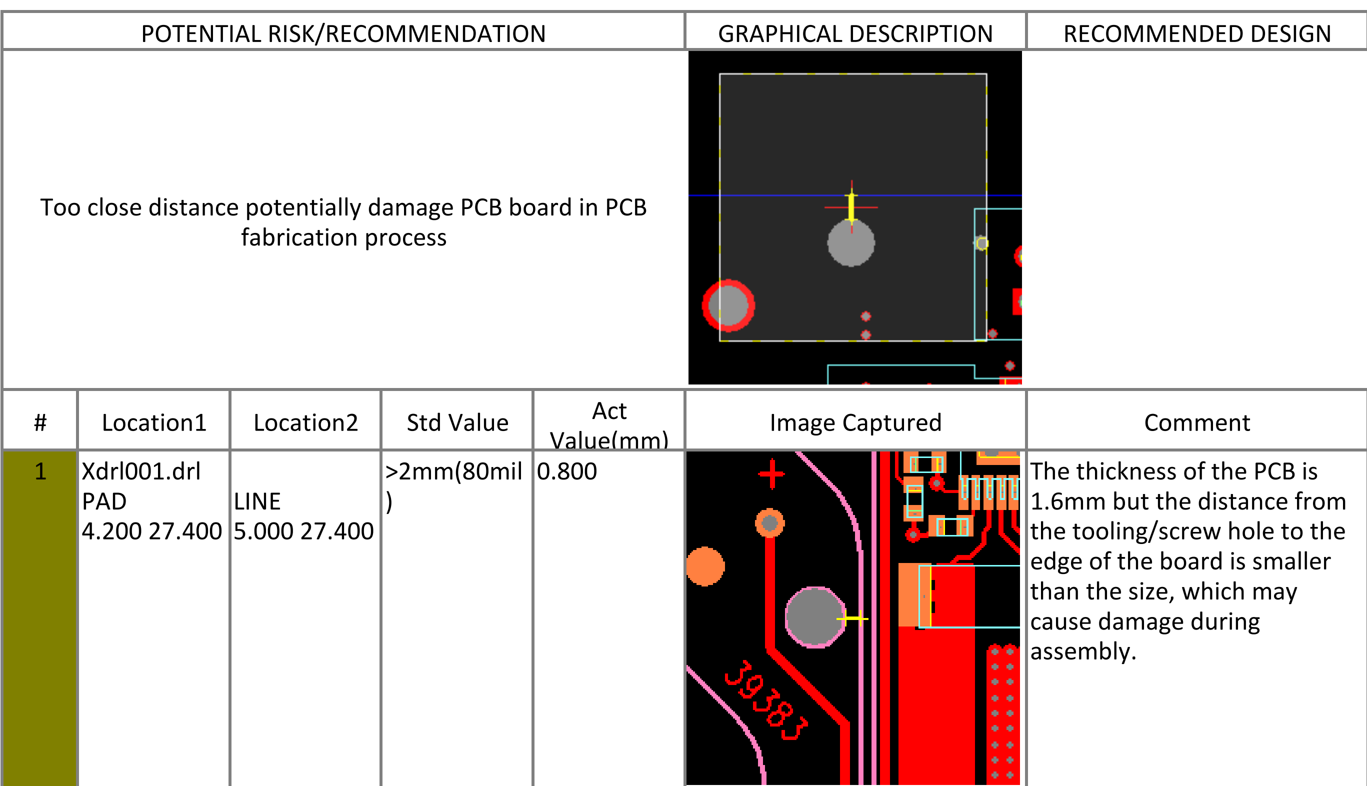
4) ChipToNPTH

Check distance of chip component to NPTH

POTENTIAL RISK/RECOMMENDATION					GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Too close distance potentially interferes with assembly equipment, or even damage component for surface tension issue						
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Top Component 2018 62.400 25.900	Xdrl001.drl PAD 62.750 25.900	=>40mil(1mm)	0.350		The small chip component and the NPTH hole are too close to each other, which can cause offset in reflow process (especially hot air reflow soldering equipment).

5) HoleToProfile (BigHole)

Check distance of large hole to PCB profile

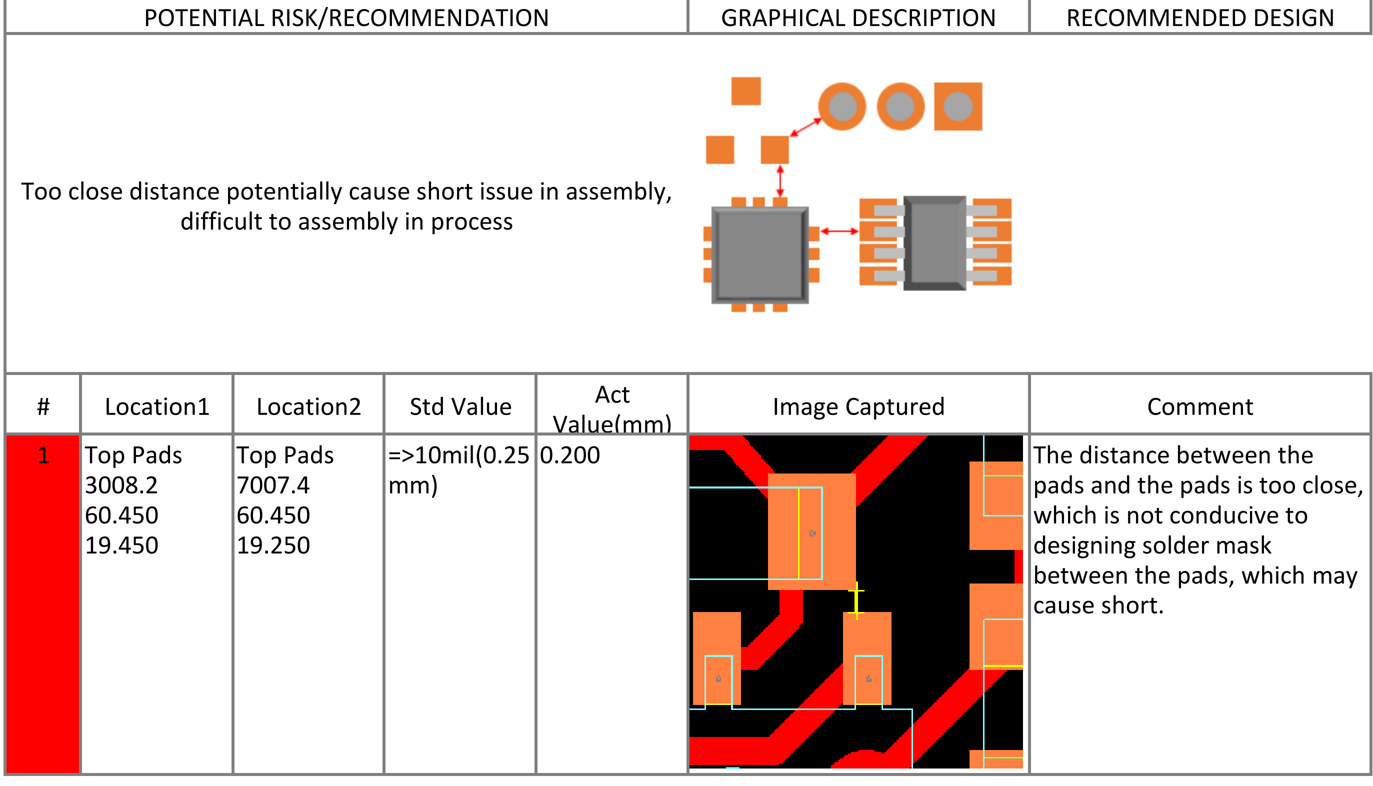


POTENTIAL RISK/RECOMMENDATION					GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Too close distance potentially damage PCB board in PCB fabrication process						
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Xdrl001.drl PAD 4.200 27.400	LINE 5.000 27.400	>2mm(80mil)	0.800		The thickness of the PCB is 1.6mm but the distance from the tooling/screw hole to the edge of the board is smaller than the size, which may cause damage during assembly.

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
2	Xdrl001.drl PAD 8.300 51.175	LINE 8.300 52.400	>2mm(80mil)	1.225		
3	Xdrl001.drl PAD 36.900 51.175	LINE 36.900 52.400	>2mm(80mil)	1.225		
4	Xdrl001.drl PAD 1.225 2.600	LINE 0.000 2.600	>2mm(80mil)	1.225		

6) FootprintToFootprint

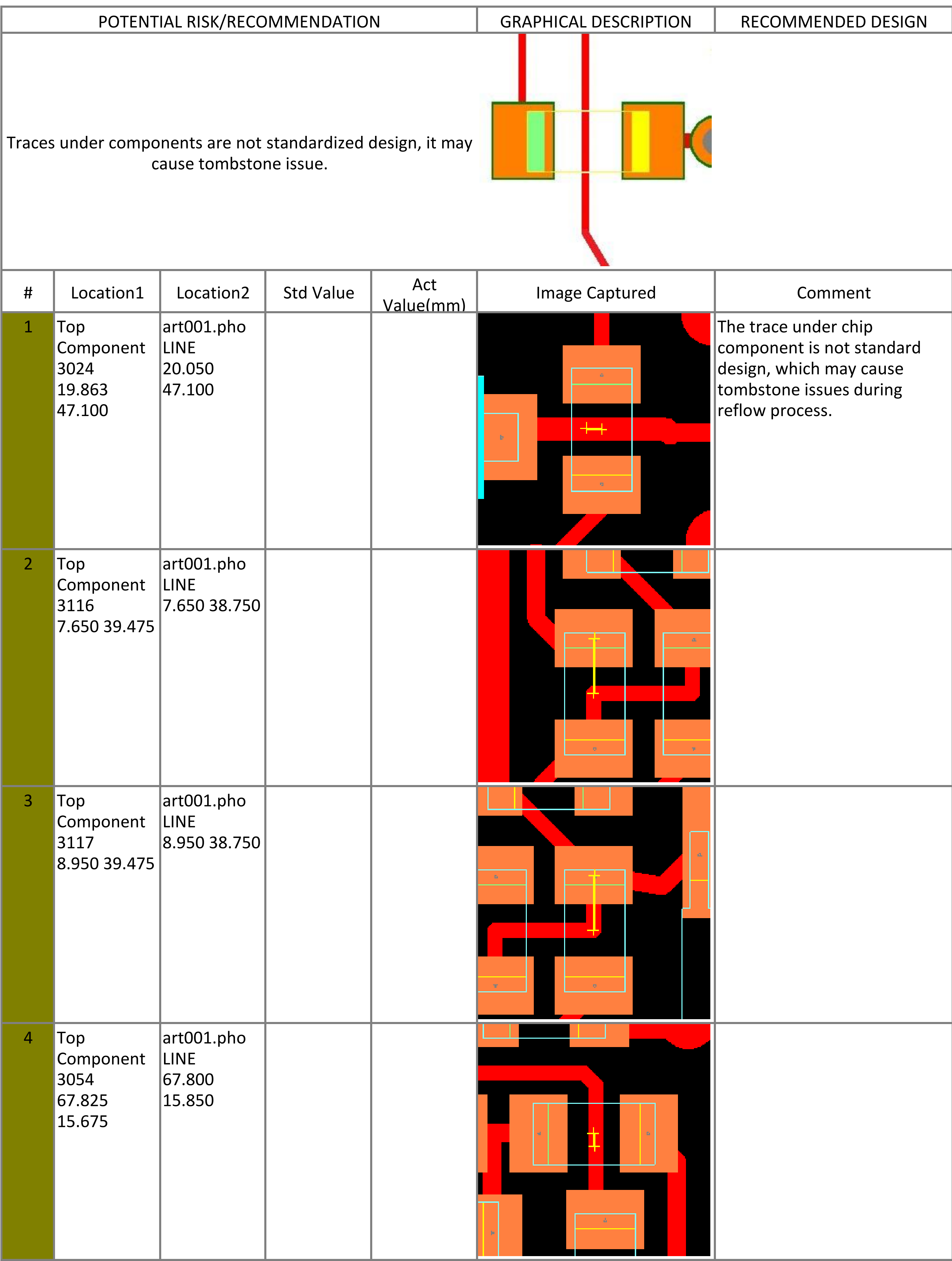
Check distance of footprint to footprint for different parts



POTENTIAL RISK/RECOMMENDATION					GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Too close distance potentially cause short issue in assembly, difficult to assembly in process						
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Top Pads 3008.2 60.450 19.450	Top Pads 7007.4 60.450 19.250	=>10mil(0.25 mm)	0.200		The distance between the pads and the pads is too close, which is not conducive to designing solder mask between the pads, which may cause short.

7) TraceUnderComp (Chip)

Trace under chip component



POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Traces under components are not standardized design, it may cause tombstone issue.		

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Top Component 3024 19.863 47.100	art001.pho LINE 20.050 47.100				The trace under chip component is not standard design, which may cause tombstone issues during reflow process.
2	Top Component 3116 7.650 39.475	art001.pho LINE 7.650 38.750				
3	Top Component 3117 8.950 39.475	art001.pho LINE 8.950 38.750				
4	Top Component 3054 67.825 15.675	art001.pho LINE 67.800 15.850				

SMD land pattern standard (IPC7351)(Assembly)

1) NoLibraryComp

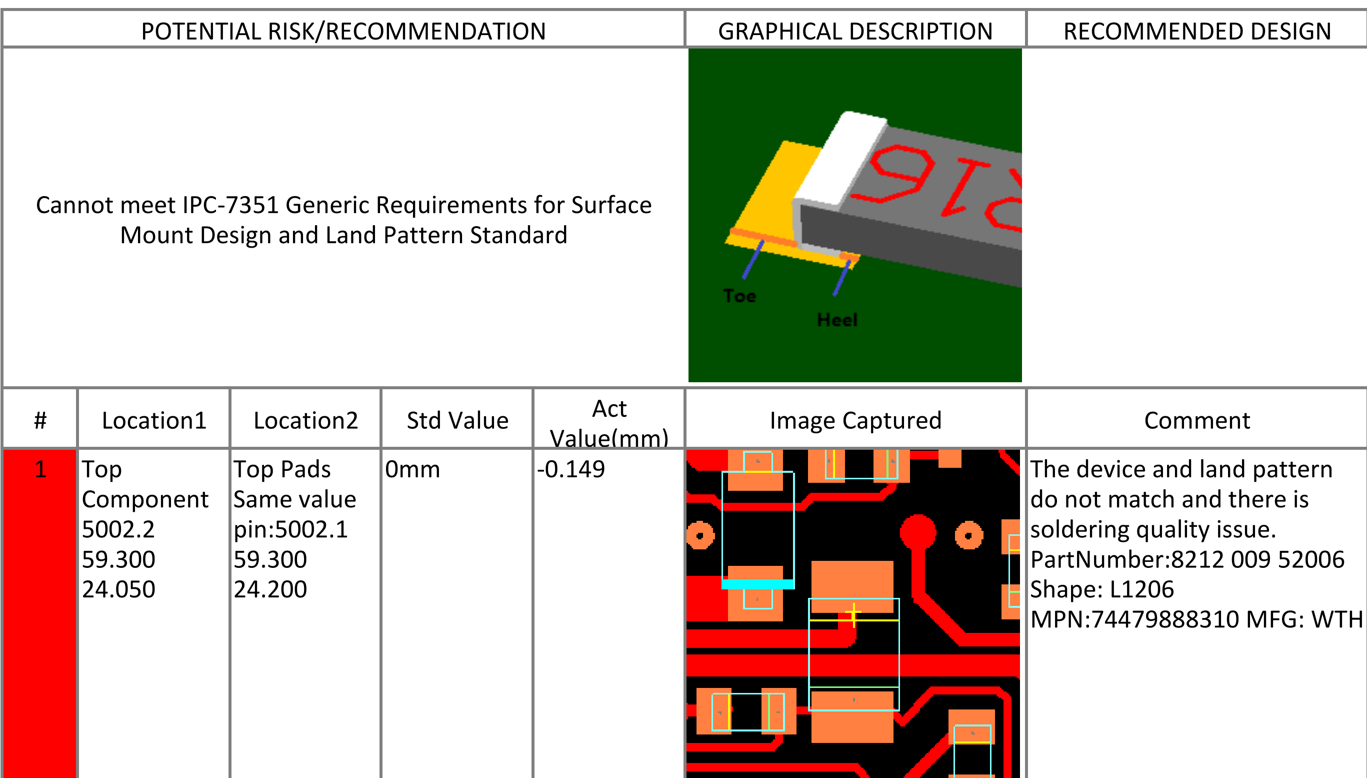
Report no component library part location

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
No available library for this part, it will cannot verify land pattern design		

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	TOP 2004 35.650 26.750	2004 35.650 26.750				We did not find the datasheets for these devices, so we did not check its assemblability, which requires human confirmation.
2	TOP 2011 34.000 36.450	2011 34.000 36.450				
3	TOP 3065 8.350 36.700	3065 8.350 36.700				
4	TOP 7002 17.000 49.900	7002 17.000 49.900				

2) Rectangular-End_IND_Heel (=>0603)

Check solder joint heel size

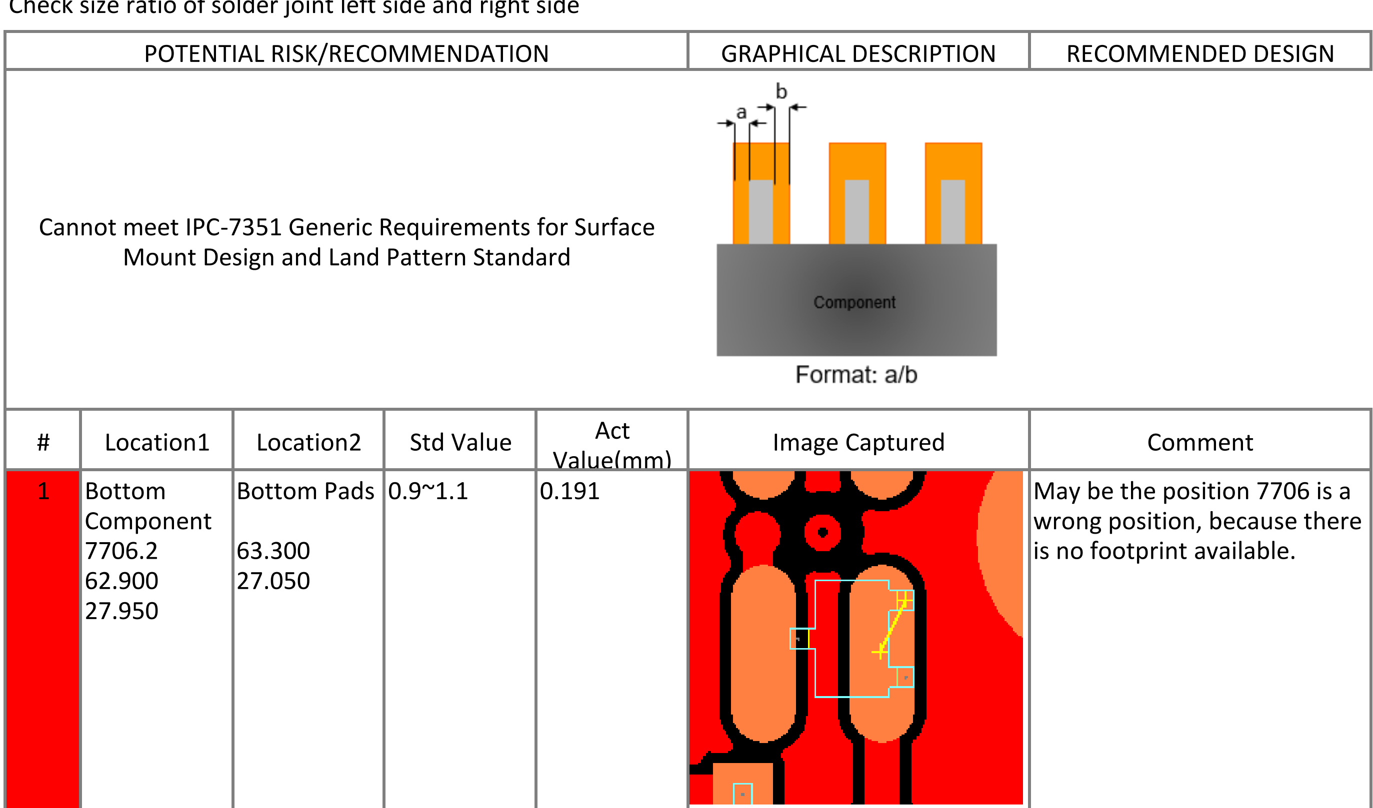


POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Cannot meet IPC-7351 Generic Requirements for Surface Mount Design and Land Pattern Standard		

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Top Component 5002.2 59.300 24.050	Top Pads Same value pin:5002.1 59.300 24.200	0mm	-0.149		The device and land pattern do not match and there is soldering quality issue. PartNumber:8212 009 52006 Shape: L1206 MPN:74479888310 MFG: WTH

3) Pin&Pad_Side_Ratio

Check size ratio of solder joint left side and right side



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	Bottom Component 7706.2 62.900 27.950	Bottom Pads 63.300 27.050	0.9~1.1	0.191		May be the position 7706 is a wrong position, because there is no footprint available.

4) Gull-Wing_SOT_Toe

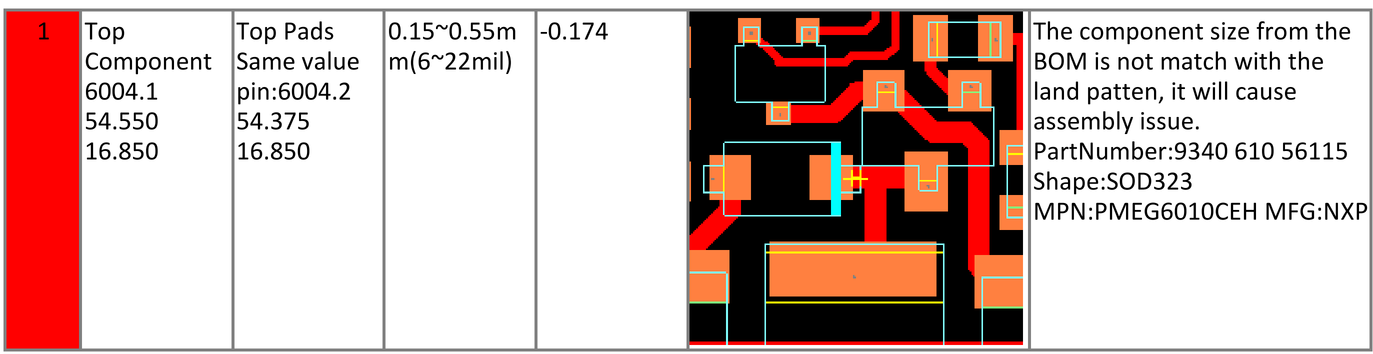
Check solder joint toe size

POTENTIAL RISK/RECOMMENDATION					GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Cannot meet IPC-7351 Generic Requirements for Surface Mount Design and Land Pattern Standard					Heel Toe	
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Top Component 7003.3 68.925 20.950	Top Pads 68.850 20.950	0.15~0.55mm(6~22mil)	-0.074		The device and the land pattern do not match, the device pins exceed the pad, and there is a soldering quality problem. PartNumber:8212 127 71798 Shape: SOT-89-5 MPN: XC6227C501PR-G MFG:TRX
2	Top Component 7019.3 29.475 49.900	Top Pads 29.500 49.900	0.15~0.55mm(6~22mil)	-0.024		PartNumber:8212 127 71798 Shape: SOT-89-5 MPN:XC6227C501PR-G MFG:TRX

5) Gull-Wing_SOD_Toe

Check solder joint toe size

POTENTIAL RISK/RECOMMENDATION					GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Cannot meet IPC-7351 Generic Requirements for Surface Mount Design and Land Pattern Standard					Heel Toe	
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Top Component 6004.1 54.550 16.850	Top Pads Same value pin:6004.2 54.375 16.850	0.15~0.55mm(6~22mil)	-0.174		The component size from the BOM is not match with the land patten, it will cause assembly issue. PartNumber:9340 610 56115 Shape:SOD323 MPN:PMEG6010CEH MFG:NXP

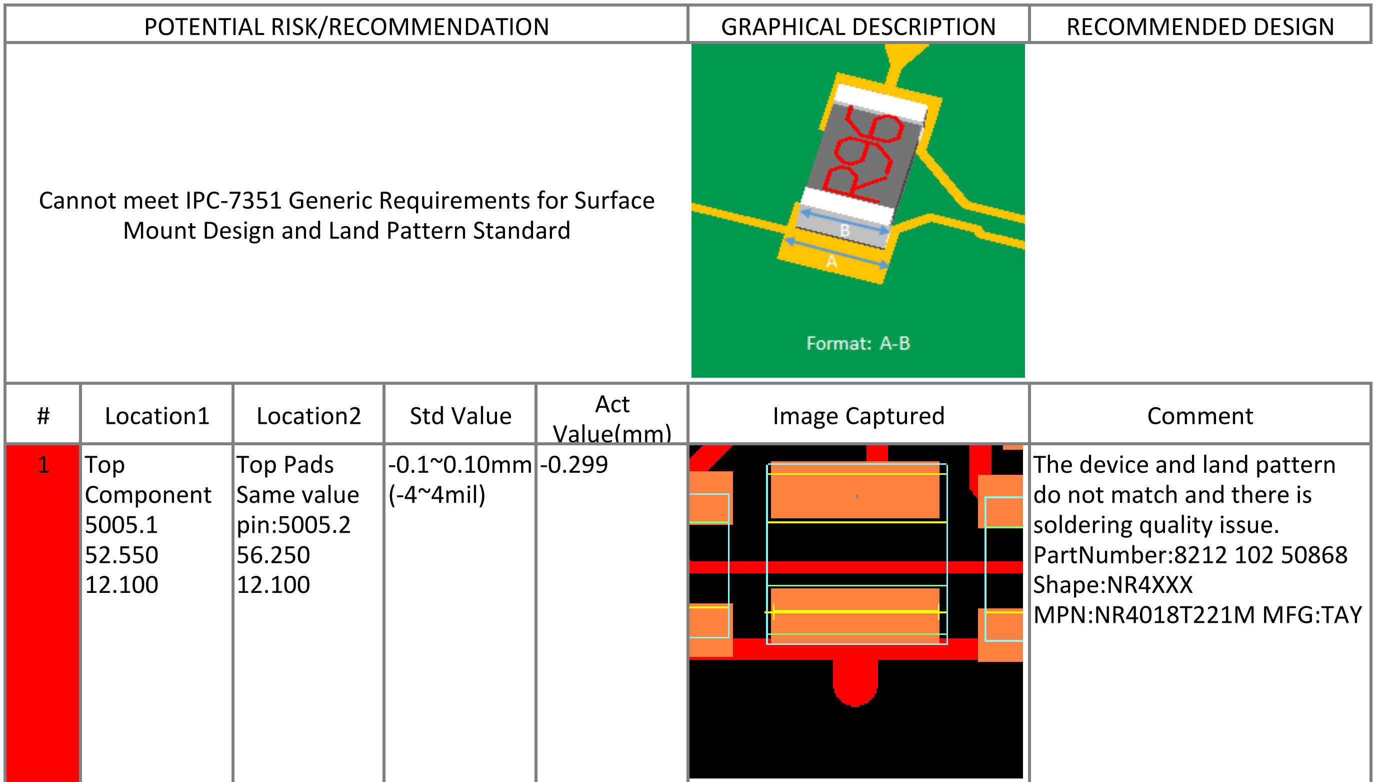
6) Gull-Wing_SOP_Side (P>0.625mm)

Check the size difference of pad width and pin width

POTENTIAL RISK/RECOMMENDATION					GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Cannot meet IPC-7351 Generic Requirements for Surface Mount Design and Land Pattern Standard					Format: A-B	
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Top Component 7024.1 15.091 39.738	Top Pads Same value pins:7024.2~7024.17 14.671 39.738	0.02~0.10mm(0.8~4mil)	0.180		The pads are too wide, which will reduce the electrical gap between the pads for fine-pitch devices. PartNumber:8212 323 71346 Shape: TSSOP-16EPG MPN: LM5088MHX-1/NOPB MFG: TI
2	Top Component 7013.1 47.775 26.662	Top Pads Same value pins:7013.2~7013.14 48.225 26.662	0.02~0.10mm(0.8~4mil)	0.210		PartNumber:8212 000 70202 Shape: TSSOP-14A MPN: LM2901PT MFG:ST

7) Rectangular-End_IND_Side (=>0603)

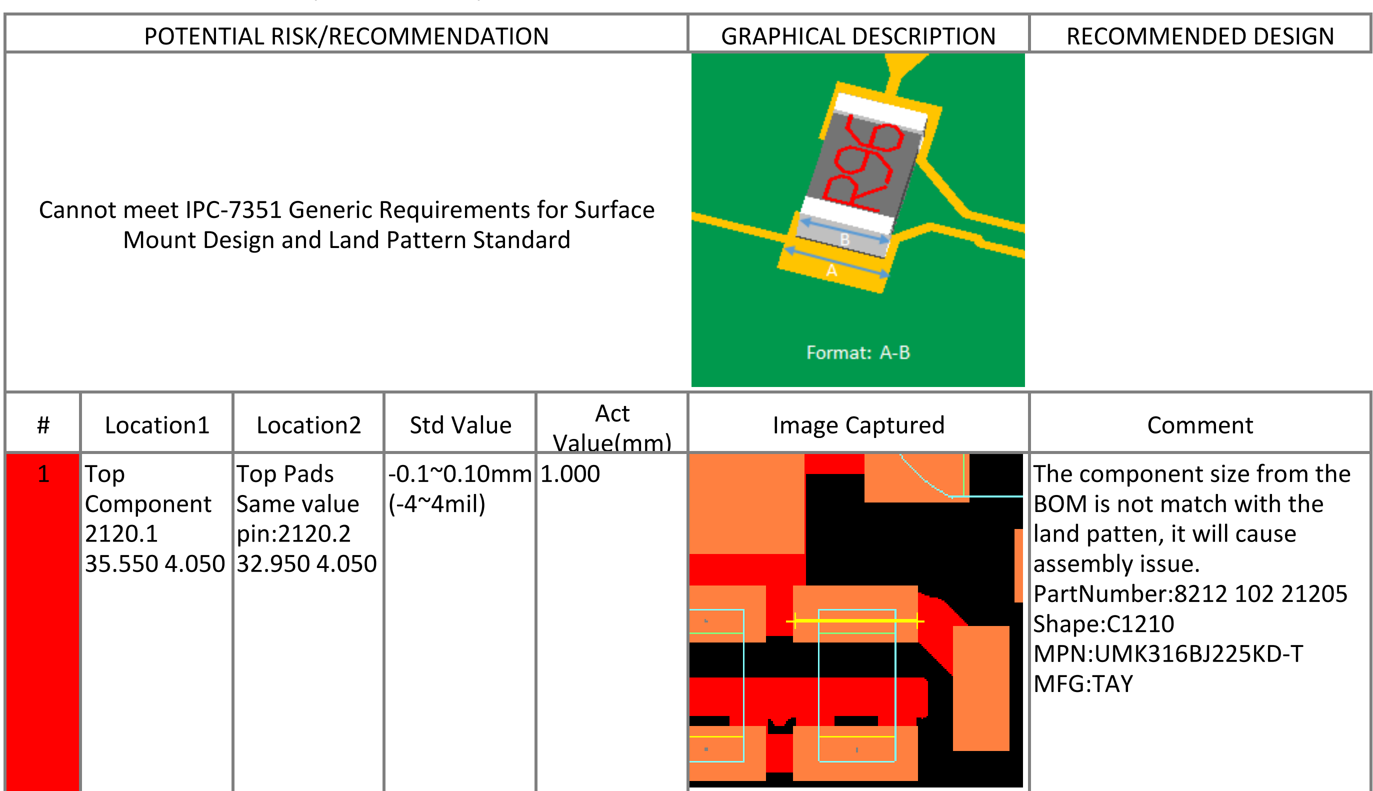
Check the size difference of pad width and pin width



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 5005.1 52.550 12.100	Top Pads Same value pin:5005.2 56.250 12.100	-0.1~0.10mm (-4~4mil)	-0.299		The device and land pattern do not match and there is soldering quality issue. PartNumber:8212 102 50868 Shape:NR4XXX MPN:NR4018T221M MFG:TAY

8) Rectangular-End_CC_Side(=>0603)

Check the size difference of pad width and pin width



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 2120.1 35.550 4.050	Top Pads Same value pin:2120.2 32.950 4.050	-0.1~0.10mm (-4~4mil)	1.000		The component size from the BOM is not match with the land patten, it will cause assembly issue. PartNumber:8212 102 21205 Shape:C1210 MPN:UMK316BJ225KD-T MFG:TAY

Drill analysis (Fabrication)

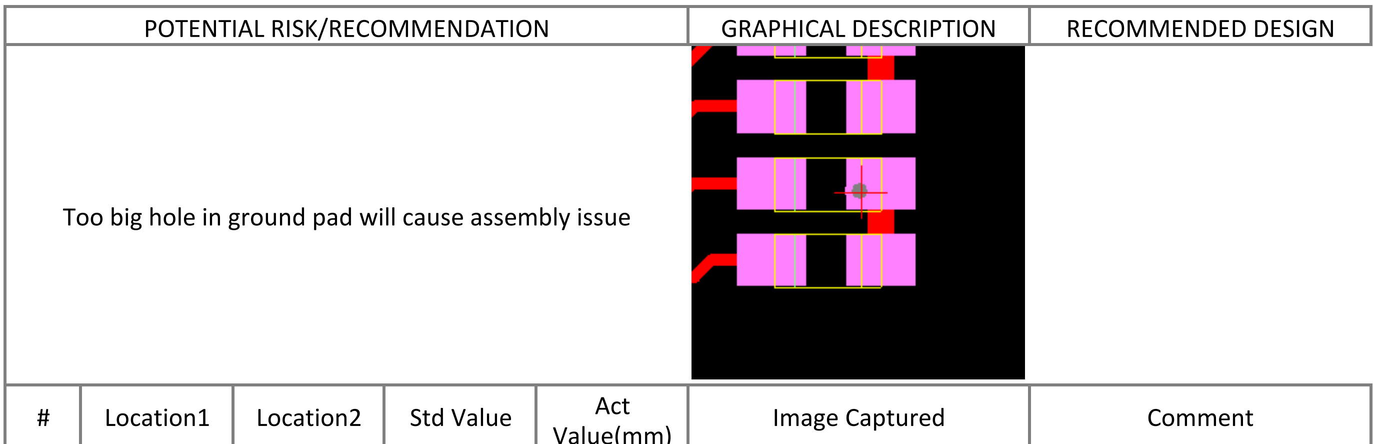
1) PadAnnularRing(NPTH)

NPTH has copper annular ring

POTENTIAL RISK/RECOMMENDATION					GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
NPTH with copper annular ring should be a design issue						
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Xdrl001.drl PAD 15.800 13.350	Top Pads PAD 14.900 13.350		0.900		the NPTH hole has the copper annular ring, it is not a standard design, it may be a wrong design, please check and confirm in the file Xdrl001.rep
2	Xdrl001.drl PAD 15.800 13.350	Bottom Pads PAD 14.900 13.350		0.900		

2) HoleInGroundPad

Check big via hole in ground pad



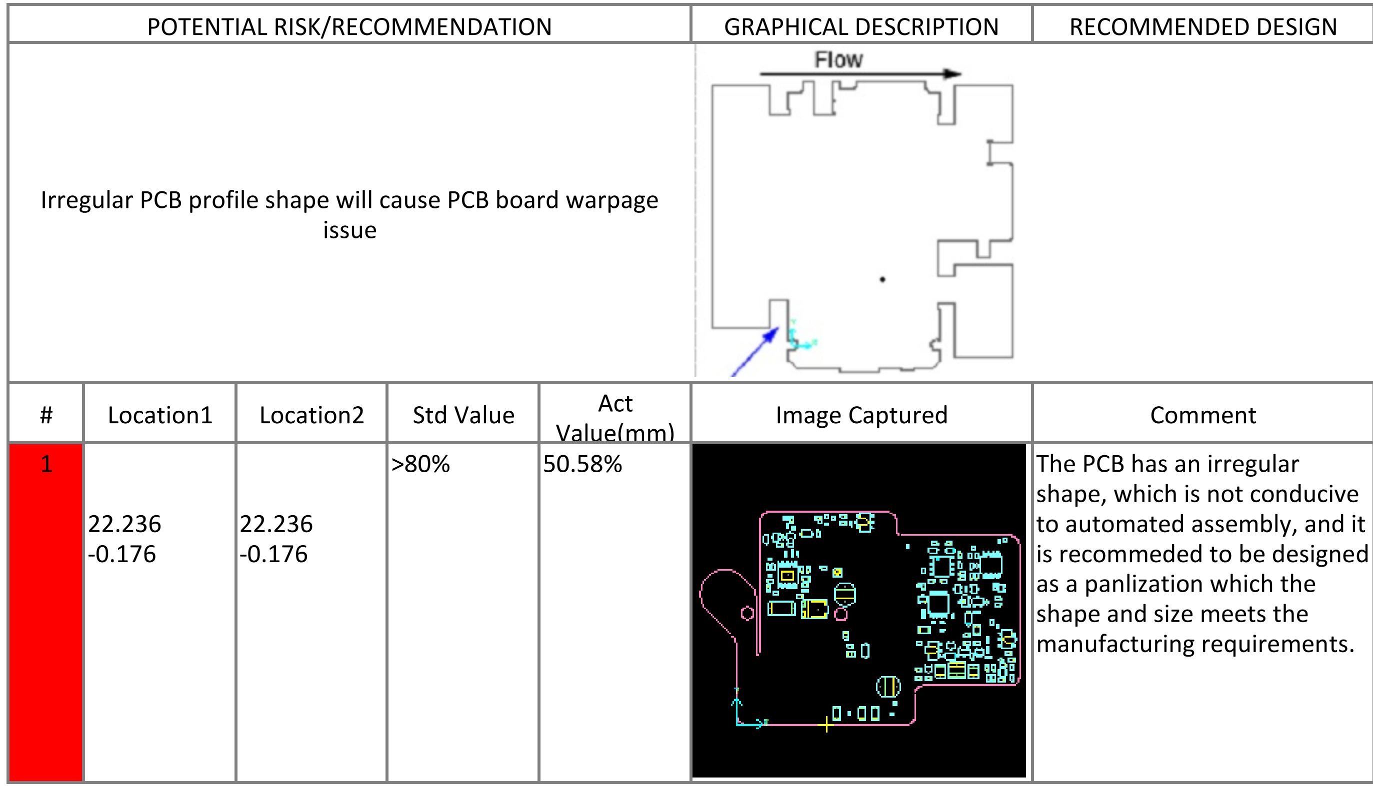
POTENTIAL RISK/RECOMMENDATION					GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Too big hole in ground pad will cause assembly issue						
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment

1	Xdrl001.drl PAD 11.850 36.050	Top Pads 12.606 36.830				The hole size is too big on position 7024's thermal pad, it may cause solder issue, the recommended hole size on thermal pad is 0.012~0.016 inch. Notice: current hole dia.=0.037 inch

Profile analysis (Fabrication)

1) IrregularPCBProfileShape

Check PCB profile shape



POTENTIAL RISK/RECOMMENDATION					GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Irregular PCB profile shape will cause PCB board warpage issue					Flow	
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	22.236 -0.176	22.236 -0.176	>80%	50.58%		The PCB has an irregular shape, which is not conducive to automated assembly, and it is recommeded to be designed as a panlization which the shape and size meets the manufacturing requirements.

Outer signal layer analysis (Fabrication)

1) FootprintWideSideConnection

The trace is not drawn from the narrow side of the pad

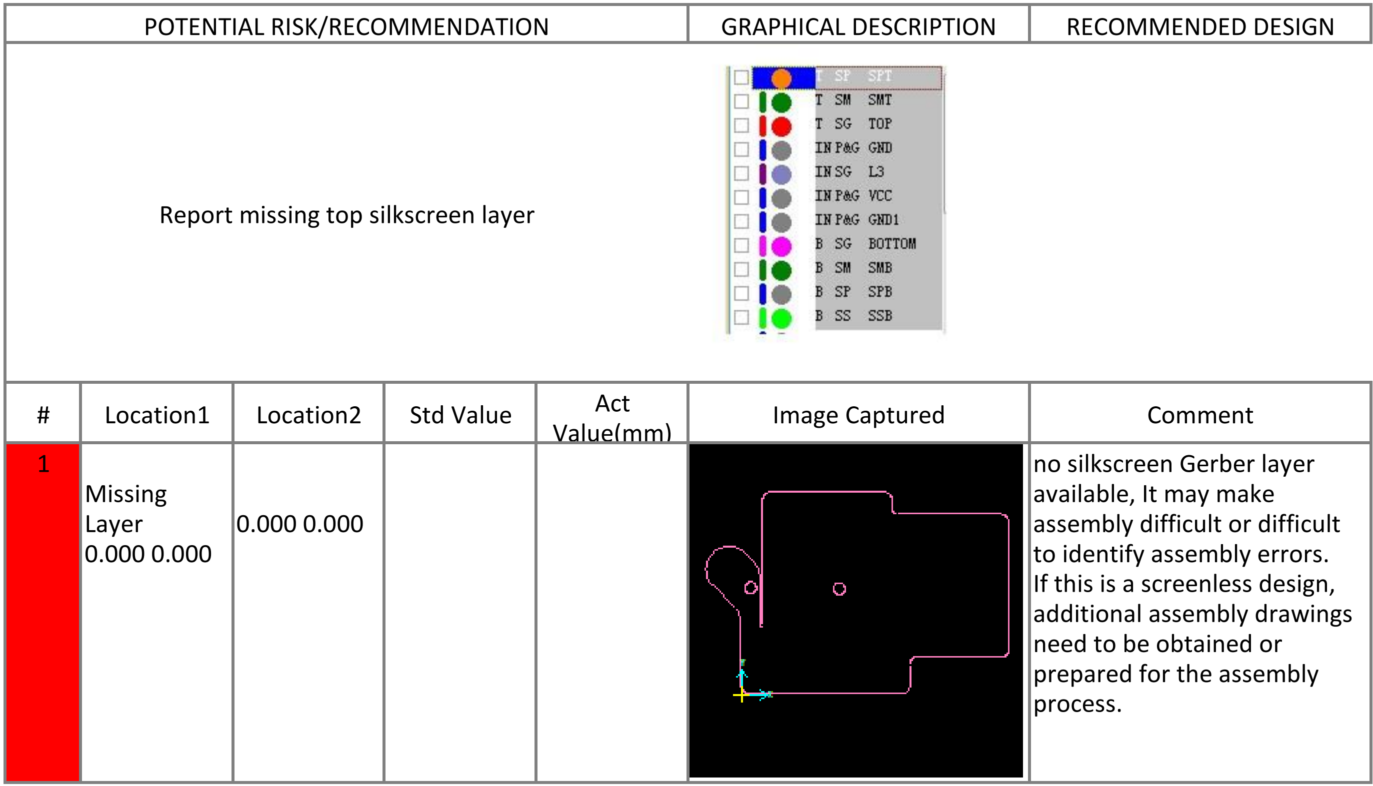
POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
The trace is recommended to pull from the pad wide side.	5 6 7	

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Top Component 7009.14 60.850 42.180	art001.pho LINE 60.825 42.350				Layout the trace from the wide side of the pad is not a standard design, which will cause abnormal solder joint shapes. The recommended design is to lead from the narrow side of the pad.
2	Top Component 7009.1 60.850 36.420	art001.pho LINE 60.475 36.500				
3	Top Component 7013.14 48.000 32.438	art001.pho LINE 47.200 31.800				
4	Top Component 7013.14 48.000 32.438	art001.pho LINE 48.325 32.500				

Silkscreen layer analysis (Fabrication)

1) MissingTopSilkscreenLayer

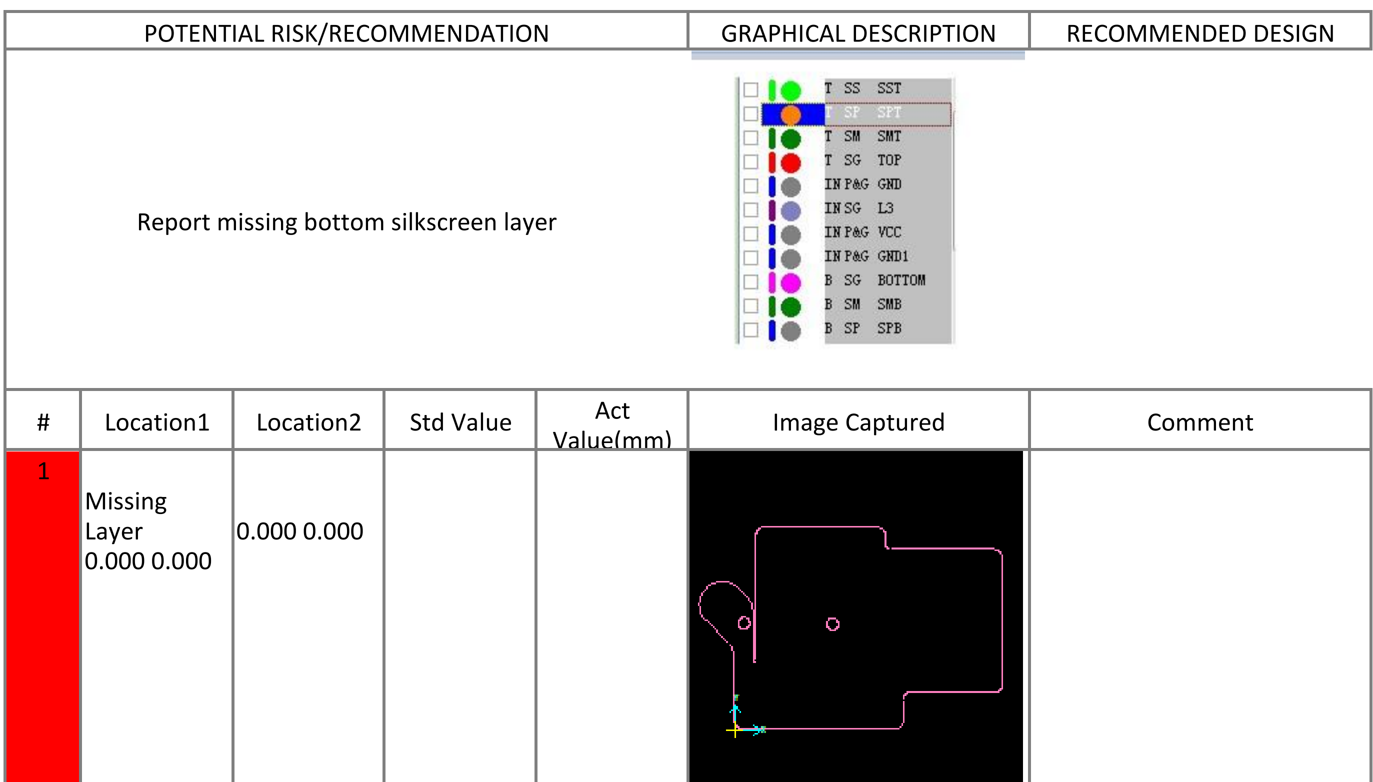
Check PCB missing top silkscreen layer



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				no silkscreen Gerber layer available, It may make assembly difficult or difficult to identify assembly errors. If this is a screenless design, additional assembly drawings need to be obtained or prepared for the assembly process.

2) MissingBotSilkscreenLayer

Check PCB missing bottom silkscreen layer

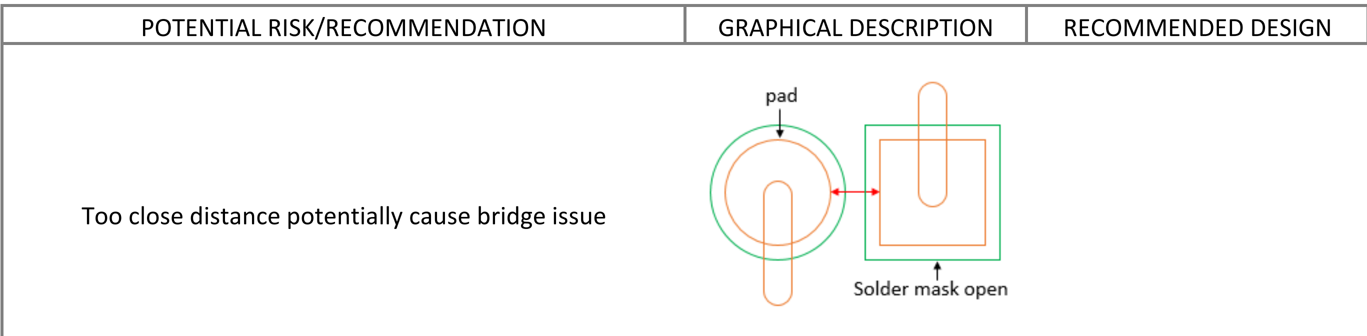


POTENTIAL RISK/RECOMMENDATION					GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Report missing bottom silkscreen layer						
#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Missing Layer 0.000 0.000	0.000 0.000				

Solder mask layer analysis (Fabrication)

1) ExposedPadToExposedPad

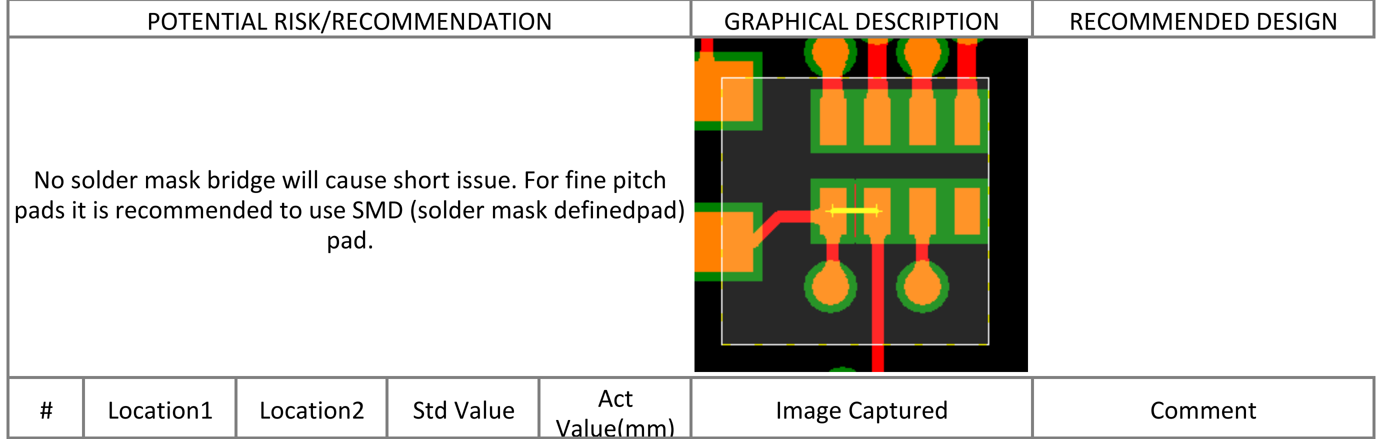
Check the distance of exposed pad to exposed pad

POTENTIAL RISK/RECOMMENDATION	GRAPHICAL DESCRIPTION	RECOMMENDED DESIGN
Too close distance potentially cause bridge issue		

#	Location1	Location2	Std Value	Act Value(mm)	Image Captured	Comment
1	Top Pads 7009.1 61.075 37.075	sm001021.pho 7009.2 61.275 37.075	=>7mil (0.178mm)	0.200		The distance between the exposed pads is too close, and there is no solder mask between the pads, which may cause a short issue. The recommended design is to add a solder mask between the pads. For this device, if the pad spacing is not enough to design a solder mask, the width of the pad can be reduced (the width of the pad on the device is too large and can be reduced by 0.1mm).
2	Top Pads 7013.10 50.375 33.225	sm001021.pho 7013.11 50.175 33.225	=>7mil (0.178mm)	0.200		

2) 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	art001.pho 58.548 29.309	sm001021.pho 57.391 29.348				It is recommeded to add a solder mask bridge between the two pads.

Appendix1: Parts Information Summary

PN Information

TOP QTY（SMD）	85
Bottom QTY（SMD）	6
Total QTY（SMD）	91
TOP QTY（PTH）	0
Bottom QTY（PTH）	0
Total QTY（PTH）	0

Part Information

TOP QTY（SMD）	149
Bottom QTY（SMD）	14
Total QTY（SMD）	163
TOP QTY（PTH）	0
Bottom QTY（PTH）	0
Total QTY（PTH）	0

Pin Information

TOP QTY（SMD）	0
Bottom QTY（SMD）	0
Total QTY（SMD）	0
TOP QTY（PTH）	0
Bottom QTY（PTH）	0
Total QTY（PTH）	0

Package Information

Minimum Pitch（SMD）	0
Minimum Pitch（PTH）	0
Maximum Pin QTY（SMD）	0
Maximum Pin QTY（PTH）	0
BGA QTY	0
CGA QTY	0

Appendix2: BOM Verification Report

BOM Verification Report

BOM & CAD Verification Report

Mismatch parts (in BOM not in CAD)

RefDes	PN	Description
1018	328 81263	AUDIO SOUNDER WASHABLE WT-1205 (SOB)
2117	012 21597	ELCAP PC 63V 100U PM20 (JAU) R
5018	225 51533	POWER-CHOKE 39U (ALT) R

Unloaded parts (in CAD not in BOM)

2140

Appendix3: 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	290 21871	16 PIN SOCKET JVT205/H43-2X8P	JVT2051H436T-2	JVT
2	290 21881	10 PIN SOCKET JVT205/H43-2X5P	JVT2051H436T-2	JVT
3	103 70862	NMOSFET 2SK3482-Z (NEC) R	2SK3482-Z	NEC
4	206 21049	ELCAP SM MV 10V 33U PM10	MV330M010C05	CPN
5	242 21623	ELCAP SM VK7 10V 470U PM20	VK7EM1A471M	YMN
6	306 71831	IC SM PIC16F15324T – I/ST (MCP)	PIC16F15324T –	MCP
7	321 70976	IC SM NJM2406	NJM2406	NJR
8	999 99999	RESERVED ITEM		

No Library RefDes List

#	RefDes	PN	Package
1	1001	290 21871	DIL16_SMT_2X2MM
2	1003	290 21881	DIL10_SMT_2X2MM
3	7014	103 70862	TO-252
4	2011	206 21049	6.3X5
5	2004	242 21623	CAP10X10.8
6	7009	306 71831	TSSOP-14
7	7002	321 70976	SOT-23-5
8	3065	999 99999	R0603