



All Issues Summary

Project: SPX19432

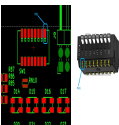
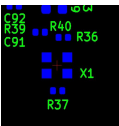

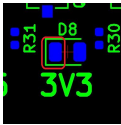
Order# 1306986-iRW Part# TRELIS

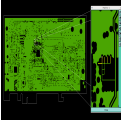
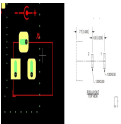

Sunnyvale, CA

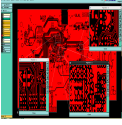
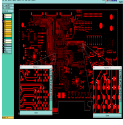

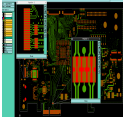
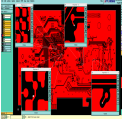
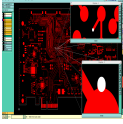
(800) 763-7503

www.protoexpress.com

This is a summary report about issues found in your data during DFA/DFM checks. To see details or to resolve any Open issues, kindly log in to your Customer Portal and go to Project SPX19432

ISSUE	ISSUE TOPIC	ISSUE DETAILS
IST138180 Mon, 09/16/2019 10:47 AM RESOLVED	Assembly-Mounting orientation confirmation needed	<p>For the DFA checks, we need you to advise if the marked pad of components mentioned below is pin-1: SW1 (Line Item# 73) [MPN: CHP-081TA]</p>  <p>Capture.JPG</p>
IST138179 Mon, 09/16/2019 10:47 AM RESOLVED	Assembly- No Polarity marking	<p>We need you to advise for the PIN1 marking for below mentioned components as it is missing in silkscreen layer. X1 (Line Item# 95) [MPN: ASE-12.000MHZ-LC-T]</p>  <p>X1.jpg</p>
IST138178 Mon, 09/16/2019 10:47 AM RESOLVED	Assembly- Cathode Mark confirmation needed	<p>We need you to advise if the marked pad of components mentioned below is cathode: D27 (Line Item# 22) [MPN: LH R974-LP-1] D8,D2,D26,D4,D5,D6,D7,D9,D10,D11,D12,D13 (Line Item# 24) [MPN: LG R971-KN-1]</p>  <p>D27.jpg</p>  <p>D8.jpg</p>
IST137318 Mon, 09/09/2019 10:37 PM RESOLVED	Data Discrepancy-Gerber related	<p>While performing DRC on your order, we observed stubs(Un-terminated traces) on inner layer4 at 5 locations. Please confirm can we proceed as is or else advise. Screen shot attached for your reference</p>

		 <p>160832-stubs on layer4.png</p>
<p>IST137275 Mon, 09/09/2019 5:53 PM</p> <p>RESOLVED</p>	<p>Data Discrepancy- Manufacturing specs</p>	<p>@Jeff --After running DRC on the design, we have observed an issue. One of the documents received by our assembly team calls for bevelling of boards at the bottom edge of the board. As discussed with you over phone, there is no beveling required as copper is very near to the board edge in layer 2,7 & 8 and beveled edges will expose the copper. As you don't want the copper to be pulled back, you have agreed to without beveling. Please reply for the confirmation.</p>
<p>IST137183 Mon, 09/09/2019 12:52 PM</p> <p>RESOLVED</p>	<p>Assembly- Mechanical/Package dimension details needed</p>	<p>For the DFA checks, we need you to provide the mechanical dimensional datasheet (Footprint information) of components mentioned below for footprint verification as we could not find one. U15 (Line Item# 79) [MPN: LFE5UM5G-85F-8BG756C] This datasheet must contain the mechanical dimensions of the component such as pitch, length, width etc.</p>
<p>IST137182 Mon, 09/09/2019 12:52 PM</p> <p>RESOLVED</p>	<p>Assembly- BOM components missing in Gerber and placement files</p>	<p>DFA checks of your files show that components mentioned below are present in BOM but missing in Gerber files and placement files. Kindly review and advice assembly location. J8,J14 (Line Item# 35) [MPN: 9774025151R]</p>
<p>IST137180 Mon, 09/09/2019 12:52 PM</p> <p>RESOLVED</p>	<p>Assembly- Foot Print mismatch</p>	<p>DFA checks of your files show a footprint mismatch of components J1 (Line Item# 29) [MPN: PJ-037A] in GERBER files w.r.t. mechanical dimensions as per datasheet. As shown in image, Gerber footprint is not matching with recommended land pattern. Kindly provide an alternate part# for this or advise if we need to make component DNI.</p>
		 <p>j1.JPG</p>
<p>IST137106 Fri, 09/06/2019 9:57 PM</p> <p>RESOLVED</p>	<p>Assembly- Other</p>	<p>For the DFA checks shows most of the components are missing in XY data, but present in BOM. Please find the attached report for your Reference. Kindly review and provide us the updated XY data, hence we can proceed with your jobs. Until then, your order is kept On-Hold for DFA checks.</p>
		 <p>XY Report.xlsx</p>
<p>IST137090 Fri, 09/06/2019</p>	<p>Assembly- IBOM issue resolution not</p>	<p>As per information received to us during online quotation, your boards are to be cleaned after assembly. But the below components are Non-Washable. Please</p>

<p>8:01 PM</p> <p>RESOLVED</p>	<p>complete</p>	<p>advise if we must follow clean process or proceed with clean such that the non-washable components will be soldered manually after assembly. SW1 (Line item# 73 MPN: CHP-081TA)</p>
<p>IST137082</p> <p>Fri, 09/06/2019</p> <p>7:19 PM</p> <p>RESOLVED</p>	<p>Assembly- XY data missing</p>	<p>Pick-and-place file is missing in your provided set of gerber files. Please provide us the pick-and-place file containing reference designators, assembly side, X location, Y location and rotation information so that we can validate your gerber files for assembly. Until then, your order is kept On-Hold for DFA checks.</p>
<p>IST137063</p> <p>Fri, 09/06/2019</p> <p>5:45 PM</p> <p>RESOLVED</p>	<p>Design Issue- Gerber related</p>	<p>While performing DFM validation on your order we observed that 3.042 mil spacing on your design (top and bottom layer) but web spec calls 4 mil spacing. Also less than 4Mils spacing is not qualified for Web orders. This may required custom quote for which our sales team will get in touch with you.</p> <div style="display: flex; justify-content: space-around; align-items: center;"> <div style="text-align: center;">  <p>bottom layer.png</p> </div> <div style="text-align: center;">  <p>top layer.png</p> </div> </div>
<p>IST137059</p> <p>Fri, 09/06/2019</p> <p>5:45 PM</p> <p>RESOLVED</p>	<p>Design Issue- Solder mask related</p>	<p>While performing DFM validation we observed that, Gang mask opening is provided in both solder mask layer. Find attached snap shot for your quick reference. This job is going to be assembled at our facility and we prefer individual mask opening. Please allow us to remove Gang relief and go ahead with individual mask opening. Please confirm.</p> <div style="display: flex; justify-content: space-around; align-items: center;"> <div style="text-align: center;">  <p>bottom mask.png</p> </div> <div style="text-align: center;">  <p>top mask.png</p> </div> </div>
<p>IST137053</p> <p>Fri, 09/06/2019</p> <p>5:45 PM</p> <p>RESOLVED</p>	<p>Design Issue- Gerber related</p>	<p>While performing DFM validation on your order, we observed few un-terminated traces (stubs) in top and bottom layer. Please confirm whether this is by design or else advise. Attached snap shot for your quick reference.</p> <div style="display: flex; justify-content: space-around; align-items: center;"> <div style="text-align: center;">  <p>bottom layer.png</p> </div> <div style="text-align: center;">  <p>top layer.png</p> </div> </div>
<p>IST137050</p> <p>Fri, 09/06/2019</p> <p>5:45 PM</p> <p>RESOLVED</p>	<p>Other</p>	<p>While performing DFM validation, we observed finger like features are present but bevel information is not provided. Can we proceed as per standard (15 mils bevel depth with 45 degree angle)? Please confirm.</p>
<p>IST137045</p>	<p>Design Issue-</p>	

<p>Fri, 09/06/2019 5:45 PM</p> <p>RESOLVED</p>	<p>Solder mask related</p>	<p>While DFM validation, we observed partial mask opening for rectangular pad at one location in top mask layer. Find Enclosed snap shot for your quick reference. Can we provide complete mask opening at this location(Pad size+4Mils) OR would you like us to proceed as is? Please confirm.</p> <div data-bbox="624 342 746 465" data-label="Image"> </div> <p>top maks.png</p>
<p>IST137041 Fri, 09/06/2019 5:45 PM</p> <p>RESOLVED</p>	<p>Design Issue- Drill related</p>	<p>While performing DFM validation we observed that, web spec calls for number of holes per board: 120, but gerber data received has 3367 number of holes. Can we proceed as per gerber data? Please confirm.</p>
<p>IST137030 Fri, 09/06/2019 5:45 PM</p> <p>RESOLVED</p>	<p>Missing information- Type of Gold/Gold thickness</p>	<p>While performing DFM validation, we observed finger like features are present but web Spec calls for gold finger as "None". Please confirm whether you require hard gold plating for the fingers or you want the whole board to be ENIG finish.</p> <p>FYI: In case of hard gold plating requirement for the fingers, we need to revise the quote for which our sales team will be in touch with you.</p> <div data-bbox="624 1093 746 1216" data-label="Image"> </div> <p>Screenshot.png</p>
<p>IST137024 Fri, 09/06/2019 5:45 PM</p> <p>RESOLVED</p>	<p>Design Issue- Gerber related</p>	<p>While performing DFM validation on your order we observed, your data contain 12 slots(11 plated and 1 non-plated) but as per web spec only 3 plated and 7 non-plated slots are allowed. This may require to revise your quote for which our sales team will be in touch with you.</p>

All Rights Reserved. Sierra Circuits, Inc.

To access the full report use the Customer Portal and go to Sierra Project SPX19432 by logging into your account at

www.protoexpress.com