**Engineering Questions**

|  |  |
| --- | --- |
| **Customer: Sig Sauer, Electro-Optics**  **Part #: 7404337**  **Revision: 00**  **PO#: 704754**  **SO#: 165438** | **Summit Tool#:d7432**  **Account Mgr: Tina Spence**,  **Prepared By: Jeshua Reyes**  **Date: 1/23/2023** |
| |  |  |  |  |  | | --- | --- | --- | --- | --- | |  | **X** | **Y** |  |  | | **BOARD SIZE** | **.7600** | **.9500** |  |  | | **ARRAY SIZE** |  |  | **# UP ON ARRAY** |  | |  |  |  | **# of layers** | **2** | | |  |  | | --- | --- | | **FAB SPEC:** | **IPC-6013 CLASS ii** | | **MATERIAL TYPE:** | **Rf-775RA / LFB Pyralux** | | **BD THICKNESS:** | **6.4 Mils at Connector / Various** | | **SURFACE FINISH:** | **ENIG** | | **IMPEDANCE PRESENT:** | **NO** | |

**\*\*\*Discrepancies noted below must have customers resolution filled in prior to manufacturing\*\*\***

|  |  |
| --- | --- |
| Item # 1: | See stackup attached for your approval with the following Requests / Suggestions   1. Fab Drawing and Separate stackup call out for Coverlay for Layer 1 and Layer 4, SMD pads on layers 1 and 4 have very tight webs that cannot be supported by coverlay, coverlay requires a minimum of 5 mil webs to prevent material from ripping apart when routing 2. Stackup calls out for extra plating, we use a non-additive process to barrel plate holes only, ED copper is not recommended on Flex Boards |
| Summit Interconnect  Recommendation: | 1.- Request to use green Flexible solder mask on layer 1 and 4 to help maintain tight webs, also since clearance comes as 1:1, we will open up mask 2 mils per side to prevent mask on pads, is this, ok?  2.- Request to start and end at Half ounce, this is also how job was quoted |
| Customers resolution: | 1. This cannot be soldermask, it must be a coverlay. In areas where there would be too small of sliver you can make it all one cutout. I can fix this in the board design and send a new file. Please advise is I should do this or if you will fix it on your side? 2. Yes I had already discussed the plating with someone, it is ok to not have that as long as the thickness of the board are roughly the same. 3. In Flex\_3 section what is the red triangle? Shown in below picture in green circle? 5. 1/24/23 This can be a normal via, it is only showing up that way because it is a 2 layer in this area. I have changed it back to a 1-4 layer via in my design so it won’t do that anymore. 6. 1/24/23Your coverlay fixes look fine |
| Summit Response To Customer Feedback | 1. If coverlay is a must then we will go ahead and add coverlay, we can fix the openings ourselves, please see attached pictures of what needs to be ganged opened, per note 12 we will go ahead and use Yellow coverlay since black coverlay is unavailable and will have to be ordered 2. Thank you for your approval thickness to be within tolerance 3. The Red triangle indicates the three vias that will be drilled from the bottom and stop in layer 3 |

|  |  |
| --- | --- |
| Item # 2: | Fab drawing gives indication of where A, B, C, And D zones are, however fab shows that stiffener goes all along section A (main) on bottom side but Section B falls in the middle |
| Summit Interconnect  Recommendation: | please confirm “B” indicates the stiffener cut out |
| Customers resolution: | Yes there should be a stiffener cutout only on the back side stiffener. Our stackup has 4 sections and B section is the same as the A section only it is missing the stiffener because it should be cut out. See some images below. This shows up as output in the pdf, it must not be outputting in other file types. Do I need to get a mechanical drawing for this?    This 3D picture is viewed from the bottom the other image are looking through the top |
| Summit Response To Customer Feedback | Thank you for your confirmation, no need to provide mechanical drawing, we can follow gerber File for dimensions |

|  |  |
| --- | --- |
| Item # 3: | Section “D” indicated where Bottom PSA is to be added however Fab assembly drawing indicates that PSA should go beyond the outline of Section “D” |
| Summit Interconnect  Recommendation: | Please advise if we should follow assembly drawing for PSA location |
| Customers resolution: | The fab drawing pdf seems to be incorrect. It is very preliminary. I think you are talking about lines not matching up in the drawing. Correct? It should just be on the rectangular area Shown in second picture in green. |
| Summit Response To Customer Feedback | Thank you for your confirmation, we will go ahead and follow Green rectangle for PSA placement |

|  |  |
| --- | --- |
| Item # 4: | Bottom Legend falls in between stiffener cut out and flex |
| Summit Interconnect  Recommendation: | Request to move legend on top of stiffener |
| Customers resolution: | I am not sure what you mean, I need to understand what you are saying better. This oval shape in your picture is the stiffener cut out. Those pins are on the bottom of the board and allow us access by having the stiffener cutout.  You can just not place it at all. We don’t need to see it for this design. Thanks |
| Summit Response To Customer Feedback | We are talking about the legend “J3” and the indicator “Dot”, if we place this legend on flex, it will then be partially covered by the stiffener and look something like the below picture where the yellow is the stiffener and the legend is on purple, we are asking to add legend ON the stiffener away from cut out  Before    After |

|  |  |
| --- | --- |
| Item # 5: | Pad size for via .008”, drill .0059” pads .015” is too small and will cause break out |
| Summit Interconnect  Recommendation: | Request to increase pad size and add tear drop to meet annular ring per class II |
| Customers resolution: | I am not sure what class II is. When I google it, I think I have found that it need to be 18, is this correct? If I change all of them in my design to be 18, I get some errors for net, vias etc. being to close. I may have to create a new layout first by moving some things.  If you do not have any drc issues you can move forward. The issues I was seeing was spacing but I have mine set to 10 mils, I think lower than 10 mils must be ok for your process. Thanks |
| Summit Response To Customer Feedback | The pads need to be increased to .016”, per our manufacturing guidelines we need at least 5 mils per side over the drill size to allow for possible miss registration, IPC allows no greater than 90° breakout of hole from land, we did not see any spacing issues after we ran our DRC’s please advise if its ok to increase to .016” |

|  |  |
| --- | --- |
| Item # 6: | As I mentioned before, we need minimum 5 mil webs for coverlay, Clearance on layer 3 has less than 5 mil webs |
| Summit Interconnect  Recommendation: | Request to gang open clearance on this location |
| Customers resolution: | Yes, sorry our production board house has a limit of three, so yes anywhere it is too small you can combine cutouts. |
| Summit Response To Customer Feedback | Thank you for your approval |

|  |  |
| --- | --- |
| Item # 7: | Some dimensions are not present on fab drawing |
| Summit Interconnect  Recommendation: | Request to follow Gerber data for all dimensions |
| Customers resolution: | Yes, you can use the gerber. I can also have them added to the drawing if you can tell me what you are missing. |
| Summit Response To Customer Feedback | We can Follow Gerber, thank you |

The above recommendations and resolutions were approved by: Date: