S1NN GmbH & Co. KG

(Automobile Connectivity Module 1.1)

By:

Tyler Alexander Ball

A Senior Project submitted

in partial fulfillment

of the requirements for the degree of

Bachelor of Science in Manufacturing Engineering

California Polytechnic State University

San Luis Obispo

Graded By:______________________ Date of Submission:______________

Executive Summary:

The results of this report show how, when diagnostics were run on the Automobile Connectivity Module 1.1, the problems discovered such as potential short-circuiting, 90-degree copper tracing angles, and components overheating/tombstoning, were addressed. By using the method of “DMAIC,” which stands for Design, Measure, Analyze, Implement, Control, successful corrections were made. In the first design of the Printed Circuit Board (PCB), 1454 critical issues were found. With the newly designed PCB modifications being implemented, only 12 critical issues remained, which were later addressed and corrected. The results from this report are a functional Automobile Connectivity Module 1.1.

Introduction:

S1NN GmbH & Co. is a highly competitive, sophisticated company specializing in the field of Infotainment Connectivity and Audio for the automobile industry worldwide. With S1NN's main facility located in Stuttgart (Germany), the ease of collaborating with major German car company clients facilitates development of professional relationships. One of S1NN's corporate goals is to make a high-end product that surpasses its competition. S1NN develops and delivers to its customers both software and hardware devices. S1NN ensures its production is fully functional by the time the final completion deadline arrives. With countless hours spent researching, designing, developing, and testing each product, S1NN aspires to be the leading edge force in the automotive industry with regard to on board entertainment systems.

With consumer demand for reliable and quality products at an all time high, there is a need to increase quality control throughout production facilities. Minimizing overall product defects and ensuring optimal design for manufacturability (DFM) requirements to be met are key components in obtaining maximum product satisfaction for both customer and producer. S1NN has customers in the automotive industry both in Europe and the United States. While working with the staff at S1nn GmbH & Co., one of its projects was assigned to my area of responsibility: an automobile Connectivity Module (CM) then being designed for production. The CM 1.1 is a device that connects mobile devices such as iPhones and iPads to the entertainment systems in cars as well as supplies enough power to charge the connected devices. This particular CM 1.1 order was for an American Car Manufacturer, consisting of strict deadlines throughout the calendar year. The task was to create and ensure a fully functional CM to provide for S1nn's customer overseas. The project requirements/processes was a perfect match to the courses I have taken here at Cal Poly. The project consisted of multiple DFM issues, a field of study that I had recently taken in school. The DFM issues of the C.M.'s Printed Circuit Board (PCB), which are also later addressed in the contents of this paper, consisted of too small of space between copper tracings and components, as well as certain components overheating. In order to accomplish a solution to these problems, modifications to the PCB needed to be addressed and implemented. Some of these modifications consisted of increasing spacing dimensions between PCB tracings. My intentions for this project were to provide insight and skills learned at Cal Poly to correct DFM issues, helping to create a functional PCB. The deliverables for this project included a prototype for testing and recommendations for a better product design that would help produce a fully functional Connectivity Module. I met each of the objectives for fixing the CM by participating in weekly meetings with other workers on the project to oversee how new modifications to the PCB performed. The items that will not be included in this report, due to non-disclosure agreements, are: Bill of Materials (BOM), Product Process dates, PCB images, PCB issues graphs/charts, and the PCB hardware casing images. The main tasks that I completed on for this project were critiquing already developed designs as well as feedback from S1nn's outsourced production partner Flextronics Inc. to provide valuable information as to why certain designs needed to be modified/revised as well as how to implement those changes. This report will also discuss a Printed Circuit Board's functionality as well as a more in depth look into what my internship efforts provided. Due to classified material and names within the company, only a limited number of names and materials can be disclosed in this report; classified names will be noted as common names and included as such in this report. The name of that project that underwent multiple adjustments and DFM checking while I was working at S1NN is called the American Original Equipment Manufacturer (OEM) Connectivity Module (CM) 1.0 and 1.1 series. The 1.0 series sample was the first generation model used by Company "F", with the hardware aspect made by S1NN. With customer requests/requirements changing from SYNC 1.0 to 1.1, changes were implemented to the hardware stages. Due to these changes, re-tracing, re-placing, and certain components were added and/or subtracted to the Printed Circuit Board (PCB) design.

Due to the changes from C.M. generations 1.0 to 1.1, problems arose while testing the copper tracings and components via specialized computer software. With deadlines for delivering a correctly made revised SYNC device nearing, insight about PCB's and DFM techniques had to be implemented in order to create a successful product.

The objectives of this project I was tasked with were to accomplish the following:

- Give insight learned at Cal Poly about possible solutions to help fix the issues about DFM at hand; and
- Collaborate with other team members to create a report, describing the product as its functionalities for future teachings and usage.

The expected deliverables for this project consist of:

- A fully functional Connectivity Module (CM) 1.1 device for S1NN to supply to an American Car Manufacturer.
- Modified Bill of Materials (BOM) with key component names/numbers changed or removed for security purposes.

- Design layout (modified due to non-disclosure agreement)

The key tasks required to complete this project comprise the following:

- Collaborate with other SYNC team members to discuss possible solutions to fix discovered issues about the PCB design.
- Implement the chosen solutions and run tests on the PCB to ensure proper functionality.

Developing new technology, whether it is software or hardware, is a complicated task. Choosing the best option to take in order to make a working prototype or final product can be difficult. An essential idea to remember when constructing a product is to know and understand the consumer base that will be using the product. Making the product "user-friendly" is a vital concept to remember. The remainder of the report describes the processes and steps taken to produce a working product for American OEM to be able to install in their cars for the automotive market.

Background:

S1NN develops both hardware and software for necessary communication technology, known as Connectivity, in the automotive industry The company was founded in 2004 and today stands as an ISO 9001 and TS 16949 certified systems supplier in the field of Infotainment as well as audio products for the automotive industry. Since 2004, S1NN has gained valuable customers in the automotive industry such as: Audi, Bentley, Daimler, Ford, Seat, Skoda, Tesla, and Volkswagen. Currently S1NN employs about 101 employees throughout the entire company, with the majority of the workforce in Stuttgart, Germany and smaller offices in Ettlingen and San Diego. Although S1NN develops hardware and software in its facilities, the physical manufacturing of the products is outsourced to Flextronics International Ltd., the second biggest electronics manufacturing services for PCB assembling in the world behind Hon Hai Precision Industry Co., Ltd. (Foxconn Technology Group).

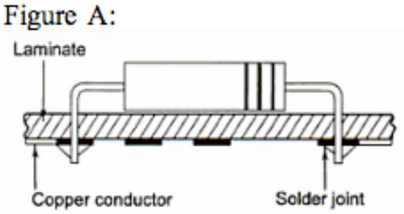
Flextronics is a Fortune 500 PCB manufacturer on a global scale with headquarters in Singapore and satellite production facilities throughout Asia, Europe, and the Americas. With the company expanded over 30 countries in 4 continents, the workforce totals approximately 200,000 employees. While touring one of Flextronics' facilities during a product test design project, I learned that Flextronics utilizes both six-sigma and LEAN manufacturing. LEAN manufacturing and six-sigma are essential in producing quality products while maximizing profits. Flextronics specializes in PCB assembly production and verification of the schematics of each PCB design sent in for production.

Literature Review:

Printed Circuit Boards:

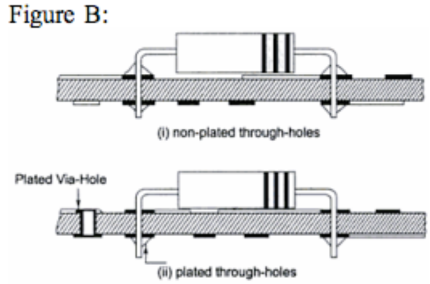
According to *Printed Circuit Board Designer's Reference: Basics* by Christopher T. Robertson, "a PCB is used primarily to create a connection between components, such as resistors, integrated circuits, and connectors" (Robertson, 2004). Copper tracings that are used to connect components together on the PCB are etched from copper sheets. There are two main different kinds of PCBs, double-sided and single-sided. Double-sided PCBs have components and copper tracings on both top and bottom of the boards whereas single-sided PCBs only have the tracings and components on one side of the boards. Double-sided PCBs consist of a greater number of components and are more complex in terms of copper tracing designs. They are also more difficult to manufacture due to different soldering techniques that are used. As stated in the book *Printed Circuit Boards: Design, Fabrication, Assembly and Testing* by R.S. Khandpur, single-sided PCBs are used when "wiring is available only on one side of the insulating substrate" (Khandpur, 2005). Khandpur goes on to state how single-sided boards are to be used when the PCB possesses simple circuitry and minimum costs are desired for the manufacturing aspect. Figure A, which appears in Khandpur's book on page 5, shows part of a single-sided PCB. The bottom side of the picture is known as the "solder side" and the top part is known as the "component side." Unlike the double-sided PCBs, the copper tracings and components are kept on different sides on the single-sided PCBs. Shown on the following page (Figure B), is a double-sided PCB illustration from

Figure A:



Khandpur's book. In the category of double-sided printed circuit boards, there are two different types of double-sided PCBs. The first type can be seen on the top of Figure B and the second type is on the bottom of Figure B. The top type is called a Non-Plated Through Hole (NPTH) PCB. The bottom board is called a Plated Through Hole (PTH) PCB. NPTHs are used for mounting and/or attaching the PCB to another piece of hardware (such as a housing device) via screws or other attachment methods. Plated Through Holes are used to connect components to the PCB, ensuring proper electrical connectivity. A visible difference between the two kinds of holes is the PTH has an annular ring around the hole on both sides of the board whereas the NPTHs don't have annular rings around their holes as well as no metal in the hole. Having no metal in the NTPH allows for easier screw threading to attach the PCB to a device without cause loose metal shavings becoming attached to the PCB, potentially causing disruptions/shorts to the PCBs.

Figure B:



Usability in Practice:

In *Usability in Practice: How companies Develop User-Friendly Products* by Michael E. Wiklund, the International Standards Organization describes usability as "the degree to which specific users can achieve specific goals within a particular environment; effectively, efficiently, comfortably, and in an acceptable manner'" (Wiklund, 1994). Wiklund believes that the vital elements going into a new design are commonly overlooked the first time. The user's ability to operate a newly designed system and/or technology is just one of the many aspects Wiklund believes the creator needs to keep in mind. Wiklund shared some common ideas with another author, G.C. Stevens. Stevens states user-friendly computer systems should follow the rule of "helping a person do a job in a natural way that is easy to understand and use." Research and development are crucial during innovations. Throughout the process of creating a new type of software, Stevens covers the idea behind participative design and why it is important to use while creating a new idea. "Involving users fully in the whole process of analysis, design, evaluation, and implementation of the system" (Stevens, 1983). Four main points mentioned by Stevens that a new technology must carry with it in order to be successful are ease of use, ease of understanding, helpfulness, and naturalness.

Mentor Graphics:

Mentor Graphics is a corporation based in the US that deals with electronic design automation (EDA) for electronics. The company strives to help their customer better electronic products faster and more cost effectively. Through this software, the Printed Circuit Boards (PCB's) could be created on computers and tested to ensure the copper traces did not overlap and cause short-circuits in the completed product. In order to timely and effectively check each trace on the PCB's, "Hyperlynx DRC" provides a set of rule checks to help prevent Electromagnetic Interference (EMI), Signal Integrity (SI), and Power Integrity (PI). Hyperlynx also checks for optimum distance between both components and copper traces. Custom violation graphics are incorporated into the software for ease of finding and diagnosing problematic circuit boards. Not only will the software show where the rule violation is occurring, but also why it is occurring. This detail in the software being used is perfect for developing professional grade PCB's in a timely manner. Hyperlynx allows the customer to browse through the different problems and issues of the PCB panel much easier. Hyperlynx offers addition testing on the desired PCB by testing the thermal aspect of the design. This test allows engineers to simulate the thermal and power integrity analysis. The purpose of this is to gain a better understanding of the effect of power distribution network current densities on board temperatures. Hyperlynx Thermal helps analyze heat-transfer mechanisms such as conduction, convection, and radiation. Due to this aspect of the software, product quality increases and the average time between each PCB failure decreases as much as 50%.

Design:

Tracing/Component Guidelines:

Precision in design of a PCB is critical. A well-made PCB requires acute awareness of part placements and adherence to copper tracings rules. Some of the ideas that need to be considered when manufacturing a PCB product are: placements of NPTH and PTH on the PCB, The uniformity of the component connection pads, and ensure that copper tracings follow basic guidelines.

Improper placements of Printed Through Holes and Non-Printed Through Holes can cause issues with the copper tracings in the immediate surrounding areas. The recommended distance between a copper tracing and a NPTH is 300 micro-meters (µm), with a minimum distance of 270 micro-meters (Flextronics Inc.). If the space between the NPTH and the copper tracings is smaller than the distance stated above, then the hardware could short to the PCB plane or the copper tracings could be cut, causing undesirable performance.

Uniformity of product placement among a components' pad is also critical in order to prevent "tombstoning" among components causing re-work. The "pad" is the area of which the PCB component rests until solder is applied. According to the book *Reflow Soldering Processes and Troubleshooting: SMT, BGA, CSP, and Flip Chip Technologies* by Ning-Cheng Lee, "Tombstoning is also known as the Manhattan effect, Drawbridging effect, or Stonehenge effect" (Lee, 2002). Tombstoning is given the explanation of when one of the two ends on a component that has gone through reflow soldering is pulling on the component at an uneven force when compared to the other side. Stated in Lee's book, is an overview about what precautions to take in order to prevent tombstoning:

> "The pad spacing, pad size, chip termination dimension, and thermal mass distribution play an important role in affecting tombstoning. Inadequate spacing between the two pads of the chips [PCB components] can cause tombstoning. Too small a spacing will cause floating of chips over the molten solder caps. Too large a spacing will cause easy detachment of either end from the pad." (Lee, 2002)

The copper tracings for PCBs have specific requirements to ensure proper functionality. Copper tracings cannot cross paths with other tracings, otherwise the PCBs will have short-circuits created and the components will not function properly. According to the book *Switching Power Supplies A to Z* by Sanja Maniktala, copper traces can vary depending on the weight of the PCB. "The so-called '1-oz' board in the United States is actually equivalent to 1.4-mils copper thickness (or 35µm) on the board" (Maniktala, 2012).

Connectivity Module 1.0:

The Connectivity Module (CM) generation 1.0 was the first generation S1NN worked on for this particular American automobile manufacturer (Company "F"). The official purchase order date of the CM 1.0 took place in February 2013 with the understanding the device being able to perform as an in-vehicle communications entertainment system; allowing users to make hands-free telephone calls (i.e. Bluetooth technology), control music, and perform other functions through the use of voice commands. The CM 1.0 had the following:

- Voice-activated, hands free calling using the "Push to Talk" button on the steering wheel.
- Automatic phone book transfer from mobile device to automobile
- Audible SMS messages
- Digital music player support: "CM" can connect to digital music players via Bluetooth or USB connections. The user can browse though music collections by genre, album, artist, or song title with the use of voice commands.
- The device can be used in various countries due to the multilingual design feature.

The features stated above had undergone hardware planning checks by S1NN and DFM with "Flextronics," S1NN's outsourcing agency, based in Paderborn (Germany), Hungary, and China. Flextronics' specialty is in PCB assembly production and verifies the schematics of each PCB design sent for production for their customers.

Connectivity Module 1.1:

Though the Automobile Connectivity Module 1.0 is useful in a majority of applications, new consumer technology requires adaptation to already existing hardware (i.e. CM 1.0). The reason for the desired change from the CM 1.0 to 1.1 was due to the new consumer devices that are on the market that now need to be considered when producing a product such as an automobile. The changes that were desired to be effectual in the CM 1.0 into a new CM 1.1, were updated:

- Performance increase (new microcontroller and new Flash and RAM)
- Increase in power output for the USB connectivity from .5 amps to 2.0 amps, allowing the charging of iPads, tablets, ect…
- Dual USB assembly possible
- Overheating protection sensor
- Added updated key components such as the new Apple Authentication IC and Bluetooth IC

S1NN deemed these new features critical in producing a desirable product for the changing consumer market. The Connectivity Module 1.1 is a more complex, better performing product than its predecessor. With this new technology, multiple devices are compatible with connecting to the automobile entertainment system. On the following page is a product progress by date for the CM 1.1 product.

Month/Year:	Description:
10/2012	Company “F” request for quotation of new product version
11/2012	Offer (Calculated cost of production) placed by S1NN to company “F”
02/2013	Purchase Order and “Kick off” of project
04/2013	“C1” (1st prototype) concept, schematics, and Bill of Materials (BOM) made.
05/2013	“C1” layout sent to Flextronics for verification. DFM and layout review feedback from Flextronics. ⇒ issues discovered with DFM.
06/2013	“C1” samples produced and received from Flextronics in China.
End of 06/2013	Hardware testing on products produced
End of 07/2013	Validation tests, Electromagnetic Charge (EMC) tests, Environmental tests. ⇒ Discovered hardware & software related issues with EMC tests.
08/2013	Completion of “C2” (i.e. updated C1) sample, which includes: fix of DFM issues, improvements of USB high-speed signal quality, and an overheating protection precaution.
09/2013	“C2” sample for product validation available.
End of 10/2013	Product validation (will be) completed.

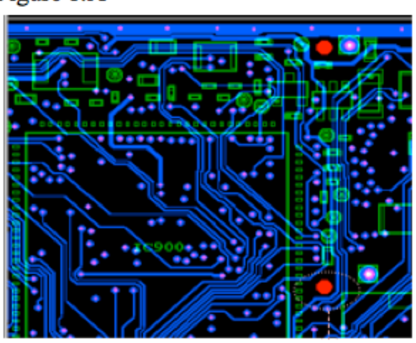
Methods:

By incorporating the skillset that I had achieved at Cal Poly in the courses of IME 144 (Introduction to Design and Manufacturing), IME 157 (Electronics Manufacturing), IME 352 (Advanced Manufacturing Process Design), and IME 418 (Product Process Design), I was able to identify problems with the CM 1.1, develop alternative solutions to fix existing problems, and implement those alternative solutions as the company saw fit. Testing of the developed PCB was outsourced to Flextronics Inc., allowing for accurate and valuable feedback of revised CM 1.1 PCB designs. When feedback of the PCB designs arrived at S1nn from Flextronics, new ways to improve the design were constructed, critiqued, and implemented. Problems that arose in current CM 1.1 designs were flagged and placed into severity categories based on how they would affect the functionality of the CM 1.1. These categories were: critical, recommended, design improvement, and closed. Critical issues were issues that needed to be corrected in order for the CM 1.1 to work properly. Recommended issues were issues that would allow the CM 1.1 to perform but could pose problems during the lifespan of the product. Design improvement category was a section to note where designs could be modified/changed but was not necessary for the CM 1.1 to properly work over the expected lifespan of the product in which the PCB was installed. The "closed" category was meant to show which problems were addressed and corrected. To remove all "critical" issues was essential and was our top priority when design corrections/improvements were being discussed and implemented.

As an Intern at S1NN, the role and responsibilities assigned were to give advice and input for the CM 1.1 design for manufacturability. During the CM 1.1 feedback from

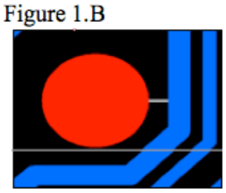
Flextronics, issues arose from the original layout created by S1NN. In order to produce an effective, working part, modifications had to be made in order for the CM 1.1 to functionally operate. Working and communicating well with other project managers and team members on the CM 1.1 plays a crucial role in the quality of the product and the timely manner in which the CM 1.1 gets properly produced. When receiving the revised DFM spreadsheet back form Flextronics, some critical errors arose for PCB component parts, copper tracings, and Non-Plated Through Holes (NPTH). In meetings with the project manager, the problems for each critical DFM feedback were addressed and prioritized according to severity for product functionality. One of the main problems discovered about the PCB components was how some of the surface mount pads were not uniform for the component that was being placed on the pads. This issue could cause an uneven force on the component, therefore causing a "tombstoning" effect. These uneven surfaces, once heated up, can cause greater stress on one side of the component. This

Figure 1.A



uneven stress can detach the weaker side from the PCB, preventing electric current from passing through it. To fix this problem, rework had to take place with ensuring all component pads were uniform and consisted of proper dimensions. Another problem faced with was some copper tracings were too close to NPTHs. Due to the small gap (229 micro-meters), the copper tracings could either be cut into by the hole's placement and/or the hardware

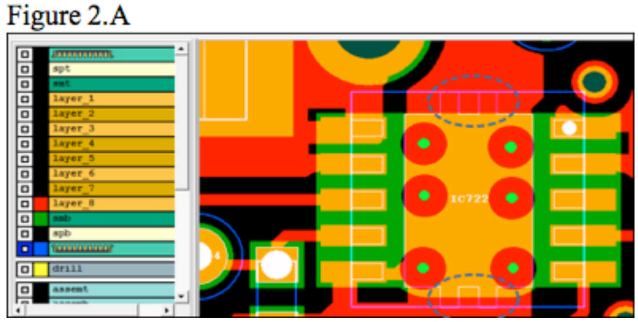
Figure 1.B



could short to the plane (Figures 1.A and 1.B). To solve this problem, the specific location of the NPTH had to be determined if the placement of the hole was critical to that exact spot, or could be adjusted slightly. Once determining the hole had to remain in the place where it rests now, re-tracing of the copper tracings around the NPTH had to be re-worked to obtain a minimum gap of 270 micrometers. Overseeing that this task was completed correctly was crucial to ensure correct DFM reports modifications. The DFM report is an extensive spreadsheet outlining critical issues, recommendations, and design improvement issues to make a more efficient PCB. Critical issues are the most important issues to attend to. These issues can cause function failure in the CM final product, causing unnecessary expenditures for re-work. Critical issues need to be fixed before mass production of the product is started.

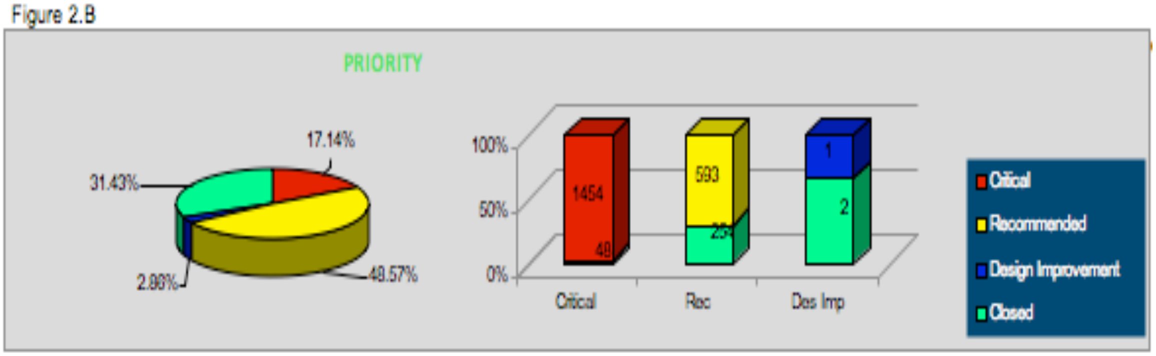
Another critical issue in the DFM report was a missing soldermask opening for a thermal pad's side pins. The reason this crucial DFM issue needed to be fixed before mass part production was that without the soldermask on the two small pins (Figure 2.A on the right), no solder would be applied to them causing a missed connection.

Figure 2.A



S1NN sends in a PCB design layout to be used for either a test batch size or mass production size order. Recommendations consist of suggestions that should be considered when looking over the design layout. The recommendations are not necessarily needed to include in the redesign stage of the PCB, although some recommendations can improve product functionality. An example of a recommended issue seen on the DFM report was the "un-terminated traces" suggestion. In the hardware layout sent to Flextronics by S1NN, there were copper traces that did not consist of start and end points. The problem with having a start point on a tracing but no end point is that a "reflection" can occur. A reflection is when information is sent via a tracing and cannot be received by the end point due to either interrupted tracing connection or no end point for receiving data. In order to find these issues mentioned above, Flextronics uses their software to analyze the PCB designs for proper functionality and feasibility. With the feedback from Flextronics, problems were addressed and solution processes were created. Displayed below in Figure 2.B is a pie and bar chart, showing both the percent and numbers of issues found in the layout of the PCB for CM 1.1.

Figure 2.B

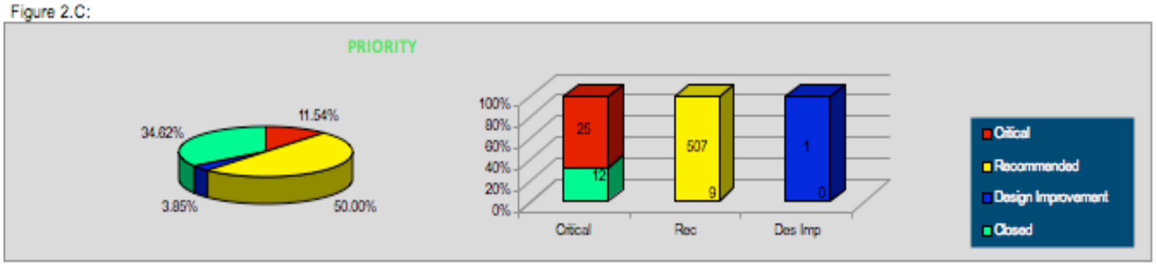


Altogether, there were 36 different issues noticed with the design of the CM 1.1 generation. Some of the issues that were not mentioned above consist of:

- Part to part spacing is too close when compared to guidelines.
  a. Solder bridging can occur because component toeprints are too close together and can ultimately result in difficult repair and/or short-circuiting.
- Part lead doesn't match Computer-Aided Design (CAD)
  a. BOM doesn't match parts being used in CAD.
- Component on top of another component.
  a. Components that are located under other components cannot be repaired or replaced without removing the top component, increasing repair costs.

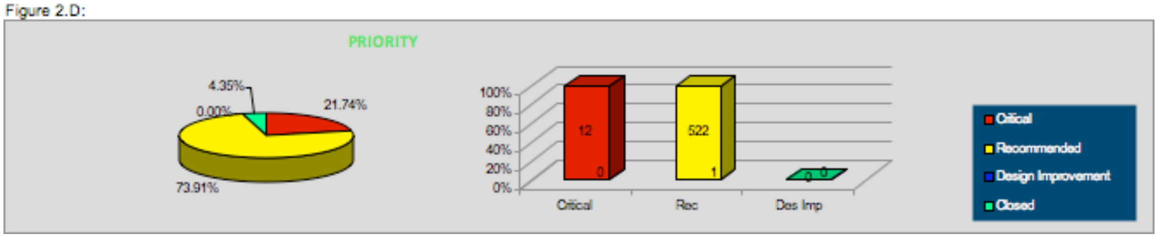
Once critical issues were addressed and corrected, a revised version CM "C2" was sent back to Flextronics for a second series of design functionality tests. Receiving the updated spreadsheet two weeks later (shown in Figure 2.C), the 1,454 "critical" issues were reduced to 25. A week later after fixing the remaining critical issues by removing "unterminated traces" and rearranging copper tracings, only 12 critical issues remained

Figure 2.C:



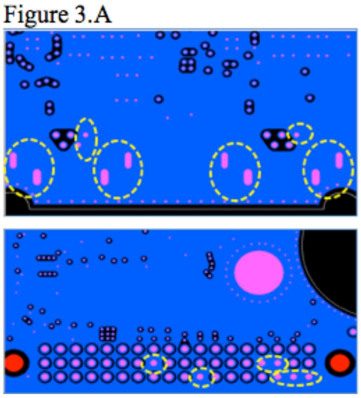
(shown below in Figure 2.D). The majority of the 12 critical issues remaining consist of the connections for soldered through hole pins not having proper thermal relief at the plane layer connections. This poses a problem because without thermals, heat sinking

Figure 2.D:



(devices used to dissipate heat from PCB components) is excessive and causes the molten solder to cool down as it passes the plane, in turn, stopping the holes' fill below minimum requirements. To solve this issue, the use of thermal connection spokes was incorporated in order to allow for more heat flow, making the PCB easier to solder. An example of the location points where problems occurred can be seen in Figure 3.A and the solution using the thermal spokes can be seen in Figure 3 (both shown on next page). Once the 12 critical issues were addressed and corrected, the schematics of the PCB were tested for errors again by Flextronics and it was noted that the PCB design was feasible for production.

Figure 3.A



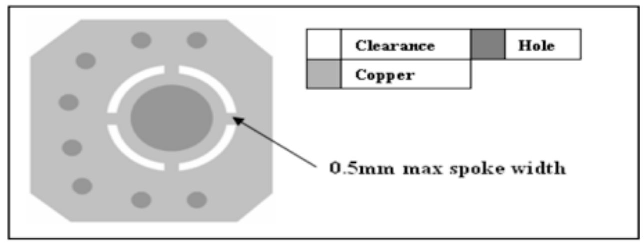


Figure 3 – Example of Isolated Internal Layer Island

Results:

By incorporating the skillset that I had achieved at Cal Poly in the courses of IME 144 (Introduction to Design and Manufacturing), IME 157 (Electronics Manufacturing), IME 352 (Advanced Manufacturing Process Design), and IME 418 (Product Process Design), I was able to determine problems with the CM 1.1, develop alternative solutions to fix existing problems, and implement those alternative solutions as the company saw fit. Testing of the developed PCB was outsourced to Flextronics Inc., allowing for accurate and valuable feedback of revised CM 1.1 PCB designs. When feedback of the PCB designs arrived at S1nn from Flextronics, new ways to improve the design were constructed, critiqued, and implemented. Problems that had arisen in current CM 1.1 designs were flagged in the put into severity categories based on how they would affect the functionality of the CM 1.1. Those issues were assessed, re-tested, re-assessed and tested to arrive at a successful PCB product. To remove all "critical" issues was essential and our top priority when design corrections/improvements were being discussed and implemented.

Cost Analysis:

Although S1nn is producing the hardware for Company "F," S1nn is not held accountable for the costs based on developing the product. The costs that S1nn encounters are labor costs and overhead costs. Due to a non-disclosure agreement, the costs of the previously mentioned items are unavailable to share.

Conclusion:

S1nn GmbH & Co. KG is becoming a globally recognized name among the automobile industry, producing high-end infotainment systems as well as software and hardware developments. The nine-week internship has taught numerous important techniques and skills to be successful in the evolving automotive industry world. Dealing with deadlines and being able to meet those deadlines played an important yet significant role in my experience. Developing a functional C.M. 1.1 was an incredible opportunity and experience. By collaborating with other S1nn employees to develop alternative solutions to the problems with previous Connectivity Module PCB designs, I was able to help ensure that the critical issues that inhibited the C.M. 1.1 from being functional were corrected. Again, these issues were addressed by using the DMAIC approach: Define, Measure, Analyze, Implement and Control. The results from using this approach were a significant decrease in critical issues (1454 issues reduced to 12), leading to a fully functional C.M. 1.1.

S1NN Internship Certificate:

S1nn GmbH & Co. KG
Gropiusplatz 10
D-70563 Stuttgart
T +49 711 901219-0
F +49 711 901219-29
www.s1nn.de

Geschäftsführer:
Philipp Popov
Andreas Heim
Registergericht Stuttgart
HRA 212366

s1nn

WORKING STUDENT CERTIFICATE

S1nn GmbH & Co. KG develops high-tech products in the field of Infotainment, Connectivity and Audio for the automotive industry worldwide.

Mr. Tyler Ball completed a voluntary internship in our company from July 8th, 2013 until September 6th, 2013 in Project Management, Software-Test and Manufacturing-Test.

His tasks covered as follows:

- Certificate verification and updating authorized validation periods for all concerning countries
- Radio diagnostics and testing for the 918 Porsche radio
- Device validation testing for Volkswagen (electric sound)
- Gave insight and helped determine proper solutions to fix hardware design problems for the Ford Sync like component layout and copper tracing routings
- Participation in software diagnostics and get to know the manufacturing operation at our contract manufacture
- Participation in conference calls with different automobile manufacturer

During the internship, Mr. Ball demonstrated excellent technical skills with a self-motivated attitude to learn new things. He was very interested in His performance exceeded our expectations.

We wish him all the best for his future endeavors.

Stuttgart, September 6th, 2013

Franziska Tokai
Human Resources

References:

Khandpur, R. S. *Printed Circuit Boards: Design, Fabrication, Assembly and Testing*. New Delhi: Tata McGraw-Hill, 2005. Print.

EFD. "Tombstone Troubleshooting." Nordson, n.d. Web. <http://www.nordson.com/en-us/divisions/efd/Literature/White-Papers/Solder/Nordson-EFD-Tombstone-Troubleshooting.pdf>.

Lee, Ning-Cheng. *Reflow Soldering Processes and Troubleshooting: SMT, BGA, CSP, and Flip Chip Technologies*. Boston: Newnes, 2002. Print.

Wiklund, Michael E. *Usability in Practice: How Companies Develop User-friendly Products*. Boston: AP Professional, 1994. Print.

"HomeFlextronics." *HomeFlextronics*. N.p., 1990. Web. 24 Aug. 2013. <http://www.flextronics.com/>.

"S1nn." S1nn GmbH & Co. KG â Infotainment, Connectivity, Audio.