ISTANBUL TECHNICAL UNIVERSITY ★ GRADUATE SCHOOL OF SCIENCE ENGINEERING AND TECHNOLOGY

DESIGN OF A PSA TEST DEVICE BY USING PRINTED CIRCUIT BOARD TECHNOLOGIES

M.Sc. THESIS

Atike DEMIRALP

Department of Electronics and Communication Engineering

Biomedical Engineering Programme

MAY 2015

ISTANBUL TECHNICAL UNIVERSITY ★ GRADUATE SCHOOL OF SCIENCE ENGINEERING AND TECHNOLOGY

DESIGN OF A PSA TEST DEVICE BY USING PRINTED CIRCUIT BOARD TECHNOLOGIES

M.Sc. THESIS

Atike DEMIRALP

(504101412)

Department of Electronics and Communication Engineering

Biomedical Engineering Programme

Thesis Advisor: Assist. Prof. Dr. İsa YILDIRIM

MAY 2015

İSTANBUL TEKNİK ÜNİVERSİTESİ ★ FEN BİLİMLERİ ENSTİTÜSÜ

BASKILI DEVRE TEKNOLOJİLERİ KULLANILARAK PSA TEST CİHAZI TASARIMI

YÜKSEK LİSANS TEZİ

Atike DEMİRALP

(504101412)

Elektronik ve Haberleşme Mühendisliği Anabilim Dalı

Biyomedikal Mühendisliği Programı

Tez Danışmanı: Yrd. Doç. Dr. İsa YILDIRIM

MAYIS 2015

Atike DEMİRALP, a **M.Sc.** student of ITU **Graduate School of Science Engineering and Technology** student ID **504101412,** successfully defended the thesis entitled **"Design of a PSA Test Device by Using Printed Circuit Board Technologies"**, which she prepared after fulfilling the requirements specified in the associated legislations, before the jury whose signatures are below.

Thesis Advisor :	**Assist. Prof. Dr. İsa YILDIRIM** Istanbul Technical University	....
Jury Members :	**Prof. Dr. Aydın AKAN** Istanbul University	....
	Assoc. Dr. Mustafa Ersel Kamaşak Istanbul Technical University	....

Date of Submission : 04.05.15

Date of Defense : 29.05.15

To my parents, Fatma and Osman,

FOREWORD

I hereby want to show my gratitude towards those who directly or indirectly made contributions to the preparation of this thesis. It is really tough thanking to my advisor Assist. Prof. Isa YILDIRIM, who, amidst his numerous works, did dedicate his time to my study ,and guide me throughout the preparation of this thesis. I should admit that, in many respects, he inspired me with his mightily positive personality and served as a sound example of a respectable educator.

I also want to thank to Dr. Yıldız ULUDAĞ who voluntarily help us when we are in need of her guidance and Tübitak Bioelectronic Group who do not hesitate when I was in need of their help.

Very special thanks goes to Hakan SALMANLI for his support and patience.

Ultimately and most importantly, I want to mention about my parents and my sister Ezgi DEMİRALP. I know that their role in my life cannot be expressed by words. What I can say is that they always supported me and showed me the right way. They always and in many respects constitute a good model in my life. I am tremendously proud of them, and do not know how to thank to them.

May 2015

Atike DEMİRALP
(Electronics Engineer)

TABLE OF CONTENTS

Page

ABBREVIATIONS

PCB **:** Printed Circuit Board
PSA **:** Prostate Specific Antigen
EMI **:** Electromagnetic Interference
BGA **:** Ball Grid Array
CPU **:** Central Processing Unit
USB **:** Universal Serial Bus

LIST OF TABLES

LIST OF FIGURES

DESIGN OF A PSA TEST DEVICE BY USING PRINTED CIRCUIT BOARD TECHNOLOGIES

SUMMARY

Printed circuit boards are such plates made of different insulating materials that they feature copper lines on the surface in order to integrate electronic circuit elements and contain holes coated inside with conductors between the surfaces. Those having only single surface can be produced unprofessionally using flat iron whereas those with multilayer require professional installations.

Pcbs designed and manufactured employing the old technologies are so big and correspondingly enlarge physical dimensions of the devices designed.

Multilayer printed circuit boards allow us to design cards in small dimensions. Placing electronic component both on upper and lower layers ensures flexibility in design. It is also possible to create capacitive effect inside the inner layers of high density cards. This offers an opportunity to reduce number of capacities occupying excessively more space in design. Transition holes coated inside with conductors must not develop a short-circuit with the copper lines of other layers. In order to prevent this, printed circuit board manufacturers developed such conceptions as blind and embedded holes. Blind hole is the one that starts from the uppermost or lowermost layers, advance several layers and end there. Embedded hole is a conducting hole that starts from one inner layer and extends to another, and can be interfered in no way. Embedded holes, as in the case of blind holes, also provide flexibility in design.

Printed circuit boards are divided into three categories as rigid, flexible and flexi-rigid combination. These card categories also are divided into 3 subcategories as single surface, double surface and multilayer. It is certain to meet rigid cards almost always in every electronic device. Flexible pcbs are employed in the moving systems, cell phones, medical devices, in brief in every field where cabling is considered to be an obstruction. Flexi-Rigid Cards are the electronic cards composed of one part fully flexible material and the other part rigid material.

Flexible cards are indispensable for medical areas. Its production, compared to rigid cards, is partly more challenging and expensive. Flexible cards have taken place at certain implant applications.

Bisens device, is a medical device that can determine pathogenic microorganisms, identify toxin, and detect biosign of cancer. In this dissertation PSA test device’s mainboard was redesigned by using new pcb technologies.

Rigid, double sided pcb made of through holes has been designed previous version of the mainboard. Old technology employed in the production of printed circuit

boards and choice of big components and footprints caused the printed circuit board to become excessively large, and in turn, device enclosure became bigger. As a result, a prototype product much bigger than it was thought it would be, was obtained.

Aim was to reduce printed circuit board size, introducing improvements in the mainboard of the medical device mentioned in this thesis study. Multilayered blind holes have also been employed in this design for such purposes.

BASKILI DEVRE TEKNOLOJİLERİ KULLANILARAK PSA TEST CİHAZI TASARIMI

ÖZET

Baskılı devre kartı elektronik devre elemanlarını monte etmek için üzerinde bakır yollar bulunan, yüzeyler arasında ise içi iletken kaplı delikler içeren değişik yalıtkan malzemelerden yapılmış plakalardır. Tek yüzlü olanları ütü ile amatörce üretilebilirken, çok katlı olanları üretmek için profesyonel tesisler gerekmektedir.

Eski teknolojiler kullanılarak tasarlanan ve üretilen kartlar çok büyük olmakta bu da tasarlanan cihazların mekanik ölçülerini büyütmektedir.

Çok katlı baskılı devre kartları küçük ebatlarda kartlar tasarlamamıza olanak sağlar. Elektronik malzemeleri hem üst hem de alt katmana yerleştirmek tasarımda esneklik sağlar. Bazı yüksek yoğunluklu kartlarda iç katmanlarda kapasitif etki yaratmak bile mümkündür. Bu da tasarımda fazlaca yer kaplayan kapasitelerin sayısını azaltabilmeyi sağlar. İçi iletken kaplı geçiş deliklerinin diğer katmanlardaki bakır yolları kısa devre etmemesi gerekir. Bunun engellemek için baskılı devre üreticileri kör ve gömülü delik denilen bir kavramı geliştirmişlerdir. Kör delikler en üst veya en alt katmandan başlayıp birkaç katman ilerleyip orada sonlanan deliklerdir. Gömülü delikler ise iç katmanda başlayıp yine iç katmanda biten, dışarıdan hiçbir şekilde müdahele edilemeyen iletken deliklerdir. Kör delikler gibi gömülü delikler de tasarımda esneklik sağlar.

Baskılı devre kartları sert, esnek, esnek ve sert malzeme birlikte olmak üzere 3'e ayrılır. Bu kart çeşitlerinin her biri tek yüzlü, çift taraflı ve çok katlı olmak üzere kendi içerisinde de 3'e ayrılır. Sert kartlar hemen hemen elektronik her cihazın içerisinde muhakkak vardır. Medikal alanda anten kartları, tanı ve teşhis cihazlarının hemen hemen hepsinde en az bir adet elektronik kart mevcuttur. Bu baskılı devre kartlarının tasarım teknolojileri cihaz kabuğunu, kullanılabilirliğini etkileyen önemli etkenlerden biridir. Esnek kartlar ise hareketli sistemlerde, cep telefonlarında, medikal cihazlarda özetle kablodan kurtulmak istenen her alanda kullanılabilir. Sert-Esnek kartlar ise bir kısmı tamamen esnek malzemeden diğer kısmı ise sert malzemeden oluşan elektronik kartlardır.

Esnek kartlar medikal alanların vazgeçilmezidir. Üretimi kısmen sert kartlara göre daha zor ve daha pahalıdır. Bazı implant uygulamalarında da esnek kartlara yer verilmiştir.

Bisens, elektrokimyasal yöntemle patojenik mikroorganizma tespiti, gıda ürünlerinde mikotoksin tespiti, medikal alanda PSA biyoişareti tespit edebilen bir tanı cihazıdır. Mısır, fındık, buğday gibi besinlerde bazı bakteriler çoğalınca

zehirli kimyasallar üretir. Deoxynivalenol ve aflatoksin tanısında 15 dakika gibi kısa bir sürede sonuç vermektedir. Bunun dışında tatlı sularda bulunabilen toksik madde salgılayan planktothrix agardhii dnası tespiti yapabilmektedir. Medikal alanda ise PSA (prostate specific antijen) biyoişareti tespitinde hızlı ve yerinde sonuç alma imkanı tanımaktadır. Bu tezde Bisens'in PSA teşhisindeki versiyonunda çalışılmıştır.

Kandaki PSA seviyesinin normal limitleri 4ng/ml dir. Kandaki PSA seviyesinin normalin üzerine çıkması prostat kanserinin bir göstergesidir. PSA testi için günümüzde insanlar hastahanelere gidip kan verip yoğunluğa göre saatlerce veya günlerce sonuç beklemektedir. Bisens, laboratuvar alt yapısı olmayan, şehire uzak yerler veya sağlık ocakları için 15 dakika gibi kısa bir sürede hızlı sonuç verebilen bir tanı cihazıdır. İçerisinde özel kimyasal sıvılar bulunduran kartuşa hastadan alınan kan enjekte edilir, cihazın kapağı kapatılıp ölçüm butonuna basıldıktan yaklaşık 15 dakika kadar sonra ölçüm sonucu ekranda gözükür. Bu hızlı sonuç verme kabiliyeti, hastanın doktorun yanındayken bile kan testi sonucunu almasına olanak sağlar.

Cihazın anakartı; güç entegreleri, wifi ve usb modülleri, motor sürücü devreleri ve çevresel birim konnektörlerinden meydana gelir. Anakartın ilk versiyonu; çift taraflı, bütün deliklerden oluşan sert baskılı devredir. Üretiminde 1.6 mm kalınlığında her iki yüzeyi ve delik içleri de bakır kaplı FR4 plaka kullanılmıştır. Anakartının teknolojisinin eski olması, büyük malzeme ve kılıflarının seçilmesi, baskılı devre kartını fazlaca büyütmüş, bu yüzden cihaz kabuğu da büyümüştür. Sonuç olarak, ortaya hedeflenenden çok daha büyük prototip bir ürün çıkmıştır. Cihaz, el cihazı olma amacından uzaklaşmıştır.

Bu tez çalışmasında bahsedilen PSA test cihazının anakart tasarımı gözden geçirilmiş, yeni bir baskılı devre tasarımı gerçeklenmiştir. Tasarımda iki katlı yapıdan, çok katlı baskılı devre yapısına geçilmiştir. Bu yapı içerisinde tasarımda kör delikler kullanılmış, bu sayede kartta yerden kazanç sağlanmıştır. Devrede empedans uyumlu olması gereken hatlar dikkate alınmış, yeni bir stackup çalışması yapılmıştır. Tek hatlarda 50 ohm empedansını sağlamak için Altium Designer, SaturnPcb yazılımları aracılığıyla simülasyonlar yapılmıştır. 50 ohm ve 100 ohm empedansları sağlamak amacıyla gerekli olan hat kalınlıklarına, baskılı devre kalınlığına, bakır ve FR4 kalınlıklarına karar verilmiştir.

Bu çalışmada elektromanyetik girişimin etkilerini en aza indirecek konfigürasyonlar seçilmiştir. Konnektörler ve işlemci dahil olmak üzere her malzemenin baskılı devre kartındaki ayakizi tekrar tasarlanmıştır. Bir çok elektronik malzemede delikli yapıdan yüzey montaj teknolojisine geçilmiştir. Böylece deliklerin oluşturduğu diğer katmanlardaki ölü alanlar kullanılabilir hale gelmiştir.

Baskılı devre kartında kullanılan minimum hat kalınlığı ve minimum delik çapları simülasyon programları aracılığıyla hesaplanıp tasarımda minimum 0.1 mm hat genişliği ve minimum 0.1 mm hat aralığı kullanılmıştır. Geçiş delikleri 0.15 mm çapında seçilmiştir. Yüksek akım çeken bölgelerde geçiş deliklerinin çapları 0.3 mm ye kadar arttırılmıştır.

Anakartın tasarımında gerçeklenen iyileştirmeler sonucunda, anakartın boyutları 101 mm x 165 mm ölçülerinden 49 mm x 95 mm olacak şekilde küçülmüştür.

Yaklaşık 4 katlık küçülme cihaz kabuğuna da aynı oranda yansıtılabilir. El cihazı olma amacıyla prototipi üretilmiş olan Bisens, önceki versiyonunda anakartın büyüklüğü sebebiyle elde taşınabilir ancak oldukça büyük bir prototip ürün olmuştur. Bu tez çalışması sayesinde, anakartta gerçeklenen tasarım revizyonu cihaz kabuğuna yansıtıldığı taktirde var olandan yaklaşık 4 kat daha küçük PSA testi yapabilen el cihazı ortaya çıkacaktır.

1. INTRODUCTION

Bisens is a diagnostic device that can determine pathogenic microorganisms with electrochemical method, identify micro toxins in food products, and detect biosign prostate specific antigen (PSA). Some bacteria growing inside the nutrients such as corn, hazelnut and wheat generate toxins. It provides results within 15 minutes with regard to diagnosis of deoxynivalenol and aflatoxin. In addition to this, it is capable of detecting DNA of planktothrix agardhii living in fresh waters and secreting toxic substance [1]. In the medical field, it offers an opportunity to provide fast and befitting results in the determination of biosign PSA. The increase in prostate specific antigen (PSA) levels in serum above the normal limits (4ng/ml) is the primary indication of prostate malignancy; therefore PSA is used as a biomarker for the diagnosis and prognosis of the prostate cancer [16,17]. In this thesis, competences of Bisens in the medical sense will be discussed and improvements will be introduced to the PCB of motherboard of the device serving such purpose.

At the beginning of the 20th century, developments regarding techniques employed in printed circuit board of today has started. Albert Hanson, a German inventor, characterized flat foil conductor placed in multiple layers on an insulating board in 1903. In 1904, Thomas Alva Edison has made experiments on plating conductors onto linen paper with chemical methods. Arthur Berry in 1913 patented a print-and-etch method in Britain and in the United States Max Schoop obtained a patent to flame-spray metal onto a board through a patterned mask. In 1927, Charles Durcase obtained a patent with regard to a method of electroplating circuit patterns [30].

Paul Eisner, an engineer from Austria, invented the printed circuit which was used in a radio set while he was working in England in 1936. And in 1936, PCB is used for commercial purposes.

A printed circuit board consists of conductive traces, pads and other features etched from copper sheets laminated onto a non-conductive substrate. PCBs can be single sided, double sided or multi layer as well. Conductors on different layers are connected with plated-through holes called vias. Every electronic product contains at least one printed board. These products range from spacecraft, submarines, medical devices and other useful consumer products. PCBs require the additional design effort to lay out the circuit. Printed board designers make decisions and follow IPC guidelines that set the stage for proper functioning and reliability of electronic products [26].

Boards can be single sided, double sided, or multi layered. They may be manufactured with special cores intended to distribute the heat more evenly or restrict movement of the organic laminate.

Early PCBs benefited from through-hole technology, where electronic components were mounted using leads inserted through holes on one side of the board and soldered onto copper traces on the other side. Such boards may be of single-sided, having an unplated component side, or more compact double-sided, having components soldered on both sides. Owing to the fact that holes must pass through all layers to the opposite side, through-hole manufacture adds to board cost by requiring many holes to be drilled precisely and restricts the routing area to be available for signal traces on layers immediately below the top layer on multilayer boards. Surface-mount technology introduced in the 1960s, gained speed in the early 1980s and by the mid-1990s became commonly used. Components were mechanically redesigned in order to have metal tabs or end caps that can be soldered directly onto the PCB surface, instead of wire leads to pass through holes. Components became quite smaller and placement of such components on both sides of the board became more common than those with through-hole mounting, paving the way for much smaller PCB assemblies with much higher circuit densities. Surface mounting makes great contribution to a high degree of automation, reducing labor costs and increasing production rates to a large extent.

The better the design of printed circuit board is, the more useful and optimum will be the design of device. In this thesis, motherboard of Bisens which is a medical device, has been improved and existing and developing PCB technologies have been examined. The most optimum solution has been adapted to the motherboard. Aim was to reduce overall dimension of the device with this solution. At this stage, in place of the old type through hole components used in the motherboard, SMD footprint components providing same function have been chosen. Card design has been updated as multilayer type. Blind holes were used in order to save space.

1.1. Motivation

For PSA test today, people go to hospital to give blood samples and wait for hours or days in order to get the result. Bisens is a diagnostic device that quickly provides results within a short time such as 15 minutes for the places or community health care centers without laboratory infrastructure and away from the city center. Small scale heath care centers without laboratory infrastructure take the blood sample from patient and send it to the more competent institutions for test. At this stage, results of the test may be mixed or samples may be lost. Moreover, one has to wait results for days. Bisens can help in handling these issues by providing fast and reliable results.

Motherboard of Bisens is composed of power integrated circuits, Wi-Fi, and USB module, and a circuit driving motors. In the early version of the motherboard design, a lot of through hole components were used. Components having big footprints were chosen and a double-sided card design has been realized. Based on that, its prototype design has been much bigger than it would be required.

1.2. Overview of Research Contributions

In this dissertation, new version main board was developed by using better pcb technologies with new small components. Smaller mainboard was obtained using a multilayer stackup with blind vias technology. The new mainboard is nearly four times smaller than the previous version.

1.3. Organization

The organization of this dissertation is as follows:

In Chapter 2, PCBs are mentioned in the medical field and in brief how Bisens operate are discussed.

In Chapter 3, printed circuit board types, rigid and flex board models, materials, multi layer pcb design methods and via types are mentioned.

In Chapter 4, design process of the pcb is mentioned. Finally, the dissertation is concluded in Chapter 5.

2. BACKGROUND AND RESEARCH OVERVIEW

In this Chapter, PCBs are mentioned in the medical field and in brief how Bisens operate is discussed.

2.1. Application of PCBs in Medical Products

Biomedical telemetry systems developed for such purposes of collection and assessment of desired parameters in one center using radio signals without restricting the movements of patients, has become the focus of interest of recent studies. Thanks to the biomedical telemetry systems in our day, monitoring many diseases can be achieved effortlessly. One of the major components of such systems are the antennas transmitting the biological signals of patient without cable to the external devices. In this regard, antennas that will be developed in order to be employed in the biomedical systems, are required to be biocompliant, compact and having low power output. Antennas become prominent in the biomedical applications taking all of these requirements into consideration [20,21]. Antennas employed in the biomedical applications are divided into three categories as those that can be implanted into the body, swallowed, and worn [22]. Implant antennas being placed at a certain fixed position inside the body, are used to ensure data transmission for microwave imaging, irregular heartbeat, diagnosis and treatment of cancer. Swallowable antennas are employed in the transmission of some parameters (for example, heat) exhibiting changes in the body, and in the diagnosis of colon cancer. Wearable antennas observed recently are placed onto the surface of the body and can be used in the transmission of such values as cardiac rhythm, blood oxygen level, blood glucose level [23]. Since implant antennas will be placed inside the body, materials to be used and antennas to be designed are required to be small in size, with low power consumption and compliant with the body [21].

Zengin, et al. designed of a small size implantable antenna. The aim of designing antenna is to obtain physiological information to the person. Simulation measurements of antenna were obtained in body with CST Studio Suite programme.

After that, in vitro measurements were performed. They performed measurements on antenna by making an artificial material. Artificial material shows electrical features of human skin tissue, to verify measurement results. Obtained measurement results and simulative results are compared and good aggrement is observed. They used 2 layer rigid pcb. Top layer contains the antenna traces, bottom layer contains solid RF ground plane [5].

Karacolak, et al. designed a new dual band implantable antenna for wireless telemetry in the MICS/ISM spectra is and tested. Antenna is tested in the real rat skin sample. The results are compared with the simulations and good aggrement is observed. They used 2 layer rigid pcb for antenna board [9,10].

Gabman, et al. developed a pcb technology which allows hybrid integration of both electronic and fluidic components at low cost in order to measure micro flow injection analysis. They used multilayer pcb which contains fluid elements and electronics for the control and evaluation [18].

Bradford, et al. developed a batteryless and wireless blood pressure sensor for monitoring aneurysm and intrasac blood pressure levels. This pressure sensing medical implant uses RF power energy. The implant consists of a low power microcontroller, an array of 4 silicon pressure sensors and antenna. All these components used with flexible Pcb. The flexible pcb material is chosen to be biocompatible [6].

Uludağ, et al. developed a new sensing platform which is called Real Time Electrochemical Profiling. This sensing platform relies on real time electrochemical immunoassay detection. This platform consists of new electrode array,microfluidic system, sensor cassette and 2 layer rigid small pcb. This pcb makes connections from electrode surfaces to the motherboard [1].

Another expanding area of printed circuit board application is in catheter-deployed therapies for treating heart and circulatory system problems. Flex pcbs have been used for electrophysiology studies, vascular diagnostic applications. The use of pcbs in in-vivo applications by applying nobel metals as conductors on biocompatible pcb materials has demonstrated in the pcb magazine. For short term implants the use of gold plated copper tracks has proven well results but long term implants should be completely copper-free structures [28-29].

2.2. Bisens

In our day, tests for disease biomarkers are often carried out in central laboratories that utilize large automated clinical analyzers. Tests for food and environmental safety, likewise, are typically performed by central laboratories which utilize expensive laboratory equipment such as Gas Chromatography (GC), GC-Mass Spectrometry (MS), Liquid Chromatography (LC)-MS or Enzyme-linked immunosorbent assay (ELISA). These processes, which involve high costs, include some requirements such as sample transportation, increased waiting time and trained staff. In this context, it becomes necessary to develop a portable detection system which enables on-site testing, uses low sample and reagent volumes and which, at the same time, is easily operated, cheap and capable of quick processing.

Biosensors once used to be viewed only as an analytical tool for biomolecular interactions, but now they are widely utilized in diagnostics, biological defense, environment and quality assurance in agriculture and food sectors. Drawing upon lab-on-a-chip technologies and nanotechnology, biosensor devices have aroused a significant interest in biological defense research and development during the recent years. In the face of numerous publications and patents available, however, commercialization of biosensors technology has significantly failed to keep up with the research output [2,8]. As analytical devices, biosensors consist of a biological recognition element and a proper transducer, and they are mostly connected to a compatible data processing system. The element of biological recognition might be an enzyme, a micro-organism, a tissue or a bioligand such as antibodies and nucleic acids.

Collaboration of a multidisciplinary group including scientists and engineers from various disciplines such as biotechnology, molecular biology, material science, microfluidics, electronics and software engineering is necessary for building an integrated biosensing platform . Multidisciplinary dimension of biosensor prototype design and system integration is shown in the Figure 2.1 [3].

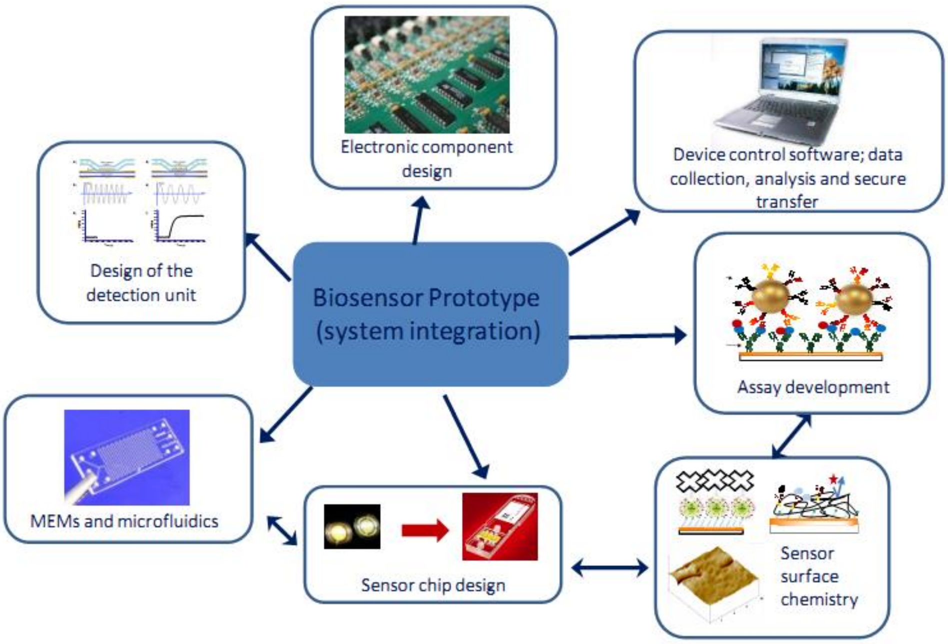

Figure 2.1 : Multidisciplinary dimension of biosensor prototype design [3].

Different biosensor types have been tried for detecting cancer biomarkers. Electrochemical biosensors, optical biosensors are used for PSA analysis [24,25]. A detection technology called "Real Time Electrochemical Profiling" (REP) was developed by Y. Uludağ et al. Although this technology draws upon the fundamental basics of electrochemical immunosensing, especially amperometry, it has some key features such as a new electrode array, a microfluidics-based assay and real-time amperometric measurements during the flow of an enzyme substrate. They developed a sandwich assay with an antigen and detection antibody/enzyme-modified nanoparticle) during microfluidic flow. Subsequently, a TMB reagent is also injected on surfaces at a specific flow rate while current is measured against a potential and plotted versus time [2,8].

Bisens, which is PSA test device, consists of biosensor, microfluidics, step motors, cartridge, main board pcb and peripheral pcbs. Bisens is shown in the Figure 2.2. Different peripheral modules like wifi module, cpu, motor drivers or usb operates different voltage levels. Because of system's input voltage is 12 Volt, voltage regulators are used.

Figure 2.2 : Bisens

General logic of the device operation: Catridge is installed to the device, along with the start command, step motors operate in the following order, and pushes the liquids in cartridge towards microfluidic canals.

1. Motor 1 Buffer Cleaning Fluid
2. Motor 4 Serum Sample
3. Motor 2 Antibody + Enzyme modified Au nanoparticle
4. Motor3 TMB substrat

Each liquid passes over the electrode surface and is collected in the disposal sump.

Figure 2.3 shows the cartridge. Cartridge is composed of 5 partitions. 4 of the partions are like miniature syringes. Waste chemical liquids are collected in the 5th partition. There is a cleansing chemical in its first partition. That liquid pushed by a step motor after every cycle cleans the surface of electrode. Blood sample taken from the patient is put to the last partition of cartridge. Each cartridge is single use only. Blood sample is sent to the sensor surfaces when device is started. Mainboard measures the PSA levels in serum and send the results to the display. This process takes only 15 minutes. The increase in PSA levels in serum above the normal limits is the primary indication of prostate malignancy.

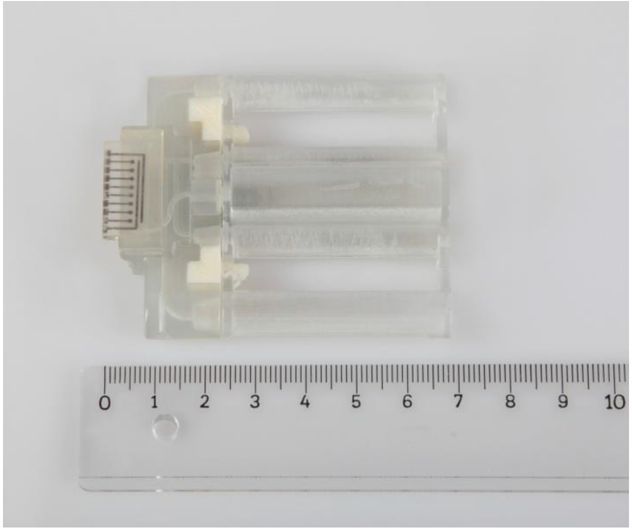

Figure 2.3 : Cartridge

PSA capture were injected across sensor surfaces. This step is followed by the injections of PSA antigen and PSA detection antibody/HRP modified Au nanoparticle. To obtain an electrochemical signal, a REP assay has been performed by applying -0.1 V potential to the electrode array and the current was measured continuously. Initially the current measurements were taken during PBS buffer injection and this signal was recorded as a baseline. The subsequent injection of 250 µL TMB incurred a current change and the substraction between the current obtained during TMB and buffer was used as sensor response [2]. This process is shown in Figure 2.4.

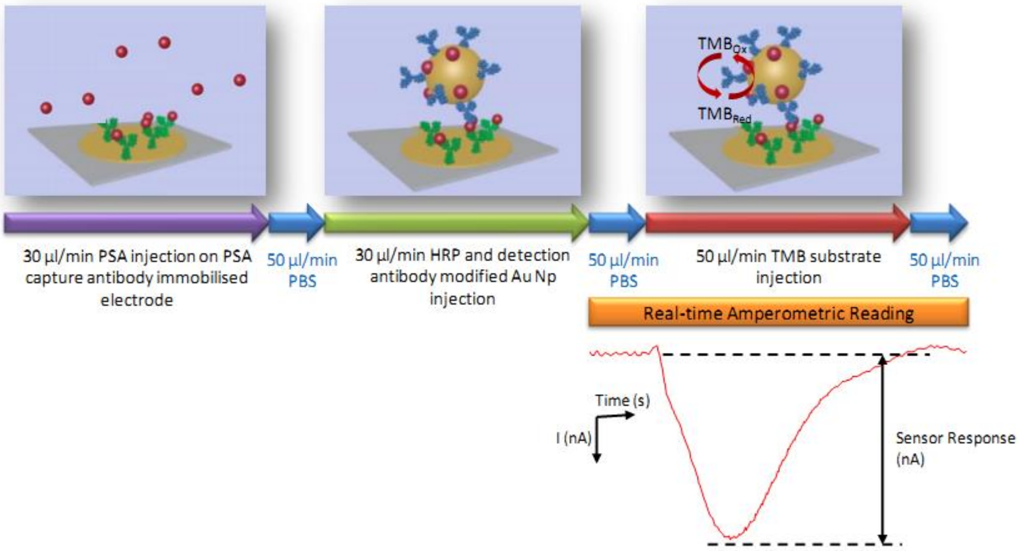

Figure 2.4 : The steps of the REP assay [2].

2.3. Summary

As demonstrated above, pcbs are increasingly important part of modern medical devices. In this chapter uses of PCBs in the medical field were discussed. Bisens which is a PSA test device was introduced.

3. PRINTED CIRCUIT BOARD LAYOUT REQUIREMENTS

In Chapter 3, printed circuit boards types, rigid and flex board models, materials, multi layer pcb design method and via types are mentioned.

3.1. Printed Circuit Board

A printed circuit board consists of conductive traces, pads and other features etched from copper sheets laminated onto a non-conductive substrate. PCBs can be single sided, double sided or multi layer. Conductors on different layers are connected with plated-through holes called vias. Every electronic product contains at least one printed board. These products range from spacecraft, submarines, medical devices and other useful consumer products. PCBs require the additional design effort to lay out the circuit. Most of the systems consist of multilayer boards of up to 20 layers or sometimes even more. Components are mounted on the top or bottom layers. The design of the pcb can be as important as the circuit design to the overall performance of the system. An example of printed circuit board is shown in the Figure 3.1.

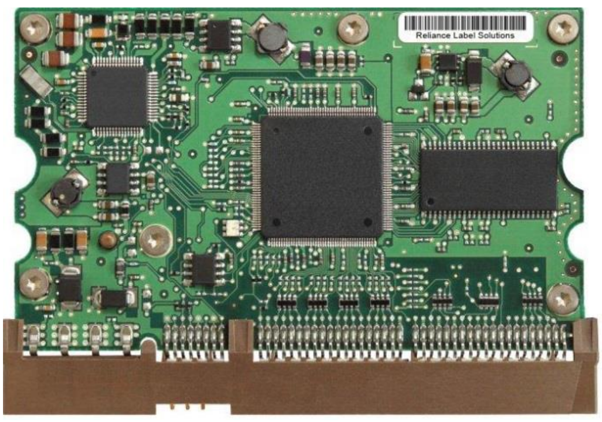

Figure 3.1: Printed circuit board with green solder mask and its components

3.2. Materials

The designer has several material choices to consider, ranging from standard materials to highly sophisticated materials. The designer should determine what requirements the pcb meet when selecting materials firstly. Printed circuit boards consist of three parts which is named the resin, the reinforcement and the copper foil. The most popular reinforcement is woven glass. It provides structural strength to the resin. A sheet of woven glass that has been coated with resin is referred to as prepreg. This material can be combined with other sheets and then laminated under heat and pressure to form the base material. Core material is made of the same materials but fully cured. Core materials typically has copper clad on either one side or both sides. Multilayer boards are made up of a number of processed cores, with sheets of prepreg material between core. They are laminated and after the lamination core and prepreg materials are indistinguishable. Base materials may be rigid of flexible. These require product flex or rigid boards [26,27]

3.3. Multilayer PCB Stackup Planning

Planning the multilayer pcb stack up is the most important issue to achieve the best possible performance of the system. An inappropriately selected materials can degrade electrical performance of signal transmission and can also make the product more susceptible to external noise. These issues can cause reducing the long term reliability of the products. When selecting a multilayer stackup we should consider the following:

- A signal layer should always be adjacent to a plane.
- A power plane can be used for the return path of the signal.
- Determine the return path of the signals. Fast rise time signals take the path of least inductance which is normally closest plane.
- Cost

A typical four layer rigid pcb stack up is shown in Figure 3.2. The characteristic and differential impedances were calculated using ICD Stack-up Planner.

UNITS: MIL — Total Board Thickness: 61.2

Layer		Material	Dielectric		Copper	Trace		Current	Impedance	Edge Coupled	Broadside Coupled	
Number	Name	Type	Constant	Thickness	Thickness	Clearance	Width	(Amps)	Characteristic(Zo)	Differential(Zdiff)	Differential(Zdbs)	Description
		Dielectric	3.3	0.5								Soldermask
1	Top	Conductive			0.7	15	15	0.49	51.93	90.98		Signal
		Dielectric	4.3	9								Prepreg
2	GND	Conductive			1.4							Plane
		Dielectric	4.3	39								Core
3	VCC	Conductive			1.4							Plane
		Dielectric	4.3	9								Prepreg
4	Bottom	Conductive			0.7	15	15	0.49	51.93	90.98		Signal
		Dielectric	3.3	0.5								Soldermask

Figure 3.2 : Typical 4 layer stack up

A typical six layer rigid pcb stack up is shown in Figure 3.3.

UNITS: MIL — Total Board Thickness: 63

Layer		Material	Dielectric		Copper	Trace		Current	Impedance	Edge Coupled	Broadside Coupled	
Number	Name	Type	Constant	Thickness	Thickness	Clearance	Width	(Amps)	Characteristic(Zo)	Differential(Zdiff)	Differential(Zdbs)	Description
		Dielectric	3.3	0.5								Soldermask
1	Top	Conductive			0.7	12	12	0.42	50.81	89.45		Signal
		Dielectric	4.3	7								Prepreg
2	GND	Conductive			1.4							Plane
		Dielectric	4.3	15								Core
3	Inner 3	Conductive			1.4	12	12	0.69	52.65	88.5	66.91	Signal
		Dielectric	4.3	12								Prepreg
4	Inner 4	Conductive			1.4	12	12	0.69	52.65	88.5	66.91	Signal
		Dielectric	4.3	15								Core
5	VCC	Conductive			1.4							Plane
		Dielectric	4.3	7								Prepreg
6	Bottom	Conductive			0.7	12	12	0.42	50.81	89.45		Signal
		Dielectric	3.3	0.5								Soldermask

Figure 3.3 : Typical 6 layer stack up

The total number of layers required for a design is dependent on complexity of the design. Signal nets must break out from the BGA package. Variety of the power signals should have different power planes. This decisions increase total number of layers.

3.4. Holes in PCBs

Through holes are used for mounting leaded components or may be used as a via to make a connection between various layers. The via is a plated through hole which is used as an interlayer connection [26].

3.5. Blind and Buried Vias

Blind vias extend only to one surface of the pcb. Blind via holes should be filled with a polymer or solder resist to prevent solder from entering them.

Buried vias extend only between conductive layers. Most commonly the interconnection is between two adjacent layers. Typical blind and buried vias shown in Figure 3.4. Number 1 is the blind via, number 2 is the buried via and number 3 is the through via.

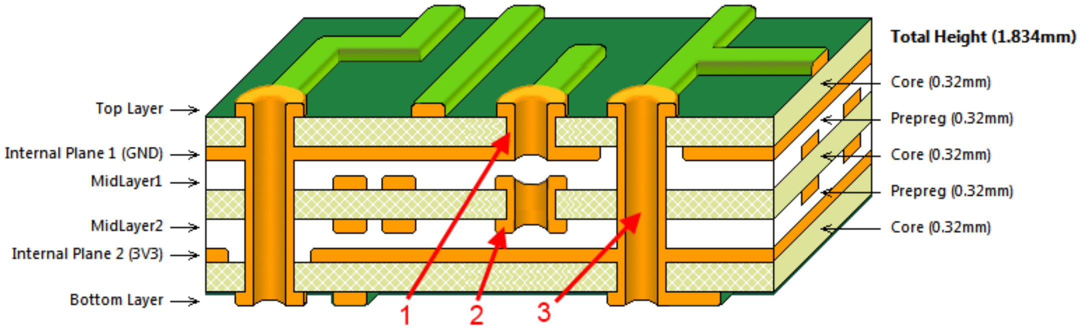

Figure 3.4 : Multilayer Stack Up

3.6. Flex PCBs

Flex circuits are used from simple consumer products to spacecraft. They are critical elements in such diverse products as medical equipment, keyboards, hard disk drivers, printers and mobile phones. Every day, flexible circuit technology creates new opportunities for engineers and designers. Flexible circuits provide best and practical solutions to interconnecting electronic systems. Figure 3.5 shows a flexible pcb assembled with surface mount components.

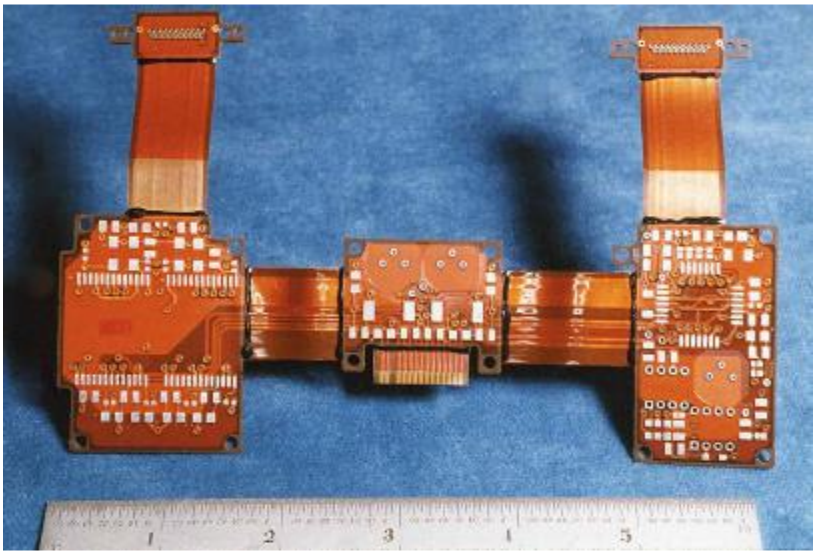

Figure 3.5 : Flexible PCB

Flexible pcbs can be single or double sided and multilayered flex pcb like rigid pcbs. Single sided flex pcbs consist of single conductor layer or conductive polymer on a flexible dielectric film. Double sided flex pcbs have two conductor layers. Multilayer flex pcbs have three or more conductive layers [26].

Flexible electronics are light, portable and so thin and flexible that they can compatible with the human body. They represent a burgeoning, but fast-growing industry.

Such devices have been successfully fabricated for short term implants like blood glucose sensors, balloon catheters and diagnostic catheters. Examples of implantable pcb layouts and corresponding applications are shown in Figure 3.6. For short term implants the use of gold plated copper tracks has proven good results but long term implants should be completely copper-free structures. Short and permanent implant's pcb layouts are shown in Figure 3.7. Short term implants have hybrid pcb layout but permanent implants are completely biocompatible [29].

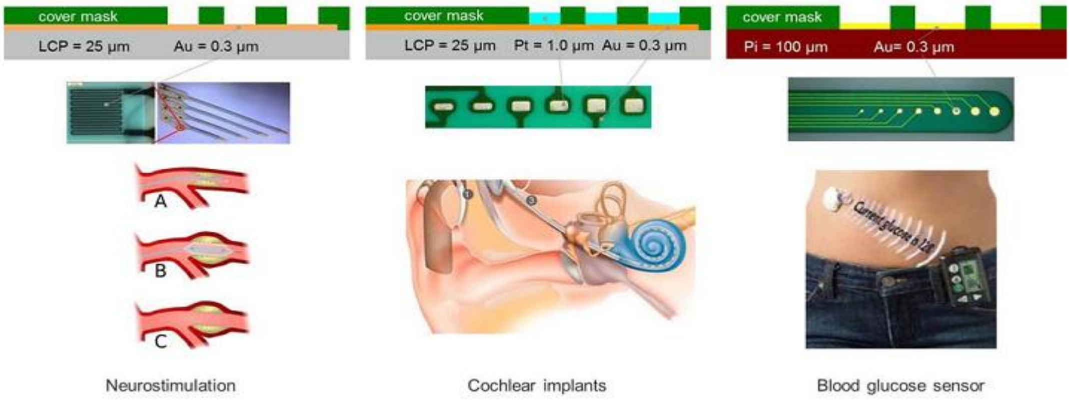

Figure 3.6 : Examples of implantable pcb layouts and their corresponding applications

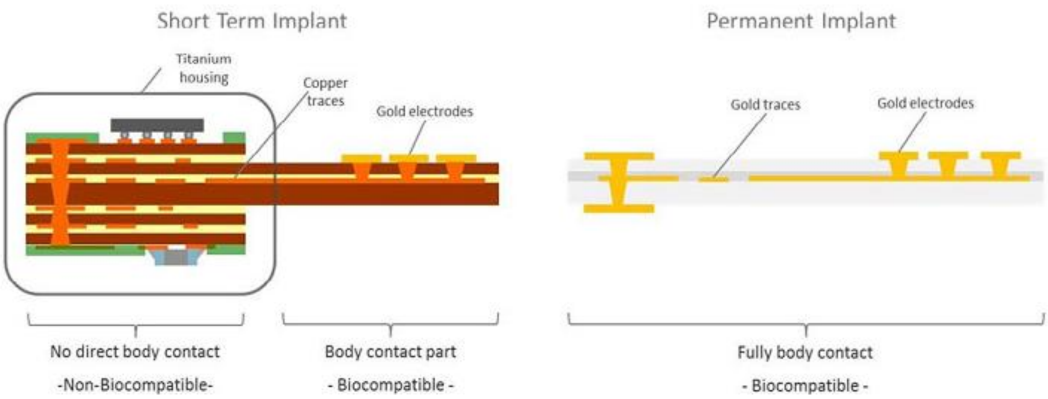

Figure 3.7 : Short term and permanent implants

3.7. Rigid Flex PCBs

Rigid flex pcbs are consist of rigid and flexible substrates. The rigid flex circuits have hybrid constructions. These type of pcbs are employed in the moving systems, cell phones, medical devices, in brief in every field where cabling is considered to be an obstruction. Figure 3.6 shows a rigid flex pcb.

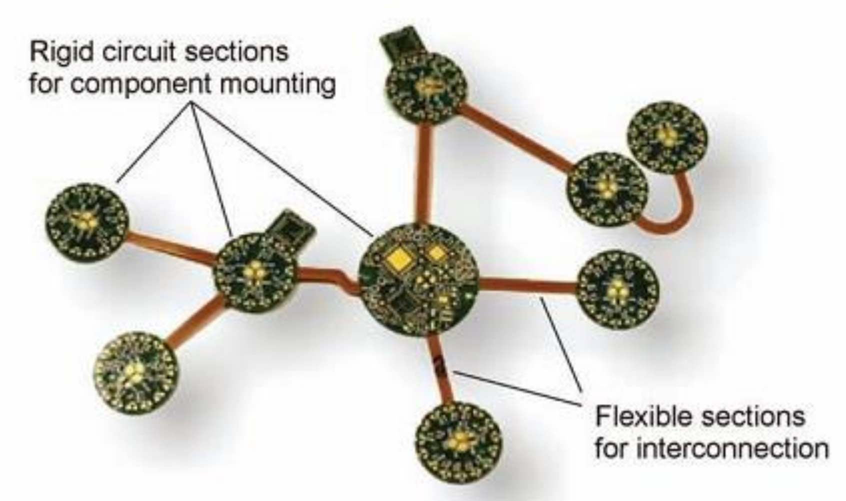

Figure 3.8 : Rigid-Flex Pcb

3.8. Summary

In this Chapter, printed circuit types via types and materials are discussed. Following that multilayer pcb stack up planning is given and several examples of stack up are considered briefly.

4. PRINTED CIRCUIT BOARD DESIGN TECHNIQUES

4.1. Routing Topologies

Several techniques exist for designing multilayer pcb. Two basic topologies are microstrip and stripline topology [12,13].

Microstrip is a well liked method to acquire trace controlled impedance on pcb. Microstrip lines are exposed to both air and a dielectric material. The approximate formula for microstrip is provided in Eq. (4.1a) and Eq. (4.1b). Microstrip topology is shown in Figure 4.1.A .

$$Z_0 = \left(\frac{87}{\sqrt{\varepsilon_r + 1.41}}\right) \ln\left(\frac{5.98\,H}{0.8W+T}\right)\Omega \quad \text{Valid for} \quad 15 < w < 25 \quad \text{mils} \qquad \textbf{(4.1.a)}$$

$$Z_0 = \left(\frac{79}{\sqrt{\varepsilon_r + 1.41}}\right) \ln\left(\frac{5.98\,H}{0.8W+T}\right)\Omega \quad \text{Valid for} \quad 5 < w < 15 \quad \text{mils} \qquad \textbf{(4.1.b)}$$

where H is the distance between signal trace and reference plane, W is the width of the trace, T is the thickness of the trace.

Embedded microstrip, which is shown in Figure 4.1.B, has the same conductor shape but the effective dielectric constant is different because of the embedded conductor layer. Embedded microstrip is described in Eq. (4.2).

$$Z_0 = \left(\frac{87}{\sqrt{\varepsilon_r' + 1.41}}\right) \ln\left(\frac{5.98\,H}{0.8W+T}\right)\Omega \quad \text{where} \quad \varepsilon_r' = \varepsilon_r \left\{1 - e^{\left(\frac{-1.55B}{H}\right)}\right\} \qquad \textbf{(4.2)}$$

where H is the distance between signal trace and reference plane, W is the width of the trace, T is the thickness of the trace.

A stripline, which is shown in Figure 4.1.C, has a thin narrow conductor. The conductor layer is between two AC ground planes. Since all electric and magnetic field lines are kept down between the planes, this configuration is useful for suppressing EMI. Crosstalk between circuits is also lower than microstrip case. A stripline is described in Eq. (4.3).

$$Z_0 = \left(\frac{60}{\sqrt{\varepsilon_r}}\right) \ln\left(\frac{1.9\,B}{(0.8W+T)}\right)\Omega = \frac{60}{\sqrt{\varepsilon_r}}\left(\frac{1.9(2H+T)}{(0.8W+T)}\right) \quad \textbf{(4.3)}$$

Dual stripline is shown in Figure 4.1.D. When using the dual stripline, both signal layers must be routed orthogonal to each other to prevent crosstalk. A dual stripline is described in Eq. (4.4).

$$Z_0 = \left(\frac{80}{\sqrt{\varepsilon_r}}\right) ln\left[\frac{1.9\,B}{(0.8W+T)}\right]\left[1 - \frac{H}{4(H+D+T)}\right] \quad \textbf{(4.4)}$$

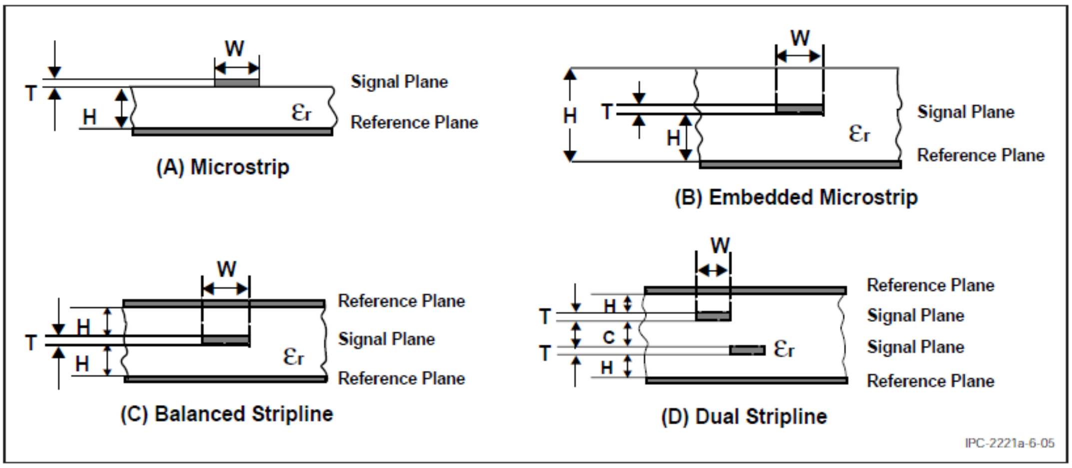

Figure 4.1 : Printed board construction

4.2. Crosstalk

One of the several important aspects of the PCB design that must be taken into consideration during design cycle is crosstalk. Crosstalk means electromagnetic coupling that will not be desired between traces, trace to wire, components and any other electrical components. Crosstalk is an undesirable phenomenon, in general, associated not only with clock or periodic signals, but also with data, address and I/O traces. Figure 4.2 shows crosstalk phenomenon. Energy from one trace is transferred to an adjacent trace based on electromagnetic coupling.

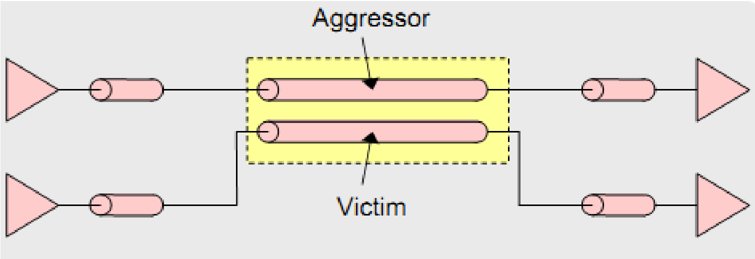

Figure 4.2 : Energy from one trace is transferred to an adjacent trace based on electromagnetic coupling

To prevent crosstalk within the pcb, designers care some extra important layout techniques. These are listed in below. Crosstalk separation is shown in Figure 4.3.

- Minimize physical distance between components
- Minimize parallel routed trace lengths
- Avoid routing of traces parallel to each other. Provide sufficient clearance between traces to minimize inductive coupling
- Route adjacent layers orthogonally to prevent capacitive coupling between the planes.

The associated local magnetic field strength drops off with distance. Eq. (4.5) shows that to minimize crosstalk, minimize H and maximize D.

K is a constant and always less than one.

H is the distance between signal trace and reference plane, W is the width of the trace, T is the thickness of the trace.

$$\text{Crosstalk} \approx \frac{K}{1+\left(\frac{D}{H}\right)^2} = \frac{K(\mathrm{H})^2}{H^2 + D^2} \qquad \textbf{(4.5)}$$

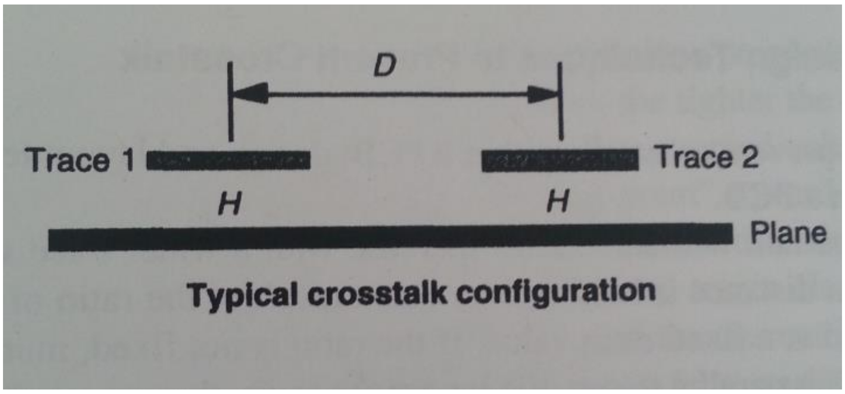

Figure 4.3 : Calculating crosstalk separation

Montrose easily explained 3-W rule in his book. This rule minimizes coupling between traces. 3-W rule states that the distance separation between traces must be three times the width of a single trace, measured from centerline to centerline. Figure 4.4, Figure 4.5 and Figure 4.6 show distance rules to prevent crosstalk.

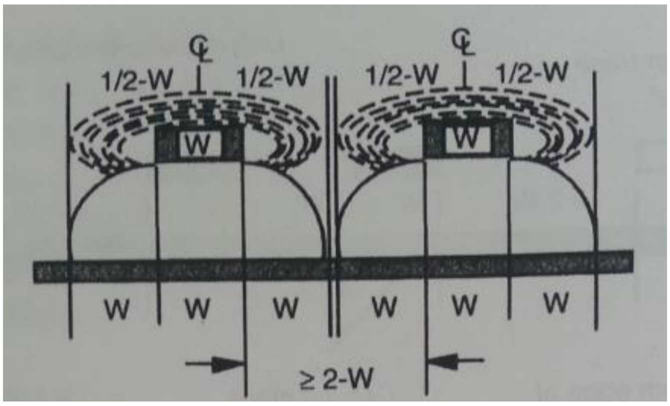

Figure 4.4 : The distance spacing between both traces must have a minimum overlap of 2W

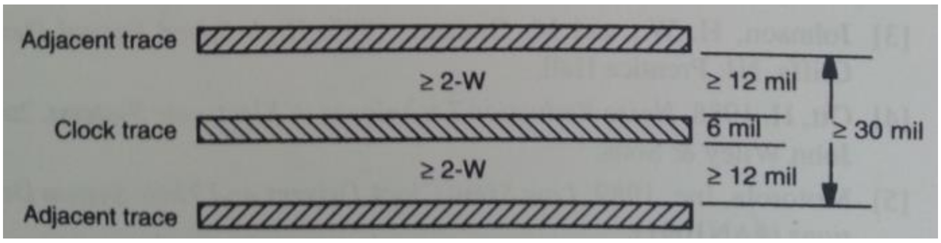

Figure 4.5 : 3-W spacing between the traces

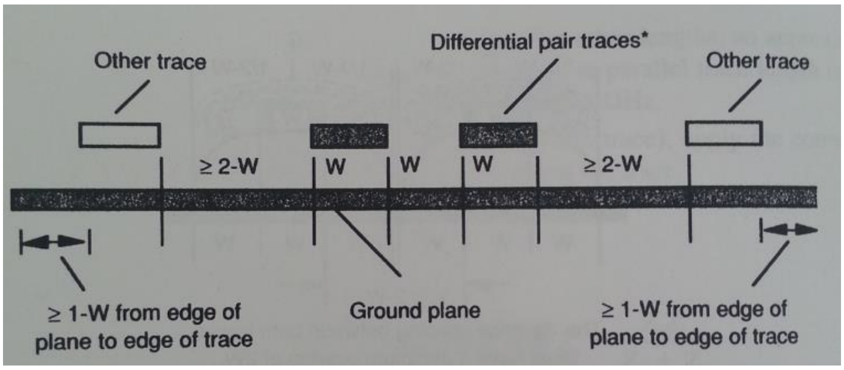

Figure 4.6 : Parallel differential pair routing and the 3-W rule

4.3. Configure Via Types

Holes are produced mechanically or by laser. Minimum drilled hole size is important for pcb design cycle. Otherwise pcb could not be manufactured. In either case aspect ratio must be considered for plating process. Small and deep blind vias are difficult to plate. For this reason, the aspect ratio of the plated through holes plays an important role ability of the manufacturer to provide sufficient plating. According to IPC standards, minimum drilled hole size for blind and buried vias is shown in the Table 4.1 and Table 4.2.

Table 4.1 : Minimum Drilled Hole Size for Buried Vias

Layer Thickness	Class 1	Class 2	Class 3
<0.25 mm [<0.00984 in]	0.10 mm [0.00393 in]	0.10 mm [0.00393 in]	0.15 mm [0.00591 in]
0.25 - 0.5 mm [0.020 in]	0.15 mm [0.00591 in]	0.15 mm [0.00591 in]	0.20 mm [0.00787 in]
0.5 mm [0.020 in]	0.15 mm [0.00591 in]	0.20 mm [0.00787 in]	0.25 mm [0.00984 in]

Table 4.2 : Minimum Drilled Hole Size For Blind Vias

Layer Thickness	Class 1	Class 2	Class 3
<0.10 mm [<0.00393 in]	0.10 mm [0.00393 in]	0.10 mm [0.00393 in]	0.2 mm [0.0079 in]
0.10 - 0.25 mm [0.00984 in]	0.15 mm [0.00591 in]	0.20 mm [0.00787 in]	0.3 mm [0.012 in]
0.25 mm [0.00984 in]	0.20 mm [0.00787 in]	0.30 mm [0.0118 in]	0.4 mm [0.016 in]

4.4. Printed Circuit Board Performance Classification

IPC accepts that rigid printed boards are subject to classifications according to intended end item use. Classification pertaining to producibility depends on complexity of design and precision required to produce a certain printed board. There are three general end product classes established. The printed board user is responsible for determination of the class to which product belongs. According to IPC standard Class 1 contains general electronic products. This class includes consumer products for example some computer and computer peripherals. Class 2 contains dedicated service electronic products. In this class there are communication equipments, sophisticated business machines, instruments and machines where high performance and extended life is required and for which uninterrupted service is desired but is not critical. Class 3 contains high reliability electronic products. This class contains the equipment used for commercial and military products where continued performance is most critical. Equipment downtime is intolerable, and must function when required such as life support items [26].

4.5. Methodology

Bisens which is a medical device, has been improved, and existing and developing PCB technologies have been examined. The most optimum solution has been adapted to the mainboard. Aim was to reduce overall dimension of the device with this solution. At this stage, in place of the old type through hole components used in the mainboard, SMD footprint components providing same function have been chosen.

Which class the card must bear according to the IPC standard has been decided before beginning the design. Since an uninterrupted performance is expected from the card as well as instantaneous disruptions are not also of prime importance, performance class of the card was chosen to be Class 2. Circuit design in the study has been carried out by means of Altium Designer software.

Components have been placed on only one surface of the old version motherboard though it was designed as double-sided, and this caused dimensions of the device to be extremely big. In the new version, components are installed on both top and bottom layers. Samtec's SFM/TFM series connectors with SMD footprints which can be installed onto the surface have been used instead of female connectors. Footprints of these connectors have been redefined.

Rigid flex cards have been designed instead of cables outgoing for screen and other peripheral devices. This solution has saved labor cost of cabling works but increased overall cost of the device since flexible cards were expensive.

In order to save much more space, blind via has been chosen as via type. Stackup configuration has been designed so as to ensure impedances to be required by the circuit. ICD Stackup Planner software and Altium Designer has been employed in realizing this design. Total pcb thickness is 1.5mm. 12 layer with blind vias method performed. Stackups are shown in the Figure 4.7 and Figure 4.8.

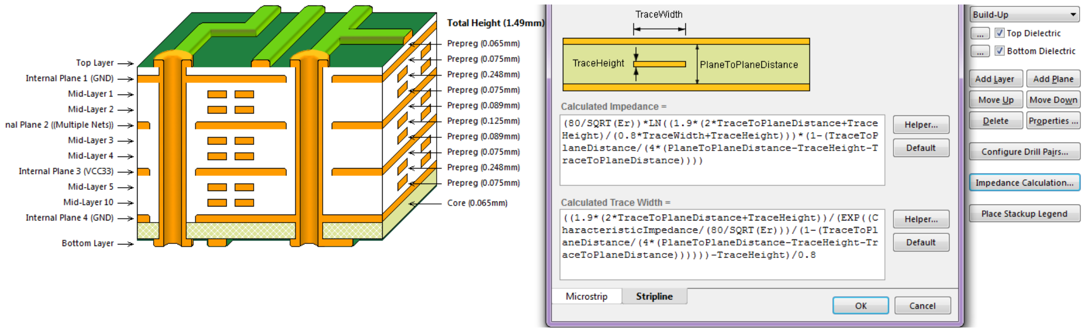

Figure 4.7 : Planning stackup and impedance measurement with Altium designer software

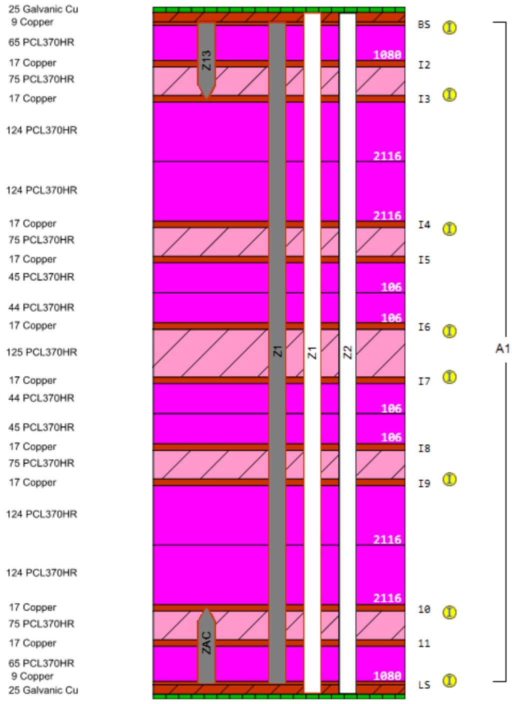

Figure 4.8 : Production stackup with blind and through via types

Minimum through hole diameters were chosen as 0.15 mm taking process of copper plating and maximum current to be carried into consideration. Diameters of the vias for the power section have been changed to 0.3 mm.

3-W rule has been applied between lines running parallel in order to minimize crosstalk. Single stripline topology has been chosen mostly, and in this way, EMI became minimum. For EMI precaution, no signals have been carried from top and bottom layers. At the center of the board dual stripline is performed because of the symmetry.

Reference planes of critical signals have been taken into consideration, and critical signals have been carried through inner layers.

Thermal vias and broad copper surfaces called polygon have been used in order to prevent heating at that region, taking maximum currents driven by motor drives into consideration.

Minimum conductor width is calculated at 20^{o}C. Pcb's maximum current will be 3 A and has 1 oz copper. According to IPC 2152, 3A requires 125 sq-mils cross sectional area. 125 sq-mils cross sectional area requires minimum 0.2 inches width. This is equal to 5.08 mm. This means maximum current region will be minimum 5.08 mm width.

For the processor BGA package was used in the new design whilst LQFP package had been employed in the old design. Use of BGA package has provided 4 times more advantage in terms of usage area, but increased the cost of card.

Because of using BGA package, there is an extra recommendation for vias between bga pads. Filling vias with solder mask or another filling materials is important issue to protect vias. Filling phase provides to put the vias very close to the component mounting pads and protect vias from any chemical leakage during the assembly process. We use filled and covered via which is renamed Type VI according to IPC4761 in our pcb design [19].

Mainboard consists of 8 signal layers and 4 power planes. Copper thickness is 17 micron on the inner layers and 34 microns on the outer layers. FR4, prepregs and copper thicknesses are shown in Figure 4.8.

When designing a pcb, designers think about that electrically long traces. Maximum signal edge is 2 ns in our system. Nearly 18 cm is a maximum trace length boundary on microstrip lines and 14 cm is a maximum trace length boundary on stripline traces. Our pcbs dimensions is 49 mm x 95 mm. Because of the small pcb, signal integrity simulation is unnecessary for this situation.

Top layer, which is named primary side according to IPC standard, has component's footprints. CPU, wifi and usb modules, peripheral connectors, motor drivers are placed on the top layer side. Top layer of the mainboard is shown in Figure 4.9.

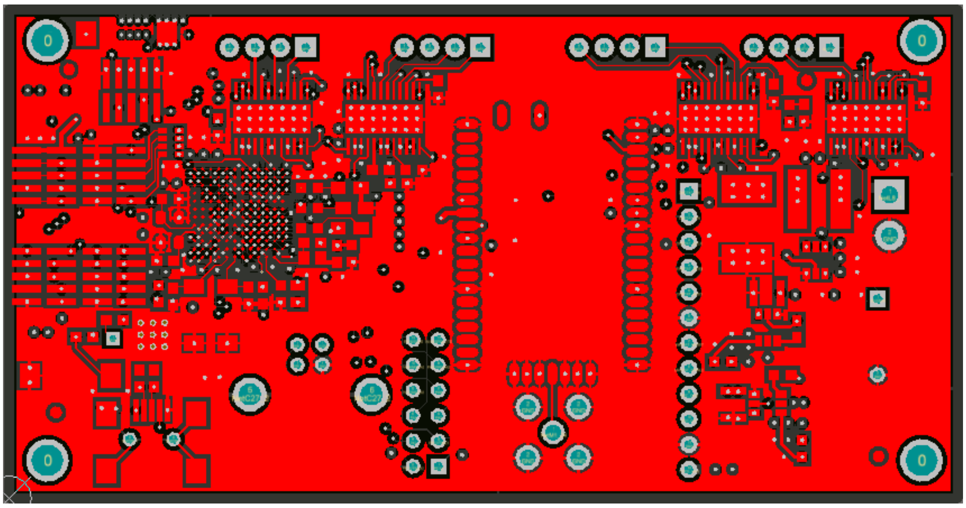

Figure 4.9 : Top layer of the mainboard

Bottom layer, which is named secondary side according to IPC standard, has component footprints like top layer. This layer includes some active and passive smd components. Bottom layer of the mainboard is shown in the Figure 4.10. There is an extra ground polygon. This polygon's clearence is 0.5 mm.

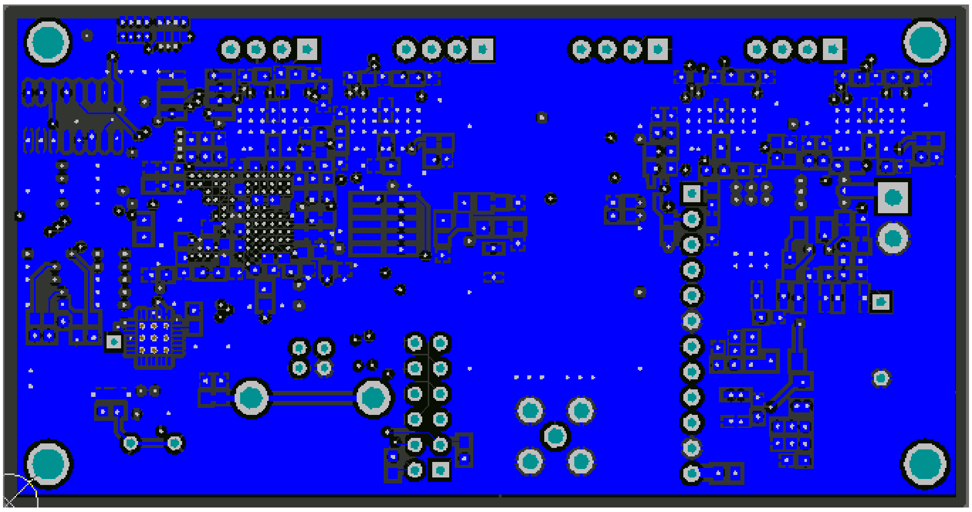

Figure 4.10 : Bottom layer of the mainboard

Interplane 1 is shown in the Figure 4.11. Interplane 1 layer has only ground net. This is the reference plane.

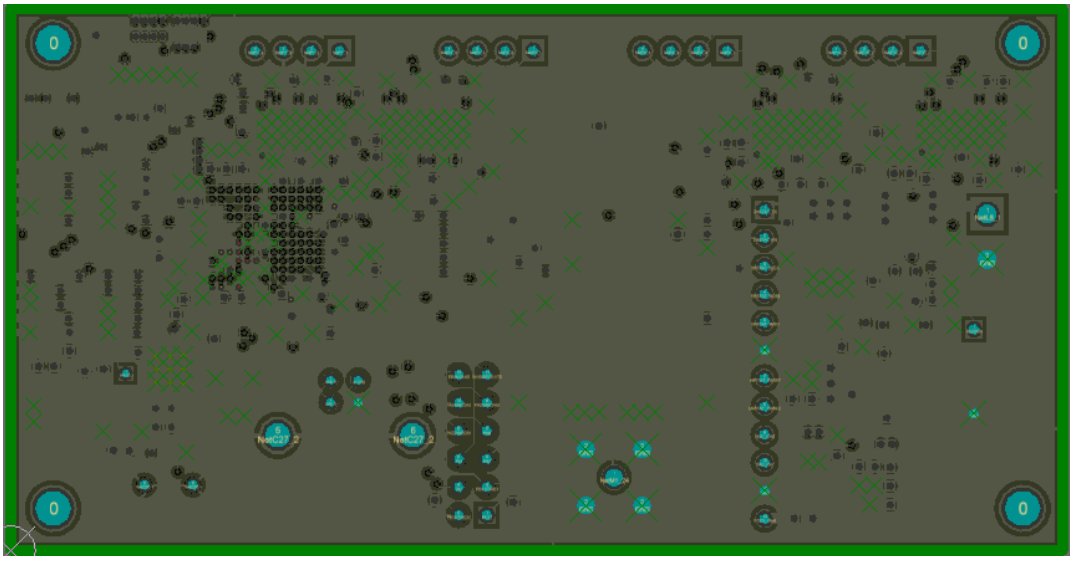

Figure 4.11 : Interplane layer 1

Mid-layer 1 is shown in the Figure 4.12. This layer has signal nets, and usb differential signals which have 100 ohm differential impedance.

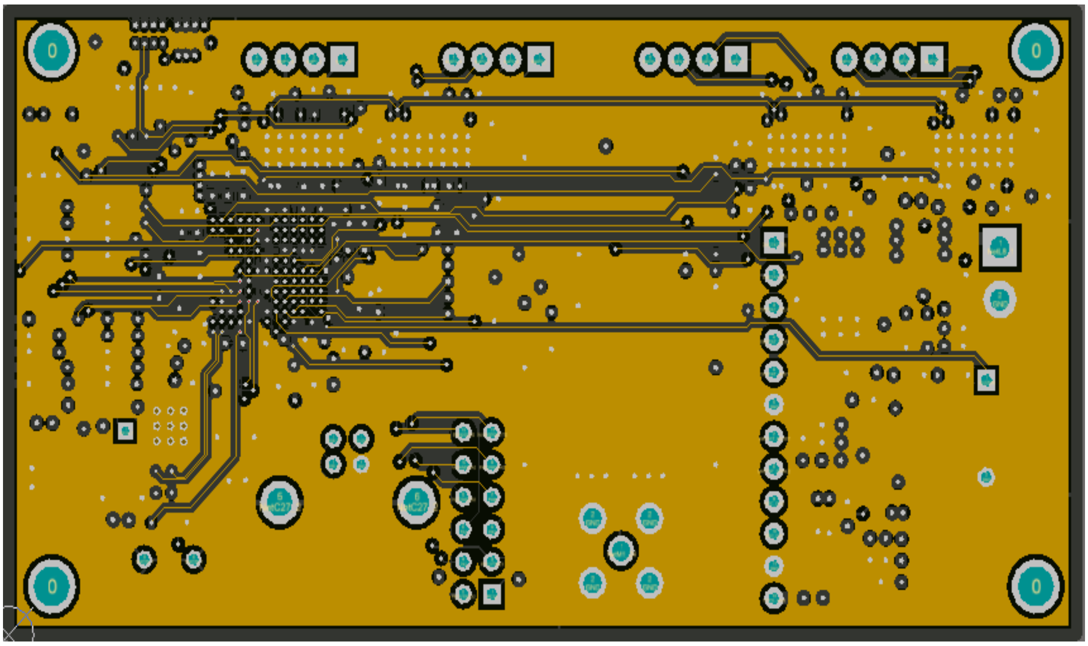

Figure 4.12 : Midlayer1

Mid-layer 2 is shown in the Figure 4.13. This layer has 5 Volt solid polygon.

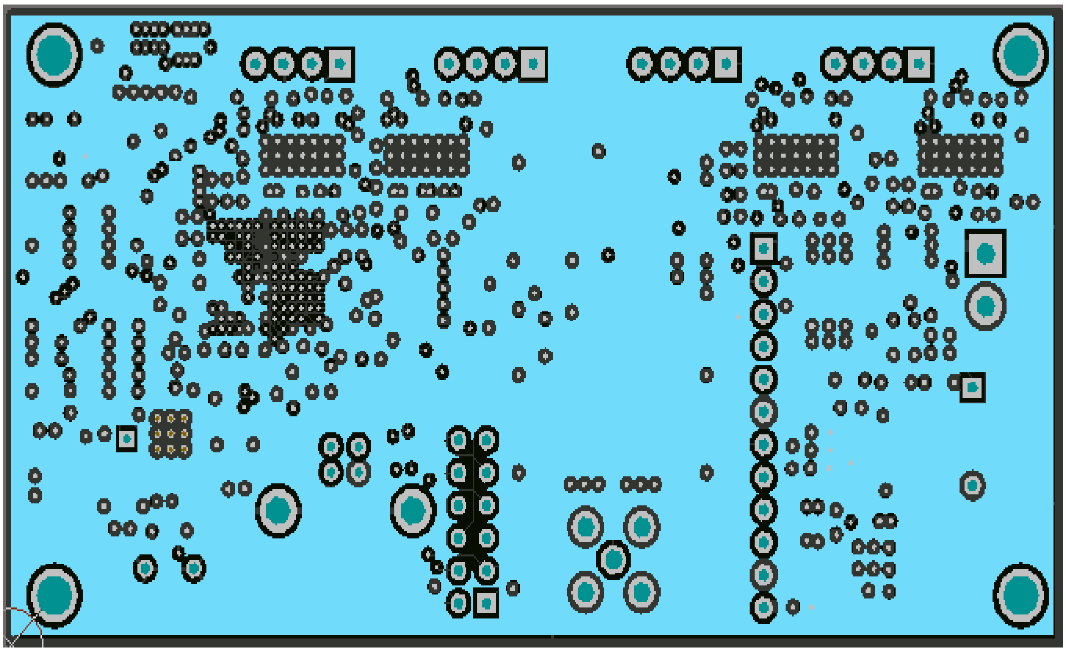

Figure 4.13 : Midlayer 2

Interplane 2 is shown in the Figure 4.14. It contains Vddout Vmot and Vin voltages. This layer has 3 different voltage levels for motor drivers, cpu and system's input voltage.

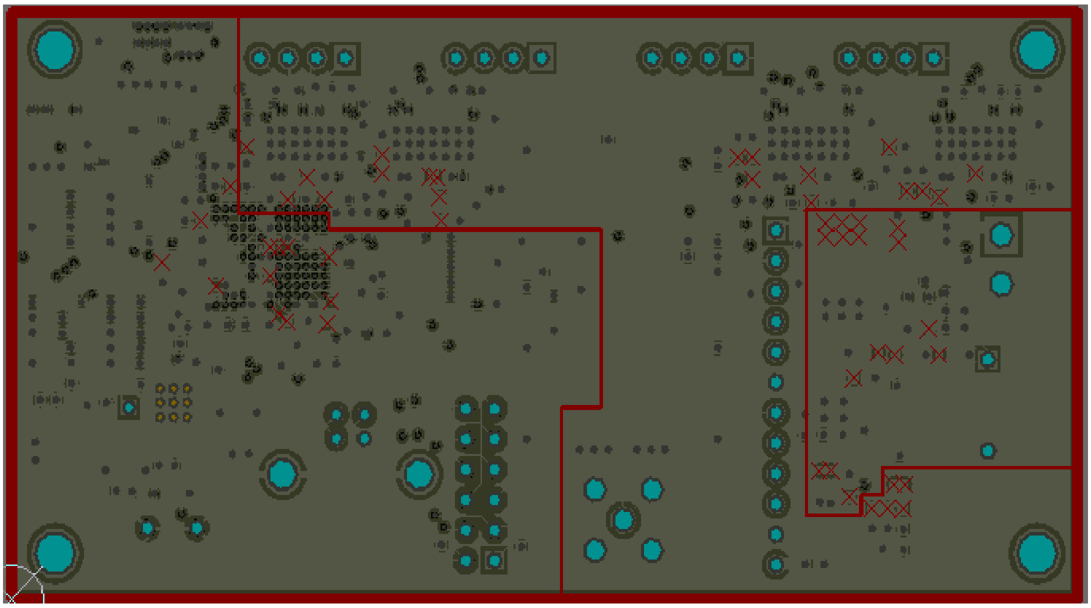

Figure 4.14 : Interplane Layer 2

Midlayers 3-4-5-6 are shown in the Figure 4.15. These layers are other signal layers which is required to break out the Cpu's BGA package.

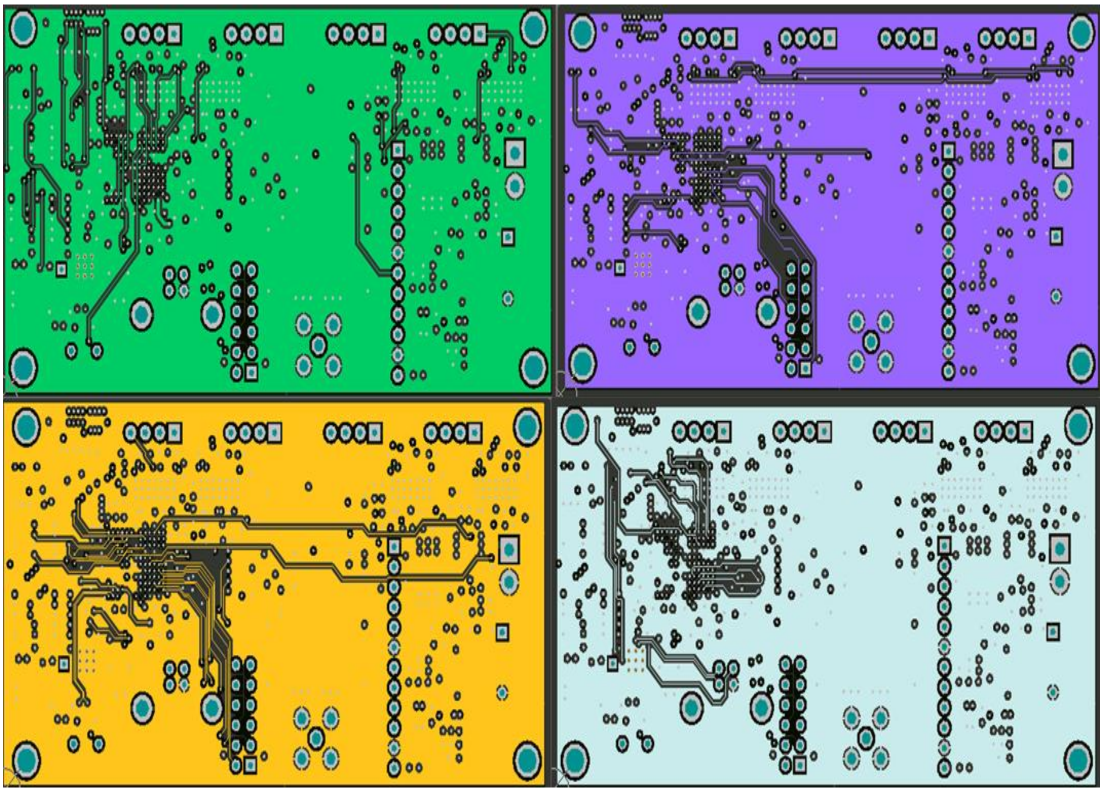

Figure 4.15 : Midlayer 3-4-5-6

Interplane 3 and 4 are shown in the Figure 4.16. Interplane 3 is 3.3 voltage plane. Interplane 4 is a reference ground plane.

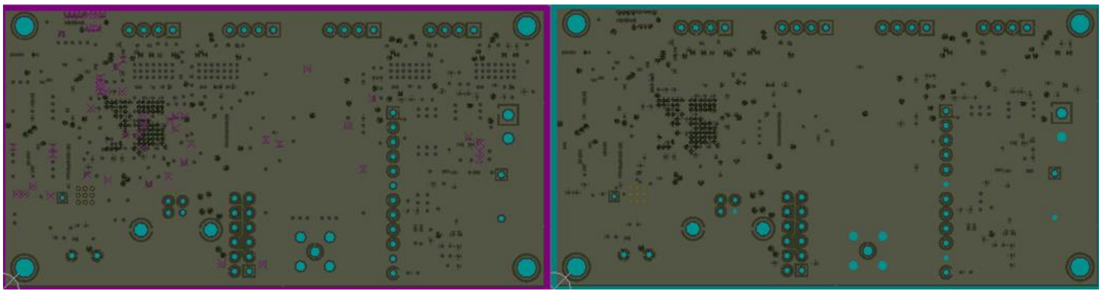

Figure 4.16 : Interplane layers 3 and 4

Mainboard of Bisens device has two layers and a dimensions of 101mm x 165mm on the previous version. In this thesis, design has been reviewed and size of the card has been reduced in order to be 49mm x 95mm applying new technologies to the PCB. As can be seen from the Figure 4.17 smaller one is the new version of the mainboard, bigger one is the previous version of the motherboard.

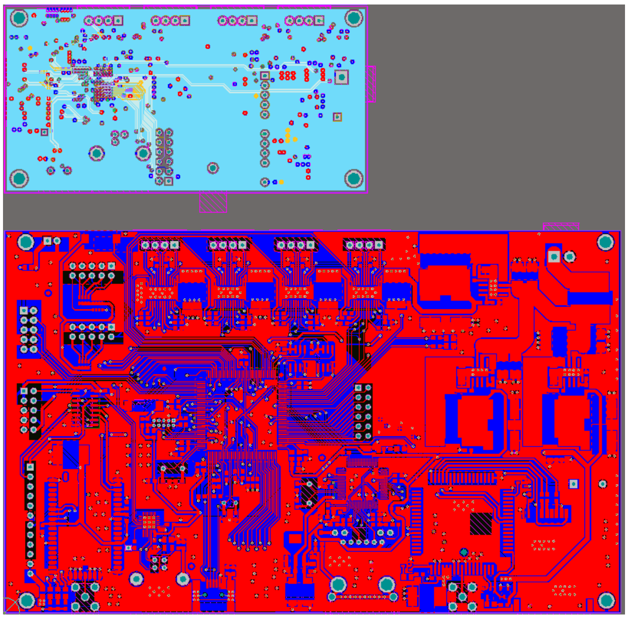

Figure 4.17 : Mainboard of Bisens v1 and v2

Figure 4.18 shows 3-d view of the mainboard. Every component's 3-d step file has been imported its own footprints.

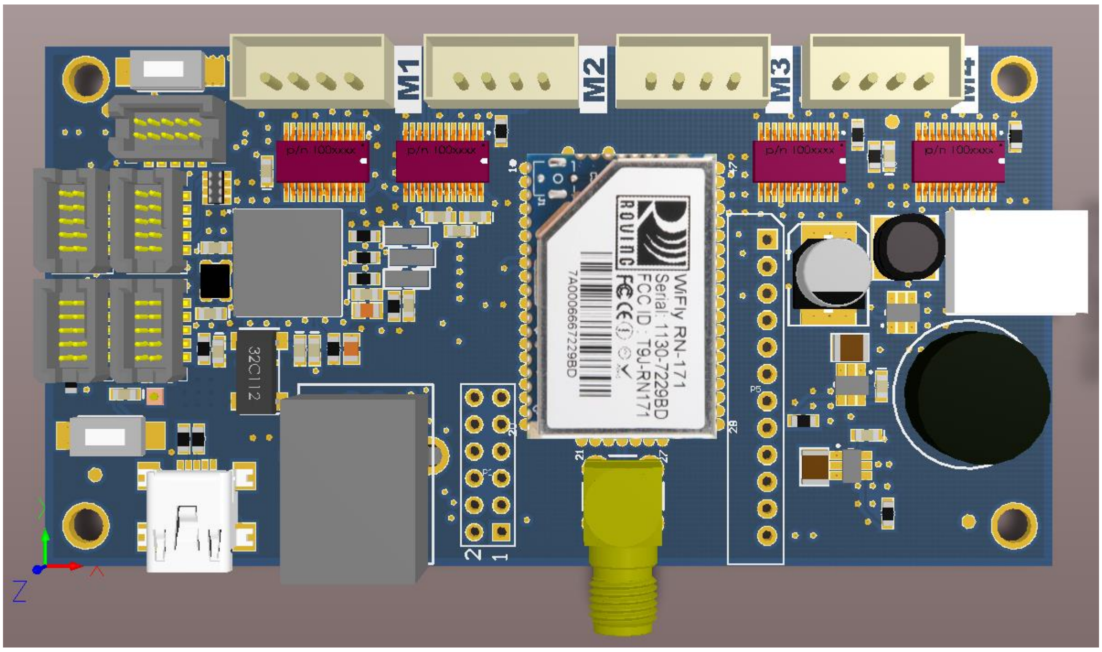

Figure 4.18 : 3D View of The Mainboard

4.6. Summary

In this Chapter, 12 layer pcb with blind vias was designed. Nearly four times smaller mainboard was obtained than the previous version.

5. CONCLUSION

In this dissertation, we designed new version of the mainboard with Altium designer software. We arranged new stack up configuration according to circuit impedance requirements. Single 50 ohm, differential 100 ohm impedance lines were calculated and decided to line thicknesses, board thickness and prepreg thicknesses.

We considered EMI issue and used stripline topology. We calculated minimum trace conductor widths at 20^{o}C according to IPC2152.

We selected 0.1 mm via diameters for signal nets and 0.3 mm via diameters for power vias. We simulated via diameters with Saturn Pcb Software considering maximum current capacity rule. We applied 3-w clearance rule to prevent crosstalk.

We use filled and covered via which is renamed Type VI according to IPC4761 in our pcb design.

Although previous version of the mainboard was produced before, it was too large in dimensions. Mainboard of Bisens, which is PSA test device, was double sided board. Dimensions of the mainboard was 101mm x 165mm on the previous version. Because of the big printed circuit board, prototype product was too big as well.

In this thesis, design has been reviewed and size of the card has been reduced in order to be 49mm x 95mm. Because of the small printed circuit design signal integrity simulation is unnecessary for this situation.

Because of the miniaturization Bisens can be hand carry diagnostic device. After the mainboard miniaturization, we will select new and smaller lcd, smaller step motors and syringes.

As a result of the miniaturization; peripheral components, device mechanical drawing will be smaller than before.

Tests are typically performed by central laboratories which utilize expensive laboratory equipment such as Gas Chromatography (GC), GC-Mass Spectrometry

(MS), Liquid Chromatography (LC)-MS or Enzyme-linked immunosorbent assay (ELISA). These processes, which involve high costs, include some requirements such as sample transportation, increased waiting time and trained staff. In this context; Bisens, which is a PSA test device, is a portable detection system. It is easily operated, cheap and capable of quick processing. This device enables on-site testing, uses low sample. Small scale heath care centers without laboratory infrastructure takes the blood sample from patient and sends it to the more competent institutions for test. At this stage, sometimes results of the test may be mixed or samples may be lost, and sometimes one has to wait results for days. Bisens is a PSA test device that quickly provides results within a short time such as 15 minutes for the places or community health care centers without laboratory infrastructure and away from the city center.

REFERENCES

[1] Olcer, Z., Esen, E., Muhammad, T., Ersoy, A., Budak, S., Uludag, Y., (2014). Fast and sensitive detection of mycotoxinsin wheat using microfluidics based Real-time Electrochemical Profiling. Biosensors and Bioelectronics, vol.62, pp.163.

[2] Uludag,Y., Köktürk, G., (In Press) Cancer biomarker detection in serum samples using nanoparticle amplification and lab-on-a-chip based amperometric biosensor**.**

[3] Uludag,Y., (In Press) Critical stages of a biodetection platform development from sensor chip fabrication to surface chemistry and assay development.

[4] Zengin, F., Türetken, B., Akkaya, E., San, S.E., (2010). Ekit (implant) Ultra Wideband Antenna Design for Implant Applications. IEEE, vol.978-no.1 pp. 4244-6382-4/10.

[5] Zengin, F., Akkaya, E., Türetken, B., San, S.E., (2011). Design and Realization of Ultra Wide-Band Implant Antenna for Biotelemetry Systems. IEEE, vol.978, no.1- pp.4244-6051-9/11.

[6] Bradford, B., Krautschneider, W., Schröder, D., (2010). Wireless Power and Data Transmission for a Pressure Sensing Medical Implant. BMT, OCTOBER 2010.

[7] Uludag, Y., Olcer, Z., Sagiroglu, M.S., (2014). Design and characterisation of a thin-film electrode array with shared reference/counter electrodes for electrochemical detection. Biosensors andBioelectronics, vol.57 pp.85–90.

[8] Uludag, Y., Tothill, I. E., (2012). Cancer Biomarker Detection in Serum Samples Using Surface Plasmon Resonance and Quartz Crystal Microbalance Sensors with Nanoparticle Signal Amplification. American Chemical Society, vol 84, pp. 5898−5904.

[9] Karacolak, T., Cooper, R., Topsakal, E., (2009). Electrical Properties of Rat Skin and Design of Implantable Antennas for Medical Wireless Telemetry. IEEE TRANSACTIONS ON ANTENNAS AND PROPAGATION, VOL. 57, NO. 9, pp. 2806-2812.

[10] Karacolak, T., Cooper, R., Butler, J., Fisher, S., Topsakal, E., Senior Member, IEEE (2010). In Vivo Verification of Implantable Antennas Using Rats as Model Animals. IEEE ANTENNAS AND WIRELESS PROPAGATION LETTERS, VOL. 9, pp. 334-337.

[11] Karacolak, T., Member, IEEE, Cooper, R., Ünlü, E.S., Topsakal, E., Senior Member, IEEE (2012). Dielectric Properties of Porcine Skin Tissue and In Vivo Testing of Implantable Antennas Using Pigs as Model Animals. IEEE ANTENNAS AND WIRELESS PROPAGATION LETTERS, VOL. 11, pp. 1686-1689.

[12] Mark I. Montrose (2004). EMC and the Printed Circuit Board Design, Theory, and Layout Made Simple. IEEE Electromagnetic Compatibility Society.

[13] Bruce R. Archambeault (2002). Pcb Design for Real World EMI Control.Kluwer Academic Publishers.

[14] Stephen C. Thierauf (2004). High Speed Circuit Board Integrity. Artech House.

[15] E.P. Diamandis,Trends in Endocrinology and Metabolism,9 (1998) 310.

[16] M.K.Brawer,P.H.Lange, World Journal of Urology,7 (1989) 7.

[17] Gaßmann, S., Ibendorf, I., Pagel L., (2007). Realization of a flow injection analysis in PCB technology. Sensors and Actuators A, vol.133 pp.231–235.

[18] IPC-4761 (2006). Generic standard on printed board design, IPC Association Connecting Electronics Industries, Northbrook, IL.

[19] Kiourti, A. Konstantinos, A. Psathas, J. R. Costa, C. A. and Konstantina S. N., "Dual-Band Implantable Antennas for Medical Telemetry: A Fast Design Methodology and Validation for Intra-Cranial Pressure Monitoring", Progress In Electromagnetics Research, 141, 161-183, 2013.

[20] Sondaş, A., Uçar, M.H.B., Uras, E., (2014). ÇİFT-BANTLI MİKROŞERİT İMPLANT ANTEN TASARIMI A DUAL-BAND IMPLANTABLE ANTENNA DESIGN. IEEE 22nd Signal Processing and Communications Applications Conference.

[21] Topsakal,E. " Antennas for Medical Applications : Ongoing Research and Future Challenges", International Conference on Electromagnetics in Advanced Applications ICEAA '09,890-893,2009.

[22] Zengin,F., Türetken, B., Akkaya,E. and San,S.E., "Ekit (implant) Uygulamaları İçin Geniş Bantlı Anten Tasarımı", 15th National Biomedical Engineering Meeting, 1-4,2010

[23] P. Sarkar, P.S. Pal, D. Ghosh, S.J. Setford, I.E. Tothill, International Journal of Pharmaceutics, 238 (2002) 1.

[24] G.A.J. Besselink, R.P.H. Kooyman, P.J.H.J. van Os, G.H.M. Engbers, R.B.M. Schasfoort, Analytical Biochemistry, 333 (2004) 165.

[25] IPC-2221 B (2012). Generic standard on printed board design, IPC Association Connecting Electronics Industries, Northbrook, IL.

[26] IPC-4101 C (2003). Generic standard on printed board design, IPC Association Connecting Electronics Industries, Northbrook, IL.

[27] Baker, G., (2014). Wearable Electronics: The Shape-Shifting Future of Medical Devices, The PCB Magazine, Medical Electronics, Vol.4 pp 34-36.

[28] Schulze, D., Tölke, R., (2014). Noble Metal Pcb Manufacturing for Direct Implants, The PCB Magazine, Vol.4 pp 12-20.

[29] Petherbridge, K.,Evans P., Harrison, D., (2005). The Origins and Evolution of the Pcb.

.

CURRICULUM VITAE

Name Surname: Atike DEMİRALP

Place and Date of Birth: Kocaeli, 1988

E-Mail: ademiralp@itu.edu.tr, atikedemiralp88@gmail.com

B.Sc.: Electrical and Electronic Engineering, Erciyes University,

Work Experience : TÜBİTAK-BİLGEM (2011- Working)