PAK-TAB – DESIGN AND DEVELOPMENT OF AN EMBEDDED HARDWARE PLATFORM FOR ANDROID APPLICATIONS

By

NC Zeeshan Zafar

PC Sidra Zakreen

PC Shaher Bano Kazmi

PC Muhammad Bilal Javaid

Project Supervisor

Lt. Col. Dr. Adnan Rashdi

Submitted to the Faculty of Electrical Engineering, Military College of Signals National University of Sciences and Technology, Rawalpindi in partial fulfillment for the requirements of a B.E. Degree in Electrical (Telecom) Engineering

JUNE 2014

CERTIFICATE OF CORRECTNESS AND APPROVAL

Certified that the work contained in this thesis titled "Pak-Tab - Design and Development of an Embedded Hardware Platform for Android Applications", carried out by Zeeshan Zafar, Sidra Zakreen, Shaherbano Kazmi and Bilal Javaid under the supervision of Lt.Col. Dr. Adnan Rashdi for partial fulfillment of Degree of Bachelors of Telecommunication Engineering, is correct and approved.

Approved By

Lt.Col. Dr. Adnan Rashdi

EE Department

Military College of Signals, NUST

Dated: June 2014

ABSTRACT

PAK-TAB– DESIGN AND DEVELOPMENT OF AN EMBEDDED HARDWARE PLATFORM FOR ANDROID APPLICATIONS

This project carried out the design and development of an embedded hardware circuit board for android applications using Samsung S5PV210 Processor. The designed hardware possesses GPS, Wi-Fi, capacitive screen, G-sensor and will have ports/interfaces like USB, HDMI, SD-card and Ethernet. The embedded board may also be used with WinCE and Linux. Altogether the embedded hardware board constitutes a tablet that offers various useful functions.

The steps that were involved in making a tablet included the designing of schematics and PCB layout, debugging of that PCB layout and then its fabrication. Furthermore, Device-Drivers and Android applications were installed onto the tablet.

The schematics were made using Altium Software using the knowledge of linking the sub-sheets to the main sheet. The PCB layout was designed and manually routed according to the Schematics provided, so that minimum errors are present in the PCB. Further Debugging of the PCB layout was done to ensure that there were no mistakes. When the circuit was verified, it was sent for Fabrication. Methods of installation of Device drivers and booting techniques were conducted on the kit. Furthermore, a study of android basic applications for verification of tablets was conducted.

DECLARATION

No portion of work presented in this thesis has been submitted in support of another award or qualification either in this institution or anywhere else.

In The Name Of Allah, the Most Benevolent, the Most Merciful.

DEDICATION

Dedicated to my family who have been my emotional anchors through not only the uncertainties and challenges of graduate school, but my entire life.

-Zeeshan

I would like to express my gratitude and deep appreciation to my family for their constant encouragement and support.

-Bilal

This thesis is dedicated to my loving family, my hardworking group-mates and my outrageously energetic friends without whose kind words and support, I could not have achieved this milestone in my life.

-Sidra

I dedicate this thesis to my supporting parents, brother and fiancé for their unconditional support, and inspiration. Furthermore I would like to dedicate this to all my group members, friends and teachers for their irrevocable and persistent faith in all of us.

- Shaher bano

ACKNOWLEDGEMENTS

All praises for Allah Almighty who gave us the strength to accomplish this mighty task despite abundant hitches and adversities on our way.

We are grateful to our Supervisor, Lt Col Dr. Adnan Rashdi for his constant guidance and encouragement. The project wouldn't have been possible without his guidance, unfaltering patience and invaluable efforts.

We would like to express a special word of thanks to Mr. Arsalan Munir and Miss Amna Hasan for their invaluable help and guidance.

We are also grateful to our parents, family and friends for their admirable support. Lastly, we acknowledge the Lab attendants of SDR Lab who rendered their help during the period of our project work.

Table of Contents

List Of Tables

LIST OF FIGURES

LIST OF ABBREVIATIONS

AC	Alternating Current
App	Application
ARM	Advanced RISC Machines
COM (port)	Communication Port.
DC	Direct Current
DMM	Digital Multi Meter
DTE	Data Terminal Equipment
LCD	Liquid Crystal Display
MHz	Mega Hertz
NAND	Not And
PCB	Printed Circuit Board
SD card	Secure Digital Card
USB	Universal Serial Bus
UTP	Unshielded twisted pair
Wi-Fi	Wireless Fidelity
PHY	Physical Layer Device

Chapter 1: Introduction

1.1 Project Motivation

1.2 Project Description & Salient Features

1.3 Scope of Work

1.4 Objectives

1.5 Project Specifications

1.6 Deliverable

1.7 Organization of thesis

Introduction

1.1 Project Motivation

There has long been a need for a computer that people could take with them so that they could do their work outside of the office. There are of course other options for this but laptops are not really as portable as most people need them to be and smart phones don't have the capability to do all of the things that people need done. A tablet on the other hand is ideal since it is small enough to be easily portable and it can handle most of the tasks that people need to be able to do. Pak-tab is designed such that it can be used for specialized purposes such as high speed video processing, tracking, surveillance DTE tool and also be able to handle multiple systems connected to the board.

Pak-tab runs on Google's Android operating system (OS). This OS is called open-source, meaning it can be customized and optimized by tablet manufacturers and users to suit their own specifications. This is one of its main features, which set it apart from Apple's iOS. Pak-tab users can alter the appearance by enabling widgets to display information like emails and calendar appointments on the home screen. This makes it more complex than iOS and much more flexible. New applications can be downloaded from Google Play store, (previously called Android Market) which has a good range of applications available. Different tablets run different versions of Android. They all have a dessert-themed name and a version number. Pak-tab runs on Android 4.0 Ice-cream Sandwich.

We have chosen this project as it offers us a chance for us to learn a myriad of engineering skills related to both hardware and software.

This project requires understanding of multi-layered PCB designing. Altium Software is to be used for the tablet circuit simulations as the overall circuitry of the tablet is very large and it has features such as auto routing for testing and debugging that makes PCB designing a simpler task

1.2 Project Description & Salient Features

The project aims at development of an embedded systems board capable of high speed processing and supporting WinCE, Android and Linux OS so that it can be used as a general purpose tablet.

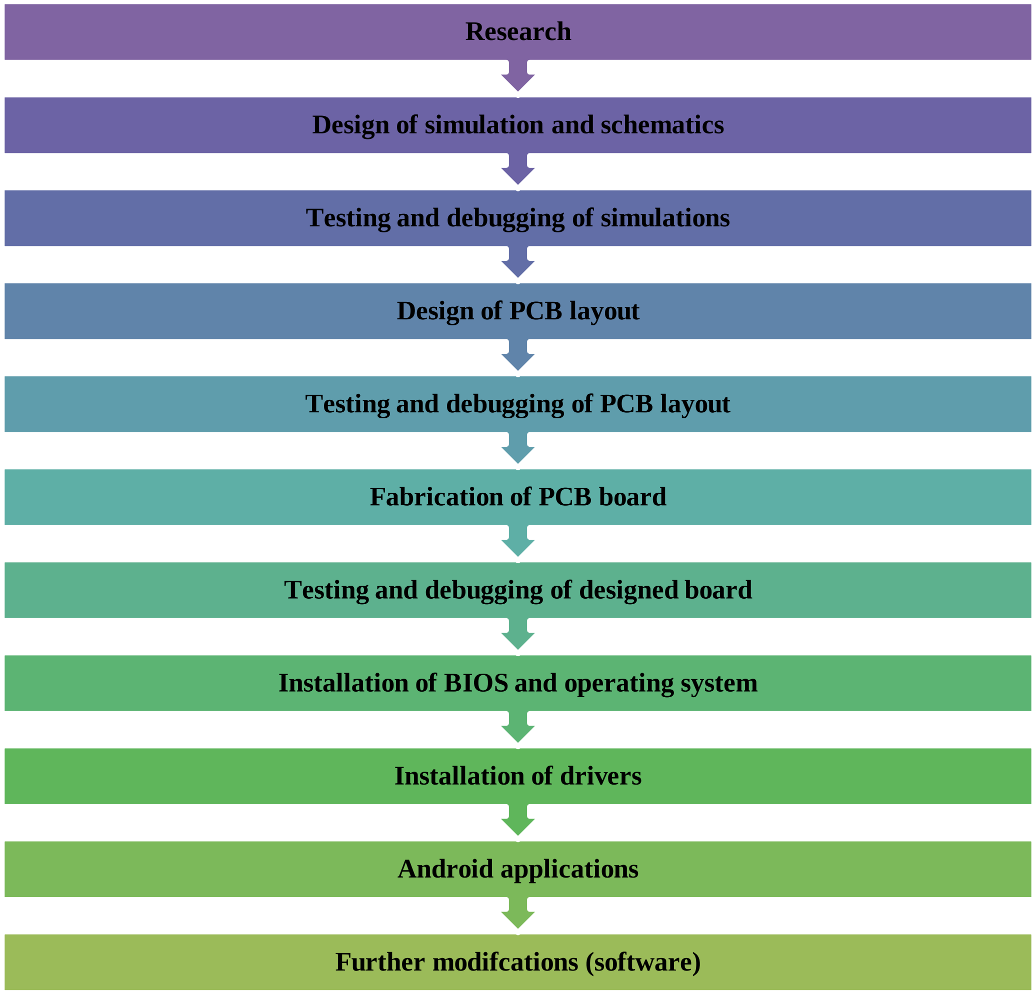

Table 1 Flowchart of the Project

1.3 Scope of Work

Design of a high speed tablet with inherent low cost in order to attract a large percentage of potential customers.

The project will involve:

a. Making a detailed schematic on Altium software

b. Verification of schematic of the tablet

c. Transferring the schematic onto the PCB

d. Creating footprints of the associated components

e. Determination of suitable routing that would lead to the appropriate layering

f. Ultimately, resulting in the complete design of the printed circuit board

g. The next step would be the testing and debugging of the circuit simulations

h. After the testing and debugging, the circuit would be sent for fabrication

i. Installation of BIOS and operating system on the embedded board

j. Writing drivers for modules attached

k. Furthermore, selected android applications would be developed to be incorporated in the tablet

This project was chosen as it offers a chance for us to learn a myriad of engineering skills related to both hardware and software. It requires the understanding of multi-layered PCB designing, learning computing languages such as java that can be used for android programming. Altium Software is to be used for the tablet circuit simulations as the overall circuitry of the tablet is very large and it has features such as auto routing for testing and debugging that makes PCB designing a simpler task.

1.4 Objectives

Designing of a carrier board using a high performance Cortex A-8 1 GHz processor (S5PV210). The tablet is specifically being designed to run H.264 protocol to aid efficient video compression.

1.5 Project Specifications

Design and development of embedded circuit board using Samsung S5PV210, 1GHz Processor. The designed hardware will have GPS, Wi-Fi, capacitive screen, G-sensor and will have ports/interfaces like USB, UART, HDMI, SD-card and

Figure 1 Tablet-PC

Ethernet. The board will be capable of handling Android, WinCE and Linux operating systems.

1.6 Deliverable

The Deliverable of this project is a carrier board that uses a high performance Cortex A-8 1 GHz processor (S5PV210) that is able to run android applications

1.7 Organization of thesis

Chapter 1 It described the project's motivation, its scope and objectives. It also mentions the project specifications and deliverables

Chapter 2 Provides the literature review of the project, including the research related to multi-layered PCB designing required in making Tablet PCs

Chapter 3 Gives the design and development details of the project comprising of the method to create circuit schematics and PCB layouts that are made using the Altium Software

Chapter 4 States and explains the circuit diagram of all the modules of Pak-Tab and their corresponding PCB layouts

Chapter 5 Explains the method for installation of OS on the carrier board

Chapter 6 Analyses and evaluates the performance of Smart Café on this basis of results of various tests carried out. Basing on the performance benefits of having a Smart Café are also discussed

Chapter 7 Discusses some future enhancements and finally concludes the chapter

At the end of the document are the appendixes giving additional relevant details about Smart Café, a user manual and few application codes in Android and PHP. The list of reference material used while preparing this document follows the Appendixes.

Chapter 2: Literature Review

2.1 Stages of Development of Pak-Tab

2.2 Research

2.2.1 Multi-Layered PCB designing

2.2.1.1 Layer Stack Up

2.2.1.2 Trace

2.2.1.3 Plane

2.2.1.4 Silkscreen (also called Screen Print)

2.3 S5PV210 Processor

2.4 Part Research & Selection

2.5 Schematic Capture

2.5.1 Simulation

2.6 Board Layout

2.7 Verification & Validation

2.8 Software Research

2.1 Stages of Development of Pak-Tab

We can split the hardware development of Pak-Tab into four steps: Research, Schematic Capture and Simulation, Board Layout, and Verification and Validation. [5]

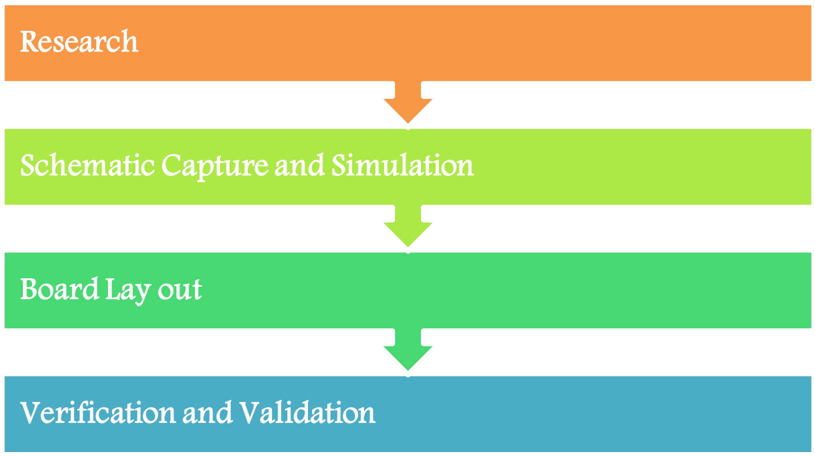

Figure2. Stages of Development of Pak-Tab

2.2 Research:

The research was divided into three distinct parts. First, we had to do research on the Multi layered PCB designing. Secondly, we had to research the processor parameters. Next, based on the processor parameters we had to do part research and selection. These steps are further described below.

2.2.1 Multi-Layered PCB designing:

A PCB can be called as a layered structure, usually with several copper and insulating layers. The main portion is an insulated material usually made from fiber glass, and epoxy. Pak-tab is a 4- layered PCB that can be made up of two double-layered PCBs laminated (sandwiched) together with a core material in between.

2.2.1.1 Layer Stack Up: Layer stack up can be defined as the copper organization of multiple layer PCBs with the intent of having specific signal and ground planes on certain layers for routing convenience and electromagnetic shielding purpose [8]. Pak-tab has the following layer structure, in which the top and bottom layers are reserved for signal routing and the inner layers are reserved for ground and power planes:

- Copper Top
- Inner 1
- Inner 2
- Copper Bottom

2.2.1.2 Trace: A portion of the copper foil that is remaining on the PCB after the etching process for a signal connection (net) from point A to point B.

2.2.1.3 Plane: A plane is defined as a large portion of the copper foil that is remaining on the PCB for a signal connection that attaches to many components [8]. This is a layer where little copper is etched away. In Pak-tab power and ground signals are connected to a plane, since the power and ground needs to route to many components on the PCB. Since a plane has a lot of copper, and the plane covers a large area, traces are routed to the plane with an effective lower resistance from a connection point to the plane. This will create smaller voltage drops (versus placing discrete traces) when the components conduct power and ground current

2.2.1.4 Silkscreen (also called Screen Print): This is the final layer applied over the top solder mask and also over the bottom solder mask (if required) to display part outline and reference designator information on the board. Besides displaying part outline and reference designator information, the silkscreen can display orientation information (such as anode/cathode direction) and any other symbol or textual information the user wants to be displayed on the board.

2.3 S5PV210 Processor

Pak-Tab is built around Cortex A8, arm architecture version 7, S5pv210 processor that has a clocking speed of 1GHz. It requires a voltage of 5V and a current of 2A to operate. The processor is broken up into 13 stages meaning that a single instruction would have to go through 13 stages to finish executing. This is beneficial over a single stage processor because an instruction goes through each stage quicker and once an instruction completes a stage, a new instruction can immediately enter that stage.

CENTRAL PROCESSING UNIT		
CPU Clock Speed: 1,000 MHZ	CPU Cores: 1 CORE(S)	Number of CPU Pipeline Stages: 13 STAGES
Primary CPU	ARM Cortex-A8 processor core	
L1 Instruction Cache	32 KB/Core	
L1 Data Cache	32 KB/Core	
L2 Cache	512 KB/Core	
Word Length	32 bit	
Data Bus	64 bit	

Table 2 Characteristics of S5PV210 Processor

This processor is based on the Reduced Instruction Set Computing (RISC) design strategy enabling instructions to execute faster, as opposed to the Complex Instruction Set Computing (CISC) design strategy, which is generally slower at executing due to lengthy instructions.

2.4 Part Research & Selection

Before designing, we needed to select components that will make a circuit function. There are a number of semiconductor manufacturers including Texas Instruments[6], Analog Devices National Semiconductor [7] etc. that create devices that meet varying specifications. Our

Processor requirements were that it had a voltage of There are a number of ways to research parts including:

1. Datasheets on semiconductor performance[3]
2. Simulation tools that can be downloaded and used to evaluate components from that manufacturer.

2.5 Schematic Capture

A schematic is a collection of electronic symbols connected together with virtual "wires"[1]. When fabricating a printed circuit board (PCB), a schematic is needed to provide input (a netlist) to the layout and routing tool. Schematic capture involves taking symbolic representations of components (resistors, capacitors, amplifiers, comparators etc.) and wiring them together into a visual diagram which can be easily viewed to understand the functionality of the circuit. The purpose of the schematic is to view the building blocks of the circuit from a theoretical perspective. These schematics were drawn on Altium Software as it handles large circuitry more effectively. The schematic consists of **component symbols** and **wires (also called nets).**

2.5.1 Simulation

Simulation can be used to evaluate the behavior of real world components in a virtual environment, thereby allowing us to do advanced analysis of a design earlier in the design flow[4]. By visualizing the behavior of the circuit at the earliest stage of construction, one can reduce errors and improve performance. We built the schematic of Pak-tab, and then simulated and visualized its characteristics on Altium Software.

2.6 Board Layout

A schematic when completed is transferred to board layout. Each symbol in a schematic is associated with a footprint [2]. A footprint visually represents the physical dimensions of an IC or semiconductor package. The footprint translates the symbol of a component (such as an Operational Amplifier) to an 8 pin rectangular package. During the board layout stage the design is defined as it will eventually look when the tablet is made. This means defining the board outline, placing parts, creating connections between parts in copper, and then finally exporting for fabrication.

2.7 Verification & Validation

Once the board is fabricated, the engineer needs to validate the behavior of the Pak-tab. Once it is validated, it can be handed off to product development and be prepared for manufacturing. Validation requires taking measurements from the board and ensuring that real performance meets the simulated performance (or specifications).

2.8 Software Research

For software research we had to search for tools that could help us download the ROM image of the tablet. A ROM image is a data file that contains information used on a Read Only Memory chip. ROM image is a data folder that contains files and code needed to boot the device up and run Android on it. But this is only part of a ROM. A ROM also contains a GUI (graphical user interface), required and useful applications, support files for those applications and the kernel.

For the Booting process we have used **Fastboot** and **DNW** softwares. They help us in installing the ROM image in the tablet.

Chapter 3: Design & development

3.1 Construction of Schematics

3.1.1 Defining Parameters of designed components

3.1.2 Verification of circuits

3.1.3 Designing Component footprints

3.2 Creating a New PCB layout

3.2.1 Transferring the Design

3.2.2 Setting up the PCB Workspace

3.2.3 PCB Workspace Grids

3.2.4 Configuring the Display of Layers

3.3 PCB Fabrication

3.4 PCB Debugging

3.1 Construction of Schematics

The schematics of different modules to be incorporated with S5PV210 processor have been chosen on the basis of power requirements, their interfacing with the processor and availability. The schematics are designed using Altium designer software. Altium software provides different approaches towards the design of schematics like flat, hierarchical and global approach (sheet - sub sheet approach) in which the processor is placed at the top of hierarchy and different modules can be connected to the main sheet. The modules will be designed individually on the sub-sheets.

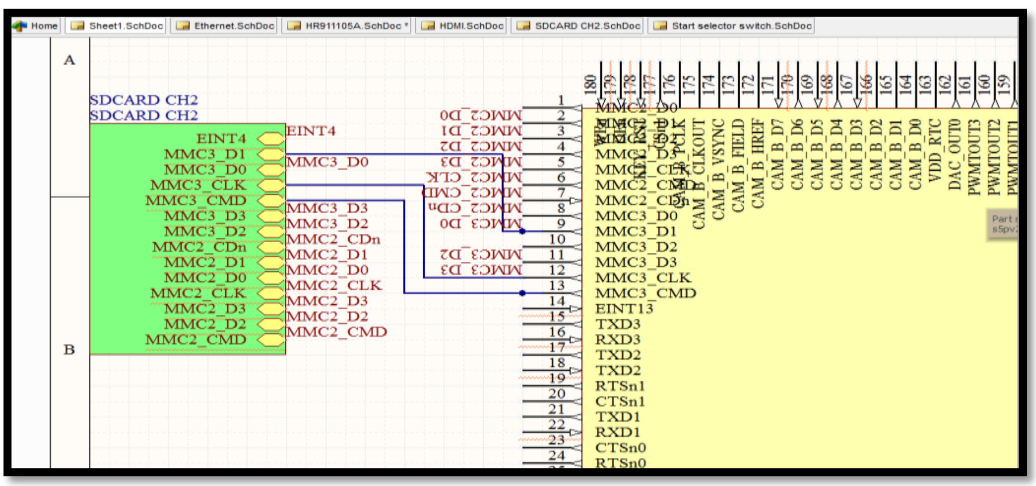

Figure 4 Using Sub-sheets in Altium; white background is of the main sheet while green is for sub sheets

3.1.1 Defining Parameters of designed components:

After the modules are decided and their power requirements and compatibility with the board is ensured, the next step before going to the design of schematics is to ensure that all the used components are available in the market as per desired dimensions and power requirements. This step involves detailed reading of data sheets of each component being used.

The following things are to be kept in mind while looking at the data sheets.

a. **Dimensions:** Height, Width and Length of the component are vital as it affects the overall dimensions of the board.

b. **Power Demand:** The power requirements of the component should match those defined in the design of that module.

c. **Pitch:** Pitch is the distance/ gap between the pins of the IC and components.

d. **Temperature tolerance:** it defines that how much temperature a component can endure and work properly.

The parameters are defined by Double clicking at a component.

For developing this particular project, we added the following parameters:

a. **Category:** It defines the type of the component

b. **Manufacturer/part number:** the part number of the component present in the manufacturer's directory.

c. **Package/case:** the specific PCB footprint package number

d. **Size/dimensions:** the length, height and width of a component.

e. **Supplier part/number:** It defines the part number in the supplier's directory.

f. **Value:** operating value of the components.

3.1.2 Verification of circuits: After the accomplishment of first step, the next step is to verify the designed schematics by checking the various connections and ports.

3.1.3 Designing Component footprints: The third step moves towards the design of a PCB layout. It involves making the footprints of the components used in schematics. The footprints are simulation of copper layout of the components to be placed on the carrier board. They have been designed using the "IPC compliant footprint design wizard" available at the Altium software. ICs are made by using the "line" or "rectangle" button. Pins are added by clicking "Place" and then "Pin". The "Hot End" of each pin must be placed outside the body of the component. After the footprints were finalized they were linked to their respective components.

3.2 Creating a New PCB layout:

Before transferring the design from the Schematic Editor to the PCB Editor, we needed to create the blank PCB with a board outline. We created a new PCB design in Altium Designer by using the PCB Board Wizard, which allows one to choose from industry-standard board outlines as well as create one's own custom board sizes. We designed our own custom based PCB with dimensions 172mm x 122mm. At any stage one can use the Back button to check or modify previous board designs in the wizard.

To create a new PCB using the PCB Wizard, we were to do the following steps:

- Display the Files panel. The default location for this panel is docked on the left side of Altium Designer.

- Create a new PCB by clicking on PCB Board Wizard in the New from Template section at the bottom of the Files panel. The PCB Board Wizard opens with an introduction page.
- On the second page of the wizard one selects the board outline one wishes to use; we selected Custom from the list of board outlines. In the next page we entered the custom board options. We selected Rectangular for the Outline Shape and typed 112mm in the Width and 172mm in the length fields.
- The next page allows one to select the number of layers in the board. We will need 4 layers and 3 power planes.
- For the Via style, we selected Thruhole Vias only.
- Then we were to set up some of the design rules for track width and via sizes that are applied to our board.
- The PCB Board Wizard has now collected all the information it needs to create a new PCB board.
- The PCB Editor will now display a new PCB file. The PCB document displays with a default sized white sheet and a blank board shape.

3.2.1 Transferring the Design: The process of transferring a design from the capture stage to the board layout stage is launched by selecting Design then Update PCB Document.PcbDoc from the Schematic Editor menus, or Design, then Import Changes from PCB Document.PcbDoc from the PCB Editor menus -when one does the design, the design is compiled and a set of Engineering Change Orders is created, that will perform the following steps:

A list of all components used in the design is built, and a footprint is required for each component. Altium Designer will attempt to locate each footprint in the currently available libraries, and place each into the PCB workspace. If the footprint is not available, an error will occur.

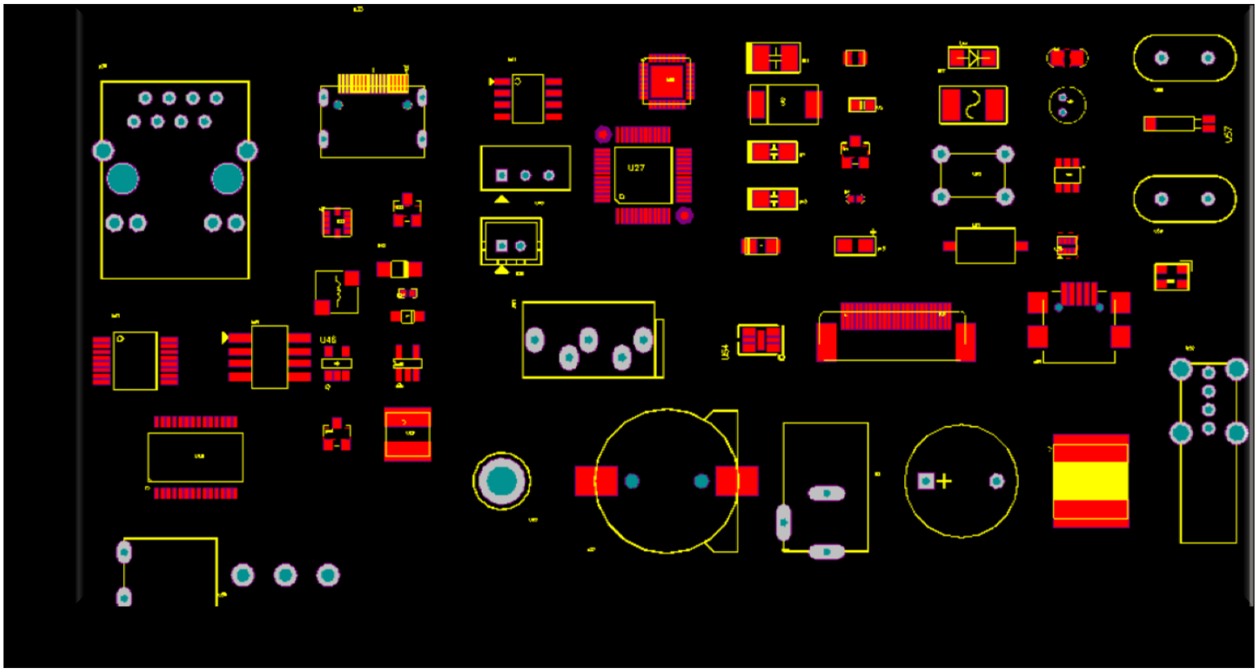

Figure5. Footprints

For the footprints that were not available, the appropriate footprint was downloaded from the internet and added to the existing libraries or drawn according to their datasheets. A list of all nets (connected component pins) in the design is created. Altium Designer will add each net to the PCB, and then attempt to add the pins that belong to each net. If a pin cannot be added an error will occur - this happens when the footprint was not found, or the pads on the footprint do not map to the pins on the symbol. Addition design data is then transferred, including placement rooms, net and component classes, and PCB design rules.

3.2.2 Setting up the PCB Workspace: Before positioning the components on the board we needed to configure certain PCB workspace and board settings, such as the grids, layers and design rules.

3.2.3 PCB Workspace Grids: We need to ensure that our placement grid is set correctly before we start positioning the components. All the objects placed in the PCB workspace are aligned on a grid called the snap grid. This grid needs to be set to suit the routing technology that we plan to use.

To set the snap grid, complete the following steps:

1. Select Design » Board Options, to open the Board Options dialog.
2. Click the Grids button down the bottom right to open the Grid Manager.
3. Altium Designer supports multiple user-defined grids.
4. The grid was set for 0.5mil for the entire tablet.
5. Click OK to close the dialogs.

3.2.4 Configuring the Display of Layers: There are three types of layers available in the PCB Editor:

1. **Electrical layers** - includes the signal layers and internal power plane layers.
2. **Mechanical layers** - there are 32 general purpose mechanical layers, used for design tasks such as dimensions, fabrication details, assembly instructions, or special purpose tasks such as glue dot layers. These layers can be selectively included in print and Gerber output generation.
3. **Special layers** - these include the top and bottom silkscreen layers, the solder and paste mask layers and drill layers.

3.3 PCB Fabrication

After the compilation of PCB design, as shown in the figure below, the next step is the process of fabrication. Design will be sent abroad to a PCB fabrication company.

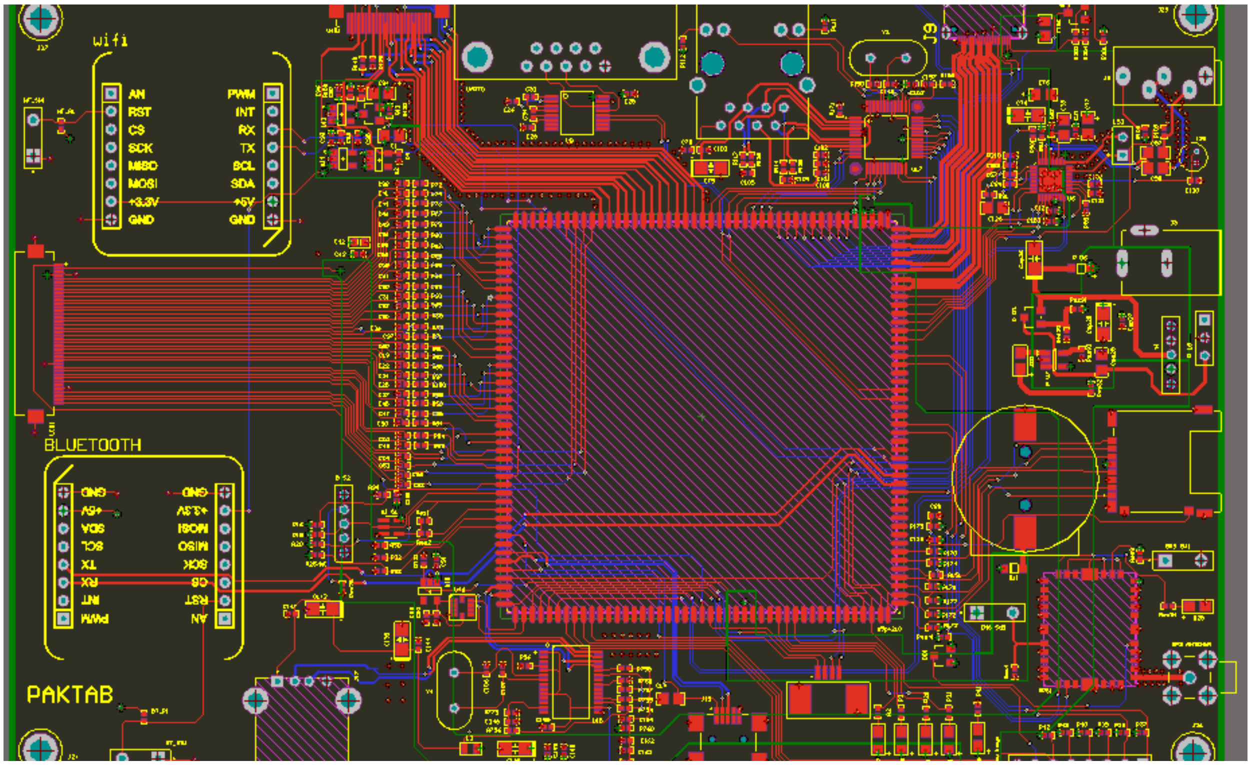

Figure 6 .Designed PCB board

3.4 PCB Debugging: After the successful designing and printing of Printed circuit board (PCB) the next step is to do the debugging and testing of the arrived board. The method is further described in chapter 5.

Chapter 4: Circuit simulations and PCB layout of Modules

4.1 Start Selector Switches Circuit

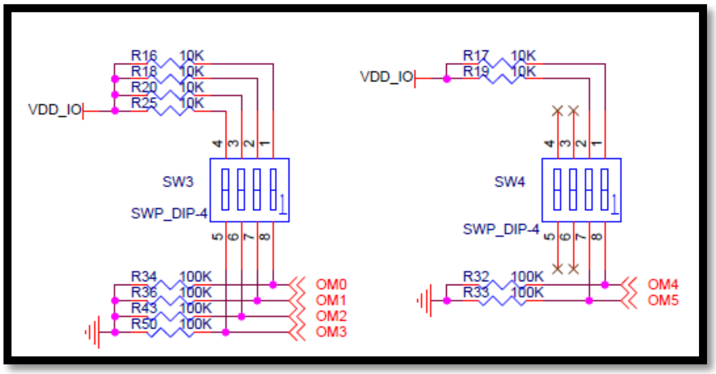

Figure 7: A Schematic showing the Start Selector Switches

Start Selector Switches circuit is as shown above.

The main purpose of this circuit is the switching of connections of the processor pins. The main advantage of these switches is that they can be turned to make connections with the contacts in that particular position.

In the above circuit, we have to start selector switches labeled as SW3 and SW4.

The first four pins of SW3 are connected to the VDD_IO. The pins 5, 6, 7, 8 are connected to pins 124, 123, 122, 121 of the processor via labels as OM0, OM1, OM2, and OM3 respectively.

4.2 UART (Universal Asynchronous Receiver/transmitter) and GPS

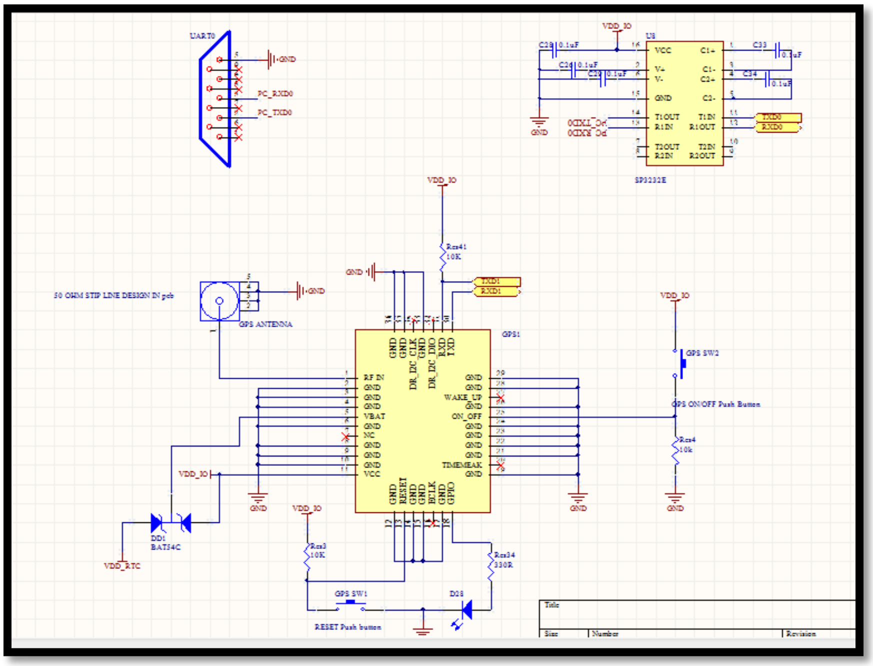

Figure 8: A Schematic showing UART and GPS

A Universal asynchronous receiver/transmitter, is a piece of computer hardware that translates data between parallel and serial forms. The "universal" designation in UART indicates that the data format and transmission speeds are configurable.

The SP3232E is a Small Outline IC and it has dimensions with a minimum width of 3.8mm and maximum width of 5.8mm with a length of 9.8mm. It is an RS-232 standard, transceiver solution intended for portable or handheld applications. It has a high-efficiency power supply that requires only 0.1µF capacitors in 3.3V operation.

The PCB layout of UART is as shown below.

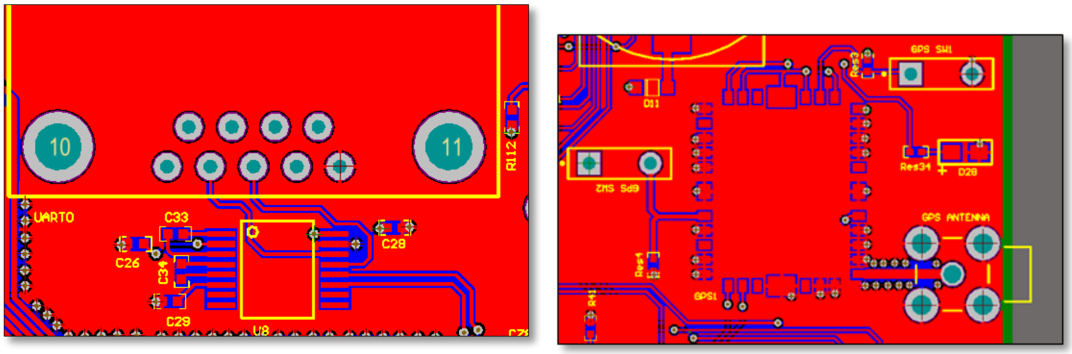

Figure9: PCB layout of Universal Asynchronous Receiver/Transmitter & GPS

4.3 <u>ADC (Analog to Digital Converter)</u>

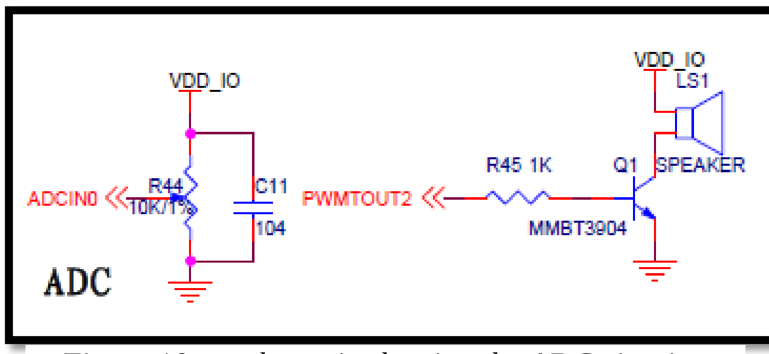

Figure 10: a schematic showing the ADC circuitry

An analog-to-digital converter (abbreviated ADC, A/D or A to D) is a device that converts a continuous voltage to a digital number that represents the quantity's amplitude. This conversion involves quantization of the input, so it necessarily introduces a small amount of error. Instead of doing a single conversion, an ADC often performs the conversions ("samples" the input) periodically. The result is a sequence of digital values that have converted a continuous-time and continuous-amplitude analog signal to a discrete-time and discrete-amplitude digital signal.

The MMBT3904 transistor is designed as a general purpose amplifier and switch. The useful dynamic range extends to 100 mA as a switch and to 100 MHz as an amplifier and then it passes the result on to the speaker.

4.4 **LCD**

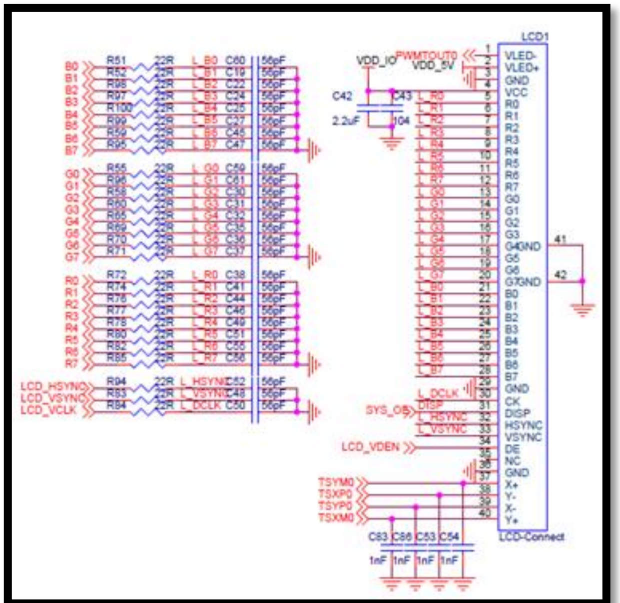

Figure 11: a schematic showing the LCD circuitry

A capacitive touchscreen panel consists of an insulator such as glass, coated with a transparent conductor such as indium tin oxide (ITO). As the human body is also an electrical conductor, touching the surface of the screen results in a distortion of the screen's electrostatic field, measurable as a change in capacitance. Different technologies may be used to determine the location of the touch. The location is then sent to the controller for processing.

The pins 5 to 28 of LCD connect are all connected to the respective capacitors which are further connected to the processor. While pins 37 to 40 are all grounded, via 1nF capacitors and they are getting inputs from the processor.

4.5 Camera

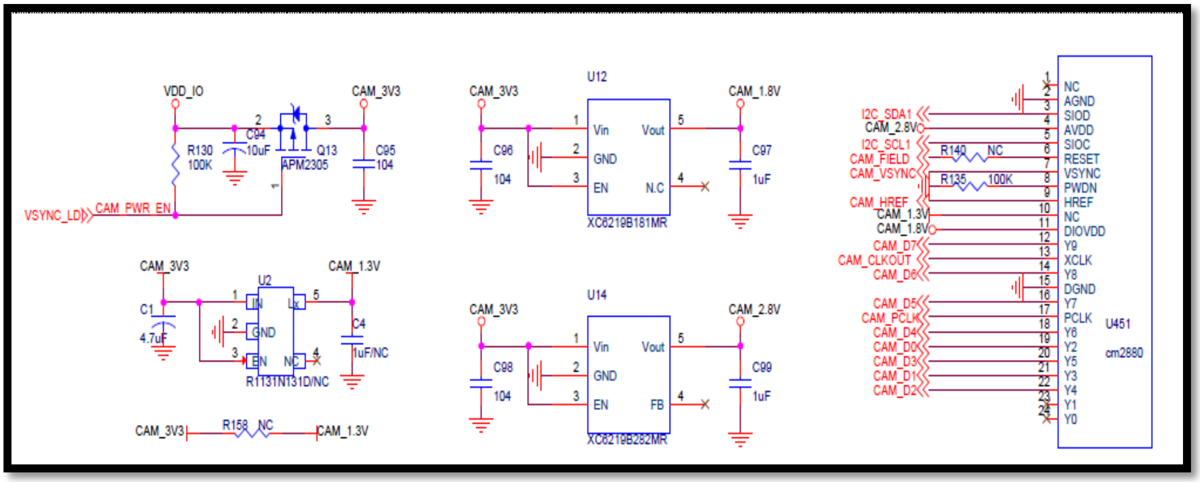

Figure 12: a schematic showing the Camera circuitry

The R1131N131DNC Series, used in this camera module, are CMOS-based low voltage regulator ICs with output voltage range from 0.8V to 3.3V. To prevent the destruction by over current, a current limit circuit is included. Standby mode realizes ultra-small consumption current. The output voltage of these ICs is internally fixed with high accuracy.

The APM2305 MOSFET acts as a power management tool in the camera circuit. It is of reliable and rugged nature and hence, is necessary for battery powered circuit.

The XC6219 ICs used in this circuit are highly accurate, low noise, CMOS LDO Voltage Regulators. They offer low output noise, high ripple rejection ratio and very fast response. The XC6219 ICs are fully compatible with low **ESR** ceramic capacitors, reducing cost and improving output stability.

4.6 <u>Ethernet</u>

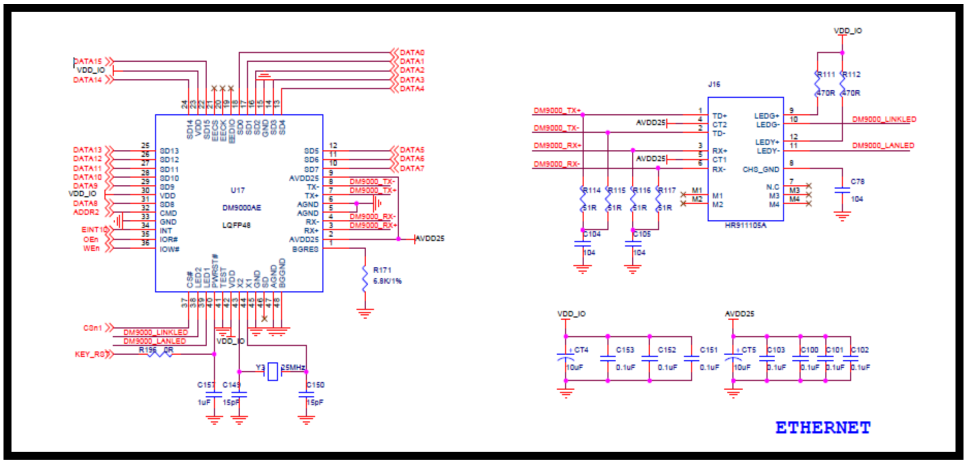

Figure 13: a schematic showing the Ethernet circuitry

The Ethernet module consists of two ICs:

1. HR911105A
2. DM9000AE

4.6.1 <u>HR911105A</u>

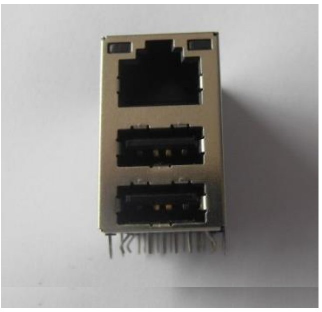

Figure 14: The HR911105A Ethernet port

The J16 (as shown in the above diagram) is a single port RJ45 connector which will enable the Paktab to support cable Ethernet interface, and the user can through the cable Ethernet to surf on Internet.

4.6.2 <u>DM9000AE</u> The DM9000A is a Fast Ethernet controller with a general processor interface supporting data transfer rates of 100 Mbps and a 10/100M PHY. It is designed with low power and high performance process that supports 3.3V.

4.7 G-Sensor

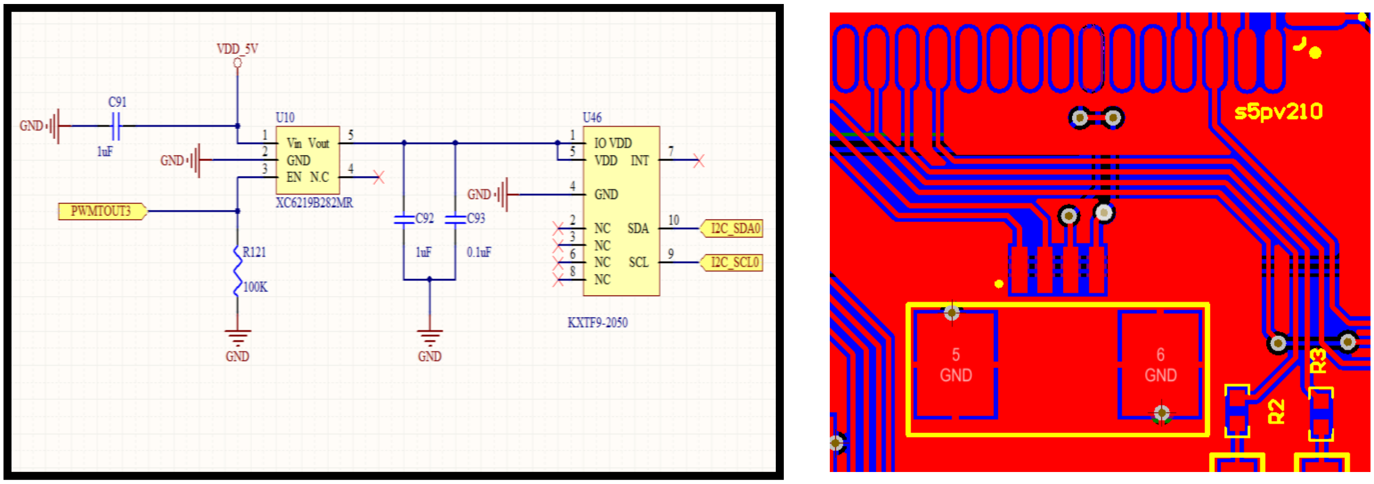

Figure 15: The Schematic and PCB layout of G-Sensor

The G-sensor works by using a low noise, CMOS **low drop out voltage regulator** in conjunction with a silicon digital accelerometer.

The accelerometer is in the form of a 3 x 3 x 0.9 mm **LGA** plastic package operating from a 1.8 – 3.6V DC supply. It is an electromechanical device that will measure acceleration forces. These forces may be static, like the constant force of gravity, or they could be dynamic - caused by moving or vibrating the accelerometer.

4.8 Capacitive Touch Screen

Pins 1 to 4 of the capacitive touch screen are connected to pin 127, 135, 138 and 139 respectively, of the processor while pins 5 and 6 are connected to ground.

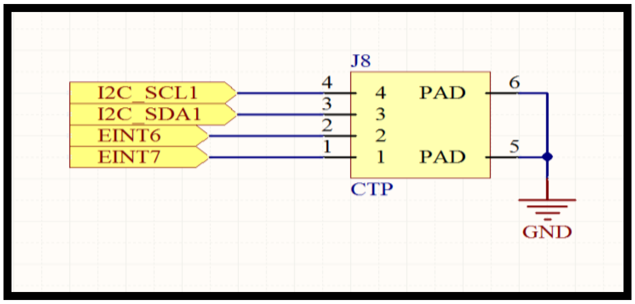

Figure16: The Schematic of Capacitive touch Screen

4.9 <u>USB Device</u>

USB On-The-Go, often abbreviated USB OTG or just OTG, is a specification that allows Paktab to act as a host, allowing other USB devices like a USB flash drive, digital camera, mouse, or keyboard to be attached to it.

USB OTG systems can drop the hosting role and act as normal USB devices when attached to another host. This can be used to allow a mobile phone to act as host for a flash drive and read its contents, downloading music for instance, but then act as a flash drive when plugged into a host computer and allow the host to read data from the device.

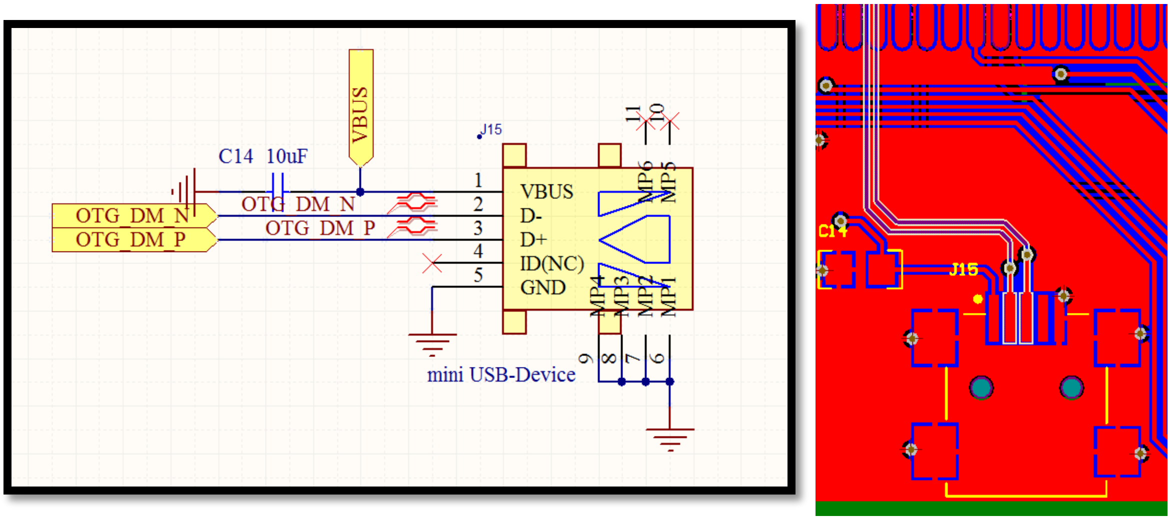

Figure 18: The Schematic of a USB OTG System and the PCB layout of USB OTG

The traces that connect to the processors through OTG_DM_N and OTG_DM_P are on an inner layer of the PCB and not on top of the board to avoid connection with other traces on the top layer.

4.10 <u>USB Host</u>

USB HOST supports USB devices such as mouse, keyboard, USB WIFI, USB Bluetooth, USB 3G, etc.

The S5PV210 provides only one channel USB HOST, however, it can be expanded to four channels via a USB expansion HUB chip AU9254, but since this would mean 4 times more occupancy of area on board, it was avoided.

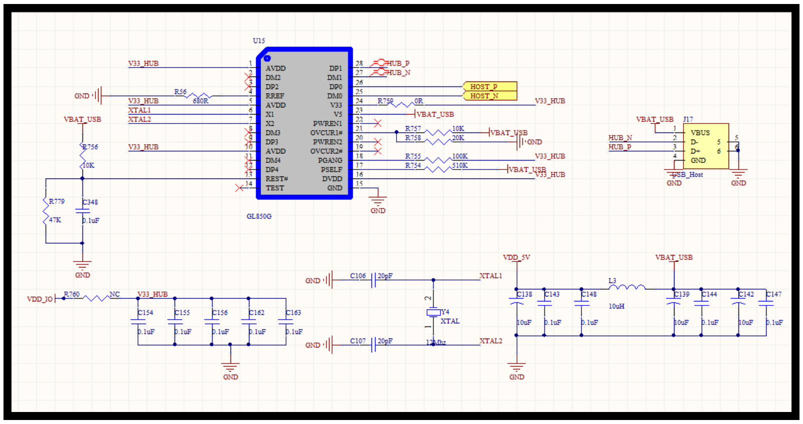

Figure 19: A Schematic showing USB Host

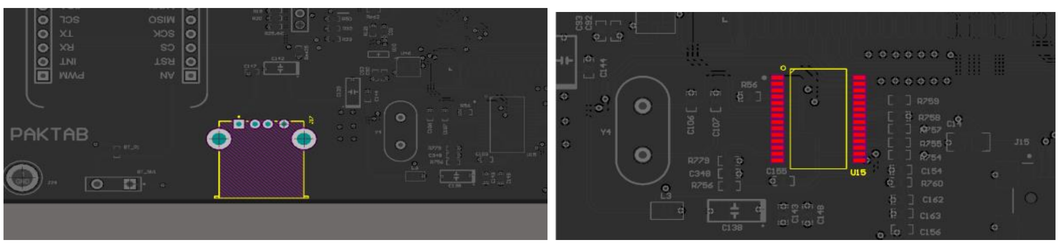

Figure20: Footprints of the USB port and AU9254 in the layout, respectively

4.11 **Bluetooth**

Bluetooth is a short-range radio technology (or wireless technology) aimed at simplifying communications among Internet devices and between devices and the Internet. It aims to simplify data synchronization between devices with one another, and the seamless communication between devices, local applications and the cloud.

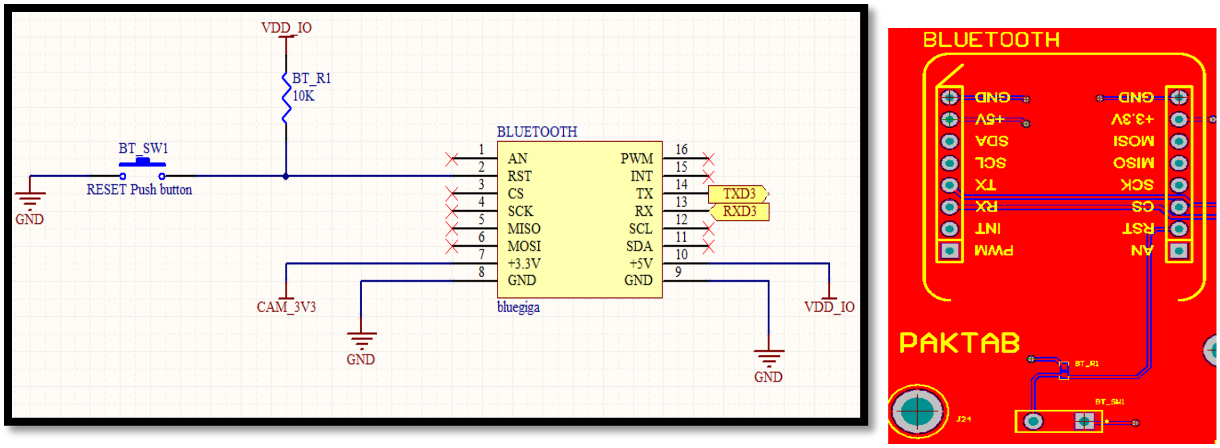

Figure 21: The Schematic drawing and PCB layout of Bluetooth

1.12 <u>Wi-Fi</u>

Wi-Fi refers to wireless networking technology that allows computers and other devices to communicate over a wireless signal using radio waves. This Wi-Fi module translates data into a radio signals and transmits it wirelessly. A wireless router receives the signal and decodes it. The router sends the information to the Internet using a physical, wired Ethernet connection. The process can also work in reverse, with the router receiving information from the Internet, translating it into a radio signal and sending it to Paktab's Wi-Fi module.

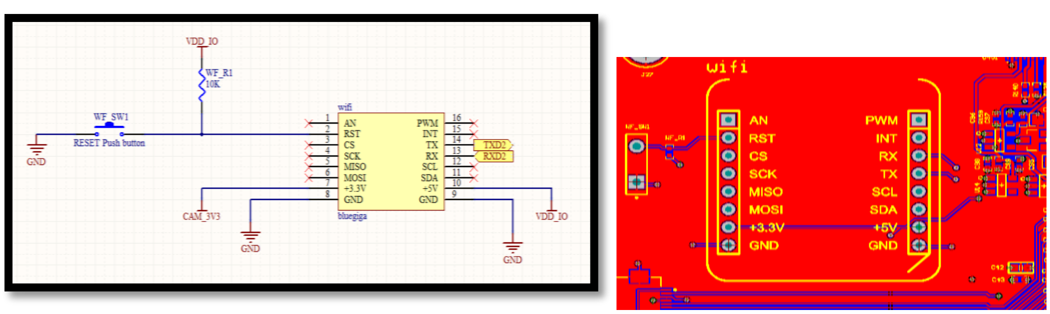

Figure 22: The Schematic drawing and PCB layout of Wi-Fi

Chapter 5: Software Design and Development

5.1 Software Design

5.1.1. System Block Diagram

5.1.2. Layered Model of Smart Cafe

5.2 Software Development

5.2.1. Installing Android OS on ARM11 Board

5.2.2. Developing a Café Management System

5.2.3. Developing an Android Food Menu App

5.2.4. Portals Development

5.2.5. Web Designing

5.2.6. Database for keeping Revenue Records

5.3 Software Quality Attributes

5.1 Installation of OS on the carrier board

The carrier board should only be powered up after all the connections and the routes are double checked

As the project demands development of a board around the processor, the processor being used will be imported with 180 interfaces/pins to provide communication between the processor and the modules designed on the board. The processor itself consists of a main processor chip along with cache, inand, nand and supporting circuits.

The next step will be to provide an OS to this board on which the board can run in future and further module drivers can be installed and tested. The S5PV210 processor supports Linux, WinCE and android. The operating system that will be transferred to the embedded hardware lies in the category of **embedded operating system**. In contrast to an operating system for a general-purpose computer, an embedded operating system is typically quite limited in terms of function, depending on the device in question, the system may only run a single application. However, that single application is crucial to the device's operation, so an embedded OS must be reliable and able to run with constraints on memory, size and processing power.

Booting the operating system in the newly fabricated board is the first step which is often referred to as the "first program". The booting can be done through various interfaces like USB, SD-card and serial port etc., by changing the configuration of DIP switches.

The standard Booting method used for embedded operating system installation will be followed which uses Microsoft visual studio 2010.

Chapter 6: Analysis and Evaluation

6.1 Debugging

6.1.1 Hardware debugging

6.2 Booting via Burning the Android Firmware Image under Windows

6.2.1 Requirements of PC

6.2.2 Requirements of Hardware

6.2.3 Preparations before Burning Image

6.2.4 USB driver installation

6.2.4.1 Download "uboot.bin" for running in the RAM

6.2.5 Burning the image into nand OR inand

6.2.5.1 Formatting the flash

6.2.6 Android driver installation

6.2.6.1 Burn the Android Image using fastboot

6.2.7 Configure the Boot Mode from either nand or inand

6.2.8 Set the operating system startup parameters

6.1 Debugging

The debugging and testing is one of the crucial steps involved in the design of embedded circuit as the success of the board depends upon it. The debugging is done at two levels: HARDWARE debugging and SOFTWARE debugging.

6.1.1 Hardware debugging

The design of PAK-TAB consists of a processor in the middle and different modules attached to the core to provide functioning. These will checked and verified as follows:

1. The first step will be to check for the availability of ground and power voltages which can be done by using a DMM on different test points existing on the board. A test point is usually an otherwise un-needed hole or via in a PCB that can be used to attach a multi-meter or scope probe to measure the signal at that point.
2. The design of PAK-TAB consists of power planes that provide required voltages to components throughout the board. The voltages floating in the board are named, VDD-5V, VDD I/O and CAM_3V3. The input jack of the board will be checked to ensure that the battery connected is providing 5V input for operation, and that the current requirements are met.
3. It is essential to check whether each module is getting the required voltage for its operation which can also be done using a simple DMM check method.
4. The traces and routes of PCB on the board are confirmed by tracking them to their location on the board. This is a lengthy process but is necessary as it ensures that no

tracks are short circuited. The traces are confirmed by placing probes at the start and end of a trace and short circuiting would indicate that no trace breakage is present in the path.

5. The next step is to check the board for damaged components. Capacitors tend to look brown if they have been overheating. One can also use a multi-meter across each capacitor to look for short circuiting. If there are no obvious damages or short circuiting, continue to inspect the other components.

6. The exact placement of components (as indicated by the schematics and the Layout designed in the Altium designer) are also ensured.

7. After ensuring that the power requirements, traces and the components are attached correctly. The next step will be to test each module separately and check whether it is working properly or not. This involves both hardware and software debugging to be done in parallel. The software drivers of each module will be installed and the working of modules will be checked by using inbuilt commands of the software.

Since, the tablet will run initially on android, simple testing android applications will be designed that can be used to check all the inputs, outputs and the results of that particular module.

6.2 Booting via Burning the Android Firmware Image under Windows

Booting is a startup sequence that starts the operating system of a computer when it is turned on. A boot sequence is the initial set of operations that the computer performs when it is switched on. Every computer has a boot sequence. The average computer doesn’t understand the boot sequence but it is important to know for customizing and troubleshooting your computer.

6.2.1 Requirements of PC

- Windows XP
- Install burn tools: DNW, fastboot
- Drivers for USB and Android 4.0.4
- Image files

6.2.2 Requirements of Hardware

- Carrier Board of the Tablet
- A USB type-A to mini-USB cable
- A 9-pin serial cable
- 9V DC power adapter

6.2.3 Preparations before Burning Image

1. The fastboot software is used to burn the image file that was created after the source code compilation. Place the burn tool *fastboot.exe* and the driver files *AdbWinApi.dll*, *AdbWinUsbApi.dll* into folder *D:\fastboot*
2. Power the development board, and turn on the power switch
3. Using the USB to mini-USB cable: Connect the mini-USB plug to the mini-USB receptacle of the tablet, connect the type-A plug to the USB port of PC using the 9-pin serial cable
4. Configure boot mode via DIP switches SW3 and SW4 according to the table below:

Set the DIP switch as 1000 01

PIN	1	2	3	4	5	6	7	8
PORT	OM0	OM1	OM2	OM3	OM4	OM5	NC	NC
STATUS	VCC	GND	GND	GND	GND	VCC	NC	NC

Table 4: Configuration of DIP switches

6.2.4 USB driver installation

This is done by first connecting the board to a PC via the onboard mini-USB interface through the USB cable. The on-board power supply is then turned on.

The PC recognizes the device connected to it which can be seen in the Device Manager window.

The device driver that needs to be installed on the Tablet must be saved in the PC. It can be selected by browsing the folders to get to the specific location where it has been saved.

The DNW software which is a software tool that recognizes the setting up of a connection between the two devices. The DNW status bar would now show [USB : OK], as shown in the figure below:

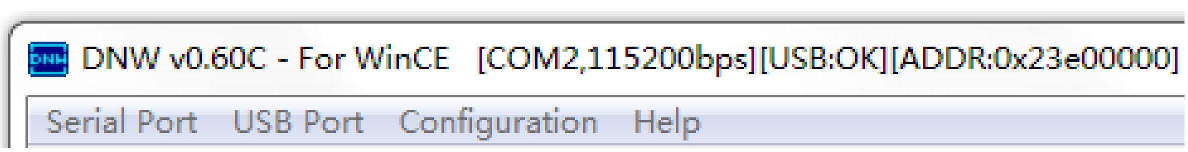

Figure 23: DNW software

DNW transmits the image file that was created after compilation of the device driver’s source code after which its status bar will show [USB: x] first, then [USB: OK], indicating the **USB interface initialization** was successful.

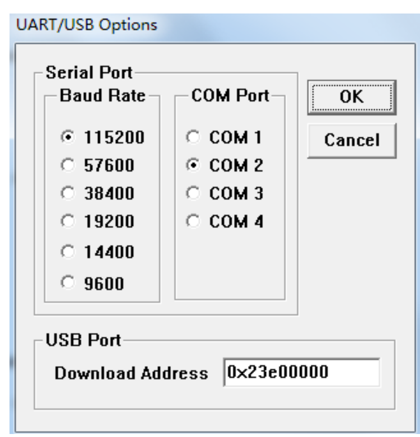

Figure 24: UART/USB OPTIONS

6.2.4.1 Download "uboot.bin" for running in the RAM

The DNW can be configured as shown in the diagram and the "uboot.bin" image file is transmitted. When the uboot starts running, a 3 seconds countdown message will show up.

6.2.5 Burning the image into nand OR inand

This section will describe how to burn the image onto the nand / inand

6.2.5.1 Formatting the flash

1. Formatting the nand flash

   Use the **nand scrub** command to format the whole nand

2. Formatting the inand flash

   Use the **fdisk –c 0** command to format the whole inand flash

6.2.6 Android driver installation

A USB device will be detected by the computer, open "Devices Manager", double-click [Android 1.0] and select the android driver.

6.2.6.1 Burn the Android Image using fastboot

1. Enter "fastboot" in the DNW window.
2. A USB device will be detected by the computer.
3. Open "Devices Manager and select the appropriate driver

Now the device is installed successfully

1. Copy the fastboot file folder to the PC, as it was placed in D:\fastboot
2. Launch the CMD, change directory to fastboot
3. Executive "fastboot devices" to search the current devices

Use the **fastboot erase userdata** and **fastboot erase cache** commands to erase user data and cache data respectively.

6.2.7 Configure the Boot Mode from either nand or inand

The configuration of DIP switches SW3 and SW4 for booting from nand is 110000 while that from inand uses 101100.

6.2.8 Set the operating system startup parameters

Restart the development board, in the 3s countdown, press space bar and set the start of the operating system parameters for programming android4.0 to inand version.

Restart the development board.

Chapter 7: Future Work and Conclusion

7.1 Recommendations for Future Work

7.1.1 Software Domain

7.1.2 Hardware Domain

7.2 Conclusion

7.2.1 Overview

7.2.2 Objectives achieved

7.1 Recommendations for Future Work

Pak-tab contains possibilities for both Software and Hardware domains. These possibilities are discussed below:

7.1.1 Software Domain

The future work that could be done on Pak-Tab is designing of a software application that makes it more task-specific. In this project different modules have been designed around s5pv210 processor. Overall, the designed hardware possesses GPS, Wi-Fi, capacitive screen, G-sensor and has ports/interfaces like USB, HDMI, SD-card and Ethernet. These modules offer a chance to install a variety of different applications on Pak-tab, making it more generic. The person who wishes to continue the project in the Software domain can create an application that can use the aforementioned modules; standalone or in a combination.

7.1.2 Hardware Domain

For Students who are more interested in the hardware domain, they would need to redesign the entire tablet, as the existing embedded hardware cannot change. Redesigning could be done in the following three ways

- **Tablet size is reduced**

  We can achieve this by adding in more layers.

- **Installing new modules**

  Modules such as TV-out etc. can be installed.

- **Different Processor**

  A dual core or Quad core processor could be used for faster processing.

7.2 Conclusion

7.2.1 Overview:

This project involved the development of a tablet pc that can be used in homes and offices replacing the more common desktop computers and laptops. Tablets are currently the pinnacle of mobile technology. They are compact, very lightweight and extremely easy to carry. Pak-Tab can be ideal for those who browse the Web casually, such as read the news or popular websites etc. The design procedure involves the circuit schematics and PCB layout stages which lead to the manufacture of Pak-Tab's base-board. After passing through the hardware debugging phase we come to the OS and drivers' installation phase and upon its completion, the system can be booted and finally Android Applications can be installed on it using the .apk file format.

7.2.2 Objectives achieved:

The objectives that were aimed for and met, were the core features of the tablet namely GPS, Wi-Fi, USB Port, SD CARD slot, Audio/Mic, G-sensor, Capacitive screen, HDMI, Analog to Digital Converter and Ethernet. The touchscreen is sensitive enough to accurately register light swipes and taps with a fingertip or stylus. Pak-Tab is designed for portability; it's sleek, lightweight and durable. Pak-Tab also contains a webcam in addition to the other features, including USB, HDMI and Ethernet ports; auxiliary jack; built-in microphone; and memory card reader.

GPS Facilities: Global Position System is a service that pinpoints the exact location of the user. GPS facility is achieved in the tablet via a module that incorporates internal antenna and is connected to the processor through UART.

Wi-Fi services: Wi-Fi services help the user to connect to the internet through a wireless medium. This allows for the mobility of the user within the network coverage area. For this purpose a Wi-Fi module has been attached to pins 170 and 171 of the processor.

Ethernet services: Ethernet is used to provide communication between peers. Via Ethernet, data is shared between individuals that are connected to each other through a wired medium. Ethernet would be connected to pins 51 to 67, and to pins 132, 177, 179 and 180 of S5PV210 processor.

Video Communication: Pak-Tab can act as a video communication platform. It is designed to have modules such as capacitive touch Screen, Audio/Mic, and Camera that allow for efficient video transmission and reception. Video communication can be done between two users using Internet through either WiFi or Ethernet connection. For bigger images or for better vision, the tablet can also be connected to a larger screen through an HDMI interface on the tablet.

Achievements: Pak-Tab's greatest achievement is its performance. The performance of a tablet is determined by the type of processor the device has, the amount of RAM, hard drive or solid-state drive capacity, wireless capabilities (Wi-Fi) and other features. The technical specifications determine how powerful and fast a tablet PC is and how well it performs.

Here are some the performance measures of Pak-Tab:

CPU	S5PV210
Operating Frequency	1GHz
Memory	512MB /1GB DDR2
inand flash	4GB inand

Table 4: Performance Measures Of Pak-Tab

Contributions: The references given at the end of this document were a great source of help and guidance throughout the PCB designing phase. Moreover, Stanford University's online courses provided free access to information that proved beneficial towards making us meet our goals when designing Pak-Tab.

Limitations: The major limitation faced in designing Pak-Tab was that after the Gerber files were sent for the manufacture of Hardware platform, we had to wait for about two months until it was finished and sent back. However, we utilized this time by learning about the installation process of OS and device drivers.

Applications: The tablet will be designed such that it is available at a cheap rate and can be used for specialized purposes such as high speed video processing, tracking and surveillance DTE tool and also be able to handle multiple systems connected to the board.

One major advantage of a tablet computer is that they are portable. There has long been a need for a computer that people could take with them so that they could do their work outside of the office. There are of course other options for this but laptops are not really as portable as most people need them to be and smart phones don't have the capability to do all of the things that people need done. A tablet on the other hand is ideal since it is small enough to be easily portable and it can handle most of the tasks that people need to be able to do.

Furthermore, a tablet is ideal for entertainment. Due to their small size, it can be held in different positions; that makes them good for things like watching movies or television shows. They are also good for reading books because they can be held in a position that is ideal for being able to read.

In addition, availability of applications for a tablet is more as compared to laptops or desktop computers. This means that one can constantly add new features to one's tablet to make it more useful. The huge number of apps that are being written for tablets has meant that they have become a lot more adaptable than other computers. In general there are not apps for PC's or laptops, instead one has to rely on the software companies to develop new products.

Appendix

1. ***Appendix A – The Tablet / Hardware interface***
2. ***Appendix B – Code to display Menu categories***
3. ***Appendix C – Code to display list of dishes***
4. ***Appendix D- Code to display details of a dish***
5. ***Appendix E- Code to send and receive orders***
6. ***Appendix F- Code to print receipts from Portal***
7. ***Appendix G- User Manual***

APPENDIX-A

The Tablet / Hardware interface

TABLE OF CONTENTS

The Tablet / Hardware interface

The tablet would comprise the following interfaces:

1. **Serial ports for Debugging**: Our tablet provides us with a serial port for debugging.
2. **HDMI interface:** The tablet supports HDMI output via the onboard mini HDMI interface. Audio/Video signals can be presented on terminal devices which support HDMI1.3 protocol, such as TV sets, monitors, etc.
3. **Camera interface:** The camera interface is a 24-PIN general interface, supports all OV serial cameras, and eliminates the need of camera adaptor.
4. **Earphone interface:** The onboard earphone jack provides audio output to earphone or power amplifier.
5. **Recording input:** The tablet supports sound recording via onboard MIC, no extra MIC is needed.
6. **DIP switches:** The DIP switches SW3 and SW4 are used for boot mode configuration.
7. **SD card slot:** The tablet provides with one SD card slot.
8. **ETH:** The tablet supports a cable Ethernet Interface; the user can surf the Internet through the cable**.**

APPENDIX-B

Definitions

AUTO-MDIX: Auto-Sensing Medium Dependent Interface crossover enables every port on the switch to automatically detect the Ethernet cable type being used and adjust to make a link over that cable.

Fast Ethernet: Fast Ethernet is a collective term for a number of Ethernet standards that carry traffic at the nominal rate of 100 Mbit/s, against the original Ethernet speed of 10 Mbit/s.

LDO: A low-dropout or LDO regulator is a DC linear voltage regulator which can operate with a very small input–output differential voltage.

LGA: The land grid array (LGA) is a type of surface-mount packaging for integrated circuits (ICs) that is notable for having the pins on the socket rather than the integrated circuit.

LQFP stands for Low-profile Quad Flat Package and is a type of surface mount ICs

USB OTG: USB On-The-Go, often abbreviated USB OTG or just OTG, is a specification that allows USB devices such as digital audio players or mobile phones to act as a host, allowing other USB devices like a USB flash drive, digital camera, mouse, or keyboard to be attached to them.

Bibliography

1. ***Bibliography***
2. ***References***

Bibliography

- How To Do Everything: Ubuntu by Jeffrey Orloff
- Microprocessor and interfacing by Douglas V.Hall [first published 1991]
- Embedded systems by Raj Kamal [2008]
- Microelectronics By Sedra Smith [2010]
- Hardware Manual (X210II Rev1.0) [2012]

References

[1] Pcb Design By CHUNXIALIU1, 2013

[2] M. Chincholkar and J. Herrmann. Modeling the impact of embedding passives on manufacturing system performance, September 2002.

[3] Datasheet Forum, datasheetcatalog.com

[4] The R Development Core Team. The R Reference Manual -Base Package. Network Theory Limited, 2005.

[5] T. DeMarco and T. Lister. People-ware Productive Projects and Teams. Dorset House Publishing, 1999.

[6] http://www.ti.com/

[7] http://www.analog.com/en/index.html

[8] http://edn.com/design/pc-board/4424239/PCB-design-basics

[9] http://system-on-a-chip.findthebest.com/l/341/Samsung-Intrinsity-S5PV210

[10] http://www.xda-developers.com/tag/fastboot/

[11] http://forum.xda-developers.com/wiki/index.php?title=ROM-VS-Kernel

[12] http://www.lifehacker.com.au/2013/09/rooting-your-android-phone-everything-you-need-to-know/

[13] http://www.nxp.com

[14] http://www.monolithicpower.com/Lists/Product_Synchronous%20Switcher_5717/DispForm.aspx?ID=97

[15] http://www.wolfsonmicro.com/products/audio_hubs/WM8976/

[16] http://www.vishay.com/search?query=DO-35_02&search-inventory-submit.x=13&search-inventory-submit.y=7&type=inv

[17] http://www.kionix.com/accelerometers/kxtf9

[18] http://www.freprt.com/index.php?page=shop.product_details&flypage=yagendoo_VaMazing_2.tpl&product_id=39&category_id=97&option=com_virtuemart&Itemid=27&lang=en

[19] http://www.alibaba.com/product-gs/804227616/_IC_MSPN09_A0_4000.html

[20] http://www.proconn-tech.com.tw/p3-c.asp

[21] http://www.anpec.com.tw/searchProducts.aspx?lang=en&act=all_search&keyword=APM2305

[22] http://semibatteries.howtochargebatteries.com/and9031-constant-current

[23] http://www.nxp.com/products/diodes/switching_diodes/series/1N4148_1N4448.html

[24] http://www.webopedia.com/TERM/K/kernel.html

[25] http://www.dsoftware.us/category/operating-systems/linux

[26] http://www.nxp.com