ELECTRONICS ASSEMBLY
AND
PRINTED CIRCUIT BOARD LAYOUT

A graduate project submitted in partial fulfillment of the requirements for the degree of Master of Science in Electrical Engineering

By

Syed Shayan Alam

August 2013
The project of Syed Shayan Alam is approved:

_________________________________________  ________________
Dr. Xiaojun Geng                   Date

_________________________________________  ________________
Hamid Gholami                   Date

_________________________________________  ________________
Dr. Ali Amini, Chair             Date
# TABLE OF CONTENTS

SIGNATURE PAGE ........................................................................................................... i ii

LIST OF FIGURES .......................................................................................................... iv

LIST OF TABLES ........................................................................................................... ix

ABSTRACT ...................................................................................................................... x

CHAPTER 1: INTRODUCTION ......................................................................................... 1

1.1 INTRODUCTION ....................................................................................................... 1

1.2 ARDUINO MICROCONTROLLER ............................................................................ 2

1.3 XBEE WIRELESS MODULE ................................................................................... 4

CHAPTER 2: DESIGN PHASE ......................................................................................... 7

2.1 PROTOTYPE DESIGN ............................................................................................. 7

2.2 SCHEMATICS .......................................................................................................... 21

2.3 PCB LAYOUT .......................................................................................................... 41

2.4 DOCUMENTATION & DESIGN STANDARDS ....................................................... 58

CHAPTER 3: MANUFACTURING ...................................................................................... 62

3.1 QUOTATION & PURCHASE ORDER ................................................................. 62

3.2 RECEIVING & STOCK ......................................................................................... 66

3.3 PRODUCTION ....................................................................................................... 69

CHAPTER 4: PRODUCT ANALYSIS ............................................................................... 76

4.1 DESIGN FOR MANUFACTURING REPORT ....................................................... 76

4.2 FUTURE ENHANCEMENTS (SMT) ...................................................................... 78

CHAPTER 5: COMPLIANCES ......................................................................................... 87

5.1 STANDARDS AND CERTIFICATIONS ................................................................. 87

5.2 QUALITY MANAGEMENT ....................................................................................... 90

CHAPTER 6: CONCLUSION .......................................................................................... 95

REFERENCES .............................................................................................................. 96

APPENDIX: SOURCE CODES ...................................................................................... 97
LIST OF FIGURES

Figure 1-1 Arduino Duemilanove ................................................................. 2
Figure 1-2 Xbee Shield (Without Xbee) ......................................................... 4
Figure 1-3 Xbee 802.15.4 (With external antenna) ........................................ 6
Figure 2-1 Sample Complex Design on a Bread Board ................................. 7
Figure 2-2 Transmitter Block Diagram ......................................................... 9
Figure 2-3 Receiver Block Diagram ............................................................. 9
Figure 2-4 Hard-Wire Design (Non-Wireless) ............................................. 10
Figure 2-5 Xbee attached to USB Adapter .................................................... 11
Figure 2-6 XCTU Graphical User Interface ................................................ 12
Figure 2-7 Modem Configurations ............................................................. 13
Figure 2-8 Good Communication Message ............................................... 14
Figure 2-9 C Programming (Example of Multiplication) ............................ 14
Figure 2-10 Arduino Programming (Example of Multiplication) ................. 15
Figure 2-11 Arduino Software Logo .......................................................... 15
Figure 2-12 Arduino Coding Environment Window ................................... 16
Figure 2-13 Receiver Code ............................................................... 17
Figure 2-14 Transmitter Code .............................................................. 18
Figure 2-15 Prototype Receiver Module ...................................................... 19
Figure 2-16 Prototype Transmitter Module .............................................. 20
Figure 2-17 Boarduino Kit ............................................................. 22
Figure 2-18 Boarduino Kit (Parts Soldered) .................................................. 22
Figure 2-19 Boarduino Schematics ......................................................... 23
Figure 2-20 Xbee Schematics ............................................................ 24
Figure 2-21 Original Power Supply ................................................................. 25
Figure 2-22 Power Supply Modified (USB Interface Removed) ...................... 25
Figure 2-23 Xbee Part from Xbee Schematics .............................................. 26
Figure 2-24 Adding Xbee Using Copy/Paste Function ................................. 27
Figure 2-25 Xbee Attached to ATmega ......................................................... 28
Figure 2-26 Adding Part to EAGLE Schematics ......................................... 29
Figure 2-27 Adding Part to EAGLE Schematics ......................................... 30
Figure 2-28 Part Dimension Overview ....................................................... 33
Figure 2-29 LED Connection to ATmega .................................................... 34
Figure 2-30 Electrical Rule Check .............................................................. 35
Figure 2-31 Xbee VCC Floating ................................................................. 36
Figure 2-32 Zero ERC Errors ................................................................. 36
Figure 2-33 Receiver PCB Schematics ..................................................... 37
Figure 2-34 Edit Package ......................................................................... 38
Figure 2-35 Drill Hole Menu ..................................................................... 39
Figure 2-36 Drill Hole Properties ............................................................. 39
Figure 2-37 Transmitter PCB Schematics ............................................... 40
Figure 2-38 Switch to Board EAGLE Function ....................................... 41
Figure 2-39 First Time Schematics to Board ........................................... 42
Figure 2-40 EAGLE PCB Layout Software ............................................. 42
Figure 2-41 Placement of Major Parts ..................................................... 43
Figure 2-42 Receiver PCB Placement ..................................................... 44
Figure 2-43 Polygon Drawing ................................................................. 45
Figure 2-44 Naming the Polygon ................................................................. 46
Figure 2-45 Ratsnest Symbol ................................................................. 46
Figure 2-46 Ratsnest Result ................................................................. 47
Figure 2-47 Net Classes Declaration ..................................................... 47
Figure 2-48 Auto Router EAGLE Function ......................................... 48
Figure 2-49 Zero ERC or DRC ............................................................. 49
Figure 2-50 Component Layer Processor .............................................. 50
Figure 2-51 Solder Layer Processor .................................................... 51
Figure 2-52 Component Silk Layer Processor ....................................... 52
Figure 2-53 Solder Silk Layer Processor .............................................. 53
Figure 2-54 Component Stop Layer Processor ..................................... 54
Figure 2-55 Solder Stop Layer Processor ............................................. 55
Figure 2-56 Drill Layer Processor ....................................................... 56
Figure 2-57 CAM Processor Output Files .......................................... 57
Figure 2-58 BOM (Transmitter PCB) .................................................. 59
Figure 2-59 BOM (Receiver PCB) ....................................................... 60
Figure 2-60 Assembly Drawing (Sample) ............................................ 61
Figure 3-1 GERBER Verification Using ViewMate ................................. 63
Figure 3-2 Quotation Review Form ..................................................... 65
Figure 3-3 Quality Certificates (PCB Bare-board) ................................. 67
Figure 3-4 Bare PCB (Transmitter Left / Receiver Right) ..................... 67
Figure 3-5 Humidity Indicator ........................................................... 68
Figure 3-6 PCB Marking ................................................................. 68
Figure 3-7 Illustration of Through-Hole (left) & SMT (Right) Components .......... 69
Figure 3-8 Traveler from Accurate Electronics ........................................... 70
Figure 3-9 Wave Soldering Machine Process Illustration ............................... 72
Figure 3-10 Wave Solder Machine ............................................................... 72
Figure 3-11 Types of Masking (Chemical-left, Tape-middle, Die Cut-right) ........ 73
Figure 3-12 Pallets for Wave Solder Machine ................................................ 73
Figure 3-13 Selective Soldering (Machine-left, Soldering-right) ....................... 74
Figure 3-14 Final Product (Receiver PCB) .................................................... 75
Figure 3-15 Final Product (Transmitter PCB) .................................................. 75
Figure 4-1 Desiccants .................................................................................... 76
Figure 4-2 Design for Manufacturing (DFM) Report ..................................... 77
Figure 4-3 SMT Machine (MyData) ............................................................... 78
Figure 4-4 SMT Common Packages ............................................................. 79
Figure 4-5 Integrated Circuit Packages ......................................................... 80
Figure 4-6 Ball Grid Arrays (BGA) ................................................................. 81
Figure 4-7 Quad Flat No Lead (QFN) ............................................................ 81
Figure 4-8 SMT Placement Illustration .......................................................... 82
Figure 4-9 Paste on PCB ............................................................................... 82
Figure 4-10 Stencil for Solder Paste (20” x 20”) .......................................... 83
Figure 4-11 Stencil Printer (Ekra) ................................................................. 83
Figure 4-12 Solder Wetted (0402 Package – left, BGA Package – right) ........... 84
Figure 4-13 SMT Reflow Oven Recipe ........................................................... 84
Figure 4-14 SMT Reflow Oven (15’ long with 9 zones) ..................................... 85
Figure 4-15 Zone Control Window for Reflow Oven ........................................... 85
Figure 4-16 Xbee SMT Package (Through-Hole – left, SMT – Right) .................... 86
Figure 4-17 ATmega SMT Package (Through-Hole – left, SMT – Right) ............... 86
Figure 5-1 Accurate Electronics (Logo & Certifications) ........................................ 89
Figure 5-2 Database Software GUI ................................................................. 90
Figure 5-3 NCMR (Non-Conformance Material Report) ....................................... 93
Figure 5-4 QMS Forms at Accurate Electronics .................................................. 94
LIST OF TABLES

Table 1-1 Arduino Duemilanove Specifications .................................................. 3
Table 1-2 Xbee Variety Specifications ............................................................... 5
Table 2-1 LED Ratings ....................................................................................... 31
Table 2-2 Electrical / Optical Characteristics .................................................... 32
ABSTRACT

Electronics Assembly and Printed Circuit Board Layout

By

Syed Shayan Alam

Master of Science in Electrical Engineering

The purpose of this project is to convey the knowledge of making final electronics assembly for Commercial, Consumer, Medical and Aerospace & Defense usage. Engineering students graduating from many universities are far away from the reality of designing and manufacturing of electronics assembly. The standards, certifications of a product and the quality of a product required to assemble an electronics assembly will be explained in great detail. To fully understand and explain the manufacturing process a non-complex product was designed with all aspects of today’s engineering design. Designing of a wireless device was chosen to control the intensity of an LED. This project is designed on an Arduino Microcontroller platform equipped with the Xbee wireless controller. This project elaborates each phase from designing of a Printed Circuit Board (PCB) all the way to each process of manufacturing.

PCB layout is performed using the Eagle freeware layout software. Xbee wireless controller is programmed using the Xctu software and the Arduino microcontroller is programmed using the Arduino’s proprietary software. All software will be explained in great depth. The production of the PCB will be processed at Accurate Electronics located in Chatsworth, California.
Chapter 1: Introduction

1.1 Introduction

Today’s graduating engineers are far away from the knowledge of electronics assembly. It is a subject that is not enlightened in many engineering schools of today. Many engineering schools along with California State Universities does focus on the making of Integrated Circuits (ICs) however neglects the methodology of making electronics devices that are used on daily basis around the world. The end users for these electronic assemblies includes the regular consumer, commercial (industrial), medical facilities, and/or Aerospace & Defense. This project will walk through all the necessary phases of making of electronics assembly. First, a product will be designed and then processed through the manufacturing facility. The end user for this product will be set to consumer ratings to reduce the cost of manufacturing. The aim is to learn all the major process of designing to manufacturing.

The best way to explain and understand the manufacturing process will be to choose a non-complex design while maintaining all aspects of today’s engineering design. Therefore, designing a wireless device to control the intensity of an LED is a perfect choice. The major feature of this design includes wireless capability (RF), analog to digital control and coding of the microcontroller. The design uses Arduino Microcontroller and Xbee wireless controller.
1.2 Arduino Microcontroller

For the prototype phase the Arduino Duemilanove development kit will be used. The Arduino Duemilanove is a 2009 version and it is very popular for prototyping many microcontroller based applications. The microcontroller is based on the ATmega IC. Due to its many input output (I/O) ports the designing has immersive possibilities. The development board is equipped with 14 digital I/Os. Amongst those 6 can be used as a Pulse Width Modulator (PWM) additionally the board also has 6 analog inputs, followed by a Universal Serial Bus (USB) connection. The microcontroller can be simply programmed using the proprietary software by Arduino.

![Arduino Duemilanove](image)

**Figure 1-1 Arduino Duemilanove**

The Arduino Duemilanove can be powered and programmed using the USB connection. To just power the unit, the DC input can be used with two options of either using the AC (Alternating Current) to DC (Direct Current) power wall outlet or power using a 9 volt battery. The Duemilanove is fully operational with the range of 7 to 20 DC volts input. The memory of the Duemilanove is depended on the ATmega IC installed on
the socket. Larger scale applications require more space therefore the Duemilanove is capable of handling two types of ATmega, one has 16KB called the ATmega168 and other has 32KB called the ATmega328. For this project the suitable candidate is the ATmega168.

Programming the ATmega on the Duemilanove is hassle-free since the ATmega is pre-burnt with the bootloader. The coding is written on the Arduino environment and loaded on the IC using the USB connection through the STK500 protocol. Below is the specification table for the Arduino Duemilanove.

<table>
<thead>
<tr>
<th>Microcontroller</th>
<th>ATmega168</th>
</tr>
</thead>
<tbody>
<tr>
<td>Operating Voltage</td>
<td>5V</td>
</tr>
<tr>
<td>Input Voltage (recommended)</td>
<td>7-12V</td>
</tr>
<tr>
<td>Input Voltage (limits)</td>
<td>6-20V</td>
</tr>
<tr>
<td>Digital I/O Pins</td>
<td>14 (of which 6 provide PWM output)</td>
</tr>
<tr>
<td>Analog Input Pins</td>
<td>6</td>
</tr>
<tr>
<td>DC Current per I/O Pin</td>
<td>40 mA</td>
</tr>
<tr>
<td>DC Current for 3.3V Pin</td>
<td>50 mA</td>
</tr>
<tr>
<td>Flash Memory</td>
<td>16 KB (ATmega168) or 32 KB (ATmega328) of which 2 KB used by bootloader.</td>
</tr>
<tr>
<td>SRAM</td>
<td>1 KB (ATmega168) or 2 KB (ATmega328)</td>
</tr>
<tr>
<td>EEPROM</td>
<td>512 bytes (ATmega168) or 1 KB (ATmega328)</td>
</tr>
<tr>
<td>Clock Speed</td>
<td>16 MHz</td>
</tr>
</tbody>
</table>

Table 1-1 Arduino Duemilanove Specifications
1.3 Xbee Wireless Module

The Xbee also known as Zigbee is a wireless module that is designed for applications that requires less data communication while having long range capabilities and also using less power consumption. The Xbee is very friendly with the Duemilanove since it can be attached using a shield especially designed for easy connection. Xbee Shield acts like a daughter board which attaches on top of the Duemilanove shown below is the picture of a Xbee Shield.

![Xbee Shield (Without Xbee)](image)

**Figure 1-2 Xbee Shield (Without Xbee)**

Xbee comes in 16 different varieties for all kinds of application needs. For this project we would be choosing the Xbee with the lowest power consumption. Table 1-2 shows Xbee of all kinds with all ranges of frequency and also the power consumption along with the maximum range of communication in line of sight.
<table>
<thead>
<tr>
<th>Product</th>
<th>Frequency</th>
<th>Power Output</th>
<th>Maximum Range</th>
<th>RF Data Rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>XBee ZB</td>
<td>2.4 GHz</td>
<td>1.25/2 mW</td>
<td>120 m</td>
<td>250 Kbps</td>
</tr>
<tr>
<td>XBee-PRO ZB</td>
<td>2.4 GHz</td>
<td>63 mW*</td>
<td>3.2 km</td>
<td>250 Kbps</td>
</tr>
<tr>
<td>XBee ZB SMT</td>
<td>2.4 GHz</td>
<td>3.1/6.3 mW</td>
<td>1200 m</td>
<td>250 Kbps</td>
</tr>
<tr>
<td>XBee-PRO ZB SMT</td>
<td>2.4 GHz</td>
<td>63 mW</td>
<td>3.2 km</td>
<td>250 Kbps</td>
</tr>
<tr>
<td>XBee 802.15.4</td>
<td>2.4 GHz</td>
<td>1 mW</td>
<td>90 m</td>
<td>250 Kbps</td>
</tr>
<tr>
<td>XBee-PRO 802.15.4</td>
<td>2.4 GHz</td>
<td>63 mW*</td>
<td>1.6 km</td>
<td>250 Kbps</td>
</tr>
<tr>
<td>XBee-PRO 900HP</td>
<td>900 MHz</td>
<td>250 mW</td>
<td>45 km**</td>
<td>200 Kbps</td>
</tr>
<tr>
<td>XBee-PRO XSC</td>
<td>900 MHz</td>
<td>100 mW</td>
<td>24 km**</td>
<td>9.6 Kbps</td>
</tr>
<tr>
<td>XBee-PRO 900</td>
<td>900 MHz</td>
<td>50 mW</td>
<td>10 km**</td>
<td>156 Kbps</td>
</tr>
<tr>
<td>XBee-PRO DigiMesh 900</td>
<td>900 MHz</td>
<td>50 mW</td>
<td>10 km**</td>
<td>156 Kbps</td>
</tr>
<tr>
<td>XBee DigiMesh 2.4</td>
<td>2.4 GHz</td>
<td>1 mW</td>
<td>90 m</td>
<td>250 Kbps</td>
</tr>
<tr>
<td>XBee-PRO DigiMesh 2.4</td>
<td>2.4 GHz</td>
<td>63 mW*</td>
<td>1.6 km</td>
<td>250 Kbps</td>
</tr>
<tr>
<td>XBee-PRO 868</td>
<td>868 MHz</td>
<td>350 mW</td>
<td>80 km**</td>
<td>24 Kbps</td>
</tr>
<tr>
<td>XBee Wi-Fi</td>
<td>2.4 GHz</td>
<td>16 dBm</td>
<td>300 m</td>
<td>65 Mbps</td>
</tr>
<tr>
<td>XBee 865LP</td>
<td>865 MHz</td>
<td>12 dBm</td>
<td>4 km</td>
<td>80 kbps</td>
</tr>
<tr>
<td>XBee 868LP</td>
<td>868 MHz</td>
<td>12 dBm</td>
<td>4 km</td>
<td>80 kbps</td>
</tr>
</tbody>
</table>

**Table 1-2 Xbee Variety Specifications**

There are two possible choices of Xbee from the chart above we can see that the lowest power consumption is the 1mW. The difference between Xbee 802.15.4 and XBee DigiMesh 900 is the type of communication protocol. Since the 802.15.4 is easy to work with therefore it will be used on the Xbee Shield.
The range on the selected Xbee 802.15.4 is only 90 meters and sometimes due to noise in medium may distort the signals therefore the best choice is to attach an external antenna on the Xbee making it stronger on communication level. Suppliers such as Adafruit.com sells pre-installed external antennas on the Xbee making it easier for the engineers to design at easier pace. Figure 1-3 shows the picture of the Xbee with an external antenna.

Figure 1-3 Xbee 802.15.4 (With external antenna)
Chapter 2: Design Phase

2.1 Prototype Design

All electronics assembly requires the prototype phase. Prototype is also known as “bread boarding”. During this process the application is built with loose components or sent through the manufacturing house if the engineer is confident enough. However, every Design Engineer chooses to go through two phases of prototypes. First is on the bread board followed by production run through manufacturing house. On the other hand, some engineer only chooses one phase of prototype via bread board design and followed by First Article (FA) approvals. First Articles are the first run of any batch ran through the manufacturing house that is also known as the “Golden Board”. The FA is explained in great detail in Chapter 3. Figure 2-1 is an example for the bread board design that is commonly used in electronics industry for prototype.

Figure 2-1 Sample Complex Design on a Bread Board
Today’s circuit designing software are very advanced and has the ability to simulate the functionality of the circuit. However, designing on a bread board gives the conformation of the electronic circuitry that it will perform for the application it is designed for. The difference between bread boarding and software simulation is that the in real design an engineer can observe the heat transfer or communication issues that are not easily discovered in simulation. Overall, the design industry of today does both software simulation of a circuit and also many prototypes before mass productions.

For this project two development kits of Arduino Duemilanove will be used. One will act as a transmitter and other as the receiver. Along come the two sets of Xbee for transmitting and receiving data. The project design is to control the intensity of a Light Emitting Diode (LED) using an analog source in our case that will be a potentiometer. This project requires analog signal to be converted into digital in order to control the light intensity. Following are the block diagram for the Transmitter and the Receiver design.
Figure 2-2 Transmitter Block Diagram

Figure 2-3 Receiver Block Diagram
The design begins by setting up the non-wireless version of the project. This way it conforms that the design will function properly. Therefore, the following block diagram was built on a bread board to see if it actually works with hard wires instead of using the Xbee wireless communication. To perform this task an LED was chosen along with a potentiometer and attached to the breadboard with DC power going through the potentiometer. The hard-wire design did function properly and intensity was controllable. Again, this confirms that the potentiometer is compatible with the LED and also it works with 9 Volt battery. In a large scale production design an engineer would rather pull out the datasheet instead of the hard-wire test to confirm compatibility. However, since this project involves parts that can be easily tested therefore it was preferred that a real test would have been performed before stepping up to the wireless design. This prevents the designer from staying in a state of dilemma at a higher level of design that weather wireless is having issue or the parts are non-compatible. It is always a wise idea to test as much as possible with real life interaction but it is not always possible in a large scale design. Figure 2-4 displays the block diagram of the hard-wire potentiometer and LED compatibility check.

![Block Diagram, Part Description, Schematics](image.png)
Now the design is to be implemented on the Arduino Duemilanove. The Arduino environment is a software that can be acquired from the Arduino website “www.arduino.cc”. The software comes with the sample codes that can be viewed to learn the technique of coding. Xbee wireless modules are to be configured so that the same communication protocols can be set on both devices. To connect Xbee to the computer a special adapter is available to quickly configure it.

Figure 2-5 Xbee attached to USB Adapter

The adapter is connectable to the computer using the mini USB cable. To configure Xbee, a software called XCTU is being used provided by Maxstreme manufacturer of Xbee. With the XCTU software the baud rate on both Xbee can be set to same, for this project baud rate has been set to 19200 bit/sec. Baud rates are measured in bits per second and it is very necessary to be same on both wireless devices. For example if there are two wireless modules and one is set to 9600 baud rate and the other is set to 19200 the data transferred from the 19200 to 9600 will miss some of the data and data will be considered lost or lagged during communication.
The XCTU software is very friendly and can also be used to upgrade or downgrade the firmware on the Xbee. The XCTU has a user friendly Graphical User Interface (GUI). The Xbee after being connected to the computer using the adapter with the mini USB gets detected by the XCTU software automatically. Followed by the current baud rates shown on the right side, flow control, data bits, parity check, and stop bits are shown. Baud rate is the only of concern for our project as long as other settings are at default.

Figure 2-6 XCTU Graphical User Interface
Setting up the Baud rate is the major step, next comes the verification of the firmware. Verify the firmware that it is exactly same on both Xbee modules. The current firmware and settings can be acquired by clicking the “Read” button the “Modem Configuration” tab. Figure 2-7 shows the “Modem Configuration” tab where the version can be seen to be 1083.

Figure 2-7 Modem Configurations
Confirm the communication changes by pressing the “Test / Query” button on the PC Settings tab. The test will return with the windows message box confirming that the communication with the modem is “OK” see Figure 2-8 for good connection. If the message box states that it is unable to communicate then very if all settings including the baud rate and firmware are exactly same on both Xbee modules.

**Figure 2-8 Good Communication Message**

Now the Xbee modules are ready to be attached to the Arduino Duemilanove. Two different codes were written, one for the transmitter and the other for the receiver. Transmitter is the unit attached with the potentiometer and the receiver is attached with the LED. Arduino environment is executed by the executable file “arduion.exe” this file opens a window where code can be written. Arduino website includes many examples of coding methods. The coding method is very similar to “C Programming” following is an explanation of the language C anatomy that is being used in Figure 2-10 for multiplying.

**Figure 2-9 C Programming (Example of Multiplication)**
Following is an example code of multiplying two integers using the anatomy of C programming language in the Arduino environment. Just remember that Arduino coding always require the “setup( )” and “loop( )” and other function can be created inside the brackets of setup and loop.

```c
void setup()
{
    Serial.begin(9600);
}

void loop()
{
    int i = 2;
    int j = 3;
    int k;

    k = myMultiplyFunction(i, j); // k now contains 6
    Serial.println(k);
    delay(500);
}

int myMultiplyFunction(int x, int y)
{
    int result;
    result = x * y;
    return result;
}
```

Figure 2-10 Arduino Programming (Example of Multiplication)

Figure 2-11 Arduino Software Logo
Arduino software executes with a window to write the code and it is equipped with the functionality of compiling and loading the code to the ATmega on the Duemilanove. Following is the first window that always executes with every run of Arduino software.

![Arduino Coding Environment Window](image)

**Figure 2-12 Arduino Coding Environment Window**

The check mark under the file menu is to verify and compile code followed by the arrow key to load the code to the ATmega chip.
The Receiver code and the Transmitter code are shown below in Figure 2-13 and Figure 2-14 but the entire code is available on the Appendix: Source Codes acquired by the Community of Robots. As shown in the Receiver code that pin 5 on the Arduino Duemilanove will be used as an output for the LED intensity. The delay is used to check for the value every 1000ms which is 1 second this way the LED will be more stable. Many “Serial.print” functions were used to verify the communication through the Xbee, using the print function the Arduino environment was able to display live changing values on the Arduino monitor.

```
// RECEIVER
byte incomingByte,pot;

void setup() {
  // start serial port at 19200 bps
  Serial.begin(19200);

  pinMode (5, OUTPUT);  // set led pin 5 to output
  delay(1000);
}

void loop() {

  if (Serial.available()) {  // is there any information available
    incomingByte = Serial.read();  // assign bytes to the variable 'in'
    Serial.print(int(incomingByte));
    Serial.println();
  }
}
```

Figure 2-13 Receiver Code
The Transmitter code is capable of reading the potentiometer value via the analog pin 0 and convert it into byte which is then stored on the “new val” and can be read by the receiver code in order to lit the LED with correct intensity. There are many ways to write both of these codes such as one way could be to only have 4 different brightness of LED where the transmitter code will send values as 1 to 4 (low brightness to high brightness).

![Transmitter Code](image)

**Figure 2-14 Transmitter Code**
The Arduino DueMilanove is now loaded with the code and attached with the Xbee Shield and are now ready for hard connections. Figure 2-15 shows the prototype of receiver module with the green LED connected to pin 5 on the DueMilanove.

Figure 2-15 Prototype Receiver Module
Next comes the Transmitter module that is transmitting the value of the potentiometer to the receiver module. Here the potentiometer is attached on the bread board because the Duemilanove does not have thee way pin connection.

![Prototype Transmitter Module](image)

**Figure 2-16 Prototype Transmitter Module**

The potentiometer is connected to the +5Vdc, ground and pin 0 for analog input from the potentiometer. This way the code reads the value and converts it into bytes and transfers it to the receiver module.
2.2 Schematics

The main purpose of this project is to assemble a product that is design specific. Which means the focus is towards low pricing and less parts. Therefore, the first step is to find out the necessary parts in order to build this circuitry design specific. In today’s design industry designers never starts from scratch since many module and sub-module parts come along with the schematics. In this case, the Xbee schematic is provided by www.ladyada.net website in many formats including the schematics for EAGLE software. EAGLE (Easily Applicable Graphical Layout Editor) is freeware software provided to non-profit organization and for educational purpose only. The software is falls in the category of CAD (Computer Aided Design) since it has two major applications which include schematics editor and PCB (Printed Circuit Board) layout. The current stable version of the software is 6.4. The drawback of using the freeware version is that it limits the size of PCB to 4 inches by 3.2 inches, the PCB can only have two layers and the schematics editor can only contain one sheet. The limitation does not affect the project’s design since it is a non-complex product. Let’s start by acquiring the necessary schematics for the Arduino Duemilanove. Ladyada’s website also provides a clone for the Duemilanove, this clone has only necessary parts required for ATmega to work properly. The clone is name “Boarduino” since it is a kit that requires soldering of individual parts to accomplish the Arduino. For this project a single Boarduino kit was purchased to verify the functionality. The Boarduino comes in two different versions, DC Boarduino and the USB Boarduino. The DC Boarduino was chosen since the final product needs to be capable of running with 9V battery. The kit comes along with loose Through-Hole parts that are soldered on to the PCB. Figure 2-17 shows the kit that comes
with all the accessories. The kit is then soldered as shown in Figure 2-18 and ready for evaluation. The method for evaluation involves the functionality verification of the design. For the efficient way of evaluating the Boarduino was to swap the ATmega chip with the Duemilanove prototype shown in Figure 2-16 this way the Boarduino controlled the intensity of the LED and proved that the Boarduino does function the same way as the Arduino Duemilanove development kit.

Figure 2-17 Boarduino Kit

Figure 2-18 Boarduino Kit (Parts Soldered)
The Boarduino is now can simply be replaced by the Arduino Duemilanove therefore the schematics provided from the www.ladyada.com will now be modified to attach the Xbee schematics. Following is the Boarduino schematics acquired from the www.adafruit.com. This schematics will be modified to be design specific.

Figure 2-19 Boarduino Schematics
The schematics can now be modified by executing the schematics on the “Schematics Editor” from EAGLE. The main focus is to remove unnecessary parts from the Boarduino and combine the Xbee module from the Xbee schematics shown on Figure 2-20.

Figure 2-20 Xbee Schematics
The first modification to be done on the Boarduino schematics is to remove the USB interface from the power supply circuitry as shown. Figure 2-21 is the original power supply and the modified power supply schematics can be seen on Figure 2-22.
Now the Xbee module from the Xbee schematics is to be copied and transferred to the Boarduino schematics. Figure 2-23 shows that the only the Xbee part has been selected from the Xbee schematics.

Figure 2-23 Xbee Part from Xbee Schematics
Note that the header of the Xbee is not needed and therefore the gray selection did not cross the header endings on Figure 2-23. Now the part is being copied to the Boarduino. The Xbee part placed on bottom right of the schematics can now be used and attached to the ATmega chip.

Figure 2-24 Adding Xbee Using Copy/Paste Function
The Xbee is now connected with the ATmega and can be controlled by the Boarduino.

The part is supplied with 5 Vdc from the 9 V battery source along with the common ground. The TXD and RXD connections are for the transmitting and receiving data from the Xbee to the ATmega.

![Diagram of Xbee Attached to ATmega](image]

**Figure 2-25 Xbee Attached to ATmega**

The Boarduino is now capable of wireless communication therefore it can now be converted into two different assemblies. It can be converted to the Receiver device by connecting the LED and can be converted to the Transmitter by attaching the potentiometer.
To add electronic part on to the schematics is very straightforward since many parts are already in the EAGLE library of components. Accessing the library is also simple and requires a single click from the left menu shown in Figure 2-26. This will open the “ADD” window and many parts are available. All these electronic components are real industry products. Parts that do not exist in the library mostly can be acquired by the manufacturer or build by the known dimension. In this project, Xbee was acquired from the manufacturer’s website and the copy paste function can be seen in Figure 2-23.

Figure 2-26 Adding Part to EAGLE Schematics
The through-hole LED was searched by the size of the LED acquired by the datasheet. The dimension of through-hole parts are really necessary to be aware and can be easily accessed by the datasheet of the manufacturer. Below is the dimension of the LED acquired from the manufacturer’s datasheet.

Figure 2-27 Adding Part to EAGLE Schematics
Furthermore, datasheet is very essential and hold many information regarding the part. By thoroughly going over the LED’s datasheet we are aware that this LED has high intensity, popular T-1 diameter package, wide viewing angle, etc. The manufacturer calling out there product “popular package” means that all variety of PCB layout software may easily use this part in their design because it is available in the libraries of the software. The datasheet also provides the maximum and minimum working conditions of the product. For the LED the datasheet specifies that operating temperature range is between -55° Centigrade to +100° Centigrade which is also same for the storage temperature. This temperature range is more than enough for this project. In some cases this information is very important especially for Aerospace & Defense projects since their electronic assemblies might be required to operate in harsh environment such as navy fleets in Atlantic Ocean or outer space in Mars. Below are the properties of LED acquired from the datasheet.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>AlGaAs Red/HER MV5094A</th>
<th>AlGaAs Red/Green MV5491A</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Continuous Forward Current</td>
<td>30/30</td>
<td>30/30</td>
<td>mA</td>
</tr>
<tr>
<td>Peak Forward Current - I_F</td>
<td>90</td>
<td>90</td>
<td>mA</td>
</tr>
<tr>
<td>(I = 1.0 KHz, Duty Factor = 1/10)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reverse Voltage - V_R (I_R = 10 μA)</td>
<td>5</td>
<td>5</td>
<td>V</td>
</tr>
<tr>
<td>Power Dissipation - P_D</td>
<td>120</td>
<td>120</td>
<td>mW</td>
</tr>
<tr>
<td>Operating Temperature - T_OPR</td>
<td>-40 to +100</td>
<td></td>
<td>°C</td>
</tr>
<tr>
<td>Storage Temperature - T_STG</td>
<td>-40 to +100</td>
<td></td>
<td>°C</td>
</tr>
<tr>
<td>Lead Soldering Time - T_SOL</td>
<td>Wave 260 for 5 sec</td>
<td></td>
<td>°C</td>
</tr>
<tr>
<td></td>
<td>Rework 240 for 5 sec</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 2-1 LED Ratings
The datasheet also provides other information that are not essential for this project. These information includes the optical characteristics and these information are essentials for the Medical and the Aerospace & Defense industry. For example, a night vision goggle is made for military purpose and the power switch requires LED visibility only up to 1 meter since it might alert the enemies if the LED’s luminous intensity is higher, then this information is important. Following is the Electrical / Optical Characteristics from datasheet.

<table>
<thead>
<tr>
<th>Part Number</th>
<th>AlGaAs Red/HER</th>
<th>Condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>MV5094A</td>
<td>MV5491A</td>
<td>20 mA</td>
</tr>
<tr>
<td>Luminous Intensity (mcd)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Minimum</td>
<td>2/2</td>
<td></td>
</tr>
<tr>
<td>Typical</td>
<td>5/5</td>
<td></td>
</tr>
<tr>
<td>Forward Voltage (V)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Maximum</td>
<td>2.8/2.8</td>
<td></td>
</tr>
<tr>
<td>Typical</td>
<td>2.0/2.0</td>
<td></td>
</tr>
<tr>
<td>Peak Wavelength (nm)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>660/635</td>
<td>660/565</td>
<td></td>
</tr>
<tr>
<td>Spectral Line Half Width (nm)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>20/45</td>
<td>20/30</td>
<td></td>
</tr>
<tr>
<td>Viewing Angle (°)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>75</td>
<td>75</td>
<td></td>
</tr>
</tbody>
</table>

Table 2-2 Electrical / Optical Characteristics

From the Table 2-2 the soldering temperature is now known to be 260°C for wave soldering and 240°C for reflow oven but no more than 5 second. Therefore, this information is essential for the manufacturing house to be aware of, since they will ensure that this part does not get higher heat in their entire process of manufacturing.
Adding the part to the EAGLE schematics is then followed by the package quick overview shown below in Figure 2-25. This requires an engineer to confirm the package dimension before placing the part on the schematics. The dimension can be seen on the right top of the window and also in the description section of the part on the left.

Figure 2-28 Part Dimension Overview
After adding the part on the schematics by clicking on the “OK” button on the “ADD” window the part is joint and placed at the necessary location. The LED is now connected to digital pin 5 from the ATmega IC as shown in Figure 2-26. Note that the LED is now represented using a diode symbol rather two through-holes since this is still a schematic. This part will convert into two through-holes once converted to PCB layout format.

![Figure 2-29 LED Connection to ATmega](image-url)
The Schematics is now to be verified by the ERC (Electrical Rule Check), a function built in EAGLE software to search for unconnected pins and therefore solving the problem of impedance.

Figure 2-30 Electrical Rule Check
After evaluating the ERC report it is noted that the Xbee is having the most issue therefore upon further investigation it was noticed that the VCC (+ DC Voltage) is not joint properly. Figure 2-31 display the error found and Figure 2-32 shows the issue is now fixed and resulting in zero ERC errors.
The Schematics for the “Receiver PCB” is now completed and the full schematics is shown below in Figure 2-33. Note that this schematics is design specific and features to reprogram on board, USB connectivity and other non-essential features were removed.

Figure 2-33 Receiver PCB Schematics
The “Transmitter PCB” is also made in conjunction with the “Receiver PCB”. The “Transmitter PCB” is not much different except the addition of potentiometer and removal of LED. However the potentiometer used on the prototype level is not available in the EAGLE’s part library therefore a similar part was dropped on the schematics and the pin holes were modified to this project’s need.

Modification can be done on any part by right clicking and selecting the “Open Device”, under this section choose “Edit Package” by right click menu on the part description.

![Figure 2-34 Edit Package](image)

The part can now be modified in any way, modification are required on the drill hole sizes as shown below by selecting the property of each hole and modified the size also shown in Figure 2-35.
The drill hole properties can be accessed for each hole by right clicking on each pin and selecting the “Properties”.

**Figure 2-35 Drill Hole Menu**

The Diameter is being modified as shown below to desired pin size inside the “Properties” window.

**Figure 2-36 Drill Hole Properties**
The Schematics for the “Receiver PCB” is now completed and the full schematics is shown below in Figure 2-37. The potentiometer can be seen below connected to the +5 V power source cojoint with the Xbee and the other end of the potentiometer is connected to the GND (ground). The middle pin is connected to the ATmega in order to transfer the analog value of voltage to the analog input PC5 pin and later converted into digital voltage and transferred to the “Receiver PCB” via Xbee wireless communication.

Figure 2-37 Transmitter PCB Schematics
2.3 PCB Layout

The “Receiver PCB” is being converted into a board. This process starts by clicking on “Switch to board” from the file menu located top left (shown in Figure 2-38).

![Figure 2-38 Switch to Board EAGLE Function](image)
If the schematic is being converted to board for the first time then a message of creating a new file message will be prompted (Figure 2-39). The file is created and then the software is switched to EAGLE PCB Layout (Figure 2-40).

Figure 2-39 First Time Schematics to Board

Figure 2-40 EAGLE PCB Layout Software
The EAGLE PCB Layout software starts with all scattered components located on the left and the PCB outlined is on the right side as shown in Figure 2-40. Now the engineer is to have some PCB layout experience knowledge before starting to place components inside the outline. The process usually starts by selecting sensitive parts such as in our case there are two, first is the Xbee and the ATmega. Therefore these parts will be assigned their own corners on the PCB. PCB layout for Aerospace & Defense and Medical are very sensitive and require much more experience with part’s characteristics such as heat issues. Parts that require heat sink can also be placed under thicker copper layers and be can avoid a physical heat-sink. Major parts are now placed inside the PCB. Note that wires can be seen all scattered which later be routed using the routing function.

Figure 2-41 Placement of Major Parts
The rest of the parts are now placed on the PCB while making sure that it is best location for the designated part by the knowledge of component engineering that comes through experience and can be acquired by evaluating characteristics of electronic components.

Figure 2-42 Receiver PCB Placement
The PCB is now complete and ready for the layer process. First process is set to draw the polygon around the PCB outline that’s the start of informing the software that this is the PCB to be layered and routed.
Since this project involves creating a simple PCB therefore the PCB will contain only two layers, GND (Ground) and VCC (Voltage). Name the polygon to “GND” as shown.

![Figure 2-44 Naming the Polygon](image)

Next step is to run the Ratsnest feature to achieve shortest possible paths for possible airwires.

![Figure 2-45 Ratsnest Symbol](image)
The Ratsnest will return with total number of airwires on the below of application.

![Ratsnest Result](image)

**Figure 2-46 Ratsnest Result**

Now the “Net Classes…” button has been used from the edit menu to set the Net Classes. For a normal commercial PCB layer width is set to 12mil, drill hole size to 20mil, and any clearance to be at least 10mil.

![Net Classes Declaration](image)

**Figure 2-47 Net Classes Declaration**
The PCB is now ready to go through the “Autorouter” feature. The basic functionality of this function is to route wires while maintaining the clearance that was declared to be at least 10mil in Figure 2-47.

Figure 2-48 Auto Router EAGLE Function
The PCB is now much stable and all the wires are routed, to confirm if every route is correct therefore DRC (Design Rule Check) and ERC (Electrical Rule Check) were executed.

![Zero ERC or DRC](image)

**Figure 2-49 Zero ERC or DRC**

The layout is now confirmed that it does not violate any rules therefore it is ready for manufacturing. The files that need to be transferred is called Gerber. This file is industry universal data exchange format for PCB manufacturers. Start the “CAM Processor” in order to create the Gerber file from EAGLE PCB layout software located under file menu. Using the CAM Processor many layers will be created. Every layer is essential for the PCB manufacturer.
Start by creating the “Component” layer as shown below. Only select the Top, Pads, and Vias layer from left. Make sure the device is set to “GERBER_RS274X” and the file is save “.cmp”.

Figure 2-50 Component Layer Processor
Next layer is the “Solder”. Bottom, Pads, and Vias layers were selected from the layer section and the file extension was changed to “.sol”

Figure 2-51 Solder Layer Processor
“Component Silk” layer is created by selecting the Dimension, tPlace, and tNames layers.

The file extension is changed to “.plc”.

Figure 2-52 Component Silk Layer Processor
“Solder Silk” is created by selecting the Dimension, bPlace and tNames and the file extension is changed to “.pls”
“Component Stop” layer is created by only selecting tStop layer. The file extension is changed to “.stc”.

![Component Stop Layer Processor](image)

**Figure 2-54 Component Stop Layer Processor**
“Solder Stop” layer is created by only selecting the bStop layer. The file extension is changed to “.sts”.

Figure 2-55 Solder Stop Layer Processor
Now comes the “Drill” layer this layer includes the Drills, and Holes layer. The Device is also changed to the industry standard “EXCELLON” and the file extension is changed to “.ncd”.

Figure 2-56 Drill Layer Processor
All the layers are now complete and the CAM Processor is ready to output the files necessary for the PCB house to manufacture. Clicking the “Process Job” outputted the files shown below in Figure 2-57.

![Figure 2-57 CAM Processor Output Files](image)

All the above steps were repeated for the “Transmitter PCB” this PCB will be controlling the LED on the “Receiver PCB”.

57
2.4 Documentation & Design Standards

The documentation phase is the most important section of any design. Many large scale companies that design electronic assemblies have designated section just for documentation. Engineers maintaining the design document are referred as Systems Engineer. These engineers are responsible to maintain all documentation in data server up to their latest revisions. Maintaining revisions for assemblies prevents all confusion and drops the level of mistakes. ISO (International Standards of Organization) is one of the most popular standard organizations offering the ISO 9001 for QMS (Quality Management System). ISO 9001 requires a company to maintain documentation for all designing and productions. Furthermore, the soldering standards for this project will be through the IPC (Association Connecting Electronics Industries). IPC-A-610E standard is for the Acceptability of Electronic Assemblies, this standard was designed to make a universal acceptance rules for accepting quality of electronic assemblies. IPC-A-610E has divided the acceptance quality of electronic assemblies into three classes. From Class 1 for the lowest possible quality while functioning, Class 2 for consumer and commercial products and Class 3 is suitable for Medical, Aerospace and Defense products. Most of the electronic manufacturers do not produce assemblies at Class 1 quality they rather produce all products at minimum of Class 2. For this project the suitable class is Class 2 due to low cost production. All products are also inspected under Class 2 acceptance standards at the production facility. Standards are explained in great detail further in the “Standards and Certification” section.
Documents necessary for production includes the BOM (Bill of Materials), GERBER files, and assembly drawing if applicable. BOM holds the entire list of the parts used in order to assemble the electronics assembly. EAGLE Schematics provides extraction for the BOM from the schematics file. First step is to open the schematics file using the EAGLE Schematics Editor software then from the file menu select to run the ULP (User Language Program) and select the file called “BOM.ulp”. The bill of material is now extracted as shown below.

Figure 2-58 BOM (Transmitter PCB)
The BOM extraction method was repeated for the “Receiver PCB” and the following BOM was extracted.

![Figure 2-59 BOM (Receiver PCB)](image)

The BOM are ready and the GERBER files are also extracted as shown above. The assembly drawing is not necessary for this project since the design does not contain any complex part installation. An assembly drawing covers all the necessary notes for the manufacturer such as applying a label or attaching a screw on an assembly. The drawing is sometime 3D or 2D depending on the complexity of a PCB.
See Figure 2-60 for a sample assembly drawing from a real manufacturing product provided by Accurate Electronics. The drawing is sometimes also referred as blue prints.

Figure 2-60 Assembly Drawing (Sample)
Chapter 3: Manufacturing

3.1 Quotation & Purchasing Order

The quotation process is the first step any manufacturer goes through because the cost is always important to the customer. The author of this project is considered as a customer. The customer also needs to verify that the production house is following the standards required by the engineer. This project requires that the manufacturer produces board with ISO 9001 and IPC-A-610 standards.

Accurate Electronic is electronic assembly manufacturer located in Chatsworth the heart of the county of Los Angeles. This manufacturer is chosen due to its capability, 25 years of service, quality and standards certified. Many companies require the audit of the manufacturer by their Quality Engineers. This way the engineer is aware of manufacturing process that the product will go through and also verify the QMS (Quality Management System). After approval of the facility, the facility gets added to the AVL (Approved Vendor List) at customer site so that future order can be placed without any interventions of Quality Engineers. For this project, the facility is considered to be AVL and the data can be sent to them for the manufacturing of the electronic assembly.

At quoting process Accurate Electronics verifies that all the files are on hand such as Gerber, and BOM. Any notes such as IPC Class 2 production is listed on the PO (Purchase Order). All GERBER files are verified by the Process Engineer at Accurate Electronics using the software called ViewMate a Pentalogix software. Accurate Electronics is supplied with all GERBER layers in “.zip” extension labeled as “Receiver.zip” and “Transmitter.zip”.
Following are screenshots of GERBER verification. The GERBER files can be drag and drop inside the software window to view the layers as shown below.

Figure 3-1 GERBER Verification Using ViewMate (Transmitter Top/Receiver Bot)
After the verification process and evaluation by the Process Engineer the quote is submitted to the customer. If the customer accepts the price, terms and conditions then a PO (Purchase Order) is submitted by the customer to the manufacturer. At quotation level Accurate Electronics also fills out an internal form “Customer Quotation Review” also referred as Form# F72001 Rev C. Note that an ISO 9001 facility is to maintain all necessary forms with a form number and document revision. This is to make sure that the facility is documenting every detail for all products. This form is used to summaries the electronic assembly detail into one page so that pricing, terms and conditions can be decided. The form is filled by the Process Engineer and contains all necessary details and then reviewed by the management to decide for Lead Time (time to assembly and ship) and price. It is very important to review all necessary steps required to manufacture an assembly since any special requirement may increase the price. Many times it has been seen that jobs that are not thoroughly studied resulted in higher cost at the time of production and the end result was no profit and even in worst cases loss to the manufacturer. Vice versa if a manufacturing company quotes the assembly at higher rate and in reality the job is not that high then the manufacturer might lose the bid since there are many other manufacturers competing for the same job. Overall, the quotation stage is the most critical stage of the production since it is the start of the job and the serious it is taken the smoother manufacturing process flows. Customer Quotation Review form filled by the Process Engineer is attached on the next page. The form shows that this job is for only 2 PCBs, the job is Turn-key (parts purchased by manufacturer) plus Consignment (parts supplied by customer), the soldering process is Lead, and the manufacturing standard is IPC Class 2. Electronic component will be provided to the Accurate
Electronics however PCB bare board will be purchased by Accurate Electronics. Further it mentions that the GERBER file is provided by the customer and it has only 20 Through-Hole parts. Lastly, the manufacturing lead time is set to be 5 days at maximum which is signed by the Manufacturing Manager.

![Figure 3-2 Quotation Review Form](image-url)

Figure 3-2 Quotation Review Form
3.2 Receiving & Stock

All products coming in or shipping out from Accurate Electronics is inspected. The Receiving & Stock department has Receiving Inspectors that are certified by IDEA (Independent Distributors of Electronics Association). This certification trained the inspectors to detect and stop counterfeit parts to enter the facility and also responsible to inspect all incoming parts with AQL (Acceptance Quality Level). Using AQL is to make judgment on batch of parts to be under conformance, the AQL requires percentage of batch to be inspected rather than 100% inspection. If any failure is encountered during AQL inspection, then the percentage of inspection increases by the inspector. Further, the Receiving Inspector also responsible to insert the amount received in the database and label parts with Accurate Electronics’ part number. The stock completes kit for job specific when all parts are received by the inspector. All incoming product requires to have paper work due to ISO documentation regulation. The parts are only accepted with COC (Certificate of Compliance) this way the making of a product is fully traceable. All Medical, Aerospace and Defense industries require high level of traceability.

This project involves both consignment and turn-key items, therefore the inspector will receive all components without an AE# except the PCB bare board. The PCB bare board will be purchased by Camtech a fabrication facility. Upon receiving of the bare PCB from the fabricator the inspector inspects only 1 out of 5 due to AQL inspection. Transmitter and Receiver bare PCBs were ordered quantity 5 of each. The inspector confirms the quantity and receives the parts in the database of Accurate Electronics. Later these parts are moved to the stock and kitted for production run.
The package label “Quality Certificate” states that the PCB is for part number “Transmitter Board”. The size can also be read to be 80 mm by 63.17 mm by 1.57 mm, since the fabrication is done in China therefore the measurement system is in metrics. Date code is 1326 which is translated to 26\textsuperscript{th} week of year 2013. Quantity is 5 and therefore only 1 PCB was inspected thoroughly. The inspector verifies the PCB with the provided GERBER from the customer.

![Figure 3-3 Quality Certificates (PCB Bare-board)](image1)

![Figure 3-4 Bare PCB (Transmitter Left / Receiver Right)](image2)
Every electronic assembly is packed with a humidity indicator card, this card can detect encountering of moisture to the PCB. The card is dropped in the packaging at the time of vacuum sealing the PCBs. Receiving inspector inspects the humidity card upon receiving of all electronics component sensitive to moisture. The bare boards used for this project were manufactured and shipped from China. Therefore the following humidity indicator was verified by the inspector at Accurate Electronics.

![Humidity Indicator](image1.png)

**Figure 3-5 Humidity Indicator**

The above humidity indicator states the PCB was not introduced to any moisture at the time from packaging to opening of this package. Now the inspector inspects other details on the bare PCB such as marking as shown below.

![PCB Marking](image2.png)

**Figure 3-6 PCB Marking**
3.3 Production

Process Engineers and Project Engineers decide the necessary steps for the product to be manufactured with. For this project there were many possibilities. Today’s electronic assemblies are divided into two different types of component packages, Through-Hole and SMT (Surface Mount Technology). The project is design with only Through-Hole parts due to quick layout. SMT is far more efficient way of manufacturing of electronic assemblies. Medical, Aerospace and Defense products are mixed assembly of Through-Hole and SMT. Following is the visual distinction between Through-Hole and SMT components.

![Figure 3-7 Illustration of Through-Hole (left) & SMT (Right) Components](image)

The production begins by the making of Traveler, it is a document that defines all steps for manufacturing and requires sign off at every stage of production. Travelers are created by the Process Engineer. Further it contains the BOM, and all other necessary documentation along with a printed copy of the GERBER file. In most cases, the GERBER file is replaced with the Assembly Drawing. Each product at Accurate Electronics has its own Traveler and which is also digitally archived after completion of
the production. Travelers consists of many pages however page 1 of Traveler is shown on below for this project.

7/30/2013

<table>
<thead>
<tr>
<th>Job Name</th>
<th>Transmitter PCB Rev A</th>
</tr>
</thead>
<tbody>
<tr>
<td>Customer</td>
<td>Syed Shayan Alam</td>
</tr>
<tr>
<td>SO No</td>
<td>19301</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Start Date</th>
<th>7/31/13</th>
</tr>
</thead>
<tbody>
<tr>
<td>Finish Date</td>
<td>8/7/13</td>
</tr>
</tbody>
</table>

### Stock Part Release

<table>
<thead>
<tr>
<th>Worker</th>
<th>Date</th>
<th>Start Time</th>
<th>End Time</th>
<th>Total Hours</th>
<th>Completions</th>
<th>Finished</th>
</tr>
</thead>
</table>

**Notes**

Load Through-Hole parts as shown in attached GERBER print. Make sure of POLARITY.

### Component Load

<table>
<thead>
<tr>
<th>Worker</th>
<th>Date</th>
<th>Start Time</th>
<th>End Time</th>
<th>Total Hours</th>
<th>Completions</th>
<th>Finished</th>
</tr>
</thead>
</table>

**Notes**

Program the machine and Run the product.

### Selective Solder

<table>
<thead>
<tr>
<th>Worker</th>
<th>Date</th>
<th>Start Time</th>
<th>End Time</th>
<th>Total Hours</th>
<th>Completions</th>
<th>Finished</th>
</tr>
</thead>
</table>

**Notes**

Solder Xbee, Make sure of orientation. Sensitive part/DO NOT WASH/USE NO-CLEAN SOLDER.

### Hand Solder & Touch-Up

<table>
<thead>
<tr>
<th>Worker</th>
<th>Date</th>
<th>Start Time</th>
<th>End Time</th>
<th>Total Hours</th>
<th>Completions</th>
<th>Finished</th>
</tr>
</thead>
</table>

**Notes**

Solder Xbee, Make sure of orientation. Sensitive part/DO NOT WASH/USE NO-CLEAN SOLDER.

### Inspection

<table>
<thead>
<tr>
<th>Worker</th>
<th>Date</th>
<th>Start Time</th>
<th>End Time</th>
<th>Total Hours</th>
<th>Completions</th>
<th>Finished</th>
</tr>
</thead>
</table>

**Notes**

Inspect the PCB per IPC CLASS 2.

Figure 3-8 Traveler from Accurate Electronics

70
The Stock department verifies all part with the part description provided by the customer stated on the BOM. Then the component is prepared and formed to ensure the Through-Hole lead does not hit the machine at any stage of production either for wave solder or selective solder machine. A wave solder machine is a fountain of solder. Solder is a metal alloy that melts at high temperature and joints the electronic component to the circuit board. Accurate Electronics uses many types of solders such as “SN63PB37” and “SN100C”. SN stands for tin and PB stands for lead and the following number is the ratio of mixture. For customers requesting RoHS (Restriction of Hazardous Substances) compliance the “SN100C” is used since it has no lead which is safe for environment. For this project “SN63PB37” will be used. Now comes the option of flux, a substance to avoid burning the alloy during the soldering process. Flux also have two major choices, first is the regular corrosive flux (Water Soluble Flux) which require aqueous cleaning (water wash) and the other is non-corrosive (No-Clean Flux) and does not require cleaning however leaves residue which some customer does not accept. For this project the choice of flux is to be decided by Accurate Electronics. The manufacturer chooses the No-Clean flux to be efficient for this project because the Xbee can not be washed.

The choice of producing the product through the wave or selective still remains to be answered. Wave solder are generally designed to run mass quantities while Selective solder machine is design to run small batches at perfection. Therefore Selective solder is chosen. The difference between Wave Solder and Selective Solder machine is shown visually below. Both machines are equipped with conveyors and pre-heaters to avoid thermal shocks to the product. Wave Solder machine contain pots of alloy solder where the mechanism helps to achieve smooth wave of melted alloy shown in Figure 3-10.
As the PCB rolls on the conveyor, the first process is to wet the PCB with flux in-order to flow the solder in the holes much smoothly, it acts like a catalyst to help solder flow more freely. Then comes the preheaters that heats the PCB at slower rate to reach approximately 240 degree celcius. Finally, the PCB touches the solder fountain at 260 degree celcius and the alloy fills the through-hole with the help of flux. At last the PCB exits the Wave machine.

**Figure 3-9 Wave Soldering Machine Process Illustration**

**Figure 3-10 Wave Solder Machine**
The downside of wave soldering machine is that the PCB contacts the wave and therefore solders the entire PCB whereever a hole is present. This causes all via holes to be filled up and the PCB encounters very high temperature. There are ways to prevent solder to fill hole, this method is known as masking. PCB masking is required when a PCB is double-sided which means components are present on both sides of PCB or in cases where solder is not required to a specific location. Types of solder mask and solders pallets are shown below.

![Image of types of masking materials](image)

**Figure 3-11 Types of Masking (Chemical-left, Tape-middle, Die Cut-right)**

The masking of a PCB takes time and therefore the efficient way of producing PCBs through wave is to use pallets. Pallets are fixtures that are design specific and non-compatible with other PCBs. Therefore, PCBs produced via wave solder requires higher cost at the time of first production run however the speed of production is a plus point which pays off later for the higher start cost.

![Image of pallets for wave solder machine](image)

**Figure 3-12 Pallets for Wave Solder Machine**
Selective soldering is another method and machine of soldering through-hole parts to the PCB. This machine is programmable which run on nozzle type soldering. The machine can be equipped with different types of nozzle depending on the PCB’s need. Selective soldering is modern way of soldering through-hole parts and requires no pallets and/or masking. The only setup time involved is programming the soldering locations. The machine is computer controlled and the nozzle travels from holes to holes soldering parts in to the through hole which is time consuming but accurate. Selective soldering method is perfect for prototype productions where the customer is unwilling to spend money for fixtures for wave solder or even Following are the visual descriptions of selective soldering machine that is being used at Accurate Electronics.

![Select Soldering Machine and Soldering Process](image)

**Figure 3-13 Selective Soldering (Machine-left, Soldering-right)**

This project is a perfect candidate for Selective Soldering and also for No-clean process since the Xbee can not be washed.
Following are the pictures for the final product built at Accurate Electronics.

Figure 3-14 Final Product (Receiver PCB)

Figure 3-15 Final Product (Transmitter PCB)
Chapter 4: Product Analysis

4.1 Design For Manufacturing Report

Accurate Electronics provides DFM (Design for Manufacturing) for every prototype in order to inform customer with issues faced by Accurate Electronics so that future run will avoid those mistakes. It is totally up to customer to implement those changes to the product. The DFM are generated by the Production Managers, Project Engineers and the Process Engineers. In simple words DFM report are recommended changes from manufacturer.

The DFM report presented by Accurate Electronics for this project states that the Stockroom department noticed that many components were loose packed and were not sealed properly with Humidity Indicators. Also the packaging was missing the Desiccants, a small pouch that absorbs humidity shown in Figure 4-1. Further the Stockroom department notices that the BOM documentation is missing the revision letter and the Xbee part number is missing from the BOM. The SMT department suggests converting all parts to SMT for higher efficiency for future run. The most important recommendation for next run is to request to add rails at the edge of PCB for the machines to handle product with a gap so that all parts can be machine soldered via selective. The last suggestion was to flip the DC Jack 180 degrees for easy connection to 9 volt battery holder.

![Figure 4-1 Desiccants](image)
The DFM report provided by Accurate Electronics is shown below for this project.

![DFM Report](image)

**Figure 4-2 Design for Manufacturing (DFM) Report**
4.2 Future Enhancements

The future enhancements for this project have enormous possibilities. The must-have enhancement includes the conversion to SMT (Surface Mount Technology) since it will significantly reduce the size of the PCB plus it will be more reliable. Second enhancement will be to reduce the size of battery and install it directly on PCB rather than external battery holder.

Switching the entire PCB to SMT is not that difficult however production of SMT PCBs are expensing since it requires setuping the SMT machines such as “Pick and Place”, Stencil Printer and Reflow Ovens. The time consuming machine setup time is involved with the “Pick and Place” machine this machine requires CAD data from the schematics to be transferred to the machine with X and Y co-ordinates to place the parts using robot. SMT machines at Accurates Electronics are capable of placing up to fifty thousand components per hour. Famous brands amongst these machines are manufactured by Fuji, MyData and Siemens. Following is a picture of SMT “Pick and Place” machine.

Figure 4-3 SMT Machine (MyData)
SMT parts are also sometimes called SMD (Surface Mount Devices). Many industries are switching designs to complete SMT components on their PCBs. However, some newer parts are still only available only on through-hole packages. Therefore, most of the industry include Aerospace & Defense products are still mixed production with combination of both SMT and Through-Hole packages. Following are the visual description of common SMT/SMD packages.

### SURFACE MOUNT PACKAGE SIZES

<table>
<thead>
<tr>
<th>Package type</th>
<th>Size in inches</th>
<th>Size in mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>0201</td>
<td>0.024&quot; x 0.012&quot;</td>
<td>0.5 mm x 0.3 mm</td>
</tr>
<tr>
<td>0402</td>
<td>0.04&quot; x 0.02&quot;</td>
<td>1.0 mm x 0.5 mm</td>
</tr>
<tr>
<td>0603</td>
<td>0.063&quot; x 0.031&quot;</td>
<td>1.5 mm x 0.8 mm</td>
</tr>
<tr>
<td>0805</td>
<td>0.08&quot; x 0.05&quot;</td>
<td>2.0 mm x 1.25 mm</td>
</tr>
<tr>
<td>1206</td>
<td>0.126&quot; x 0.063&quot;</td>
<td>3.2 mm x 1.6 mm</td>
</tr>
<tr>
<td>1210</td>
<td>0.12&quot; x 0.10&quot;</td>
<td>3.2 mm x 2.6 mm</td>
</tr>
<tr>
<td>2020</td>
<td>0.20&quot; x 0.20&quot;</td>
<td>5.08 mm x 5.08 mm</td>
</tr>
<tr>
<td>2512</td>
<td>0.25&quot; x 0.12&quot;</td>
<td>6.35 mm x 3.0 mm</td>
</tr>
</tbody>
</table>

Figure 4-4 SMT Common Packages
ICs (Integrated Circuits) are also have their package names. Following are the visual aid and description on the type of packages. Note that parts with straight leads are through hole while the others are SMT packages.

Figure 4-5 Integrated Circuit Packages
The most complicated SMT packages are considered to be lead less parts. These parts have leads on the bottom of part and can only be inspected with X-ray machines to see if the alloy completed wetted (melted) under the part. Among these types of parts there are two common ones that are called QFN (Quad Flat No Leads) and BGA (Ball Grid Array) packages. Following are visual description of BGAs and QFNs.

Figure 4-6 Ball Grid Arrays (BGA)

Figure 4-7 Quad Flat No Lead (QFN)
SMT process requires printing of solder paste on to the PCB via Solder Paste Printers. These printer uses metal alloy paste that are paste even in normal room temperature. The printer requires Stencils, a metal sheet with laser cut precise holes to spread the alloy paste on to the PCB with high precisions. After the printing of solder paste on the PCBs, the PCBs are sent to the “Pick and Place” machines to place the parts on top the paste, again with high precision.

Figure 4-8 SMT Placement Illustration

Figure 4-9 Paste on PCB
Stencils are also design specific and can not be used on different PCBs. Therefore, if an ECO (Engineer Change Order) is released requiring to change the PCB in any matter will waste the Stencil. Design Engineers are required to take in account that if they change the PCB design then they will be scrapping many manufacturing tools along the way. Following is a sample picture of an stencil along with the printer.
Solder paste printing then “Pick and Place” are followed by the step called the Reflow of solder paste. During this process the PCB is sent to the rolling oven where the PCB encounters high temperature slowly rising to the peak temperature where the solder paste is completely liquid and followed by dropping of temperature. The end result is mostly a perfectly wetted solder shown below.

![Solder Wetted](image)

*Figure 4-12 Solder Wetted (0402 Package – left, BGA Package – right)*

The reflow oven is equipped with many oven zones and the heat is set on each zone. The data sheet for the solder paste provides a general recipe to accomplish nominal wetting of the solder paste. Following is the graphical temperature guide for the oven to be configured at.

![SMT Reflow Oven Recipe](image)

*Figure 4-13 SMT Reflow Oven Recipe*
Reflow ovens are designed to achieve the temperature environment shown in Figure 4-13 with slow increments of heat by using many temperature zones. These ovens are long due to many zones of heaters. The recipes are programmed on the computer attached to the programmed and zones are set with minimal increase to the peak temperature. The main concern is to avoid the PCB to encounter any thermal shock since SMT parts are way more sensitive compared to through-hole parts. Programming the reflow ovens requires skills and techniques since there are many factors such as conveyor speeds, thermal dissipation, solder paste type, and much more. Following picture shows a 9 zone reflow oven. Each zone has two heaters, one on the top and another on the bottom.

Figure 4-14 SMT Reflow Oven (15’ long with 9 zones)

![Figure 4-14 SMT Reflow Oven](image)

Figure 4-15 Zone Control Window for Reflow Oven

![Figure 4-15 Zone Control Window for Reflow Oven](image)
Overall, the SMT enhancement to this product will definitively improve the reliability however at the prototype level it was wise to assemble the product with only through hole since it brought the cost of the product low. The first step to switch to SMT involves looking for equivalent parts in SMT packages. For instance on this product Xbee and ATmega are the most critical components since others parts are definitently available in SMT packages. Following are the visual comparison of through-hole and SMT packages of Xbee and ATmega IC.

Figure 4-16 Xbee SMT Package (Through-Hole – left, SMT – Right)

Figure 4-17 ATmega SMT Package (Through-Hole – left, SMT – Right)

The above confirms that this project can be confirmed to SMT. The product then will be more reliable can be produced in mass quantity at cheaper cost.
Chapter 5: Compliances

5.1 Standards and Certifications

Accurate Electronics is compliant with many certifications and they follow IPC standards for all of their electronics assemblies. Currently Accurate Electronics is ISO 9001, AS 9100, ISO 13485, ITAR, UL, GMP, and RoHS Compliant. All these compliant are very important for current electronic industries. ISO 9001, which stands for Quality Management, is especially designed to manage and control documentation in a facility. The ISO 9001 certification is required by many customers before they can even add a manufacturing company to their AVL (Approved Vendor List). ISO 9001 standard confirms that the product a customer receives is always consistent by implementing Quality Management Principles provided by ISO.

Aerospace & Defense customers are obliged by their companies to only order electronic assemblies from AS 9100 certified facilities. This certification is widely compulsory in aerospace and automobile industry. AS9100 is recognized by FAA (Federal Aviation Administration) as quality standards for aerospace standards. Under this certification a facility is required to have higher degree of quality while also maintaining higher traceability of a product.

ISO 13485 is a standard that is recognized by the medical industry where every detail of manufacturing is crucial since the product designed involves human factor. Medical products requires to have higher rate of traceability along with COC (Certification of Compliance) for every single item used in manufacturing of a medical device. These certifications bring more confident to customers ordering electronic assemblies for their respective industry.
ITAR stands for International Trafficking of Arms and Regulations this is a regulation controlled by the United States government. Any facility recognized by ITAR confirms that any data for manufacturing including BOM, product detail, assembly drawing, etc will not be exported outside of United States for manufacturing or for any other purpose. All tiers dealing with an ITAR product needs to be recognized by ITAR. The ITAR is enforced by Department of Defense and protected by Department of Homeland Security. ITAR customers are usually tier for military and aerospace products.

UL stands for Underwriters Laboratory which is a certification to produce products under safety regulations along with validation of a product which further includes testing and inspection. UL is also one of the approved companies to perform OSHA (Occupation Safety and Health Administration) safety testings. Many electronic products are UL certified and these product goes through strict scrutiny in which every single item is check against BOM and assembly drawing.

GMP stands for Good Manufacturing Practices. This is not a certification but rather a self paced practiced and can be adopt by any manufacturing facility. The GMP is focused toward the manufacturing process and quality of a product. It states that each manufacturing process should be clearly defined, controlled and consistency is maintained throughout the product life. Further, any kind of changes in process shall be evaluated before implementation. At last the quality department is to control and provide preventive and corrective actions for any concerns.

RoHS stands for Restriction of Hazardous Substances and it is not a certification. It is a practice to built product with lead-free substances. Also the manufacturing has to
control that lead-free product are not contaminated with lead products and this is easily done by segregating production lines for RoHS products.

Overall, electronic assemblies are manufactured under many standards and regulations depending on the location where it is being built. Many consumer products are being built in China due to lesser and lesser regulations from the government and from the customers. However, the US electronics manufacturing market still holds strong due to higher quality products and consistency for repeat order. It all breaks down toward how robust is the manufacturing process.

Figure 5-1 Accurate Electronics (Logo & Certifications)
5.2 Quality Management System

Quality Management System is the most important part of any manufacturing facility where the motive is to achieve or maintain robust manufacturing processes. The QMS requires controlling and evaluating the manufacturing process while maintaining higher quality along with traceability. To achieve robust quality system, manufacturing facilities rely on database software that tracks production at every possible stage.

Accurate Electronics uses DBA Manufacturing database software. This software is equipped with sophisticated features such as MRP (Material Requirement Planning), Travelers (process routings) and many other features to help manufacturing. Accurate uses MRP feature on all projects since the stockroom is more reliable with their current quantities therefore confirming the parts on hand is important. DBA is divided into many sections such as Item, Descriptor, BOM (Bill of Material), Sales Order, Job, Purchase Order, Note, Customer, Supplier, etc. Following is a screen shot for the GUI (Graphical User Interface) of DBA software.

![Database Software GUI](image)

**Figure 5-2 Database Software GUI**
The database system is part of QMS and it helps by keeping the past, present and future of an electronic assembly. Customer BOMs are transferred into DBA. The Item section currently has 12,310 parts. All these parts were assigned custom part numbers from Accurate Electronics. These numbers are referred as “AE#'s” and used internally only. Assigning “AE#” is considered good, the benefit is that long part numbers are referred with only 6 digit numbers rather than long manufacturing part numbers.

Travelers are produced using the DBA software which consists of process routings and steps. Each routing is labeled and consists of brief description for the process. The traveler also requires signature after completion at each step along with the date and time. The main purpose of the traveler is for traceability and also for tracking WIP (Work In Progress). Every product is required to have a Traveler before starting of manufacturing. To further assist the manufacturing process Engineers are required to create MPI (Manufacturing Process Instructions). These instructions are very detailed and contain useful information regarding the assembly whether it is testing a product or building a product. A copy of Traveler for this project can be seen in Figure 3-8.

Quality Management System is required to fulfill the ISO 9001, AS9100, ISO13485, and UL requirements. To sum up all the requirements, the QMS should be able to record all production status, follow the assembly requirement per customer and by IPC standards, use only parts listed on the customer’s BOM, and in non-coforming situations come up with CAPA (Corrective and Preventive Action) reports. AS9100 also requires that all tools used for measurement shall be calibrated properly with certifications. To manage all the requirements QMS is equipped with all types for forms. These forms hold records for many manufacturing tasks.
QMS was also responsible of establishing receiving area. Currently Accurate Electronics’ receives all incoming through the receiving area. This area is segregated and nothing comes into company without bypassing the Receiving Inspector. The Receiving Inspector is trained and certified by the quality director. The inspector verifies the parts with its proper documents such as Drawings, Datasheets, and or Purchase Orders. On the other hand, Receiving Inspector is also in charge of receiving the RMAs (Return Merchandise Authorizations) also route it through quality personnel which later gets transferred to the engineers for proper evaluation of returned goods. Furthermore, Receiving Inspector is also authorized to return Non-Conforming incoming items. However, all Non-Conformance items are required NCMR (Non-Conforming Material Report) to be filled. Figure 5-3 shows a copy of NCMR form filled at Accurate Electronic for an RMA of five PCBs.

Overall, QMS provides full quality control and train personnel to follow the right procedures of correct manufacturing process. It is a system that makes the facility under rules and regulations. QMS also trains inspectors that are responsible to inspect the quality of final product before shipping out to the customer. Accurate Electronics uses IPC standards for all electronics assemblies. Therefore all inspectors are trained with IPC certification and the QMS holds training records. Via QMS, records are collected at all stages of inspection and RMAs. From the collected data, metrics are established and thoroughly studied every quarterly meetings to ensure high quality standards. QMS is equipped with many types of form for all kinds of manufacturing needs and a snap shot of all forms are shown in Figure 5-4.
Following is a copy of NCMR form filled at Accurate Electronics for RMA of five PCBs being returned to Accurate Electronics for repair. Note that this form can also be used for internal non-conforming and incoming non-conforming items.

![NCMR Form]

Figure 5-3 NCMR (Non-Conformance Material Report)
Following are the list of some forms that are available under the Quality Management System at Accurate Electronics. Note under ISO quality systems, all form are assigned form number.

Figure 5-4 QMS Forms at Accurate Electronics
Chapter 6: Conclusion

The topic of Electronic Assemblies is a very dense and cannot be fully covered in a single project report. However, this report covers all the major steps to enlighten engineers with the making of electronic assemblies. This report helps understanding the process for manufacturing of electronic assemblies weather it is small as a cell phone or big as an airplane. All assemblies follow the same criteria of manufacture some with more high regulations and others with fewer regulations especially for consumer electronics.

This report covers the design process where the ATmega is programmed followed by the prototype phase. During this design phase the wireless communication methods are also discussed and explained that how important is to configure wireless devices to be exactly same. Further, the design is converted to schematics and explained that todays’ engineers tends not to start from scratch rather starts for sub modules as their schematics and completes by modifying the schematics to their needs. After the creation of schematics the process of PCB layout is elaborated in detail and also how the software can be freely acquired. The process of manufacturing electronic assembly is also explained. The report also explains how to choose a manufacturing facility for the design needs. The files that are required to manufacture PCBs are also shown in detail. Manufacturing process and how an electronic manufacturer operates with all the regulations are also described in great detail. All in all, this project teaches an engineer how an electronics assembly is built along with future enhancements that can make an electronic assembly more reliable.
References


<http://arduino.cc/en/Main/arduinoBoardDuemilanove>

<http://www2.ensc.sfu.ca/people/faculty/dill/COURSES/ECAD/EAGLE-2-BOM.pdf>


<http://www.faludi.com/itp_coursework/meshnetworking/XBee/XBee_firmware_upgrade.html>


Appendix: Source Codes

//Transmitter
int val;
int new_val;

void setup()
{
    // start serial port at 19200 bps
    Serial.begin(19200);
}

void loop()
{
    // read analog input
    val = analogRead(0);

    // remap values
    new_val = map(val, 0, 1023, 253, 0);

    Serial.write(254);  //SYNC char
    Serial.write(new_val);
    delay(new_val);
}

// RECEIVER
byte incomingByte,pot;
void setup()
{
    // start serial port at 19200 bps
    Serial.begin(19200);

    pinMode (5, OUTPUT);  // set led pin 5 to output
    delay(1000);
}

void loop()
{
    if (!Serial.available())
    {
        incomingByte = Serial.read();
        Serial.print(int(incomingByte));
        Serial.println();
    }

    if ((int(incomingByte) == 254))
    {
        pot = Serial.read();
        Serial.print("Pot = ");
        Serial.println(int(incomingByte));
        Serial.println();
    }

    analogWrite (5, incomingByte);  //pwm led according to pot values
}