IE.3511 – Safety and reliability

Project Report on

Metal Detector PCB development as a Military Application

Submitted by:

Vinay GARADE –61086
Chinmay kaul–60955
Qiping XU –60980
Mingyang Li –61048
Mohamed Yasar Arafath Kaja Mohaideen – 60952
Aravindhan UVARAJ –60976
Ajay kumar. Chittimilla–60482
Fábio Ramos – 61449

Index

1. Introduction

A metal detector is a portable electronic instrument which detects the presence of metal in its vicinity. Metal detectors are useful for finding metal inclusions hidden within objects, or metal objects buried underground. They often consist of a handheld unit with a sensor probe which can be swept over the ground or other objects to detect the metal. When a metal comes in vicinity of the detector the sound of the buzzer in the detector changes thus notifying the presence of metal. The metal detectors are used in wide number of metal detecting applications in industries, security services in commercial and domestic sectors, archaeology, civil engineering , defence services and etc. considering this wide number of applications we have designed a Metal detector Circuit for military applications like detecting land mines hidden inside the ground, hidden bombs and weapons. The detector circuit designed is a induction balance type metal detector circuit where the change in the inductance of the search coil changes the output of the circuit. The designed PCB board has a power supply unit followed by a metal detection circuit using IC555 and followed by amplification circuit. The Military devices are exposed to harsh climatic conditions, chemical exposures, vibrations hence we have to consider these constraints while designing the PCB. To follow the military guidelines for PCBs the standards such as IPC6018 Class 3 MIL-PRF-31032 -55110 are referred.

2. Circuit functioning

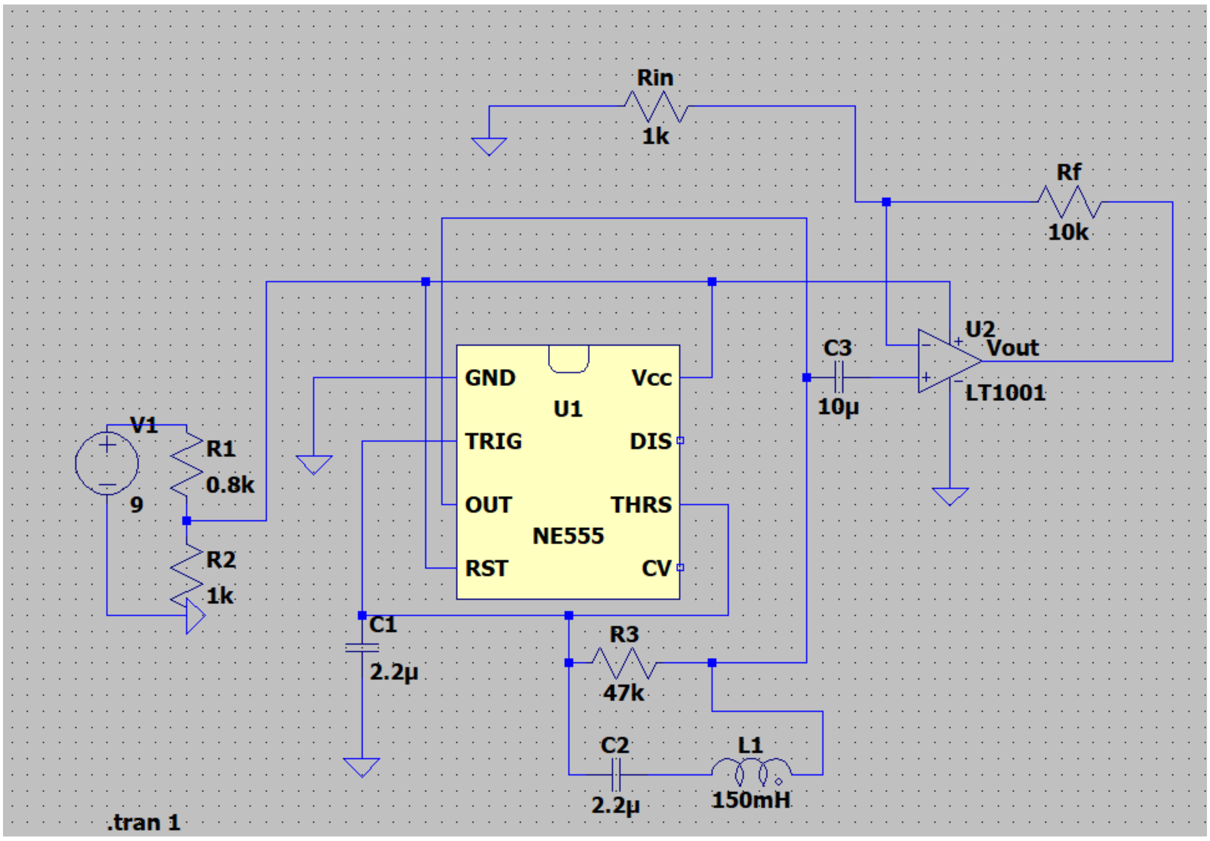

Fig 1. Metal detector circuit simulation in LT spice

The circuit basically consist of following three parts:

Power supply circuit:

The input voltage of 9 volt from external voltage source is converted into

5Vdc for the IC555. The voltage conversion is done by using a Voltage divider circuit. The resistance R1 and R2 are selected in such a way that the output voltage is 5Vdc.

Metal detecting circuit:

The Timer LMC555 IC is wired as a Mono stable multivibrator. Monostable multivibrator is characterized by giving high output when there is low pulse signal to the input trigger pin. The duty cycle of output pulse from IC 555 is proportional to the Inductor L1 and Capacitor C2 along with resistor R3 form a RLC circuit and are connected to the trigger pin of IC 555. Here the inductor we use is an Air core. When current flows through the Inductor a magnetic field proportional to the current flow develops around it. When a metal comes in vicinity of this magnetic field by the principle of electromagnetic induction emf is induced in the metal and eddy currents start flowing through it. This flowing current through the metal create a magnetic filed around it which opposses the magnetic field created by inductor as per lenzs law. As a result inductance of the inductor increase thus the impedance of overall RLC circuit changes. This results in increase in flow of current and Capacitor C1 starts charging quickly. Feeding the trigger pin of IC 555.This change in inductance results in change in duty cycle of output pulse that goes to the amplifier.

Amplification:

The output of the metal detecting circuit is amplified using a non-inverting amplifier built by using LM741 opamp. A buzzer when connected to the output of the amplifier changes the sund when metal is detected. The output can also be used further for interfacing the circuit with other peripheral devices

Simulation:

The circuit was simulated using LTspice before implementing on PCB. To simulate the change in inductance due to presence of Metal, we changed the inductance value from 50 to 150mH. In actual PCB the inductor is a wounded coil that can be connected separetly via connectors on PCB(Fig 2).

3. Component selection

Capacitor- 2.2µF, 10µF

- Specially designed for crossovers and electronics
- Extra-thick tinned copper leads

Parameters	2.2µF	10µF
Tolerance	±10%	±10%
Dissipation Factor	Max 0.001 (0.1%) @ 1 KHz	Max 0.001 (0.1%) @ 1 KHz
Operating Range	-55°C to 125°C	-55°C to 125°C
Dielectric Strength	150%	150%
Insulation Resistance	10,000 MO x µF	10,000 MO x µF
Life Test	1,000 hours @ 85°C, 150% rated voltage	1,000 hours @ 85°C, 150% rated voltage
Warranty	5 Years	5 Years
Price per unit	€ 0.2	€ 0.41

LMC555 IC

These devices are precision timing circuits capable of producing accurate time delays or oscillation. The output circuit is capable of sinking or sourcing current up to 200 mA. Operation is specified for supplies of 1.5V to 15V. With a 5-V supply, output levels are compatible with TTL inputs.

Parameters	Values
VCC (Min) (V)	1.5
VCC (Max) (V)	15
Iq (Typ) (uA)	2000
Operating temperature range (C)	-40 to 125
Price per unit	€1.2
Temperature stability	0.005% per °C

Resistors

Resistance (Ω)	Parameters	Values
1K	Power data	166 mW (1/6 W)
	Tolerance	1%
	Temperature coefficient:	- 350 PPM / C, + 500 PPM / C
	operating temperature	-55°C to 155°C
	Unit Weight	155.833 mg
	Price per unit	€ 0.09
10k	Power data	250 mW (1/4 W)
	Tolerance	1%
	Temperature coefficient:	- 350 PPM / C, + 500 PPM / C
	operating temperature	-55°C to 155°C
	Unit Weight	513.611 mg
	Price per unit	€ 0.09
47k	Power data	500 mW (1/2 W)
	Tolerance	1%
	Temperature coefficient:	- 350 PPM / C, + 500 PPM / C
	operating temperature	-55°C to 155°C
	Unit Weight	589.830 mg
	Price per unit	€ 0.11

LM741

The amplifiers offer many features such as overload protection on the input and output, no latch-up when the common- mode range is exceeded, as well as freedom from oscillations.

Parameters	Values
Number of channels	1
Total supply voltage (Max)	22Vdc
GBW (Typ) (MHz)	1
Slew rate (Typ) (V/us)	0.5
Vos (offset voltage @ 25 C) (Max) (mV)	3
Iq per channel (Typ) (mA)	1.7
Vn at 1 kHz (Typ) (nV/rtHz)	30
Operating temperature range (C)	−55°C to +125°C
Offset drift (Typ) (uV/C)	15
Input bias current (Max) (pA)	210000

CMRR (Typ) (dB)	90
Output current (Typ) (mA)	25
Price per unit	€0.85

PCB

Layer count	4 Layers
Quantity Requirements	1 - 10000+
Materials	FR-4
Plating Finish	Electrolytic Hard Gold
Certifications	IPC6018 Class 3 MIL-PRF-31032 -55110 / ISO 9001:2008
Board Thickness	.031" / .8mm"
Copper Weight	3 oz. Inner / 15 oz. Outer
Trace/Space	5/ 5 Mils
Price	€5.83

4. PCB design

The PCB is designed using the KiCad software. The software has its own predefined library for component selection which makes the designer work easy. The MIL-PRF-31032 -55110 standards of military grade circuit board were followed while component selection and PCB designing. The PCB designing aspects and constraints faced while designing are discussed below:

4.1 Designing:

The PCB is a four layer copper PCB of dimension 45mm x 45mm and height of 14.1 mm. The topmost layer is used for control signal connections, the second layer for 5Vdc and 9Vdc power connection and the third layer for ground. The resistors and capacitor are SMD components. The IC555 and LM741 are SMD socket based IC of 8 pins each. The base of these IC is a SMD socket and the ICs can be detached from the base. These provide flexibility of IC replacement in case of IC damage thus saving the cost of replacing the whole PCB.

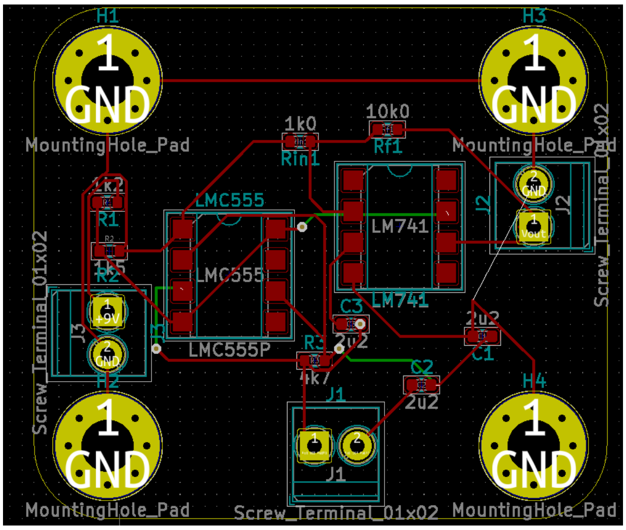

Fig 2 PCB layout after routing the connections in Kicad

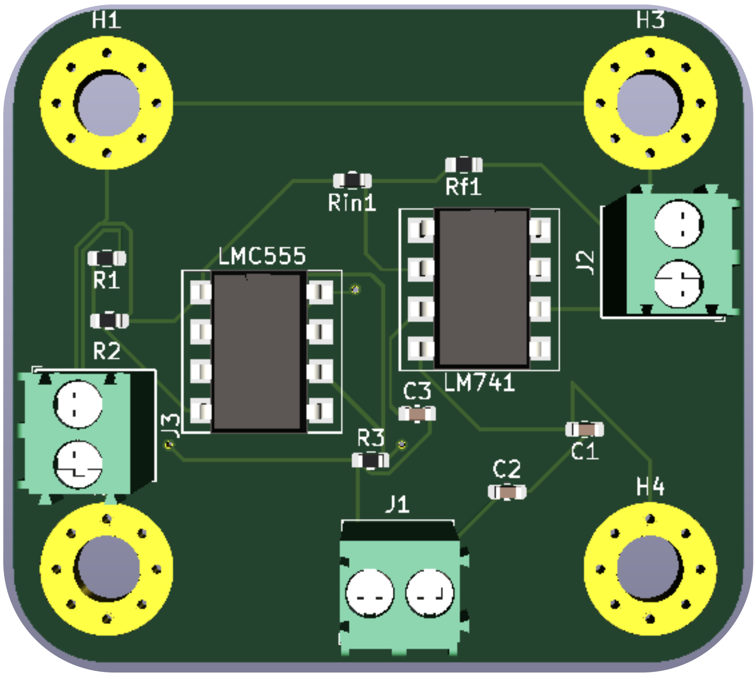

Fig 3 The 3-d view of PCB in Kicad software

The PCB has Three Screw terminal connectors as shown in Fig 3. The connector J1 is for connecting the search coil, The output of amplifier is terminated at J2 and can be used to interface other circuit or a buzzer to this PCB and J3 is for connecting 9Vdc power supply. Four mounting PAD holes with inner diameter of 4.3mm are used. The small via holes in the mounting hole pads are designed to electrically connect the mounting hole pad to the pad on the other side of the board.

4.2 Constraints:

The constrains faced during the designing process were as follows

- The component spacing on the PCB has to be done in such a way that the track length should be minimum and track routing should not be congested.
- Respecting the minimum spacing between components to 2mm and designing a PCB in 45mm x 45mm so that it should occupy less space when embedded into a system.
- The Track overlapping had to be kept as minimum as possible while routing the Tracks. This is because while manufacturing the track has to be routed through digging a hole from one side to the other side of PCB at the point of track intersection.

5. PCB Manufacturing

The selection of a contractor for manufacturing a PCB as per the demands is based upon the cost and also the contractor should comply with the required standards for manufacturing Military Grade PCB. Focusing on this Two aspects we have selected RayMing a PCB manufacturer from china.

Sub Contractor:

RayMing is a PCB manufacturer in China and has over 15 years of PCB Manufacturing experience and delivers the best quality printed circuit boards. they can build up to 36 layer prototype PCB, and up to 12 layer mass production PCB and provide widely PCB technology support like HDI PCB, Flexible PCB, Rigid flex PCB, also more PCB material option like Rogers PCB , Teflon, Arlon, Metal core PCB. Moreover, Ray PCB's capable of providing turn-key and partial turn-key printed circuit board assembly services. For full turnkey, care of the entire process, including manufacturing of Printed Circuit Boards, procurement of components, PCBA Testing, continuous monitoring of quality and final assembly. Whereas for partial turnkey, we should provide the PCBs and certain components, and the remaining parts will be handled by them.

Manufacturing Process:

Step 1: PCB Design and Gerber file

The circuit designer draws schematics in CAD software that will be used to layout the PCB design. The designer must coordinate with the PCB manufacturer about the software used to layout the PCB design so that there will be no compatibility issues. After the PCB design has been accepted for fabrication, the designer will generate a file from the design that is accepted by the PCB manufacturers. This file is called "GERBER file". The Gerber file is the standard file used by majority PCB fabricators to show the components of PCB layout like copper tracking layers, and solder mask etc. The Gerber file is the 2D vector image file. The extended Gerber delivers perfect output.

Step 2: GERBER to Photo Film

The special printers used to print the PCB photo film is called "plotter". These plotters will print circuit boards on the films. These films are used to image the PCB. Plotters are very accurate in printing technology to give a highly detailed film of PCB design. The plastic sheet come out from the plotter is the PCB printed with black ink. In case inner layer, the black ink represents the conductive copper track while the blank portion is non-conductive part.

Step 3: Inner layer printing: Photo resists and Copper

Now these photo films are printed on copper foil. The basic structure of PCB is made of a laminate board. The core material is epoxy resin and glass fiber known as substrate material. Laminate receives the copper that construct the PCB. Copper is covered on both sides. The process involves removing away the copper to show up the design from the films. The photo resist is made of photo sensitive chemical which will harden when UV radiation is applied. The black ink on the photo film will block the UV light thus preventing the copper underneath it and not hardening the photo resist under the black ink traces. The clear area will pass the UV light thus hardening the excess photo resist which will be removed. The board is then washed with alkaline solution to remove excess photo resist.

Step 4: Layer Alignment and Optical Inspection

After all the layers are ready from above mentioned steps, they are then aligned upon each other. This can be done by punching registration holes as mentioned in previous step. The technician will place all layers in the machine known as "optical punch". This machine will punch the holes accurately. Layers ones placed and error occurred cannot be reversed. The automatic optical inspection machine will detect any defects using laser and compare the digital image with Gerber file.

Step 5: Layer-up and Bonding

At this stage all the layers including the outer layer will be bonded together one upon each other. All the layers will be stacked upon substrate. The outer layer is made of fiber glass "pre-impregnated" with epoxy resin called prepreg. Top and bottom of the substrate will be covered with thin copper layer having the copper trace etching. A heavy steel table with metallic clamps is used for bonding/pressing layers. More prepreg sheets are placed similarly and finally aluminium foil complete the stack. A computer will automate the process of bond press, heating the stack and cooling at controlled rate. Now the technician will remove the packing pins and pressure plates to unpack the stack.

Step 7: Drilling

Now it is time to bore the holes in stack PCB. The precision drill achieves 100 microns diameter holes, with great accuracy. This drill is air driven drill has a spindle speed around 300K RPM. But even with this speed the drilling process takes time because each hole takes its time to be bored perfectly. The X-ray based identifier spots the drill locations accurately. The drill files are also generated by PCB designer that is given to PCB fabrication house at earlier stage. This drill file determines the micro movement of drill and spots where to bore holes.

Step 8: Plating and Copper Deposition

After cleaning, PCB now treated with chemical deposition. During this a thin layer (1 micron thick) of copper is deposited over the surface of the panel. The copper flows into the drilled holes. The walls of the holes are entirely plated with copper. The whole process of dipping and removal is computer controlled.

Step 9: Outer Layer Imaging

Same as for the inner layers, the photo resist is applied to the outer layer, the prepreg panel and black ink film joined together is now blasted with UV rays in yellow room. The photo resist hardens. The panel now goes through the machine to wash away hardened resist protected by black ink opacity.

Step 10: Plating of Outer Layer:

Electroplate panel with thin layer of copper. After initial copper plating, the panel gets tin plating, which allows removal of all the copper left on the board. The tin prevents the part of the panel required to remain enclosed with copper during the etching stage. Etching eliminated undesired copper from the panel.

Step 12: Solder Mask Application

After cleaning the panels and epoxy solder mask ink will cover the panel. The UV radiation is imposed on board that goes through the solder mask photo film. Covered sections stay unhardened and will be removed. The board is now put in oven to heal solder mask.

Step 14: Silkscreen

The PCB is in its final stage receives ink-jet printing/writing on surface. This is used to indicate important information related to the PCB.

Step 15: Profiling

The last step is to cut board from the original panel. A router is used for this purpose and it creates small tabs along the board edges which enable board to easily pop out from the panel.

Step 16: Testing

Final stage is electrical testing of final PCB. The automatic process validates the functionality of PCB to match the original design.

6. Testing and Inspection

To identify and address faults and bugs such as short circuits, opens, poor soldering, functional issues, we define the methods of testing PCB throughout the design and manufacturing processes. This step can avoid situations where designers and manufacturers realize that the product is faulty at the last minute, while the boards are in full production.

Generally, the testing involves verifying design features in terms of visual, structural, electrical and functionality. In most cases, there are various techniques for testing each of these areas. The common methods include:

- In-circuit testing (ICT)

- Automated optical inspection (AOI)

- Automated X-ray inspection (AXI)

-Environmental tests

In-circuit testing will work for most basic circuits, but as the level of complexity and component density increases AOI and AXI inspection techniques are used.

6.1 In-Circuit testing(ICT)

There are two major types of test equipment used in ICT: Flying probe and Bed of nails fixture.

Flying probe:

A flying probe is the machine used for the flying probe testing method. With software that suitable for the PCB layout, the flying probes check the signal one by one. The whole testing time is about 30 sec.

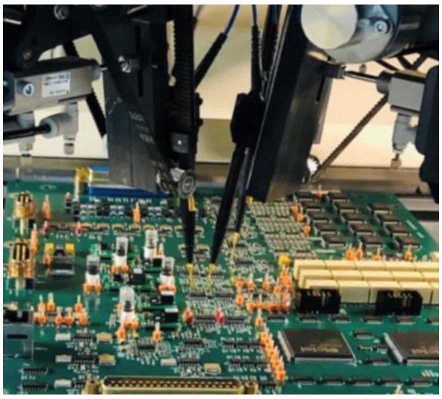

Fig 4: Flying probe testing technique

Bed of nail:

Bed of nail is a specific tool for fixture test. Each type of PCB has a bed of nail. This testing method always needs 5 to 10 seconds for one PCB. Because it is necessary to make a test fixture for each type of PCB, which is not economical for PCB prototypes.

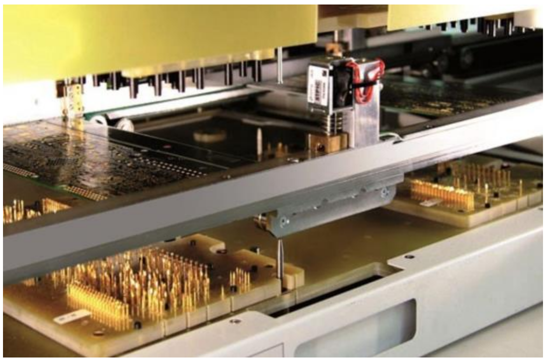

Fig 5: Bed of nails testing technique

They both have advantages and disadvantages. The flying probe test may be cheaper at initial cost, the bed of nails fixture test is a higher cost-effective for batch orders.

The flying probe has no need for fixed testing tools, and it has short delivery time, which is suitable for PCB prototyping. According to our current situation, we choose Flying probe as electrical testing.

6.2 Automated Optical Inspection (AOI)

PCB optical inspection is the use of a camera to inspect a printed circuit board. The most common utilization is automated optical inspection (AOI), which is performed during board assembly to inspect solder joint quality, component shifts, tombstoning and other issues that may require rework.

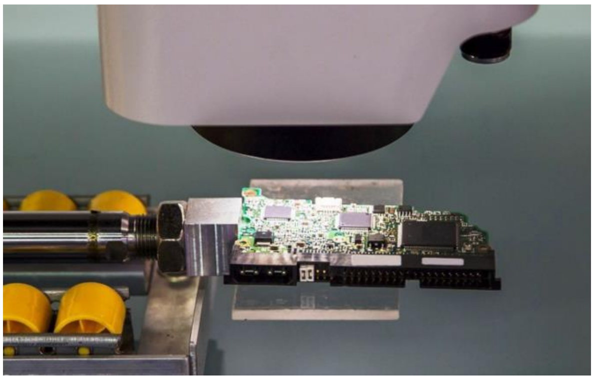

Fig 6: Automated Optical Inspection

Other than performing tests on the PCB under assembly, the AOI method can monitor the manufacturing process. Using the technology in the pick and place machines enables us to track the processes in real-time, and correct assembly defects such as potential component misplacement and misalignment.

6.3 Automated X-ray Inspection(AXI)

Automated Optical Inspection is only applicable for defects that are relatively easier found such as open circuits, solder bridges, solder shorts, insufficient solder and excess solder. However, because pins are hidden under the chip package, it's difficult to test them only based on light. That's why Automated X-ray Inspection (AXI) is required.

Belonging to the same structural test and inspection category, AXI shares the same principle with AOI, that is, inspection is carried out through capturing images. The difference between them lies in the source in that AOI depends on light source to capture images while AXI on X-ray. Materials absorb X rays according to their atomic weight proportion. Materials made of heavy elements get more X rays absorbed while those made of light elements get fewer. As a result, materials absorbing more X rays are displayed more obvious or darker in their images than those absorbing fewer rays. When it comes to PCB, solder joints are made of materials with heavy elements while other parts such as most packages, silicon ICs and component leads are made of materials with light elements. Therefore, high-quality solder joints should look darker or more obvious on images than other parts.

Based on its working principles and capabilities, AXI is applied to test circuit boards containing components with array-style packaging or fine pitch packaging such as BGAs. AXI is usually placed in the assembly process, just after the last soldering no matter wave soldering or reflow soldering. Furthermore, AXI is usually applied coupled with ICT and in order to obtain optimal inspection results.

6.4 Environmental testing

Environmental test are qualification tests that carried on Certain number of PCBs. A series of environmental tests can include shock, vibration, and mechanical tests. These tests should be performed in accordance with the IEC 60068 environmental testing standards. These tests are intended to ensure a new product can survive under when stored or transported at extreme temperatures and humidity, as well as in the operational environment. The test are carried on basis of following environmental constraints:

- Electrical
- Climatic
- Mechanical

6.4.1 Electrical

The electrical constraints include ESD (Electrical Static Discharge). ESD is a common EMC phenomenon. Static electricity refers to the instantaneous voltage generated by static charges accumulated when a specific substance touch and rub with other substances. Usually, the instantaneous voltage generated by static electricity is extremely high (>1kv), and the damage to the circuit is devastating and permanent.

There are 2 conditions where the ESD will show up: human body model (HBM) and machine model (MM). Here we have considered the HBM condition.

The HBM simulate the condition that the people carry the static electricity. It is following the standards:

MIL-STD-833C Method 3015.7

EIA/JESD22-A114-A (JEDEC, 1997)

EIA/JESD22-A114-A (JEDEC, 1997)

Here is the electrical model for testing:

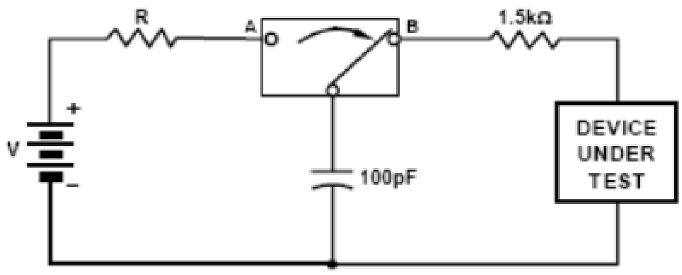

Fig 7 Electrical model for ESD Testing

We have followed the standard of MIL-STD-833C method 3015.7. If the max voltage is lower than 2kV then it is Class-1. 2k~4kV for Class-2. 4k~16kV for Class-3.

EOS usually occurs on semiconductor components. When the current or voltage of the electronic component exceeds the maximum value of the device, the EOS will happen. EOS may cause thermal damage to the entire device or one device. When the device is running with a high current or high voltage, the resistor will continue to heat up and the temperature will be too high, it will be causing damage to the components.

6.4.2 Climatic

Humidity and Corrosion

Many PCBs will be deployed in a humid environment, thus a common test for PCB reliability is a water absorption test. In this type of test, a PCB is weighed before and after being put in an environmental chamber with controlled humidity. Any water that adsorbs onto the board will increase the board's weight, and any significant change in weight results in disqualification.

When these tests are performed during operation, the exposed conductors should not corrode in a humid environment.

Thermal Shock and Cycling

Thermal testing is normally performed separately from humidity testing. These tests involve repeatedly changing the board temperature and examining how thermal expansion/contraction affect reliability. In thermal shock testing, the board is rapidly moved between two extreme temperatures using a 2-chamber system. The cold temperature is usually somewhere below freezing, and the high temperature is normally above the glass transition temperature for the substrate (above ~120 °C). Thermal cycling is performed using a single chamber, and the temperature is changed by several from one extreme to the other at 10 °C per minute.

In both tests, the board will expand or contract as the board's temperature changes. During expansion, high stress will be placed on the conductors and solder points, which accelerates the lifetime of the product and allows mechanical failure points to be identified.

6.4.3 Mechanical

Vibration testing

The PCB vibration test is used to test the reliability and anti-vibration ability of the product. In the process of transportation and use of the product, the vibration is unavoidable, and vibration testing is an important part of PCB testing.

We need to simulate various vibration environments in the laboratory. The experiment set-up is an air-cooled shaker (LDS V555 with a maximum sine force of 939 N, a maximum acceleration of 100g with $g = 9.81 \text{ m/s}^2$ and usable frequency range between 5Hz and 6300Hz). The maximum weight that the shaker can support is 5kg without counting the core of the pot and about 4kg with counting it. Furthermore, the sample needs an additional tool to be mounted on the shaker. The average weight of the support tools is about 2kg. Therefore, it is very important to take into consideration the shaker limits because increasing weight decreases acceleration limits. This shaker is shown in Fig. 8

Fig. 8 LSDV555 shaker.

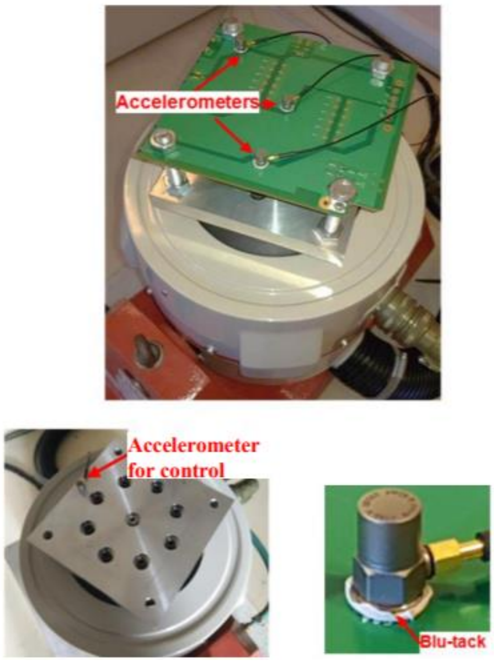

Fig. 9 The board mounting on the shaker, the accelerometers (monitors) positions on the board and the accelerometer (control) on the support tool and the blutack used for the accelerometers fixation

Four accelerometers were used, one for controlling that the shaker follows the profile and three for monitoring at different places, in the center, beside the fixation screw and on the middle side (between two screws). These accelerometers were fixed using blu-tack (blue adhesive substance) as shown in Fig. 9

7. Conclusion:

The PCB designing and manufacturing is an art in itself and has to be done with great precision and accuracy. The PCB should comply with the international standards intended for its use. The Military devices are exposed to harsh climatic and environmental conditions like Heat, Humidity, Thermal shocks and vibrations, hence the component selection is one of the most important aspect of PCB building. Selected electrical components should be reliable and withstand this harsh conditions and give the best performance. Testing and Inspection is also an another important tool as it helps to evaluate the performance and highlights the flaws in the PCB which can be corrected by improving the manufacturing methods. The metal detector PCB designed by us complies with the Military grade standards and will no doubted perform well in harsh conditions. As future advancements the plan is to include a RF transmitter circuit on the metal detector PCB and accompanied with separate RF receiver that can help to use this PCB in remote locations.