Printed Circuit Board Sensor Technology: Rigid vs. Flexible vs. Additive Manufacturing

by

Moriah Anise Reed

A thesis submitted to the Graduate Faculty of
Auburn University
in partial fulfillment of the
requirements for the Degree of
Master of Science

Auburn, Alabama
August 6, 2022

Printed Circuit Boards

Approved by

Dr. Robert Dean, Chair, McWane Endowed Professor of Electrical and Computer Engineering
Dr. Spencer Millican, Assistant Professor of Electrical and Computer Engineering
Dr. Thaddeus Roppel, Associate Professor of Electrical and Computer Engineering

Abstract

There is no denying that printed circuit boards (PCB) have integrated themselves into all sorts of technology that we have today. They are in the computers on which we work, the appliances we use, and the sensors that monitor our world; but how did the PCB come into existence? The history of the PCB is fascinating, and it reaches further back in time than one might think. Its design and manufacturing processes were influenced by many inventors such as Thomas Edison and Paul Eisler, and the PCB continues to evolve today. Traditional substrates for rigid and flexible PCBs are still viable, but as technology advances, the materials that compose PCBs are changing as well. In addition, long-established manufacturing methods are beginning to make way for additive manufacturing processes (AM). These new AM processes are not quite ready to manufacture at full capacity, but they can certainly speed up the development process of a single device.

All of this traditional and emerging technology is especially applicable to PCB sensors. PCB sensors provide a unique avenue for exploring the world. Using PCB manufacturing allows for versatility in design. The many designs allow for the creation of conductive, capacitive, or inductive measurements. However, it is important to bear in mind the particle being measured and to optimize the sensor design to be able to maintain as much contact with the measurand as possible. PCB manufacturing also creates the opportunity to test multiple designs to determine which is the best for the application. This thesis presents an in-depth look at the PCB's history,

materials, manufacturing processes, optimization, and how these specifically apply to multiple PCB sensor designs.

Acknowledgments

I would like to thank my major professor, Dr. Robert Dean, for his help, encouragement, and guidance in writing this thesis and throughout my graduate school career. I would also like to thank Dr. Roppel and Dr. Millican for their support and encouragement during this process and in class. In addition, I would like to thank my family, both biological and church, for their support, love, and prayers throughout my life, but especially during graduate school. Lastly, I would like to thank God, without whose help, I would not have been able to achieve this degree.

Table of Contents

List of Tables

List of Figures

List of Abbreviations

AM	Additively Manufactured
AMEP	Additively Manufactured Electroless Plating
AOI	Automated Optical Inspection
CAD	Computer Aided Design
CCL	Carbon Composite Laminates
CTE	Coefficient of Thermal Expansion
CTF	Ceramic Thick Film
DRF	Dielectric Relaxation Frequency
EC	Electrical Conductivity
ENIG	Electroless Nickel Immersion Gold
FF	Fringing Field
IDE	Interdigitated Electrode
LCP	Liquid Crystal Polymer
PCB	Printed Circuit Board
PWB	Printed Wiring Board
PTF	Polymer Thick Film
PTFE	Polytetrafluoroethylene
SMA	Subminiature version A
SMT	Surface Mount Technology

Chapter 1

Introduction

The printed circuit board (PCB) is one of the most influential inventions in the world. These devices are versatile in their configurations and can be found in most electronic systems manufactured today. From satellites and remotely operated vehicles to everyday electronics like computers and coffee machines, PCBs can be found in abundance.

PCBs have a fascinating history starting much farther back than one may realize. Their development was spurred on by war and technological races through the years, and they became an integral part of how our world operates. Even with the growth of other technologies over the last few years, the PCB still has the potential to continue impacting the world.

The main reason the PCB finds itself in this position is that many applications exist for these devices. The electronics world has benefited from PCBs because, without them, today's machines would not be possible. Medical instruments and devices are dependent on the technology of printed circuitry. Even safety apparatuses, such as fire alarms, are made possible by PCBs. However, with its low costs and versatile applications, the PCB truly shines in the world of scientific research in the form of sensors that allow for the exploration of Earth and the galaxy around us.

This thesis seeks to take a broad look at all PCBs by first turning the clock back to observe the PCB's development through the pages of history. Then, the different configurations, materials, and manufacturing processes of the PCB will be examined, followed by a discussion that focuses on the types of PCB sensors and how they are impacting the world today. Optimization of the PCB, along with a comparison of the rigid and flexible PCBs, will be presented. Lastly, there will be a discussion of potential future work that can be done to explore the uses of the PCB.

Chapter 2

The History of the PCB

Before discussing how PCBs can be used or how they are made, it is necessary to understand how these devices grew into the modern versions that are known today. PCBs have an intriguing history, and their primitive forms appeared much earlier than one might imagine. What, though, was the driving factor behind their development? As the old saying goes, "Necessity is the mother of invention," and PCBs are no exception.

In the late 1800s, the world was in the middle of the Information Age. By this time, the telegraph had been around for a half-century [17], and in 1876, Alexander Graham Bell invented the telephone. The first camera for the average person also became available in 1884 in the form of the Kodak camera, and this device was brought to the public thanks to George Eastman. Another notable inventor that was alive at the time was Herman Hollerith. He invented the tabulator in 1890. Two big names also competed for the spotlight during this time: Thomas Edison and Nikola Tesla. Thomas Edison is typically remembered for inventing the light bulb in 1879, but he has many other inventions ascribed to his name. One of those inventions was motion pictures which came on the scene in 1889. Nikola Tesla was also a prolific inventor. In 1888, he produced the electric motor, and then in 1895, he invented AC power [32]. Also, during this time, communication across the Atlantic Ocean became possible, but it was not yet commercialized [17]. All these inventions paved the way for future electronics, but to get to the electronics we have today, better circuitry needed to be designed.

The last decade of the 1800s to the 1920s marked the Progressive Era. In 1903, a German inventor by the name of Albert Hanson produced a patent for the first PCB [32]. Hansen sought to help build better circuitry for the telephone exchange [17]. Figure 2.1 shows Hansen's proposed circuit.

Figure 2.1: PCB Drawing from Hansen's Patent [32]

This PCB is quite primitive compared to the modern ones, but it was a start. Hansen's patent describes an additive process that utilized a dielectric substrate such as paraffined paper. The actual circuitry was crafted out of sheets of metal that were either copper or brass. These metal sheets would have patterns cut out to match the circuit pattern. Then, the patterns would be glued to the substrate. Hansen also first introduced the idea of through-hole technology [17]. Figure 2.2 shows the proposed through-hole connections.

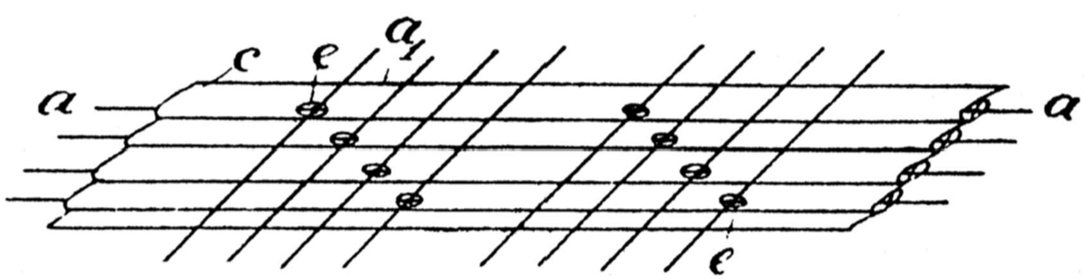

Figure 2.2: Through-hole Connections [17]

Another previously mentioned inventor, Thomas Edison, also had several theories on making these devices a reality. Two of his theories differed from Hansen's. In one process, Edison proposed drawing the circuitry line with glue on a substrate. Then, the glue would be covered with a powerderized metal. For his second theory, Edison used a silver nitrate solution for the traces. The silver nitrate could then be processed to leave behind only metal [17].

By the time 1912 rolled around, radio became another motivation to pursue PCB technology. That year, the first public radio station, KQW – San Jose, California, broke through the static. It sparked the incorporation of radio into the regular person's daily life. Over the next eight years, radio grew from that one lone station to hundreds of stations all over the United States and in most major countries around the world. Also, during this time, radio proved its worth as a safety feature, and it quickly became a life-saving standard in the maritime world [17].

1913 saw the development of the first true printing process, the Schoop Process. The inventor Max Schoop was responsible for the development of this additive process. His method involved using a mask and sprayers. The mask acted as a shield to control the metal spray as the traces were laid out [17]. Figure 2.3 shows Schoop's designs for masks and sprayers.

Figure 2.3: Schoop Process Designs [17]

The Schoop Process was a satisfactory solution for its day because it allowed for thick traces to be laid out. The traces needed to be thick to carry the massive amounts of current required to operate early circuitry [17]. However, as circuitry evolved, the traces became smaller, and the Schoop Process ultimately proved wasteful and costly [17].

During this same year, the first subtractive process of printing circuits was developed. Arthur Berry created a simple heating circuit, but he went about this in a unique way. He took metal sheets and drew the traces on them with an etchant resist. Then, he etched away the unnecessary metal [17]. Figure 2.4 shows Berry's heater circuit.

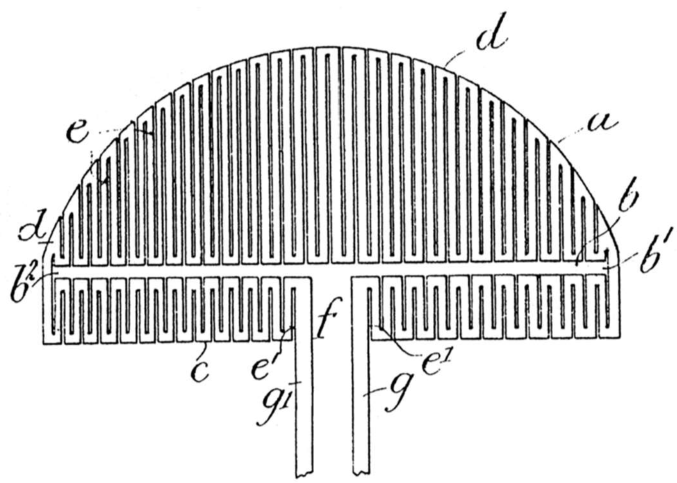

Figure 2.4: Heater Circuit [17]

It is also important to mention that thanks to E. Bassist, early photolithography emerged [17].

The subsequent significant development in PCBs came in 1925 at the hands of Charles Ducas. This invertor was also working on a heating coil, and he produced that in the form of the world's first printed wiring board (PWB). He also used a different method for making his circuits. His process involved depositing conductive inks onto an insulator, as shown in Figure 2.5 on the following page [32]. Also, like Albert Hanson, Ducas worked on the idea of multilayer circuits and how their connections might be made [17].

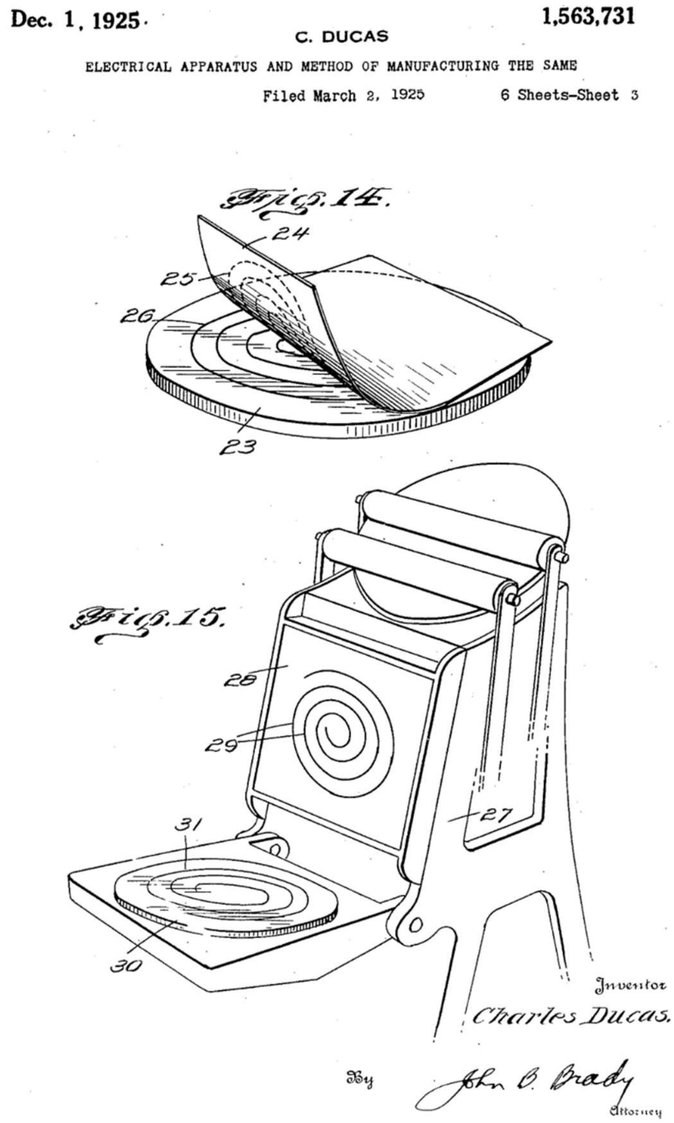

Figure 2.5: Charles Ducas' Patent [32]

At this point in the PCB's history, most of the ideas moving forward were improvements on previous inventions rather than actual new innovations. In addition, five years after Ducas' work, the Great Depression hit the United States. Industries across the country were hit incredibly hard, and the PCB industry, small though it was, could not escape. Like the rest of the United States' economy, it took a war to bring back the spark [32].

World War II brought innovation in many industries, including circuitry. The driving force behind these studies was twofold: communications and weapons. During wartime, reliable communications are paramount to a winning effort [32]. Paul Eisler is known as the Father of the Printed Circuit [17], and in 1943, he finished a radio that utilized a PCB. The PCB contained in the radio was composed of a glass substrate and copper foil traces [32].

Communication was only half the battle. The other great need in wartime was more advanced weapons than their enemies had. The world once again turned to PCBs to create bigger, badder weapons. The circuits that were produced due to this effort were hybrid circuits. These hybrid circuits were used as fuses and were composed of a ceramic substrate and conductive inks [17].

The effort to produce these new circuits during the war brought about many innovative processes to make PCBs. These processes can be broken down into several distinct categories. First, there were actual printing processes such as ceramic thick film (CTF) and polymer thick film (PTF). There was also the spraying process developed by Max Schoop that was no longer in production. However, his efforts paved the way for processes like chemical deposition and vacuum deposition to emerge in this era. Lastly, there was a die stamping process developed for PCB manufacturing [17]. Despite these major developments in PCB manufacturing, the PCB still had trouble finding its way into the average person's home [32].

However, during the Cold War and the Space Race, the PCB continued to progress. In 1963, the Hazeltyne Corporation patented plated through-hole technology. This reduced the space needed for electronics and the number of cross-over connections. Also, around this time, IBM first produced surface mount technology (SMT). Both advances helped catapult the U.S. to the moon [32]. Figure 2.6 presents a patent documenting through-hole technology.

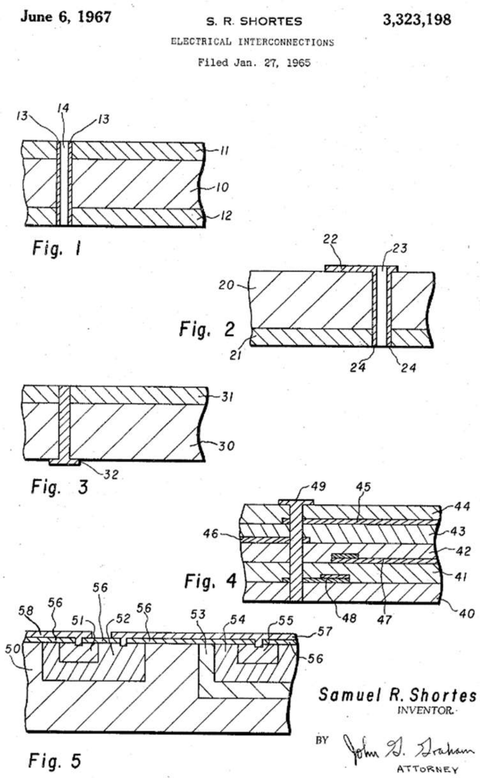

Figure 2.6: Through-hole Technology Patent [32]

Both of the previously mentioned technologies also allowed for the PCB to be truly commercialized. Due to this, PCBs allowed massive amounts of entertainment devices to become commonplace in people's homes. Gaming consoles, VHS players, and portable music players were just a few of the devices that were developed [32].

By the 1990s, the PCB had reached a plateau in its development as newer ways to create circuits and devices in silicon became more prominent [32]. However, the PCB is still progressing and making an impact in today's world.

Chapter 3

Printed Circuit Board Technology

Now that the history of the PCB has been considered, let us take a closer look at what comprises a modern PCB and how they are made. PCBs can be manufactured in two basic forms: rigid PCBs and flexible PCBs. There is also a hybrid PCB technology that is called a semi-flex PCB. Before delving into the specifics of one form over the other, factors that are determined by the circuit design should be considered. Some of these factors include the value of the material's dielectric constant, application of the final circuit, and glass transition temperature [28]. The dielectric constant of a material is directly connected to the size of the final product. Smaller circuit boards require materials with low dielectric constants to reduce the size of the traces needed [1]. The dielectric constant also comes into play when a PCB needs to handle high frequencies [28]. The glass transition temperature is entwined with the temperature that a PCB will have to handle during manufacturing as well as the number of layers that the PCB will need to be. More layers equal more time spent at high temperatures [28]. This puts the materials of the PCB under greater stress. The temperature during the manufacturing process is not the only place that temperature can impact the design of a PCB. The temperature that the PCB is expected to operate at must also be considered. Other mechanical properties of the material should also be analyzed. What kind of shear stress can the material handle? What is the tensile strength, and can it retain its shape? The last question mainly applies to flexible PCBs, and it refers to how well the PCB will deal with being bent once or multiple times over its service life [28]. Another consideration is whether or not the material can recreate the intended design with minimal shrinkage or enlargement [28]. The last consideration is the material's ability to resist fire. This is important to take into account, especially when trying to receive a UL Listing [28]. With these factors in mind, let us begin looking at the varied materials used to create the substrate for PCBs.

3.1 Rigid PCBs

Rigid PCBs can be made of many different base materials or substrates. The most popular is known as FR-4 [28]. FR-4 is made by coating an electrical-grade fiberglass weave with epoxy resin. The name itself represents two things: a name of a particular material and a grade of that material. The FR in FR-4 stands for flame retardant, and the number four is the grade [1]. This substrate is widely used, and for good reason. First, FR-4 is the most affordable option that is currently in use. Second, FR-4 features a widely applicable dielectric constant, at 4.5 [1]. It also does well in the temperature and environment categories mentioned above [28]. FR-4 and the primary material used for traces, copper, both have a coefficient of thermal expansion (CTE) of around 18 ppm/°C. This is important because both materials are expanding or contracting at the same rate [21]. This helps maintain good contact between the insulator and conductor throughout the entire PCB. It also means that the copper traces will not expand further than the substrate and cause cracks. In addition, FR-4 can be mixed with different additives to allow any number of its properties to be enhanced for different applications [28]. In general, for applications of fourteen layers or less, FR-4 is the way to go if a rigid PCB is the desired outcome [28]. There is an even cheaper version of this substrate called Perfboard. It still utilizes either FR-2 or FR-4 epoxy, but instead of laminating fiberglass, the epoxy is used to coat paper [28].

Another material that is used to create the substrate for rigid PCBs is polyimide laminate and prepreg. This material has excellent temperature resistance which makes it an ideal substance to use in harsh environments. On the electrical front, polyimide is only slightly better than FR-4. Polyimide is also much more expensive than FR-4 [28]. So, unless the PCB is going to encounter a harsh environment, FR-4 is still the best choice for the design.

Another material that can be used as a substrate for rigid PCBs is carbon composite laminates (CCL). It is important to note that CCLs are not dielectrics like the other substrates mentioned in this chapter. They are, however, electrically conductive, but this property allows them to be used as a ground plane layer. CCLs are excellent at dissipating heat. It also has a low CTE and is very rigid without adding extra weight. This rigidity allows for the PCB to perform well when introduced to shocks and vibrations [21].

The last rigid substrate that will be mentioned is known as Teflon which is chemically known as polytetrafluoroethylene (PTFE) and is ideally suited for high-frequency applications. Its unique properties do come with a substantial price tag and the need for special manufacturing. From the machines to the personnel, Teflon has different requirements, but if you need a PCB for high-frequency applications, this material will accomplish the task well [28].

Since a look has been taken at the materials involved in rigid PCBs, let us turn to the process of manufacturing these devices. Regardless of what kind of PCB you want to create, one must always start by drawing it out. When PCBs were developing, designers would draw them by hand. They accomplished this tedious task with a light board and stencils [32]. Nowadays, the process has been greatly simplified with the advent of the computer and computer-aided design

(CAD) software. The first CAD software suites were Protel and EAGLE [32], but now, the most commonly used application is IX274X that produces a PCB data format called Gerber Extended [26]. Using computer software allows one to produce an output file that contains all the pertinent information needed to manufacture the PCB. Some of the information included in the output file is the number of copper layers, the front and/or backside solder mask layers, the front and/or backside silkscreen layers, and an outline of the circuit board [40]. The output file might also contain the drill drawing and the component notations [26].

The next step in the PCB manufacturing process may seem trivial or boring, but it is extremely important. The design for manufacture check is the gatekeeper to this entire process. The manufacturer goes through the design with a fine-toothed comb to ensure that everything is in order. Tolerance violations are one of the biggest things being researched, and this process is accomplished using a computer and software as in the first step [40].

Once the design passes muster, it can move on to the next step, printing. This is not the same as printing out a document from your computer though. A plotter is used to produce the sheets, and instead of paper, the designs are printed on a film. This film closely resembles old-school projector slides. The plotters use two colors of ink: black and clear, and there are two styles of films being printed. The first style corresponds to the inside layers of the PCB. For inside layer films, the black ink represents the conductive traces, and the clear ink designates the non-conductive areas. The second style represents the outside layers, and for these films, the ink colors reverse roles [40].

The number of films to be printed is entirely dependent on the design being manufactured, but there is an extremely simple formula to help with this calculation. In this case, the number of films is equal to double the number of layers. This formula comes from the fact that there has to be a film for every layer and every etching mask, and each layer has one etching mask [40]. Once all the films have been printed, they have a registration hole punched through them. This registration hole is used to ensure that the design stays aligned throughout the manufacturing process [40].

With the films printed, manufacturing the layers can begin. It is important to note that cleanliness is of high priority throughout this process [26]. With cleanliness ensured, the inner layers are manufactured first. In order to do this, the chosen substrate is coated in a layer of copper. After the copper-covered substrate is glazed with a layer of photoresist, it is aligned with its particular film and exposed to UV light. The job of the film is to act as a selective filter to the UV light. The clear ink allows the UV to flow through and harden the photoresist. The black ink acts as a blocking agent to the UV. After exposure, the layer is washed in an alkaline solution to remove the unhardened photoresist. Another solution removes the exposed copper. After all of this, the layer is still not quite finished. It must undergo a cleaning process before is it deemed ready to move on to the next step. This process is repeated for all the inner layers [26].

The next step checks the layers for any alignment issues or defects. This process can be done by humans. However, most inspections are conducted via automated optical inspections (AOI) [26]. This process has greater efficiency and accuracy than an inspection conducted by a human [18]. The AOI machine uses lasers to first scan the layers. Next, it utilizes the scans to create an image for analysis, and lastly, it compares the image to the original design output file [26]. After passing inspection, the layers are aligned using registration holes [26] and an optical punch that inserts a pin through those holes [40].

Now, the layers are ready to be combined. This step actually has two sub-steps: lay-up and laminate [40]. To set up the lay-up stage, prepreg is layered on a press table [26]. Prepreg is a fiber-glass sheet filled with epoxy resin [35]. Next, a layer of substrate is deposited followed by a layer of copper traces. These two types of layers are alternated until all layers are present. Then, another layer of prepreg is added. Lastly, a copper plate is used on top. Registration holes and clamps are responsible for ensuring the alignment of the layers. The resulting product is called a PCB stack [40].

Now, the stack can be laminated. This is done in a laminating press that uses heat and pressure to fuse the PCB stack together. The heat and pressure melt the epoxy and bind all the layers into one [40]. With the layers combined, the pins are removed from the registration holes, and the copper topping plate is removed as well [40].

With the layers combined, the PCB is taken to the drilling station. Using an x-ray machine, the predetermined holes are located [40]. Then more registration holes are drilled, and lastly, a drill machine is used to manufacture the needed holes. This machine is run by a computer to ensure design quality since the holes are only around 100 microns in diameter [26].

The next step is to clean the product again. Now, more copper can be deposited on the stack. This step serves three purposes: to add copper to the holes that were just formed, to add another form of binding to the inner layers, and to deposit copper for the outer layer. This is done through chemical deposition to a thickness of one micron. As with the drilling, a computer is in control of this process [26]. With the copper deposited, the device is covered in photoresist, aligned under a film, and then exposed to UV light. The PCB is then put through another alkaline solution followed by another round of examination to ensure that the soft photoresist has been removed [26]. Unlike in the inner layer fabrication, the hardened photoresist now covers the unwanted copper, and the copper to be kept for the traces of the outer layers was just exposed by the alkaline solution.

To protect the desired copper traces, the device is put through two more plating processes. In the first, another layer of copper is electroplated to the surface. The second process deposits tin. This tin is used to protect the layer of copper that is to remain during the final etching. The final etching uses chemicals to remove the unwanted copper and the photoresist [26].

With the stack's inner and outer layers in place, the board can move on to the next step, solder mask application. Before the solder mask can be applied, the board must first be cleaned. Then, the board is covered in a layer of solder mask ink. Next, it is once again aligned under a film and exposed to UV light. As with the photoresist, the light hardens the solder mask that it touches, and the untouched solder mask remains soft. The unhardened solder mask is then removed. The board is then placed in an oven to cure what remains [26].

The next step is to apply the surface finish. This finish is applied to increase the solderability of the board [26]. Here are a few finishes that are available. Immersion silver is good for mitigating signal loss, and its composition is lead-free. However, this finish is prone to oxidation and tarnishing. A hard gold finish is resilient, also lead-free, but as with anything that is made with gold, expensive. The next type of finish is electroless nickel immersion gold (ENIG). This finish is fairly common and long-lasting. It is still somewhat expensive though. Another finish is the organic solderability preservative. This finish is cost-effective, but it has a small usability window. The last finish is known as lead-free hot air solder leveling. This finish is less expensive and able to be stored, and if there are mistakes in the finish, they can be fixed [40].

Once the finish has been applied, the board can have its silkscreen added. The silkscreen is done with inkjet writing, and it includes all the important details about the board that the user may need to know. Then the PCB receives one last coat and is cured [26]. With the PCB fully assembled, it goes through a final electrical test to ensure the connections are solid and the board is working as designed. If the tests are successfully passed, the board moves on to the last step: cutting out the individual PCBs [26].

Throughout the above process, there are multiple PCBs on one sheet. The final step is to release all of the individual PCBs from the sheet [26]. This step is accomplished through either routing, scoring, or cutting v-grooves along the edges of the boards [40]. Figure 3.1 illustrates the cross-section of a simple PCB, and Figure 3.2 shows an actual finished PCB.

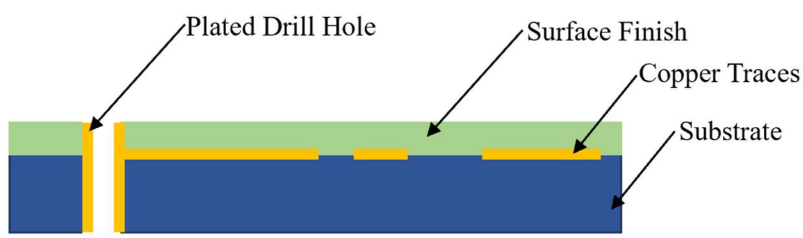

Figure 3.1: Cross-section of a PCB [40]

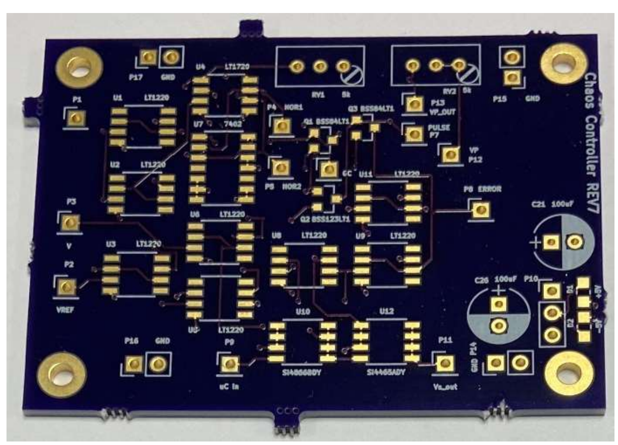

Figure 3.2: Rigid PCB

3.2 Flexible PCBs

Flexible PCB materials consist of flexible laminates. These laminates can be polyimide, liquid crystal polymer (LCP), or polyester [28]. LCP has been developed for the purposes of high-frequency applications. This thermoplastic is ideally suited for wireless applications where the frequency is 5 GHz or higher. LCP boasts a much better dielectric constant than FR-4, and it has a lower dissipation factor. In addition, it is very resistant to moisture with a moisture absorption rate that rivals glass. It does have an added cost with it, and traditional manufacturing is not always suited to this substrate's development [20]. However, in many applications the benefits often out-weigh the costs.

Despite LCP's great properties, the most popular flexible substrate is polyimide. Polyimide is a special type of thermosetting resin. This resin has material properties that allow it to resist becoming soft or flowing at the elevated temperatures that occur during manufacturing. In addition, it does not become rigid nor lose its elasticity after curing. This material does have room for improvement. Polyimide is prone to tearing and is adept at absorbing humidity which is not desirable for electronics. However, the ability of polyimide to resist hot temperatures and its good electrical properties make it the most used substrate for flexible PCBs [27].

Much of the manufacturing process of flexible PCBs follows the same process as rigid PCBs, but there are some differences. The first step to create a flexible PCB is to print the films that will be needed for the various layers. Next, a layer of copper is deposited onto a large sheet of the chosen substrate. Then, the copper-plated sheet is cleaned. It is important to note that there is special equipment that can offer support to the sheet of flexible substrate. The cleaned sheet is then coated in photoresist and aligned under the correct film. The board is then exposed to UV light to harden the appropriate sections of photoresist. The product is then put through a chemical etching process to produce the circuit pattern. Next, the sheet is taken to a drilling station. This drilling is done by automated drills, or if the holes are supposed to be extremely small, lasers are used. Once the drilling is done, the flexible PCBs in the making go through a copper plating process. At this point, the flexible PCB manufacturing process begins to differ from the rigid PCB one [14].

The next step in the flexible PCB manufacturing process is the coverlay application consisting of polyimide [14]. The polyimide acts as a protective covering for the PCB [36]. After this, the PCB goes through the lamination process. This process ensures that the coverlay is secured to the flexible circuit. Lamination uses heat, pressure, and a vacuum to bond the layers together. After this, the flexible circuit can receive an optional stiffener application. This is done by applying a stiffening material to strategic points on the circuit. If the PCB undergoes stiffener application, it must have another layer of coverlay applied and be laminated again to secure the stiffening material to the PCB. Once the PCB is completed, it undergoes electrical testing to ensure proper functioning [14].

As with rigid PCBs, multiple flexible PCBs are manufactured on one large sheet and must be punched out to complete the manufacturing process. However, due to the flexible nature of the substrate, most of the methods used to free rigid PCBs cannot be applied to flexible PCBs, but multiple techniques have been invented to assuage this issue. Routing is the method that both types of PCBs share. Another method of freeing flexible PCBs is the punch and die sets method. A third method uses a laser to cut the PCBs free, and lastly, chemically milled dies can be used [14]. Figure 3.3 on the next page presents a flexible PCB wrapped around a piece of PVC pipe.

Figure 3.3: Flexible PCB

3.3 Additively Manufactured PCBs

Additively manufactured (AM) PCBs can be both rigid or flexible, and they also do not have a true substrate depending on how they are printed [19]. The previous two processes are the traditional methods of PCB manufacturing and are both subtractive methods. In AM, the PCB is created layer by layer [19], and there are three primary ways to do this.

The first method of AM is inkjet printing. In inkjet printing, the design is delivered to the printer as a CAD file [37], and instead of paper, the printer uses a flexible material. In addition, traditional colored inks are also not used. Instead, the inks are nanoparticle or particle-free metal inks. Traditional nanoparticle metal inks have many fragments of the conductive metal in them [13]. In contrast, a combination of metal salts and a reactant(s) compose particle-free metal inks [41], and it is reduced to just the metal via a chemical reaction during the printing process [13]. Most of the time, the metal utilized is silver. If a multilayer PCB is desired, the printer uses a combination of metal inks and polymer inks to create each layer [19].

Ink jet printing PCBs is an excellent process for many reasons. First, this process allows for electrical testing after every layer is printed. This allows the manufacturer to identify any problems early, and it saves time if the process must be restarted. There is also minimal processing involved. Only two machines are used: a computer and a printer. Inkjet printing can also create very precise lines. In fact, it is common to achieve a precision of 100 microns. The minimal processing of additive manufacturing also results in less waste being produced than what is common for traditional, subtractive processes [19].

On the other side, ink jet printing also has a rather large set of problems. Even though the precision is phenomenal, it takes up a lot of time to achieve it. Metal inks can also create problems as too much ink can lead to it bleeding and then cracking as the PCB dries. As previously mentioned, the most common metal used is silver, and silver is not known for its solderability. The silver traces can be taken up by the tin in the solder during reflow. Also, standard solder pastes can harm the traces. There are some solder pastes that are accepted and useable. These pastes are usually either low-temperature tin-bismuth or indium-based pastes. Lastly, the current inkjet polymers are not yet durable enough to withstand common PCB operating temperatures [19].

The second type of AM is extrusion or single nozzle jetting. This is more along the lines of what is thought of as 3-D printing. In this style of AM, the design is delivered to the printer via a stereolithography file (.stl). For this process, the traces are first laid down on a flat surface. Then, a polymer is laid down around the traces to protect and hold the device together [19].

One of the benefits of this AM process is the ease with which multilayer circuits can be manufactured. Secondly, the conductivity and solderability of the final circuit are excellent. The thick deposits of metal for the traces result in this. There are some drawbacks to this process though. The pastes used can be difficult to control. This is especially true if the design calls for fine lines. In addition, the pastes cannot have any air pockets or bubbles in them, or else the circuit will not function properly. This process is also not very efficient because the target surface must be mapped first and then printed after, and lastly since the metal being used is still silver, the same issues presented for inkjet printing apply here as well [19].

The last type of AM is a combination of AM and electroless plating (AMEP). In this method, the design can be manufactured via stereolithography or fused deposition modeling. The object is printed with a unique metal-loaded material that can then be electroless plated. In this process, palladium is typically used as a seed metal. This is a new area of manufacturing and an advancing area of research. For example, at UCLA, using stereolithography, researchers made prints out of multiple materials that could then be selectively plated [19]. Additionally, in the UK, researchers are working on a process to use less expensive materials as seed metals [19]. The next big problem to be solved is working out a way to fabricate conductors in an object without drilling holes in the PCB or having the PCB go through the plating bath multiple times [19].

3.4 Comparing Rigid, Flexible, and Additively Manufactured PCBs

The second to last part of this section will consist of a comparison between the three types of PCBs. All three types have their pros and cons, but a lot of times, the design is going to determine the material and the process to create the device. In order to be concise, these pros and cons apply to the most common materials used in each device.

Rigid PCBs have many excellent qualities. They are low cost, have high dielectric strength, and are good insulators. In addition, they are compatible with most environments, and despite being lightweight, rigid PCBs have a good strength-to-weight ratio. They are also good at resisting moisture, and they have reliable performance in moderate to moderately high temperatures [1]. However, the rigid PCB has two drawbacks. First, rigid PCBs have a high dissipation factor which impedes signal transmission. Second, rigid PCBs have low impedance stability. This means that the dielectric constant may not change evenly over the whole PCB as the temperature changes. In conclusion, the rigid PCB is good for a wide range of general applications, but if the device is meant for a high-frequency application, a different type of PCB might be best [1].

Flexible PCBs have many advantages as well. For instance, flexible PCBs take up less space. They also have less chance of failure because there are typically fewer interconnects. They are also good in harsh environments, and as the name states, flexible [15]. One of the cons of flexible PCBs is that they do tend to cost more. Special assembly needs mean higher costs [16]. In addition, the material itself makes it more difficult to assemble [1], and they tend to be more fragile overall. The two previously mentioned disadvantages are the reason flexible PCBs are typically unsuited for complex PCB designs [1]. In conclusion, flexible PCBs are great for when you need them, and they are used widely in industries such as medicine, automotive, and aerospace, to name a few [15].

AM PCBs are good for small orders, and they can end up leading to a faster product development process. The process is sped up for multiple reasons. First, after the design has been uploaded to the machine, there is virtually no human involvement. Also, multiple designs can easily be created and manufactured, even simultaneously. This process also has a cost that is dependent on the weight of the material used not the complexity of the design. In addition, multilayer PCBs are much easier to produce. Lastly, the design can be created in any number of ways. This includes the shape of the circuit and how the traces are run. With AM, traces can be created in curves or even in 3-D [37]. The cons of this manufacturing process stem from two things. One, the process itself is still in development, and two, the materials used are limited at the moment [19]. In conclusion, at this moment, AM manufacturing processes are mainly good for product prototyping. But with advancing research, AM should become a mass-scale manufacturing method for PCB fabrication.

3.5 Sensors in PCB Technology

All of the above manufacturing techniques can be applied to PCB sensors. The process begins with the substrate as with any other PCB design. For most applications, the typical FR-4 substrate works well. However, there are plenty of applications for flexible substrates too. In addition, AM techniques can make developing sensors a much shorter process since multiple designs can be researched over a briefer period of time [37]. In reality, the true difference between a regular PCB and a PCB sensor is in how the traces are laid out on the substrate. Sensors rely on components such as resistors, capacitors, and inductors to create sensing structures to observe changes in their environment. PCB sensors take these potentially bulky components and reduce them to planar versions. These planar versions can be in one of three formats: an interdigitated electrode (IDE), a planar electrode, or a planar inductor.

The IDE is composed of two sets of teeth that fit together in the same way that the fingers can interlace as shown in Figure 3.4 on the next page. Though, instead of touching, these fingers have a predetermined space between them. As the sensor comes in contact with the measurand, the capacitance (or whatever is being measured) between and above the plane of the teeth is affected [6].

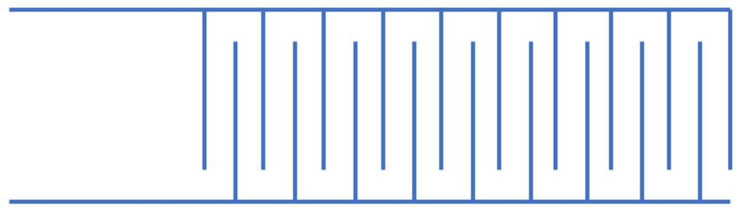

Figure 3.4: IDE Traces [6]

The planar electrode sensing element like the one in Figure 3.5 works in much the same way as the IDE. However, instead of multiple teeth for electrodes, fairly wide copper traces are applied to the substrate creating two large electrodes for conductivity sensing [5].

Figure 3.5: Planar Electrode Traces [5]

The last configuration for the traces of a PCB sensor are the ones for a planar inductor. These traces are laid out in a flattened spiral to form an inductor as depicted in Figure 3.6 [8].

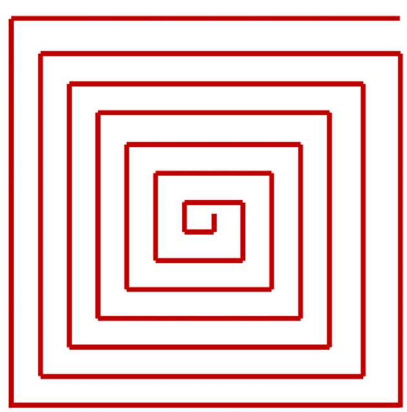

Figure 3.6: Planar Inductor Traces [8]

After the traces have been put on the substrate, holes are drilled to create vias for different connections. A lot of PCBs use subminiature version A (SMA) through-hole connections to establish a link between the sensor and the impedance analyzer. Once the hole is drilled, it can be plated. After plating, the entire sensor aside from the designated sensing area is covered in a protective layer of solder mask. The sensing area receives a layer of surface finish to protect it but still allow for the sensor to detect measurands in its environment [8]. Figure 3.7 below represents the cross-section of a PCB sensor.

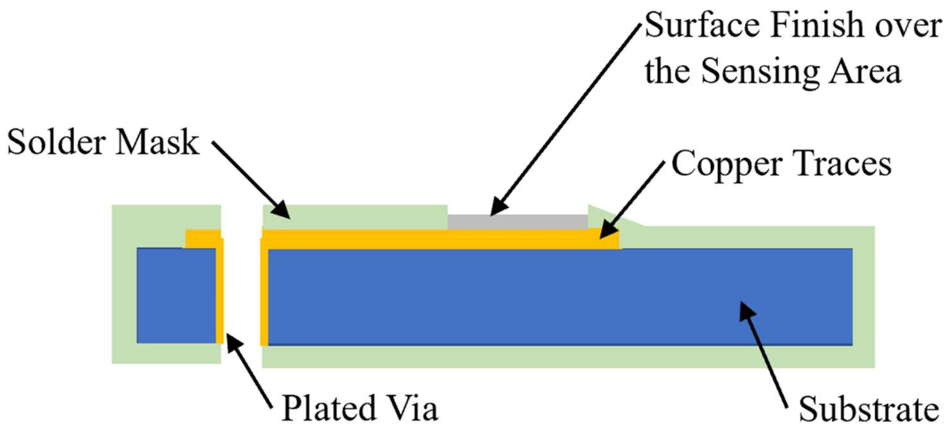

Figure 3.7: Cross-section of a PCB sensor [8]

Chapter 4

Types of Printed Circuit Board Sensors

Printed circuit boards are useful for creating all sorts of devices. However, they are especially suited for the low-cost realization of sensors, and much like the assorted styles and substrates, these sensors can be designed to measure many different variables. Depending on the desired application, the sensors can be either resistive/conductive, capacitive, or inductive. Let us look at each variation.

4.1 Resistive/Conductive Sensors

Resistive/conductive sensors detect changes in resistance. The best way to demonstrate how they work is to look at an example of one of these sensors. This example resistive sensor can be used to detect nitrates in water sources. This particular sensor utilizes an IDE array like the one mention in Section 3.5 to detect nitrates. Now, the first question to ask is how do nitrates change the water source so that they are detectable? Nitrates in a water source increase the liquid's conductivity. In addition, they cause the atoms of the liquid to form hydration shells. These hydration shells are known to have fairly low resonant frequencies. Vibration can be caused in these hydration shells by applying electromagnetic energy at the resonant frequencies. These vibrations cause detectable impedance alterations across the sensor, thus allowing the nitrates to be detected. A greater change in impedance indicates the presence of a larger number of nitrates [7].

Another good example of a conductive sensor is one that is used to detect levels of salt in samples of sand collected from different loggerhead sea turtle nesting sites. However, in this application, a planar electrode design was utilized. The salts in the sand affect the electrical conductivity (EC) of the sample. As the level of salt increases, EC also increases. This sensor allows for close study of the nests with minimal disturbance [5]. Figure 4.1 shows an example of an PCB EC sensor that utilizes a planar electrode design.

Figure 4.1: Resistive Sensor

4.2 Capacitive Sensors

As the name suggests, capacitive sensors are developed to sense changes in capacitance. These types of sensors seem to be the most popular. In particular, the IDE capacitor lends itself to many different applications, but in this case, a water level sensor is going to be discussed. This sensor is formed using an IDE capacitor. It looks exactly like the previously mentioned IDE array, but it is simply used to measure capacitance instead of resistance. Now, as the water level rises across the sensor, the capacitance starts to increase. This results from water having a dielectric constant that is around 80 times greater than that of air at room temperature. A higher water level results in a greater increase in capacitance. These sensors can be used for passive monitoring of systems to make sure that the water level remains manageable or non-existent, or they can be used to actively control a system based on the water level [22]. To further explain this principle, consider the bilge of a vessel. A little water in the bilge is not necessarily catastrophic, but it is not to be ignored. One could passively monitor the water level just to see where it is at a particular time, or the situation could be actively monitored where the bilge pump would automatically turn on to remove the excess water. Figure 4.2 presents a PCB IDE capacitive sensor.

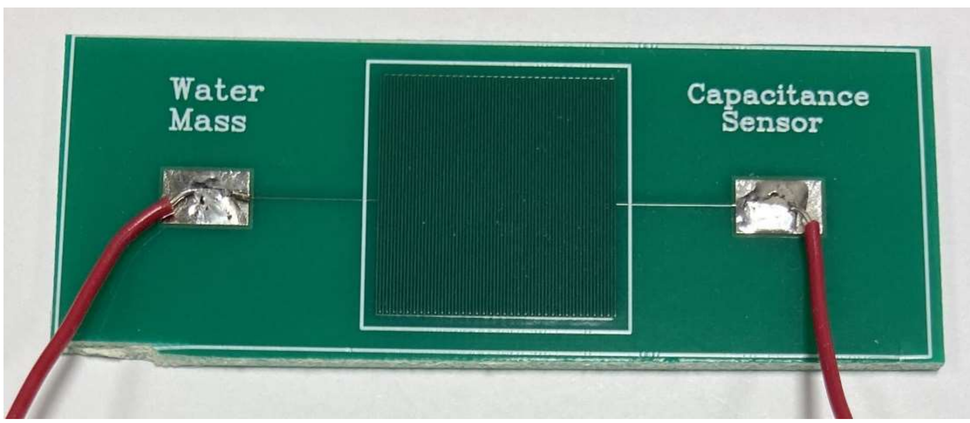

Figure 4.2: Capacitive Sensor

4.3 Inductive Sensors

Inductive sensors sense changes in inductance, but they do not seem to be as popular as the first two styles of PCB sensor design. However, they still have an important role to play especially considering the following example: an inductive sensor that detects corrosion in the form of rust. The inductor was created on a flat surface using copper traces drawn in diminishing square loops. Due to the magnetic properties of rust, changes in the inductance can be detected. The more the inductance increases, the more corrosion is present [8]. Once again, this sensor could be applied to the world of boats where detecting corrosion early is paramount. Figure 4.3 is of a PCB planar inductive sensor.

Figure 4.3: Inductive Sensor

4.4 PCB Sensors in the World Today

Now that the different types of PCB sensors have been discussed, it is important to understand the extent to which PCB sensors are impacting the world. Several applications have already been mentioned, but there are a lot more out there. Let us look at some based on the industry that they aid.

Food is something that everyone needs to survive, and PCBs can help by keeping food safe and providing valuable information about the content of said food. There are PCB sensors that are able to detect endotoxins that bacteria emit [30]. Another safety sensor that has been created is one that can sense chemicals that accompany food poisoning [33]. Engineers have also developed a sensor that can be used to non-invasively grade meat based on its fat content [24], and there is also a sensor that can report how much sugar is in solutions [3].

Before food can make it to the stores and tables, it has to come from the farms, and PCBs are helping there as well. Farmers can use a sensor that enables them to evaluate the moisture content of the soil. This can be used to provide useful information about what crops might grow best in which field [23]. In addition, after the crops are harvested and stored, too much moisture is not good for the food. For this reason, farmers monitor the moisture level in their storehouses closely, and a PCB design for this application is a great low-cost way to keep an eye on the harvest [4].

The scientific community is another area that can greatly benefit from PCBs. There are many PCB sensors to choose from that can aid in the exploration of the world. For instance, there are PCB sensors that are able to determine the air's humidity [31]. There are also sensors for collecting information about the PH level of water [34], and there are other sensors that can help track the level of salt in the water. This is especially important in areas where fresh and saltwater ecosystems are in close proximity [10]. In addition, another principal factor affecting water ecosystems is the buildup of inorganic materials. As it turns out, there is a PCB sensor that allows for the monitoring of this factor as well [39].

Many other industries can benefit from the PCB sensors that exist. In the construction industry, there is a PCB sensor that can help keep track of the moisture content in concrete [2]. For the medical world, there is a PCB sensor that can be used to detect defects in a person's DNA that can indicate the presence of certain genetic diseases or even viral infections [38]. There are also PCB sensors that can be used to increase the safety of everyone by detecting unwanted gases [25] and for monitoring the environment in general [29]. As previously mentioned, farmers need to keep an eye on the moisture level of their food stores, but this is not the only place that food monitoring is important. This technology can also be applied to the space industry and long space flights [9]. As you can see, there are many useful applications for PCB sensors in the world today.

Chapter 5

Optimizing PCB Sensors for Non-spherical Particle Sensing

An important part of designing a PCB sensor is considering what type of particle will be measured. Small particles like dirt or ones dissolved in liquid are easily measured using PCBs. They tend to cover the sensing area evenly and maintain good contact throughout the measuring process. However, when dealing with larger particles, orientation becomes a factor in how well you can measure your desired value and how accurate those measurements will be [12]. In addition, consider a three-dimensional rectangular object where none of the three side dimensions equal one another. In this case, the measurement that is received is going to be dependent on the side that is in contact with the sensing plane [12]. To further study this aspect of PCB sensor design, let us look at a sensor that is designed to measure the moisture content of corn.

For this experiment, there are several things needed. First, the testing material will need to be gathered. In this case, dried corn was utilized. Deionized water and a large beaker for soaking the corn in are also needed. A temperature and humidity sensor will also be used along with an impedance analyzer. Another beaker will be used to contain the corn for measuring its moisture content with the PCB sensor. A commercial moisture content analyzer was also used to provide a baseline for the moisture content of the corn being assessed in each round with the PCB sensor. Lastly, you will need the PCB sensor.

A rigid PCB sensor was used for these tests. An IDE was formed within the substrate to function as a capacitive fringing field (FF) sensor. The sensor's dimensions were chosen so that the PCB sensor could lie flat at the bottom of the beaker [12]. A photograph is shown in Figure 5.1 of the PCB sensor used in these tests. This PCB sensor utilizes 1.6 mm thick FR-4 as its substrate to cover an area that is 58.4 mm long and 43.2 mm wide. In addition, copper that measures 35µm in thickness forms the traces for the IDE and connections to the SMA connector. The IDE sensing area is composed of 82 overlapping electrode pairs covering an area of 1482 mm^2. The teeth are around 38.8 mm long and 152.4 mm wide each, and there is a 304.8 µm gap between opposing teeth. The PCB is protected by solder mask that is 50-75 µm thick. A different polymeric solder mask covers the sensing area. With this design's dimension, only objects that were less that 212 µm above the sensing area affected the capacitance detected by the PCB sensor.

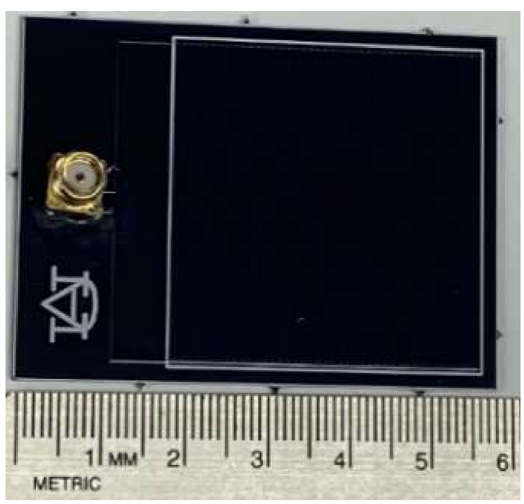

Figure 5.1: PCB Moisture Content Sensor [12]

With the materials gathered, the testing could commence. Before each set of measurements was taken, the corn was soaked in the deionized water for a specific length of time to allow the kernels to absorb moisture. Then, the testing took place with the sensor in one of two orientations. For the first orientation, the sensor was placed horizontally at the bottom of the beaker while connected to the impedance analyzer, and then, the corn was poured on top of the PCB sensor. The second orientation involved holding the PCB sensor vertically to the side of the beaker until enough corn was poured in to hold it securely in place. Once the PCB sensor was oriented, the capacitance was measured over a range of frequencies to test the moisture content. This process was repeated for various moisture contents to determine if there was any correlation between the orientation and the capacitance measured [12]. Figure 5.2 shows the experiment set up for the PCB sensor in the vertical position [8].

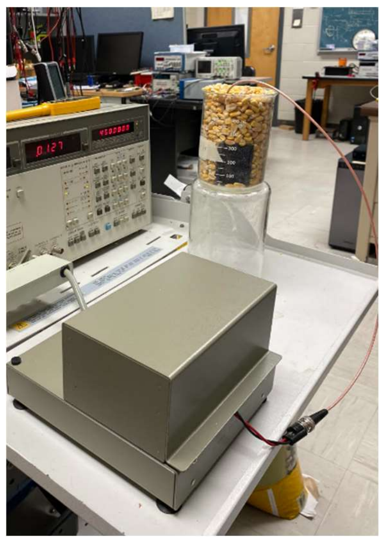

Figure 5.2: Testing Setup [8]

This experiment yielded the conclusion that the orientation of the sensor does affect the capacitance measured. At moisture contents of around 10%, the orientation of the PCB sensor did not make much of a difference. Although the capacitance readings of the PCB sensor oriented horizontally tended to be slightly higher. However, as the moisture content rose, the difference in the capacitance measured between the orientations became much greater. Figure 5.3 on the next page shows a graph depicting these results. Based on the results of this experiment, it is important to consider what the testing material is going to be when designing and orienting the PCB sensor.

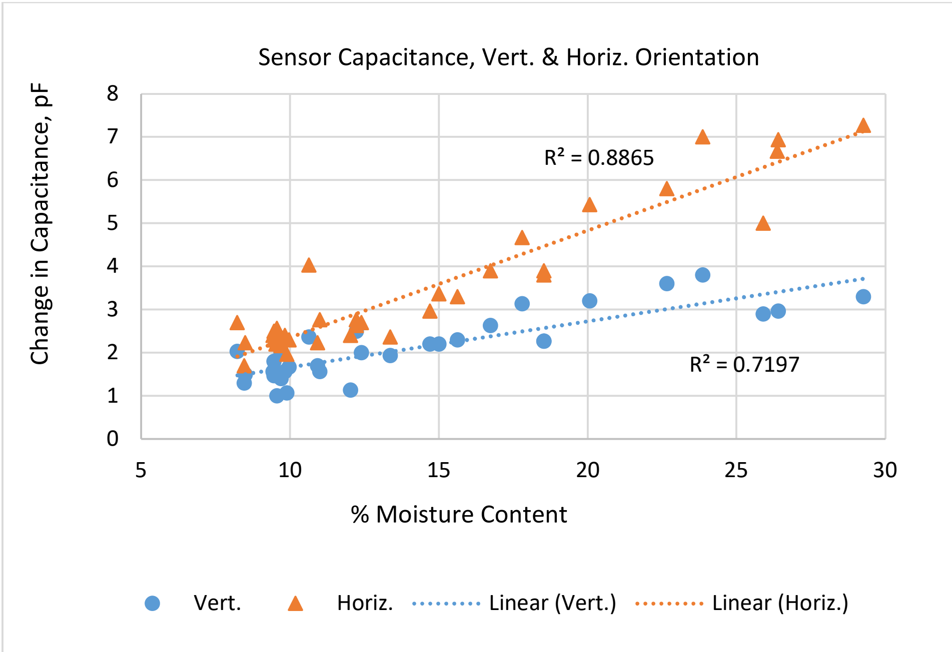

Figure 5.3: Moisture Content Test Results [8]

Chapter 6

PCB Ice Detection Sensor: Rigid versus Flexible

Ice affects many people's lives in winter, and depending on the industry that they work in, it could also affect their lives every day. Now, there are many instances when ice is a desirable outcome, but the opposite is also true. That is why being able to detect icing events is beneficial to many people. Having sensors that send an alert when the ice forms on the roads would allow for timely warnings to be issued to motorists and the appropriate machinery for salting the roads to be deployed. Ice sensors could also alert pilots to ice on the wings of airplanes or inform engineers of ice on a rocket that is about to be launched into space [11]. They could also be used to alert ship captains to the formation of ice on the deck or hull of their vessel. There have been several PCB sensors tested and developed to detect icing events [6]. In this section, two of these sensors are going to be compared. The first is a rigid PCB [6], and the second is an AM flexible PCB [11]. Let us begin, though, by discussing how scientists can differentiate between ice and water to develop these types of sensors in the first place.

6.1 Detecting Ice

Ice and water are both made up of the same components. In each molecule of either, there are two hydrogen atoms and one oxygen atom. So, how can a sensor differentiate between the two substances? The answer lies in the electrochemical properties of water and ice. Two factors that affect electrochemical properties are the dielectric constant and their dielectric relaxation frequency (DRF). Water and ice have a large gap in their DRFs. Water's DRF is 10 GHz, but ice has a DRF of 3 kHz at 0°C. This frequency just gets lower as the temperature decreases below 0°C. However, at 0°C, water and ice have about the same dielectric constant, 90 [11]. This is quite unhelpful for distinguishing between the two substances, but what causes this high dielectric constant? Well below the DRF, water and ice, since they are both H_2O, can experience proton hopping. When an alternating current is applied to H_2O at a frequency well below the DRF, the ions H_3O^+ and OH^- alternately form. These two exchange molecules as the polarity of the applied voltage alternates [6]. One could try testing at low frequencies, such as 1 kHz, but this is impaired due to high dielectric constants for both substances. However, if testing is conducted at frequencies between 10 kHz and a couple of gigahertz, there is a significant difference between the dielectric constants of ice and water since proton hopping will occur in water but not in ice. By measuring capacitances between this range of frequencies, one can determine whether the sensor is in water or in ice. When measuring at frequencies much higher than ice's DRF but below water's DRF the difference in the dielectric constants is more apparent and has an even more noticeable effect on capacitance.

6.2 Printed Circuit Board Ice Sensors

The rigid PCB sensor was manufactured using traditional subtractive methods. This PCB senor's substrate was FR-4 with traces formed from copper. The traces formed an IDE with thirty-six opposing electrode pairs, and to protect it from liquid or moisture, it was covered with a polymeric solder mask. An SMA connector provided a means of connecting the sensor to the impedance analyzer for measuring its capacitance. The total area covered by the PCB was 1.4 cm wide and 9.7 cm in length. The sensing area was 2.18 cm long and 1.11 cm wide. The sensing area was connected to the SMA connector by two copper traces [6]. This sensor can be seen in Figure 6.1.

Figure 6.1: Rigid PCB Ice Sensor [11]

The flexible PCB sensor was manufactured using both traditional and AM processes. A flexible polyimide substrate with an attached SMA connector was used. Copper traces ran to the sensing area where an AM inkjet process was used to produce an IDE, but instead of copper, conductive silver ink was used to draw the traces. This ink was Novacentrix JSB-40G silver nanoparticle ink. This printout was then connected to a traditionally manufactured flexible PCB with copper traces. Also, the sensing area was protected from moisture by an AM printed solder mask material. This PCB covered a total area of around 9.5 cm long by 2 cm wide. The sensing area was around 2.2 cm long and 1.8 cm wide. This sensor had the same SMA connector previously mentioned PCB sensor[11]. A photo of this sensor can be found in Figure 6.2.

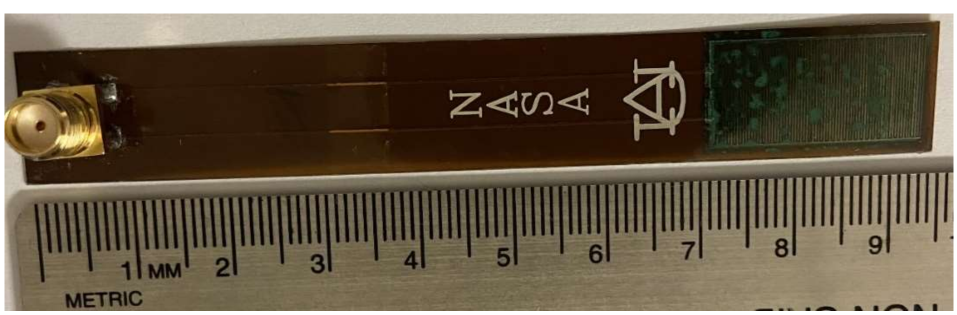

Figure 6.2: Flexible PCB Ice Sensor [6]

6.3 The Experiment

The experiment performed on both sensors had the same setup. The first step was to place the sensor along the inside of a plastic cup, as shown in Figure 6.3. The sensor needed to be placed far enough into the cup to allow the sensing area to be covered but the SMA connector to be accessible and clear of the water. Once the sensor was in position, a small clamp was used to keep the sensor in place. A digital thermometer probe was also placed in the cup to monitor the temperature of the water/ice. Next, deionized water was used to fill the cup so that the water covered the sensing area. This was then put in a beaker and placed in the freezer to allow solid ice to form. A high-power freezer was used to get the ice's temperature down to -30°C.

Figure 6.3: Experiment Setup [11]

Once the ice was ready, the setup was removed from the freezer and connected to the impedance analyzer. At this point, the two experiments followed two different paths [6][11]. The rigid PCB sensor was tested at various temperatures from -30°C to 34°C and at the frequencies of 1 kHz and 64 kHz [6]. The flexible PCB sensor was tested at temperatures from -30°C to 22°C. However, this PCB sensor was only tested at a frequency of 2.5 MHz since it was a continuation of the research explored with the rigid PCB [6] [11].

6.4 Results

The results yielded from both tests were good. The results from the rigid PCB experiment are in Table 1 below and graphically in Figure 6.4 [6].

Table 6.1: Rigid PCB Results [6]

	Capacitance (pF)	
Test	**1 kHz**	**64 kHz**
Dry, 23°C	27.7	26.7
Water, 34°C	175.2	108.7
Water, 25°C	199.2	108.7
Water, 15°C	192.2	107.7
Water, 5°C	181.7	110.2
Ice, 0°C	137.0	49.6
Ice, -2°C	123.8	41.2
Ice, -12°C	114.0	35.3
Ice, -21°C	99.0	33.3
Ice, -30°C	78.0	31.3

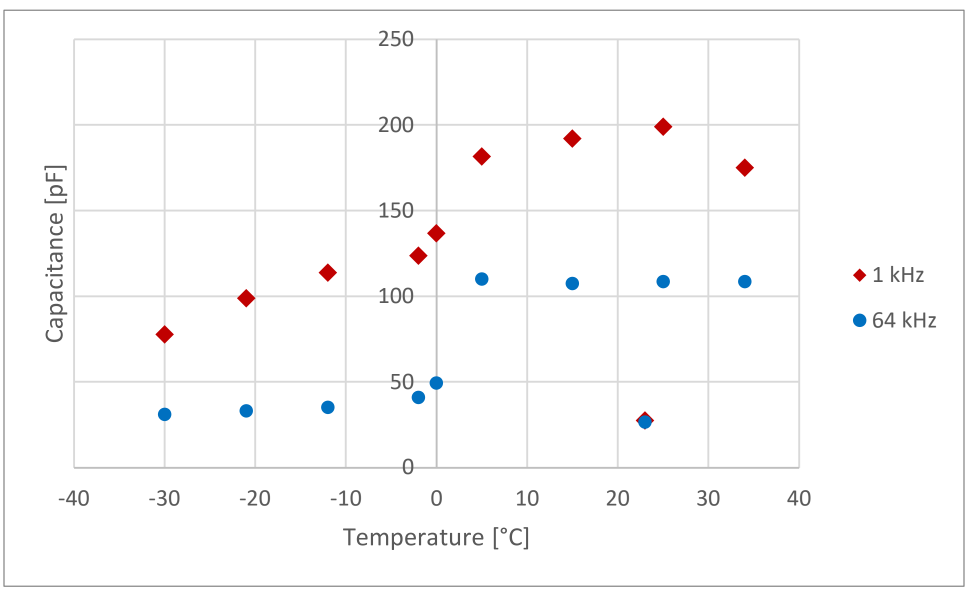

Figure 6.4: Graphical Results of the Rigid PCB Experiment [6]

As can be seen from the data in Table 6.1, as the temperature decreased, the capacitance measured also decreased. At 1kHz, the dip in capacitance as the water became ice is not too noticeable, but this is to be expected since a 1 kHz frequency is lower than the DRF of both ice and water. At a frequency of 64 kHz, the capacitance dips significantly as it passes from water to ice as theorized. Therefore, this PCB sensor successfully completed its mission and can be used to differentiate between liquid water and ice [6].

Now, let us look at the data gathered from the flexible PCB sensor in Table 6.2 and Figure 6.5.

Table 6.2: Flexible PCB Results [11]

Test	Capacitance (pF)
Water, 22°C	175.8
Water, 15°C	161.7
Water, 5°C	162.7
Ice, 0°C	20.2
Ice, -2°C	19.2
Ice, -12°C	17.9
Ice, -21°C	18.3
Ice, -30°C	17.4

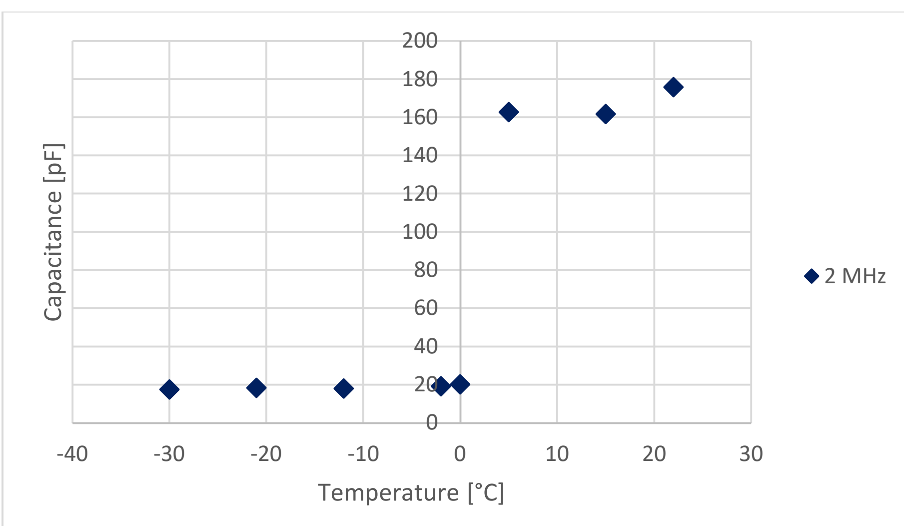

Figure 6.5: Graphical Results of the Flexible PCB Experiment [11]

In this case, there is also a direct correlation between the temperature and the measured capacitance. Once again, there is a major difference in the capacitance measured at 5°C and 0°C. However, in this case, there is a much more significant drop in capacitance [11]. This could be due to several factors. First, the flexible PCB sensor was evaluated at a higher frequency that was further away from the DRF of ice [11]. Secondly, when the ice formed in the cup with the rigid PCB sensor, a small gap developed between the ice block and the sensor. This allowed a small amount of air in between the sensor and the ice. While it was small, this could have changed the electrochemical properties being measured [6]. The flexible sensor was able to move with the ice as it formed, and this allowed better contact to be maintained between the ice and the sensor[11]. Because of this, the flexible sensor appeared better suited to the task at hand.

Chapter 7

Conclusion

PCBs are fantastic devices that can be applied in many different ways over a broad range of industries. They tend to be less expensive than other circuit implementation techniques, and they can be designed in a multitude of ways to solve any number of problems. Their history is a long one, but their development is still incomplete even today. They can be rigid or flexible, and they can be manufactured traditionally by subtractive methods, or they can be created using AM techniques. With a variety of substrates and materials for creating the traces, there is sure to be a PCB that will fit any design. These devices are especially useful for creating sensors that help derive information about the world around them, but which style of PCB is better? Is it the rigid PCB, or is it the flexible PCB? Is it PCBs that are traditionally manufactured or the ones created with AM? Each style has its pros and cons, but the design and desired application of the PCB sensor is the true driving factor in PCB sensor creation. In addition, it is important to keep the size of the particles being measured in mind. The PCB sensor design might have to be altered in order to maintain as much contact as possible with the material being measured. Now, in the ice detection sensor that was discussed, both sensors indicated a drop in capacitance as the water turned to ice. According to the rigid PCB sensor, there was a drop of 44.7 pF at a frequency of 1 kHz, and at 64 kHz, there was a capacitance drop of 60.6 pF [6]. The flexible PCB reported an even more dramatic drop in capacitance at a frequency of 2 MHz. In water at 5°C, the sensor reported a capacitance of 162.7 pF, but in ice at 0°C the capacitance was only 20.2 pF [11].

Based on the above results, both sensors did their job. However, the flexible sensor was able to do a better job. This might be due to how the ice was able to form around each sensor. The rigid sensor was prone to having the ice form gaps around the sensing area. This led to small air pockets that potentially changed the overall dielectric constant experienced by the sensor and by association the capacitance being measured [6]. However, the flexible sensor did not have this issue due to its ability to conform with the water as it morphed into ice [11]. This makes the flexible sensor the winner in this instance as far as sensor performance is concerned. Sensor cost, reliability, and manufacturing lead time were not considered in this work.

Chapter 8

Future Work

There are several areas in which this research can continue to grow. The first is quite simple: run more experiments. More experiments result in more data that can be used to further document the usefulness of PCB sensors. These experiments could be ice detection as mentioned in this paper, or they could be in any number of other areas as previously seen. There are simply so many applications for PCB sensors. Secondly, the PCB sensors could also be tested in various areas of industry. The ice sensor for example could be tested on dams that are prone to icing, on the decks of ships that operate in the North Sea, or even on cryogenics tanks that might have a tendency to develop ice on the outside. In addition, using AM, assorted designs of the sensor could easily be manufactured and then tested. This method of manufacturing would also speed up the process of prototype development. Lastly, theoretical experiments using computer software could also be completed to further explore PCB sensors design and technology. Running theoretical tests on the sensor design before it is even manufactured is another way to cut back on costs and time for viable prototypes to be developed.

Chapter 9

References

[1] *A Guide to FR4: When Can You Use It and When Can You Not*. n.d. <https://www.mclpcb.com/blog/fr4-guide/>.

[2] Alam, Md. Nazmul, et. al. "Concrete Moisture Content Measurement Using Interdigitated Near-Field Sensors." *IEEE Sensors Journal* July 2010: 1243-1248.

[3] Angkawisittpan, N. and T. Manasri. "Determination of Sugar Content in Sugar Solutions using Interdigital Capacitor Sensor." *Measurement Science Review* (2010): 8-13.

[4] Casada, Mark E. and Paul A. Armstrong. "Wheat Moisture Measurement with a Fringing Field Capacitive Sensor [Paper Presentation]." *2008 ASABE Annual International Meeting*. Providence, Rhode Island, United States, 2018, June 29 - July 2.

[5] Dean, Rebecca E. and Robert N. Dean "A PCB Sensor for Improving Loggerhead Sea Turtle Nesting Research." (n.d.).

[6] Dean, Robert N. "A PCB Sensor for Detecting Icing Events." *IEEE Sensors Letters* January 2021.

[7] Dean, Robert N., et. al. "A Low-Cost Environmental Nitrate Sensor." *IEEE* (2020): 165-170.

[8] Dean, Robert N., et. al. "A PCB Sensor for Magnetic Materials." *IMAPS Device Packaging Conference*. 2019.

[9] Dean, Robert N., et. al. "A PCB Sensor for Status Monitoring of Stored Food Stocks." *IEEE Sensors Letters* April 2019.

[10] Dean, Robert N., et. al. "A PCB Technology Electrical Conductivity Sensor for the Measurement of Saltwater Contamination." (n.d.).

[11] Dean, Robert N., et. al. "Development of a Printed Ice Sensor for Use in a Printed Wireless Tag." *IMAPS Device Packaging Conference*. 2022.

[12] Dean, Robert N., et. al. “Optimizing Fringing Field Sensor Performance with Non-Spherical Particles.” (n.d.).

[13] Dyson, Matthew. *Particle-Free Conductive Inks: An Advantageous Alternative?* March 10, 2021. <https://www.idtechex.com/en/research-article/particle-free-conductive-inks-an-advantageousalternative/23215#:~:text=In%20a%20particle%2Dfree%20ink,a%20smooth%20conductive%20metal%20layer.>.

[14] *Flex & Rigid-Flex Manufacturing Process.* n.d. <https://www.epectec.com/flex/gallery/>.

[15] *Flexible PCB.* n.d. <https://www.pcbcart.com/pcb-fab/flexible-pcb.html>.

[16] *Flexible PCB and Its Assembly Technology.* n.d. <https://www.pcbcart.com/article/content/flexible-pcb-assembly-technology.html>.

[17] Gilleo, Ken and Jerry Murray. "The Definitive History of the Printed Circuit." (1999).

[18] Helen. *9 Must-Know PCB Assembly (PCBA) Equipment*. 2019. <https://www.seeedstudio.com/blog/2019/07/10/9-must-know-pcb-assembly-pcba-equipment/>.

[19] Langnau, Leslie. *3D-Printed Circuit Boards: How They're Made and Why They Matter.* April 29, 2021. <https://www.makepartsfast.com/3d-printed-circuit-boards-how-theyre-made-and-why-they-matter/>.

[20] *Liquid Crystal Polymer Printed Circuit Board (LCP PCB).* December 12, 2020. <https://alcantapcb.com/liquid-crystal-polymer-printed-circuit-board-lcp-pcb.html>.

[21] Mangroli, Alex and Kris Vasoya. "Optimizing thermal and mechanical performance in PCBs." *Global SMT & Packaging.* December 2007: 10-12.

[22] Manut, A., et. al. "Characterization of Inter Digital Capacitor for Water Level Sensor." *RSM 2011*. Kota Kinabalu, Malaysia: IEEE, 2011. 359-363.

[23] Mizuguchi, Jaime, et. al. "Fringing Field Capacitive Sensor for Measuring Soil Water Content: Design, Manufacture, and Testing." *IEEE Transactions on Instrumentation and Measurement.* January 2015: 212-220.

[24] Mukhopadhyay, Subhas Chandra, et. al. "A Novel Planar-Type Biosensor for Noninvasive Meat Inspection." *IEEE* September 2007: 1340-1346.

[25] Omran, Hesham and Khaled N. Salama. "Design and Fabrication of Capacitive Interdigitated Electrodes for Smart Gas Sensors [Paper Presentation]." *2015 IEEE 3rd International Conference on Smart Instrumentation, Measurement, and Applications*. Putrajaya, Malaysia, 2015, 24-25 November.

[26] *PCB Manufacturing Process - A Step-by-Step Guide*. n.d. <https://www.pcbcart.com/article/content/PCB-manufacturing-process.html>.

[27] *Primary Material of Flexible PCB.* n.d. <https://www.pcbcart.com/article/content/primary-material-of-flexible-pcb.html>.

[28] *Printed Circuit Board (PCB) Materials*. n.d. <https://www.printedcircuits.com/printed-circuits-materials/>.

[29] Quintero, A. Vásquez, et. al. "Smart RFID label with a printed multisensor platform for environmental monitoring." *Flexible and Printed Electronics* 17 May 2016.

[30] Rahman, Mohd Syaifudin Abdul, et.al. "Detection of bacterial endotoxin in food: New planar interdigital sensors based approach." *Journal of Food Engineering* (2012).

[31] Rivadeneyra, Almudena, et. al. "Printed electrodes structures as capacitive humidity sensors: A comparison." *Sensors and Actuators A: Physical* (2016): 56-65.

[32] Sattel, Sam. *The History of PCBs*. n.d. <https://www.autodesk.com/products/eagle/blog/history-of-pcbs/>.

[33] Syaifudin, A. R. Mohd, et. al. "Measurements and Performance Evaluation of Novel Interdigital Sensors for Different Chemicals Related to Food Poisoning." *IEEE Sensors Journal* November 2011: 2957-2965.

[34] S, Zoolfakar A., et. al. "Design, Fabrication, and characterization of pH Transducer using Printed Circuit Board (PCB)." *2010 International Conference on Electronic Devices, Systems and Applications*. 2010. 206-211.

[35] *The Basics of PCB Design and Composition*. n.d. <https://emsginc.com/resources/basics-pcb-design-composition/>.

[36] Tome, Paul. *What It Takes to Manufacture Flexible PCBs*. June 22, 2017. <https://blog.epectec.com/what-it-takes-to-manufacture-flexible-pcbs >.

[37] *Uses of PCB Additive Manufacturing for Prototype Lead Time + More.* September 10, 2021. <https://blog.matric.com/pcb-additive-manufacturing#:~:text=Additive%20manufacturing%20involves%20making%20a,aided%20design%20(CAD)%20files.>.

[38] Van Gerwen, Peter, et. al. "Nanoscaled interdigitated electrode arrays for biochemical sensors." *Sensors and Actuators B Chemical* (1998): 73-80.

[39] Wang, Xiaolei, et. al. "Inorganic Material Detection Based on Electrode Sensor." *IEEE Sensors Journal* 1 June 2016: 4147-4148.

[40] *What is the PCB Manufacturing Process*. n.d. <https://www.mclpcb.com/blog/pcb-manufacturing-process/>.

[41] Yang, Wendong, Emil J. W. List-Kratochvil, and Changhai Wang. “Metal particle-free inks for printed flexible electronics.” *Journal of Materials Chemistry C*. Issue 48, 2019. <https://pubs.rsc.org/en/content/articlelanding/2019/tc/c9tc05463d>.