OULUN AMMATTIKORKEAKOULU

Zhenxing Xu

Creating a CAD Library for Printed Passive Components

Creating a CAD Library for Printed Passive Components

Zhenxing Xu
Master's Thesis
Spring 2022
Printed Intelligence
Oulu University of Applied Sciences

ABSTRACT

Oulu University of Applied Sciences
Printed Intelligence

Author(s): Zhenxing Xu
Title of the thesis: Creating a CAD Library for Printed Passive Components.
Thesis examiner(s): Timo Vainio
Term and year of thesis completion: 2022 Pages: **66 + 1 appendix**

The objective of this master's thesis was to create a CAD library for printed passive components. An electronic system consists of electronic components. Designing an electronic system with CAD software starts with selecting electronic components, which are the elements in a CAD library. Traditional electronics have good support from CAD software, and its electronic component libraries have rich content. Printed electronics are a new research and engineering area. However, CAD libraries for printed electronics lack content. Designing electronic systems with printed electronic components, but without support from the CAD libraries has a big challenge. The aim of this thesis is to try to find a solution by creating a CAD library for printed passive components.

Theoretical research and tool selection are necessary prerequisites for creating a CAD library. First, this thesis introduces the meaning of electronic components and the difference between active and passive components. The working principle of passive electronic components are described, especially resistors, capacitors, and inductors. Then scientific publications are summarized to find manufacturing methods, materials, and equipment for the printed passive components. During the literature research, important geometrical and electrical parameters for creating CAD libraries are recorded. After all theoretical studies, the software tools which are used for creating CAD libraries are selected. Their features and application areas are discussed and compared in a wide range.

As the kernel of the thesis, a process for creating library components is described. It demonstrated step by step to create a resistor in KiCAD and a capacitor in Altium Designer. Fusion 360 is used to create the components' 3D model. The processes in this three software represent the process of creating a CAD library for printed passive components. By using the same process with different electronic components' parameters, the library's content can be extended. In the end, as verification, a simple RLC circuit was demonstrated by using a newly created CAD library.

This work shows a solution to create a CAD library for printed passive components. However, Scientific research and engineering development are still ongoing. New printed electronic components appear every year. Libraries need to be maintained as the number of components increases. This could be the research and engineering area for the future.

Keywords: printed electronic, CAD, Altium Design, KiCad, Fusion 360, Passive Electronic Components

Contents

List of Abbreviations

Abbreviations

AC	Alternating Current
CAD	Computer-Aided Design
CAM	Computer-Aided Manufacturing
DC	Direct Current
ECAD	Electronic computer-aided design
EDA	Electronic Design Automation
EMF	Electromotive Force
ERC	Electronic rule check
LIFT	Laser-Induced Forward Transfer
MCAD	Mechanical computer-aided design
PE	Printed Electronics
PLM	Product lifecycle management

Symbols

R	Resistance
A	Cross-Sectional Area of the Conductor
C	Capacitance
d	Distance Between the Two Plates
L	Inductance
n	Number of Turns
r	Radius of Coil
s	Length of Turns
ε	Electric Constant
l	Length of the Conductor
ρ	Electrical Resistivity of The Material

List of Tables

List of Figures

1 Introduction

Electronic technology has been one of the fastest-growing technologies in the last two centuries. It transformed and made our lives better in a different way. In the 19th century, the electric stove (in 1859 by George B. Simpson), the light bulb (in 1879 by Thomas Alva Edison) was invented, and in 1985 the wireless telegraph and the radio communication became true. (casx123, 2021) Entering the 20th century, new inventions were not only new materials, various functions but also diameter. During this technological singularity diode (1904), triode (1907), the transistor radio (1954), integrated circuit (1958-1959), and the computer on a chip (1971) were discovered or invented (Constable, et al., 2003). Just when people were about to enter the 21st century, in 1998, a plastic transistor was developed (Constable, et al., 2003). This material opened a gate for printed electronics. With this new material, electrical components can be flexible and produced like the newspaper. The unique manufacturing process for electronics increased productivity and reduced the price per unit. Many new materials, technologies, and applications have been discovered in the last years. (Holden, 2020) Scientists, engineers, and people with different backgrounds have been investing with high enthusiasm in printed electronics. A lot of investigation was put into printed electronics, and many other applications were developed in the laboratory. But there is still a lot to be done to transfer the success of scientific research to large-scale industrial production.

The design of electrical devices was started from scratch on paper and draft schematics. The different computer programs are used today by electrical engineers thanks to computer technology. With the help of computer-aided design (CAD), tasks can be done in a more accessible manner than before on paper. Complicated electrical equipment can be constructed and even simulated in a virtual environment; a possible failure could be detected before the prototype. Product improvement and iteration could be accelerated. As a result, a lot of time and funds could be saved, better product quality could be more accessible, and complicated systems could be invented, which were unimaginable without CAD.

Electrical equipment consists of electronic components. Electronic components are included in the famous CAD/CAM (computer-aided manufacturing) tools as databases. This makes design and collaborative work very efficient. However, the recently appeared printed electrical components were rarely seen in the database. The work in this thesis attempts to introduce tools and processes to create CAD library for printed electronic components.

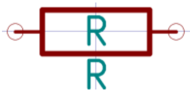
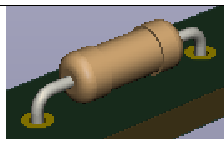
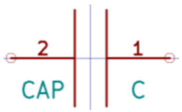
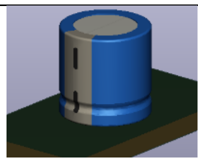
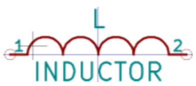
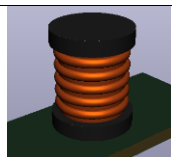
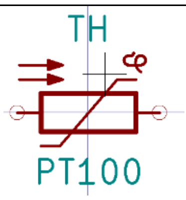
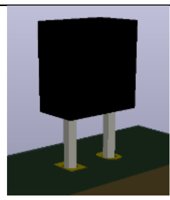
2 Electronic Passive Component

An electronic component is an essential part of electrical equipment. It helps the functioning of electrical equipment. Different electronic components can be classified into active and passive components (Sinclair, 2000) (Hughes, 2015). Active components can amplify the signal by using external energy, for instance, a power supply. Signals that pass through a passive part will become weaker. The passive component causes the loss of energy (Sinclair, 2000).

Passive components are the resistor, capacitor, inductor, and transducer. External power sources are not required for the operation of passive components. Power or energy from the circuit is employed and transformed by passive components in the form of voltage or current, and power gain can be produced. The flow of the current cannot be controlled. They act as energy receptors in an electrical system. (Hughes, 2015) (Dif21)

Table 1 presents different symbols and 3D models used in the library of electronic design automation (EDA) software KiCad.

Table 1 Passive components, symbols & 3D Models in KiCAD (Suite, 2021)

components	**Symbols**	**3D Models**
Resistor		
Capacitor		
Inductor		
Transducer (Sensor PT100)		

2.1 Resistor

Passive components have different electronic properties and are used in various applications. When current flows through the resistor, part of the electricity is converted into heat and dissipated to the surrounding area. Current flowing through the resistor is proportional to the applied voltage. An ideal resistor has the same impedance performance at all signal frequencies from direct current (DC) to alternating current (AC); it follows Ohm's law (Sinclair, 2000). Resistors can limit the current in electronic circuits and can also be found in most heating equipment. (ETechnoG, 2021)

Two factors influence the resistor of a given object. Equation (1-1) describes this relationship. ρ is related to the type of materials. Resistance is positively correlated to the length, and the larger cross-section has lower resistance.

$$R = \rho \frac{l}{A} \tag{1-1}$$

R: Resistance, Ω

ρ: Electrical resistivity of the material (Ω·m)

l: Length of the conductor, (m)

A: Cross-sectional area of the conductor, (m^2)

2.2 Capacitor

The working principle of the capacitor is explained with simplified graphics in Figure 2-1. There is a dielectric located between two conductive plates, which can be for example metal. The energy is stored in an electrostatic field; a potential between the two plates generates this. The capacitance between two plates can be calculated with the equation (1-2) (Bird, 2010).

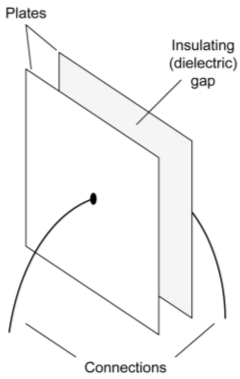


Figure 2-1 Generalized diagram of a capacitor (Hughes, 2015)

$$C = \varepsilon \frac{A}{d} \quad (1\text{-}2)$$

C: Capacitance, Farads

A: Direct facing area of the two plates, square Meters

ε: Electric constant

d: Distance between the two plates, Meters

Based on its unique structure (Figure 2-1), DC cannot pass through a capacitor. However, AC, whose direction is reversed periodically, could pass through it. Capacitors are often used to let a signal of a particular frequency pass.

2.3 Inductor

Besides resistors and capacitors, inductors make up other critical passive components. In 1831 Michael Faraday discovered a phenomenon, an electromotive force (EMF, voltage) which could be generated in a time-varying magnetic field located conductor. The inductor is an application of electromagnetic induction on the coil. (Sinclair, 2000) Unlike the capacitor, an inductor has the opposite behavior, and it allows DC to pass but impedes AC.

The unit of inductance is Henry. It represents the current change rate of 1 ampere/second and induces a voltage of one volt (Figure 2-2). (Sinclair, 2000)

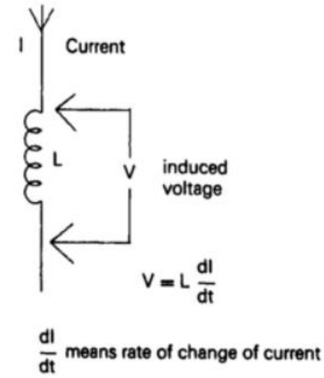


Figure 2-2 Current, voltage, and inductance (Sinclair, 2000)

For a single-layer air-core coil (Figure 2-3), an approximate inductance can be calculated with formula in (1-3) (Sinclair, 2000)

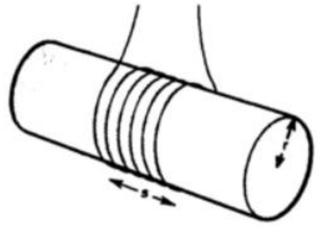


Figure 2-3 Inductors with single-layer air core coil (Sinclair, 2000)

$$L = \frac{2.8r^2n^2}{r + 1.11s} \tag{1-3}$$

L: Inductance, μH

r: Radius of coil, cm

s: Length of turns, cm

n: Number of turns

The magnetic material can be used in the inductor's core in many applications. This can increase inductance enormously. Inductors also have different shapes of construction. In the traditional and silicon-based electronic applications, inductors are primarily in 3 dimensions. Figure 2-4 shows the different variants.

Figure 2-4 Different types of inductors (McLyman, 2017)

2.4 Passive Transducers

The previous chapter describes energy conversion from electrical energy into heat by a resistor, electric field by a capacitor, and magnetic field by an inductor. Passive transducers reverse this process. It transforms between physical quantities. For instance, heat and displacement could be converted into variable resistance, capacitor, or magnetic resistance. Some real applications are thermocouples, capacitor microphones, capacitor touch sensors, or moving iron microphones.

3 Fabrication for Printed Passive Components

Traditional passive components are presented in chapter 2. The difference in function and principle between the printed passive components and the conventional passive components is limited, whereas printed components' fabrication is unique. Printed electronics (PE) is manufactured with printing technology and implements a particular printing material with an electronic function. This chapter summarizes different printing technologies used for PE fabrication and how the printed passive components are developed by utilizing printing technologies.

3.1 Overview of Printing Technologies

Printing is defined as a reproduction process by applying the printing ink to a substrate with the help of an image-carrying medium. (Kipphan, 2001) Divided by the image-carrying medium, printing technologies can be categorized into contact and non-contact printing. The technologies applied in graphic printing and printed electronics are similar, but they have different quality requirements. Graphic printing is concentrated on optical quality, and electronic properties are important in printed electronics. The commonly used printing technologies are presented in Table 2. Most of them are used by graphic printing and printed electronics, but Aerosol jet and dispensing printing are more common in printed electronics. These two technologies use a single nozzle to generate dots, lines, and surfaces on two dimensions. It also has the capability of printing three-dimensional components.

Table 2 Different Printing Technologies (Kipphan, 2001) (Tan, et al., 2016) (Panreck, et al., 2018) (Pohl, et al., 2015)

Contact printing technologies	Non-contact printing technologies
• Screen Printing • Gravure printing • Flexographic printing • Offset lithography	• Aerosol jet printing • Inkjet printing • Dispensing printing • Laser-induced forward transfer (LIFT)

Depending on the application, different printing technologies have their advantages and disadvantages. They have their unique adaptability related to the substrate and ink. For instance, screen printing utilizes high viscosity ink and can have very high layer thickness. It also has good substrate adaptability. This could benefit the production conductor, which requires higher power transfer on the substrate. In opposite, inkjet ink has very low viscosity, but it has a very thin layer thickness. It does not need a printing plate; files can be directly printed on the substrate. Table 3 presents the critical feature of different printing technologies.

Table 3 Characteristics of Printing Technologies (Kipphan, 2001) (Panreck, et al., 2018) (Kwon, et al., 2020)

	Printing plate	**Ink viscosity (cP)**	**Line width (µm)**	**Layer thickness (µm)**
Screen	with	1000 – 50,000	30 – 50	Up to 12
Gravure	with	50 – 200	10 – 50	0.8 – 8
Flexographic	with	50 – 500	45 – 100	0.8 – 2.5
offset	with	40,000 – 100,000	~ 10	0.5 – 1.5
aerosol	without	1 – 1000 PA	~ 10	0.1 – 2
inkjet	without	5 – 20	30 – 50	< 0.5
dispensing	without	1 – 1,000,000	5 – 200	-

3.2 Materials Used for Printed Passive Components

Substrate and ink are indispensable and important materials for producing printed electronics. Different substrates are found in the research and applications. Substrates from different materials have separate properties. For instance, paper (Santhiago, et al., 2017) (Siegel, et al., 2010) or fabrics (Simegnaw, et al., 2021) (Wei, et al., 2013) are porous, in which the substrate could absorb the ink; on the other hand, ink sticks only on the surface of the metal, glass (Hrehorova, et al., 2011) and plastics. Polymer materials raise broad interest in PE applications thanks to their excellent environment reliability and flexible characteristics. Passive components could be produced on different polymer substrates, such as polyimide (PI), polyetherimide (PEI) (Salmerón, et al., 2014), polyethylene naphthalene (PEN) (Correia, et al., 2018), and polyethylene terephthalate (PET) (Ostfeld, et al., 2015). Some polymer Substrates are commonly used materials in our daily lives.

For example, PET is a widely used low-cost, food-safe material in producing drink bottles. With the benefit of lower material prices, the advantages of PE products are becoming more prominent.

There are various kinds of function ink materials used for PE components. According to the different classification methods PE materials could be organic or inorganic. (Cui, 2016) Furthermore, they can be divided into conductors, dielectrics, and semiconductors. Conductors and dielectrics are commonly used materials for the fabrication of passive components.

3.2.1 Conductive Materials

Organic conductive materials are reported in the recent application. For instance, polyacetylene, polyaniline, polypyrrole, polythiophene, and poly(3,4-ethylene dioxythiophene) (PEDOT) are the most researched conductive polymer. (Wang, et al., 2019) (Liu, et al., 2003) By applying the doping process, the conduction of polyacetylene is almost comparable to copper. (Cui, 2016) Conductive materials can also be obtained by dispersion of many conductive materials, for example, metal powder or graphene in a non-conductive polymer. (Hardin, et al., 2019)

Commonly used inorganic materials are a mix of metal particles within resins to produce printed pastes, and the “paste” can be used for screen printing. Resin-containing paste has a higher post-treatment temperature, and the resistivity (1,000 to 100 μΩ·cm) is 2 to 3 orders higher than bulk metal (1 μΩ·cm). (Cui, 2016) Recently developed nanotechnology has enabled the dispersion of nanosized metal particles in solvents. With the help of surfactants, nanoparticle ink could be treated at a lower temperature, but its conductivity is much better than the “paste”; it is only a few times lower than the bulk material. (Anto, et al., 2010)

Because of its high conductivity, silver (Ag) 1.59 μΩ·cm, copper (Cu) 1.72 μΩ·cm, gold (Au) 2.44 μΩ·cm, and aluminum (Al) 2.65 μΩ·cm are mainly used as materials to produce inorganic conductive materials. Au has the best environmental stability, but it is the most expensive material compared to the other materials. Cu and Al have poor ecological stability. They both have a high tendency for oxidation. For this reason, Cu and Al nanoparticle ink are challenging to prepare. Ag becomes the most cost-effective nanoparticle ink to produce the conductive layer. (Cui, 2016)

Conductive ink can also be obtained by heavily doping a transparent oxide semiconductor to obtain transparent conductive oxide (TCO). This material has much lower conductivity than nanoparticle ink, but it has its unique transparent property; for example, a TFT panel needs transparent wires.

3.2.2 Dielectric Materials

Dielectric or insulating materials are the critical material in producing multi-layer electrical components. It prevents electrical leakage between different layers (for example, conductors or semiconductors) and makes the production of multi-layer capacitors possible. The dielectric layer must be thick enough to prevent electronic leakage. In some cases, it needs to be thinner to get better overall performance, for example, a large capacitor. (Hardin, et al., 2019)

The insulating materials can be organic or inorganic. Printable organic dielectric materials include poly(methyl methacrylate) [PMMA] (Hardin, et al., 2019), polyimide [PI] (Zhang, et al., 2016), poly(vinyl phenol) [PVP], poly(vinyl alcohol) [PVA], polystyrene [PS], poly(perfluorobutenylvinylether) [CYTOP] and benzocyclobutene [BCB] (Cui, 2016). Inorganic inks include inorganic oxides (ALO_x, ZrO_x, HfO_x, TiO_x), (Caironi, et al., 2015) ion-gel and solid-state electrolytes (SiO_2) (Cui, 2016).

Organic dielectric materials have the advantages of materials variety, surface roughness, surface trap density, compatibility with other organic materials, cost, flexibility, and low temperature. (Cui, 2016) because of these properties, it has a good potential to produce flexible electronics. Inorganic materials have better properties at high permittivity and high-frequency stability. But it requires high-quality film formation and a high-temperature post-process. (Cui, 2016) (Caironi, et al., 2015)

3.3 Fabrication of Printed Resistor

Correia (Correia, et al., 2018) reported in his publication on an inkjet-printed resistor. Teonex Q65HA poly (ethylene naphthalate) PEN film was used as a substrate. The electrode layer was printed with nanoparticle Ag ink (UT DOTs inc.). Conductive polymeric ink Poly(3,4-ethylene dioxythiophene) poly(styrene sulfonate) PEDOT:PSS was used as an active layer. The ink was printed with Fujifilm DMP-3000 with the DMC-11610 cartridge and 10 pL drop volume. Ag layer was printed as the first layer, then the PEDOT:PSS was printed after, and different layers were printed without curing in between. Ag ink was dried in the oven for 30 minutes at 150 °C, and PEDOT:PSS was cured at 130 °C for 5 minutes.

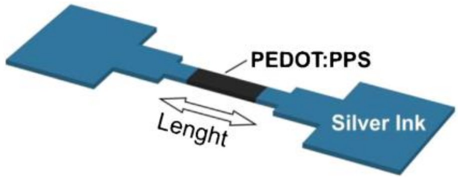


Figure 3-1 Schematic representation of printed resistor (Correia, et al., 2018)

The inkjet-printed resistors had 500 µm in width, 0.5 mm to 2 mm in length, and were printed in 2, 4, and 6 layers. After the dry and curing, the resistance was between 25 Ohm to 650 Ohm (Figure 3-2) with less than 5% tolerance.

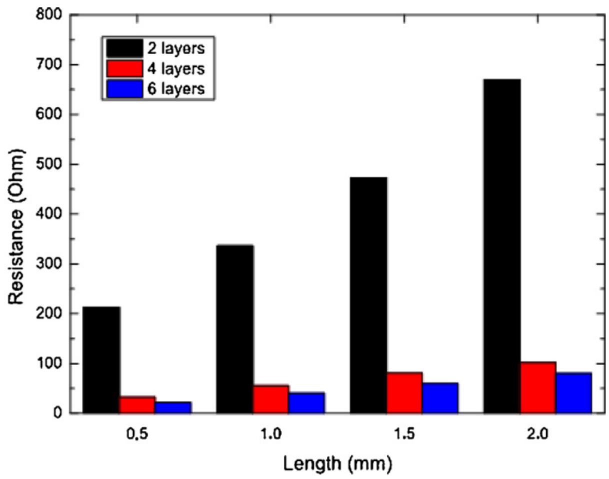
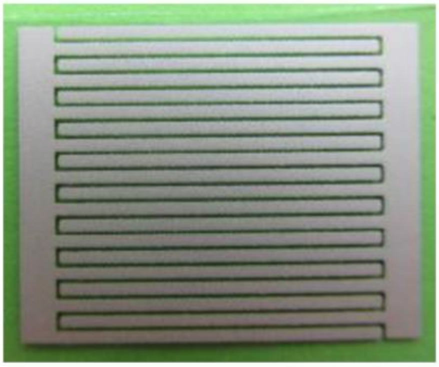
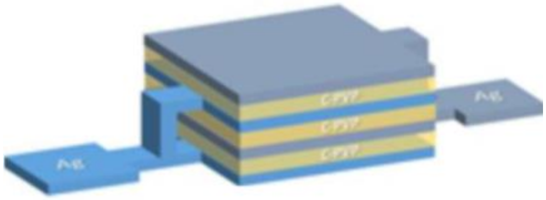


Figure 3-2 Resistance of inkjet-printed resistor (Correia, et al., 2018)

3.4 Fabrication of Capacitor

The printed capacitor can be produced differently. Interdigital capacitors and sandwich capacitors are the general constructions of printed capacitors (Blecha, et al., 2017). The interdigital capacitor (Figure 3-3 left) has only two dimensions and needs only conductive ink. The ink is printed on the dielectric substrate with the unique geometric shape of the electrode, and the two electrodes are very close, so a capacitor can be generated. A sandwich capacitor is printed layer by layer. The dielectric layers between each conductive layers are printed (Figure 3-3 right).





a) b)

Figure 3-3 Different types of capacitors a) Interdigital capacitor (Blecha, et al., 2017), b) Sandwich capacitor (Correia, et al., 2018)

Blecha (Blecha, et al., 2017) presented in his report an interdigital capacitor. It was produced with screen printing technology. The silver paste DuPont 5029 was printed on PET film. The capacitor had the dimensions (Figure 3-4) of w = s = 0.75 mm, l = 20 mm, and interdigital pairs = 20.

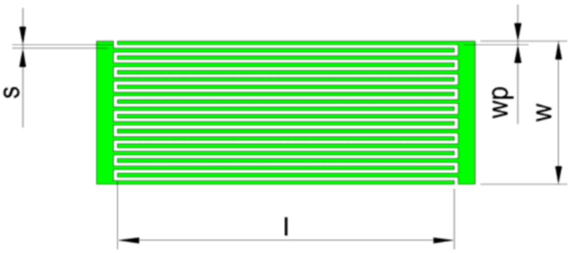
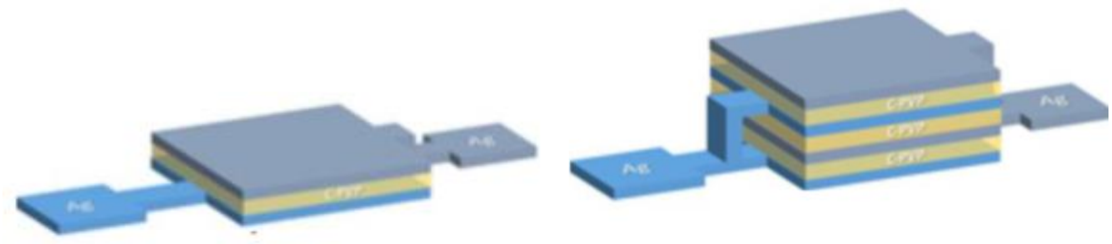


Figure 3-4 Dimensions and geometry of interdigital capacitor (Blecha, et al., 2017)

The sandwich capacitor was found in Correia's report (Correia, et al., 2018). The dielectric layer was printed with cross-linked poly (4-vinylphenol), with propylene glycol monomethyl ether acetate (PGMEA) and poly (melamineco-formaldehyde (PMFM) (C-PVP). The material used for the substrate and electrode was the same as described in chapter 3.3. The capacitors were produced with an active area of 5 x 5 mm² and 2.5 x 2.5 mm². The dielectric layer was slightly bigger with the 5.5 x 5.5 mm² and 3 x 3 mm² to guarantee good isolation. All the layers were printed with inkjet technology. The print head has a typical drop volume of 10 pL. The printer was set on 40 µm drop space for the conductive layer and 15 µm drop space for dielectric layers. Single stack and triple stack capacitors were printed with the same process. The geometry of the sandwich capacitor is presented in Figure 3-5. Both conductive and dielectric layers were cured layer by layer in the oven (Nabertherm TR240) at 150°C and 30 minutes.



a) b)

Figure 3-5 Dimensions and geometry of Sandwich capacitor a) Single stack capacitor, b) Triple stack capacitor (Correia, et al., 2018)

The capacitance was 2.0 nF/cm² for the single stack and 3.2 nF/cm² for the triple stack. (Correia, et al., 2018)

3.5 Fabrication of Inductor

Printed inductors are mainly planar spiral inductors due to the property of printing technologies. Generally, only one printed layer is required. In some cases, the dielectric layer is printed to avoid the shortcut [22] between the electrodes or protect the printed layer (Pajkanovic, et al., 2016). Planar spiral inductor used in silicon-based electronic and printed electronics has similar structures. Figure 3-6 shows the commonly used planar inductor structures for silicon-based electronics. Figure 3-7 presents the printed product of planar inductor structures from a different source.

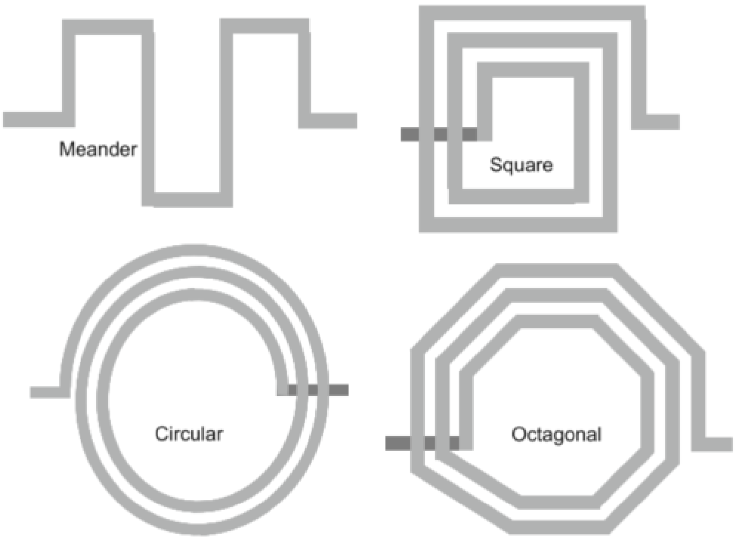


Figure 3-6 Planar inductor structures for silicon-based electronic (Haobijam, et al., 2014)

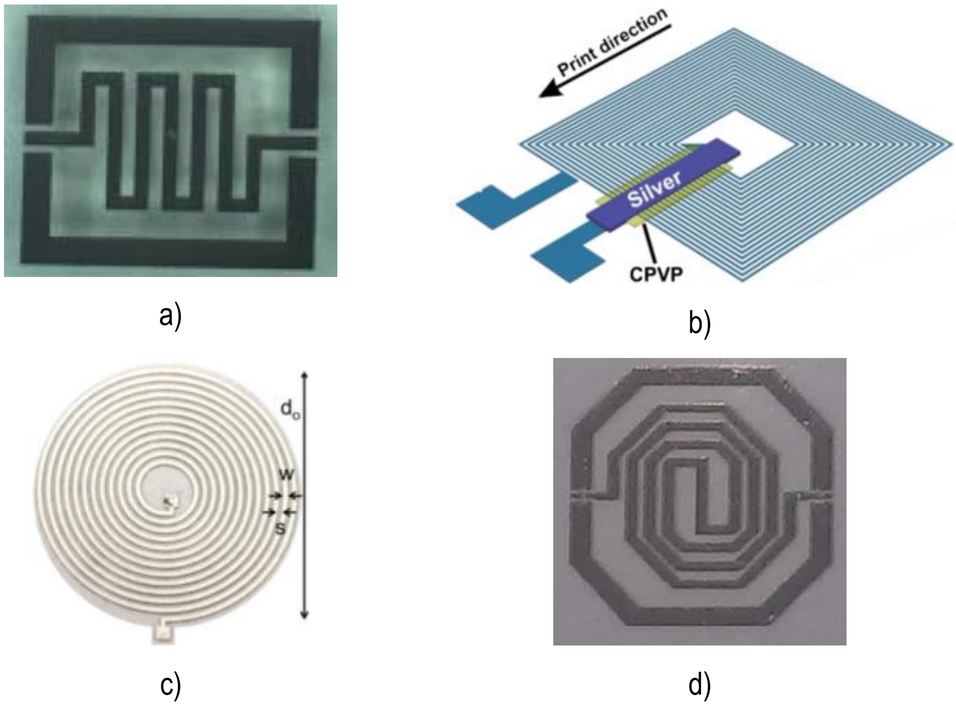


Figure 3-7 Printed inductor: (a) meander inductor (Menicanin, et al., 2015) (Menicanin, et al., 2013), (b) square inductor (Correia, et al., 2018), (c) Circular inductor (Ostfeld, et al., 2015), (d) octagonal inductor (Menicanin, et al., 2015).

3.5.1 Meander Inductor

Menicanin presented an inkjet-printed meander inductor (Menicanin, et al., 2015). In her articles, PET film (Novele IJ-220) with a thickness of 140 µm from the company Novacentrix was used as substrate. Water-based Nanoparticle silver ink Metalon® JS-B25HV (25% wt of silver) was printed as a conductive layer. Printing equipment was DMP3000 from Fujifilm. The printer was adjusted with 25 µm drop space, and the drop volume was 1 pL.

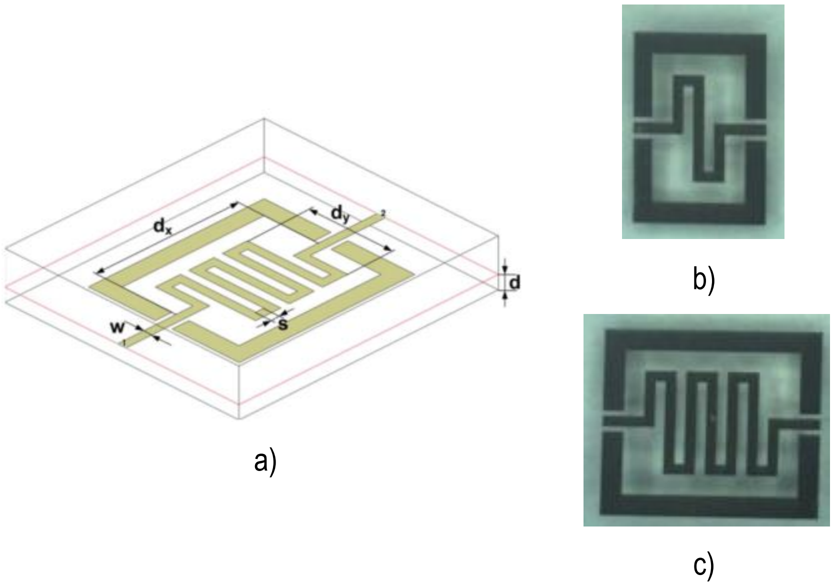


Figure 3-8 Meander inductor a) geometric model (Menicanin, et al., 2013), b) inkjet printed meander inductor one turn (Menicanin, et al., 2015), c) inkjet-printed meander inductor three turns (Menicanin, et al., 2015)

Two different meanders were printed with one and three turns. Its geometric model and actual product are presented in Figure 3-8. The width of the conductive line and the spacing between them were the same w = s = 200 µm. Inner dimension d_x was 1.8 mm for one turn and 3.4 mm for three turns. Inner dimension d_y was the same 2.0 mm for both 1 and 3 turns. A ring-shaped ground plane of 400 µm width was surrounding the inductor. Post-treatment was implemented with an oven at 110°C for 40 minutes. The average layer thickness of printed layers was approximately 800 nm.

3.5.2 Square Inductor

The inkjet-printed square inductor was found in Correia's article (Correia, et al., 2018). PEN film Teonex Q65HA was used as substrate. UT DOTs nanoparticle Silver ink (UTDAgIJ1) was used for printing the conductive layer. Dielectric material C-PVP was printed for isolation between two electrodes. Inkjet printer Fujifilm Dimatrix DMP3000 with 10 pL drop volume cartridge was used. Drop space was adjusted with 40 µm for the conductive layer and 15 µm for the dielectric layer. The square inductor was printed with 100 µm line width, 75 µm line spacing, and 25 turns. After printing, the substrate was sintered in the oven at a temperature of 150°C for 30 minutes. The square inductor was measured, and its inductor was 5.42 µH ± 13%.

3.5.3 Circular Inductor

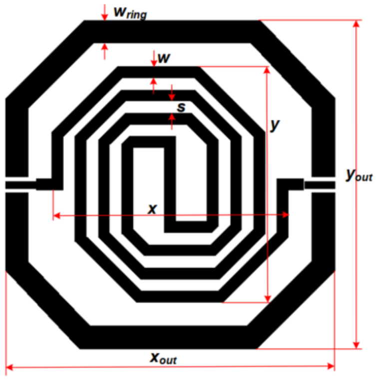
Screen printed circular inductor was introduced by Ostfeld (Ostfeld, et al., 2015). Dupont 5028 silver micro flake ink was printed on PET substrate with Asys ASP01M screen-printer. The screen was made from stainless steel with 400 threads per inch mesh size. The printing parameter was adjusted at 60 mm/s printing speed, 55 N squeegee force, and 1.5 mm snap-off distance. The printed substrate was sintered in an oven at 140°C for 10 minutes. The geometric parameters and the inductance were described in Figure 3-7 c) and Figure 3-9.

Inductor Name	Outer Diameter (cm)	Line Width (μm)	Space Width (μm)	Number of Turns	Inductance (μH)	
					Calculated	Measured
L1	3	500	500	12	2.3	2.23 ± 0.02
L2	5	500	500	12	7.2	7.06 ± 0.04
L3	5	1000	500	13	4.7	4.720 ± 0.002
L4	6	1000	500	16	8.0	7.839 ± 0.005

Figure 3-9 Geometric parameters and inductance (Ostfeld, et al., 2015)

3.5.4 Octagonal Inductor

An inkjet-printed octagonal inductor was reported in Menicanin's article (Menicanin, et al., 2015). Water-based silver ink (Metalon® JS-B25HV) was printed with the inkjet printer Fujifilm Dimatrix DMP3000 on PET substrate Novele IJ-220. The 1 pL drop volume cartridge was used, and the drop spacing was adjusted to 25 μm. After the printing, the substrate was sintered in an oven at 100°C for 40 minutes. The geometrical structure and parameters are presented in Figure 3-10. This octagonal inductor had an inductance of 20 nH.



Parameters	Dimensions
Number of turns n	4 turns
Width of the conductive line w	300 μm
Spacing between adjacent segments s	300 μm
Width of ring conductive line w_{ring}	600 μm
Inner dimension of the inductor x	6.1 mm
Inner dimension of the inductor y	6.1 mm
Outer dimensions of the inductor x_{out}	8.5 mm
Outer dimensions of the inductor y_{out}	8.5 mm

Figure 3-10 Geometrical structure and parameters of an octagonal inductor (Menicanin, et al., 2015)

4 CAD Software for Electronic Modelling

Electronic computer-aided design (ECAD) is a software tool. It is used to help engineers and designers plan electronic systems, for instance printed circuit boards (PCB). ECAD and electronic design automation (EDA) point to the same software (Team, 2020). Before the ECAD software was introduced, designing, and planning of an electronic system was done on paper. Today a simple drawing and diagram software can do the same task as it was done on the paper. Microsoft Visio is a diagramming and vector graphic software. With the help of electrical engineering templates, Visio can be used for creating electronic schematics (Walker, 2007). Visio has a similar user interface as other Microsoft office software. Figure 4-1 shows the user interface in Visio. Electronic symbols can be found in electrical engineering templates. A third-party symbol database (Visio shapes stencils) could also be imported to Visio (Herber, 2021). But Visio does not have the function included in the modern ECAD program. The popular computer-aided design (CAD) and EDA software not only can draw a schematic plan but also has an automatic processing function that can help design and plan the layout of the printed circuit.

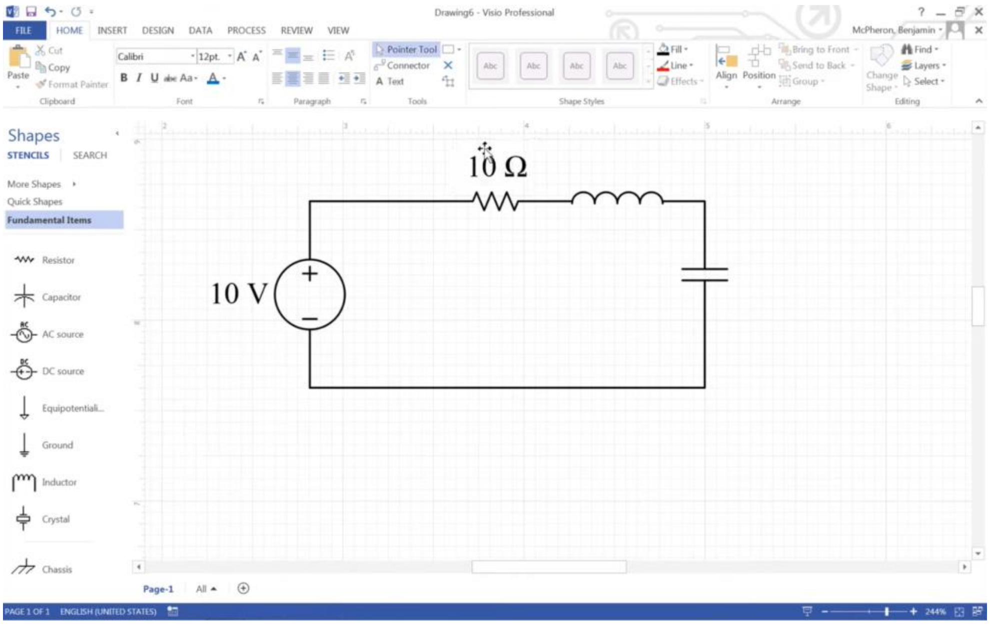


Figure 4-1 Electronic schematic with Visio (DMExplains, 2016)

ECAD software is used to create electronic systems. Mechanical computer-aided design (MCAD) is used to design mechanical systems. ECAD and MCAD software are used together to create products in real situations. A smartphone can be a good example. It is a piece of electrical equipment but has mechanical parts; all the electrical components are placed in an enclosure. Both

ECAD and MCAD are used for developing a smartphone. Figure 4-2 illustrates a simplified life cycle in electronic product development. ECAD and MCAD are both important in designing the product.

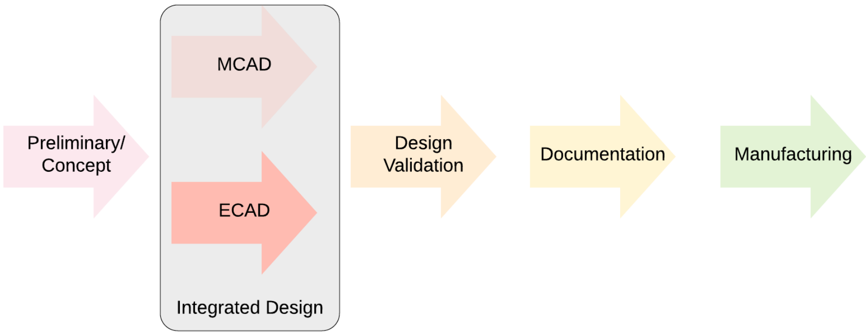


Figure 4-2 CAD in product development (Mafukidze, 2022)

Different ECAD tools are available. A comparison can be found in the literature (softwareradius.com). Depending on the application area, they all have advantages and disadvantages. For example, they may have different features, usability, system complexity, technical support, operation system compatibility, price, etc.

The PCB design workflow is similar in different ECAD software. In the first step, electrical symbols are placed on the schematic plan. The electrical symbols are a part of the software library, or they could be created according to an individual requirement. At this stage, footprints may also be connected with the related symbols when the schematic plan is complete without failure. The next step is, translate the schematic plan into a PCB layout. In this step, footprints are placed on the circuit board, and conductive tracks are placed between components. After this step, the board layout should be ready, and a PCB fabricator can fabricate it. (Keeth, et al.)

4.1 ECAD Modeling Program

4.1.1 Altium Designer

Altium Designer is an ECAD software for printed circuit board design (Designer, 2021). It has been in the market for nearly 20 years and considered to be one of the best PCB design programs in the industry (softwareradius.com). It is the most widely used ECAD application for PCB design. Figure 4-3 shows the Altium designer's working environment.

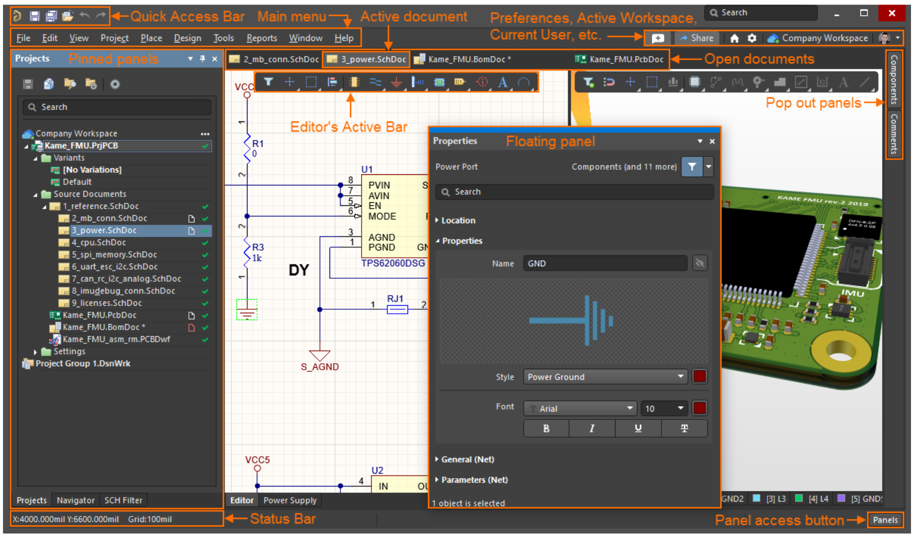


Figure 4-3 Altium designer 22 user interfaces (Designer, 2021)

Some features of this software are described below:

Hierarchical Schematic design is helpful for big projects with many designs and pages included. Each schematic design can be nested or connected with its parent or child design. The CAD software considers the logical relations between each design (Systems). Figure 4-4 shows the overview of hierarchical design in Altium design.

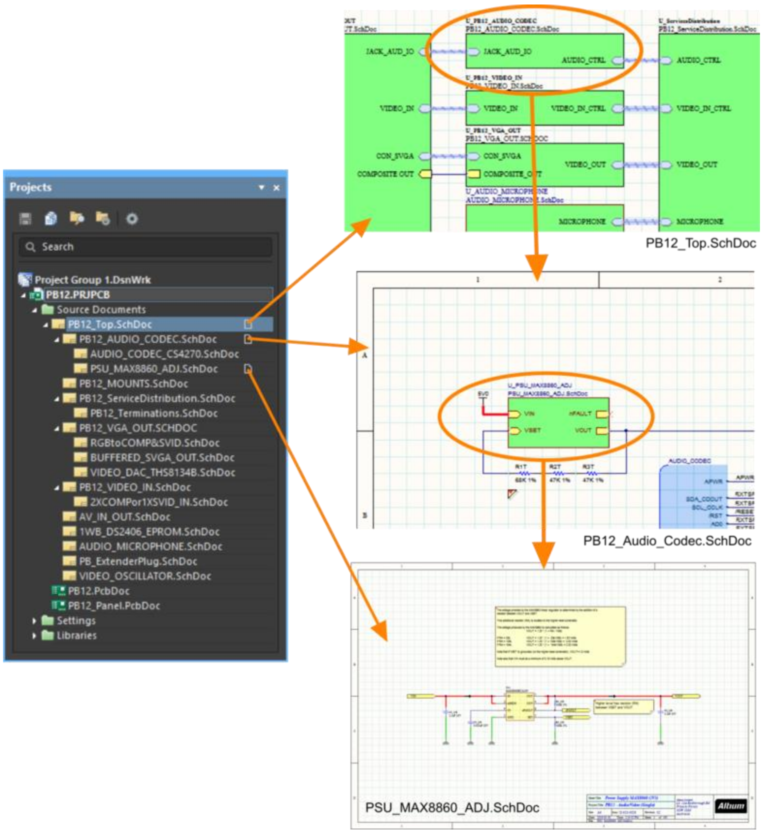


Figure 4-4 Hierarchical design in Altium design (Altium, 2022)

Schematic driver design rules and product lifecycle management (PLM): The design of an electronic system starts with schematic planning. Altium's electronic rule check (ERC) can be applied at the very beginning of the electronic system design or any step of the PLM. It can help find and eliminate mistakes in the schematic by checking the utilized rules. With the help of this tool, the designer can concentrate on the design and not spend too much time checking the failure in the design. (Carlson, 2020) (Keller, 2016)

Extensive library and components: Altium Design contains over 400,000 components in its library (softwareradius.com), it can be maintained separately, and its content is extendable. There are three ways to maintain the library. Workspace Library, Database Libraries, and File-based Libraries. The Workspace library maintains the components in your registered account on Altium Designer Workspace, which is a cloud-based infrastructure platform. The library can be saved in the company database. In this way, the Database Libraries are utilized. The file-based Libraries hold the components locally on the computer as files. (Altium, 2021) Library components can be imported from existing files and created from scratch with your unique design.

Advanced 3D visualization and ECAD-MCAD integration: Altium Designer supports a 3D PCB editor. When the schematic design is translated to PCB, the 3D view of PCB can be presented in Altium Designer. It also supports 3D file transfer between different programs (CoDesign). Combined with the MCAD program, an actual product can be virtually demonstrated on the computer display. At the same time, the geometric collision can be detected by the program, and the design failure can be avoided at a very early stage. Figure 4-5 shows an example of CoDesign. The PCB was designed with Altium, and the transparent enclosure was designed with an MCAD program. The Enclosure was imported into Altium Designer, and the finished product could be demonstrated in Altium 3D view.

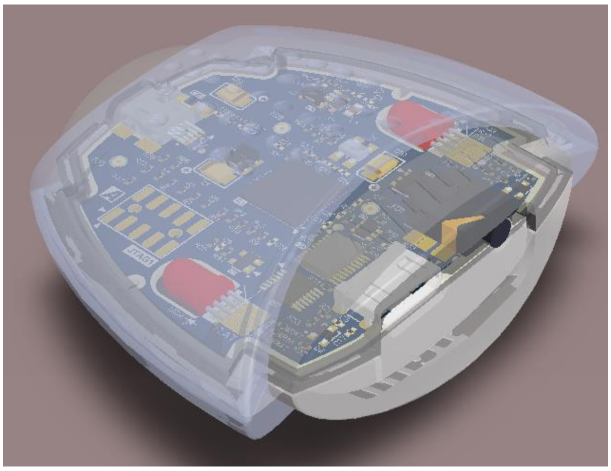


Figure 4-5 ECAD-MCAD Codesign in Altium designer (Altium, 2021)

Interactive routing: The process of defining the path of connecting between nodes means routing. The rule of routing can be defined and adjusted in many ways. This makes the process of design flexible and, at the same time, has the accuracy and correct control of routing. (Altium, 2020)

4.1.2 KiCad

KiCad was initially released in 1992 and is a widely used free software for electrical design. Since the essential features compared to the Altium Designer are the same, it can be a suitable replacement in many situations. (softwareradius, 2022) Figure 4-6 shows the project manager window in KiCad. It was the first view when the KiCad started.

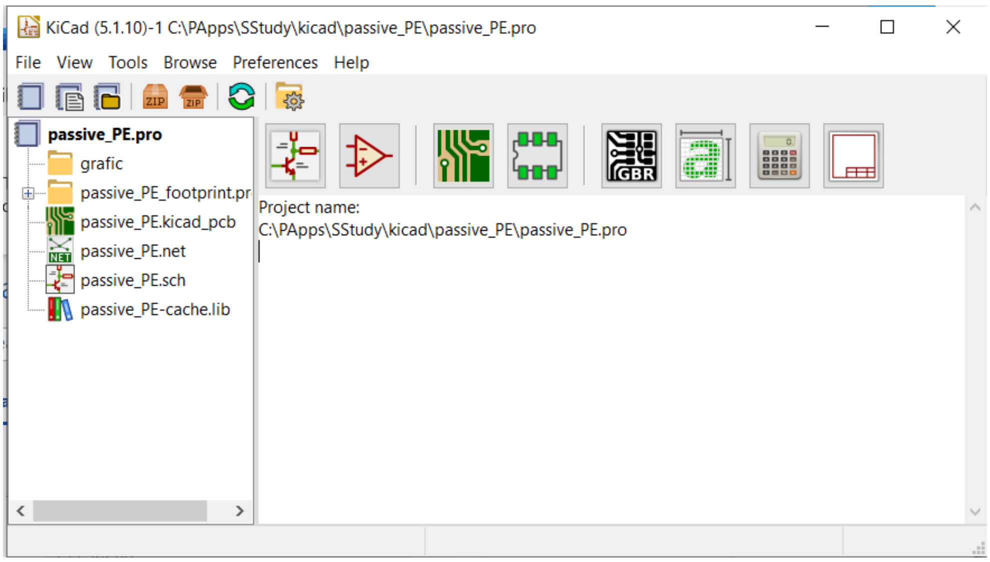


Figure 4-6 KiCad Project manager window

Some features of this software are listed below:

Cross-platform and Free: KiCad can run on different operating systems, for instance Windows, macOS, and Linux. It is free to use, and anyone is allowed to download it from the internet and use it without pay.

Easy-to-use and straightforward: Its user interface is intuitive and easy to understand. Free tutorials and documents can be found on its official website. This makes the learning curve much easier.

Design rule checking: KiCad can automatically check the design failure with defined or custom rules, for example, short cut, conflicts, or unconnected pins. (Keeth, et al.)

Library: KiCad has its library with pre-built components. It supports the import of external components, and new library components can be created by requirement.

3D viewer: 3D demonstration of the PCB board is supported by KiCad. And the 3D model can be exported as a file, which can be used in the MCAD program.

4.1.3 Different Between Altium Designer and KiCad.

Initial investment: Altium is not free software and offers various license types. Its official online store can be found at ttps://digitalsales.altium.com/. The time-based licenses start from 295 Euro/month, and a perpetual license costs 9170 Euro. The KiCad is free software, and the initial investment much lower than Altium.

Operating System support: Altium is only compatible with MS-Windows operating systems. KiCad could be supported by almost all popular operating systems (MS-Windows, macOS, Linux).

Remote collaboration: Altium supports remote-working and cloud-based cooperation. Teamwork with Altium can have a good user experience since it is officially supported. KiCad does not support collaborative PCB design by itself, but there are third-party cloud services that can be integrated with KiCad, for example, www.inventhub.io or Git (Abid, 2020). This makes the remote collaboration possible with KiCad.

Project’s complexity: Altium Designer supports Hierarchical Schematic design. A big project can be separated into related small functions. It does also support the auto-routing feature. KiCad does not support these two valuable functions. Complex and big projects are because of these features getting more convenient with Altium Designer.

MCAD support: booth Altium Designer and KiCad have 3D viewing capability, but Altium has ~~little~~ better support on MCAD, which combines ECAD with MCAD. But the MCAD future in Altium may not be good enough to replace a complete feature MCAD program.

4.2 Fusion 360 & Autodesk Eagle

Fusion 360 is a cloud based MCAD program used in design with the help of computer technology for mechanical equipment. Figure 4-7 presents the fusion 360 interfaces. Different tools, control panels, and 3D models are demonstrated from 1 to 9.

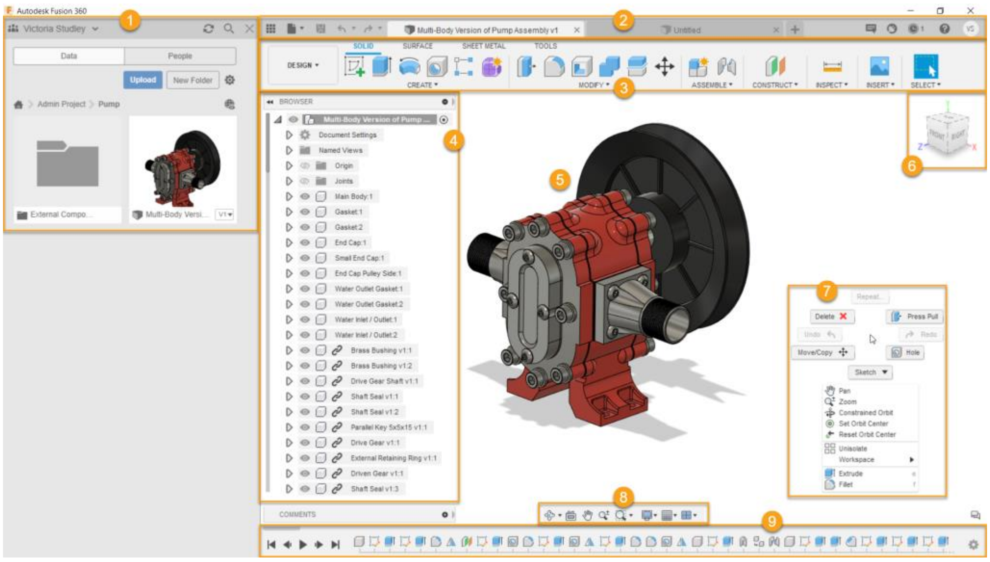


Figure 4-7 Fusion 360 interface (Autodesk, 2022)

Some features of this software are listed below:

Cloud-based storage: Working with fusion 360 requires an internet connection. The files designed with Fusion 360 are saved remotely in the fusion 360 accounts. This makes cooperation over the internet very easy, but it also supports offline working mode. Design can be done offline, and the files will be uploaded when the internet connection is available.

Support different design methodologies: There are two types of design methodologies: bottom-up and top-down. With the bottom-up method, each part of the system needs a detailed parameter before the system can be built. In the top-down methodology, the designer defines the relations between each component (such as the geometry of the parts, sketch) within an assembly. The CAD software organizes the connections between each element. Detailed parameters of the components are evident after the system is completed. Both bottom-up and top-down methodology has their advantages and disadvantages. Fusion 360 provides different possibilities for the designer.

Computer-aided manufacturing (CAM): CAM is supported by fusion 360. The design file can be directly printed with a 3D printer or generate CAM toolpaths for CNC machines. (Autodesk, 2022)

ECAD & MCAD in one program: Fusion 360 is not only for mechanical designers. In 2020 the ECAD program Autodesk EAGLE was integrated into Fusion 360 (Autodesk, 2021).

Cross-platform: Fusion 360 support different operating system and hardware platform. It can work in Microsoft® Windows®, Apple® macOS.

Different price alternatives: Fusion 360 offers various price alternatives. The software is principal proprietary software. For non-commercial and personal use, this software is free for the first year and includes essential features. The limitation is the complexity of the project. The primary feature is more than enough for a simple project with fewer components. Commercial users can use this software and pay 60 euros each month or annually, approximately 500 euros.

4.3 ECAD Library & Workflow

The ECAD software helped us design electronic equipment. Different components are placed in the design, and they are carefully selected and connected according to the engineering requirements. With the ECAD program, all the design tasks are processed virtually on the computer. An electronic component was presented in the library differently. On the stage of schematic design, a component was presented as a schematic symbol. To design the PCB layout, the parameter of the footprint was required. A 3D model of the component was used to demonstrate the actual component on the computer monitor. All the three parts together presented an actual electrical component in the library. A collection of the symbols, footprints and 3D models formed an electronic components library. Commonly used electronic components were included in most ECAD software. Figure 4-8 and Figure 4-9 show the library in Altium Designer and KiCad.

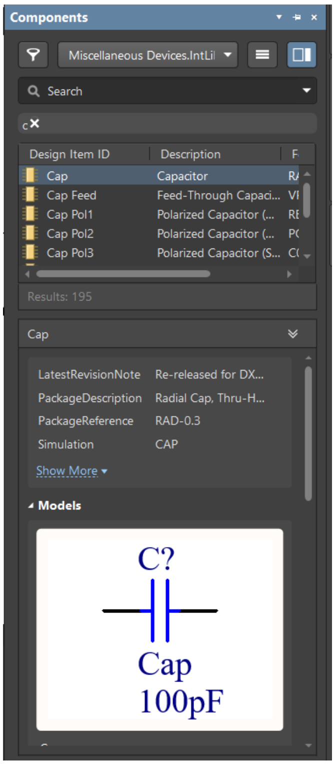


Figure 4-8 Components panel in Altium Designer

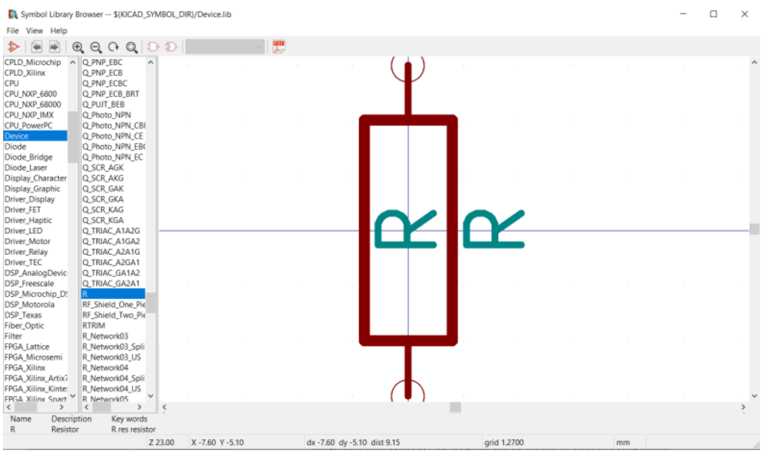


Figure 4-9 Library panel in KiCad

However, new devices and functions appear, and they may not be directly available in the software. The new components must be created in the library before using it in the design project.

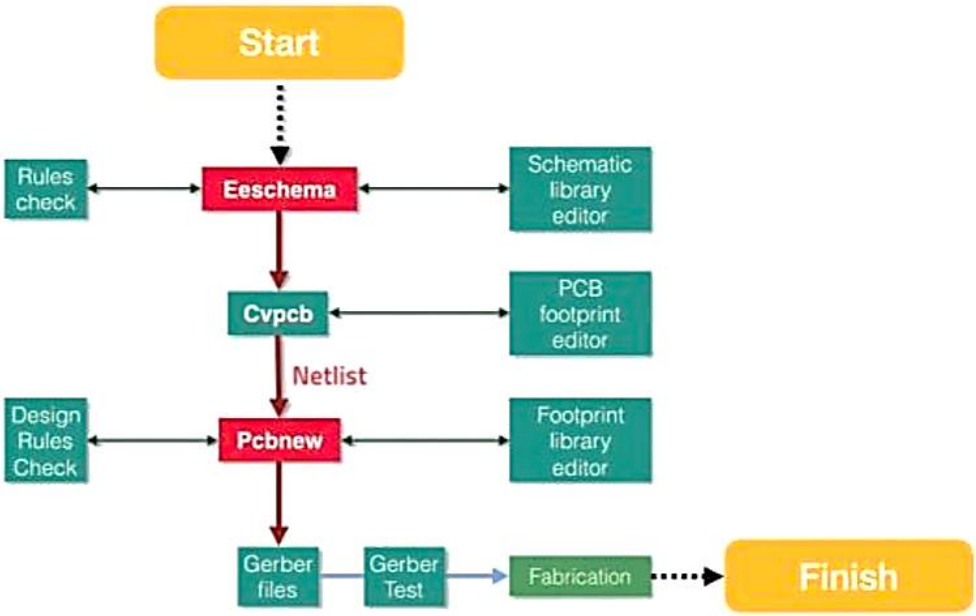


Figure 4-10 PCB design workflow (Dalmaris, 2019)

Figure 4-10 demonstrates a KiCad workflow in PCB design. The process in other ECAD programs is similar. It started with a schematic design (Eeschema). Symbols were placed from the schematic library on the schematic plan. After this, the route was designed, and it connected the pin of the symbols. The schematic plan with routing (netlist) was translated to the PCB design. In this step, the footprint library was used. The PCB layout was checked, failure could be detected and corrected, and the particular files (Gerber files) were exported. The fabrication files were submitted to the PCB manufacturer, and the PCB could be fabricated.

5 Library and Application with PE Components

Printed electronics is a new technology unlike the traditional electronic components, which were already included in most ECAD libraries. When an electronic project includes printed electronic components, the first step in the workflow is to create these components in the library. This chapter demonstrates the detailed process for creating the ECAD library's PE components.

5.1 PE Resistor with KiCad

5.1.1 New Project and Library Category

In KiCad, all the schematic design and new library components are in one project. The first step in working with KiCad is to create a project. A new project is created by following the menu "File → New → Project…" (Figure 5-1).

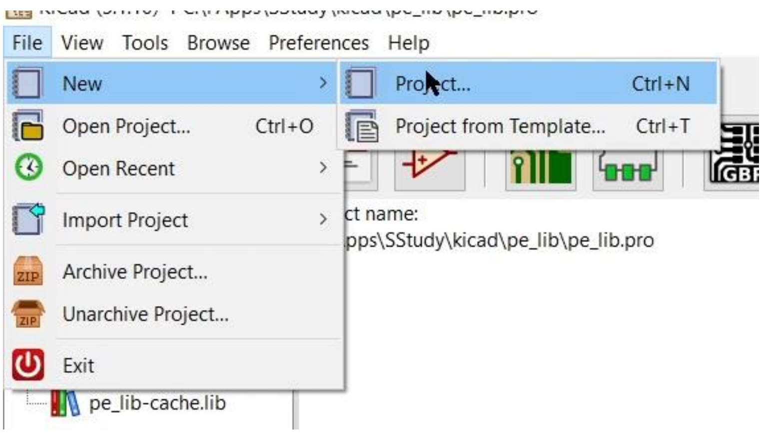


Figure 5-1 Creating a new KiCad Project

Figure 5-2 shows the KiCad interface. Different quick links and tools are marked in the red rectangle. These can be used for creating an electronic components library.

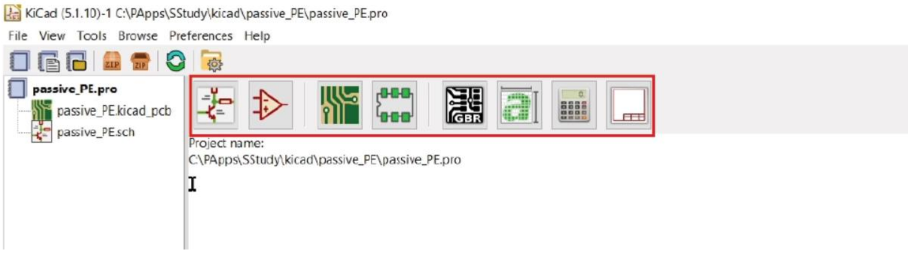


Figure 5-2 A new KiCad project

Click the footprint editor symbol (Figure 5-3), and the footprint editor program (Figure 5-4) is started.

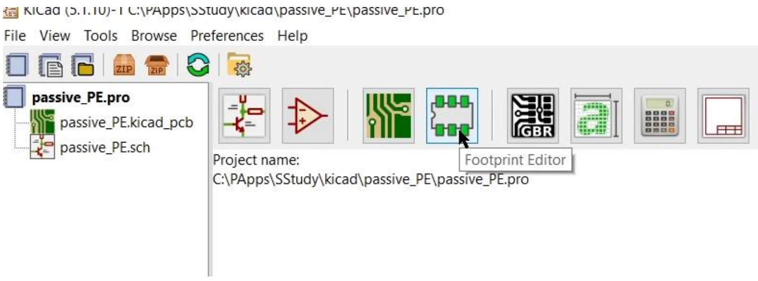


Figure 5-3 Footprint editor symbol

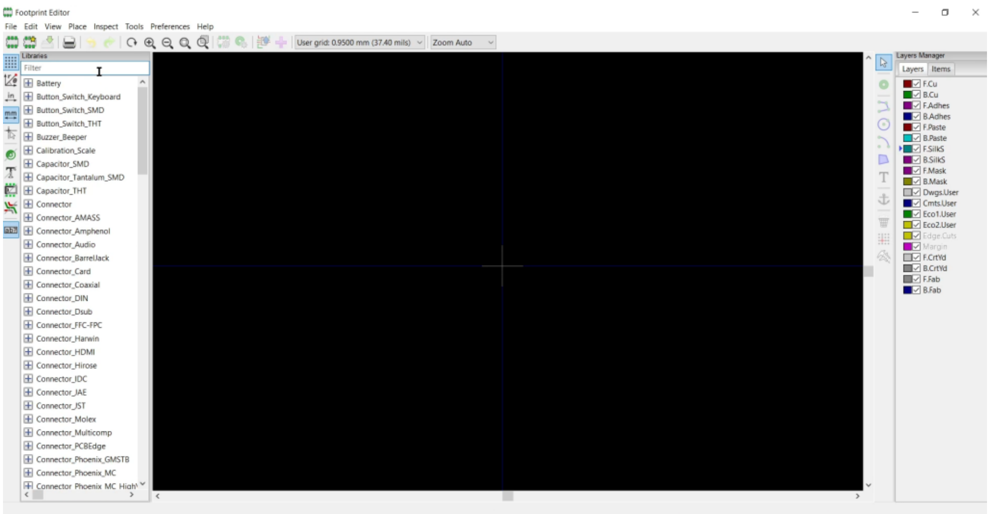


Figure 5-4 Footprint editor

Since KiCad is developed for traditional electronic design, PE components can not be found in the library. A new library needs to be created. Right-click in the local libraries (or in the menu File), then click on “New library” (Figure 5-5), and a new category of the library is created.

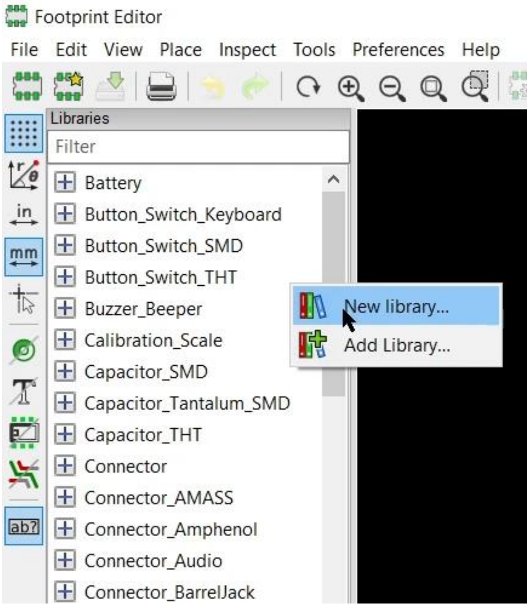


Figure 5-5 Creating a new footprint library

All the PE components were planned to be saved in the new category (Figure 5-6, "passive_PE_footprint").

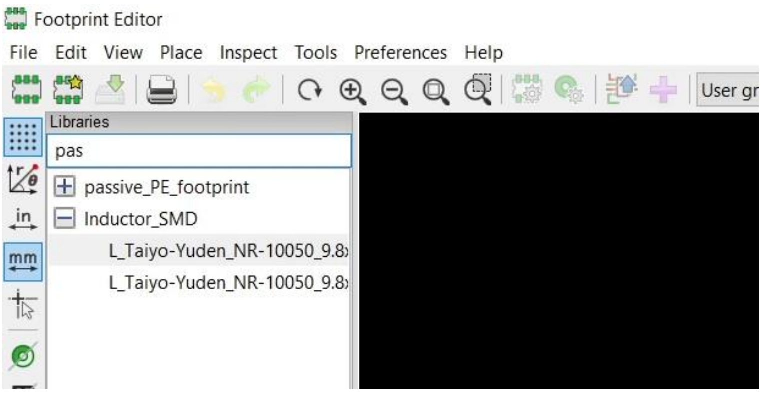


Figure 5-6 A new library for PE components

5.1.2 New Footprint

Start making a new footprint by clicking the symbol "New footprint" (Figure 5-7).

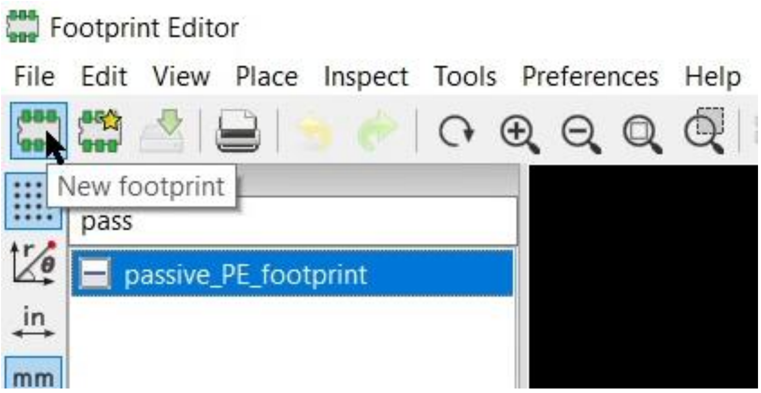


Figure 5-7 New footprint symbol

First, the “add pad” symbol () in the footprint editor toolbox is chosen. Then two pads are created in the footprint editor. Depending on the component, which will be mounted on this pad, the pad’s dimensions can be different. In this case, the pad has 3 mm in width and 2.5 mm high. Pad properties could be activated by double-clicking on the pad. Figure 5-8 shows the properties of the pad.

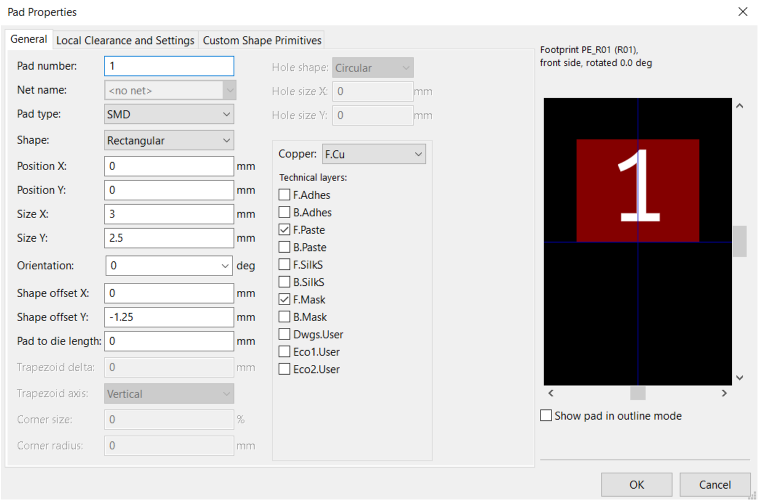


Figure 5-8 Pad properties

Figure 5-9 demonstrates a footprint for a PE resistor, which can be used for mounting the resistor mentioned in reference (Correia, et al., 2018). Its 3D view is shown in Figure 5-10.

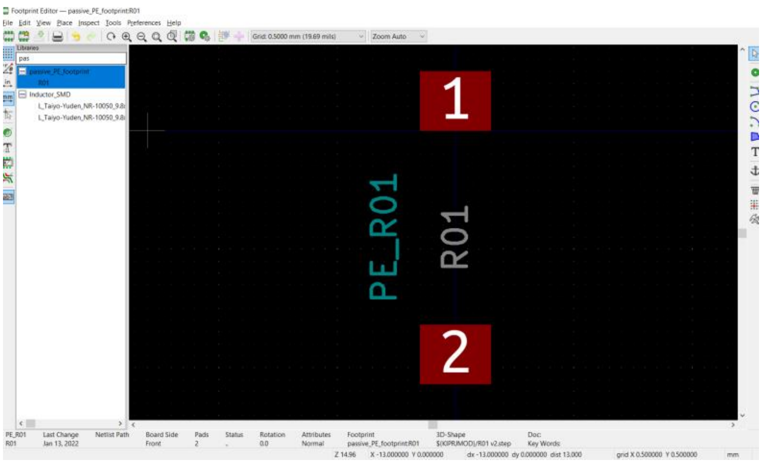


Figure 5-9 Footprints of a PE resistor

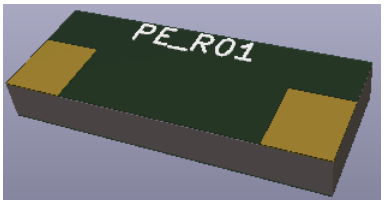


Figure 5-10 Footprints of a PE resistor in 3D view

5.1.3 3D Modelling PE Resistor

As part of the KiCad function, a 3D Model of electronic components can be virtually present. The 3D view in Figure 5-10 shows only the footprint without a related resistor. Unfortunately, KiCad does not include a 3D modeling tool for electronic components. Instead, a 3D Model of an electronic component can be imported into KiCad. To create a 3D Model for PE components, another CAD program (Fusion 360) was applied. Figure 5-11 shows the sketch tool in Fusion 360. A sketch with related dimensions was drawn. The Centre of Figure 5-11 offers the sizes of the PE resistor.

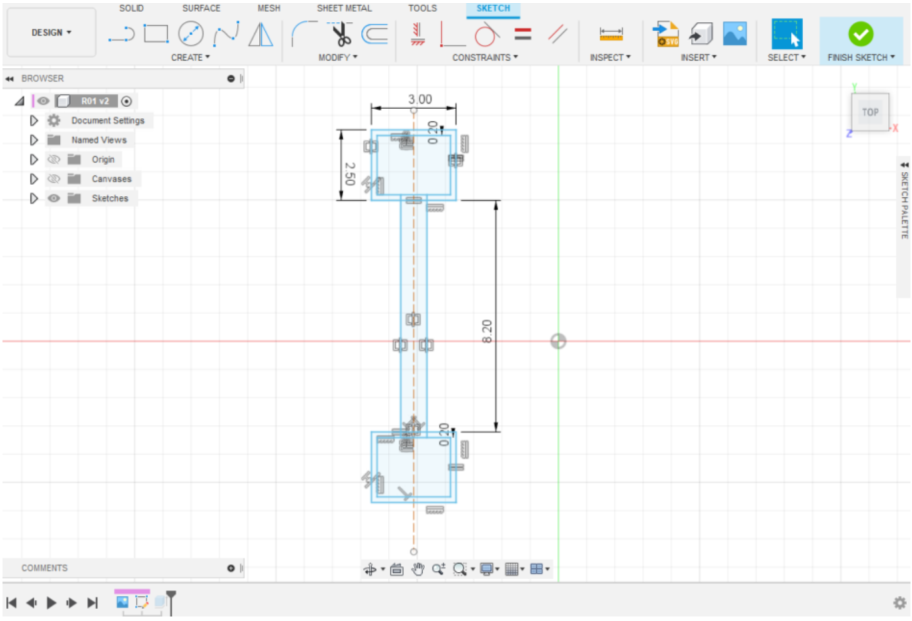


Figure 5-11 Sketch tools and PE resistor dimensions

Sketch tools defined length and width, and the thickness of the printed layer was generated with the tool Extrude (Figure 5-12). This PE resistor had a thickness of 1.6 µm (Correia, et al., 2018). Accordingly, the parameter for extruding was set at 0.0016 mm. A 3D Model for this electronic component was created.

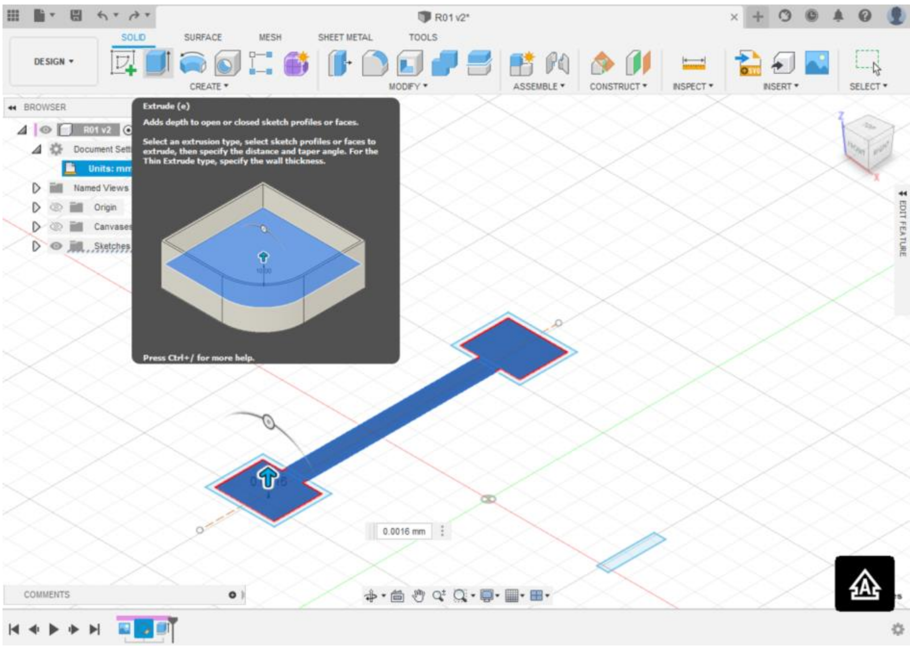


Figure 5-12 Extrude in Fusion 360

This 3D model was exported in a format that KiCad can use. In Fusion 360, select “File → Export”, and an export dialog was opened. They then chose STEP as a file format and exported the 3D model. Then the work in Fusion 360 is completed.

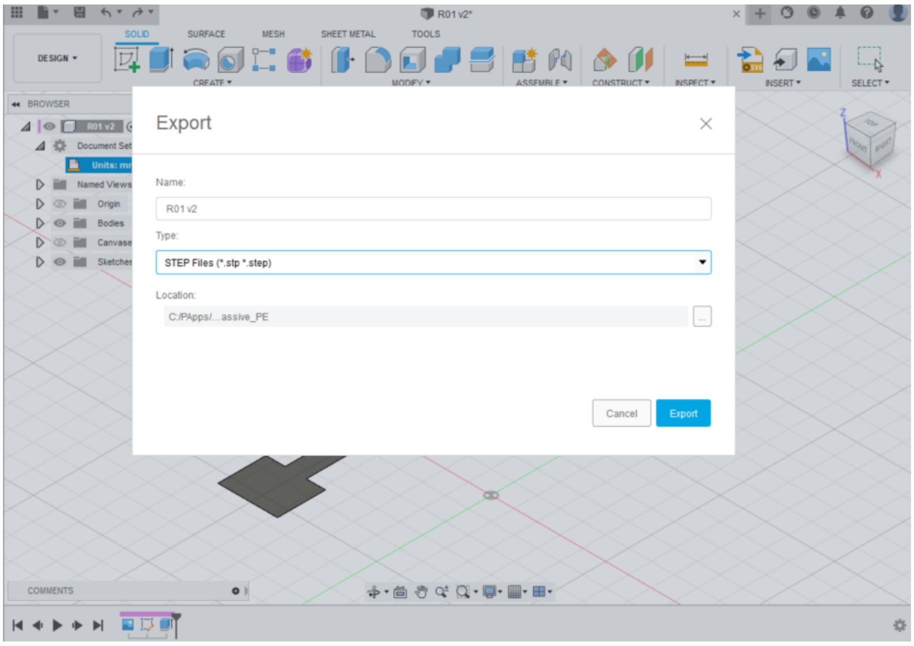


Figure 5-13 Export 3D model

The STEP file has been imported into the KiCad at this step. In footprint editor, click on the symbol “footprint properties” (Figure 5-14 first step, or from the menu Edit → Footprint properties). Then open the STEP file, which is created by Fusion 360. Following the second to fifth steps in Figure 5-14, the 3D model is imported in KiCad.

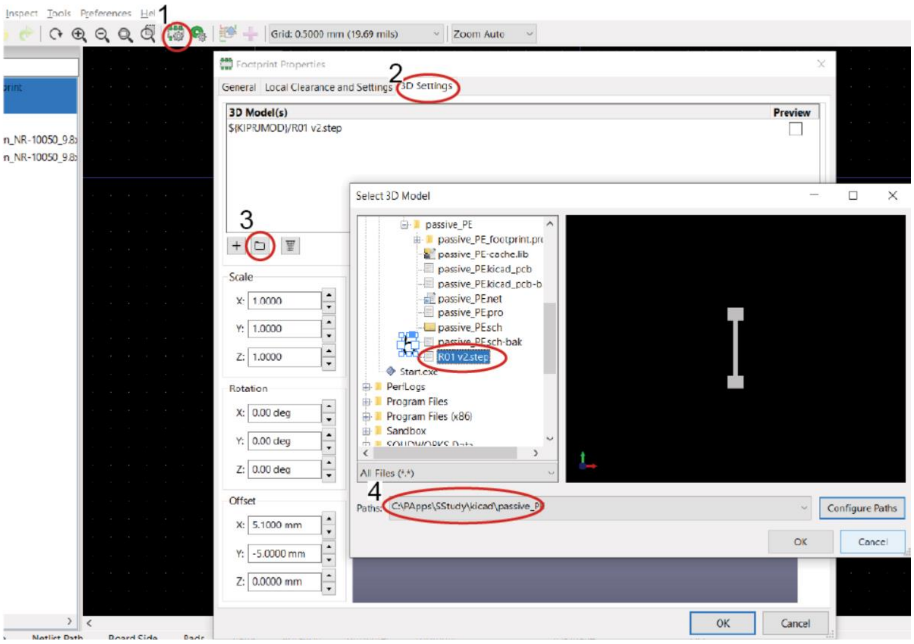


Figure 5-14 Import 3D Model in KiCad

The 3D model may not be perfectly positioned on the footprint. The component's position (Figure 5-15) has to be changed until it is correctly located on the footprint. At the same time, a preview is presented.

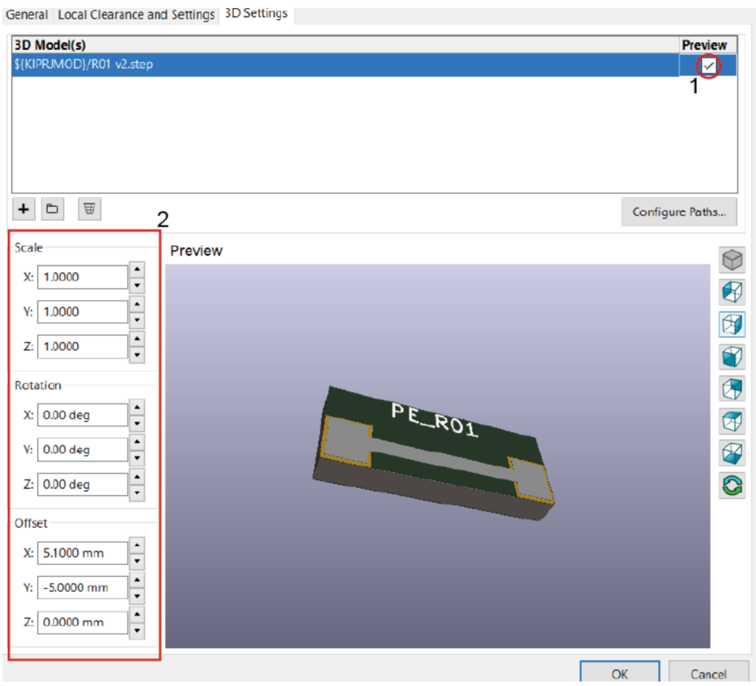


Figure 5-15 3D Model preview

5.1.4 Connecting Electronic Symbol with a Footprint

The electronic symbols are a part of the library. They are necessary for making electronic schematics. The symbol editor can be found in the “Tools → Edit Schematic Symbols.” The previous literature research on PE shows that the same symbols were used in traditional electronics and PE (Correia, et al., 2018) (Ostfeld, et al., 2015) (Menicanin, et al., 2015) (Menicanin, et al., 2013). Since passive PE and traditional electronic components have similar functions and symbols already included in the KiCad Symbol libraries, creating new symbols for passive PE components is unnecessary. However, the KiCad local libraries are not automatically connected with the newly created PE footprint. A new footprint value must be assigned for the symbols during the design process. Figure 5-16 shows the procedure for connecting electronic symbols with customized footprints. Double click the symbol in the schematic layout editor (Figure 5-16 number 1), and the symbol properties dialog opens. Then open the footprint library browser by clicking the red cycle in Figure 5-16, number 2. Finally, select the corresponding footprint in steps 3 and 4 (Figure 5-16).

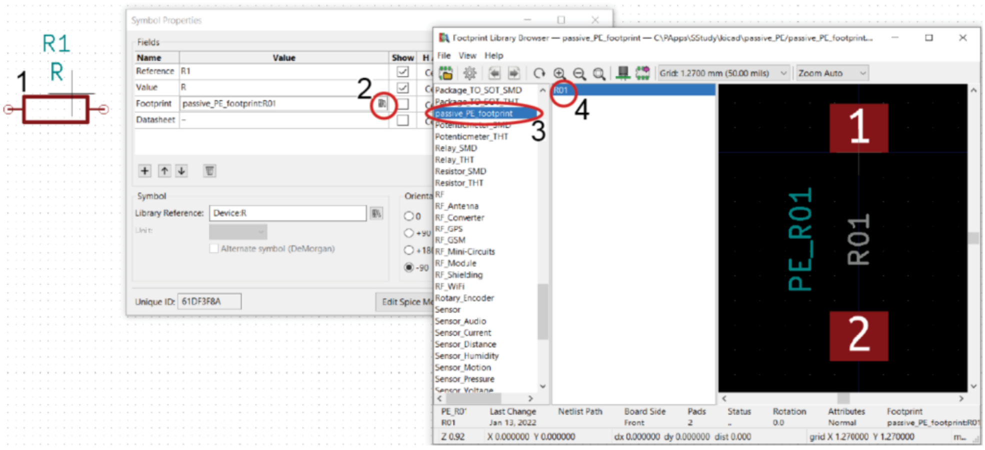


Figure 5-16 Symbol properties and footprint

5.2 Capacitor Library in Altium Designer

Making a new library component in KiCad for PE capacitor is similar to making a resistor component described in chapter 5.1. Compared to this free software, Altium Designer is famous in industrial applications. This chapter descripts the process for creating a printed capacitor library in Altium Designer. It has similar library files (Figure 5-17) as in KiCad. A schematic library is used for creating schematic symbols, which were the primary elements in the electronic schematic plan. A footprint is completed in the PCB library, and a 3D Model is a part of the footprint. (Altium, 2021)

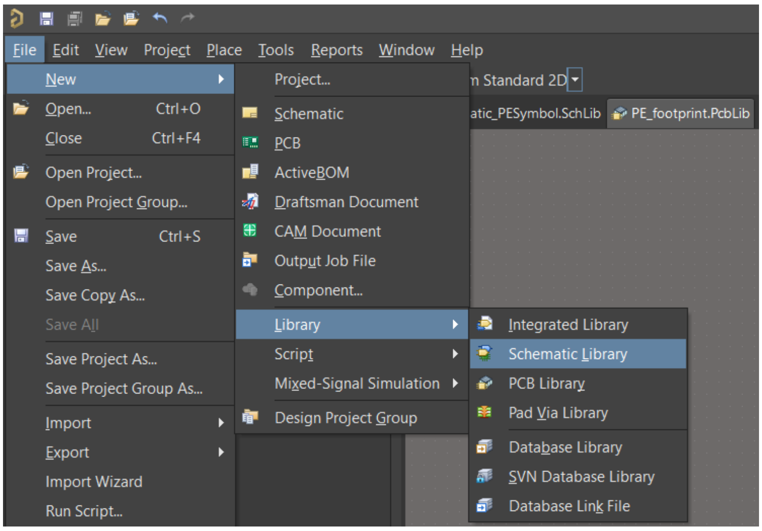


Figure 5-17 Crating the new Altium Designer library

5.2.1 Schematic Symbols

New schematic symbols need to be created inside a schematic library, a container for the schematic symbols. A new empty library was created following the menu "File ⟶ New ⟶ Library ⟶ schematic Library". The library has a file name with the suffix ".SchLib". Double click and open this library. Switch the interface between "projects" and "SCH Library". Figure 5-18 shows the different views between "projects" and "SCH Library". On the left side (Figure 5-18), a new schematic symbol can be created in the "SCH Library".

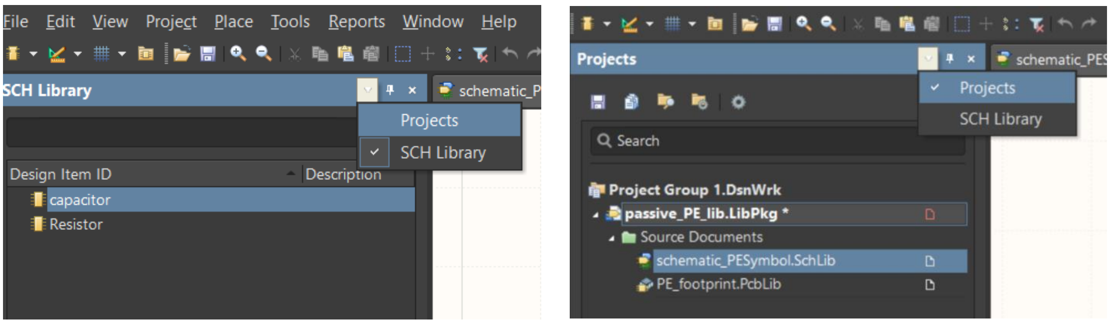


Figure 5-18 Libraries for schematic symbols

Follow the menu "Tools ⟶ New Component" and create a new component. A library component is made without a symbol. Different tools could be used for drawing the symbols. Figure 5-19 presents the tools.

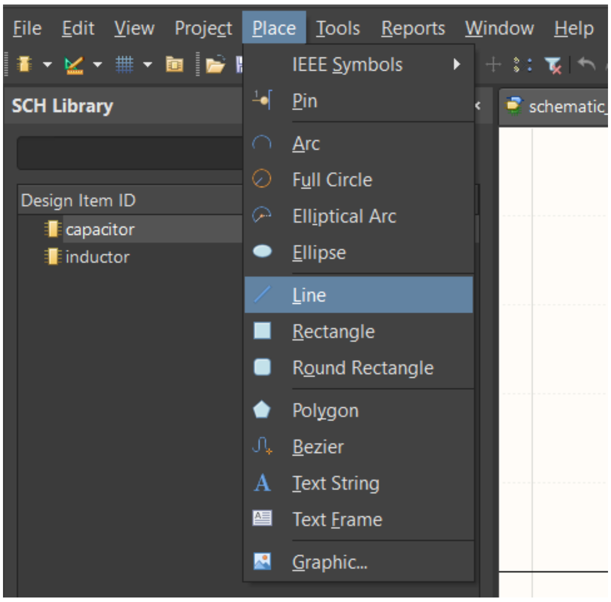


Figure 5-19 Tools for drawing the symbols

Figure 5-20 shows a capacitor symbol, which was created with the tool “Place ⟶ Line” (blue) and “Place ⟶ Pin” (black).

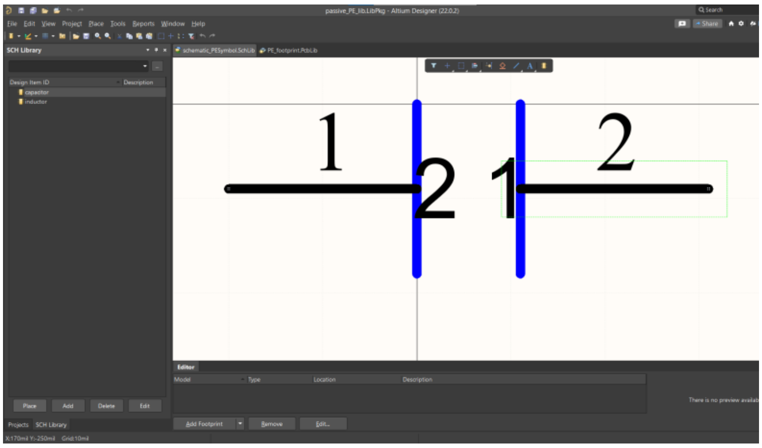


Figure 5-20 Capacitor symbol

5.2.2 Footprint

The footprint was included in the “PCB Library” because this “PCB Library” needs to be created first. Figure 5-21 makes a PCB Library (File ⟶ New ⟶ Library ⟶ PCB Library). The library has a file name with the suffix “.PcbLib”.

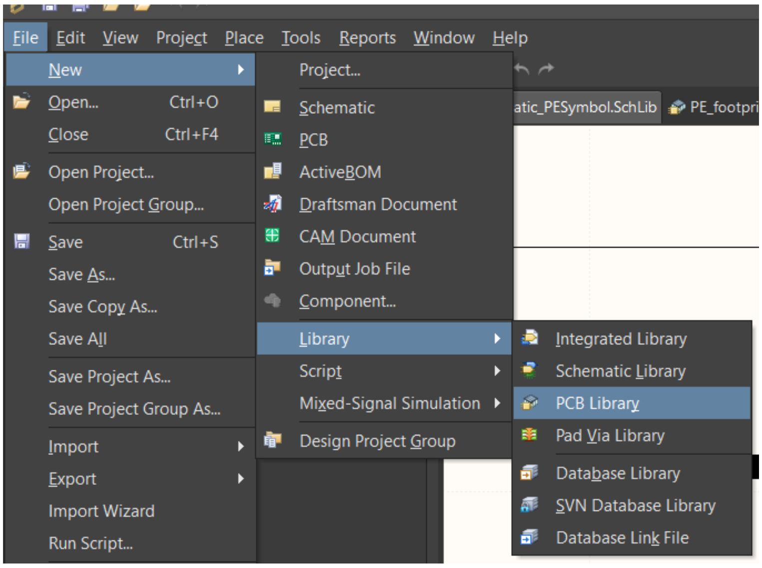


Figure 5-21 Creating a PCB Library

Open the newly created PCB Library and create a new footprint (Tools → New Blank Footprint). Two mounting pads were drawn. These will be the mounting place for the two electrodes of the printed capacitor. The tool was found in the menu "Place → Pad". The properties of the two electrodes were adapted according to the geometry of the printed capacitor. Electrode 1 was placed on the origin of the coordinate. Both x and y are zero. Electrode 2 was placed at the position x = 21.5 mm (46.457 mil), y = 0 mm. Electrodes 1 and 2 had the same width (x = 1 mm) and length (y = 19.7 mm). They were placed on the "Top Layer". The red frame in Figure 5-22 shows the properties of electrode 1.

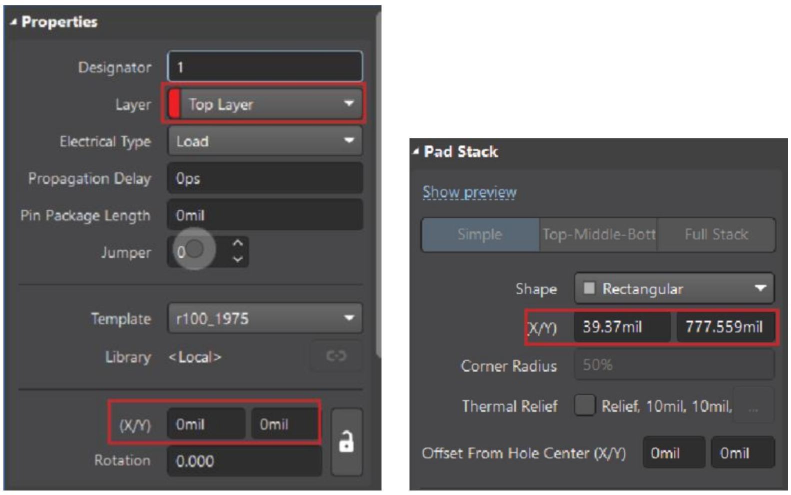


Figure 5-22 Pad properties

In the next step 3D model of the printed capacitor was imported. Select “3D Body” from the menu “Place” (Figure 5-23). Then select the 3D model (chapiter 5.2.3). The capacitor was placed on the “Pad”.

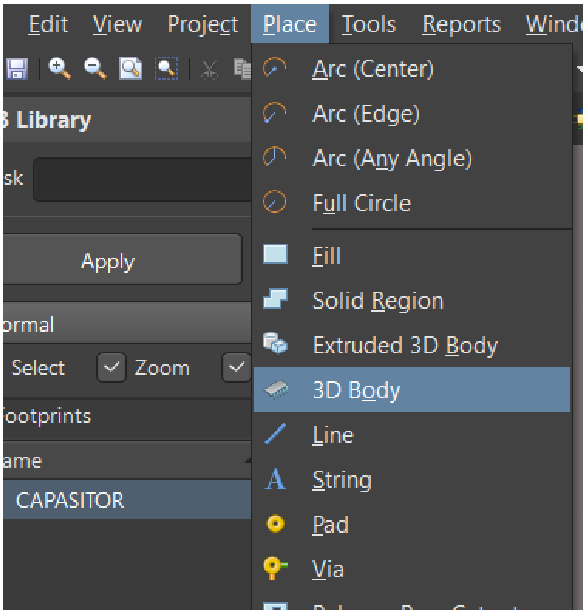


Figure 5-23 3D Body for footprint

Figure 5-24 shows the 2D and 3D footprint layouts. In the 3D layout, the capacitor was presented from the back view. The grey color is the printed capacitor, and the golden area contacts the two electrodes of the capacitor.

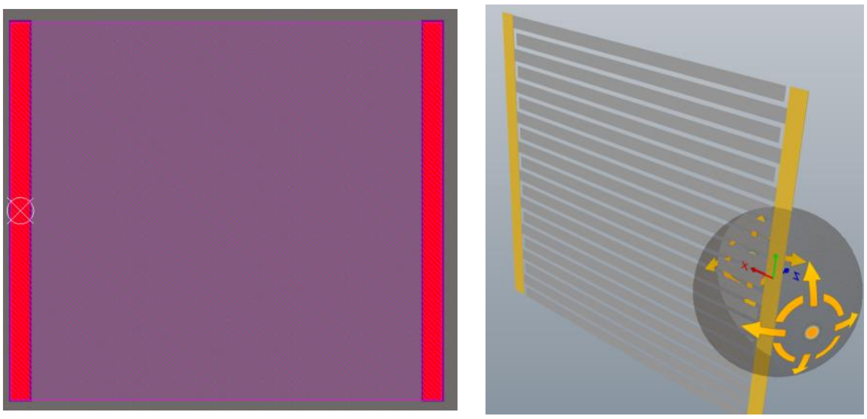


Figure 5-24 PE Capacitor footprint 2D (left) and 3D (right) layout

5.2.3 Creating a 3D Body for PE Capacitor

The 3D model of the printed capacitor was created with Fusion 360. The same technique was implemented, which was described in chapter 5.1.3. The two electrodes had a width of 1.00 mm and 19.75 height. The finger of the capacitor had the dimension 20.25 mm X 0.75 mm, and the space (not printed area) between each finger was 0.25 mm. It had 20 interdigital pairs. The size of the capacitor is shown in Figure 5-25 left. On the right (Figure 5-25), the 3D model was presented.

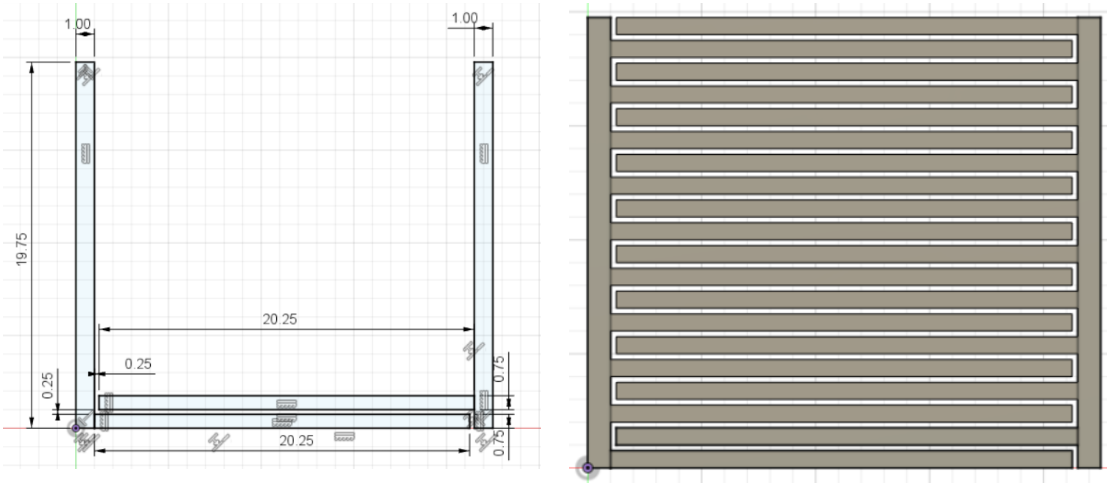


Figure 5-25 Dimensions of PE capacitor (left) and 3D Model (right)

5.2.4 Footprint for Schematic Symbol

Footprint and schematic symbols were created in the previous chapters. The footprint can be assigned to the corresponding schematic symbol. The process has different steps. They were presented with a montaged graphic in Figure 5-26.

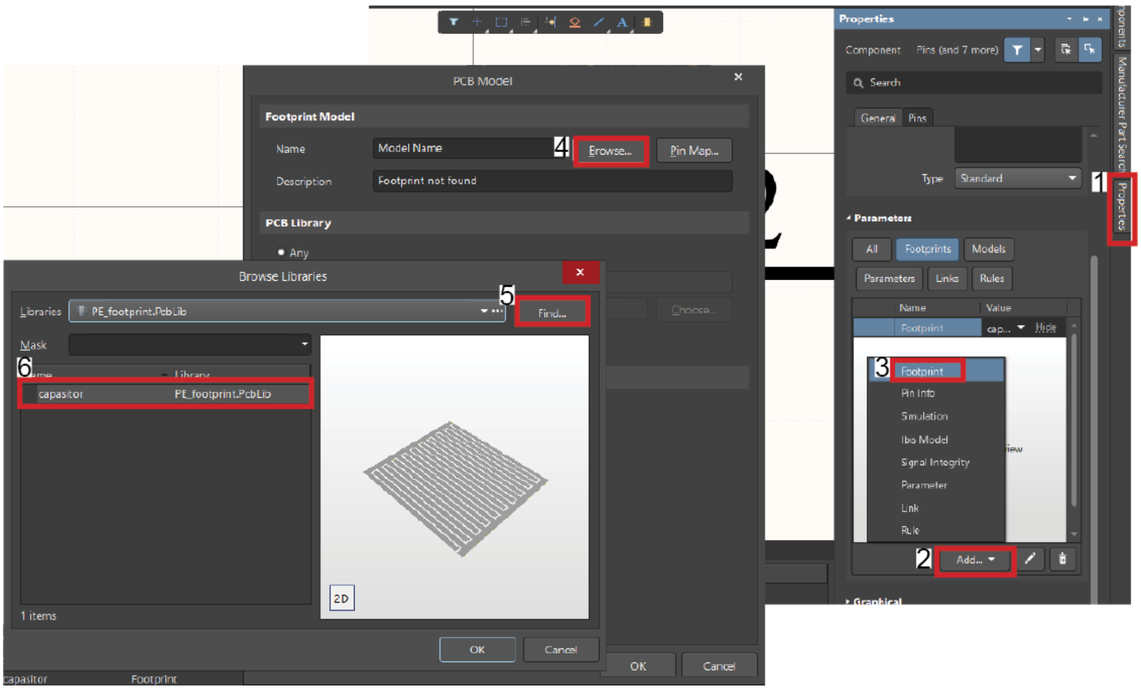


Figure 5-26 Footprints for schematic symbol

In the schematic library, select the component capacitor. Then open the symbol's properties (Figure 5-26 step 1). Then find the parameter *Footprints,* selecting the button "Add…" (Figure 5-26 second step), then select Footprint (Figure 5-26 third step). A new dialog is opened. Browse (Figure 5-26 fourth step) and select (Figure 5-26 fifth step) the PCB library, which contains the footprint of the capacitor. Select the component (Figure 5-26 sixth step), then click OK. The footprint is assigned to the symbol.

5.3 Application of PE library

5.3.1 Project Structure and New Project

Chapters 5.1 and 5.2 demonstrated the workflow for creating resistor and capacitor library components in KiCad and Altium Designer. The workflow for creating an electronic components library is similar in both software. A library component includes a schematic symbol used for the schematic plan. A footprint describes the dimension of the components and is used in planning an actual circuit board—a 3D model, which demonstrates the actual product with the monitor.

This chapter created a simple RLC circuit with Altium Designer including customized library elements. A new project was completed in Altium Designer, and this file has the file name with the suffix ".PrjPcb". Four files were necessary to create a printed circuit, and they were part of the project. A schematic document includes the schematic plan. A schematic library consists of all electronic symbols. A PCB document represents the layout of the printed circuit. A PCB library consists of each electronic component's footprint and 3D model. The red frames in Figure 5-27 show the five necessary files for creating a printed circuit.

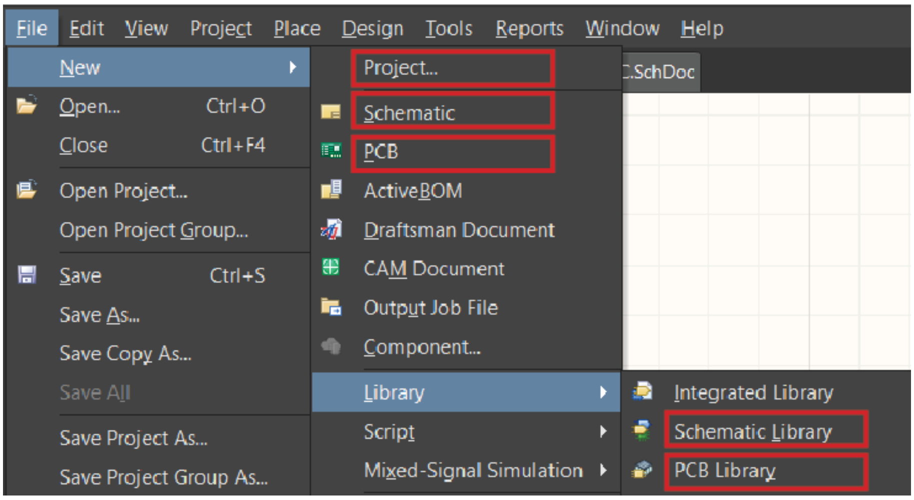


Figure 5-27 New Altium Designer Project

5.3.2 Import Existing Libraries

Since the schematic Library (schematic_PESymbol.SchLib) and PCB Library (PE_fotprint.PcbLib) were already created, they will be used in the new project. Follow the menu “File → Open” and open the libraries highlighted as in Figure 5-28.

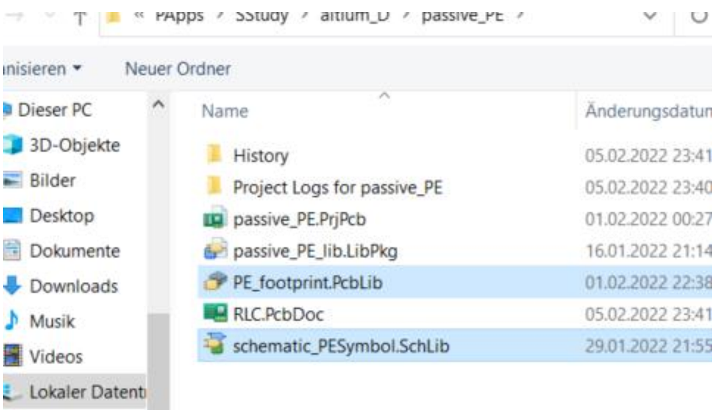


Figure 5-28 Libraries with PE components

The opened libraries were still outside of the project. Figure 5-29 left shows the files “PE_foofprint.PcbLib” and “schematic_PESzmbol.SchLib” are under the “Free Documents”. Left-click and pull the libraries into the current project “RLC_circuit.PrjPcb” (Figure 5-29 right). The libraries are imported successfully.

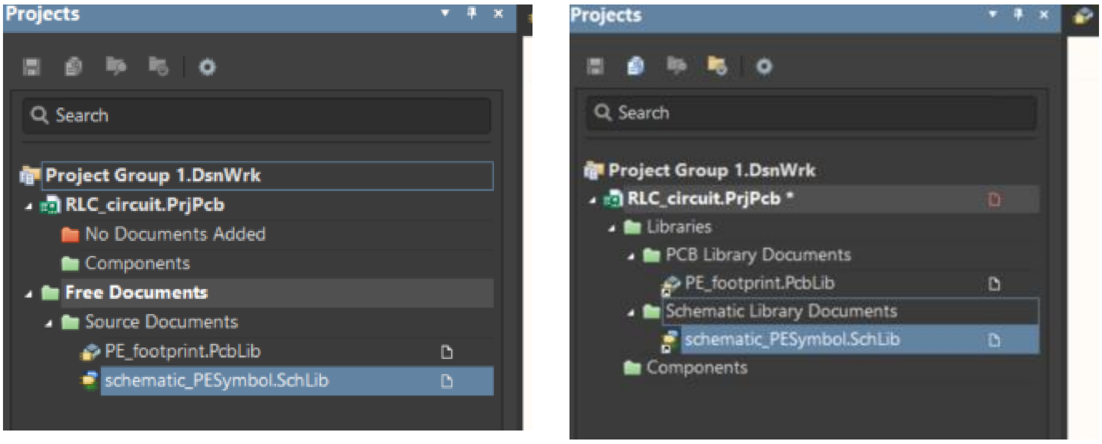


Figure 5-29 Libraries outside (left) and inside (right) the Project

5.3.3 Creating Schematic Plan

A schematic document (File → new → Schematic) and a PCB document (File → new → Schematic) were created. The structure of the project looks as in Figure 5-30. Under the project (file "RLC_circuit.PrjPcb"), there were four files "RLC.SchDoc", "RLC.PcbDoc", and "PE_footprint.PcbLib" and "schematic_PESymol.SchLib", they represented schematic document, PCB document, footprint library, and symbol library.

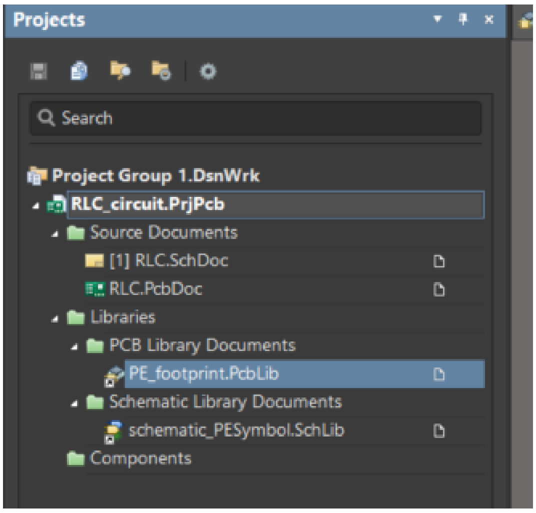


Figure 5-30 New schematic and PCB documents

Double click and open the schematic document "RLC.SchDoc" in Altium Designer. Place the resistor, inductor, capacitor, and a 2-pin header symbol in the document with the tool "place part" (or in the menu "Place → Part"). They then connect all the components by placing the wire ("Place → Wire"). The circuit shows in Figure 5-31, and the symbols are not annotated.

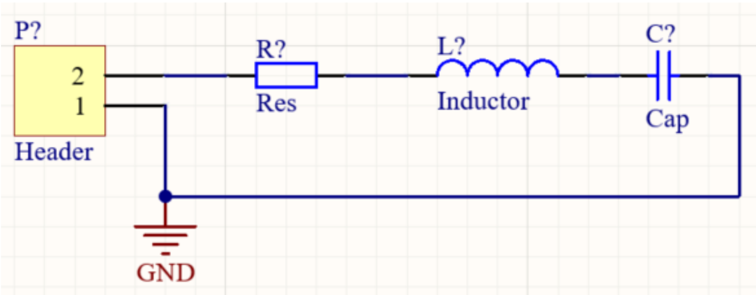


Figure 5-31 RLC circuit before annotation

The “annotation” (in the menu “Tool → Annotation → Annotation Schematics…”) was implemented (Figure 5-32). Then the number of the symbols was updated. The final schematic plan is shown in Figure 5-33.

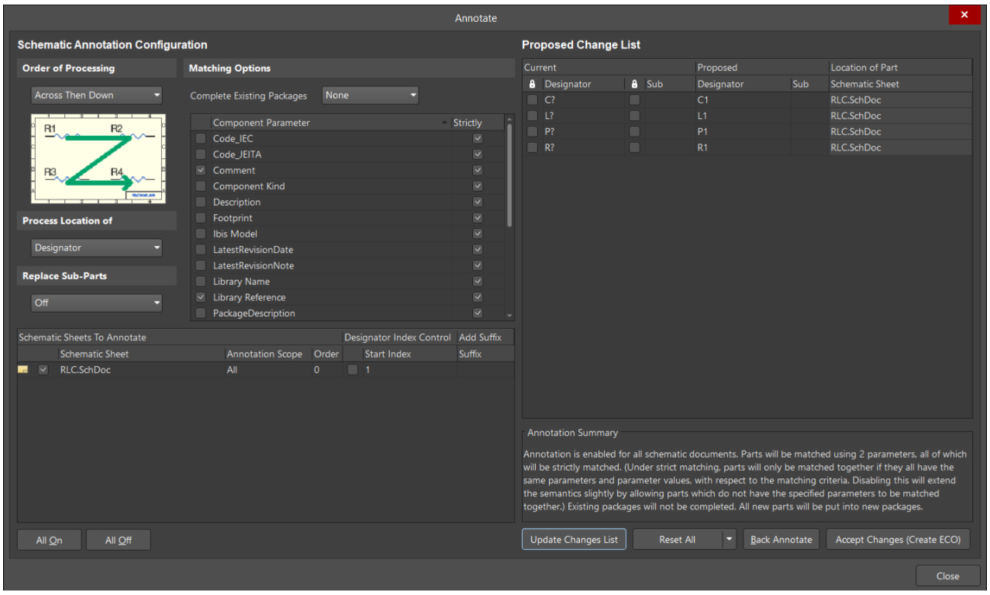


Figure 5-32 Annotate schematics

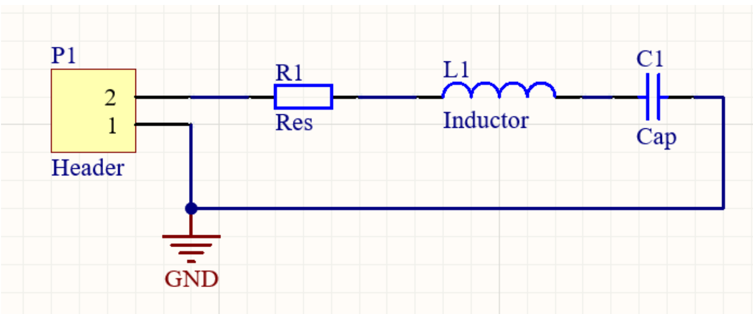


Figure 5-33 RLC circuit schematic plan

5.3.4 Layout of Printed Circuit

Before the printed layout was created, all the design information in the schematic document needed to be translated to the PCB document. A process “Update PCB Document” was executed from the menu “Design ⟶ Update PCB document”, then the changes were validated and executed (Figure 5-34). After this process, the necessary information was transferred from the schematic plan to the PCB layout document (Figure 5-35).

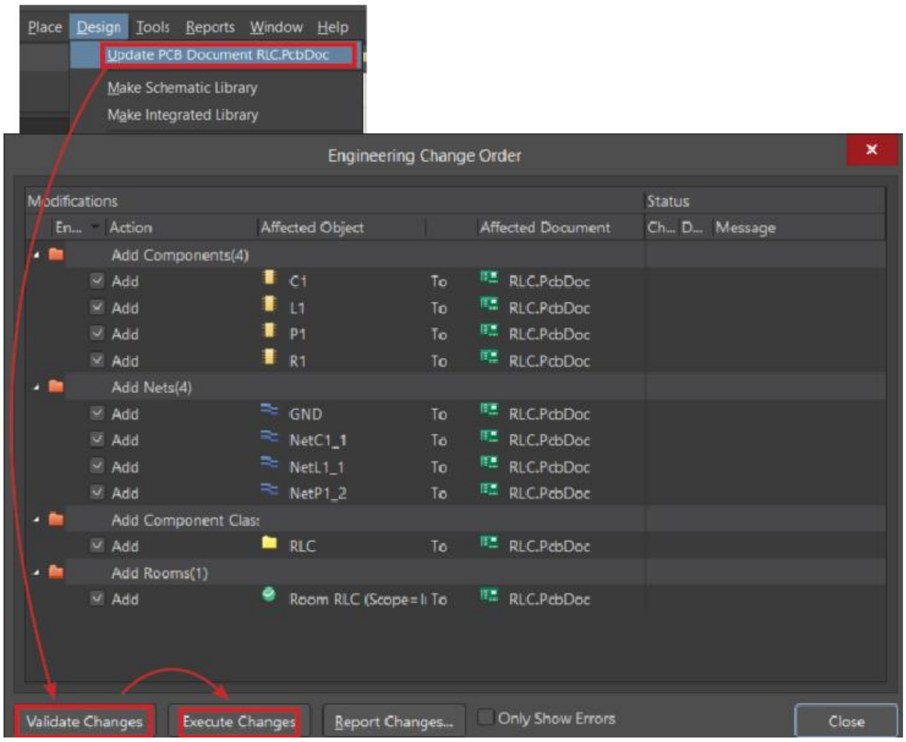


Figure 5-34 Update PCB Document

Figure 5-35 shows the printed circuit with footprint and 3D components. The automatic execution in Figure 5-34 did not wire the components together, and its position was not optimized.

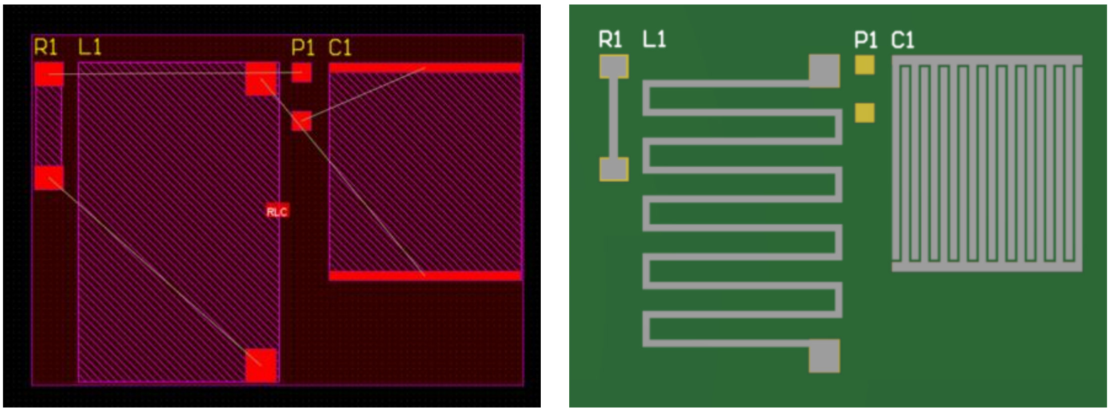


Figure 5-35 2D (left) and 3D(right) layout from schematic plan

The position of the electronic components was changed by moving them with the mouse. In this way, the wire could be placed without overlap. With the help of the tool “interactive routing” (or in the menu “Place → Track”), the resistor, capacitor, and inductor were connected. The final result is presented in Figure 5-36.

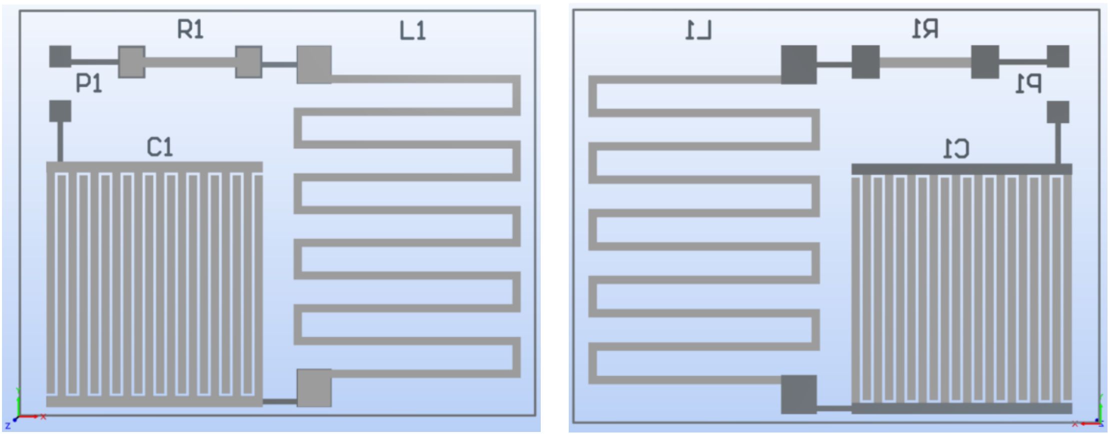


Figure 5-36 Render image of printed RLC circuit on transparent film front (left) and back view (right)

Now a printed RLC circuit with customized library components has been created.

This chapter demonstrated the essential elements (schematic symbols, footprints, and 3D bodies) in creating printed passive components. Two ECAD programs and an MCAD program were used as practical examples. Different library elements (resistor, capacitor, and inductor) were created, and a standard procedure for making a PE library was demonstrated in detailed steps. Figure 5-37 shows the relationship between each element in a CAD Library.

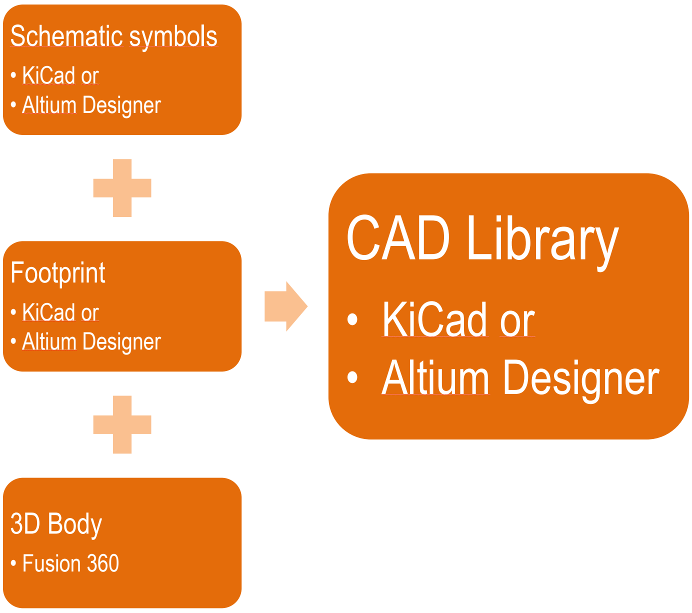


Figure 5-37 Elements of CAD Library and applied software tools

6 Discussion and Conclusion

This work presented a summary of passive printed electronic components, and the difference between traditional passive components and printed passive components was discussed. The fabrication materials, methods, and processes for the printed passive components, especially resistor, capacitor, and inductor, were researched. A way of creating the library for different passive components was demonstrated. In the end, a simple RLC circuit was created with a new passive PE components library.

The CAD software is a big family. Choosing the correct CAD tool for the PE application was a significant change. The learning curve on different CAD software was influenced by personal knowledge background. Three CAD software were introduced in this works to create a PE library. Two of them (Altium designer and KiCad) are especially designed for ECAD applications. One software (Fusion 360) was initially intended for the MCAD application, but now it also includes the ECAD module. Many other CAD programs may also be suitable for creating the passive PE library components. Because of budget and time limitations, creating the PE library for all the software was not the purpose of this work. However, Altium designer and KiCad are excellent and typically represent the other ECAD software. Both are widely used programs. Altium Designer represents industrial standard and professional quality whereas KiCad is about open source, good quality and functionality.

This work shows a potential working process for creating a passive PE components library. By the time this work was completed, a standard library for PE components could not be found yet. Printed electronics is a fast-growing industry in the current and foreseeable future. Using a standard library can promote development and cooperation in this area. Creating a PE library with widely used components and collecting the components in a standard library can be valuable works for the future.

7 References

Abid Usama Better manage KiCAD projects using Git [Online]. - Feb. 11, 2020. - March 12, 2022. - https://medium.com/inventhub/better-manage-kicad-projects-using-git-8d06e1310af8.

Altium Building & Maintaining Your Components and Libraries in Altium Designer [Online]. - Altium LLC, August 25, 2021. - March 05, 2022. - https://www.altium.com/documentation/altium-designer/building-maintaining-components-libraries-overview.

Altium Interactively Routing Your PCB in Altium Designer [Online]. - Altium, November 27, 2020. - March 06, 2022. - https://www.altium.com/documentation/altium-designer/interactive-routing-pcb.

Altium Multi-sheet & Multi-channel Design in Altium Designer [Online]. - Altium Limited, January 12, 2022. - March 05, 2022. - https://www.altium.com/documentation/altium-designer/multi-sheet-multi-channel-design.

Altium The 3D Advantage in ECAD-MCAD Integration [Online]. - Altium, December 15, 2021. - March 06, 2022. - https://www.altium.com/documentation/altium-designer/3d-advantage-ecad-mcad-integration.

Anto Bibin T. [et al.] Hydrophilic Sparse Ionic Monolayer-Protected Metal Nanoparticles: Highly Concentrated Nano-Au and Nano-Ag "Inks" that can be Sintered to Near-Bulk Conductivity at 150 °C [Journal] // Advanced functional materials. - 2010. - 2 : Vol. 20. - pp. 296-303.

Autodesk Autodesk EAGLE now included with Fusion 360 [Online]. - autodesk, November 17, 2021. - March 23, 2022. - https://knowledge.autodesk.com/de/support/fusion-360/troubleshooting/caas/sfdcarticles/sfdcarticles/DEU/Autodesk-EAGLE-now-included-with-Fusion-360.html.

Autodesk Get Started in Fusion 360 [Online]. - Autodesk, Inc., 2022. - March 23, 2022. - https://help.autodesk.com/view/fusion360/ENU/?guid=GUID-E647CA56-7187-406A-ACE4-EAC59914FAE4.

Bird John Electrical and Electronic Principles and Technology [Book]. - [s.l.] : Elsevier Ltd., 2010.

Blecha Tomas, Harant Jaroslav and Syrovy Tomas Influence of Aging on Printed Passive Components Parameters [Conference] // 2017 40th International Spring Seminar on Electronics Technology (ISSE). - Sofia, Bulgaria : [s.n.], 2017. - pp. 1-6.

Caironi Mario and Noh Yong-Young Large Area and Flexible Electronics [Book]. - Weihneim, Germany : Wiley-VCH, 2015.

Carlson Chris Eliminate Mistakes in Your Schematics with Electronic Rule Checking [Online]. - Altium, November 10, 2020. - March 05, 2022. - https://resources.altium.com/p/eliminate-mistakes-in-your-schematics-with-electronic-rule-checking.

casx123 HIstory of Electronic [Online]. - 2021. - September 01, 2021. - https://www.timetoast.com/timelines/91979.

Constable G. and Somerville B. A Century of Innovation: Twenty Engineering Achievements that Transformed our Lives [Book]. - [s.l.] : Joseph Henry Press, 2003.

Correia V. [et al.] Design and fabrication of multilayer inkjet-printed passive components for printed electronics circuit development [Journal] // Journal of Manufacturing Processes. - 2018. - Vol. 31. - pp. 364–371.

Cui Zheng Printed Electronics: Materials, Technologies and Applications [Book]. - Singapore : John Wiley & Sons, 2016.

Dalmaris Peter KiCad Like a Pro (2nd Edition) [Book]. - Austrialia : ELEKTOR, 2019.

Designer Altium Exploring Altium Designer [Online]. - August 17, 2021. - February 21, 2022. - https://www.altium.com/documentation/altium-designer/exploring-altium-designer.

Difference Between Active and Passive Components [Online]. - 09 21, 2021. - https://circuitglobe.com/difference-between-active-and-passive-components.html.

DMExplains Making a Circuit in Visio - How To Ep. 34 [Online]. - 03 15, 2016. - 01 03, 2022. - https://www.youtube.com/watch?v=-9tOoYpGBrQ&ab_channel=DMExplains.

ETechnoG Active and Passive Components Definition, Examples, Differences [Online]. - 2021. - October 04, 2021. - https://www.etechnog.com/2019/05/active-passive-components.html.

Haobijam Genemala and Palathinkal Roy Paily Design and Analysis of Spiral Inductors [Book]. - New Delhi : Springer, 2014.

Hardin James [et al.] All-printed Multilayer High Voltage Capacitors with Integrated Processing Feedbac [Journal] // Additive Manufacturing. - 2019. - Vol. 27. - pp. 327 - 333.

Herber Paul Electronics shapes Visio shapes stencils and templates [Online]. - Jul 2021. - 01 03, 2022. - https://www.paulherber.co.uk/visio-electronics/.

Holden Happy Printed Electronics a Technology of the Past and Future [Online]. - altium, April 14, 2020. - April 01, 2022. - https://resources.altium.com/p/printed-electronics-technology-past-and-future.

Hrehorova Erika [et al.] Gravure Printing of Conductive Inks on Glass Substrates for Applications in Printed Electronics [Journal] // Journal of Display Technology. - 2011. - 6 : Vol. 7. - pp. 318 - 324.

Hughes John M. Practical Electronics: Components and Techniques [Book]. - [s.l.] : O'Reilly Media, Inc., 2015.

Keeth Graham , Evans Jon and Peterson Glenn GETTING STARTED IN KICAD [Online]. - KiCad. - March 08, 2022. - https://docs.kicad.org/6.0/en/getting_started_in_kicad/getting_started_in_kicad.html#drc.

Keller Christian Electronic Rule Checking for Schematics [Online]. - Altium, November 02, 2016. - March 05, 2022. - https://resources.altium.com/sites/default/files/uberflip_docs/file_1228.pdf.

Kipphan Heimut Handbook of Print Media: Technologies and Production Methods [Book]. - Berlin : Springer-Verlag, 2001.

Kwon Kye-Si [et al.] Review of digital printing technologies for electronic materials [Journal] // Flex. Print. Electron.. - 2020. - 043003 : Vol. 5.

Liu Yi, Cui Tianhong and Varahramyan Kody All-polymer capacitor fabricated with inkjet printing technique [Journal] // Solid-State Electronics. - 2003. - 9 : Vol. 47. - pp. 1543-1548.

Mafukidze Harry THE TOP FIVE EDA/ECAD PROGRAMS FOR CIRCUIT DESIGN [Online]. - Circuit Basics, 2022. - 02 19, 2022. - https://www.circuitbasics.com/what-is-circuit-design/.

McLyman C. W. T. Transformer and Inductor Design Handbook [Book]. - Boca Raton : CRC Press, 2017.

Menicanin Aleksandar [et al.] Flexible inkjet printed CPW octagonal inductor on PET substrate [Conference] // 2015 38th International Spring Seminar on Electronics Technology (ISSE). - Eger, Hungary : [s.n.], 2015.

Menicanin Aleksandar [et al.] Fully Inkjet Printed CPW Meander Inductors on PET Flexible Substrate [Conference] // 2015 38th International Spring Seminar on Electronics Technology (ISSE). - Eger, Hungary : [s.n.], 2015. - pp. 43-46.

Menicanin Aleksandar B. [et al.] Low-Cost CPW Meander Inductors Utilizing Ink-Jet Printing on Flexible Substrate for High-Frequency Applications [Journal] // IEEE Transactions on Electron Devices. - 2013. - 2 : Vol. 60. - pp. 827 - 832.

Ostfeld Aminy E. [et al.] Screen printed passive components for flexible power electronics [Journal] // Seientific Reports. - 2015. - 1 : Vol. 5.

Pajkanovic Aleksandar, Stojanovic Goran M and Djuric Snezana M. Performance analysis of meander-type inductor in silicon and flexible technology [Journal] // Microelectronics Journal. - 2016. - Vol. 56. - pp. 57-64.

Panreck Benjamin and Hild Manfred Fully Printed Strain Gauges: A Comparison of Aerosoljet-Printing and Micropipette-Dispensing [Journal] // International Scholarly and Scientific Research & Innovation. - 2018. - 9 : Vol. 12. - pp. 678 - 684.

Pohl R. [et al.] Solid-phase laser-induced forward transfer of variable shapes using a liquid-crystal spatial light modulator [Journal] // Applied Physics A. - 2015. - Vol. 120. - pp. 427-434.

Salmerón José F. [et al.] Properties and Printability of Inkjet and Screen-Printed Silver Patterns for RFID Antennas [Journal] // Journal of Electronic Materials. - 2014. - Vol. 43. - pp. 604–617.

Santhiago Murilo [et al.] Flexible and Foldable Fully-Printed Carbon Black Conductive Nanostructures on Paper for High-Performance Electronic, Electrochemical, and Wearable Devices [Journal] // ACS Applied Materials & Interfaces. - 2017. - 28 : Vol. 9. - pp. 24365 - 24372.

Siegel Adam C. [et al.] Foldable Printed Circuit Boards on Paper Substrates [Journal] // Advanced Functional Materials. - 2010. - Vol. 20. - pp. 28 - 35.

Simegnaw Abdella Ahmmed [et al.] Review on the Integration of Microelectronics for E-Textile [Journal] // Materials. - 2021. - Vol. 14.

Sinclair Ian Passive Components for Circuit Design [Book]. - [s.l.] : Newnes, 2000.

softwareradius KiCad Vs Altium Designer - Which One is Best in [2022] [Online]. - software radius, 2022. - March 07, 2022. - https://www.softwareradius.com/kicad-vs-altium-designer/.

softwareradius.com 27+ (free) Best PCB Design Software in 2022 [Online] // software radius. - Software Radius. - February 21, 2022. - https://www.softwareradius.com/best-pcb-design-software/.

Suite KiCad EDA KiCad Libraries [Online]. - 2021. - 09 26, 2021. - https://www.kicad.org/libraries.

Systems Cadence Design Introduction to Hierarchical Schematic Design [Online]. - Cadence Legal Department. - March 05, 2022. - https://resources.pcb.cadence.com/blog/introduction-to-hierarchical-schematic-design.

Tan H. W., Tran T. and Chua C. K. A review of printed passive electronic componentsthrough fully additive manufacturing methods [Journal] // Virtual and Physical Prototyping . - 2016. - Vol. 11(4). - pp. 1-18.

Team The SnapEDA What are ECAD libraries? [Online]. - April 10, 2020. - 02 19, 2022. - https://blog.snapeda.com/2020/04/10/ecad-vs-mcad-vs-eda-everything-you-need-to-know/.

Walker Mark H. Microsoft® Office Visio® 2007 Inside Out [Book]. - [s.l.] : Microsoft Press, 2007.

Wang Binghao and Facchetti Antonio Mechanically Flexible Conductors for Stretchable and Wearable E-Skin and E-Textile Devices [Journal] // Advanced Materials. - 2019. - 28 : Vol. 31.

Wei Yang [et al.] Screen printing of a capacitive cantilever-based motion sensor on fabric using a novel sacrificial layer process for smart fabric applications [Journal] // Measurement Science and Technology. - 2013. - Vol. 4. - p. 075104.

Zhang F. [et al.] Inkjet printing of polyimide insulators for the 3D printing of dielectric materials for microelectronic applications [Journal] // Journal of Applied Polymer Science. - 2016. - 133 : Vol. 18. - p. 43361.

8 Appendices

PRINTED PASSIVE COMPONENTS LIBRARY APPENDIX 1

This appendix includes data for Altium Designer, KiCad, and STEP files of 3D models created in this work.