Reverse, Refurbish and Repurpose PCBs

Noah Harvey

May 2022

A report submitted to the University of Plymouth in partial
fulfilment for the degree of MEng(Hons) Robotics

with sponsorship from Babcock International

Abstract

Engineers within Babcock's Electronic Repair Facility (ERF) reverse engineer Printed Circuit Boards (PCBs) by hand and manually read integrated circuit values to be entered into Electronic Design Automation (EDA) software. PCBs are often damaged during operational use, at which point they are returned to Combined Weapons and Electrical Workshop (CWEW) for maintenance and repair. The information detailing the problems with the PCB's functionality can often be insufficient, making a diagnosis of the issue more difficult. This is time consuming, subject to human error and challenging when the documentation required is unavailable.

This project, comprised of both hardware and software, aims to automate the analysis of circuit boards and supply diagnostic feedback to engineers. This device can inspect the board using machine vision techniques while maintaining a component library.

Circular economy and design lie at the forefront of this project. In essence, it is the practice of designing products with component rotation in mind, rather than disposal, to create more sustainable products with smarter methods of recycling electronic materials. This product allows for not only reverse engineering the PCB, but to provide an accelerated health check of the board and its components to aid refurbishment and the means to repurpose components.

Acknowledgments

The author would like to acknowledge the industrial advisors who have provided technical knowledge and expertise throughout the development of this project. The author would like to thank Samuel Duffield, Stewart Gould and Peter Smith for their mentoring and support. Additionally, the author would like to thank Babcock International for the sponsorship involving both financial and managerial support.

The author would also like to thank Dr Ian Howard for agreeing to be the project supervisor ahead of the academic year, and for providing regular feedback and research-based support.

Finally, the author would like to thank Lainie Smith, Luke Waller and Jack Pendlebury for their continued support and encouragement throughout the year.

Contents

Glossary

API - Application Programming Interfaces

BMS - Business Management Software

CAD - Computer Aided Design

CNC - Computer Numerical Control

CSV - Comma Separated Values

CWEW - Combined Weapons and Electrical Workshop

DC - Direct Current

EDA - Electronic Design Automation

ERF - Electronic Repair Facility

E-Waste - Electronic Waste

FOV - Field of View

FPS - Frames per Second

GPU - Graphics Processing Unit

HDR - High Dynamic Range

HSV - Hue-Saturation-Value

IC - Integrated Circuit

IDE - Integrated Development Environment

LED - Light Emitting Diode

MCU - MicroController Unit

OCR - Optical Character Recognition

OEM - Original Equipment Manufacturer

OOP - Object Orientated Programming

OpenCV - Open Source Computer Vision Library

PCB - Printed Circuit Board

PLA - Polylactic Acid

PNG - Portable Network Graphics

PWM - Pulse Width Modulation

QR - Quick Response

RGB - Red-Green-Blue

SDK - Software Development Kit

STL - Standard Triangle Language

TIFF - Tag Image File Formats

TPU - Thermoplastic Polyurethane

WEEE - Waste Electrical and Electronic Equipment

1 Introduction

We collectively produce 50 million tonnes of electronic waste (e-waste) each year. This number is increasing and has been referred to as the most rapidly growing waste problem of the 21st century. Last year the UK disposed of 2 million tonnes of Waste Electrical and Electronic Equipment (WEEE) [1], with only 17% of this figure being recycled, the rest is sent to landfill or even illegally shipped to other countries. Both scenarios result in toxic substances such as mercury, lead and cadmium being absorbed into the earth.

Adopting a circular economy business model when producing electronics would decrease these adverse effects of the environment. The first principle of a circular economy is eliminating waste and pollution, starting with the design of electronics by manufacturing them with the end-of-life goal of being recycled. There is of course the additional problem of the global silicone supply shortage, a reminder of the hardships we have had to experience when the demand cannot be met. If we used a circular economy, we would have the means to share reused materials, refurbish existing products and repurpose products at the end of their life.

This project focused on the methods used to carry out accelerated circuit board inspections in order to detect defects at a component level. The PCBs were analysed in such a way that they could be classified for one of three outcomes. If the board is deemed as healthy, it could then be reverse engineered for reproduction. If the board had defects, these were highlighted and documented to explore trends. The data collected throughout this project, and the project management tools used have all been collated in this paper to evidence how the design concept was developed into a full fledged product with plans to enter operation in Summer 2022.

2 Research

The research phase involved exploring the methods in which PCBs are classified in terms of their appropriate waste stream. The alternative to sending e-waste to landfill is to dispose of them more efficiently, by sending them to a recycling centre. Recycling electronic waste involves various manual and mechanical procedures, making recycling PCBs containing hundreds of components rather difficult. Currently, the boards are crushed and separated into metals, equating to 30% of their mass, and non-metals being the remaining 70%.

The methods used to recover valuable materials from electronic components involve open-air burning, which releases fine particles of toxic materials into the atmosphere, and acid baths. There are numerous contamination risks related to the incineration process, resulting in the land, water and air harbouring chemicals detrimental to human health. On the other hand, if the e-waste is not dissembled, it is simply shipped off to landfills in other countries for them to face the consequences. These approaches will not keep up with the rapid production of new technologies, with the latest innovations becoming obsolete within months, we need to explore more viable solutions.

The project outcomes were heavily reliant on three main capabilities of the program working in unison. These involved a mechanical frame capable of obtaining images of PCBs in multiple planes, the ability to process the input images to obtain enhanced outputs and a way of creating a database capable of identifying trends in board appearance. The hardware used to create the frame was selected through the down-selection and optioneering process detailed in section nine. Each component was chosen through researching its capabilities and comparing them against not only the requirements, but other competitors in the same field. This was a lengthy process, requiring a great amount of detail and due diligence, but the outcome resulted in the best possible solution.

3 Inspiration

The inspiration for this project came from Apple's Project Liam [2], which defined the start of their journey towards becoming more environmentally friendly and outlines their goal of becoming carbon neutral in 2050. Liam was designed to discover an alternative solution to crushing and shredding components to obtain materials, as the volume and quality was greatly decreased during the process. The initial capabilities of the robot included disassembling complete and partial phones, tackling the complications that came about through innovating components to be smaller, more durable and therefore harder to isolate.

Apple understood that the recovery of materials incorporated into their electronic devices was determined by the initial design and have since kept ease of disassembly at the forefront of their designs. This allows for material isolation, purer metal extraction and a close loop system in which they can repurpose the materials acquired and incorporate them into their next design [3]. Apple now has multiple disassembly robots in operation, all having learnt from their predecessor with the aid of university interns who specialise in computer vision systems with the means to model future recycling processes.

Another inspiration was Retronix [4], a Scottish company founded in 1992, which now operates around the world. They provide companies with the means to adopt a circular economy by refurbishing and re-using their components. The services they have been offering for over 30 years include reclaiming components from scrap PCBs, re-conditioning them for re-purposing and various forms of reballing to recover high value materials.

4 Project Management

While discussing the possibility of a sponsored project with Babcock they requested a concept design, bill of materials and an outline of the design requirements. All of which had to satisfy CWEW engineers' needs as well those suggested by the University of Plymouth's project advisor in accordance with the marking rubric. Multiple meetings were held to identify what parts of the Technical Team's wishes were desirable goals that the project supervisor could justify as appropriate for a Bachelor's project before the start of third year. It was agreed that the project's progress would be reported to Babcock bi-weekly, and the university supervisor would run weekly meetings to discuss deliverables. The meetings entailed a briefing of the previous week, a discussion on the tasks completed and feedback, followed by an overview of the next steps. The weekly calls defined the structure for the week, ensuring that clear goals were outlined, and that the tasks were achievable within the time frame.

- Phase 1 focused on the the concept design, equipment bill of materials and component optioneering.
- Phase 2 focused on the hardware, assembling the frame, mounting the cameras, and setting up the Gantt chart to create a timeline.
- Phase 3 focused on the software set up and machine vision application. This is where the project semester began.

Throughout all three phases the progress was documented in the logbook []. Each week the tasks were recorded, and the work carried out was reported in detail for supervisors to access. Both industrial and academic supervisors had access to the OneDrive folder, where photographs, videos, and other tools such as a Gantt chart was stored. The Gantt chart details the development of the project in stages which relate to the deliverables defined in the design requirements, along with research times, totalling to 10 months. The Kanban board broke down the stages into more detailed, smaller tasks which were more manageable in a shorter time range.

Management Tools

- OneDrive, sharing documents
- GitHub, version control and Kanban board
- Microsoft OneNote, recording weekly progress
- Microsoft Excel, Gantt chart
- Microsoft Outlook, email, and calendar reminders
- Zoom, video calls

The Project Supervisor was able to have meetings with Babcock in attendance during Phase 1 and the continuous communication between all parties, has been extremely effective. Frequent meetings with Babcock have been greatly beneficial to all parties, an agile approach in which the customer is kept in the loop is vital in such a project and has been heavily encouraged in Babcock's approach to developmental projects such as this one.

The Agile methodology used broke the project down into 2 week-long sprints which focused on the deliverables of the current phase. This approach was applicable to both hardware and software, with evidence of progression in programming being difficult to present this method allowed for the theory behind smaller tasks to be discussed to show an understanding of the next step. The meetings were used to discuss the functionality of the code as it was being written, and as the project advanced it was clear to see how methods had developed through multiple iterations to finalise the product.

From an industrial point of view, operating in planned sprints allowed for incremental development in which the most important features of the project were outlined as desirable goals for the fortnight. The outcomes achieved were then presented to stakeholders to assure them of progress and clarify that their requirements were being fulfilled. This method simplified tasks, elevated motivation levels, and provided a regular meeting in which problems could be identified. For example, hardware delivery times would vary in length but when operating in short sprints the next steps were also highlighted. This meant that hardware and equipment was ordered with plenty of time in advance to prevent a delay in the project's progress.

5 Phase 1

5.1 Concept Outline

CWEW expressed their need for a faster way to run health checks on their PCBs and accelerate the process of reverse engineering the boards. The Babcock Business Management Software (BMS) Agility framework required a cost benefit analysis, detailed design requirements, a Concept Design Definition Document and an overview of the downselection and optioneering process.

5.2 Cost Benefit Analysis

The design requirements were investigated with the Technical Team, then presented to the Engineering Lead who reached out to stakeholders who approved funding upon the mutual agreement that the project would successfully fulfil their needs.

5.3 Concept Design Definition Document

With the budget outlined, the various design options could be explored. The complexities of the project were noted in ways that all stakeholders could understand, this involved generalisations such as using the terms microcontroller, motor driver and so forth instead of actual component names although they had been explored during downselection. The development team involved the lead engineer providing the budget and those who were playing the role of industrial advisors reviewed the concept designs. Upon agreement of the favourable design, it was then accepted for further development.

5.4 Down-selection and Optioneering

This stage involved reviewing potential options, such as software and components, to find those most suited for the application given the criteria for the task. The initial list of components identified as suitable for the task was scrutinized. The product specifications, performance and other relevant factors were reviewed in order to downselect the best fit. This process helped to identify the best components for the project specifically, which ensured that performance requirements were met.

6 Design Requirements

6.1 Physical Design Requirements

- Capable of holding 30cm by 40cm PCBs
- Bespoke bracket capable of holding both an Intel RealSense and 4K webcam
- Fully adjustable width, height, and camera placement – with capability to automate
- Modular design that allows for adjustment of PCB height
- The circuit board must be manipulated to obtain images of both sides
- Bespoke PCB for stepper motor control and external peripherals

6.2 Software Design Requirements

- Identification of identical boards and their variants
- Storage of text obtain from optical character recognition
- Generation of circuit board database
- Produce a depth map of the PCB and the bespoke mount
- Provide real time distance data to aid in camera calibration

These ideal outcomes were presented to the Technical Team, along with an in-depth discussion about the time commitment it would require. A budget of £2,000 was agreed in exchange for permission to share a case study with them that they could publish. Throughout the project, the Technical Team discussed additional features and alterations that they desired. These were influenced by the direction in which the project was going, news and articles on advancing technologies and innovative ideas they wished to explore.

It was made aware to all from the start of the project that the testing could not be carried out on PCBs from within CWEW. Without appropriate clearance and security measures required, this project would not see the ERF Team until June 2022. However, other PCBs from various members of the team were given in abundance, and additional parts were purchased to further the testing. This did not impact the performance of the project as the design requirements accounted for the situation at hand, with aims and objectives that could build a strong platform to begin board analysis with.

7 Aims and Objectives

Below is a detailed list of objectives that, with completion, indicate that the project has been successful. These were designed with the University's marking system in mind and have been broken down to illustrate the grade boundaries.

1. Construction of din-rail frame of agreed dimensions with Babcock's ERF Engineers to accept majority of their circuit boards.
2. One or more Intel RealSense cameras supported by frame to sit above the target circuit board.
3. Capable of capturing clear 4K pictures as a solid input for processing.
4. Capable of reading any pre-printed Quick Response (QR) codes previously fixated by an Original Equipment Manufacturer (OEM).
5. collection of servo motors to orientate the circuit board in 1 plane (spin) to produce a clear image to the cameras.
6. Use of Optical Character Recognition (OCR) techniques to locate and store the integrated circuit components identified into a human readable format.
7. Distance from board to camera computed to aid with calibration.
8. Design for a user interface, capable of orientating the board as the user pleases.
9. Depth map of the PCB produced by the RealSense camera.
10. Use of machine learning techniques to compare the size and shape of the circuit board with previously seen circuit boards, to identify whether the solution has seen variants of the same board previously and identify trends in board condition.

As an overall outcome, this project includes a bench-top device that manipulate PCBs under a 4K camera to obtain the best possible image. The software can provide detailed information of the configuration of board, read integrated circuit values, and create a components list in a library dedicated to that board.

8 Software Optioneering

8.1 Integrated Development Environment

The software element of this project was developed using Visual Studio rather than Visual Studio Code or STM32 CubeIDE. Visual Studio contains features similar to the other Integrated Development Environments (IDEs), such as a debugger for source and machine code, autocompletion and the option to incorporate additional tools or extensions into the platform. For this project, the decision was defined by compatibility with the hardware, along with the possibility to integrate OpenCV and OCR software.

The Intel RealSense depth camera required the Software Development Kit (SDK). It contained the software, Application Programming Interfaces (API) tools and language wrappers required to obtain data from the depth camera. Visual Studio accommodated this SDK very well, where as the STMCube IDE could not at all.

The decision to use Visual Studio was also made with Babcock's technical team in mind. The team are adept in the use of Visual Studio as their industry standard, with their business package including the full version. The free version used in this project provided the necessary tools for the software development, and the opportunity to explore the full capability of Visual Studio remained open.

8.2 Optical Character Recognition Tools

OCR analyses the text from an image and produces an output of machine-readable text, such as a digitized document. Research uncovered two OCR tools that met the first requirement of being open-source. The decision to use Tesseract [5] was finalised when Calamari required dependencies which were difficult to integrate into the project, and required another tool to carry out the actual text processing.

Tesseract was well documented, included clear download instructions and possessed the capability to represent the text in multiple ways. The recent update for Tesseract had increased its accuracy to 92.9%, reinforcing the decision further.

8.3 Further Software Dependencies

The latest release of the Open Source Computer Vision Library (OpenCV 4.5.5) was used [6] with the C++ interface to develop the computer vision application for image processing. OpenCV provides access to over 2,500 algorithms enabling users to deploy various machine learning and computer vision capabilities into their work.

The RealSense SDK-2 [7], was used to support the D455 depth sensing camera. The SDK was capable of supporting a variety of programming languages and development platforms, while providing tools to obtain depth streams, visualize point clouds and camera configuration. The calibration tools were a necessity for the D455 as they were required to maintain optimal performance.

Leptonica [8], an open source library consisting of software commonly used for image processing and a requirement for OCR through the use of Tesseract. Leptonica was used for opening input images of various formats, such as Portable Network Graphics (PNGs) or Tag Image File Formats (TIFFs). TIFFS were explored as an option for the project as they are used to describe, store and interchange raster image data. The format is often used by photographers as they provide a convenient way to store high-quality images.

9 Hardware Optioneering

9.1 Cameras

Two cameras were required for this project, one to carry out accurate depth analysis and another to capture high resolution images. It became apparent relatively early in the project that depth perception cameras did not meet the requirements when it came to red-green-blue (RGB) image resolution, and so the prospect of an additional camera was explored. The process of optioneering the cameras was lengthy, the specifications were deeply scrutinised and the budget allocated to these products was by far the largest. For the individual requirements, two cameras capable of different tasks were put forward for downselection.

The resolution of the camera was the most important specification. The first round of research for selecting the most suitable camera for capturing images of the PCBs involved short listing those with a resolution of 4K. The compatibility came next, it had to be interfaced with a laptop operating on Windows 10 but also compatible with other operating systems, such as Linux. The size and shape was important as it had to be mounted on a bracket so full size cameras and obscure shapes were omitted from the selection.

The Mokose 4K USB camera was a strong contender at £159, designed for livestreaming and compatible with Windows, macOS and Linux. It offered the ability to connect multiple cameras to the same device and fine control over the images it captured. The shape was rather cubic with long lenses but the field of view range was between 30 and 60 degrees. It did offer white balance, but stated that lots of manual adjustment was required to obtain perfect pictures. This project could not incorporate fine tuning of the camera into a routine where multiple pictures had to be captured efficiently, and so this camera was removed from the downselection list.

The Logitech Brio Stream webcam priced at £219 operated on Windows, macOS and Chrome OS. The pricing was initially out of budget but regular offers were applied and at the time of research it was £160. The field of view ranged from 60 to 90 degrees, the design incorporated the common tripod mount screw hole and it was highly reviewed for how well it handled different lighting situations. The High Dynamic Range (HDR) footage obtained contained much more detail, with the ability to balance out shadows and account for new environmental effects, it was selected for capturing pictures of PCBs.

Figure 1: Logitech Brio Stream

Deciding on a depth sensing camera was even more complex. Set-up, power and processing had to be considered. The top three sensors fell into a similar price range but had varying performance and two had very little information published. However, the details obtained and specifications found were sufficient enough to inform the decision.

Firstly, the Stereolabs Zed 2i camera priced at £400 stood out due to its main features being neural depth sensing and spatial object detection. It was capable of using deep learning methods to estimate per-pixel depth and the uncertainty from a continuous video stream. However, it required additional drivers and software which further increased the cost, as long as the requirement of a powerful Graphics Processing Unit (GPU). The disadvantages began to outweigh the strengths when further reading uncovered that there was no open-source SDK available, and this camera was deemed inappropriate for the task.

Microsoft Azure Kinect DK was the runner up depth camera researched, with similar system requirements and the ability to operate on both Windows and Linux through a USB 3.0 port. It was placed at a price of £355 with options for a wide or narrow field of view, with similar system requirements to the Zed 2i but without the need for a GPU. The Kinect DK was capable of running all of the processing within the device itself. The 1MP depth sensor stream could be paired with the 12MP RGB video to produce a fully aligned video, excellent for the application within the project. The shape of the device was interesting, with a notably large depth that was difficult to mount. The point cloud examples included in their demonstrations were of high accuracy, except they could not overcome shiny surfaces. At a resolution of 512x512 at 30FPS, this sensor operated at a lower performance level than the Zed 2i and D455.

Figure 2: Intel's RealSense D455 Depth Camera

Intel's highly reputable D455 was being sold for £314 with their second free cross-platform SDK that had been integrated with Open3D, an open-source library designed for processing 3D data and supporting 3D machine learning. The RGB camera can obtain the best resolution out of the three that were researched, obtaining 1280x720 at 90FPS. The D455 did not require an external power supply, it was capable of on chip depth processing with a USB 3.0 interface to display the data. The overall design was compact, however this did come at the cost of having larger amounts of noise in open spaces. In relation to the task at hand, in which a small area underneath the frame was the region of interest, the D455 was an excellent choice and so it was purchased for this project.

9.2 Development Board

The decision regarding which IDE to use for programming external peripherals was influenced by the development board used. Due to product shortages, size limits of the controller pad design and the need to begin manipulating boards under the camera earlier on in the project, the board had to be readily available.

An optioneering table was created by stating how essential the parameter was in the class weight column. A score of zero meant it was not required at all, whereas 10 meant that it was of vital importance to the project. Each board was then allocated a weighting score in relation to how well the specification met the requirements. Totalling up this score produced an overall rating regarding the boards capabilities, cost and how well it met the criteria of the project.

Parametric	Class Weight	STM32 F303K8		STM32 L432KC		Arduino ATmega328P	
		Weight	Score	Weight	Score	Weight	Score
Cost	3	6	18	7	21	5	15
Ease of Use	5	10	50	10	50	8	40
Number of interfaces	8	7	56	8	64	7	56
IDE Cost	4	10	40	10	40	10	40
Processor	8	6	48	8	64	3	24
Flexible Power Supply	5	8	40	8	40	6	30
Low Power	2	2	4	8	16	2	4
Maximum Clock Speed	2	4	8	7	14	4	8
Number of PWM pins	5	6	30	8	40	4	20
Maximum Data Size	3	4	12	7	21	4	12
Maximum Program Size	3	4	12	8	24	4	12
Total			**318**		**394**		**261**

Figure 3: Development Board Optioneering

The development boards were selected based on their size, then other factors such as the number of pins, particularly those capable of Pulse Width Modulation (PWM). These pins were required to drive stepper motors, LEDs and other external peripherals. It was made evident that the Arduino Nano (ATMega328P) was incapable of supporting these peripherals. It was also over double the price of the other developments boards, and although the range of open-source libraries available was vast, it would have failed to meet other requirements entirely.

The STM development boards both scored very well, but the L432KC had a better specification and was more qualified in terms of satisfying the criteria. The FlexPowerControl meant that it could remain in an ultra low power state when not in operation, with a 2nA shutdown mode, and it was capable of being powered from the 5V output that stepper motor drivers considered for the project commonly included.

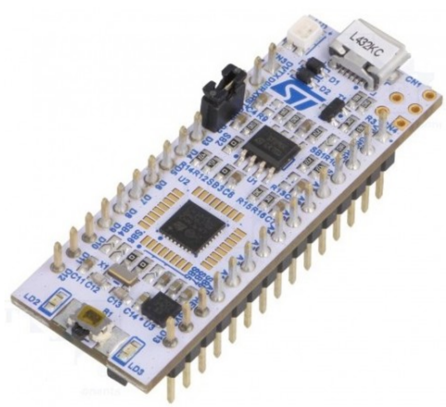

Figure 4: Nucleo L432KC Development Board

9.3 Motors

A stepper motor was used to manipulate the board due to their remarkable accuracy and control, especially their stopping accuracy tolerance of +/-0.5 degrees unloaded. The motor was not required to operate continuously, but the precise position and holding torque of the motor selected was of high importance.

Brushed motors were a cheaper option with high efficiency and the ability to easily control fast moments. The application of these motors however, was not suitable for the job. The brushless DC motors explored were both efficient and durable due to less internal friction than brushed motors but they faced the same criticism of poor performance in the intended application of holding a mechanism in one position for extended periods of time. For these reasons, stepper motors were explored in more detail.

The two main types of stepper motor were those with permanent magnets and those with variable reluctance. Permanent magnet stepper motors were capable of producing more torque per unit of power, whilst already requiring less power to run in the first place. Variable reluctance stepper motors housed gears with teeth that produced a higher degree of angular resolution. Both of these properties were deeply important to the functionality of the project and so further research was required when optioneering a stepper motor to manipulate the PCBs.

NEMA 17 Model	Price	Step - Angle	Weight	Rated Current	Holding Torque
1701HS140A	£9.90	1.8	0.15kg	1.40A	1.5kg.cm
1702HS133A	£10.75	1.8	0.22kg	1.33A	3.2kg.cm
1703HS168A	£11.56	0.9	0.28kg	1.68A	4.4kg.cm
1704HSM168A	£15.55	0.9	0.35kg	1.68A	5.5kg.cm
1705HS200A	£15.95	0.9	0.55kg	2.00A	8.2kg.cm

Figure 5: Stepper Motor Optioneering

The NEMA 17 stepper motor was widely available from multiple suppliers, was applicable to functions similar to the way in which the PCB was going to be manipulated in the project and the various specifications made downselection easier. The calculations, evidenced in Week 14 of the project logbook heavily influenced this decision, and so a table of characteristics was made to compare models.

Some models did not meet the necessary requirements, and so they were omitted, leaving two remaining models. The 1703HS168A model was selected for its higher step angle accuracy, lower price and ability to hold and rotate the weight required.

Figure 6: NEMA 17 Stepper Motor

9.4 Motor Drivers

The stepper motor drivers were required to transform a pulse signal from the development board into an angular displacement signal. This was therefore necessary to drive the NEMA-17, which needed a driver that had a maximum current rating higher than 1.68A. This component needed to be of high quality, and the budget was increased to take this into account.

Firstly, the SparkFun EasyDriver, priced at £18.74, was investigated as an appropriate option. It was capable of providing any voltage level required from 6V to 30V, with an onboard regulator capable of outputting 5V or 3.3V. The motors attached could use 4, 6 or 8 wires, the driver itself could cater to a variety of steppers but only power one at a time. It incorporated sleep, enable and micro-stepping pins which suitably met the criteria outlined in the hardware requirements. There were alternative versions, such as the Big EasyDriver, capable of driving multiple boards for a price of £30 but this was not an essential feature and so for the price this option was eliminated.

The RAPS128 stepper motor driver housed a large heatsink and full capability with 32-bit microcontrollers such as the L432KC. It was capable of taking 128 microsteps per full revolution, and had a higher maximum current output of 2.2A but did however cost £18.45. The instruction manual provided detailed information on how to obtain the highest number of microsteps to maximise accuracy, as well as how to interface the drivers with microcontrollers for optimal control. This was a clear winner, with a lower price in relation to other options, and specifications outweighing the EasyDriver.

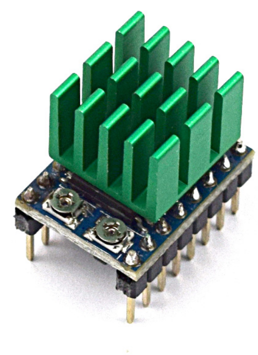

Figure 7: RAPS128 Stepper Motor Driver

In an effort to progress the project while waiting for the intended driver to arrive, the L298N H-Bridge was used. It had a peak current per phase of 2A, was capable of taking 12V but outputting a supply of 5V to the development board. This was vital to creating a control panel that could encase all the necessary hardware to control the external peripherals. Through testing it was evident that holding PCBs in a certain position increased the temperature of the driver noticeably, with the intention being to position larger boards within the PCB mount, drivers with large heatsinks were explored.

10 Frame Materials

The frame had to have a height of 0.8m, with a base of 0.6mx0.6m. This meant that the materials used could not falter or bend while holding a load at a significant height, and that the base would have to be heavy enough to remain grounded while other components moved. The PCB holder had to be capable of extending up to 0.4m at a minimum, with the additional option to increase the height at which the PCB was held.

10.1 Frame Skeleton

The skeleton of the frame needed to comprise of materials that allowed for easy adjustment but also allow for certain sections to be permanently fixed. Aluminium extrusion was affordable, lightweight but still strong. The slots offered two main uses, one to facilitate bearings to allow for smoother movement of rails and another to lock in t-nuts for the more permanent fixtures.

10.2 Joints and Miscellaneous

A combination of tools and materials were used to form joints within the frame. PLA was used to print bespoke parts for varying fits. For example, corners with had to be fixed in place used a tight fighting 3D printed model with a t-nut to lock it in place. The camera mount was 3D printed with space to decrease the friction and have the bracket mounted on a bearing that would slide horizontally with ease.

Jigs were used to aid with cutting the extrusion into sections, these were 3D printed too. The jigs were then reused to create tight sealing security bars to lock multiple sections of the frame in place. This allowed the user to lift the entire frame, increased its portability and increased the strength when mounted to a worktop.

10.3 PCB Mount

The PCB mounting device consisted of aluminium extrusion, 3D printed components to adjust height and grip, the stepper motor selected and bearings for smooth rotation. A modular design was utilised to enable easy stacking of blocked to accommodate to PCBs with larger depth measurements. Alternatives to using PLA for the PCB mount would not have performed as well, and the process of assembly would have been less time efficient.

11 Sustainability

11.1 Hardware

Components bought as bulk orders were purchased with further development in mind and so all parts will be put to good use. For example, multiple motor drivers were bought as a package deal so they could be used later on in the project.

The microcontroller was selected with future enhancements to the project in mind. This method of planning has prevented waste due to components no longer meeting the requirements outlined. Although the board was small, it had enough pins to operate two stepper motors, four buttons, three LEDs and a buzzer with six spare pins remaining.

11.2 Materials

The PLA used to 3D print was purchased from a supplier that uses cardboard reels to cut down on plastic waste. The waste PLA produced from testing, misjudged tolerances and general iteration of design changes was recycled. This was done through handovers to a company capable of melting the filament back down and extruding the material back onto a reel to be used again. TPU was considered for printing the grips but research uncovered that PLA is more environmentally friendly.

11.3 Waste

The unused components of this project have been kept due to their general functionality. These parts include bolts, nuts and other miscellaneous. The waste extrusion was used to create the PCB mount as it allowed for adjustable widths of the boards by gliding one end across.

12 Hardware Interface Concept Design

The Hardware Interface Diagram below illustrates how the hardware was connected. The optional step involved connecting the microcontroller to the laptop or dock in order to upload the program to it. It was also used to output serial messages via USB to debug and initialise codes during development. However, for general use, once the code had been uploaded to the board it only required power from the stepper motor driver to operate external peripherals.

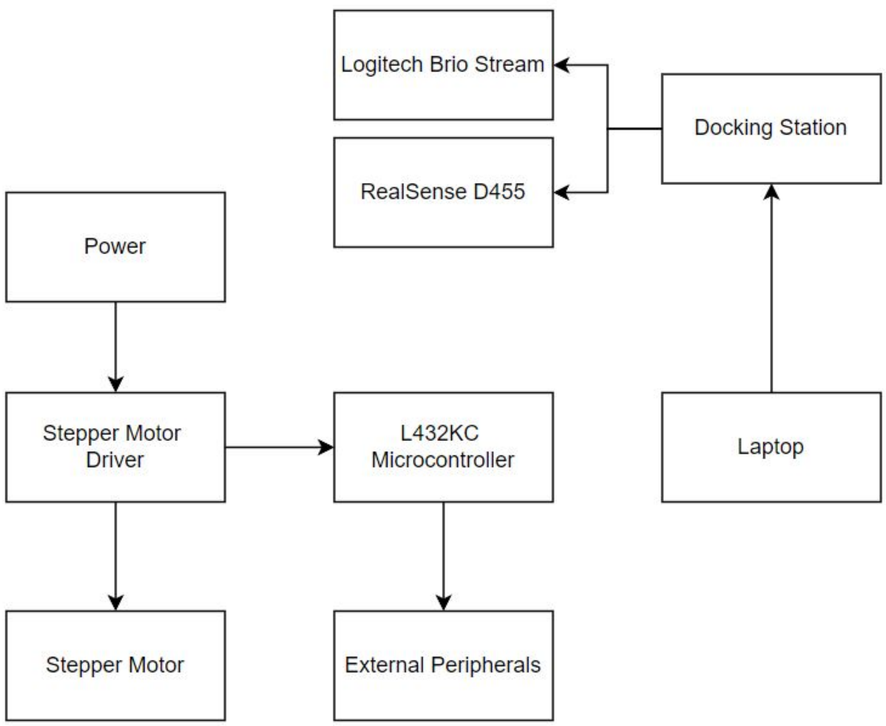

Figure 8: Hardware Interface Diagram

12.1 Microcontroller and External Peripherals

It was decided that Mbed would be used to develop the program for controlling the stepper motor and external peripherals. The engineers required control of the board via physical means at the hardware end, and expressed a want for controlling the PCB orientation from their seat at the laptop. The approach taken meant that both options could be used, as the docking interface would be close to the frame and allowed for USB Serial communication. A bespoke PCB was designed to mount the microcontroller along with buttons, LEDs and a warning buzzer.

12.2 Camera Interface

The cameras could not be supported by the average USB hub without decreasing the resolution of the pictures and so the HP USB C Dock G5 was used. It kept the desktop clean and organised, while expanding possibilities for additional monitors to display more information and supply power to the laptop at the same time.

13 Frame Design

13.1 CAD Software

In total, 22 parts were designed using Autodesk Fusion 360 in order to assemble the multiple frames and various components. The design itself had multiple iterations which incorporated the upgrades required to overcome the disadvantages of the previous version. Fusion kept version control of all the parts, enabled rapid prototyping and was capable of providing stress tests and other validation tools at the design stage. It could also export the parts as Standard Triangle Language (STL) files for 3D printing.

Once the STL files were obtained, Ultimaker Cura was used. Cura had a high compatibility with a range of 3D printers, translating the STL files into a format that the Creality Ender 3 printer could utilize. The software was easy to use and provided more advanced settings to alter the way in which the print would be created. For example, the smaller parts were printing with a 0.4mm nozzle and fine, intricate settings to produce prints of a high quality which were still strong. The larger prints were created with a 0.8mm nozzle, which halved the printing time and allowed for more rapid prototyping.

The main advantage of 3D printing was the ability to produce parts when required, reducing the production time. The project was developed with constant access to a printer and so strong and lightweight parts could be manufactured rapidly. 3D printing prevented creating waste material, where alternative methods such as Computer Numerical Control (CNC) machining are referred to as subtraction manufacturing. Subtraction manufacturing refers to a block of material requiring sections to be cut away to create the part required. This process produces dimensionally accurate components from an array of materials such as metal, wood and plastics. In an effort to keep production waste to a minimum, and with the tolerances produced by a 0.4mm nozzle on a 3D printer being sufficient, CNC machining and other highly wasteful methods such as laser cutting were omitted as inappropriate manufacturing methods.

13.2 Frame

The frame consisted of five parts of aluminium extrusion and 14 3D printed components. The jigs designed to bolt onto the extrusion to aid the cutting process were used to lock various sections of the bars in place. This enabled the frame to be lifted and transported by holding the two vertical bars.

Figure 9: Frame and Additional Parts

1. Corner stands
2. End covers
3. Horizontal bar supports
4. Base joint supports
5. Jigs to cut aluminium extrusion

The way in which the bar supports were designed accounted for a tight fitting tolerance that allowed movement upon a certain degree of force. Locking tee nuts were used to permanent fixtures, such as the base. The end covers utilized a tight tolerance but were not permanently fixed. This was to allow for general adjustments and interchangeability between various parts.

The testing of the frame involved software simulations, infill strength tests and general robustness of the frame while being transported. At the start of the project during Phase 1, there was no permanent workspace dedicated to the project, and so the frame was regularly assembled and disassembled at the end of the day. This had no impact on the frame integrity as the stress tests ran during the design process highlighted weak spots. These were overcome by adding fillets to sharper corners and increasing the infill percentage.

To move the horizontal bar down, with additional support clamps attached, two arms would have to exert force down onto the bar to move it the slightest amount. The weight in relation to this force would equate to around 10kg, the aluminium bar plus additional future prints remained under this threshold. A metre of 2040 aluminium extrusion weighed 1kg, the length of the horizontal bar was 0.6m and therefore weighed 600g. The 3D printed attachments were estimated to weigh 180g at 30% infill. The rest of the weight came from the camera mount, detailed below.

13.3 Camera Mount

The optioneering of the cameras ensured that their weight and size remained smaller to increase the ease of which they could be mounted. The Logitech camera weighed 64g and the RealSense camera weighed 115g.

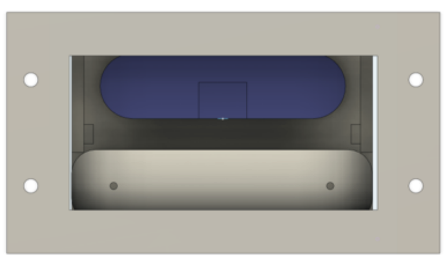

Figure 10: Camera Bracket - Cameras Fitted

1. Camera face plate
2. Camera housing
3. Horizontal sliding mount
4. Camera alignment clips
5. Bearing supports

The completed assembly of the camera mounting bracket plus additional nuts and bolts weighed 240g, with the cameras inside the total mass came to 419g. This measurement was well below the acceptable limits for what the horizontal bar could hold.

The design process for this component was relatively straightforward, exact dimensions of the cameras were obtained and space for USB Type-C cables within was accounted for. Additional holes were implemented into the design to lock the face plate in securely. These additional screws strengthened the joints and the camera alignment clips were used to hold the cameras in their aligned position while the bracket was moved along the bar.

The functionality of this design was tested throughout day to day use as sections of the frame were adjusted. As first a soft surface was placed beneath the camera mount but the robustness proved reliable given the extreme movements the bracket underwent to prove the concept.

13.4 PCB Mount V1

The PCB mount design consisted of two separate assemblies capable of supporting the circuit boards from either end. The left assembly incorporates a fixed stand to mount the stepper motor in place. The right side had a fit with more slack to allow for adjusting to fit the width of the board. It did not require a stepper motor and so the right hand gripper rotated freely on a screw.

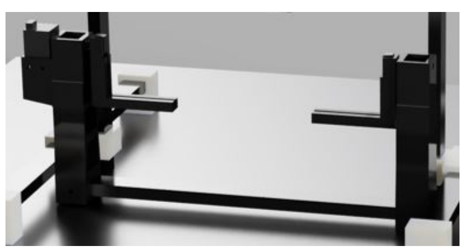

Figure 11: PCB Mount - Version 1

The mount testing involved exploring tolerances for the board grippers to ensure a tight fit to to lock the board in place while rotating. These grippers provided additional support along with width of the boards, which was advantageous when mounting boards with a larger mass.

13.5 Complete Frame Version 1

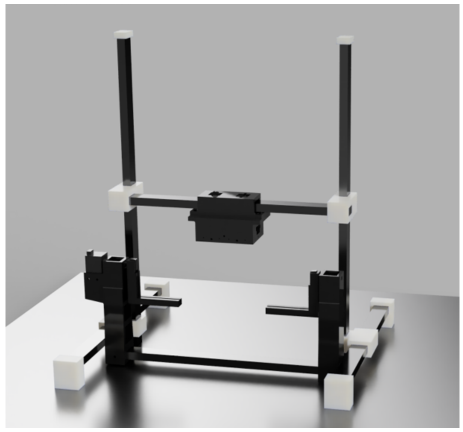

Figure 12: Complete Frame with Mounts - Version 1

Version 1 of the frame depicted above had separate assemblies, the frame and PCB mount remained unattached during the testing period. This was favourable when working on individual sections, for example when testing the stepper motor and external peripheral code the PCB mount could be easily separated from the entire project.

13.6 Complete Frame Version 2

Several improvements were made to the PCB mounting assembly, all of which arose from every day use that provided an insight into how the project's hardware aspects would perform in the workplace. It became evident that more grippers were required to suit a variety of boards, and so a range of sizes were developed with the ability to attach them made simpler. The free moving stand design was upgraded to incorporate bearings into the gripper mount to reduce the friction. In-lays were introduced into the joint compartments to hold nuts in a more secure manner, greatly increasing the structural integrity of the entire assembly.

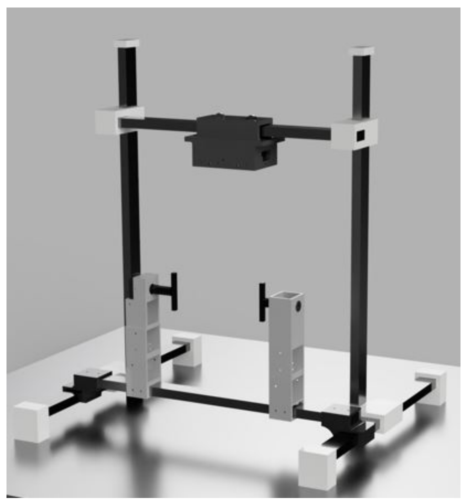

Figure 13: Complete Frame with Mounts - Version 2

14 Phase 2

14.1 PCB Manipulation

14.1.1 Stepper Class

A class dedicated to the control of the stepper motor was written with project specific methods. The methods included allowed for stepper motor initialisation, manipulation of the boards orientation and safety stops. The 180 degree rotation method enabled the user to rotate the board to capture images of the front and back faces. The smaller adjustment method rotated the board in five degree increments and the hold method would set the enable pins high, keeping the stepper motor activated and holding the board in the current position until other instructions were received.

The emergency stop method was linked to an interrupt enabled button that upon pressing cut off power to the steppers and set the enable pins low. The system would only receive power to resume operation after a hard reset, which was made easier by assigning a physical button to restart the microcontroller.

14.1.2 Hardware Class (External Peripherals)

The hardware class initialised four buttons, three LEDs and a buzzer. Three settings were developed, each scenario being displayed by a traffic light system. The first was to show full system initialisation upon power being supplied to the motor, which was indicated by an amber light. Another to show that the board was being rotated, with a red warning light and a buzzer sounding. The final was to display a green light while the board was being held in place.

14.1.3 Breadboard Testing

Testing was carried out on a breadboard to ensure that the pins assigned to controlling both stepper motors and external peripherals were fully functional. The microcontroller pin assignment was adjusted to optimise the capabilities of the pins individually to ensure correct operation and set up.

Through the use of the breadboard circuit, the stepper motor could be controlled to manipulate the PCBs under the camera to aid with calibration in the second phase of the project. It also allowed for testing of the microcontrollers other methods, such as ensuring the circuitry had been done correctly. This was of high importance as methods such as the emergency stop had to be reliable, and the reset button had to be wired correctly as the microcontroller reset button would not be accessible once assembled within a casing.

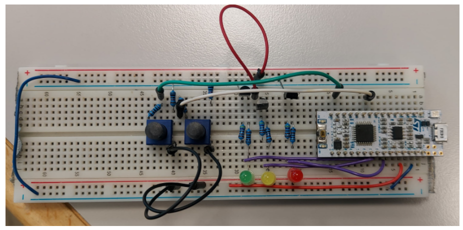

Figure 14: Microcontroller Program - Breadboard Testing

14.1.4 PCB Design

The final schematic produced was optimised to produce as much functionality from the board as possible, with spare pins made easily accessible. The various improvements included driving the two enable pins required by the stepper motor through the same pin and connecting the operational LED and buzzer.

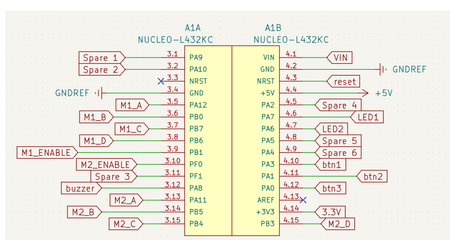

Figure 15: L432KC Pinout

Screw terminals were added to accommodate for use of the spare pins, as well as enabling simpler means of powering the board from the motor driver to utilize the 5V voltage pin by shorting the jumper pin. This greatly reduced the amount of wiring and allowed for a case to be designed that protected all of the circuitry.

14.1.5 PCB Assembly and Testing

The bespoke PCB for control of the stepper motor and external peripherals was designed using KiCad EDA. The software was free, simple to use and capable of capturing schematics and designing PCBs. The 3D viewing feature was used to obtain accurate dimensions for creating a case. KiCad's footprint libraries included all of the components required except the buzzer. This was a straight forward footprint to create due to the simplicity of the component. The full schematic and PCB design file are located in the OneDrive folder listed in the appendix.

14.1.6 PCB Control Board

The system control panel below was created to provide a user friendly interface with the PCB mount and stepper control. It was also created to display the traffic light system and provide more control over the stepper motors to increase the capability of the overall system.

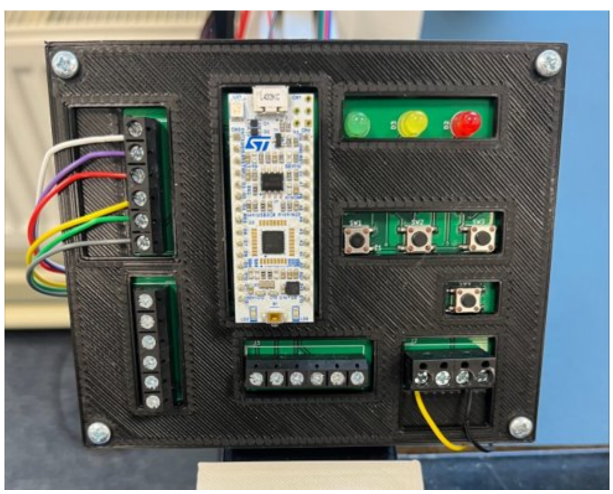

Figure 16: System Control Panel

14.2 Software

14.2.1 Camera Calibration

The RealSense D455 had multiple tools available for on-chip calibration, such as OEM for targets and dynamic to restore back to factory performance. The calibration routines would restore depth performance, increase the accuracy and correct conditions which created degradation in performance overtime. The depth precision was tested regularly to ensure optimum performance. The standard test routine involved pointing the camera at the flat table top surface where a small region of interest was selected, an increased amount of noise would be the main identifier of degradation in precision occurring. If the quality of the depth map had reduced, the self-calibration routine was completed and the corrections were saved to the depth sensor.

The Brio Stream camera routine required member functions from the *VideoCaptureProperties* class [9] that the *OpenCV* created. The use of these functions is discussed in more detail in the Capture Class section in Phase 3.

14.2.2 Software Flowchart

A flowchart was created as a visual aid to understand the flow of the data within the project. Before the images of circuit boards could be saved to the library, they had to be processed to ensure that optical character recognition would work. The flowchart also highlighted the main decisions that had to be made within the program. In order to save variants of boards and identify new ones that had not been seen by the system before, a likeness calculation routine was developed. The percentage of likeness for two boards is compared against a threshold, meaning that if the boards were very similar or had identical components, they could be variants of the same board and were treated as such.

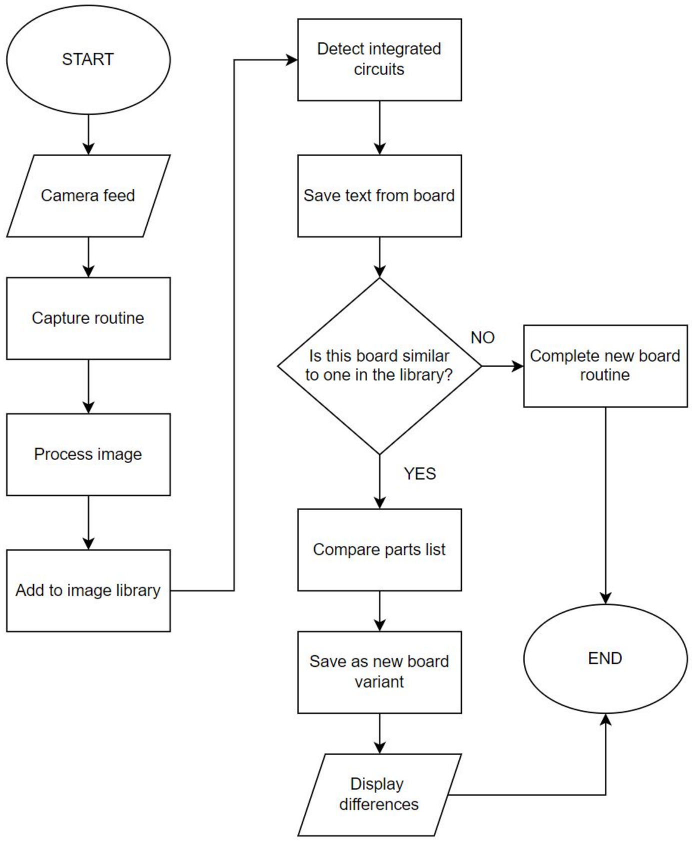

Figure 17: Software Flowchart

15 Phase 3

15.1 Software Development

15.1.1 C++ and OpenCV

The primary interface of OpenCV is in C++, a language designed for Object Orientated Programming (OOP). This meant that the programming model focused on organising the software design around the manipulation of the data. This model was well-suited to projects such as this one, where the programs were complex and required regular retrieval and maintenance of data. Other benefits of OOP involve scalability, efficiency and the ability to reuse code. The flowchart was used as a starting point to develop simplified data modelling diagrams, which were used to identify the objects and how they related to each other.

15.1.2 Classes

The structure consisted of classes, objects, methods and attributes. Four classes were created to bring together their relevant methods. These methods were functions created to describe the behaviour of the objects created with specifically defined data. Encapsulation insured that all of the important and relevant information was captured inside an object. Abstraction was used to share methods with other objects, where they could be used in specific appropriate scenarios.

15.1.3 Image Capture

This class contained three methods, two of which were used throughout the entire program. The first method initialised the camera and displayed a live stream of the input feed at a resolution of 4K. The display window was resized but the content of the stream was not saved to the PCB Library. However, the *saveImage* method was created to test various video capture properties.

1. Frame width and height
2. Rectification
3. Sharpness
4. Auto-focus
5. Auto-exposure
6. Contrast
7. Hue

All of these properties played a vital role in the cameras ability to adapt to different environments to ensure a clear template image has been captured. An additional step taken to ensure that the images being captured were in fact in 4K involved the Laplacian Operator [10]. It is commonly used to find edges within an image by highlighting sudden changes in intensity of neighbouring pixels. It combines an image with a mask, and through acting as a zero crossing detector it can identify edges and outlines of shapes.

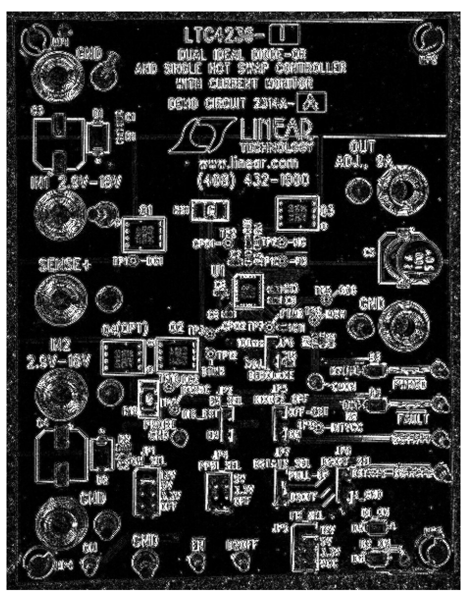

Figure 18: Laplacian Image

The Laplacian operator was applied to the original image of the circuit board when it was first captured and converted to greyscale. The results illustrated more defined edges and silhouettes, with more successful results in areas the image with a higher contrast.

The *saveImage* method was then improved, by cycling through a series of images to identify the one with the best degree of focus. This was implemented by calculating the variance of the Laplacian result image, and so if the returned value of sharpness is too low the image would not be saved due to being inadequate.

15.1.4 Library Management

The Library Management class maintains the PCB database and component library. This was achieved through creating various methods that initialised new folders dedicated to the PCB currently being examined. The routine involved taking an input string to name the folder, and using this string to create a miniature library to document further data related to that PCB. Upon folder creation, the PCB was also checked for QR codes. Once the QR code was located it would save the relevant information to a text file while saving a cropped copy of the QR itself for future reference.

The entire database was managed through a dedicated library to save images of the PCBs in all orientations, relating these images by using the card name and ID data provided by the user. These details are added to a Comma Separated Values (CSV) file, where the numbered images remain related to their original naming pattern. This allowed the program to iterate through ID numbers easier and identify a match first, then locate the ID number from the database CSV file and return the corresponding row of data containing the name of the matched board. As the program developed further, each row also contained the components present on the board, through methods used in the *ImageProcessing* class.

15.1.5 Image Processing

Images of the PCBs were processed before certain routines were carried out. These routines involved comparing PCBs, identifying components and OCR. In the case of comparing a PCB, once an image was captured and saved with appropriate pre-processing carried out as mentioned in the *ImageCapture* class, copies of the card were saved. These included a greyscale image, multiple rotations by 90 degrees to cover all orientations of the board, and a Hue, Saturation and Value (HSV) copy [11].

Operating in the HSV colour space meant that parts of the board could be more easily segmented by colour, as appose to the RGB colour space where colours are coded using three channels. For example, the hue channel was used in identifying Integrated Circuits (ICs) through detecting rectangular blocks, of which a high percentage were black. The variation of saturation levels ranged from shades of grey to full saturation, and the value channel provides a description the colours brightness or intensity. Neither of these actually influenced the hue value, and so once the correct range was found, the ICs were regularly identified.

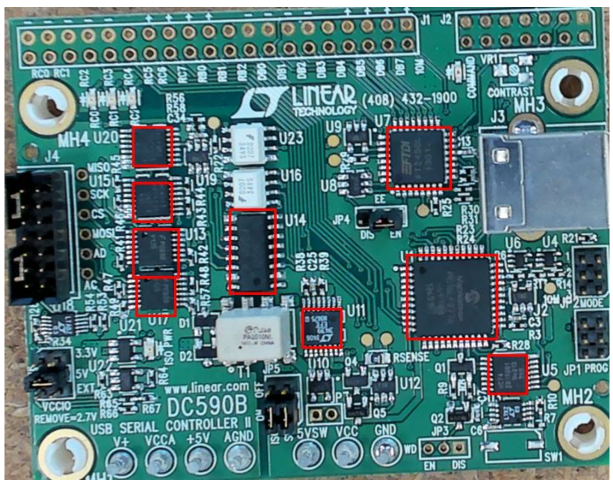

Figure 19: Component Identification

As previously mentioned, images of the boards required reprocessing to improve the accuracy of Tesseract's OCR. The images were resized using the *INTER_ AREA* interpolation algorithm [12], which re-samples using pixel area relation. Research uncovered that this method gives *Moiré-free*, overcoming a common problem with resizing high resolution images. *Moiré* is a form of spacial aliasing in which false patterns are observed within an image, often associated with false colour readings.

Various image smoothing techniques were used. The first trial involved *Gaussian blurring* [13] to mask the processing image with the specified Gaussian kernel to remove noise. This method caused a decrease in edge definition, which was counteractive to identifying text. As a result, although it was slower *Median blurring* was used to preserve edges. As appose to linear filters, the pixel values were replaced with the median value of neighbouring pixels.

Figure 20: Median Blur combined with Laplacian Operator

The image below was obtained through the application of the thresholding result of 100.107 from Otsu's algorithm [14], typically used to perform adaptive thresholding. A threshold value is calculated in order to separate the pixel values into two classes. Anything above this threshold was automatically set to 255 (black) and anything below remained as 0 (white). Adaptive thresholding was also used on larger boards that experienced different levels of lighting in certain sections [15].

Figure 21: Otsu's Thresholding Algorithm - Step 1

The *bitwise not* function was then used to invert the image by flipping pixel values. All of the pixels greater than zero became zero, and those that were zero became 255.

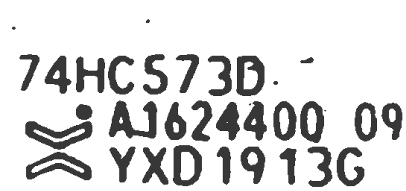

Figure 22: Otsu's Thresholding Algorithm - Step 2 Inversion

15.1.6 Board Comparison

Multiple methods were made to compare the boards against each other, whether that was with the entire library or just variations of the same board. In order to compare a new board with all previously seen circuit boards, they were referred to by PCB ID numbers as they were easier to iterate through. Every new board added to the database obtained a dedicated folder and copies of the board were saved into a database as mentioned in the Library Management subsection. The library comparison routine iterates through all of the boards within the data base, performing a basic match. Obtaining a match from boards that had their images captured at different times of day in a variety of environments was difficult, and so the process had to be upgraded to carry out a more in depth comparison.

At first, the *template matching* [16] was used to find areas of one image of a PCB that matched another area in another image. This method required a template image and a source image. While iterating through the database those images were defined as the source images, with the new image acting as a template. This new template image is compared to the sources by iterating through PCB ID numbers in a loop, checking every pixel in every image in the database. The method used two images with multiple channels, the result was a single channel image of averages, which made it easier to analyse.

This approach was also used to locate various components, so that an identical set of components being found on the PCB could result in a match as well. This was done through using *minMaxLoc* [17]. Given a template image, the function would search the PCB for matching minimum and maximum elements. Every pixel of the image was tested for potential matches, if the component template picture was located on the PCB it was highlighted with a red rectangle. The user was then notified of each successful component detection, with an alert for missing components. The components not found were highlighted by displaying an image of the component and a separate window containing a rectangle around the section of the PCB where it should have been.

Histograms of images were developed to graphically represent the distribution of data, such as pixel intensities [18]. The number of pixels which fell into ranges of various bins were tracked and represented as a histogram. The data obtained did not just involve the colour intensities, but the gradients and directions too. Using this method, the image of the PCB was divided into three separate planes of red, blue and green. The data was correlated into a histogram, which could then be compared to other PCB's histograms to identify matches.

The difference between the two histograms was analysed to understand the change in colour and intensity distribution, produced a percentage of likeliness score between the two images. The application of this method was used to understand how the environment impacted the pixel's illumination levels in order to understand how the images should be processed.

15.1.7 Depth Map and Distance Measurements

Distance measurements were obtained through initialising the depth sensor and a frame pipeline to configure, start and end camera streaming. The pipeline was then able to deliver frames as they came in, where it was then fed into the depth frame interface. Given the frame width and height, it was able to obtain the distance to the central region of interest defined as the PCB on the mount. The frame was then released, the distance was obtained and displayed to the user.

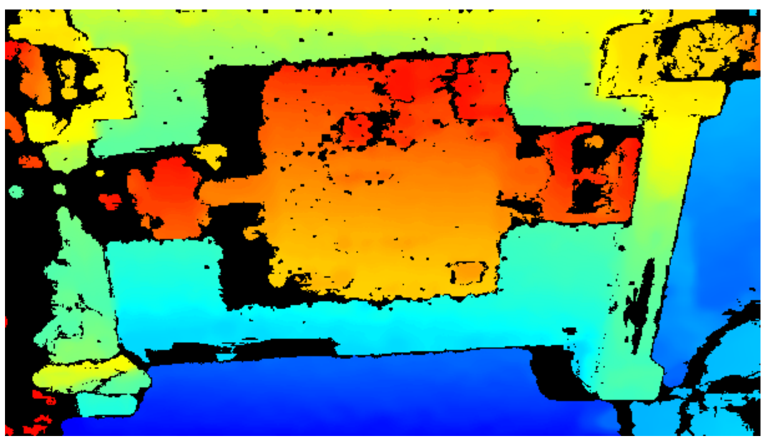

Figure 23: RealSense Depth Map

The depth map was obtained through developing the above method to declare an additional depth map colourizer to enhance the visualisation of the depth map. The depth frame was then obtained and aligned with the colourizer to generate a near and far mask that would isolate the PCB from the background. The *grabCut* algorithm was then able to generate the resulting image below by extracting foreground pixels based on the refined mask.

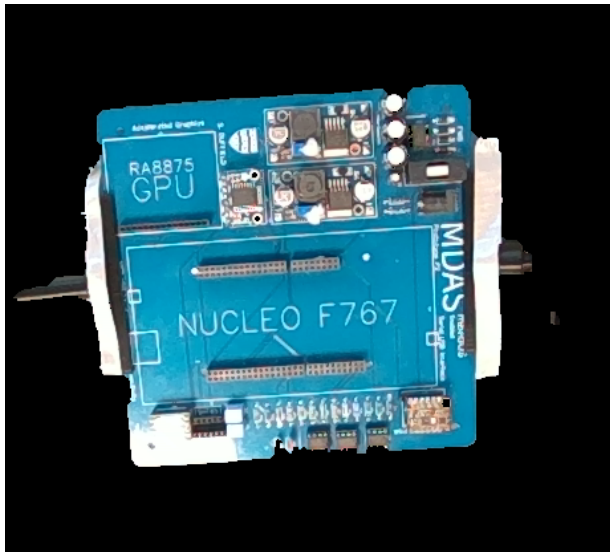

Figure 24: Background Removal

16 Evaluation

16.0.1 Hardware Testing

The testing of the hardware was discussed throughout Phase 2, with new iterations of designs being created as a result of operating the frame itself. The final frame and PCB mount design were used during the last few sprints of the project. The most noticeable change in the design through every day use was the increased robustness. As the workspace became more populated, the frame was transported more frequently. New users who had not seen the product before were asked to manipulate the boards and compliments were given for its ease of control. The bespoke panel designed to house the microcontroller, driver and external peripherals was easy to use. The safety features were also tested by a range of people to see if the traffic light system was clear to understand.

16.0.2 Software Testing

Around 3,000 images of 50 boards were combined to create the data set. These images were taken from different perspectives, with different surrounding environments and various stages in the processing algorithm. The ability to recognise a board based on feature localisation was tested for multiple components on each board, increasing the Component Library to roughly 5,000.

Further testing was carried out on PCBs of multiple colours and sizes. Out of the four colours of circuit board processed, blue was the colour that required the most processing. The program was still successful in regards to identifying the board and its components but improvements needed to be made to make it more robust. The image processing required was adapted depending on the level of lighting in the room, leading to the discovery and implementation of Otsu's method discussed in Phase 3.

To test how well the board identifier worked, small sections of the same board were edited to illustrate that even the smallest components could be detected. The same approach was taken to investigate how well the OCR methods operated in challenging environments. This was done on a box of 50 transistors, all placed at slightly different angles and alignments. The results indicated that those laying at 0, 90, 180 and 270 degrees to the cameras orientation were captured every time. However, those in between often could not be recognised or had characters incorrectly identified due to the misalignment in orientation. This was not believed to be a problem as of yet, since the text on components and circuit boards did not often occur at obscure angles, but in terms of future developments this was a challenge to be overcome.

16.0.3 Reliability

To investigate the reliability of the program, 30 boards of the same type were purchased from three different suppliers. Each board was individually added to the PCB Library, and so 30 tests on the first step of the program resulted passed. Each board was successfully added to the system with its own dedicated folder and additional requirements below.

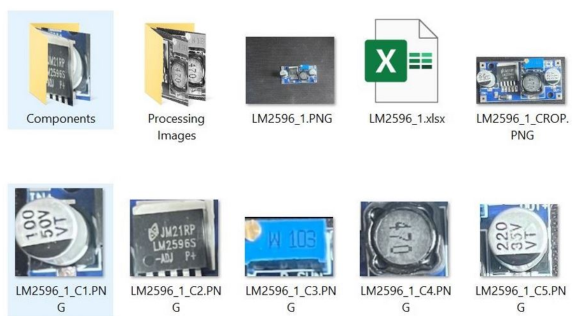

Figure 25: PCB and Component Folder Contents

Then the components were identified, and the OCR routine was applied to obtain the text from the different components. The image below demonstrates how the same boards are identified, and how their individual components are also documented in the larger database. The program's mistakes were highlighted in red in *Figure 26*.

In order to evaluate the reliability of the system, certain criteria was critically evaluated in the table below. The library creation was an important part of the routine. If mistakes were made during this stage it would result in corruption of the entire database. However, all 30 PCBs had their own library created successfully. The following step was the identification of features, being the components, on each of the boards.

	A	B	C	D	E	F
1	LM2596_1	100V 50V VT	JM21RP	W 103	470	220 35V VT
2	LM2596_2	100V 50V VT	JM21RP	W 103	4	220 35V VT
3	LM2596_3	100V 50V VT	JM21RP	W 103	470	220 35V VT
4	LM2596_4	100V 50V VT	JM21RP	W 103	47O	220 35V VT
5	LM2596_5	100V 50V VT	JM71RP	W 103	470	220 35V VT
6	LM2596_6	100V 50V VT	JM21RP	W 103	470	220 35V VT
7	LM2596_7	100V 50V VT	JM21RP	W 103	470	220 35V VT
8	LM2596_8	100V 50V VT	JM21RP	W 103	470	220 35V VT

Figure 26: PCB Component Database

One situation arose where the system identified a region of interest that did not contain the same text as the other boards did. The figure below was taken to present an example of the cases in which component four had returned a reading of anything other than "470". The fourth PCB in the image shows a component reading "SS34", where as all of the others read "SS14". This difference in the text was not picked up by the human-eye, but instead raised as a discontinuity in the board design.

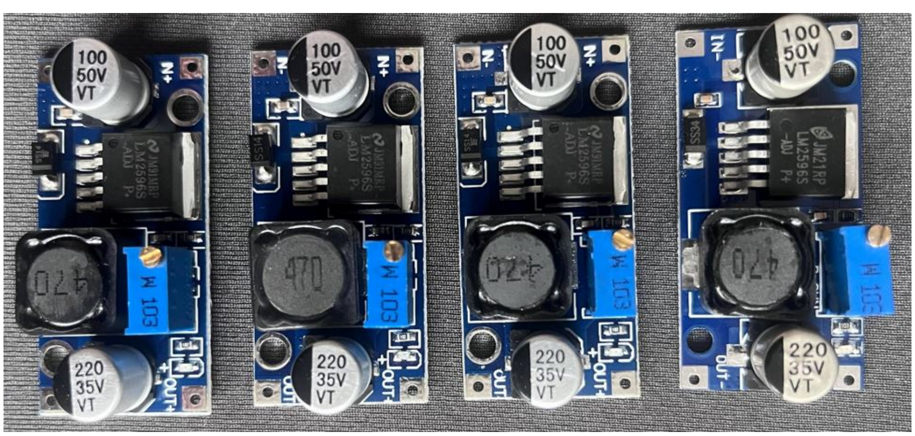

Figure 27: PCBs with Failed OCR

The PCBs shown above were harder to examine with the system due to damaged components. The boards were purchased as brand new, so this error cannot be accounted for, instead the algorithm identified the components as unmatched to alert the user that the component was worth investigating. This fulfils the aims and objectives that the product had to achieve.

Routine	Successes	Failures	Reliability Rating
Library Creation	30	0	100%
Component Identification (5 per board)	29	1	97%
Component 1	30	0	100%
Component 2	28	2	93%
Component 3	30	0	100%
Component 4	26	4	87%
Component 5	29	1	97%
Board Text	28	2	93%
General OCR Accuracy	171	9	95%
Total Reliability			96%

Figure 28: Reliability Scoring System

The reliability scoring system was used to record the accuracy of the system in relation to the various stages of the program. A total reliability score of 96% was achieved by totalling the successes and failures of each routine. The lowest score of 87% reliability was due to manufacturing differences and so it was evident that a larger data set was required to build up a representation of that specific component to understand how the appearance varied.

On the other hand, the component was flagged by highlighting it to the user, and so a second to look at the component would have confirmed that this was indeed the correct. This approach was still faster than a visual inspection of each component, and so having to check one was not seen as a cause for concern, only an area of interest into further developing the algorithm to adapt to those situations. 171 out of 180 components were correctly identified by just taking an image of the boards, and although they were added individually to the system, this was more time efficient than visually inspecting each of them manually.

16.0.4 Improvements and Future Enhancements

The physical improvements to the project involve future plans to automate the camera brackets positioning based on the feedback of real time distance from the RealSense camera. Automation of board feeding and capturing into the system would greatly increase the throughput. A conveyor belt concept design had been pitched, with the ability to flip the board to view that under side would allow the user to focus on other work instead of manually feeding the system circuit cards. Manually adjusting the height of the camera was a time consuming process which with little future work could be resolved. The depth perception and live distance reading program was completed with the integration of screw threaded steppers to adjust the camera height in mind.

Additional perspectives of the board could be obtained through implanting a camera mounting system with more degrees of freedom. For example, a camera mounted on a single arm that can follow a planned route around the PCB mount would be capable of reading text on a wider variety of components. These type of future developments require a more robust safety system to be implemented. The possibility of using the real time object detection capabilities of the RealSense camera was investigated. Intel have used this camera to develop hand tracking systems [19], and so not only could the camera detect hands entering the active workspace and shut off power to the system, the capability of controlling the system with hand gestures could be additionally explored.

Tesseract was chosen for its high speed recognition of the characters in the alphabet. However, other open source OCR engines were known for having a higher accuracy when recognising numbers. Although the latest release of Tesseract had the neural network system implemented to obtain better OCR results, the pre-processing routine required more enhancements to increase to robustness.

17 Further Considerations

17.1 Health and Safety

A risk assessment was produced before the project began. This was used to highlight any hazards and the level of risk associated with each task. The risk assessment details the steps taken to mitigate the risk and the further actions required to ensure that the work is carried out safely.

17.2 Commercial Risk Evaluation

The risk analysis involved highlighting the aims and objectives of this project that could have changed due to an event. For example, if Babcock were no longer able to sponsor the project and act as industrial advisors. This event would have had a major effect on the quality of the hardware and access to the high standard cameras may not have been possible. The impact of this was mitigated by acquiring the sponsorship money in advance of the project semester starting. This meant that the equipment of high monetary value was ordered months before, reducing the risk of having no access to the tools as and when they were needed.

Most of the materials and components required for the project were purchased through Babcock's supply chain. The length of time this took was varied and so the approach of placing orders weeks ahead of the planned work, as illustrated in the Gantt chart, was used to overcome this. The smaller orders were placed with the University in order for them to arrive in a more timely manner.

17.3 Legal Requirements

This project had no legal requirements per say, but the mutual agreement that all parties would benefit from the final product directly, or aspects of the project. While the PCB and Component Library created contained no images of circuit boards from the ERF the project remained unclassified.

The project sponsorship agreement was made with the understanding that the student's project would be advertised in a commercial manner. This entailed a case study to advertise an overview of the work completed to be shared by Babcock.

The legislation outlined by Babcock's BMS was applicable to the United Kingdom. This was sufficient as the development only took place in the UK, with the product's operation only intended for use in CWEW - Devonport, Plymouth.

17.4 Environmental Impact

The environmental impact of this project was researched during Phase 1, with the information obtained influencing the decisions made during the down-selection and optioneering of materials and components. This project aimed to reduce electronic waste and so its environmental impact was at the forefront of the decisions made.

17.5 Ethical Conduct

The PCB and Component Library only contained images of boards which were either available to the general public as purchasable products or donated for testing means with expressed permission to do so in writing. Those who donated the boards containing their own designs were aware of how they were to be used, including the fact that were unassembled to varying degrees.

17.6 Social Context

This project did not effect the general public, it did however have an impact on those working in the University laboratory. The frame, being of substantial size, was attached to a movable surface so it could be easily moved to accommodate to other laboratory users.

17.7 Product Safety and Liability

The manufacturing processes used throughout the project were risk assessed, with multiple actions taken to reduce the chances of incidents occurring. The product itself has safety features implemented into the design. This involved safety routine shut down of the system and appropriate Personal Protective Equipment (PPE).

17.8 Internationalisation - Global Market

The product was created with components which would not be restricted in most countries, and so different custom laws were not a concern. However, the images captured of PCBs could have been prohibited or unreleased to other countries and so the product may have required adjustments to remove some of the technology from testing images or videos from advertisements.

18 Conclusion

This project was one of the first attempts to automate the inspection routine of PCBs within CWEW. Prior to this product, the engineers in the ERF were manually inspecting circuit boards with little to no information on them. However, this project greatly decreased the time taken to carry out a visual inspection while documenting the trends in condition of the boards within its database. The future enhancements and program improvements discussed will greatly increase the both the robustness and functionality of the product. The capability to overcome the challenges experienced throughout the project has been provided by Babcock's decision to continue development in Summer 2022, where access to a larger dataset will be acquired.

The ability to locate components through board identification and tracking using the PCB Library was achieved with the addition of QR code identification. This meant that the cards they marked with the same QR code were already matched and immediately highlighted within the system. The PCB and Component Library created provided a simplistic approach to data acquisition that achieved a reliability score of 96% upon further testing.

The project consisted of several phases which involved concept design, CAD, bespoke PCB design and circuitry analysis. The range of image processing techniques and methods used varied greatly, resulting in an abundance of new knowledge and constant enhancement of the program. Alternatively, soft skills such as critical and creative thinking were progressively developed throughout the project in order to visualise the data flow and improve the routes taken.

19 Appendix

19.1 OneDrive

PROJ324 OneDrive Folder

19.2 OneNote

OneNote containing Logbooks, Meeting Minutes and Bill of Materials

19.3 Github

Code Repository

19.4 YouTube Videos

Reverse, Refurbish and Repurpose PCBs

References

[1] HSE. *Waste Electrical and Electronic Equipment Recycling*. May 2022. URL: https://www.hse.gov.uk/waste/waste-electrical.htm#:~:text=Every%20year%20an%20estimated%202,and%20companies%20in%20the%20UK.

[2] Joe Lessard et al Charissa Rujanavech. *Liam - An Innovation Story*. September 2016. URL: https://www.apple.com/environment/pdf/Liam_white_paper_Sept2016.pdf.

[3] Apple. *Environmental Progress Report*. March 2022. URL: https://www.apple.com/environment/pdf/Apple_Environmental_Progress_Report_2022.pdf.

[4] Retronix. *Retronix*. May 2022. URL: https://retronix.com/.

[5] Tesseract. *Tesseract*. March 2021. URL: https://github.com/tesseract-ocr/tesseract.

[6] OpenCV. *OpenCV Github Repository*. December 2021. URL: https://github.com/opencv/opencv.

[7] Intel. *Intel RealSense SDK-2*. May 2022. URL: https://www.intelrealsense.com/sdk-2/.

[8] Dan Bloomberg. *Leptonica*. September 2021. URL: https://github.com/danbloomberg/leptonica.

[9] OpenCV. *Flags for Video I/O*. May 2022. URL: https://docs.opencv.org/3.4/d4/d15/group__videoio__flags__base.html#gaeb8dd9c89c10a5c63c139bf7c4f5704.

[10] OpenCV. *Laplace Operator*. May 2022. URL: https://docs.opencv.org/3.4/d5/db5/tutorial_laplace_operator.html.

[11] OpenCV. *Thresholding Operations using inRange*. May 2022. URL: https://docs.opencv.org/3.4/da/d97/tutorial_threshold_inRange.html.

[12] OpenCV. *Geometric Image Transformations*. May 2022. URL: https://docs.opencv.org/4.x/da/d54/group__imgproc__transform.html.

[13] OpenCV. *Smoothing Images*. May 2022. URL: https://docs.opencv.org/4.x/dc/dd3/tutorial_gausian_median_blur_bilateral_filter.html.

[14] LearnOpenCV. *Image Thresholding*. August 2020. URL: https://learnopencv.com/otsu-thresholding-with-opencv/.

[15] OpenCV. *Image Thresholding*. May 2022. URL: https://docs.opencv.org/4.x/d7/dd0/tutorial_js_thresholding.html.

[16] OpenCV. *Template Matching*. May 2022. URL: https://docs.opencv.org/3.4/de/da9/tutorial_template_matching.html.

[17] OpenCV. *Operations on Arrays*. May 2022. URL: https://docs.opencv.org/3.4/d2/de8/group__core__array.html#gab473bf2eb6d14ff97e89b355dac20707.

[18] OpenCV. *Histogram Calculation*. May 2022. URL: https://docs.opencv.org/3.4/d8/dbc/tutorial_histogram_calculation.html.

[19] Abhishake Kumar Bojja et al. "HandSeg: An Automatically Labeled Dataset for Hand Segmentation from Depth Images". In: *2019 16th Conference on Computer and Robot Vision (CRV)*. 2019, pp. 151–158. DOI: 10.1109/CRV.2019.00028.