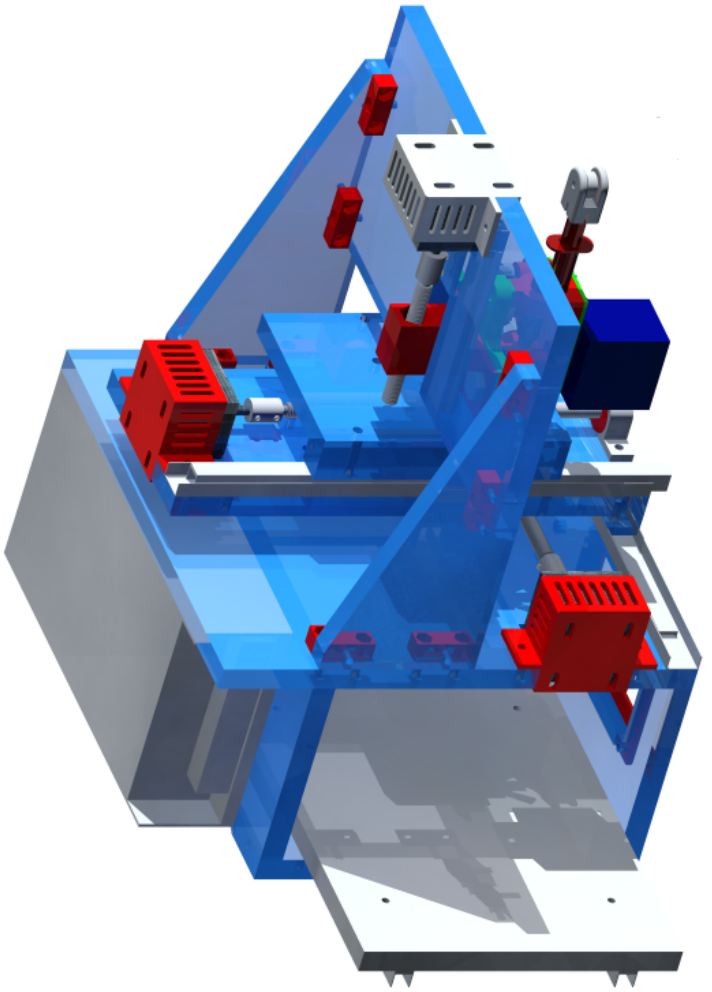
4th Year Project: Automated soldering/solder paste machine (C-PJGL2-7)

James Glanville

11th June 2013



Contents

1 Introduction **5**

2 Project Concept **6**
- 2.1 Existing Solutions . . . 7
  - 2.1.1 Methods using solder in wire form . . . 7
  - 2.1.2 Methods using solder paste . . . 7
- 2.2 Evaluation of project approach . . . 9
- 2.3 Possible expansions of project scope . . . 9
- 2.4 Project specifications . . . 9
  - 2.4.1 Device placement . . . 9
  - 2.4.2 Board size . . . 9
  - 2.4.3 User interaction . . . 9

3 Design **9**

4 Machine motion **10**
- 4.1 Requirements . . . 10
- 4.2 Design selection . . . 10

5 Electronics **11**
- 5.1 Specification . . . 11
- 5.2 Power Supply . . . 12
- 5.3 Stepper motor drivers . . . 12
- 5.4 PC interface . . . 12
  - 5.4.1 USB-serial cable . . . 12
  - 5.4.2 Bluetooth link . . . 12
- 5.5 Microcontroller and firmware . . . 13
- 5.6 Command storage . . . 13
- 5.7 Firmware . . . 14
- 5.8 LCD and user interface . . . 14
  - 5.8.1 Hardware . . . 14

6 Solder paste extruder **14**
- 6.1 Requirements . . . 14
- 6.2 Initial tool . . . 15
- 6.3 Design improvements . . . 16
  - 6.3.1 Paste extruder using bevel gears . . . 16
  - 6.3.2 Paste extruder using worm gears . . . 17

7 Device placement tool **17**
- 7.1 Requirements . . . 17
- 7.2 Design . . . 17

8 Spindle **18**
- 8.1 Requirements . . . 19
- 8.2 Design . . . 19
- 8.3 Prior art . . . 19
- 8.4 Chosen design . . . 19
- 8.5 Improvements . . . 19

8.6 Milling 20
8.6.1 Gear CAD 21
8.6.2 Milling Settings 21

9 Hot air tool **21**
9.1 Requirements 23
9.2 Calculations 23

10 Machine Handbook **24**

11 Host computer software **24**
11.1 Improvements to pcb2gcode 24
11.1.1 Adding solder paste support 24

12 Tools **26**

13 Managing devices prior to placement **26**

14 Vacuum Placing **27**
14.1 Prototype 27
14.2 Conclusions 28

15 Operating Speed **28**

16 Measuring Accuracy **29**
16.1 Testing methodology 29

17 Operating Steps **32**

18 User interface **33**
18.1 Pronterface 33

19 Zeroing the axes **33**
19.1 X and Y axes 33
19.2 Z axis 33
19.2.1 Vacuum needle 33
19.2.2 All other tools 34

20 Future Work **34**
20.1 Double-sided boards 34
20.1.1 Challenges 34
20.1.2 Envisioned solutions 34

21 Conclusions **35**
21.1 Machine Performance 35
21.1.1 Machine axes 36
21.1.2 Spindle 36
21.1.3 Isolation Routing 36
21.1.4 Pick and place 36
21.1.5 Hot air tool 36
21.2 Project Management 36
21.3 Evaluation of method of PCB manufacture/population 37

21.3.1 PCB manufacture 37
21.3.2 Solder paste extrusion 37
21.4 Final Conclusions 37

Appendices **38**

A Servo tester **38**

B Calliper serial output modification **39**

C SMD component types **40**
C.1 Two-terminal passive components (mostly resistors and capacitors) 40
C.2 SOT: Small-Outline Transistor 40
C.3 SOIC: Small-Outline Integrated Circuit 40

D Heater Calculations **41**

E Part List **42**

F Risk assessment retrospective **43**

Glossary **44**

1 Introduction

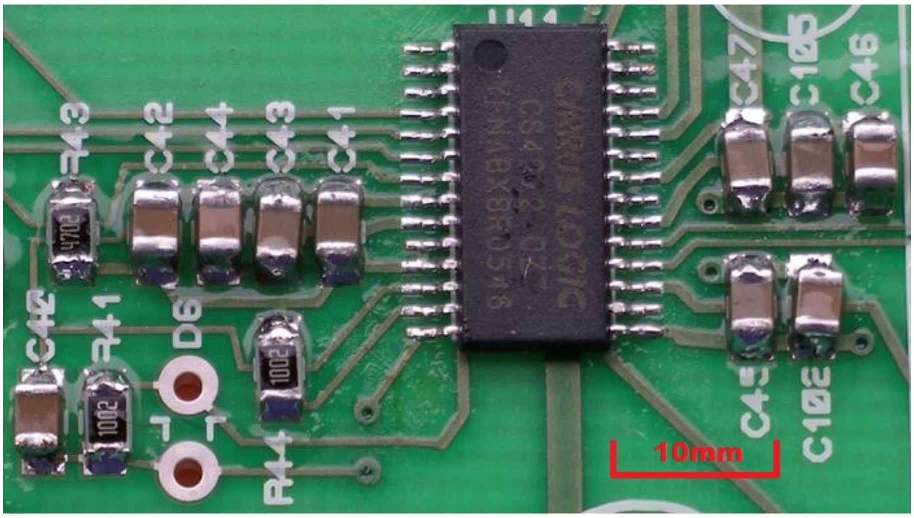


Figure 1: Example surface mount board.

The improvement in the electronics industry has led to a reduction in size of many components. Many components are now only available in surface mount packages (see figure 1). The surface mount devices (SMDs) are designed to be soldered to PCBs without requiring holes to be drilled through the board. The combination of smaller devices and no drill holes allow a much higher density of devices to be placed on a PCB. This is both commercially attractive and has facilitated dramatic reductions in the sizes of electronic equipment. However, the use of such devices necessitates expensive equipment for assembly as the device placement tolerances are reduced.

In the prototyping and hobbyist markets, assembly is usually carried out by hand, a process which requires significant time and skill. Automated systems (For example that shown in figure 2) cost upwards of £10k, which is cost-prohibitive for the majority of users.

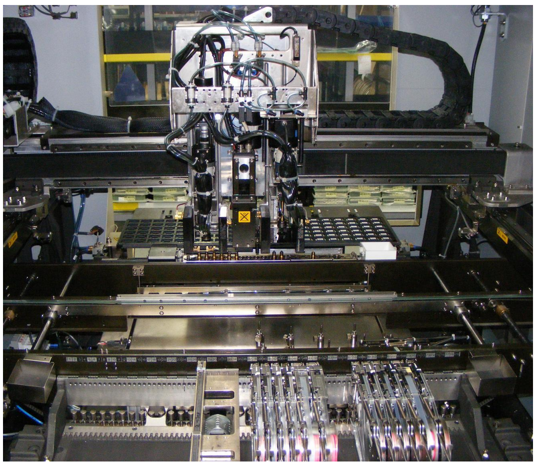


Figure 2: Juki KE2010L pick and place machine, approximately £50000.

The aim of this project is to develop ways to assist electronics hobbyists in working with SMD electronic components and circuit boards. This must be achieved in a cost-effective way to ensure it is a viable option.

As an example of the need for such a machine, see Figure 4. A USB isolator was required (to make USB connections to devices with non-isolated power supplies), which uses an ADUM3160 SOIC-28 part (See full schematic, Figure 5). Without an automated method of board manufacture, the traces had to be hand drawn in etch-resist on each side of a double-sided board. Each component had to be carefully placed with tweezers and then soldered. This was slow, and as figure 4 shows, does not give neat results.

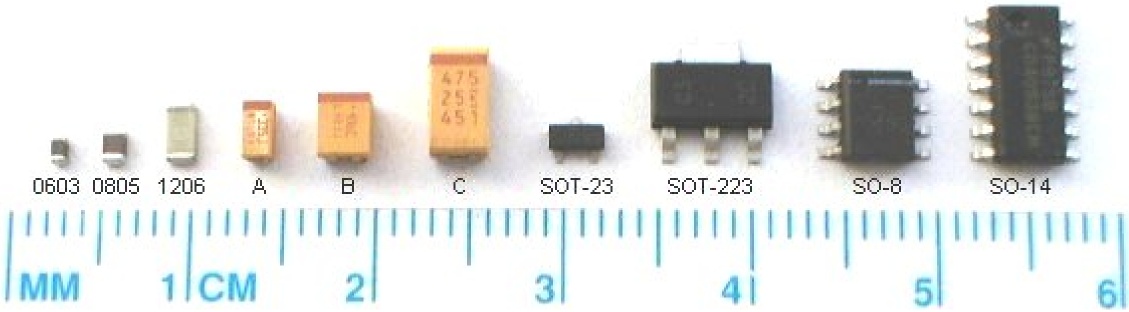


Figure 3: Example SMD components (From [7])

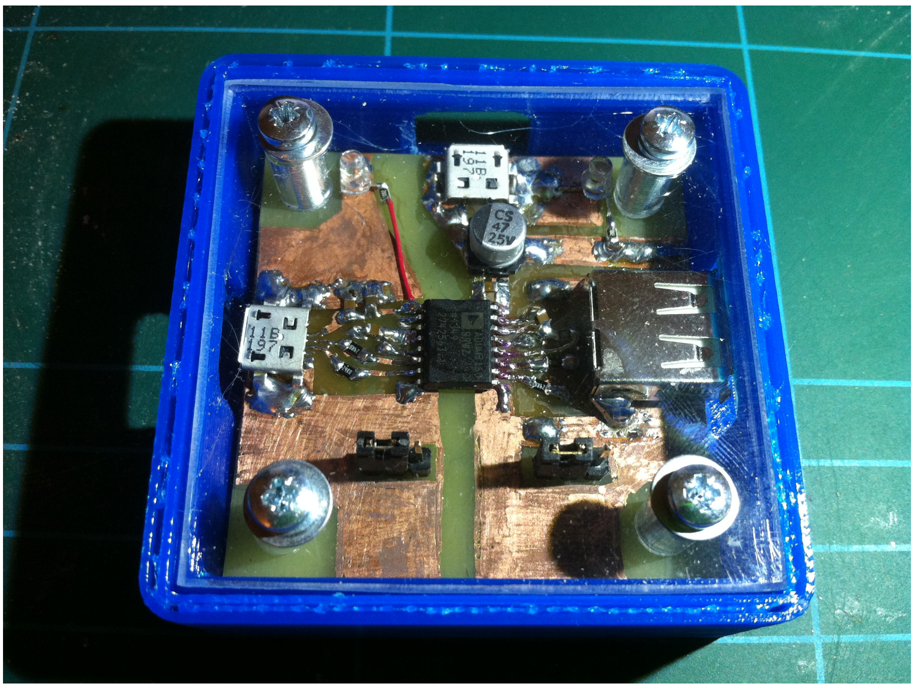


Figure 4: A USB isolator PCB with hand-drawn traces.

2 Project Concept

The focus of this project is easing both the placement and the soldering of SMDs. To this end a small machine was envisioned to automate the two tasks. This would reduce the amount of time taken to populate PCBs, and be of use to end-users who do not wish to carry out the manufacturing process by hand. The envisioned end user of such a machine is a hobbyist or academic user who wishes to produce small numbers of boards with as little user time taken up as possible.

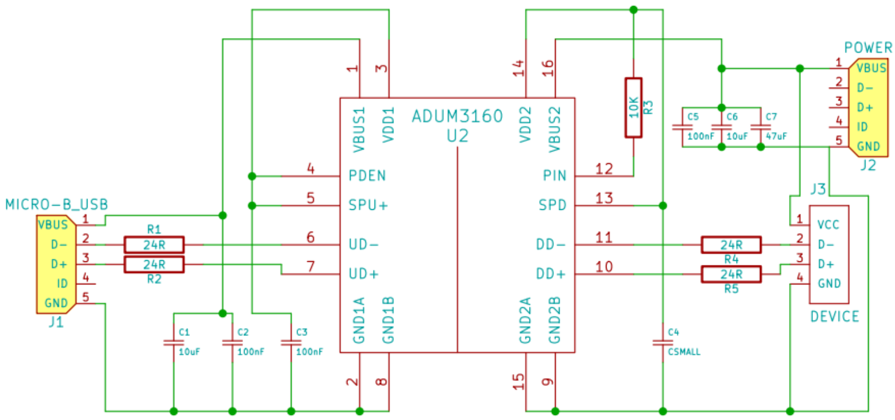


Figure 5: USB isolator schematic.

2.1 Existing Solutions

In order to identify potential improvements in manufacturing SMD boards research was carried out in to the current methods used by hobbyists. The methods were broadly divided into two categories: those using standard solder in wire form and those using solder paste.

2.1.1 Methods using solder in wire form

Despite the increased complexity, many SMD parts can be soldered by hand. This is the lowest cost process which requires the least equipment: a finer soldering iron tip may be all that is required. There are three main methodologies for this process:

- The pins are soldered individually. This is mainly suitable for larger SMD parts. A steady hand is required, as well as good eyesight. Binocular microscopes are often used to assist this process.
- The process of "drag soldering" is used. The device is placed on the board, and a small amount of solder is placed on the tip of a soldering iron. The iron is slowly "dragged" along the pins. If done correctly, surface tension effects cause the majority of the solder to remain on the tip of the iron, while the necessary electrical connections are soldered.
- The pins are roughly soldered, making sure each pin is at least soldered to the board. Many short circuits between adjacent pins will be present. Solder wick is then used with copious flux to remove excess solder between pins.

These methods are imprecise, and will often necessitate the use of solder wick or a solder sucker to remove solder bridges (See Figure 6). More importantly, they are wholly unsuitable for use with "leadless" packages. Instead of metal pins, these devices only have metal pads on their bases. Without exposed metal when the device is placed, it is not possible to solder the connections by hand.

2.1.2 Methods using solder paste

Solder paste consists of powdered metal solder suspended in flux. It is used commercially to manufacture boards with SMD components. The solder paste is applied to the board where connections are to be made. The devices are then placed on top (the paste has slight adhesive

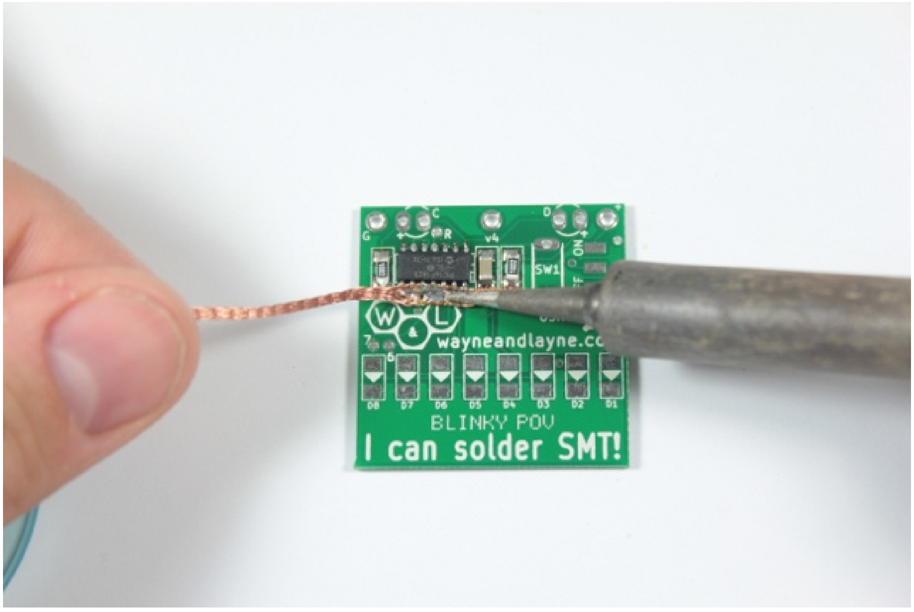


Figure 6: Removing excess solder with solder wick.

properties which limits undesirable component movement). Finally the paste is heated either directly or indirectly which causes it to melt and form solder joints. The paste is placed in a number of ways:

- Solder stencils: A thin stencil is manufactured, usually from Mylar (trade name for BoPET) or stainless steel, with holes which correspond to locations on the board that require solder. This is used to screen print solder paste to the board. This a very quick process, and suitable for production quantities, but has the expensive set-up costs of stencil manufacture.

- Solder paste can be applied by hand using a syringe. However, this requires high accuracy while placing paste and is time consuming.

- Automated solder paste extrusion uses a CNC (Computer Numerical Control) platform to manoeuvre a solder paste extruder around the board. Small amounts of paste are extruded where necessary. This is not the fastest method of manufacture, but is suitable when only small numbers of boards are required.

Once the solder paste has been placed and the devices placed on top, it must be heated to form solder connections. This can be achieved in several ways:

- A hot air gun can be used manually to locally heat the required areas of the board. Hot air guns are inexpensive, but the process requires care so as not to overheat any components. It can also be relatively difficult to get the correct temperature profile - some solder joints may be overheated while some may not fully form.

- Entire board heating involves placing the board in an oven which is heated to the correct temperature. This can vary from large commercial equipment to the use of a converted toaster oven.

2.2 Evaluation of project approach

It was decided that automating the soldering of devices using solder wire would prove too challenging, as the various methods all require a great deal of feedback and error correction to correct poor connections or shorts.

The project approach was therefore to be a CNC machine which could both place the solder paste and the devices. This was to be achieved with minimal human interaction. The solder paste would be placed by extruding it through a sufficiently small needle that could be moved in three dimensions compared to the board.

2.3 Possible expansions of project scope

Once the project concept had been chosen as a CNC machine with a syringe tool and a placement tool it became clear that such a machine may be capable of manufacturing the PCB itself without a great increase in cost or complexity if an extra spindle tool could be designed. This would allow the use of a milling bit for isolation routing through copper-clad PCBs, and a drilling bit to drill any holes for through-hole devices.

2.4 Project specifications

2.4.1 Device placement

A device pitch (distance from centre of one pin to the next) of 0.4mm is typical of small SMDs. Using solder paste, devices will shift slightly upon heating to self-align due to liquid solder's surface tension. A requirement is that the device should be placed to a small fraction of its pitch, so there is no danger of incorrect connections. An accuracy of $\pm$ 0.05mm in placement was chosen as a project goal. This should be sufficient to ensure the correct connections are made, but may be difficult to achieve in practice.

2.4.2 Board size

A maximum board size of 100x100mm would be sufficient for the vast majority of boards. 100x100mm was chosen because that is the maximum board size using the free version of EAGLE, a popular design package. As a result, there are a lot of designs that use the maximum size, and it would be useful if this project allowed those designs to be populated. The PCBs needed for the IDP fit within the 100x100mm area, so this project may be of use in the engineering department for IDP board manufacture.

2.4.3 User interaction

User interaction should be minimal. While this will not be an entirely automated process (for example the SMD parts may need to be manually loaded into a tray) the less attended time required the better. All user interfaces should be user-friendly, and all processes designed to reduce the user's workload.

3 Design

The design of the machine is split into a number of subsystems that are described in sections 4 through **??**. These are:

- Machine motion. A three-axis CNC subsystem is required to move the various tools in three dimensions relative to the PCB.

- Electronics. An electronic module is required which can drive both the machine's axes and the tools, while responding to user interaction.
- Solder paste extruder. This must be capable of accurately dispensing small volumes of solder paste.
- Device placement tool. This must be capable of picking up and placing the SMDs in their required locations.
- Spindle. This powers the milling and drilling bits to manufacture the PCB itself.
- Hot air tool. This heats the solder paste to reflow it and form connections.

4 Machine motion

This section describes the requirements for machine motion and how they are implemented in the design. There was the choice between moving the various tools over a stationary bed, or keeping the tools stationary and moving the bed. For this project only the bed will be moved, because it means that the pick and place functionality does not have to move the parts, simply lift and lower them. This should reduce the risk of the parts shifting on the needle, or falling off.

4.1 Requirements

The axes all have the same requirements of ± 0.05mm accuracy (as described in section 2.4.1). The greatest forces are encountered during the isolation routing operations, which were measured to be less than 2N. The system accuracy must be maintained under this condition.

The Z axis has the smallest required travel distance as the tools need only move high enough to clear any devices. A minimum travel of 50mm was chosen which is more than sufficient for the tallest devices. The X and Y axes must not only be capable of moving over the maximum 100x100mm board, they must also allow travel to where the devices are stored prior to final placement. A total area of 100x130mm was chosen as a compromise between maximum device storage and total machine size.

A major design philosophy was to be that all of the axes should be as similar as possible. This would result in the least amount of development work possible, as well as the most similar axis characteristics. This was later important during testing so it was not necessary to separately characterise the errors of each axis.

4.2 Design selection

Two possible drive methods were considered for use in this project:

- Stepper motors: These are brushless motors which can be controlled down to the nearest "step" (commonly 1.8^0). They require stepper motor drivers to provide the alternating magnetic field. Provided the maximum torque is not exceeded, they can be run open-loop.
- DC motors + feedback: A feedback mechanism such as a rotational encoder or linear potentiometer can be used to provide feedback for closed- loop control of DC motors. A microcontroller adjusts the current to the motor accounting for its current position and desired position.

Stepper motors were chosen as they facilitate a simple implementation. Coupled to the drive mechanism, they are driven open-loop under the assumption they will rotate precisely proportional to the number of electrical pulses. Their extra cost compared to DC motors is offset by the simpler design and lack of feedback elements to consider. They also have the advantage that they can easily be controlled by existing electronic module firmware (see section **??**). The type of motor was chosen to be a type of NEMA17 (standard motor size) motor with a 2.2kg/cm holding torque. This was found to be sufficient for an axis speed of 10mm/s without any skipping of steps.

A number of methods are available to translate the rotational driving motion to the linear motion required for the axes. Those considered for this project are as follows:

- Toothed belts and pulleys: These are simple and inexpensive. However, despite the fibreglass or steel strengthening cables, they will still elongate under load. As a result, they are not suitable for any routing/milling operations.
- Threaded rod and ordinary nuts: This is an inexpensive solution that relies on the threaded rod having a constant pitch. Strong springs are added between each pair of nuts to provide an anti-backlash effect.
- Ball screws: These are similar in operation to threaded rod and ordinary nuts, but use ball bearings between the nut and rod for smoother operation. These are sprung against the rod for an anti-backlash effects. They are higher performance, but more expensive.

Threaded rod and ordinary nuts were chosen for use in this project because of their low cost compared to ball screws, and their strength compared to toothed belts. A thread size of M8 was chosen as it provided sufficient accuracy at a high enough speed.

The axes are required to move the bed in three dimensions. However the threaded rod is only capable of providing accurate motion if the bed is constrained to move only in the axis direction. To this end a slide mechanism is required. Three such mechanisms were considered:

- Drawer slides: Inexpensive drawer slides contain pre-loaded ball bearings which reduce wobble.
- 3d printed bushings and steel rod: If 3d printer existence assumed, then very cheap solution. These bearings will wear over time, so replacement must be trivial.
- LM8UUs and steel rod: About 50p each, simple to use and accurate.

Drawer slides were chosen because of their cost advantages as well as the fact they can easily be coupled to flat boards.

5 Electronics

The electronics subsystem must be capable of driving the machine's axes and tools, carry out the required commands for PCB manufacture, and respond to the user's input and provide feedback. This section includes the firmware which will run on the electronics system.

5.1 Specification

The stepper motors used in the project are rated to a maximum of 2.5A drive current which must be provided by the electronics.

The machine should ideally be capable of standalone operation (i.e. without an attached PC). This allows the machine to operate at a further distance from the user which would alleviate concerns about noise and dust. This will require the use of some form of storage for the manufacturing commands.

The spindle motor is a brushless motor with a maximum current draw of 20A. This requires a controller to provide the necessary switching signals.

The PC interface should be compatible with all major operating systems.

5.2 Power Supply

The operating voltage of the stepper motor drivers and the spindle motor is 12V. Assuming the current draw from the electronics is small, the peak current can be calculated to be 27.5A (20A for the spindle, 2.5A for each axis). In practice the average current will be far lower as the spindle draws a much smaller current when not accelerating. A 12V 30A power supply was chosen to power the machine. It has over-current and overheating protection built in, but as a failsafe, a 30A fuse is added in-line with the +12V cable. An automotive-style fuse was used for the lowest cost, and ready availability of suitable fuse ratings.

5.3 Stepper motor drivers

Stepper motor drivers are required to translate between logic-level motion commands and the high current pulses required to drive the stepper motors. This type of drive is frequently required in commercial products such as printers, scanners and automated machinery and as a result a great number of suitable ICs are available.

For this project Toshiba TB6560AHQ drivers were selected as the lowest cost part with sufficient drive current. These need only two input signals from the electronics module, **step** and **direction** and provide the necessary control to the motor. They require minimal support circuitry, merely decoupling capacitors and current sensing resistors.

5.4 PC interface

For simplicity, the communication between the PC and the electronics board is implemented as a serial link. An arbitrary baud rate of 57600 has been chosen, this is sufficiently fast so that the electronics board is not slowed by waiting for instructions (as the firmware buffers incoming commands in a small buffer in RAM), and sufficiently slow that the error rate is assumed to be zero. The operating voltage of the microcontroller is 5V, so the serial link is not RS232-compliant. Instead either a USB->serial cable or a bluetooth module is required:

5.4.1 USB-serial cable

A USB to serial cable based on the FTDI FT232R chipset can be plugged into the serial header on the electronics board. This is a cheap (<£5) device that is a complete solution. Driver support for the FT232R is available for all platforms that have been considered (Windows/OS X/Linux/Android). A usb serial cable was chosen instead of an integrated chip because it allows more flexibility for using other interfaces instead. For example, a real RS232 port may be used with a trivial MAX232 level shifter.

5.4.2 Bluetooth link

A bluetooth module can be installed to provide wireless communication which may be more convenient. The HC-05 bluetooth module was chosen because of its low cost (£3.50) and simplicity. The board requires a 3.3V supply, which is not present on the electronics board. A small breakout board was manufactured for the module, including an AMS1117-3.3 3.3V linear regulator (£0.10) and link status LEDs. The 3.3V TX output from the module is sufficient for the AVR despite its higher 5V supply. To avoid damage to the bluetooth module, a potential divider consisting of two 1k resistors is used. This 2.5V output is again sufficient for the bluetooth module.

To configure the baud rate for the bluetooth module, it was connected to a pc using a USB-serial cable. The bluetooth module is powered up whilst the "KEY" pin is held high (3.3V). This starts the module in AT configuration mode. The baud rate was then set by sending the string "AT+UART=57600,1,0" at 38400 baud. This configuration is only required once, the module stores all configuration in non-volatile storage.

5.5 Microcontroller and firmware

The electronics subsystem requires a microcontroller running firmware that can translate incoming commands from the serial port into commands to the three axes and the various tools.

An Atmel ATmega1284P chip was chosen as the controller for the following reasons:

- It has sufficient pins to drive all of the stepper motors, tools and user interface.
- It is capable of running at the 5V logic voltage of the stepper motor drivers.
- It is a common choice for running CNC tools, and as a result a great deal of existing firmware is available to hasten development.

5.6 Command storage

In standalone mode the required commands must be stored on the machine. SD cards are ideal for use with microcontrollers, as they have a simple protocol (SPI), and are inexpensive. They also have wide compatibility with other devices. It would be useful if the GCODE could be stored on an SD card and read by the machine.

SD cards can operate in SPI mode, using only four connections (MISO, MOSI, CLK, CS). However, they operate at 3.3V, and the AVR operates at 5V. To power the card, an AMS1117-3.3 linear regulator is used. Level translation must be used to convert the four connections to the required voltages. There are three options for this process:

- Potential dividers. This is the simplest option, using only resistors. The CS, MOSI and CLK connections send data to the SD card. A potential divider is used to reduce the voltages from 5V to 3.3V. The MISO connection receives data from the SD card. The AVR has a high input threshold of above 2.6V. MISO can therefore be connected as it operates up to 3.3V. Although this is the simplest option, the high impedance output from the potential divider can cause problems at fast SPI speeds, as the SD card presents a relatively high capacitive loading.
- Discrete transistors can be used to translate between 3.3V and 5V. This has the advantage of faster operation, and increasing the noise margins (reduced when using 3.3V with a 5V chip). Two transistors are needed for each connection to ensure the signal is not inverted.
- A level translation IC can be used. This needs no additional components (except from a decoupling capacitor).

Here a level translation IC was chosen.

5.7 Firmware

The microcontroller requires firmware which can fulfil all of the subsystem requirements. Usefully such firmware already exists to drive 3d printers and other CNC machines, so this could be used without significant modification. The "Marlin" firmware ([12]) was chosen as it readily supports operation from SD cards, as well as an LCD interface.

5.8 LCD and user interface

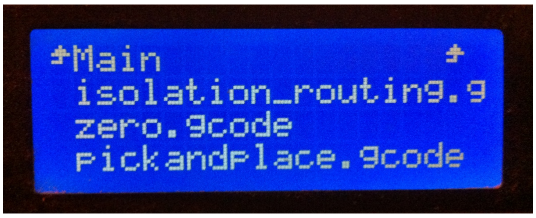


Figure 7: LCD user interface.

If the machine is to be used in a standalone capacity, there must be a way for the user to choose which GCODE file to send. Figure (7) shows the machine's user interface.

5.8.1 Hardware

A 4x20 LCD is used to display information about the current job, as well as allow the user to select GCODE files. This is an inexpensive part based on the ubiquitous HD44780 controller chip. It is operated in 4-bit mode (as opposed to 8-bit mode, which requires four additional microcontroller pins). Six connections are required to the microcontroller: RS, EN, D4, D5, D6, D7. As the electronics board does not have many spare connections, it was decided to use an MCP23017 i2c port expander to connect the LCD. The Marlin firmware already supports this, and needs just two connections (SDA and SCL) to the driver board. See Figure 8 for the schematic.

A rotary encoder is used to provide input. This uses quadrature outputs to determine motion. The encoder also has a push switch inside for selecting options. This therefore require three microcontroller pins in total.

6 Solder paste extruder

The solder paste extruder is a tool which dispenses controlled volumes of solder paste on to the board.

6.1 Requirements

The tool must be capable of accurately extruding very small volumes of solder paste. As an example, a 0.4x0.2mm pad to be coated with a thickness of 0.4mm of paste requires just 32nl of paste.

Solder paste deteriorates in contact with the air. It is supplied in syringes from which it should be directly extruded, avoiding transfer to other containers.

The tool must not extrude in places it is not required to, i.e. dribble. The pressure applied to the syringe must therefore be rapidly released at the end of extrusion.

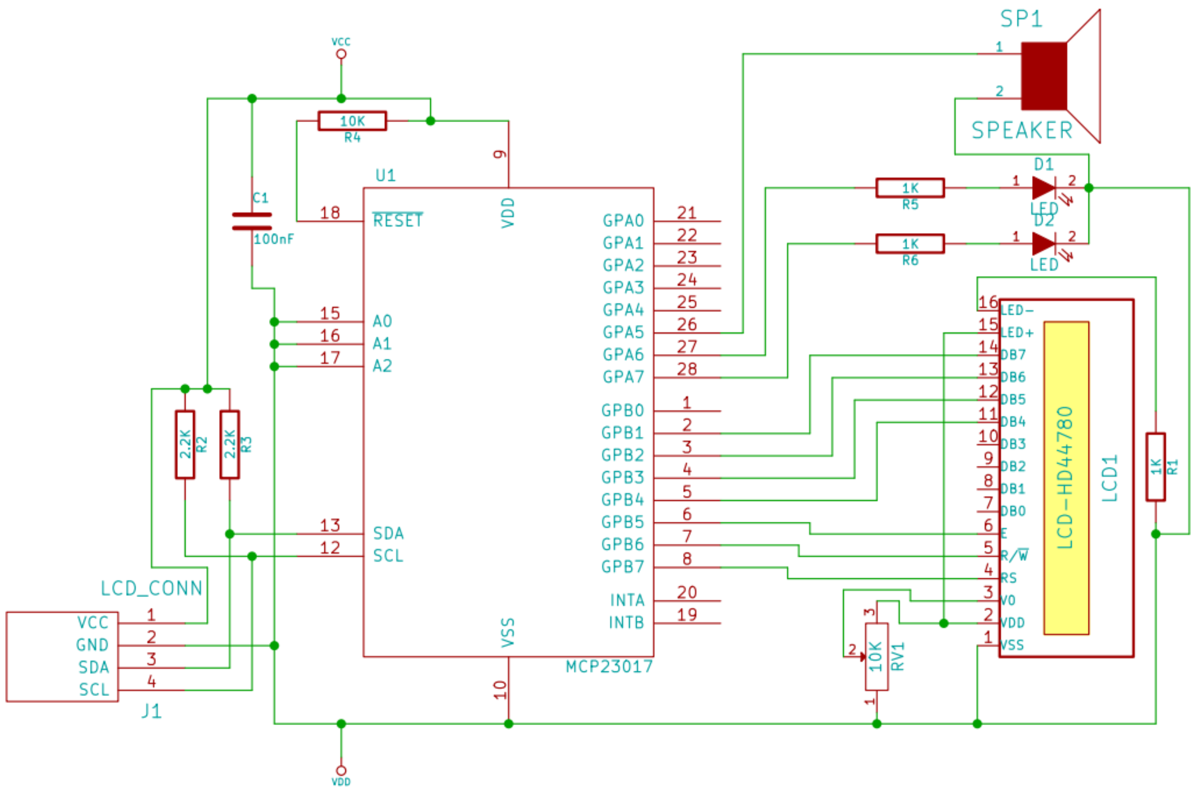


Figure 8: Schematic of i2c ->LCD board.

6.2 Initial tool

Initially, a freely available design for the paste extruder was used [4].

The extruder works by applying pressure to the top of the solder paste syringe. Pressure is applied by tightening a pulley made of T5 timing belt with a geared stepper motor. This design was chosen because the extruder is compact, and the design allows for fast reduction of pressure, important for clean extrusion and to avoid the solder paste dribbling.

To calibrate the extruder, two parameters are needed: steps/volume extruded and retraction steps. The first is easy to calculate:

(All dimensions in mm,mm^2,mm^3)

The solder paste is supplied in a 2.5ml syringe, with a 66.1276mm^2 cross-sectional area.

$$\begin{aligned}
\text{Volume extruded / step} &= \frac{\text{cross-sectional area of syringe} \times \text{T5 pitch} \times \text{number of teeth of T5 pulley}}{\text{number of microsteps} \times \text{steps/revolution} \times \text{gear reduction ratio}} \\
&= \frac{66.1276 \times 5 \times 10}{16 \times 200 \times 12} \\
&= 0.0861 mm^3 \\
\text{Steps / mm}^3 &= \frac{1}{\text{volume extruded / step}} \\
&= \frac{1}{0.0861} \\
&= 11.61
\end{aligned}$$

The retraction steps must be experimentally determined. The choice is a trade-off between minimising solder paste dribble (large number of retraction steps) and reducing total process time (small number of retraction steps). It must be enough that negligible pressure is exerted on the syringe when retracted.

Some method to "prime" the extruder will be needed. Assuming the extruder is initially hand-tightened so that the belt has little slack, a possible method is to repeatedly advance the extruder and then retract by a slightly smaller amount, until the user notifies the machine that solder paste has been extruded. At this point, the extruder should pause after retracting. The solder paste deposition process can then commence since the exact point of extrusion has been established.

6.3 Design improvements

The solder paste extruder design that was initially used was unnecessarily bulky and complex. This made it both harder to fit to the tool mounting mechanism, and increased the distance between the mount and the tool, amplifying any wobble in the mount. It was therefore deemed necessary to redesign it, optimising for size. Operation and performance is similar between all designs however.

6.3.1 Paste extruder using bevel gears

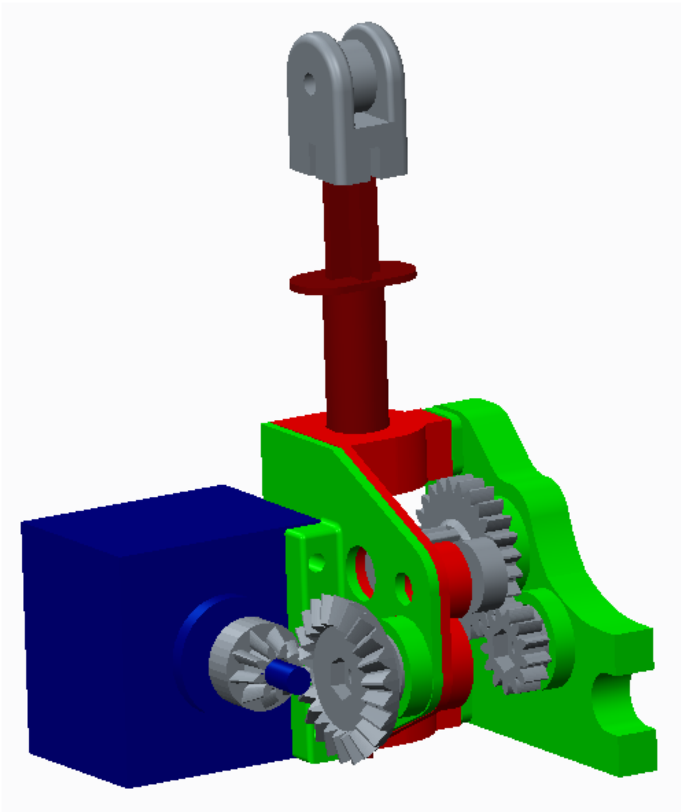


Figure 9: Placeholder image of bevel extruder

The first attempt to reduce the size of the extruder used bevel gears to shrink the size of the gearbox.

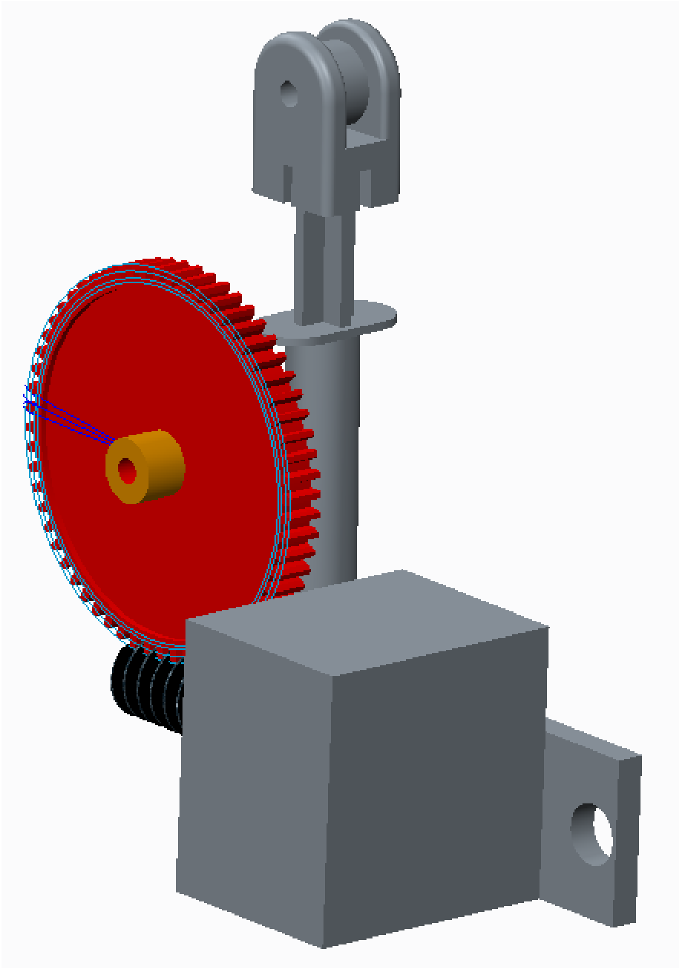


Figure 10: Placeholder image of worm extruder

6.3.2 Paste extruder using worm gears

Worm gears would give the same gear ratio in a smaller gearbox, so a new design was made using them. The improved design is as small as possible and performs as well as the much larger extruder that was originally used. See figure (11) for a rendering.

7 Device placement tool

In order to precisely place the SMDs on to the board, a tool is required which can lift them from a known location and deposit them where necessary.

7.1 Requirements

The components must be placed to the specified $\pm$ 0.05mm.

7.2 Design

A vacuum pump was chosen to power a fine hollow needle. The needle is placed over an SMD, and the vacuum pump switched on. The device is then "stuck" to the needle, which can be then lifted and moved to the correct position. Once there, the pump is switched off and the

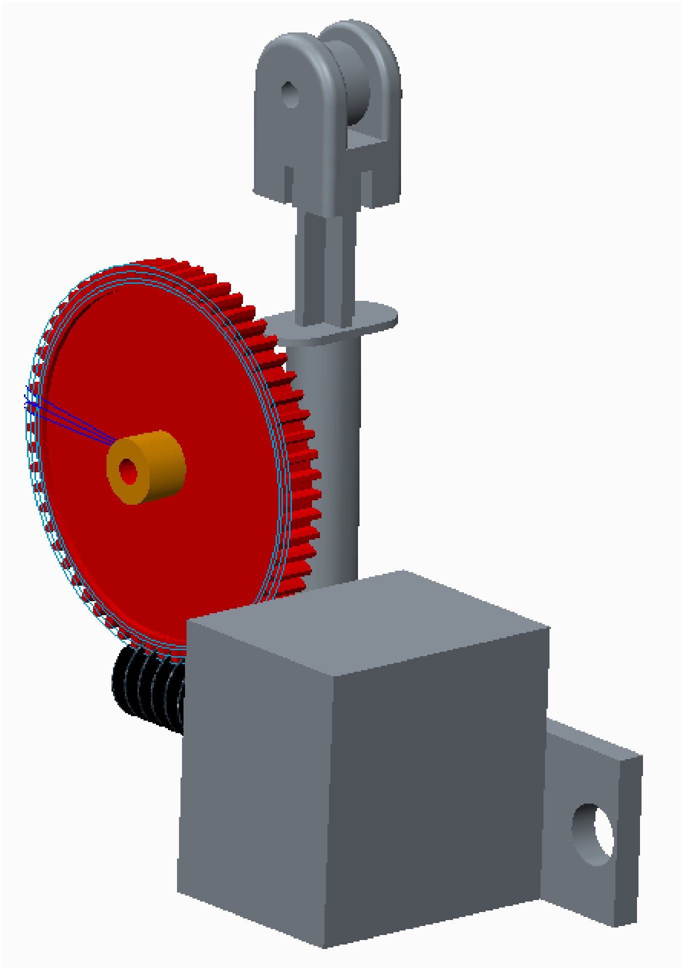


Figure 11: Placeholder image of worm extruder

device will remain stuck to the solder paste in position. This is an inexpensive solution, and also that used by commercial pick and place machines.

As each component has a different height, a microswitch is used above the needle to detect component physical contact to the needle. At this point the vacuum pump can be switched on.

A power MOSFET is used to control the pump, with a reverse-biased diode to suppress back-EMF. The MOSFET was chosen to have a small V_{GS} so that it could be fully switched on by the 5V microcontroller output.

8 Spindle

The spindle is used for isolating routing, milling and drilling operations.

Isolation routing is a method to mill traces from copper-clad pcb. A conical engraving bit is used to cut very narrow tracks through the copper, to leave disconnected pads as required. Figures 15 and 16 show a small SOIC breakout board that has been isolation routed using the machine.

Milling operations would be useful to cut the final PCB to size and to cut out any larger interior holes. While not the central focus of this project, the use of the machine to perform light milling operations would be significantly useful for many users.

Drilling holes in pcbs is a time-consuming process to do by hand. It can also be difficult to locate the holes precisely, which can be a problem for holes for small vias, or when holes must be aligned exactly (for example to fit a row of header pins). If this machine could drill the holes automatically, it would reduce the total time taken significantly.

8.1 Requirements

When isolation routing, the fine tip of the bit has very small flutes. As a result it can only manage small chip loads (the amount of material cut by each pass of a flute). To cut in reasonable time, a high speed spindle is required - up to 40000rpm.

The bits experience significant sideways forces (Approximately 2N when isolation routing, and 10N when milling). The spindle should have minimal runout under these conditions.

8.2 Design

A small brushless motor is ideal for such a spindle. The hobbyist radio controlled aircraft market provides low cost, high speed brushless motors along with the necessary speed controllers. A 25 Amp ESC (Turnigy plush 25A, £9) powering a small brushless motor (Turnigy 2217 20turn 860kv 22A outrunner, £10) is capable of milling steel (in the author's experience), so smaller, cheaper motors should be capable of routing/drilling copper. For this project a 3800KV motor was chosen for a maximum unloaded spindle speed of 45600rpm. In practice the achieveable speed will be lower due to losses in the ESC, as well as load from the cutting operations.

The biggest design challenge was to mechanically couple the motor to the required bit. There is almost zero allowance for axial misalignment, as the fine bits are liable to shatter otherwise.

An alternative design would be to use a low cost rotary tool, which already has the bearings, collet and spindle motor. However this approach would likely be too bulky, and many rotary tools do not have sufficiently low runout for use with fine bits.

8.3 Prior art

Many CNC mills use a collet to hold the bits. This is tightened around the bit, holding it in place with friction. The problem with this application is that using a collet system would require custom metal work, something not necessarily available to the envisioned end user.

The iModela cnc ([5]) is an example of an inexpensive desktop CNC mill. This uses a small ball bearing for the shaft, with custom machined shafts. The milling bits fit inside the shaft, and are secured with a single grub screw. This seems to work, though it would need a very carefully machined shaft so that axial misalignment did not shear the end of the milling bit, particularly with the very fine tools in use for this project.

8.4 Chosen design

A novel design was tested for the spindle. Instead of securing the bit in a collet surrounded by a ball bearing, the bit would surrounded by a number of ball bearings, sprung to provide a strong clamping force against the bit. The rotational coupling was achieved with a thick elastic O-ring, which allowed for the motor to be misaligned whilst still transferring rotation to the bit. In testing this performed excellently, and was capable of routing fine tracks without measureable runout.

8.5 Improvements

The spindle was redesigned to be stronger and neater. See figure (12) for a rendering. The new design encloses the moving parts more for safety.

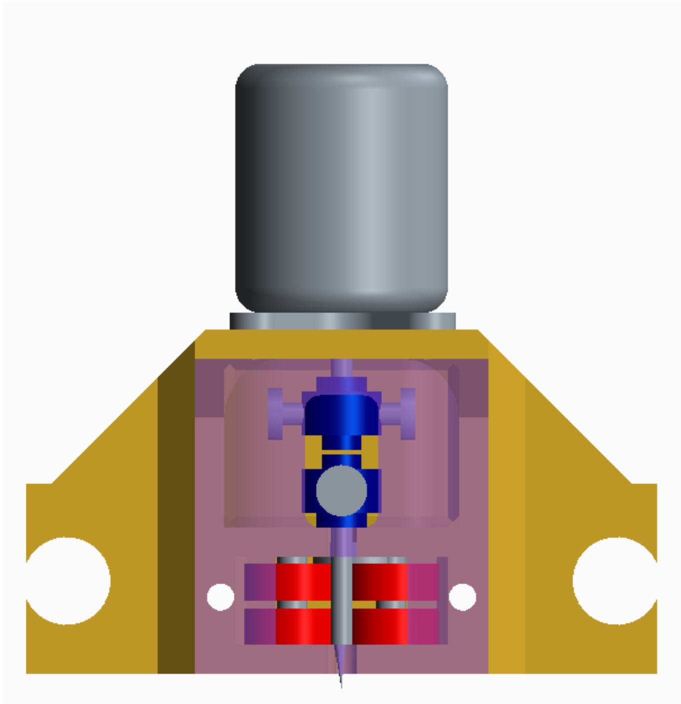


Figure 12: Rendering of improved spindle design

8.6 Milling

There are a number of uses for a machine capable of light milling:

- Manufacturing PCBs with arbitrary shapes. For example, It is possible to build connectors from PCBs that lie on the circuit board if they can be milled to the correct shape. For example, USB connectors. This reduces part cost.
- Improving the finish of 3d printed components. In cases where the rough edges of 3d printed parts are not satisfactory, a milling pass over the piece can increase the flatness of edges.

The spindle on the machine is quite capable of light milling, with some limitations:

- Milling bits only protrude from the spindle by approximately 8mm (depending on the bit). This means that this is the deepest cut the machine is capable of making.
- The axes do not have the rigidity required for milling tough materials. Plastics and wood are possible, while metals are not.
- The spindle only supports parts with a 3mm shank.
- The spindle speed is variable from 1000->40000rpm unloaded. The speed controller attempts to maintain the rotational speed setpoint, however it prioritises correct motor commutation. Therefore at higher speeds under heavy loads care must be taken that the spindle speed does not drop significantly. In practice this is unlikely to be a problem, as the motor has a maximum power of 250W and has significant moment of inertia.

As this functionality is not a central focus of this project, only a simple demonstration of milling ability was carried out. The spur gear used in the gear extruder is cut from 6mm acrylic:

8.6.1 Gear CAD

The Creo design was exported as a .DXF file. The closed source program Cambam ([6]) was used to generate GCODE for milling due to the author's familiarity with the software, but many open-source alternatives are available (eg dxf2gcode [8]).

8.6.2 Milling Settings

The following settings were used to mill the green acrylic gear (Figure 14):

Setting	Value	Comments
Sheet material	3mm green acrylic	
Cutting bit	1mm diameter endmill	Cambam estimated maximum endmill size to be 1.4mm diameter.
Depth increment	1mm	Chosen to produce part quickly at the expense of edge quality.
Cut feedrate	70mm/min	Very slow feedrate chosen for initial tests.
Plunge feedrate	70mm/min	Slow feedrate to avoid chips clogging.

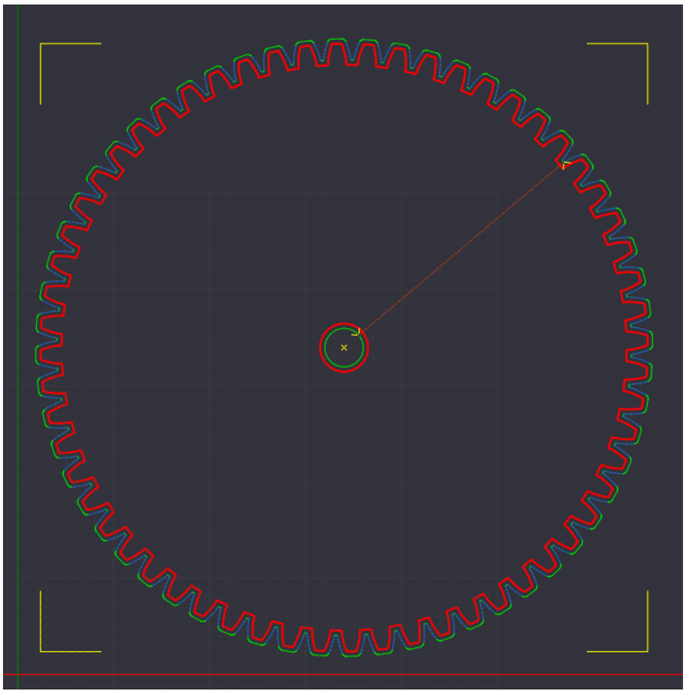


Figure 13: Gear outline shown in red. Toolpath shown in green/blue.

9 Hot air tool

Once the board has been pasted and the parts deposited, the solder paste needs to be heated up so that it will melt and form good connections. There are a few possibilities for this process:

- Oven: a small oven can be used to heat the entire board (see: ToasterSMD [10], Figure 17). This appears to work well, however the oven should not be used for food after that, since the components of solder paste are toxic. As a result, it can be an expensive method since it requires dedicated, expensive hardware.

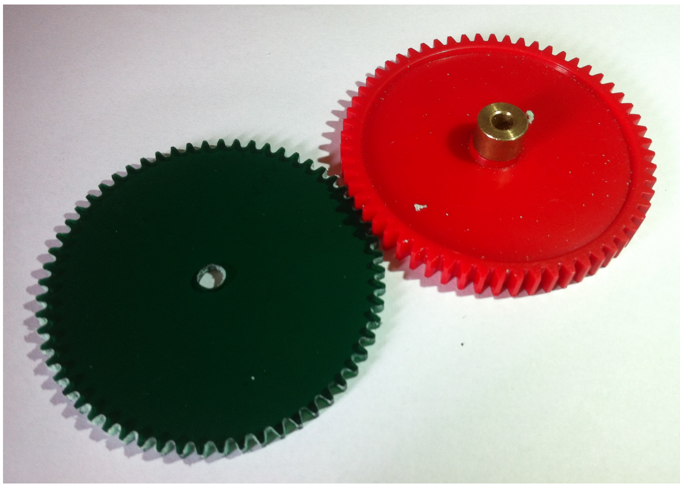


Figure 14: Milled gear (left) compared to original gear (right).

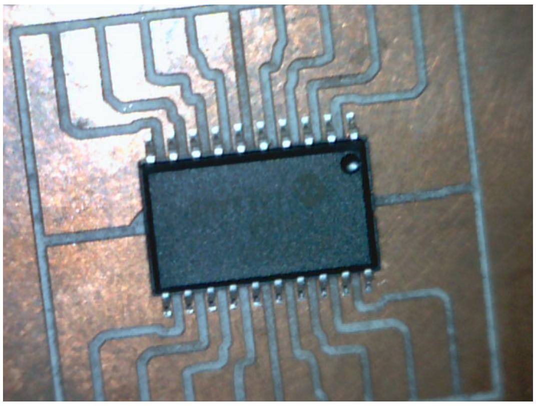


Figure 15: Isolation routed SOIC board.

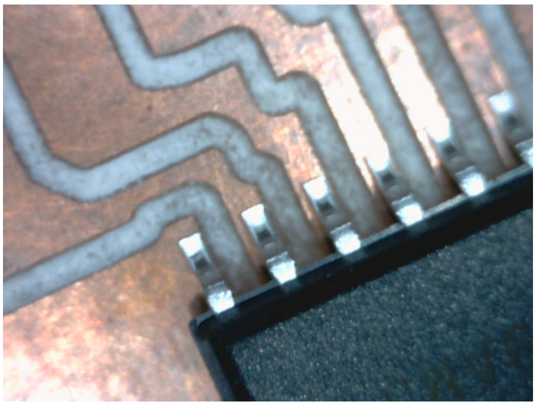


Figure 16: Closeup of breakout board.

- Hand-held hot air gun: A small hot air gun can be used to gradually melt the solder for the components (Figure 18). There is a requirement for the hot air gun that it emits sufficiently hot air at a sufficiently low velocity that the solder will melt, but also that the devices will not shift under the force. This does require the most human interaction (since it is less likely that the flow of hot air will cover the board uniformly, but it is not a particularly demanding process.

- Hot-plate. A small hot-plate can be used to heat the board from underneath (see Hotplate tutorial [3], 19). An advantage compared to the oven method is that a piece of thermally conductive metal can be used between a hotplate and the board, so that the hot plate will not become food-unsafe.

The choice of method to heat the solder paste may well depend on the end-user - one solution is not clearly superior to the others. During the course of this project the hand-held hot air gun will be used most often due to equipment availability. It would however be useful if this aspect of board manufacture could also be automated. A hot air tool has been designed that travels over the board, reflowing all of the connections.

Figure 17: SMD board being reflowed in a toasted oven.

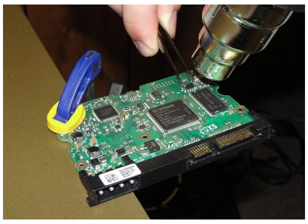


Figure 18: SMD component being soldered with a hand-held hot air gun.

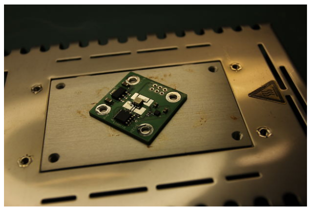


Figure 19: SMD board being soldered with a hotplate.

9.1 Requirements

A hot air tool used for soldering has the following requirements:

- Ability to heat solder paste up to 240 °C
- Slow, steady airflow
- Controllable temperature (i.e. not open-loop)

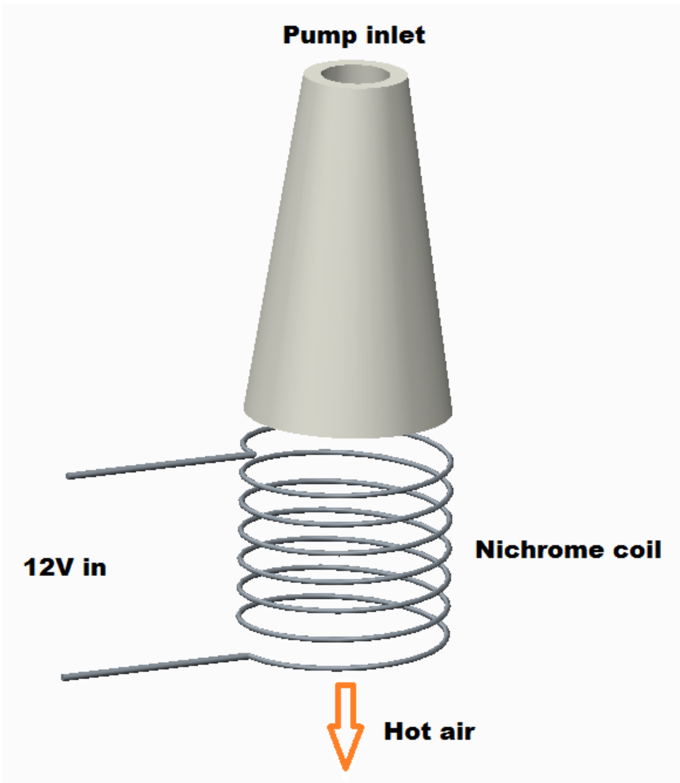


Figure 20: Simplified diagram of hot air tool.

9.2 Calculations

Refer to Appendix D for the heater calculations.

10 Machine Handbook

As the machine operation is complex, a handbook has been produced that detail the steps needed to operate it. This forms a separate document which is attached to this report.

11 Host computer software

Software will have to be written to translate board cad data to gcode. Some of this already exists in the public domain, some will have to be made as part of this project.

Isolation milling and drilling: pcb2gcode (https://github.com/festlv/pcb2gcode-metric). Takes gerber designs and generates isolation routing/drilling GCODE. It is also capable of voronai isolation (functionally equivalent routing that cuts the least amount of material). The only configuration will be to generate a millproject file with the correct cutting depths.

Solder paste placement: No freely available software.

Device placement: No freely available software.

The solder paste placement should be relatively straightforward. The program will simply need to find the locations of the pads, calculate their area, and then apply a suitable amount of paste to them. For ICs, the paste will not need to be separately applied to each pin (due to the high surface tension of molten solder), so some method of detecting ICs on the board layout will be needed.

The code to place devices will be the most challenging. However, provided there is a sensible area in which to hand-deposit the devices, it should not be too hard to calculate how to transport them to the required location.

In my opinion, the most challenging part of writing the conversion code will simply be to extract the necessary data from the cad files. Once this has been obtained, the GCODE should be easy to create.

11.1 Improvements to pcb2gcode

pcb2gcode was forked on github, and the improvements from this project are available at [2]. To demonstrate the effects of the improvements, pictures of a SOIC-20 breakout board being processed are included (Figures 21 , 22 , 23).

11.1.1 Adding solder paste support

Solder paste information is contained in the GTP (Gerber top paste) file. It was decided to extend pcb2gcode to read this file, and output GCODE suitable for solder paste extrusion.

The GTP file is in the same RS-274X extended Gerber format as the other files that pcb2gcode can already read. For simplicity, the existing Gerber processing was used. This is retrieved in

```
NGC_Exporter::export_layer(see ngc_exporter.cpp)
```

. The data is stored as a vector of vertices of each pad to be filled with solder paste.

An assumption is currently made that all pads are rectangular, and that the paste can be suitably applied in a single place. This may need to be improved in the future if large areas of solder paste are required.

Process of determining where to deposit paste:

1. Vertices of pad stored in array.

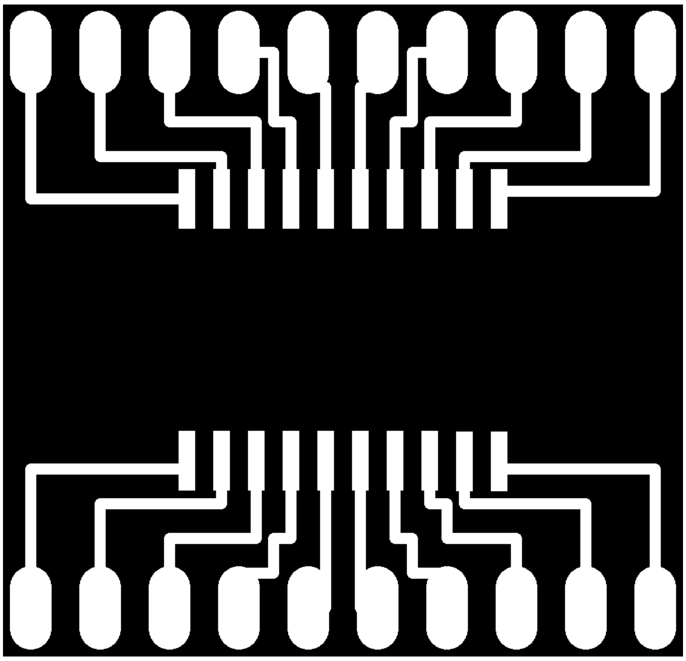


Figure 21: Input to pcb2gcode, showing the copper traces in white.

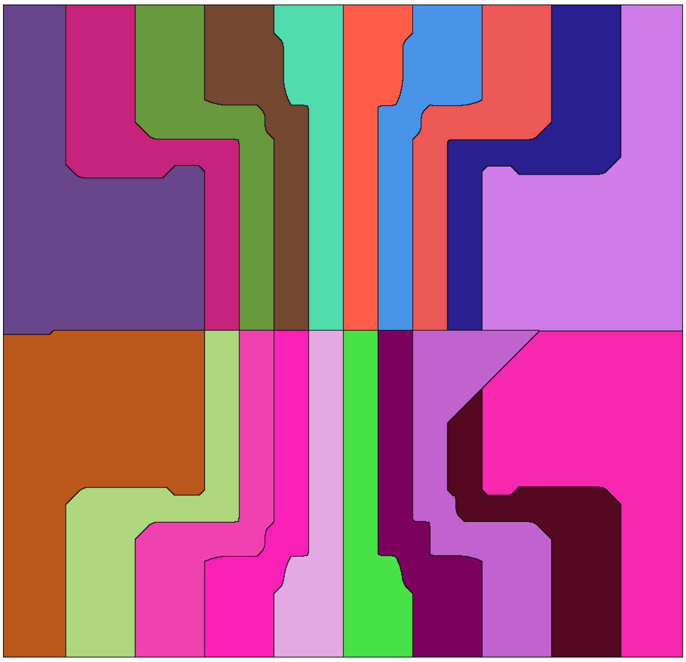


Figure 22: Output from pcb2gcode, showing isolation routing pattern (milling to be run between coloured sections).

2. Area of pad calculated.
3. Center of pad calculated.
4. From the area of the pad, the diameter of the needle, and the required paste thickness, the extrusion distance is calculated.
5. GCODE is written to extrude paste at the center of the pad.

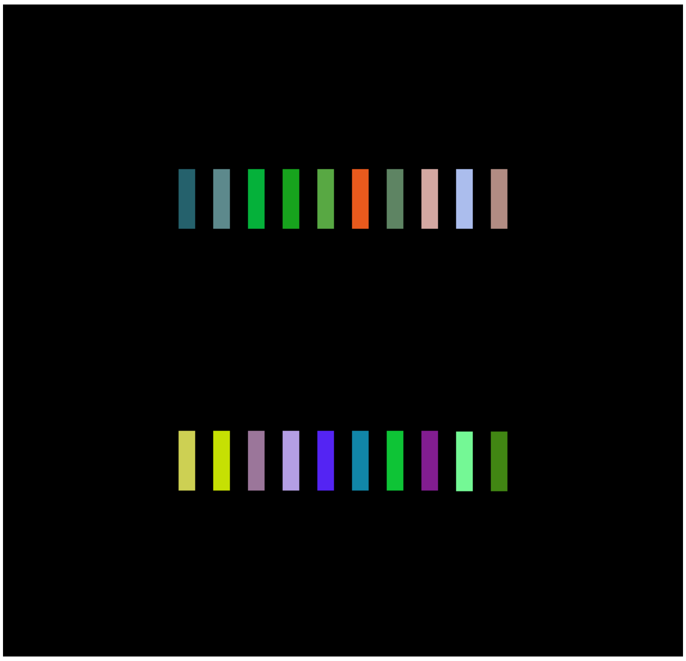


Figure 23: Solder paste output from pcb2gcode, showing where paste is to be deposited.

Retraction:

The belt on the paste extruder is slightly springy. This results in the extruder continuing to extrude a small amount of solder paste after the motor has stopped moving. This has an adverse effect on board quality. To counter this, the extruder should be retracted slightly to remove all pressure from the syringe. The distance to retract as well as speed of retraction should be experimentally determined for optimum results. Retraction support has been added to the solder paste output of pcb2gcode.

12 Tools

In the course of this project, access to a small mill, a 3d printer and a lathe was essential. It may be worth considering that a lot of people/schools have access to a laser cutter and perhaps mill/lathe, so if possible all custom components could be 3d printed or laser cut.

13 Managing devices prior to placement

There will need to be a way to place the devices in a known position, so they can then be picked up and placed in their final locations. Commercial pick and place machines take the parts directly from the reels they are supplied in. While this is optimal for that application in that it is a very fast way to operate, the cost of reel dispensers is prohibitive for this machine. Instead, the parts will have to be roughly placed by hand. There are a number of ways this could be done:

- Embossed outlines of all necessary device footprints.
- Rectangular embossed outline, devices to be located in one corner.

It was decided that the latter would be simplest to implement. A rectangular hole will be milled into the bed. Each part can then be manually placed inside, then pushed to one corner.

A consistent system will be needed to ensure correct orientation. The following set of rules is to be used:

- ICs will be placed with pin 1 touching the datum corner.
- Devices will always be placed with the longest side (if applicable) touching the long side of the rectangle.
- Devices such as MOSFETs will be placed with the corner between the longest side and side with most pins touching the datum corner.
- Polar 2-lead devices with always have the anode nearest the datum corner.

Given the current component to be placed, and the device footprint, it will be trivial to find the coordinates of the centre of the part. Some consideration will have to be given as to how to determine the centre of mass of parts which are not symmetrical in two directions (e.g. MOSFETs).

14 Vacuum Placing

14.1 Prototype

To test how effectively small parts could be placed with a vacuum, a simple test setup was fabricated:

- 12V diaphragm air pump (£8.89 - eBay)
- 6mm OD/4mm ID silicone tube (£2.69/m - eBay)
- various needles (need more info here)
- MOSFET
- 330Ω resistor

Figure 24: Pump with hose and needle.

The pump draws 200mA at 12V, which is easily controlled with a MOSFET. Using a 1.8mm ID needle, SOIC-20 parts were picked up, and had significant friction against the needle - essential to avoid slipping or rotating. It was important to place the needle as centrally as possible on the part to avoid generating any imbalance that allowed the part to fall.

14.2 Conclusions

A vacuum needle solution is workable, and a good choice for this project. Things that need to be considered for use in the final project:

- The pump takes a certain time to generate a vacuum because of the relatively large internal volume of the pump and silicon tube. It will be important to measure this, and add a margin of error so that full suction is applied to each device before any movement takes place. Similarly, when turning off the pump, the pressure must return to atmospheric before the device can be considered to have been placed successfully. In testing, the pump took less than a second to leak sufficiently for this to occur, but if another pump were to be used, it may be necessary to place a tiny hole in the air line so that pressure drops quickly.
- It is important to place the needle very near to the centre of mass of the device. The majority of devices will be symmetrical, which will make this easier.
- The needle must be able to be raised and lowered to the correct heights. It may be possible to push the needle down with a weak spring, with a micro servo that raises it - this would remove any need to know the exact height of each device. This will need to be tested.

Figure 25: SMD resistor being picked up.

15 Operating Speed

For the machine to be practical, there are some limitations on the maximum time that a board can take to manufacture. This is particularly true of the device placement stage, as this requires human intervention. The total time can be divided into a few main sections:

- X/Y travel.

- X/Y cutting.
- Z movement.
- Picking/placing components.
- Hot air reflow.

The X/Y travel will take the most time in total, so the estimated times should be calculated and optimised if possible. Assuming a travel speed of 10mm/s, the total time is simply total distance (mm) / 10.

16 Measuring Accuracy

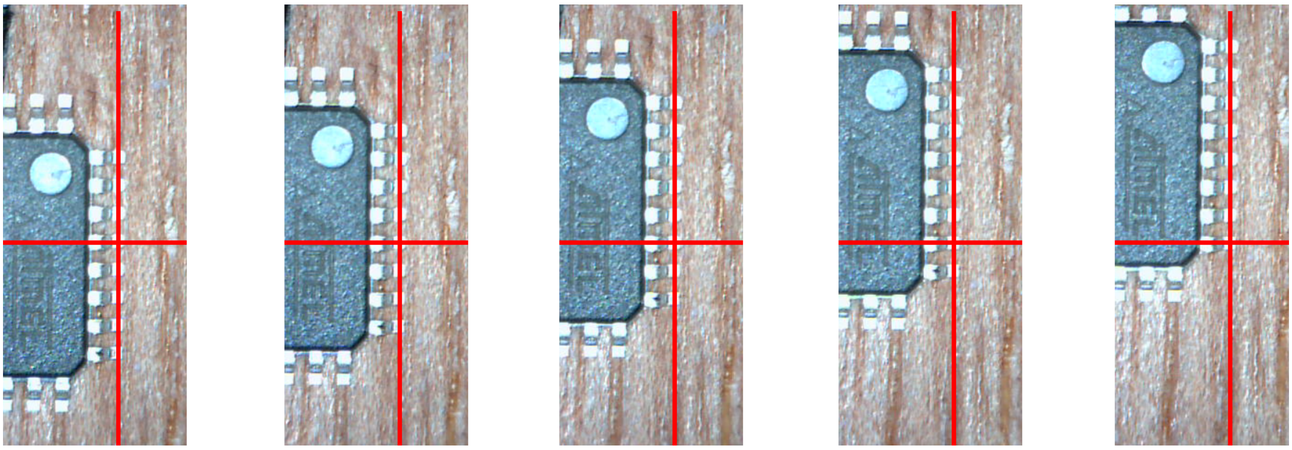


Figure 26: Visual alignment test using one side of a TQFP-32 Atmega328.

It is important to measure the accuracy of the axes to ensure that they can reach the specified +-0.05mm. This will need to be measured at all points on the bed (as the accuracy will worsen when the bed is central due to increased rod deflection.). Of particular concern will be the deflection under the load of the isolation routing bit. The solder paste and device placement will not involve any significant force. The drilling will generate a force, but the tolerance required of a 1mm hole is not nearly as demanding as that of the isolation routing cutter. The exact forces exerted by the routing bit have yet to be measured, but are estimated to be 5-10N. This means that the anti-backlash nut will need to exert the same or greater force to ensure that the board does not move. This in turn places extra resistance on the motor movement.

16.1 Testing methodology

In order to characterise the accuracy of the machine, it is important to have a consistent measuring regime. A dial gauge is used to measure relative positions of the axes. See Figure 27 for the experimental setup. It is important to ensure the worst-case error is acceptable. The causes of error and their effects are:

Effects are shown with reference to Figure 28, noting that the X and Y referred to in the figure do not necessarily represent the machine axes.

Figure 27: Measurement setup (This is a placeholder image).

- Z displacement: It is possible the surface of the bed will move in the vertical direction, something that should not happen ideally. This will occur if the surface over which the idler bearing runs is not flat, or due to manufacturing tolerances in the linear slides.
- Rotation: The drawer slides do not eliminate rotation completely, and this error dominates. Rotation is relative to the small ball bearing array which slides along the drawer slides, so this source of error depends on the absolute slide position. Figure 28 shows the largest error possible, with the ball bearings far away from the location of maximum error, where rotation effects dominate.
- Motion parallel to the axis direction. This error is mostly caused by flex in the stepper motor mounts, the motor->threaded rod couplers and the anti-backlash nuts.
- Motion perpendicular to the axis direction. This error is caused by small amounts of slop in the drawer slides.

As it is only important to measure the worst-case error, the point shown in purple in Figure **??**errornotation) is the only point which will be measured.

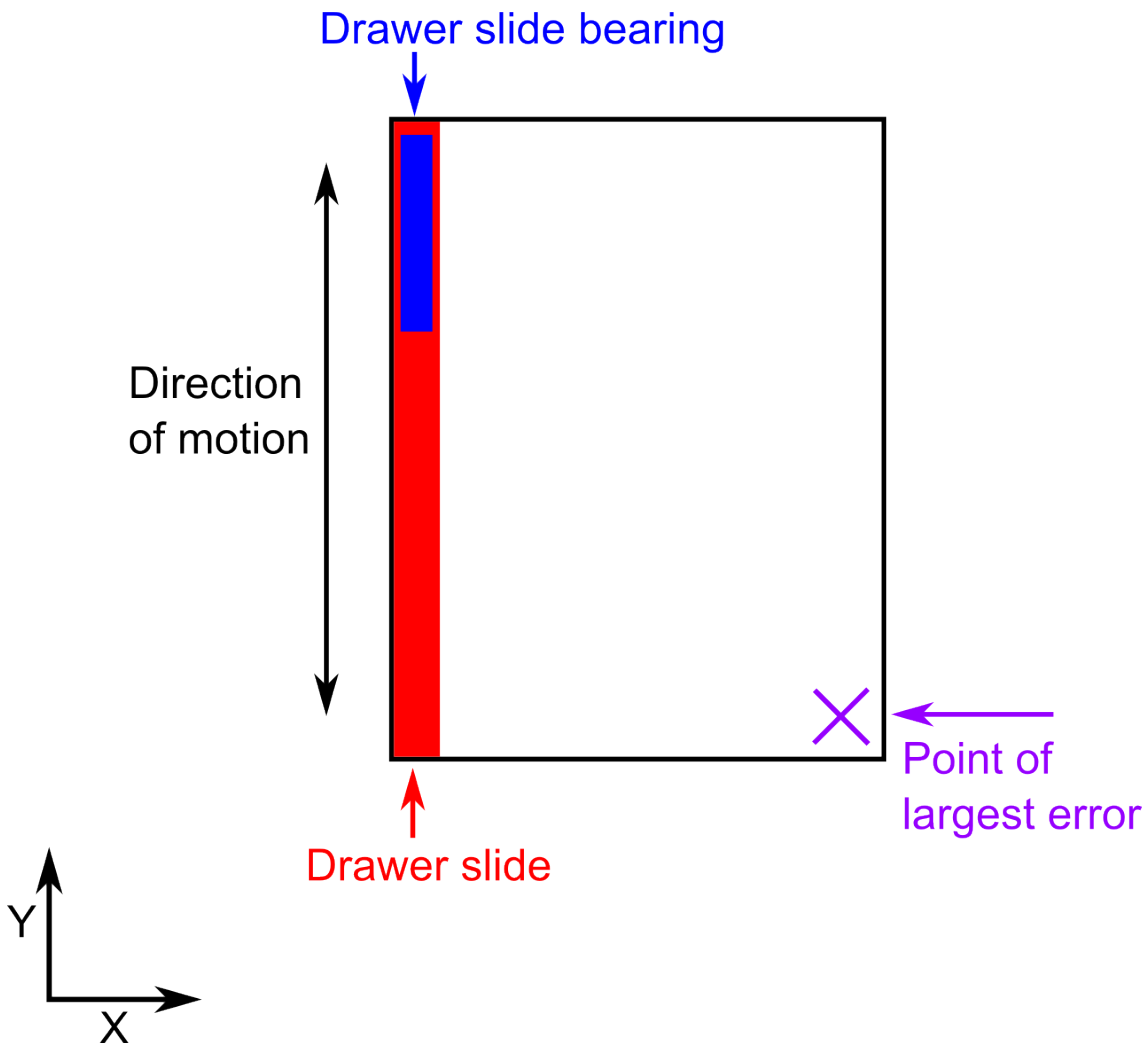
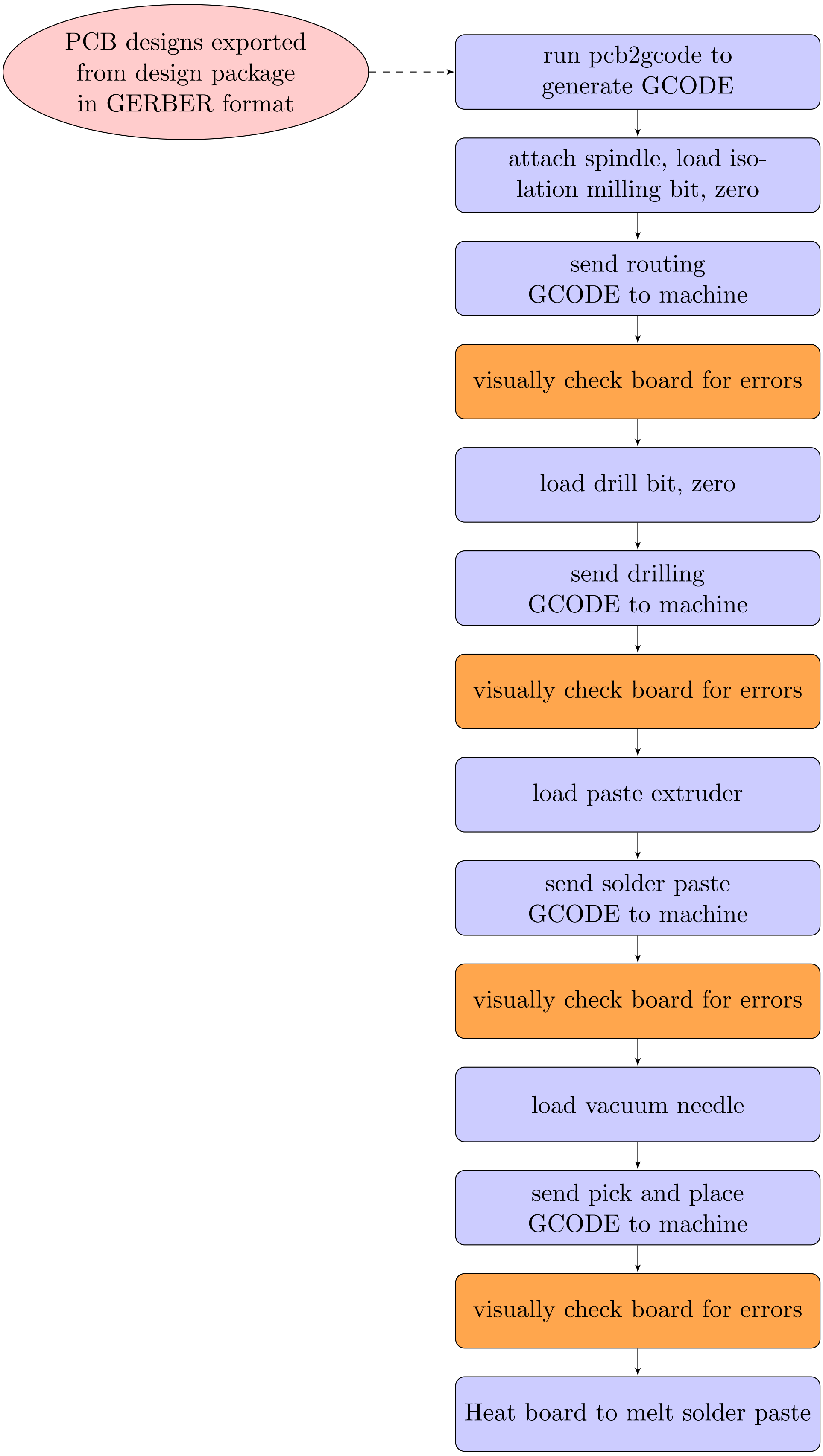


Figure 28: Measurement setup, showing error notation.

17 Operating Steps



On the host pc, custom software will have to be written to translate the design into a GCODE file to be sent to the machine. Usefully however, the machine's firmware will not need to be written or modified significantly.

18 User interface

The machine has two main user interfaces: the interface used for the machine in standalone operation (see 5.8), and that used when connected to a host computer. As the standalone operation has already been described, this section will only detail the host computer interface.

18.1 Pronterface

Pronterface was chosen as the user interface as it is a mature GCODE visualiser and sender.

INSERT PICTURE

Once the correct serial port and baud rate have been selected, pronterface will connect to the machine. GCODE can be loaded using the **Load file** button. It is rendered to the center of the window (however some operations are not shown correctly). The program is primarily intended for operating 3d printers, so to send the GCODE to the machine the **Print** button must be used.

19 Zeroing the axes

When the machine is used it is necessary to "zero" the axes to a known datum point on the board. This is particularly important when swapping tools, as there may be a small amount of slop in the tool mounts which will vary between tools.

19.1 X and Y axes

In order to zero the machine in the X and Y directions, an "L-shaped" piece of metal has been added to the bed (insert pictures/diagrams). All of the tools are metal, and therefore conductive. The axes are first manually moved to be near-zero. The tips of the tools are all electrically connected to an input pin on the AVR, with a weak (10K) pullup to 5V. The "L-shaped" piece of metal is connected to ground. When instructed, the firmware slowly moves the X or Y axis in the negative direction until a contact is made, and the input goes LOW. At this point, the exact position of the tool is known. For convenience it is important that the location of the centre of the tool is known, this can simply be calculated as the diameters of all of the tools are known and fixed.

19.2 Z axis

Slightly different mechanisms are used for the zeroing of the Z axis when used for different tools:

19.2.1 Vacuum needle

A small microswitch is part of the vacuum needle assembly. It is triggered when the needle is touching either the bed or the top of a component. As a result, zeroing can be carried out simply by moving the tool downwards until it is triggered.

19.2.2 All other tools

The other tools are zeroing electrically using the same mechanism as for the X and Y axes. The tools are lowered until contact is made between the PCB and the tool. A temporary contact must be made from ground to the copper pcb, this is achieved with an alligator clip.

20 Future Work

This section contains envisioned improvements to the machine that were not implemented due to time constraints.

20.1 Double-sided boards

It is common to use double sided PCBs in order to achieve more compact circuits. The machine will not currently have the ability to process such boards, although this would be possible in a later version.

20.1.1 Challenges

There are a number of challenges associated with working with dual sided boards:

1. Clamping the PCB to the bed of the machine is more complex with the second side of a double sided board. The components that have already been populated will not allow the board to sit flat and level.

2. The PCB material is a moderately good conductor of heat. When the second side of the board is heated to melt the solder paste, there is a danger that components on the bottom will be loosened and fall off.

3. Both sides of the board must be precisely aligned so that any through-holes or vias line up.

4. Any vias that are required will add considerable complexity. Commercially they are manufactured in a multi-stage chemical process which is not feasible for the hobbyist.

20.1.2 Envisioned solutions

1. As isolation routing places the greatest stresses on the board mounts, it is unlikely that the reverse side of the board can be manufactured after the first side has been populated with components. Therefore the isolation routing should be carried out on both sides first. Any holes through the board should also be drilled, and cutting the board out also (leaving tabs to hold it in place). When this has been done, the rest of the manufacturing process can be continued with less secure mounting from the board edges.

2. Melting solder on one side of the board while leaving the other side solid will require active cooling of the bottom of the board. Commercially components are often glued so that all the solder paste can be reflowed in one process, but the application of adhesive adds a complex extra process to the machine which should be avoided.

3. Alignment may be achieved by the use of a small USB microscope. The user would be able to see an image of the board, and select datum points (for example the center of a pad). Once the board was flipped, the same datum points could be located. From this,

a transformation matrix can simply be computed and applied to the coordinates of the GCODE.

4. A reasonable substitute for vias is to drill a hole through the board, place a short piece of wire into it and solder both sides to the copper. It is likely that the only practical way to achieve this is manually once the board has been manufactured. The ability to perform light milling operations with the machine could be used to mill the ends of the wire much closer to the surface of the PCB than could be achieved with wire cutters. In this way it is likely that thermal vias under components could be achieved (Vias used for the purpose of heatsinking are placed under the thermal tab of an SMD component to conduct heat to copper on the reverse side of the board).

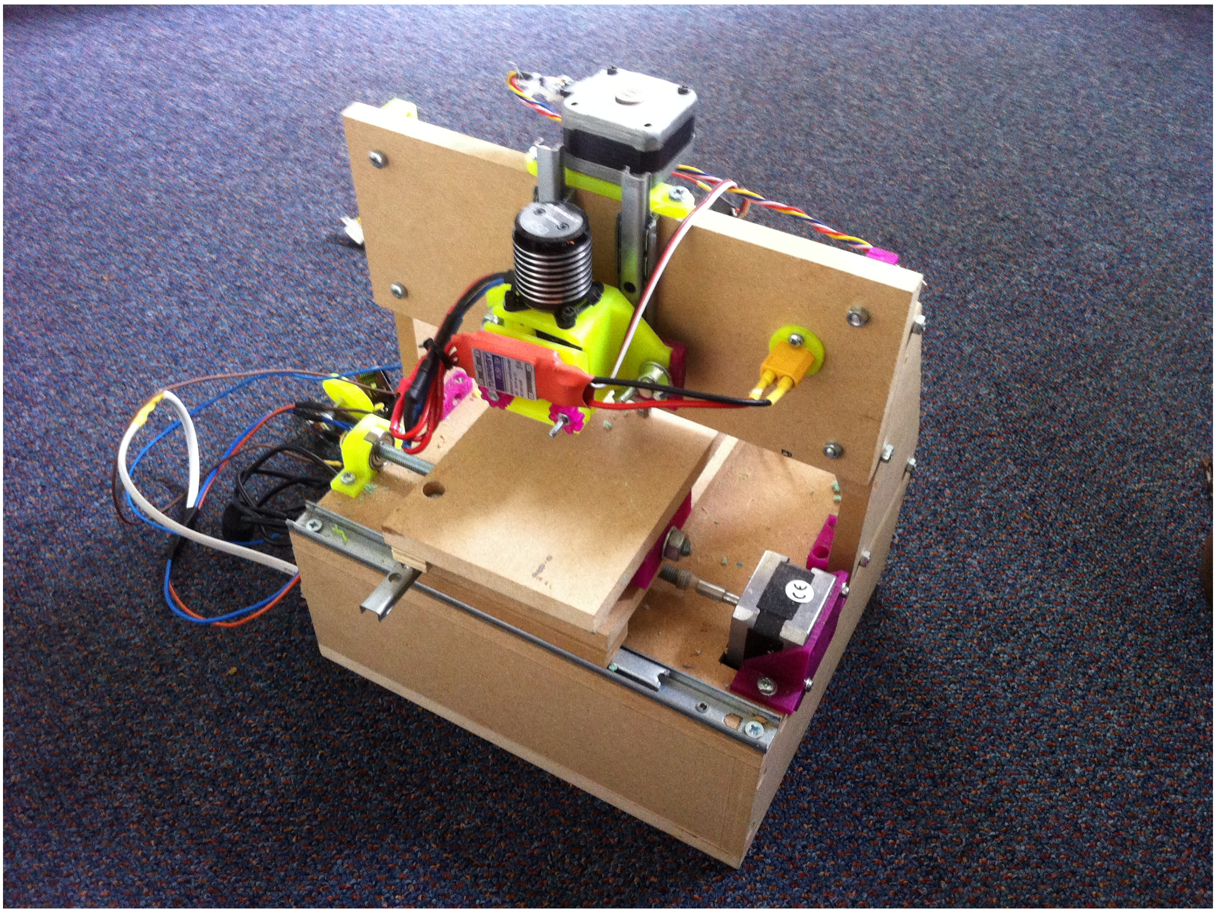


Figure 29: Finished wooden version of the machine.

21 Conclusions

As this has been a complex project, the conclusions are separated into a number of sections:

21.1 Machine Performance

The greatest challenge has been to ensure that the machine's operation is reliable. All machine subsystems operate correctly, but for convenient operation user interaction must be minimised.

21.1.1 Machine axes

Obtaining the requires $\pm$ 0.05mm accuracy has been mostly achieved. It has proved hard to eliminate the occasional $\pm$ 0.07mm error. While this is not ideal, the accuracy appears sufficient for the parts that have been tested.

During the planning stage, linear bearings were considered as a possible axis slide. These were measured to have a smaller error than the drawer slides, but the extra cost was not felt to be justified. The error inherent in the drawer slides however suggest it may have been better to use the linear bearings. The machine design is however conducive to replacing the slides with bearings with little redesign.

21.1.2 Spindle

The novel spindle design has proven to perform excellently. It is greatly superior to the bearing design used in inexpensive rotary tools (eg the Dremel) that are sometimes used in CNC projects. In the course of the project, the limitation of the shaft size needing to be 3mm diameter has not proved to be a hindrance. While outside the scope of this project, it would be interesting to evaluate the relative performance of the design against the preloaded single-bearing spindles of desktop CNC mills.

21.1.3 Isolation Routing

The machine has proved as capable at PCB milling as the reference CNC mill used for comparison (A converted Proxxon MF-70 CNC mill). The inexpensive brushless motor had sufficient power to cut through the copper at high speed. In a production version of the machine, it is likely that the cost could be reduced by using a less powerful motor and speed controller.

21.1.4 Pick and place

The pick and place tool has been shown to operate satisfactorily. One improvement that is suggested would be to introduce a small rubber shroud for the vacuum nozzle to ensure a tighter seal and hence stronger force. This should reduce the movement of the part relative to the needle.

21.1.5 Hot air tool

The hot air tool has not proved to operate satisfactorily. Various attempted designs have suffered from overheating of the nichrome wire, or insufficient heat transfer to the air. This was however an extended project goal, and it is entirely possible to use a commercial hot air tool by hand to reflow the parts. It is likely that a future improvement would be to replace the hand made hot air tool with a commercial product.

21.2 Project Management

A recurring theme during the project has been that while all of the machine aims were attainable, it took more time than was expected to improve the processes to be user-friendly and reliable. In hindsight it may have been more efficient to use an existing three-axis system for testing, so that the tool heads could be used and tested at an earlier stage.

21.3 Evaluation of method of PCB manufacture/population

21.3.1 PCB manufacture

There are limitations inherent in milling PCBs (compared to the more usual chemical etching) - the process is incredibly sensitive to positional errors. It proved difficult to reliably manufacture traces of less than 0.03mm wide. This was also true of the reference PCB mill. Small flakes of milled copper have a tendency to bridge traces, and are difficult to reliably remove.

While the machine is capable of manufacturing boards with undemanding features, it has not been shown to be better than a wet chemical etch. This process is relatively inexpensive and does not have the problem of copper flakes (which are chemically dissolved).

21.3.2 Solder paste extrusion

Solder paste extrusion must be assessed by comparison with the only two practical alternatives:

1. Solder paste mask. This is a more reliable method and does not suffer from the start/stop extrusion dribble. It is not however feasible to produce a mask for small runs of boards due to the cost.

2. Hand application. This requires a steady hand which may be a problem, for example in a classroom or learning environment.

The machine performed acceptably during solder paste extrusion. The main problem that was seen during paste extrusion was small amounts of ”dribble” when the pressure was not reduced sufficiently quickly at the end of extrusion. The use of solder paste is relatively forgiving in this aspect however, and this was primarily an aesthetic problem with traces of solder on the finished board.

21.4 Final Conclusions

The machine's operation has been shown to be generally acceptable. However the process requires more human intervention and testing than was envisioned. In its current state the machine shows limited advantages over more manual methods of PCB manufacture, but given more development time and testing this is expected to improve dramatically.

The novel spindle design has far surpassed the design expectations, and it would be worthwhile to characterise its performance in more demanding applications, however this is outside the scope of the current project. The steps taken to produce GCODE (e.g. running pcb2gcode) will always need a PC, but once the GCODE has been produced it is possible for the machine to read this from connected storage.

Appendices

A Servo tester

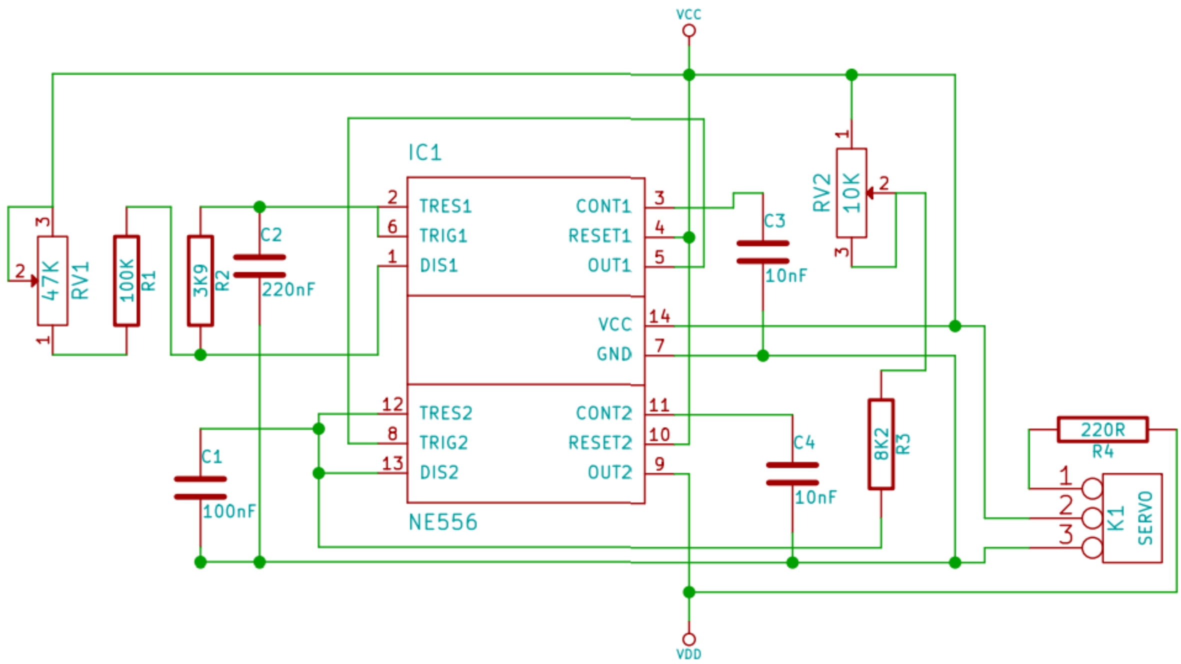


Figure 30: Servo tester schematic.

The ESC used to control the brushless spindle motor is controlled with standard ”servo” control. This consists of variable length pulses sent approximately 20ms apart. The length of pulses corresponds to the position of the servo, or in this case the desired operation speed of the brushless motor. The pulses are specified to be 1-2ms, which correspond to the minimum and maximum desired speeds.

During testing it because apparent that it would be useful to have manual control of spindle speed. A simple circuit was designed to provide the necessary pulses (See Figure 30).

The circuit works by using one 555 timer to generate a square wave of approximately 50Hz. This triggers a second 555 timer, which operates in monostable mode, with a pulse length that is dependent on the position of a potentiometer. The 50Hz wave satisfies the condition of pulses that should be sent approximately every 20ms, and the pulses are varied between 1-2ms. The speed controllers contain a 5V linear regulator which is connected to one pin of the servo connection. This powers the 555 timers so that no other power supply is required for operation. The circuit performed as expected and has been useful in project development.

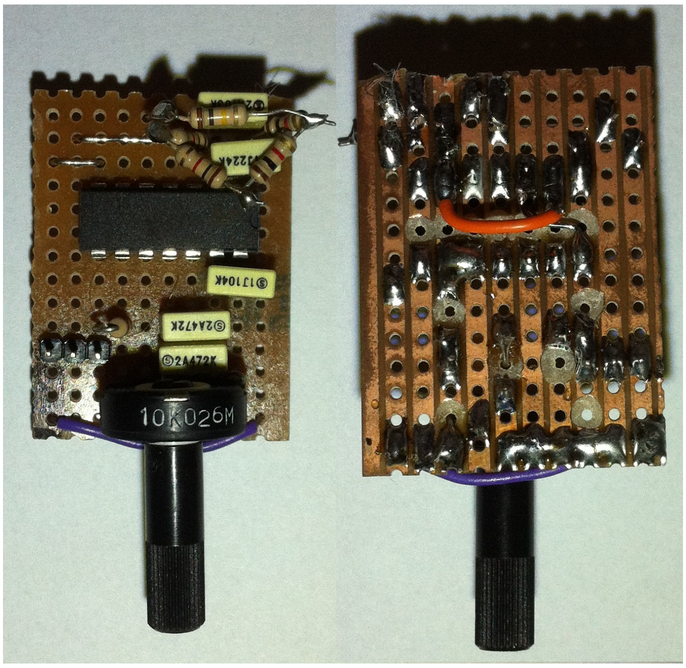


Figure 31: Servo tester board (front and back).

B Calliper serial output modification

During the testing of accuracy it was necessary to take a large number of positional measurements in all directions. To speed up this process, an inexpensive pair of callipers was modified to output a serial data stream which could be logged against the position according to the firmware. Instructions located here: [1].

The positional data can be read from four traces on the calliper internal PCB. For convenience, the +V, ground, data and clock lines were brought out on to four 2.54" pin headers. As the callipers are powered by a 1.5V battery, NPN transistors are used to amplify the signals to 5V. An ATTINY2313 decodes the data stream and converts it to a serial stream, sent via a USB-serial cable to the PC.

Figure 32: Modified callipers (Data port highlighted in red).

C SMD component types

A large variety of SMD types and sizes are available. A selection is described here (from [9])

C.1 Two-terminal passive components (mostly resistors and capacitors)

Package types are given a four digit code, which describes the length and width in mm.

Example:
0603 is a 0.6 x 0.3mm rectangular package.

C.2 SOT: Small-Outline Transistor

Despite the name, these devices are not limited to single transistors, other types (eg linear regulators) are common in the same packages.

Example:
SOT-223 is a 6.7x3.7x1.8mm body, three terminals plus a heat transfer pad.

C.3 SOIC: Small-Outline Integrated Circuit

Dual-in-line devices with 8 or more gull-wing lead form pins, spacing of 1.27mm

D Heater Calculations

Assumptions:

- Specific heat capacity of dry air is not significantly affected by temperature for estimates of power required.
- 50% of energy into heater is used to heat hot air stream (other radiative losses)
- Flow rate of air pump = 18 cm^3 / s
- Specific heat capacity of dry air = 1.006 J/gK. This is assumed to not change significantly with temperature (acceptable to ± 10%).
- Room temperature of 20^0C, temperature required is 240^0C. Delta T = 220K
- Density of air = 0.001225 g / cm^3

Power required to heat air (ideal) = (18*0.001225)*220*1.006 = 4.88W Power required to heat air (assuming losses) = 4.88*2 = 9.76W

For simplicity, it is assumed that the heater requires 10W of input power.

The design is based on a small coil of nichrome resistance wire as a heating element. Calculations must be made to find the size of wire required:

Size of coil (Chosen to be similar in diameter to the air tube with 6mm OD/4mm ID)

Diameter = 5mm Length = 12mm Wire spacing = 2mm Angle of wire = $\tan^{-1}$ (2/(5* pi)) = 7.26 ^{0} Turns of wire = 12/2 + 1 = 7 Length of wire = 7 * sqrt ((5 * pi $)^2$ * 2^2) = 110.84mm

Resistivity of nichrome at room temperature = 1.0 * 10E-6 ohm m Power supply voltage = 12V Current draw = 10 / 12 = 0.83 A Resistance required = 12^2 / 10 = 14.4 ohms Cross sectional area of wire = 1.0 * 10^{-6} * 110.84 * 10^{-3} / 14.4 = 7.70 * 10^{-9} m^2 = 7.70 * 10^{-3} mm^2 Diameter of wire = sqrt(7.70E-3/pi) * 2 = 0.099mm

This is the minimum diameter of wire. Next the maximum diameter of wire is calculated, assuming a maximum current draw of 10A.

Resistance required = 12 / 10 = 1.2 ohms Cross sectional area of wire = 7.70 * 10^{-3} * (14.4/1.2) = 0.092 mm ^{2} Diameter of wire = sqrt(0.092/pi) * 2 = 0.34mm

Therefore wire diameter is between 0.099mm and 0.34mm The most commonly available suitable wire is sold as 30 SWG, equivalent to 0.315mm diameter.

Resistance of 110.84mm of 0.315mm diameter nichrome wire:

$$\text{Resistance} = \frac{\rho L}{A} = \frac{1.0 * 10^{-}6 * 110.84 * 10^{-}3}{\pi * (\frac{0.315*10^{-}3}{2})^2} = 1.42\Omega$$

$$\text{Maximum current draw} = \frac{\text{V}}{\text{R}} = \frac{12}{1.42} = 8.45\text{A}$$

E Part List

Auto-generated from "Part List.xlsx" using [11].

Part	Number required	Total Cost	Supplier
base_board	1	-	-
topboard	1	-	-
bed	1	-	-
zsupport	2	-	-
zpanel	1	-	-
base_back	1	-	-
base_sides	2	-	-
base_bottom	1	-	-
electronics_tray	1	-	-
drawer slides	6	8	
m8 threaded rods	3		
m8 brass nuts	6		
anti-backlash springs	3		
electronics board	1	30	
nema17 motors	3	18	Zapp Automation
Vacuum tube	1	1	
Turnigy plush 30A	1	8	Hobbyking
Turnigy 450 3800KV	1	8.56	Hobbyking
prop savers	2	1.84	Hobbyking
m4 nuts and bolts	40		
12v 30A psu	1	23.58	
nuttrap	3		
motorholder	3		
pasteextruder	1		
needleholder	1		
zholder	1		
rightangle	4		

F Risk assessment retrospective

The risks highlighted in the risk assessment were as follows:

- Electrical: only low voltages used (¡=12V)
- Hazardous substances: solder paste toxic, ensure hand washing after use
- Use of workshop equipment.
- Mechanical: minor pinching hazards
- Other: hot air for soldering, heated 3d printer extruders.

The risk assessment accurately reflected the hazards that were encountered during the project. All safety precautions were carried out as specified, and no accidents occurred at any point. If starting the project again, no changes would be made to the risk assessment.

Glossary

ESC Electronic speed control. A device which controls a high-current motor from a logic input.. 19

PCB Printed circuit board. A circuit board made of an insulating sheet covered in conductive traces to which components are soldered. FR-4 is the common grade of glass-reinforced epoxy used in most PCBs, and as such is the only material covered by this project.. 5

SMD Surface mount device. A component with no through-hole leads which is designed to be soldered to a PCB. In some cases the components are glued to the board for strength.. 5

References

[1] David Cook. *Interfacing with Digital Calipers*. URL: http://www.robotroom.com/Caliper-Digital-Data-Port.html (visited on 05/12/2014).

[2] James Glanville. *JamesGlanville / pcb2gcode-metric*. URL: https://github.com/JamesGlanville/pcb2gcode-metric (visited on 05/12/2014).

[3] Hobbytronics. *Hotplate Surface Mount SMD Soldering Tutorial*. URL: http://www.hobbytronics.co.uk/hotplate-smd-soldering (visited on 05/12/2014).

[4] Richard Horne. *Universal Paste Extruder for 3d printers*. URL: http://www.thingiverse.com/thing:20733 (visited on 05/22/2014).

[5] TechSoft UK Limited. *iModela, the smart compact milling / engraving machine*. URL: http://www.imodela.co.uk/ (visited on 05/12/2014).

[6] HexRay Ltd. *Cambam*. URL: http://www.cambam.info/ (visited on 05/26/2014).

[7] Jean P. Nicolle. *SMD (Surface Mount Device) / SMT (Surface Mount Technology)*. URL: http://www.fpga4fun.com/SMD.html (visited on 05/12/2014).

[8] various. *dxf2gcode: A program to convert a dxf drawing into G-Code*. URL: https://code.google.com/p/dxf2gcode/ (visited on 05/26/2014).

[9] various. *Surface-mount technology*. URL: http://en.wikipedia.org/wiki/Surface-mount_technology (visited on 05/12/2014).

[10] Tom Walsh. *ToasterSMD*. URL: http://openhardware.net/Misc_Stuff/ToasterSMD/ (visited on 05/12/2014).

[11] Eric Wood. *excel =>LaTeX*. URL: http://ericwood.org/excel2latex/ (visited on 05/22/2014).

[12] Eric Zalm. *Marlin 3d Printer Firmware*. URL: https://github.com/ErikZalm/Marlin (visited on 05/26/2014).