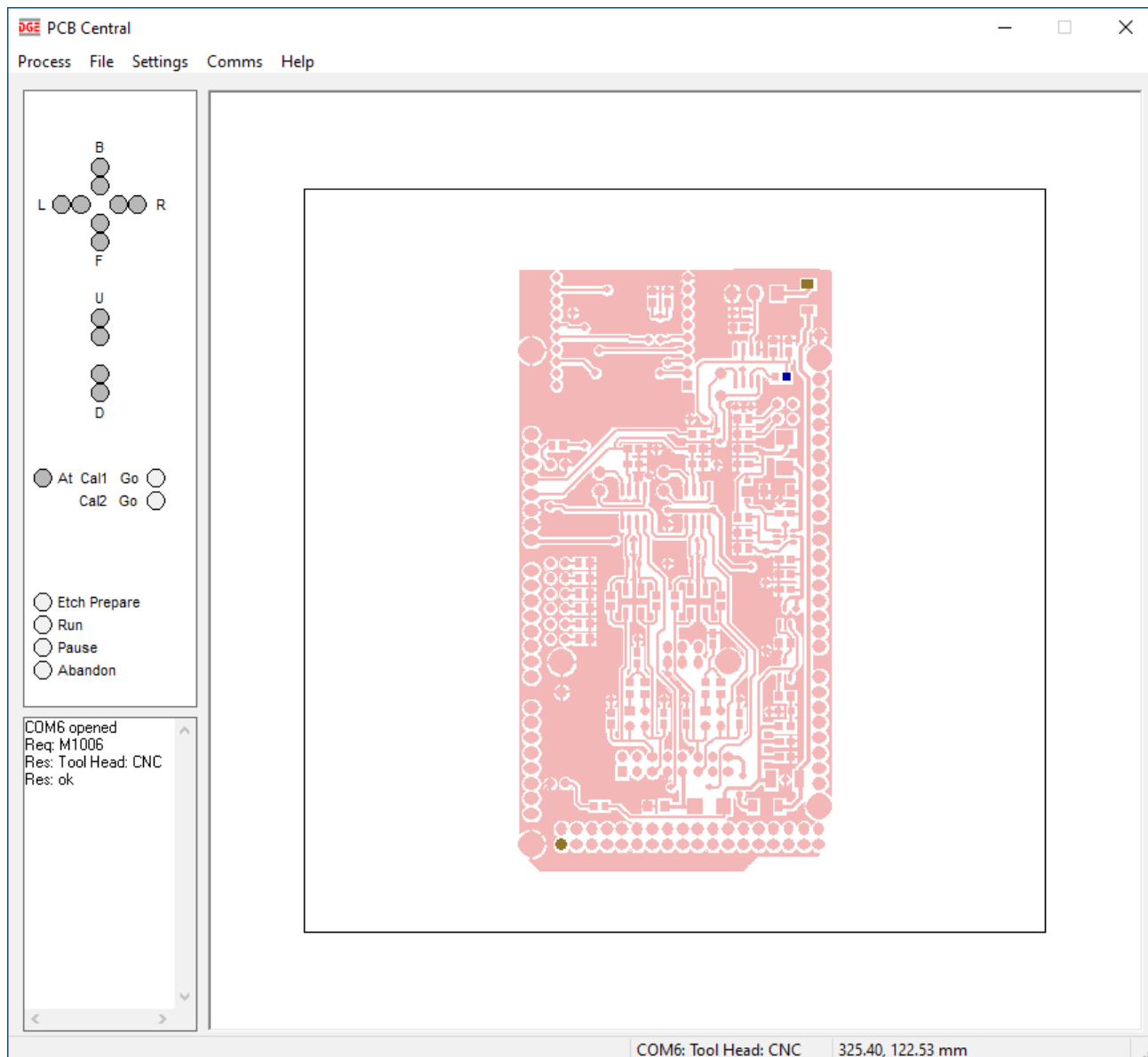


## PCB Central User Guide



### Introduction

**PCB Central** provides PCB related services of solder paste dispensing, stencil cutting, PCB drilling and to a limited extent, etch resistant ink printing and SMT assembly assistance.

**PCB Central** takes in PCB manufacturing files and outputs a **PCB Central** process file. The user will add information to the process file to accomplish a result. From a process file **PCB Central** generates G code statements to interact with the CNC machine.

Each of **PCB Central**'s services has a user interface, the set of controls presented to the user accords with which service is selected from the process menu.

## Legalese

**PCB Central's** fitness for purpose is entirely user determined. Promise of performance is not offered nor implied. Non-desirous consequences of utilising **PCB Central** is at the user's risk.

The user is to be made aware the CNC machine's warranty may be infringed when using **PCB Central**.

The user should always be near the CNC machine's on/off switch. While ever there is machine movement or machine spin, the user must always maintain attention and caution.

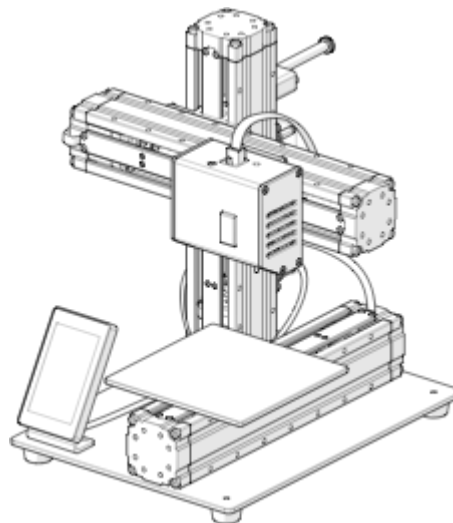
The user should know that **PCB Central** only generates and executes G code statements to move a tool. The capability of the tool will govern results.

**PCB Central** is a free program - there are no restrictions on usage (other than perhaps, letting the author know how it goes).

## Background

**PCB Central** started out as a utility to apply solder paste dispensing using generic 3D printers.

That is, until the author invested in a Snapmaker 3 in 1 machine (Snapmaker Original). The option to mount Laser modules and CNC modules extended what could be possible. The Laser module for stencil cutting perhaps? CNC module for PCB drilling? Why not?



Why not have the mounting of custom tools to deliver other services? Imagination ignited.

This is why **PCB Central** has become somewhat Snapmaker centric. This is not to say other machines, moved by Marlin firmware, could not be employed.

References to **Snapmaker** in this document infer the **Original** model.

### **About Snapmaker And Tool Heads**

The Snapmaker controller firmware is able to detect the presence or absence of a tool head. The absence of a tool head has Snapmaker unresponsive to incoming G code statements.

The type of tool that is present is determined by measurements taken on the module's RJ45 connector pins. The measurements distinguish between printer/carver/laser modules.

The solder dispense service and the assembly assist service require a custom tool which must present itself to Snapmaker as a CNC (carver) tool head. Non special electronics is necessary to achieve this, but the user is cautioned that Snapmaker's warranty could be contravened if a non Snapmaker tool is attached.

### **Dry Run Testing**

With Snapmaker's carver (CNC) module it is possible to 'dry run' **PCB Central's** services. Users could mount a soft plastic 'needle' in the CNC's chuck and go through the procedure of PCB alignment, generating G code statements and executing a service.

As the service is exercised the CNC head will travel over the service movements to give the user a review of how **PCB Central** operates.

For those able to do so, the mounting of a pen or pencil would allow the tracing of CNC movements for solder pasting by print, stencil cutting and etch printing.

### **About Snapmaker And The X Arm Issue**

The Snapmaker forum reveals users experiencing inconsistency in the Z distance between the X arm and the bed. There is difference in Z distance between the left side and the right side with the distance being wider on the right side.

**PCB Central's** services may be affected by this difference. For this reason it is recommended that the PCB layout be 'portrait' such that the X width is the smaller dimension so as to minimize the X travel.

Note: While Snapmaker software has a bed levelling facility, it applies only when 3D printing. In CNC/Laser mode, no such facility is provided. To determine the amount of Z level difference, **PCB Central** has a bed level checking utility.

## Bed Level Check

When connected to the CNC machine, **Menu->Process->Bed Level Check** may be invoked. The user is asked to jog around the bed's perimeter and record readings. The steps are...

Z jog up to a level that is clear of the bed.

XY jog to a point at the SW corner of the bed. A precise XY position is not necessary. Z jog down to a level where the tool tip touches the bed surface. The SnapMaker manual refers to feeler gauges or a sheet of paper as measuring assists.

The keyboard '1' is then pressed to have **PCB Central** register this as an origin point. Zero co-ordinates will be reported in the SW display.

Z jog up to be clear of the bed and XY jog to a NW point. Z jog down for surface test and press '2' to have the CNC's position recorded.

Repeat for NE and SE corners, pressing the '3' and '4' keys.

Bed Level Check

0.00, 110.00, -0.10	110.00, 110.00, -0.40
2 NW	3 NE
1 SW	4 SE
0.00, 0.00, 0.00	110.00, 0.00, -0.30

XY jog to SW, Z jog to bed, hit the '1' key.  
XY jog to NW, Z jog to bed, hit the '2' key.  
XY jog to NE, Z jog to bed, hit the '3' key.  
XY jog to SW, Z jog to bed, hit the '4' key.  
Use feel gauge (A4 paper) for Z jogs.  
As a key is hit the co-ordinates are reported.

OK Cancel

This example shows the NE point as 0.4mm below the SW reference point, and the SE corner as 0.3mm lower than the reference point.

## CNC Settings

Cnc Settings

Absolute Park Position

X 62.50 mm Absolute position to place CNC head after a process has been completed.

Y 62.50 mm

Z 62.50 mm Actionable only in 3D printer mode.

Relative Z Park

Z 25.00 mm Z distance to lift tool on process completion.

Boundary

X Min 0.00 mm X Max 125.00 mm

Y Min 0.00 mm Y Max 125.00 mm

Z Min 0.00 mm Z Max 125.00 mm

Co-ordinates of the CNC's boundary

Communications Timeout

32000 mSecs Period to wait for a response from the CNC.

OK Cancel

3D printers have XYZ limits which must be made known to **PCB Central** to test boundary indiscretions. The boundary settings is machine specific. For the SnapMaker this is 125mm and is set in the above dialog.

### A Caution About CNC Timeout

The timeout period is important. Marlin 3D printer firmware lacks lock/step transactions, that is, an acknowledgment is given only after a request has been accepted and performed.

Marlin accumulates requests in a buffer, acknowledging immediately. When the buffer is full acknowledgments cease and will recommence when buffer space becomes available.

The time required to execute the accumulated requests can be many seconds, too short a period will cause a timeout at **PCB Central** due to a lack of an acknowledgment. The service will then terminate.

## Comm Port Settings

Comms Setting

Port COM6

Data Bits 7 8

Baud 9600 19200 38400 56000 115200 256000

Parity None Even Odd

Stop Bits 1 2

OK Cancel

Note: Snapmaker requires a Baud of 115200, any other setting will have Snapmaker unresponsive. The setting for Data Bits, Parity and Stop Bits has no effect.

Snapmaker must be online for its Virtual Comm Port (VCP) number to be seen. See notes and end of this guide.

## PCB Manufacturing Files And Process Files

**Pcb Central** takes in Gerber, Excellon and Pick & Place files and creates process files.

The process files are...

Process	Ext	PCB EDA Input File
Paste	.pcp	Gerber paste file (.gtp, .gbp)
Stencil	.pcs	Gerber paste file (.gtp, .gbp)
Drill	.pcd	Excellon drill file (.txt)
Etch	.pce	Gerber layer file (.gtl, .gbl)
Assembly	.pca	Pick & Place file (.txt)

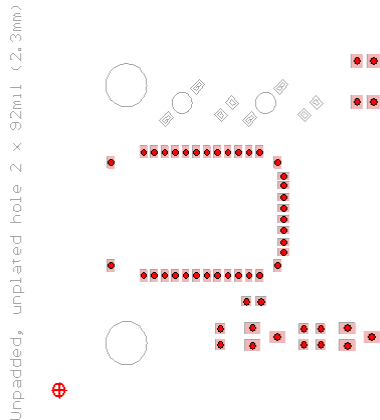
The process data is stored to file in binary form. The file includes data size and a checksum. When asked to open a process file, **PCB Central** first confirms the validity of the process file before accepting the file's data.

### About Gerber Files And Non-Orthogonal Pads

Gerber files deliver information on orthogonal PCB pads in the form of an aperture definition. The aperture definition has details of width and height for rectangular pads and the diameter of circular pads. Aperture definitions are also applied to PCB track widths.

However, if the PCB design has pads that are not orthogonal, the Gerber file represents the pad as a sequence of PCB tracks. For non-orthogonal pads, **PCB Central** cannot discern the pad nor the pad's centre point.

For these pads the user must input data to inform **PCB Central** where to execute the service. In case of dollop paste dispensing the user nominates a centre position. For paste dispensing by print, the defining outside tracks of the pad are identified.



Example of Gerber paste file with 8 non-orthogonal pads.

This example shows eight non-orthogonal pads. There are also artefacts the PCB designer used to communicate to the PCB fabricator the requirement of non-padded/non-plated holes.

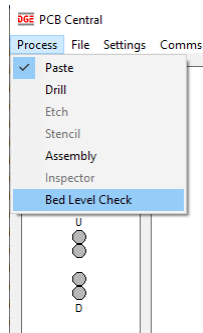
### Luer Tip Data

Solder paste is delivered through a Luer tapered syringe tip. Tapered tips help reduce dripping.

Colour	Gauge	Dia (mil)	Dia (mm)
-----	-----	-----	-----
Clear	27	8	0.20
Red	25	11	0.28
Lt Red	24	13	0.33
Blue	22	17	0.43
Pink	20	24	0.61
Green	18	35	0.89
Silver	16	48	1.22
Beige	14	63	1.60



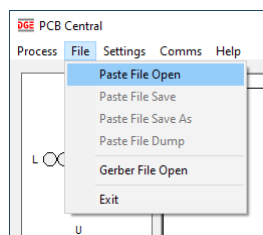
## Selecting A PCB Service



The type of PCB service to be run is selected from the Process Menu. Selection of a service has **PCB Central** aware of which EDA input file and which process output file are involved.

## Creation Of A Process File

In Menu->File the user chooses the EDA input file. For paste, stencil and etch services this will be a Gerber file, the drill service an Excellon file and for assembly assist, a pick and place file.



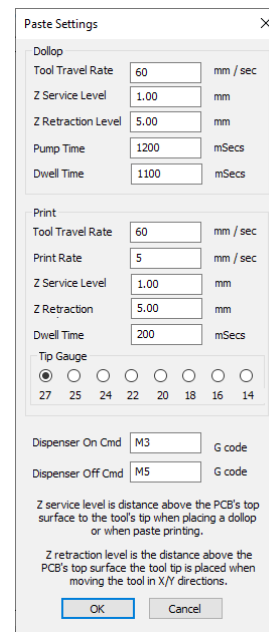
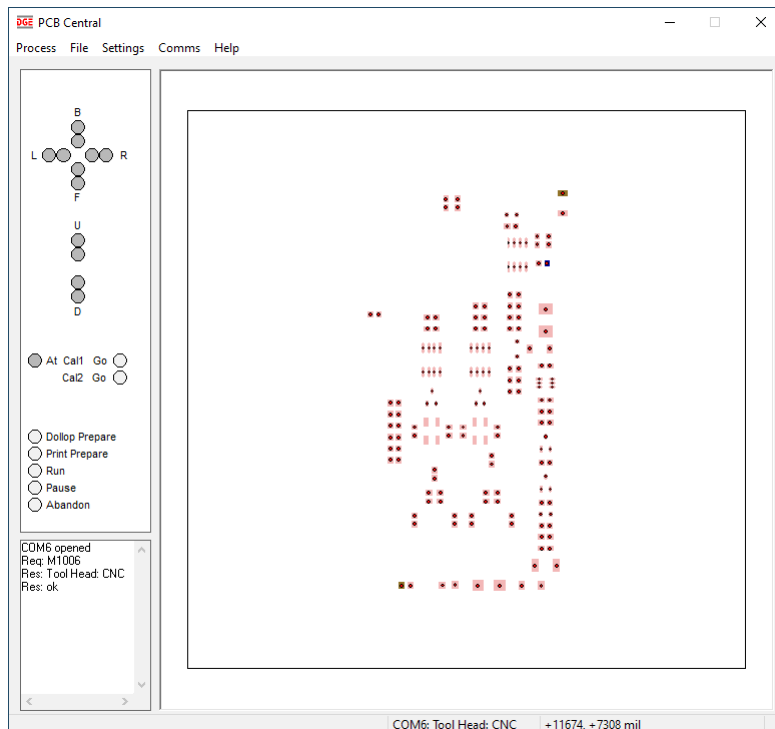
Once the input file is selected it is read in and information extracted to create the process data. To assist troubleshooting the process data may be 'dumped' to a text file.

## Solder Paste Dispensing

**PCB Central** takes in Gerber paste files (.gtp, .gbp) from which details are extracted and a 'Paste' process output file (.pcp) is created. On the first creation of a paste process file only orthogonal pads are automatically included for solder paste dispensing.

For non-orthogonal pads the user must indicate to **PCB Central** where paste is to be dispensed.

Two methods of dispensing are provided.



### Paste Dispense By Dollop

Paste by dollop places solder paste at a single point.

Left mouse clicking over an orthogonal pad will toggle that pad's inclusion/exclusion for dollop paste dispensing. With the shift key held, left mouse clicking allows arbitrary placement/removal of dollops. Typically this would be to place a dollop on non-orthogonal pads but may also be used to place multiple dollops on large pads.

### Paste Dispense By Printing

Print dispensing fills in the area of orthogonal pads with solder paste. Pads are excluded in the same manner as for dollop pasting. That is, the dollop is removed.

For non orthogonal pads the user nominates perimeter PCB tracks that enclose the print area.

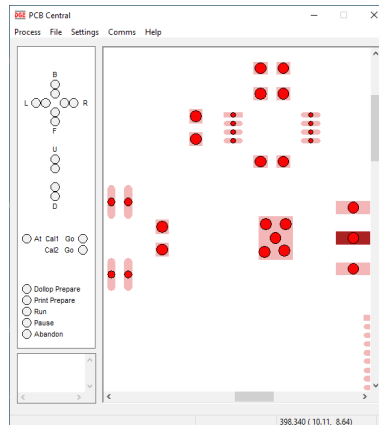
The granularity of the paste placement is governed by the dispensing tip's diameter. In the **Paste Setting Dialog** the user nominates the dispensing tip which bears in the calculation of displacement when service travelling. That is, the decreasing rectangles and decreasing circles to cover a pad is governed by the tip's diameter.



### Orthogonal Pad Selection

When mousing over an orthogonal pad the pad is highlighted. Left mouse clicking over the highlighted pad toggles an exclude/include flag on the placement of a dollop at the pad's centre. An excluded pad not pasted. When pasting by print an orthogonal pad is not pasted if the exclude flag is set.

An orthogonal pad may have one or more dollops applied. Holding down the **Shift** key when left mouse clicking places/removes arbitrary dollops.



Multiple arbitrary dollops on an orthogonal pad.

### Non-Orthogonal Pad Selection

When pasting by dollop the user places/removes an arbitrary dollop at the centre of the non-orthogonal pad by a **Shift** key + left mouse click.

To include a non-orthogonal pad in a paste by print the user holds down the **Alt** key and left clicks the PCB tracks that make up the pad's paste area.

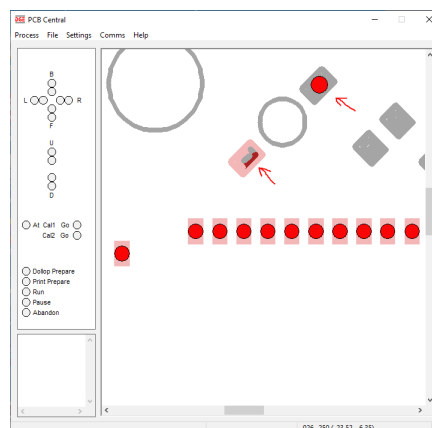


Image shows a dollop placed in the centre of a non-orthogonal pad and also shows partial coverage of PCB tracks to be included when pasting by print.

## PCB Drilling

**PCB Central** takes in an Excellon drill file and produces a 'Drill' (.pcd) process file, from this process file **PCB Central** interacts with the CNC machine.

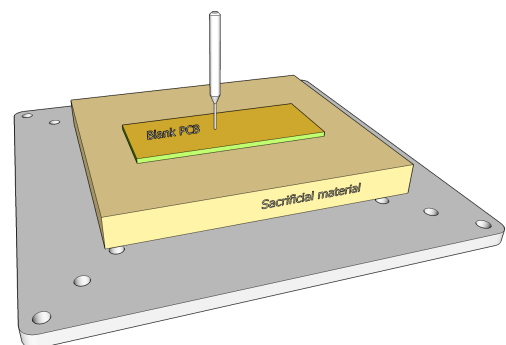
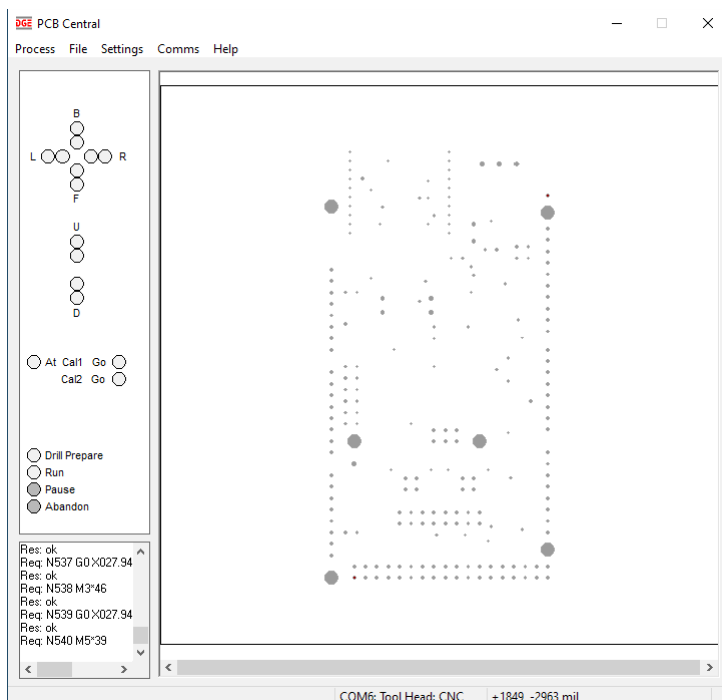
Drill holes may be excluded/included by left mouse clicking over the hole. When excluded, the hole is shown in a lighter colour.

## Calibration

As with all **PCB Central** services a calibration process must be followed. Calibration at a first point is mandatory so that **PCB Central** is aware of the PCB's top surface and the offset between PCB and CNC co-ordinates. The second calibration point would be necessary to ensure coverage.

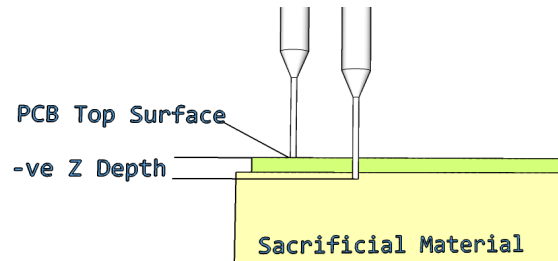
If a traversal to the second calibration point is exercised it is essential the CNC bed be level, otherwise there is a risk of the fragile drill bit snapping off when descending to the second calibration point on an uneven bed.

For that reason it may be prudent to raise the drill bit slightly before applying the first calibration point and adjusting the depth level in the **Drill Settings Dialog**.



## Drill Settings

The Z depth is given as negative number to indicate the distance below the PCB's top surface that the drill bit is to descend to.



Drill Settings	
Tool Travel Rate	30 mm/sec
Drill Feed Rate	10 mm/sec
Z Depth Level	-2.00 mm
Z Retraction Level	10.00 mm
Dwell	100 mSec
Drill On Cmd	M3 G code
Drill Off Cmd	M5 G code
Do not pause for drill change <input checked="" type="checkbox"/>	
<small>Z depth is the distance below the PCB's top surface the drill bit drops to. For 1.6mm PCBs, a nominal -2mm is suggested.</small>	
<small>Z retraction is the distance above the PCB's top surface the tool travels at.</small>	
<small>The drill job pauses to allow a drill bit change, the pause may be overridden by checking the above box.</small>	
<div>OK Cancel</div>	

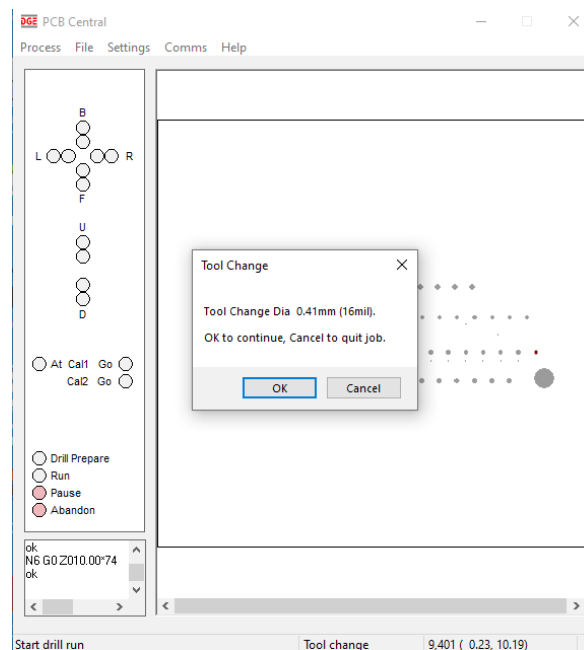
## Drill Jobbing

When preparing a drill job, statements are embedded as G code comments. These comment statements indicate to the jobbing run process when there is a change in the hole size.

The user is then prompted to affix a different drill bit. At the prompt the user may ignore the drill bit change requirement or quit the drill job.

In the **Drill Settings Dialog** the user can bypass the drill change prompt by checking the 'Do not pause for drill change'.

The user may care to change drill bits but it is suggested the same drill be used throughout the drill process. Typically this would be a drill of small diameter. Later in the PCB process, drills of larger diameter can use this smaller hole as a pilot.



To continue the process without changing the drill bit the user 'OK's the prompt.

## Etch Service

The etch service is intended to paint etch resistant ink in accordance with a Gerber top/bottom layer file (.gtl, .gbl). At the time of this document there is no satisfying method to implement this process.

The author has mounted a 0.3mm (12mil) propelling pencil to follow the job run, and while the result was sensible, the ability to trace over the track sequences, each having line widths to consider, is beyond a simple 'print run' at a fixed pen tip diameter.

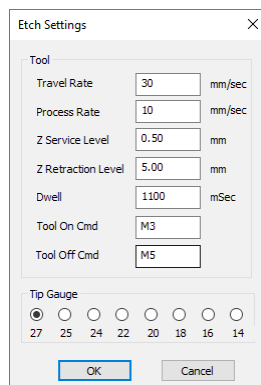
However, should the PCB copper coverage not be too fine the etch service may be of appeal.

## About Gerber Layer Files

PCB Central inputs Gerber top and bottom layer files (.gtl,.gbl) for the etch service. As with the Gerber paste file, orthogonal pads are directly known through aperture definitions while non-orthogonal pads are represented by PCB tracks. Other copper areas on the PCB are also represented by PCB tracks.



The above shows the top layer (red) and the bottom layer of an Arduino Due Shield. There are approximately 12,000 G Code statements to express non-orthogonal pads, component interconnect and ground plane coverage for the top layer and a similar number for the bottom layer.



As with the other services the Z level is the distance from the top surface of the PCB to place the pen when writing.

Users may find appeal in loading their Gerber top/bottom layer file for a visual review.

### About Circular Pads, Snapmaker And G2/G3 Issues

Snapmaker's arc/circle drawing capability is limited. Seems for a radius greater than approximately 1.0mm the circle is properly formed. As the radius is reduced different results occur. It is not known if this is Snapmaker specific or Marlin specific. The author mounted an 0.3mm propelling pencil to obtain the results of three G2 requests.

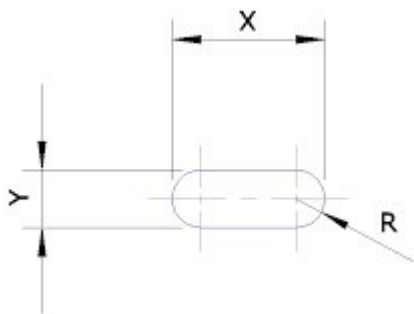


Left	G2 I3.00 Y0.00
Middle	G2 I1.00 Y0.00
Right	G2 I0.53 Y0.00

A typical PCB will have many circular pads less than 2.0mm diameter. **PCB Central's** etch service processes these pads a series of deflating octogons within the pad.

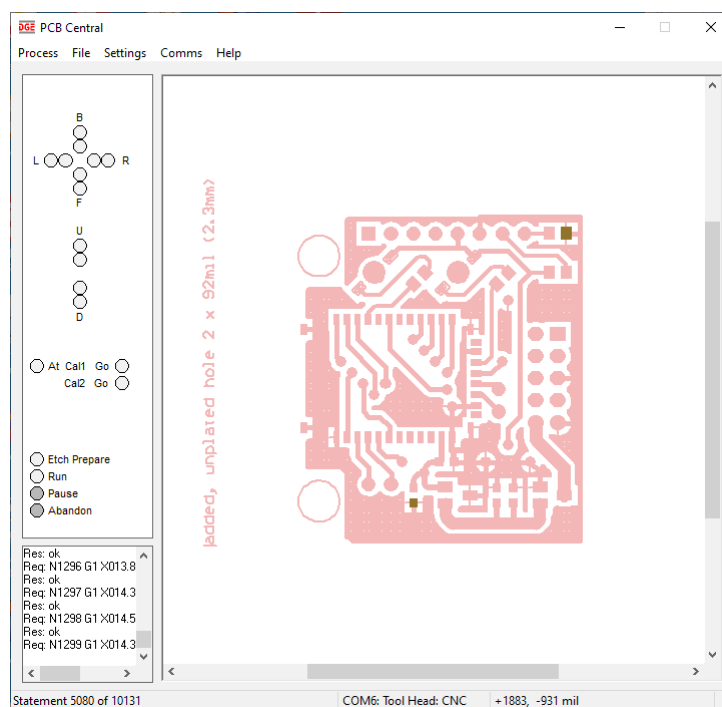
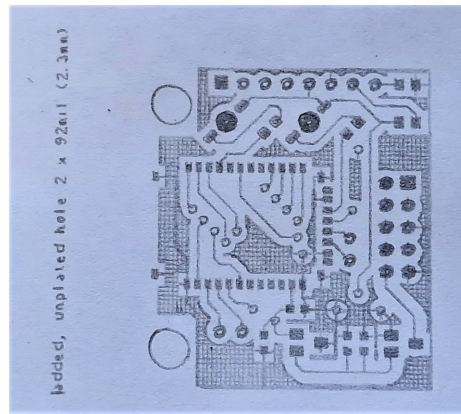
### About Obround Pads

Obround pads have parallel lines on the longer dimension and hemispheres at each side.



**PCB Central** processes the pad as a rectangle of  $( ( X - Y ) \times Y )$  with circles at each side.

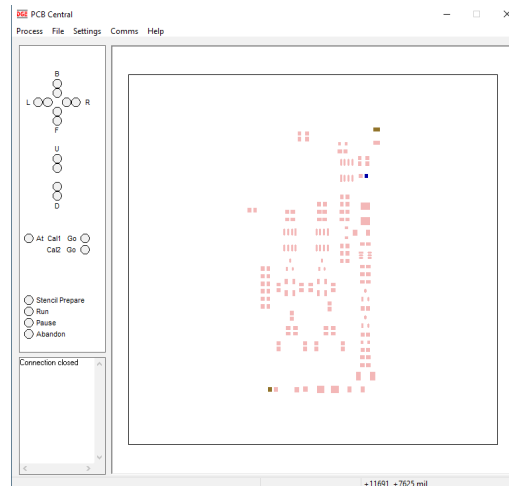
The result of tracing the pads and tracks with an 0.3mm propelling pencil is shown below. While coverage is there it can be seen the signal tracks are not as wide as expressed in the PCB document.



For this small PCB, 10,131 G Code statements were required to have the pencil follow the pads and tracks.

## Stencil Cutting

The stencil service takes Gerber paste files (.gtp, .gbp) and builds a stencil process file (.pcs).



## Calibration

The first calibration point process must be executed so that **PCB Central** is aware of the film's surface. The Z level the Laser is to be placed is user set in the **Stencil Settings Dialog** and reflects a value which best suits Laser cutting.

As the stencil film is an open surface there is no necessity for X/Y alignment. However the user may exercise the traversing between calibration points to ensure coverage.

## Laser Cutting Orthogonal Pads

The G Code produced by the stencil service has the Laser tool travel the boundary of each included orthogonal pad. A **Cut In** parameter may be set to account for the Laser dot diameter.

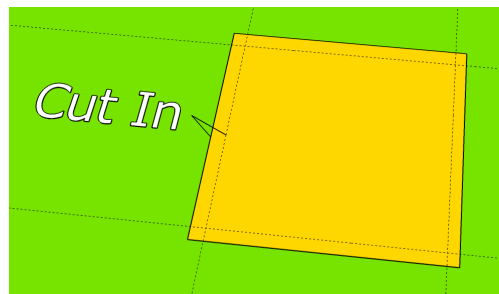
Stencil

Tool Travel Rate	30	mm/sec
Tool Process Rate	10	mm/sec
Z Service Level	1.00	mm
Z Retraction Level	5.00	mm
Cut In	0.10	mm
Dwell	100	mSec
Laser On Cmd	M3	G code
Laser Off Cmd	M5	G code

For stencil cutting the Z service and Z retraction can be the same.

The cut in parameter is the internal distance from a pad's border that the cutting tool travels. The accuracy is determined by the Laser tool's diameter.

OK Cancel



Orthogonal pads may be excluded/included in a stencil cut by left mouse clicking on the pad. Excluded pads are shown greyed out

### Laser Cutting Non-Orthogonal Pads

PCB Central has no knowledge of what constitutes a non-orthogonal pad. These pads are represented as a series of PCB tracks which are indistinguishable from other PCB tracks.

To include non-orthogonal pads in a stencil cut the user indicates to **PCB Central** those PCB tracks which make up the perimeter of the pad.

As the user mouses over tracks within a non-orthogonal pad the track is high lighted. Left mouse clicking on a high lighted track includes/excludes that track in the Laser cutting process.

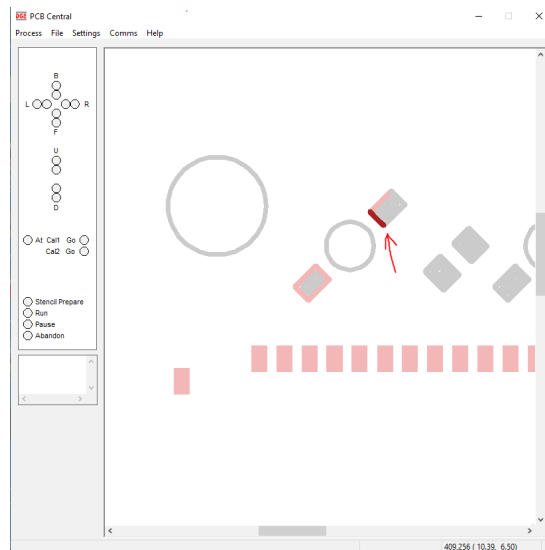


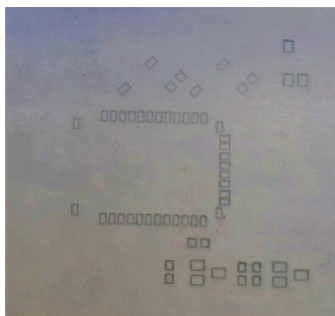
Image shows an non-orthogonal pad with its perimeter tracks selected and a pad track undergoing selection.

Note: At the creation of a stencil process file, all orthogonal pads are included and all PCB tracks are excluded.

### Caution

The user is cautioned to always wear eye protection when jobbing stencil cuts.

Note: A stencil cutting job cannot be paused/resumed, only abandoned.

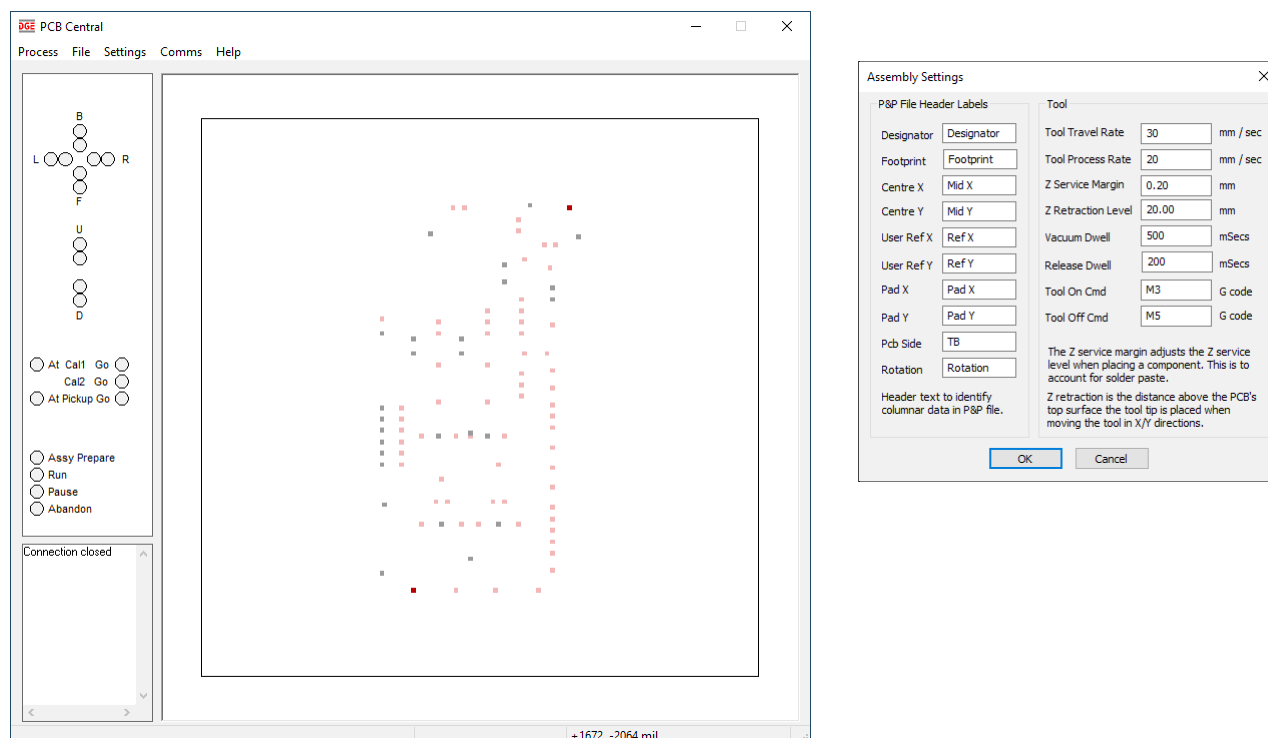


0.3mm pencil plot of a stencil cut. It can be seen both orthogonal and non-orthogonal pads are traced.



## Assembly Assist

**PCB Central's** assembly service is an assist utility. It takes in a pick and place text file (.txt) and extracts component placement details. The output is an assembly process file (.pca).

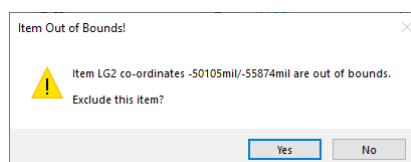


## Pick & Place EDA Files

There seems to be no consistency in how CAD programs output P&P information. The order in which the data is presented, and the labels given to the data can vary. **PCB Central** attempts to ease the transfer of data between the P&P file and the process file.

By tabling the P&P file's labels, **PCB Central** tries to determine the columnar order of information. The user can open the P&P file in a text editor to ensure spelling and capitalization matches. Note PCB Central expects the space character to separate the columns.

## Truant Components



**PCB Central** catches components having unrealistic co-ordinates, the user is notified with the option of exclusion. See end of this document.

## Reordering the P&P Components

The assembly assist service has the user place a component at a calibrated pickup point.

The order of the components is as the P&P file dictates. Since the P&P file is a text file the user may carefully edit the file to rearrange the component order. If the order is...

First by component type ie R, C, D, Q, U, L

Then by footprint

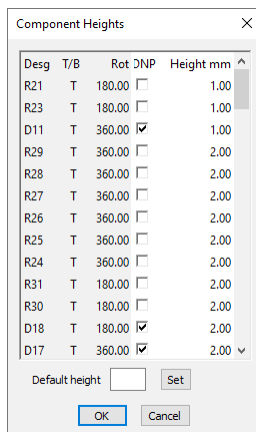
Then by component value

... then the pickup procedure is eased. Components of greater height should be placed last in the list, typically these are inductors.

## Assembly Assist Settings Dialog

Rather than placing a component at the PCB top surface where solder paste would be displaced, it would be better to place the component just above the PCB surface. The settings dialog has an '**Z Service Margin**' to allow adjustment of the level where the component is placed.

## Component Settings Dialog



The **Component Settings Dialog** allows exclusion (Do Not Place, DNP) of a component. It is flagged by checking a box. Typically these would be through hole components.

For components not excluded, **PCB Central** must know the component's height. The height has bearing on Z level calculations when picking up a component and when placing the component on the PCB.

A component's height information may be found in components' datasheet or by manually measuring the component (in mm).

## Top Side / Bottom Side Selection

Pick and Place files have placement information for components mounted on both sides of a PCB. The '\*' key on the keyboard/keypad toggles between top placement information (shown in red), and bottom placement information (shown in blue).

**Note:** **PCB Central** shows components' mid point as rectangles. This should not be confused with pads as depicted in other PCB services. .

**Note:** The display will be blank if...

There are no top side components. This is the side shown when the assembly assist process file is first opened.

The user has inadvertently struck the '\*' key to show the bottom components and there are no bottom side components.

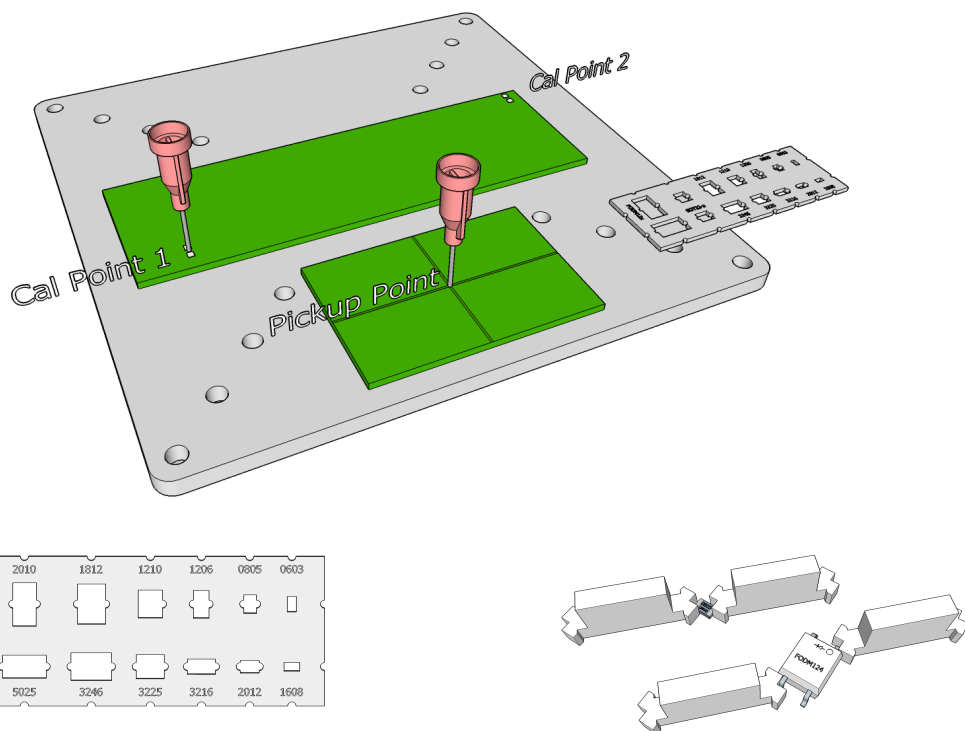
### PCB Alignment For Assembly Assist

When calibrating alignment of the PCB for assembly assist service, it is the component's centre point that is focused. If an SMT assembly session is executed immediately after a solder paste session (the PCB alignment on the CNC bed is unchanged), the calibration procedure is still necessary as the calibration points are now the centre of the nominated components.

The points chosen for calibration should not have solder paste applied. These may be pads for through hole components, fiducials or other locations.

### SMT Pick Up Point

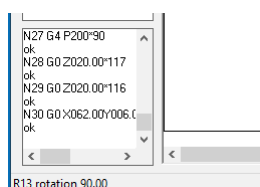
Through XYZ jogging the user indicates to **PCB Central** a point for SMT component pick up. The pick up point must reside within the CNC machine's XYZ space.



Aids when placing an SMT component at the pick up point

### Job Run

Once the PCB alignment and pick up point is established G Code can be prepared, and a job run executed.



Job runs are a sequence of...

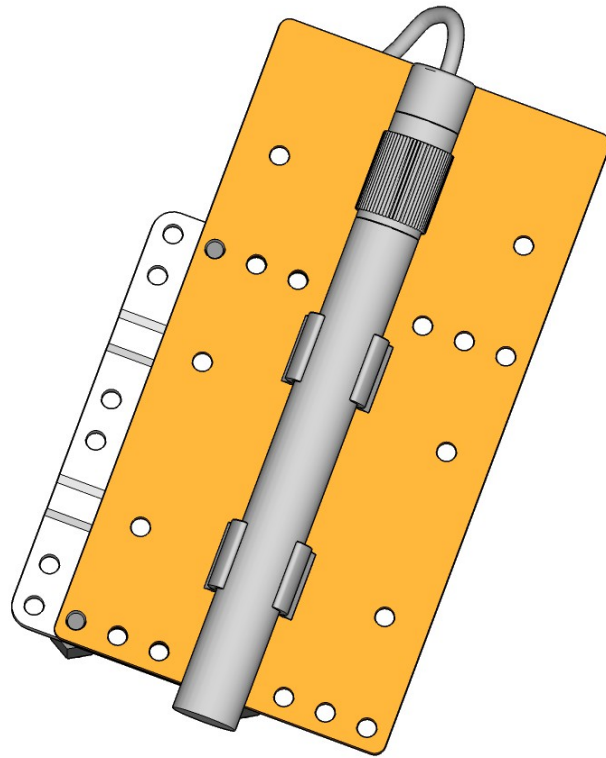
User is prompted to place first/next component at the pickup point at the right rotation.

User places the component and hits Return.

### **Inspection Service**

The user may find purpose in mounting an USB microscope on the CNC machinery for the inspection of a PCB and solder joints.

By using **PCB Central's** jogging keys, or the CNC's jogging keys, the microscope may be moved over the PCB surface to places of interest.

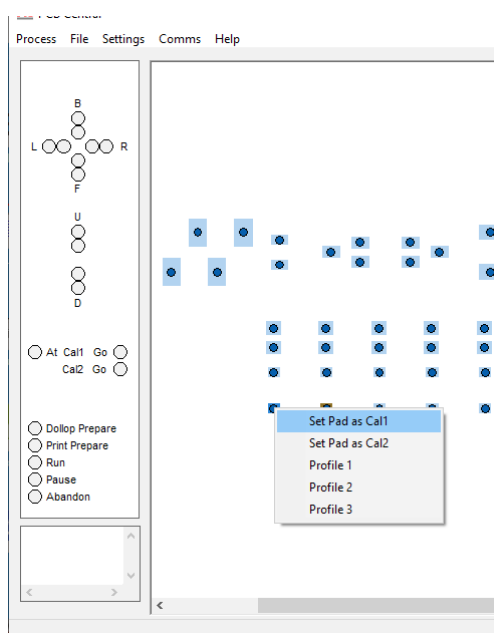


## PCB Placement And Alignment Procedure

When placed on the CNC bed the PCB alignment must be as shown on the **PCB Central** plot window. Alignment procedures can only be executed if...

- A process has been selected (Process Menu)
- A process file has been created or opened (File Menu)
- Two calibration points have been defined in the process file
- The CNC machine is online

## Setting The Calibration Points



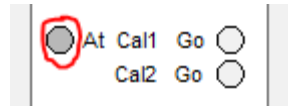
Two points of calibration are nominated by the user, typically these would be at opposite corners of the PCB. The first calibration point is where **PCB Central** assigns the XYZ origin and gains the knowledge of the offset between PCB and CNC co-ordinates.

Right mouse clicking over a pad, drill hole or component mid point invokes a pop-up menu from which the pad, hole or mid point may be set as a calibration point.

The first calibration point is mandatory where the offset between PCB co-ordinates and CNC co-ordinates are established. The second calibration point is for PCB alignment purposes.

The calibration points are saved in the process file.

Having nominated calibration points the user will first XY jog the CNC tool tip over the first calibration point. The tool tip is then Z jogged down to just touch the PCB top surface. This may be done with **PCB Central**'s jogging buttons or done with the CNC's jogging buttons.



Having placed the tool tip at calibration point 1 surface, the user left mouse clicks on the 'At' button shown above. **PCB Central** then reads in the tool tip position, establishing a reference point and offset between PCB co-ordinates and CNC co-ordinates.

The user can now click on the 'Cal2 Go' button, the tool tip will ascend, move to calibration point 2 and descend to the Z service level (as established in the settings dialog for the current process). 'Cal1 Go' causes the tool tip to ascend, travel and descend on calibration point 1. A number of iterations may be necessary to ensure PCB alignment.

Once the PCB is aligned it is secured on the CNC bed. If multiple PCBs are to be processed a 'holder' arrangement such as two L shaped brackets could be used.

Note: A calibration procedure is relevant only during the current session. On each invocation of **PCB Central** a new calibration procedure is necessary.

### About Z Levels

**PCB Central** needs to know the Z level of the PCB top surface. This is determined at the first calibration point. Further, **PCB Central** needs to know the distance from the PCB top surface to where the tool is to be placed when executing a service.

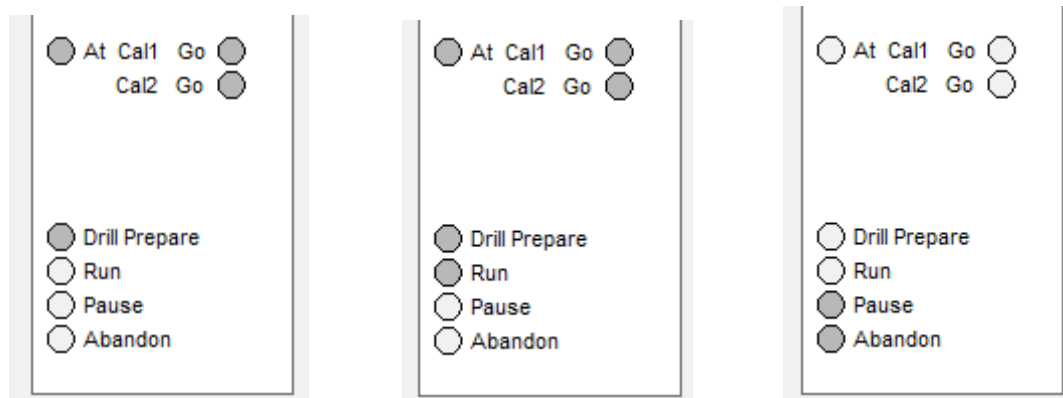
Likewise **PCB Central** must know the distance from the PCB top surface to where the CNC tool is to retract to when travelling non-service movements.

The Z service level and retraction level information is user defined in a settings dialog for each process.

### Travel And Process Feed Rates

CNC equipment express feed rates as mm/minute. This is not intuitive, and **PCB Central** has the user express feed rates in mm/sec. Conversion is done when generating G Code.

## G Code Generation And Running a Service



Once calibration is in place the 'Prepare' button becomes enabled. When pressed, G code statements are generated and stored internally. During the preparation co-ordinates are checked for boundary indiscretions.

Where necessary **PCB Central** embeds prompts in the G code statement list to signal the user of actions to be taken. In the case of the drill service, prompts are given for a drill bit change. For the assembly assistance service, the user is prompted to place components at the calibrated pick up point.

Once prepared, a job run can be started. When running a job only the Pause/Resume button and the Abandon button are active. Job runs may be repeated within a session.

Note: For stencil cutting the pause/resume button is not enabled. The stencil process can only be abandoned.

Note: G Code generation is session based as there can be no guarantee a PCB's exact XYZ position will be replicated in subsequent sessions. The G Code statements however, are written to file for checking purposes. The Etch service is not written to file due to size and time.

### About Boundary Indiscretions

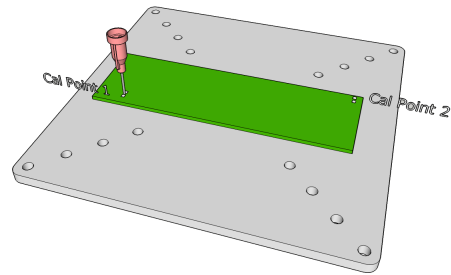
Calibration point 1 is assigned the zero co-ordinate point. This has no relationship to the CNC's XYZ centre point and there can be no knowledge by **PCB Central** if a CNC co-ordinate exceeds the XYZ space. See **Notes And Comments**.

During G Code preparation checks are made only if an absolute CNC co-ordinate exceeds the boundary limit (as set in the **Cnc Settings Dialog**).

### Solder Paste PCB Alignment

User XYZ jogs to the centre of the pad nominated as the Cal1 pad and the 'At Cal1' button is pressed.

Once this is completed the 'Go' buttons are enabled and the CNC head can be made to travel between the two calibration pads to check PCB alignment.

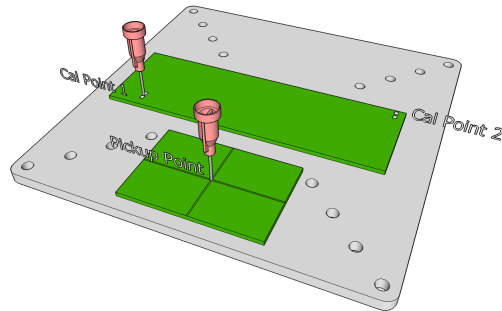


### Assembly Assist PCB & Pickup Point Alignment

This has the same PCB alignment procedure but the centre of the component is the calibration point, rather than the centre of a pad.

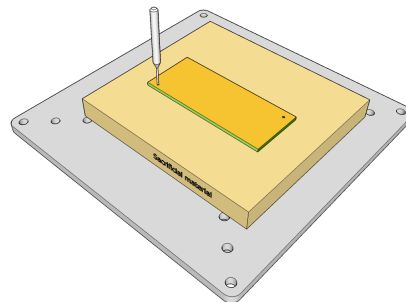
Additionally, a pickup point is set by XYZ jogging to the point and pressing 'At Pickup'.

The CNC head may then be made to travel between the three points for alignment testing.



### Drill PCB Alignment

'Cal1' calibration is mandatory. To ensure the PCB material covers the drill area 'Cal2' should be tested.



### Stencil Alignment

Since the stencil is a open film surface there is no need for alignment other than to ensure film coverage. Calibration point 1 is mandatory.

### Etch Alignment

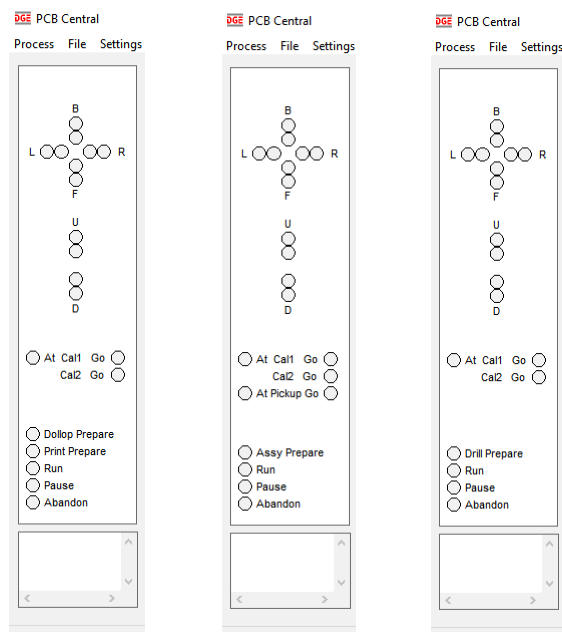
If the PCB is pre-drilled then both calibration points are mandatory. Otherwise an alignment would be conducted to ensure PCB coverage.

**Note:** All of the above is possible only when a process is selected, a process file opened, calibration points selected and the CNC machine is online.



## PCB Central CNC Control Panel

The control panel presented to the user reflects the process selected. In all processes CNC jogging may be performed only when the CNC machine is online.



## Jogging And Keyboard

All **PCB Central** processes allow user jogging of the CNC head by mouse clicking on the CNC buttons in the control panel or by keyboard keys, or on the CNC machine.

Jogging using the keyboard or keypad.

Key	CNC Head	By	
Left Arrow	Jogs Left	0.1mm	X-
Right Arrow	Jogs Right	0.1mm	X+
Up Arrow	Jogs To Rear	0.1mm	Y+
Down Arrow	Jogs To Front	0.1mm	Y-
PgUp	Jogs Up	0.1mm	Z+
PgDn	Jogs Down	0.1mm	Z-
Key + Shift		1.0mm	
Key + Ctrl		10.0mm	

## Jogging Caution

When jogging it is advisable to apply a single jog and wait for its completion before issuing a next jog request. Marlin firmware seems not to have an interactive mode, that is, a lock/step capability where an acknowledgment is issued only after an action has completed.

Marlin maintains a buffer to store incoming G Code statements, while ever there is buffer space statements are stored, and acknowledged immediately. Acknowledgments cease when the buffer is full and recommence when there is buffer space.

This allows the build up of jog statements. If the jogging step distance is large, moments of time are required to travel the distance. If the user is susceptible to impatience the tendency is to hit the jog button repeatedly, thus storing the requests. Too many jog statements can have the tool head hit the X/Y end stops. In the case of downwards Z jogging the tool tip may crash onto the bed surface.

### **Run The Service**

On 'Run' invocation, the Pause/Resume and Abandon buttons are enabled. All other buttons on the CNC panel then become disabled, as does keyboard strokes.

The CNC tool will move to the first XY position at the Z retraction level. The tool then descends to the Z service level to apply the process. If a dwell period is specified the tool waits before ascending to the Z retraction level and moving to the next position.

### **Pause And Abandon**

These buttons become enabled once a process run is started

At any time the 'Pause' button may be pressed. The pause is recognized only when the CNC head is at or is ascending to, the Z retraction level. That is, after the completion of a current process. The paused state remains until 'Resume' is actioned.

An abandon is actioned only after a current process has completed. On abandonment the tool ascends to a work space level indicated in the process's settings.

The Pause/Resume may also be toggled with the keyboard 'P' key. Likewise the keyboard 'Esc' key can be pressed to abandon a process.

Note: A stencil cutting job cannot be paused/resumed, only abandoned.

### **Repeat Job Runs**

While ever a session is active jobs may be repeated, even after an abandonment.

### The Dan M Dispensing Tool

The external tool used with Snapmaker when testing the paste service and assembly assist service is a device from Mexico. For details see...

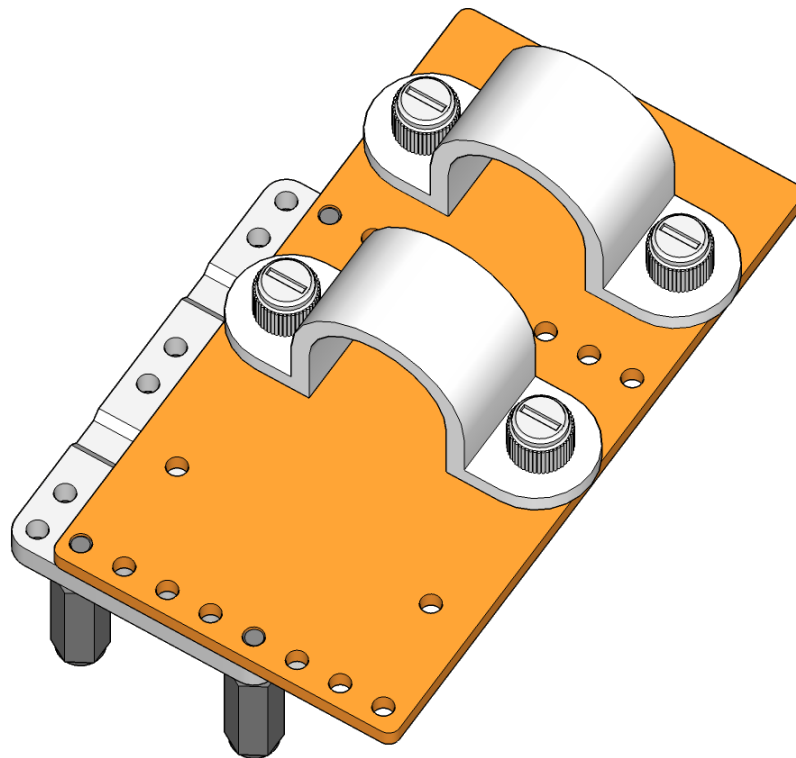
<https://www.tindie.com/products/danm/dm-solder-paste-and-adhesive-dispenser/>

The dispensing tool has its own controller which provides user settable values for the dispensing of paste/liquids either as a dollop or as continuous print.

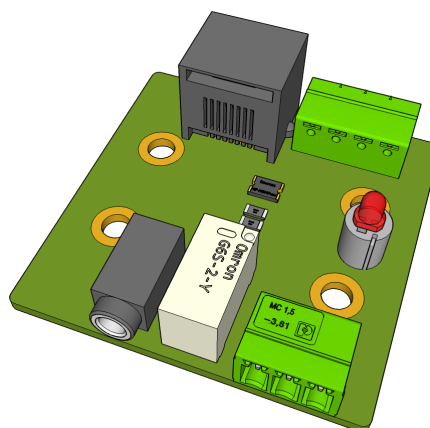
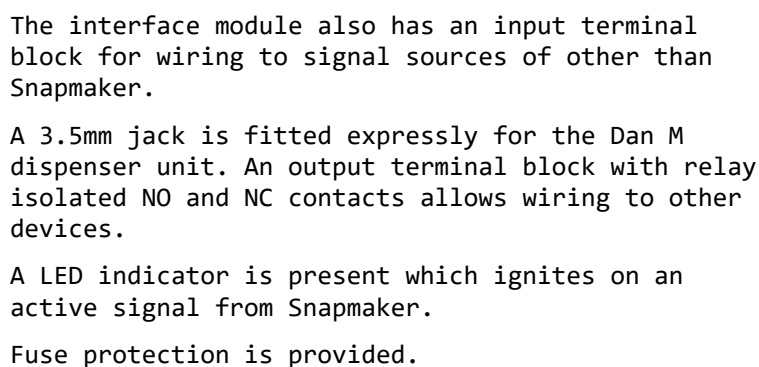
The tool also has a vacuum pick up facility.

### Tool Mounting

A mounting plate was constructed that has the dispensing tool tip approximate the XY position of the 3D printer nozzle and the CNC spindle. This is necessary for coverage over the CNC bed.



The interface module mounts on top of the Z arm and accepts the RJ45 cable that would normally plug into the Snapmaker modules. The interface module is seen by Snapmaker as a CNC tool head.



## Comments & Notes

### Pick And Place Truant Components

The author created a 'Company logo' library component made up of PCB top overlay tracks and arcs. The component had no pads to make a reference from, and the author failed to nominate a reference point within the artwork. This resulted in unyieldly number for the co-ordinates, effecting the pick and place file.

### Snapmaker Original Lack Of Boundary Detection

There is no method known to the author to have Snapmaker Original automatically home to its XYZ centre point. There are no limit switches to gauge when an end stop is met.

This bears on **PCB Central's** inability to determine if a CNC co-ordinate is outside of the space.

### Snapmaker Original And Baud Rate

If the Baud is not set to 115200 Snapmaker is unresponsive. This was confirmed using Tera Term.

### Vagaries Of Virtual Comm Ports

If **PCB Central** becomes unresponsive or not appear when first invoked, either power off the Snapmaker or disengage/re-engage the USB connector. **Windows Task manager** may be invoked to see if a **PCB Central** worker thread is active.

The USB VCP's presence may be checked through Window's Device Manager. The port will be seen as "USB-SERIAL CH340 (COMxx)".



Document Revision

1.00    xx-Feb-2023    Initial release.



**DGE Sales**

**Newcastle, NSW, Australia**

**[www.dge.com.au](http://www.dge.com.au)**

**[info@dge.com.au](mailto:info@dge.com.au)**