## APPLYING FLATCAM FOR PCB CNC MILLING

FlatCAM takes standard data files from PCB design software packages and converts them into CNC milling and drilling GCodes that can be run on a CNC milling machine. The PCB design can be single or double sided. FlatCAM provides the tooling setup to align double sided designs. FlatCAM also provides the board outline profiling GCode and the coding for any cut outs needed within the board area.

FlatCAM up to version 8.5 was created by Juan Pablo Caram but subsequent updates are supported by Marius Stanciu. At the time of this write up the current issue level was 8.991. The original documentation by Juan Pablo Caram describing the application of FlatCAM is to a reasonable standard but not always intuitive. This write up is an attempt to clear the mists and is now been updated to be based on version 8.991. The FlatCAM versions after 8.5 are somewhat more complicated but do have features that more advanced users might require.

Overview

This write up will cover the production of double sided boards and includes details of experiences regarding work holding and also milling tool selections. Single sided board are much less complex to produce and can be produced using a cut down version of these details.

FlatCAM needs Gerber files of the top and bottom copper trackwork and the board profile (.gbr) together with Excellon drilling information (.drl). These are industry standard format files for PCB manufacture and any PCB design software will allow export of Gerber and Excellon information. I use the Electrical design module in Fusion 360 for my PCB design (this is based on Autodesk's incorporation of Eagle). This means in the one package I have 3D design, CNC CAM and PCB design. Once the board design in finished in Fusion I can see the design as a 3D model with representations of the parts in place. I can then design a housing around this model. Fusion creates a manufacturing ZIP compressed export of all the files needed for third party board manufacture and the contents are shown below.
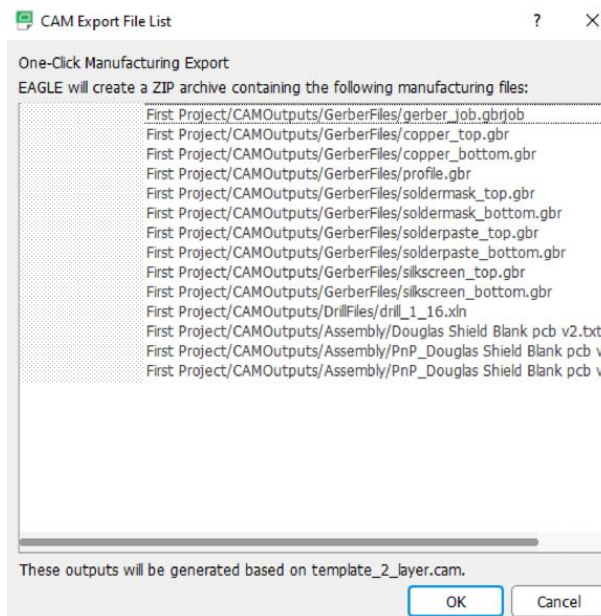
Note that like Fusion, most PCB design software packages will provide many this style of manufacturing output. Most of these files are of lesser interest to hobbyist producing simple designs in the home workshop. Should you wish to progress to having the design made by a third part manufacturer then this link is a good source of info on the various files.

https://www.allaboutcircuits.com/technical-articles/PCB-design-how-to-generate-manufacturing-files-custom-printed-circuit-board/#:~:text=The%20most%20widely%20used%20file,that%20contain%20Gerber%2Dformatted%20data.

The manufacturing files created by the PCB design package will have their data stored in ASCII format unlike a CNC machine which requires GCode format. FlatCAM provides a conversion process between these two formats. The process is not a single click on a button. The Gerber files (.gbr) first get converted to an ISO file, then the ISO becomes a CNC ready file before running the CNC conversion to GCode. The Drill file (.drl) follows a similar process. FlatCAM does streamline this process.

Board Side Naming and Mirroring

At this stage of the write up it is worth taking time to consider the logistics of the milling process that impact on the Gerber data.

PCB design software has traditionally created the PCB artwork 'looking down through' the board when viewed from component side.   This is a historic method when components had 'legs' that passed through the board material to be soldered to the copper tracks on the lower side.   As technology has progressed boards became populated on both sides and now with surface mount devices there is little to choose as to which is 'component side' and which is 'track side' except perhaps which side you are most likely to see when the board is mounted in the product.   The choice of side definition will probably be defined by the designer. He or she will be focussed on such things as connectors which will usually be on component or top side. To avoid confusion I will refer to the two sides as 'top copper' and 'bottom copper' rather than component related. This is the term used by FlatCAM.

The other aspect to consider is that the bottom copper artwork is as viewed in the design process ***down through the board from the top side***.   If the bottom side copper Gerber file is used as a basis of the milling code then the machine instructions will be incorrect.   The bottom copper instructions must be based on 'looking at' the copper artwork from the bottom side not down through from the top side.   To correct this the bottom copper artwork must be converted to a mirror image.   This can be done in the PCB design package but FlatCAM will also create this process for you.

Double Sided Milling Process

This is the simple work flow when producing a milled PCB in the home workshop : -

> The PCB material is clamped on the milling table.

> The top side artwork is milled.

> The various through holes are drilled (component related or mounting holes etc).

> The PCB material is turned over and the bottom side artwork is milled.

> The PCB is profiled with breakout tabs and then broken free from the parent blank material.

The success of this process depends on the ability, when turning the PCB parent material over to run the bottom copper, that it aligns correctly with previously milled top side copper. To do this requires the use of tooling pins that will reference the board physically 'in sync' for both top copper and bottom copper processes.

FlatCAM has a function to aid this. It allows tooling holes to be positioned as needed and then creates drilling CNC GCode for the hole locations.   The process steps as listed above will therefore have an added step to do this and this will need completing early in the manufacture.

The GCode for the tooling holes is created in FlatCAM. Providing nothing is accidentally changed on the mill X0Y0 reference, these tooling holes will allow the board to be flipped from one side to the other and maintain top and bottom artwork alignment. (Clearly for single sided boards this is not needed or relevant).

My process using a vacuum table is slightly modified on this but we will come to this later.

You can position the tooling holes where you wish but some discussion will now follow about the mirroring process.

Mirroring the Board

To mirror an object, you need an axis about which to mirror it. In CNC terms this could be about the X or Y axis.   Where you position this axis (I will call it a mirror axis) is important. If you chose to use the left hand vertical edge of the top copper artwork (Y axis) then the bottom side copper will 'hinge' on this edge like a page in a book and be placed immediately to the left of the top copper. This means the PCB blank material would have to be physically moved across on the milling table before work on the bottom side could commence.   If you make the mirror line exactly down the centre of the PCB artwork then the PCB blank will not move its symmetrical central position as you turn it over to mill the bottom copper. Once you have chosen your mirror axis the tooling pins will make sure the alignment is correct and repeatable.

To create the tooling hole drill code, you specify where you would like the tooling hole(s) to be located based on the top side view and your chosen mirror axis. When you mirror the bottom side copper you don't just get the copper mirrored but you will also get a mirrored position of the tooling holes – a chosen hole position creates a mirrored second. The location of the second mirrored hole is created in FlatCAM based on the mirror axis that you have specified.   The mirror axis can be specified as an extrapolated line from a point (e.g. X0Y0) or based on a box outline (generally this will be the periphery outline of the PCB). In my experience you need to specify 2 tooling holes for FlatCAM to create 2 further ones as mirror locations. Having two (which will create four after mirroring) keeps the board more rigidly held.

The effect of mirroring takes some understanding.   See the following images. The first image represents the parent material with the PCB layout central on the blank.   A single tooling hole has been placed and a Y axis has been chosen as the mirror line.

You can simulate this.

Take two pieces of A4 or similar paper which will represent the parent blank PCB material. Draw a rectangle on the first piece of paper to represent the PCB artwork outline. Draw a single circle to represent the tooling hole somewhere outside the PCB artwork area.   Draw a dotted line on the artwork to represent the mirror axis.   Make a photocopy of this onto the second piece of paper. Superimpose the two pieces of paper so that they represent the top side as viewed and the bottom side turned over.   Align the rectangles so they both sit on the mirror axis line and otherwise match. Unless you made the mirror axis sit exactly mid way on the rectangle you will find that the two sheets of paper must be offset for the tooling holes to align with each other.
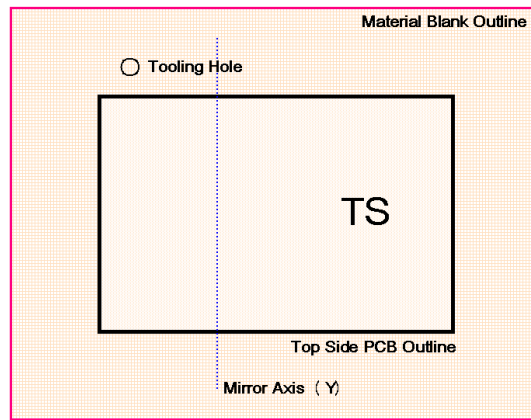
The next image is the mirrored image representing the Bottom Side as viewed from the bottom rather than down through the PCB layout from the Top Side. This is the view that the milling machine needs to mill the Bottom Side artwork.
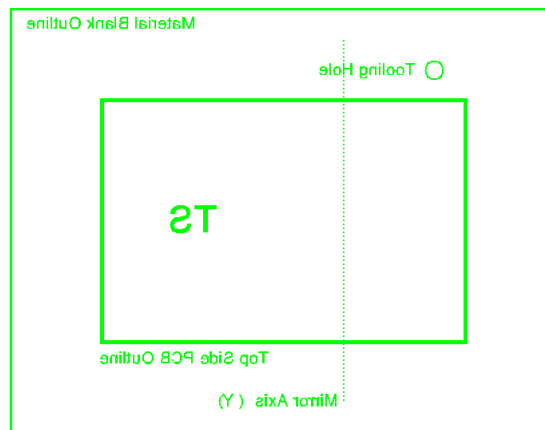
The next image shows how the tooling hole is mirrored about the mirror axis.   The two tooling holes will be drilled from the Tooling drill file created by FlatCAM.   In order for the PCB parent to locate onto the two tooling pins the PCB will have to be offset between Top Side and Bottom Side activity as shown.
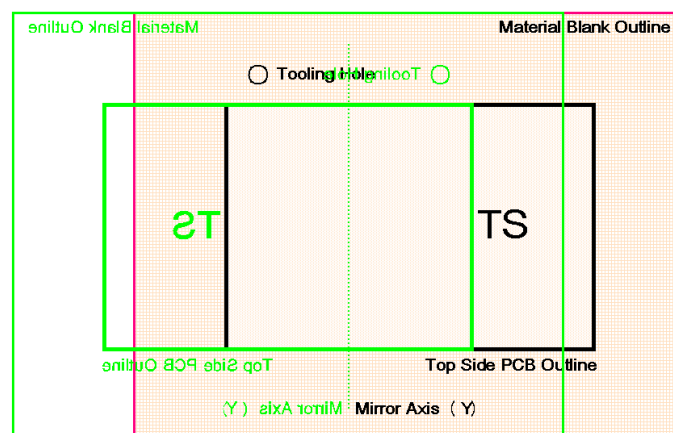
If the mirror axis is exactly central on the PCB outline all the problems go away as can be seen in the next image.
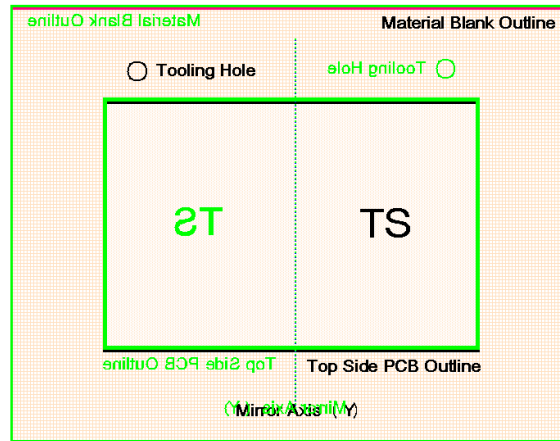
Why is this mirroring set up so important ?

Suppose you have a fixed frame tooling jig to clamp the parent material down and you do not have the artwork geometry centred at X0Y0   When you try to mill the bottom side copper you will have to move the frame to match the movement in the parent material relative to the tooling holes. This is hassle and could lead to alignment errors.

I suggest that in order to create less hassle it is prudent to make the mirror axis symmetrical either in X or Y on the PCB design.   In order to facilitate this it is important to make the centre of the PCB artwork design equal to the X0,Y0 coordinates. This can be done in the PCB design package where you should be able to set the PCB design origin point. It can also be done in FlatCAM.

I mentioned earlier that the mirror can be defined in FlatCAM using a Point (X,Y) or can be from a Box.   The Point method is already described where you create a 0,0 reference in the centre of your PCB design and use this as the Mirror either about X or about Y.   If you use Box, unless the artworks for top and bottom have a common geometry outline designed on them, FlatCAM looks at the artwork design and decides where it believes the centre point of the artwork is. This could be completely different between top and bottom copper designs. *If you need to use Box then make sure you have an identical border size border positioned in the same relative position on all layers of the PCB design.*

To help familiarity I suggested that an arbitrary pad is placed on the artwork on both top or bottom sides that sits on the axis mirror line you have chosen.   This should be outside of the formal design area. This pad gives you a reference that all is well with the mirroring of the bottom side. This will hopefully become clearer when we move into the process in FlatCAM.

PCB Design Files Needed

The file requirements from the PCB design software now breaks down into the following : -

> Board outline (Gerber file *.gbr)
> Top side copper artwork (Gerber file *.gbr)
> Bottom side copper artwork (Gerber file *.gbr)
> Drill file based on top side view (Excellon file *drl)

Note that in order to avoid confusion I find it better to not mirror any of these in the PCB design package export.   Keep this instead just as a FlatCAM process.

Here is a screen shot of the FlatCAM opening screen in version 8.991. A bit overpowering to say the least.   I have never used any of these icons so don't be put off.
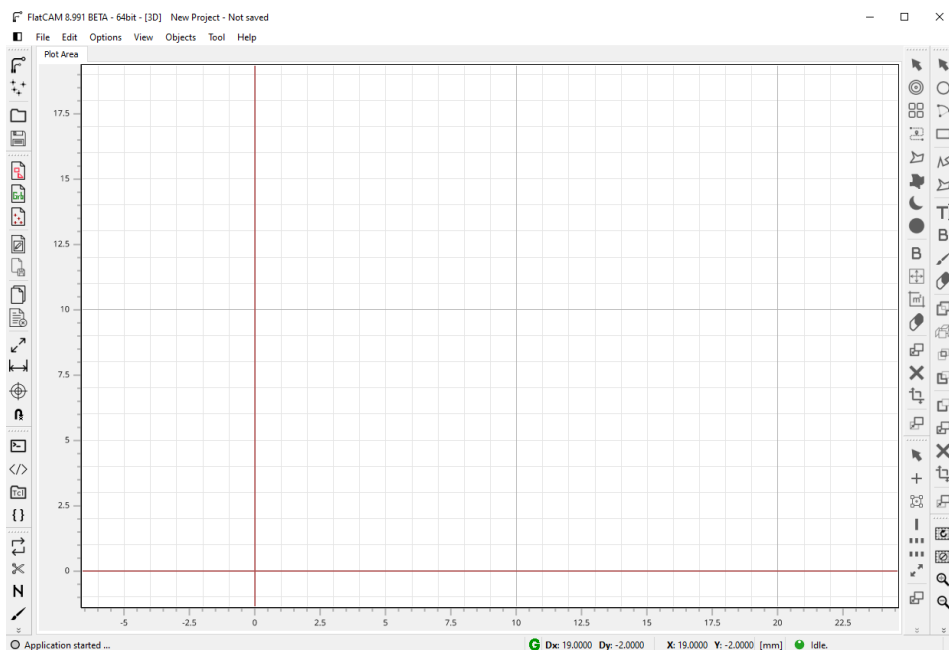
Navigation

You can move around the FlatCAM screen using your mouse.

To zoom in and out of the image area you can rotate the mouse wheel.   The image will enlarge or shrink centred on the current mouse cursor position on the PCB image.

To move the viewing area and re-centre the image press and hold down the scroll wheel and drag the image to where suits. You can also use the Zoom Fit button below the File menu.

Clicking with the left button anywhere in the drawing area will automatically transfer the X and Y co-ordinate values of that point to the computer clipboard. This is important to note.

Stage – 1 Importing Data

Because electrical components have traditionally had their physical packages created in an Imperial unit's world, it is likely that your PCB design package will create the layout on an Imperial based grid. As parts have got smaller and smaller this does get complicated.   If you wish to process the files within FlatCAM as a Metric based activity (perhaps your milling tools are Metric based) then you can set the preferences file in FlatCAM to work in Metric. Go to Edit/Preferences (and be totally overawed by the resulting screen. Yes this really is the first of 8 Preferences tabs).   Before you break out into a cold sweat, Select MM or IN and Save.
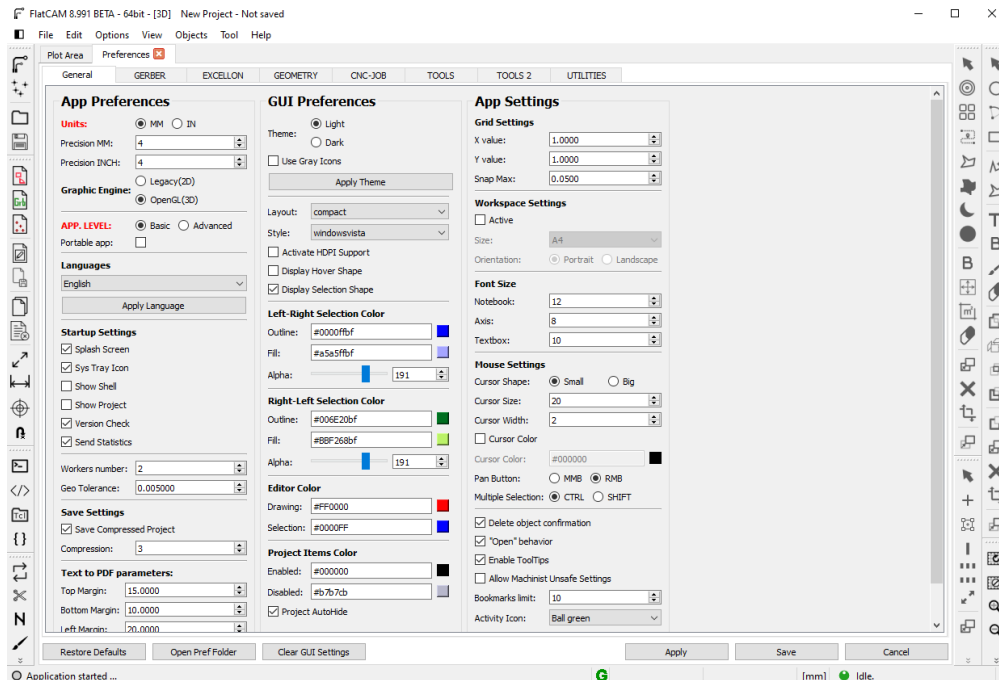

Create a folder on your computer and store the exported Gerber and Excellon files from the PCB design package. If as from Fusion this is a ZIP file then extract the files into this folder. For the purposes of this write up these will be called TOP.gbr, BOTTOM.gbr and DRILL.drl. As you read further you will realise that there is an advantage in also importing a BORDER.gbr file having just the geometry of the border outline of the finished PCB. Ideally the board artwork should be designed and referenced centrally as X0Y0 but if this is not the case then the next section will show you how to modify this.

Again I recommend that the files exported from the design software should not be mirrored in anyway.

Load the files using the FILE/OPEN dropdown menu to bring in the three GERBER and one EXCELLON file.   The later versions of FlatCAM allow you to select and open a number of Gerber files at once using Control and clicking them.

If you now click on the Projects tab you will see the files you have chosen are listed and they will also be visible graphically stacked on top of each other in the main graphic area. Here are some simple shapes created from different Gerber files imported into FlatCAM.   There is a PROFILE called Plot_1.gbr, a TOP and BOTTOM gerber and a DRILL file.

Note that all files are superimposed on each other and are all referenced to 0,0 in the bottom left hand corner.   The XY readout is at the bottom right edge of the screen. X and Y values are absolute and the Dx and Dy are relative values from the point you last clicked. The demonstration object is a 25mm square.
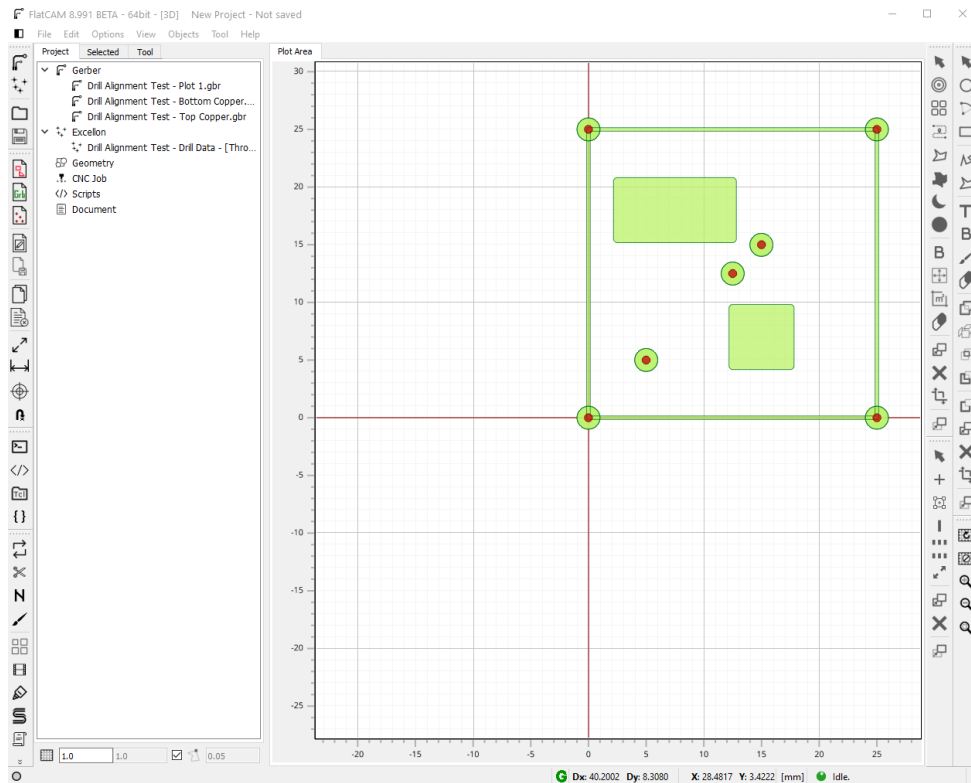
Stage 2 – Resetting the PCB X0Y0 Reference

If you have a design where the Gerber files you have imported are not referenced centrally to X0Y0 then you need to change this.   Normally the PCB design package would make the bottom left corner of the PCB layout equal to X0Y0. If this corner is nowhere near the X0Y0 and you choose to continue, you could have a situation where the code will demand a large movement of the spindle from its reference position to the working area for the milling of the board.   This could be outside of the milling envelope which is a problem and potentially a tool crash. Even if the board is referenced bottom left at X0Y0 this is not ideal as it has implications for the double sided board mirroring process. Our sample objects are all referenced to 0,0 and need to move them to have 0,0 central in the square.

Follow this sequence : -

1   If you have not already done this, go to File /Open and navigate to the folder where you have the files for the project stored and select and import all the files.   The file images will now appear stacked on each other in the Graphic area and will be listed in the Project tab listing area.

2   Click on the PLOT_1.gbr file in the Project tab to make it the file of interest. This file will define the maximum outline size of the board to be milled.   Use your mouse to zoom into the bottom left and top right corners of the border outline and read off the X and Y coordinates of these two points as accurately as you can.   If you know your board size from your design activity this helps.

3   Now for some tricky maths. You need to move the design to have X0Y0 sat at the dead centre of the board.
         Let's call the bottom left coordinates XL and YL and the top right XR and YR.

If you subtract the values (XR – XL) and (YR – YL) this will tell you the difference in coordinates between the bottom left and the top right of the BORDER outline. The centre point of the board will be these values divided by 2.

Let's assume the simple case where XLYL was at X0Y0. To move the design to make XLYL at the centre of the board we need to create an offset of X = -(XR-XL)/2 and Y = -(YR-=YL)/2. This can be entered by going to PROJECT tab, clicking on PLOT_1.gbr and then going to the SELECTED tab. At the bottom of this tab is OFFSET. In the VECTOR box enter the negative values you have calculated. Note that when entering these values you do not need to put the X and Y but solely the numbers with the X first and Y second with a comma between. Copy the values you have entered into the box using Control + C. Click the Offset button. The PLOT_1.gbr will now jump on the display area to sit centrally about 0,0. If it doesn't then recheck your maths.

In our example the square is 25mm on each side so we need to (-12.5,-12.5) in the Vector box.
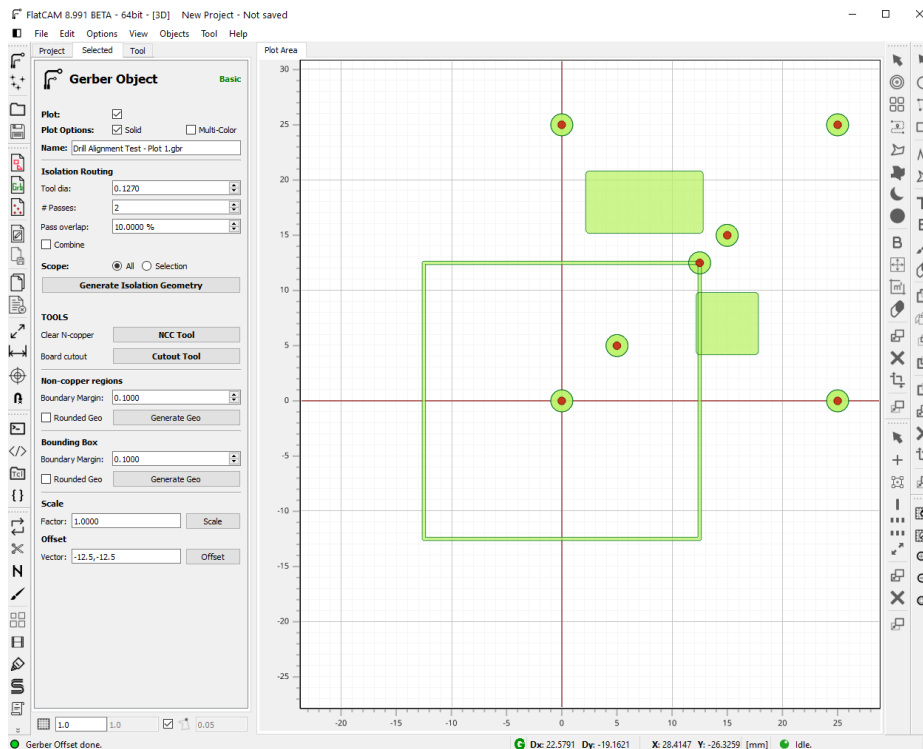
4   If the co-ordinates for the bottom left corner of the PLOT_1.gbr image were not sitting at X0,Y0 the calculation must take this into account. In effect we must move the image in two steps rolled into one. The first step is to move the corner from wherever it sits (let's call this XW, YW to X0Y0 and then make the shift to sit the image centrally as per (3) above. The maths are similar.

First the move from XV to YW is a simple negative value of -V and and -W and then the move as in (3) making the new equation : -

$$X \text{ Vector move} = -(V+((XR-XL)/2)$$
$$Y \text{ Vector move} = -(W+((YR-YL)/2))$$

Enter these values in the OFFSET Vector box, copy the values and then click the Offset button.

5   Repeat this process for the TOP.gbr, BOTTOM.gbr and DRILL.drl. Because you copied the values, time will be saved by copying the values into the Vector box using CTRL+V.   If you do not Copy the previous values there is a danger of entering a wrong value. This will mean that the artworks will no longer be correctly stacked one on top of the other. Note that the Excellon dialogue window is slightly different but there is still an Offset vector section at the end of the settings section.
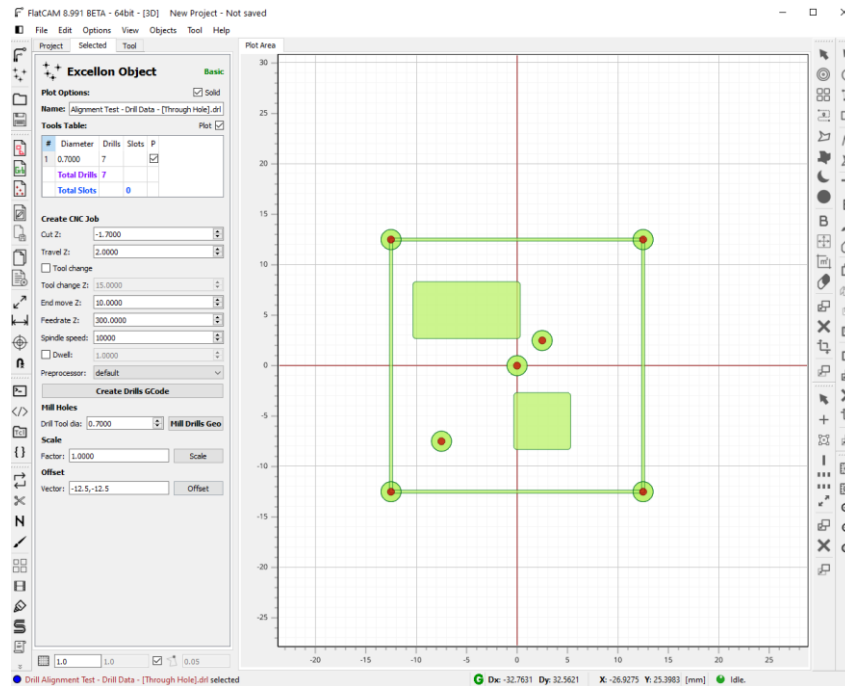
6   You should now have all four artworks located centrally and symmetrically about X0Y0 and stacked on top of each other.

Stage 3 - Mirroring the BOTTOM.gbr file

As mentioned earlier, the four images of TOP, BOTTOM, PLOT_1 and DRILL should now be shown in the viewing window and should be superimposed on each other and symmetrically positioned about X0Y0. The data for BOTTOM.gbr is as viewed down through the PCB when looking from the top of the board and this needs to be reversed (mirrored). Without this process the GCode file for the milling process will be the opposite of what it should be. It is important to mirror BOTTOM.gbr before you continue.

Click on Tool on the top menu strip (not the Tool tab) and select Double Sided PCB Tool. This will now take you to the Tool tab where the mirroring and tool functions are selected.

The first selection is to pick the artwork that is to be mirrored in the GERBER box. Usually this will be the BOTTOM.gbr artwork. Do not press Mirror yet.

Next select the Mirror axis. As described earlier this can be done about the X axis or the Y axis.   A mirror on the X axis will flip the layout vertically and a Y axis mirror will flip the layout horizontally.

You must now enter where you want the mirror line to sit on the artwork. Providing the artworks have all been referenced to have X0Y0 as the dead centre you can enter 0,0 as the Mirror point and press Add and then Mirror

Stage 4 - Tooling Pins

Assuming you have created and entered the mirror axis you now must enter one or two positions for tooling pins to hold the PCB in the correct place for BOTTOM and TOP milling.   These can by anywhere around the PCB but outside of the artwork geometry. When marking these positions keep in mind the working area of the milling spindle and ensure that the tooling holes are well out of the way to avoid a crash. You can click on the position you want to use and then copy the value into the Alignment Hole box or manually write the values in the box. Think about the mirror position implications when choosing the locations.

Enter two values so each X and Y is bracketed and the two values are separated by a comma and you put a decimal point in each value [(Xa.a,Ya.a),(Xb.b,Yb.b)]. You can click in the box and edit your copied values if you wish to change them.

Once you have your two chosen locations entered, enter a value for the tooling pin hole diameter and press Create Excellon Object.

If the mirror is symmetrical you will now see four red circles on the display representing the tooling pins (the two you entered and the mirrors of these) and these will be symmetrical about the PCB mirror line you have chosen (X or Y).   If the mirror axis is not symmetrical on the board geometry then the four tooling holes will be asymmetrically placed and you need to recheck things.

Here is the new view that is created.
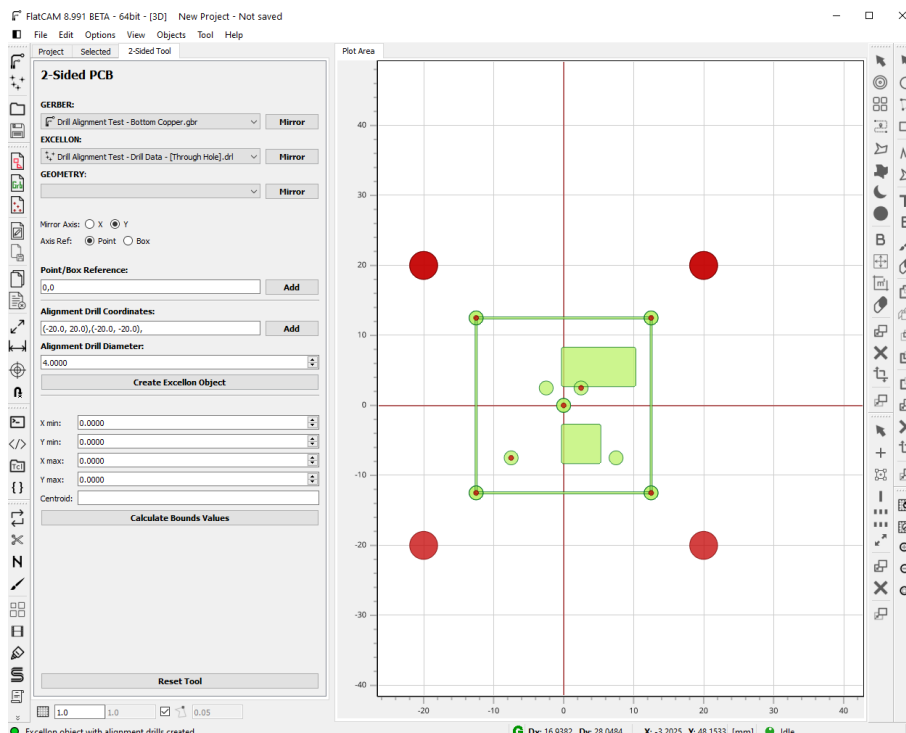
If you check the PROJECT listing tab you will now have an added file called Alignment Drills. If you decide you want to change the locations of the tooling pins you can delete the Alignment Drills file and re-do the positions.

Do not re-mirror the object or the BOTTOM.gbr will no longer be correct.

Stage 5 – Gerber Object Dialogue

You may have already used the lower area of the Gerber Object window to move the artworks to X0Y0 using the Offset Vector section. For this simple introduction we will ignore Non Copper Regions and Scale sections. We will focus instead just on Isolation Routing and later on we will use the Boundary Box section.

1   Go to the PROJECT tab and select TOP.gbr and then the SELECTED tab to see the Gerber Object working area.
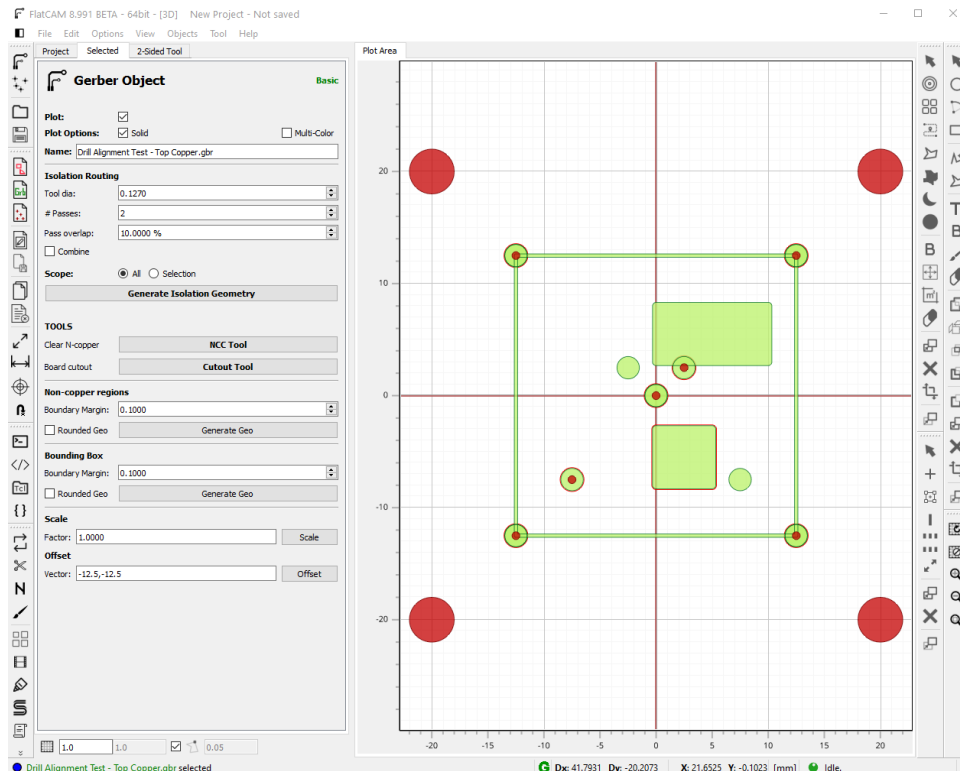
2   Enter the milling tool diameter you will use to cut the copper in Tool Dia. Take care that you use the same measurement standard as the Preferences selection.

3   Set the number of passes that will occur around the geometry and how much each pass should overlap the prior one.

4   Press Generate Isolation Geometry and the objects on screen should now have red borders. In the example this is the pads at each corner and the three diagonal pads across the screen and the small square pad.

5   You now have some further selections to set. Cut Z is the depth of cut, if you select multiple depths then the tool will pass over the artwork a number of times (not usually needed for cutting the copper traces), travel Z sets how far off the board the spindle starts at before moving down to cut the copper, (ignore tool change and associated Z), end move is the amount Z moves up at the end of a run, feed rate is the spindle movement speed, feed rate Z is the up and down speed and Spindle speed is what it says but chose the fastest speed your spindle can deliver.

6   Press Generate CNCJob object. The objects will now change to have a blue outline and will show orange tool paths to represent the movement of the spindle. A new dialogue screen will appear.   This confirms the tool chosen and allows you to enter some extra GCode commands at the beginning and the end of the GCode file. (The ones displayed are my standard Preferences).
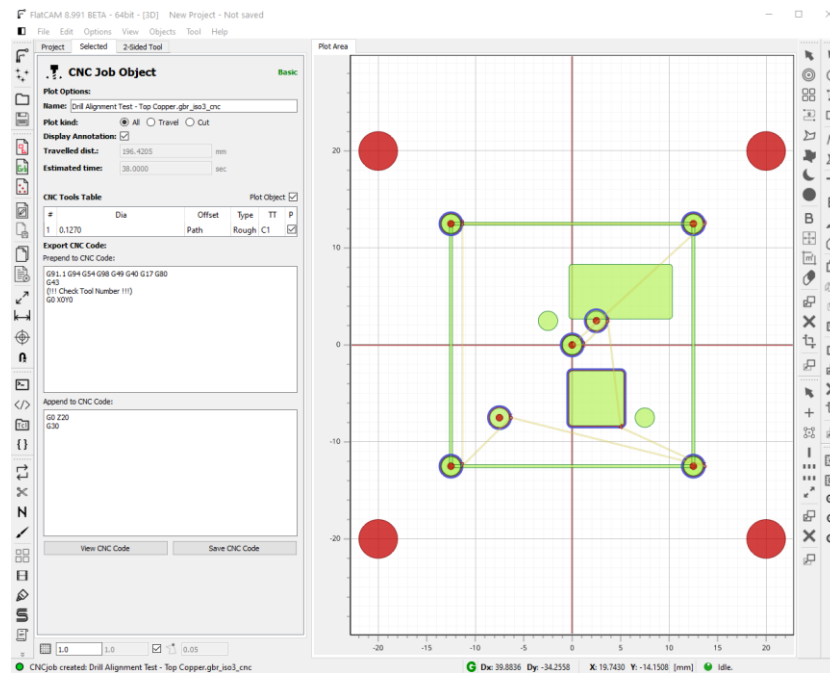
7   When completed press Save CNC Code and select a destination and name for the file you have created. This should have a .nc extension.

8   Go back to the Project tab and. You will notice in the Project tab that there is now a file called TOP.gbr_iso and a CNC file.

9   Repeat this process for your other Gerber files.

10   If you have other Gerber files repeat this process until all files have matching ISOs and the associated artworks have red outline and blue outlines. Note that the BOTTOM.gbr file does not get changed after mirroring.

11   There will now be a TOP.gbr_iso_cnc file in the Project tab listing.

12   Repeat this process for all the other ISO files.

13   When all ISO_CNC files are ready go back to the Project tab and select the TOP.gbr_cnc file.

14   Go to the Selected tab and you will see a window for the CNC Job Object.   The file name should match in the top box.
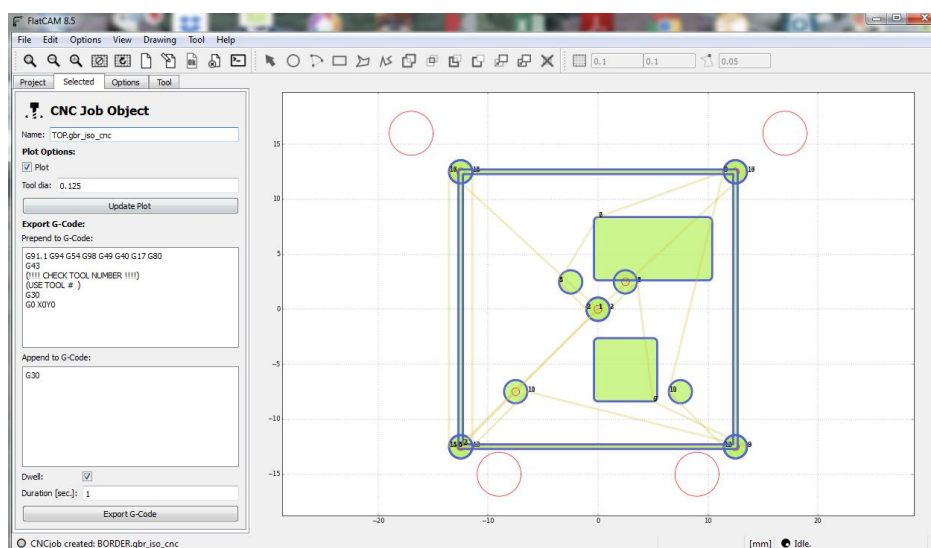
Stage 6 - Excellon Object Window

The processing of the .drl files is slightly different.

1   Go to Project and click on the DRILL.drl file. Go to the Selected tab. The Selected tab is now different being an Excellon Object window. At the top of the Excellon Object window you will see the file name.   If this isn't DRILL.drl go back to Project and select it so it appears in the window.

2   You should have Plot ticked and Solid ticked only if you want to see a red dot representing the holes.

3   In the Tools window you will have a listing of the tools that have been defined in the DRILL.drl file for use drilling the holes on the PCB. Click on the # symbol to include all tools or select the ones you only want to be included for this CNC activity.

4   Go to the Create CNC job section and note the following : -
   a. Cut Z – this is the depth of cut from the top side copper through the board by all the drills you have chosen to use. It is always a negative value as it is cutting into the PCB material from its top surface.
   b. Travel Z – this is the height that the tool tip moves up to while moving rapidly around the board to jump to another drilling position.
   c. Feed rate – this is the rate at which the spindle will move while performing the drilling process.
   d. Tool Change – untick this for now.   It is the height that the spindle must move to so you can access the collet to change a tool in mid process.
   e. Spindle Speed – this is the spindle speed while drilling.   I think you must enter a value in this box or you will not be able to create the CNC code. Choose the fastest speed your machine can handle.

5   Click on Generate

6   You will now see the pattern of the spindle movement and the order in which holes will be drilled and a new file will appear in the Project window – DRILL.drl_cnc.

7   Select the DRILL.drl_cnc file and go to the Selected tab and you will see a window for the CNC Job Object the same as the you had for the Gerber files.

8   Plot should be ticked and you should check that the tool diameter matches your expected size and if you edit this click on Update Plot.

9   The next two boxes allow you to add GCode commands to the start (Prepend) and finish (Append) of the GCode that is going to be created.   These are usually machine specific start and end commands and will depend on your machine requirements. Details of these settings are given later in the text.

10   Tick the Dwell command and leave at 1 second. This defines how long the program sits waiting before running. This gives you a bit of breathing space to hit the big red button.

11   Click on Export G-Code and name the resulting file before saving ready for loading into your milling machine. I suggest you give the file a .nc extension name.

12   Repeat this process for the Alignment Drills file but the depth of cut now needs to be the thickness of the PCB material plus (say) 80% of the thickness of the sacrificial material.
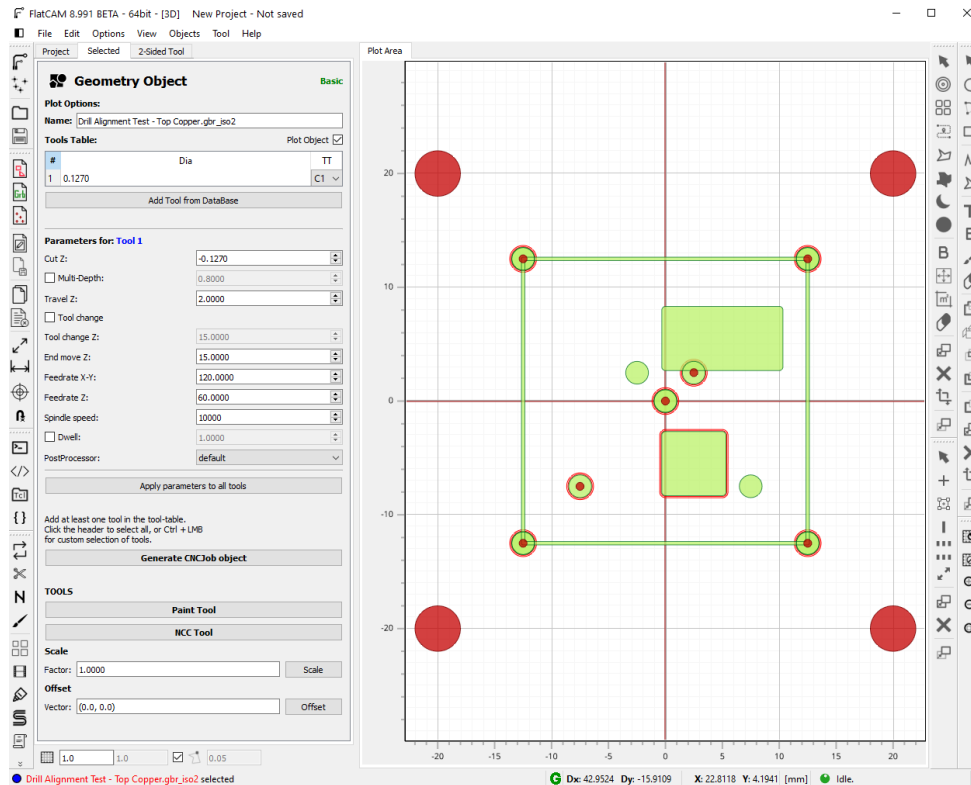
Stage 7 - Board Outline

There is a facility within FlatCAM to mill the board profile and leave breakout tabs. This requires a Gerber file that has the outline of the board contained in it.   Alternatively, this can be a standalone Gerber that just has the outline geometry. For this example there is a file called BORDER.gbr.

    1   In project select BORDER.gbr and then click on the Selected tab.

    2   You will see the following image from the Gerber Object details.
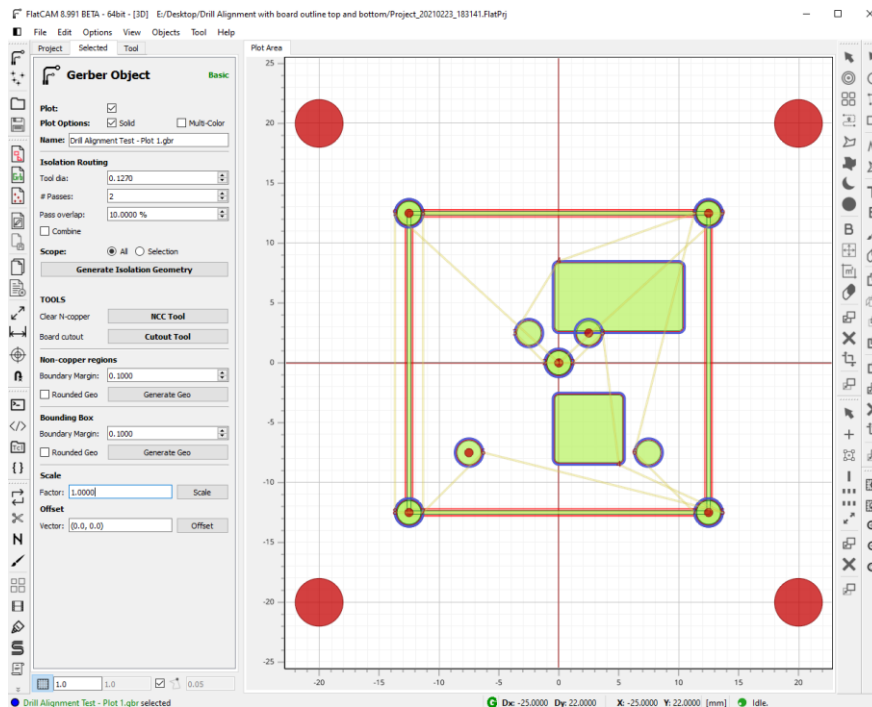
           Click on the Cutout Tool button..

3   Make sure that the PROFILE file is shown in the top box.   Set the tool width and depth of cut if not already picked up from Preferences.

4   You might wish to make the cut using multiple depths to save stress on the tool in use.

5   The Margin box allows a gap between the PCB outer edge geometry and the router path.

6   Gap Size is the width of the tabs that will be left by the routing.

7   Automatic Bridge Gaps sets the number of tabs. There are various selections for this but I usually leave it as 4. If only two tabs are chosen then you have the choice of on which axis these will be.
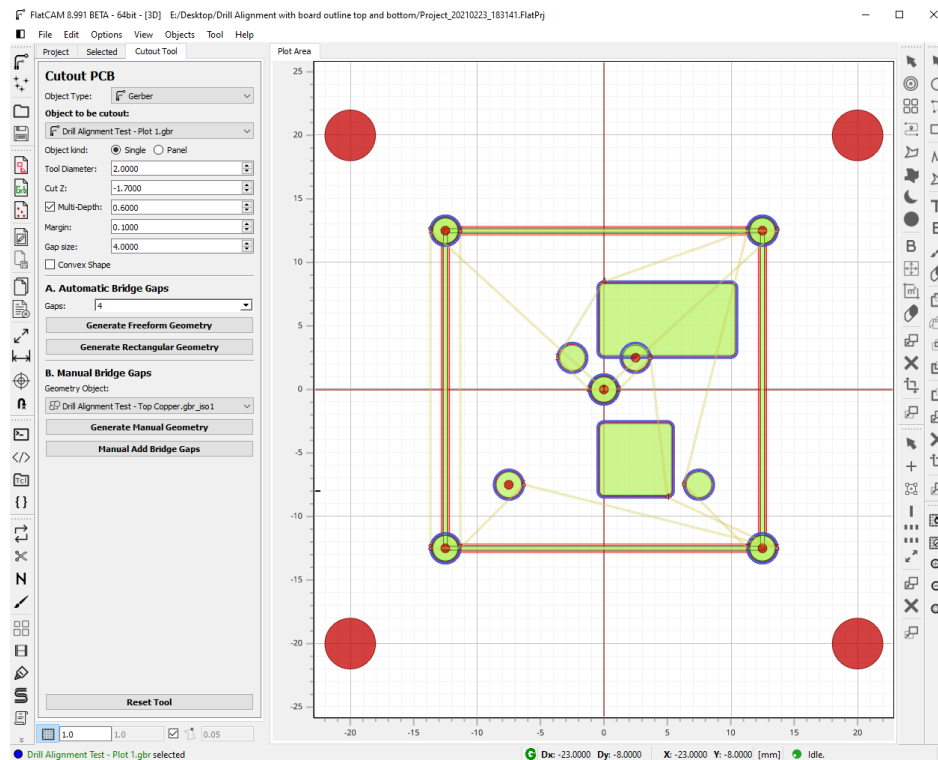
8   On completion of these four sections you can Generate Rectangular Geometry to create the BORDER.gbr_cutout file. You will see four profile lines appear on the layout to represent the cut geometry.

9   The left hand panel will now change and you should select the PROFLE file now in the Geometry section.process now follows as for the other files.   Select the BORDER.gbr_cutout in Project and then go to the Selected tab to set the Geometry Object. Check the Parmeters for Tool #1 match the previously entered settings.

10   Click on Generate CNCJob button.

11   The image will refresh with a blue band indicating the cut path and showing the tabs. Check any additional GCode settings and then click Save CNC as per the other files and store the file with an appropriate name.

Process Summary

This write up is long but I hope you will see that the process boils down to having the correct Gerber files from your PCB design package and following repeat steps.

a) Import the Gerber(s) and Excellon file(s) from the PCB design package.
b) Move all files to be symmetrically centred on X0Y0.
c) Go to the Double sided PCB tool.
d) Select the Bottom.gbr
e) Add the Mirror Axis and Point reference.
f) Add the Tooling hole(s) and drill size.
g) Mirror the BOTTOM side.
h) Create ISO files of each Project file
i) Create ISO_CNC files of each Project file
j) Create a GCode output file for each Project file.

If you are not happy with a file for any reason then go to the Project tab and delete it and re-run the processes associated.   The one exception is the mirror of BOTTOM which should be done just once. The problem with the mirror action is that the BOTTOM file does not get re-named in any way to give you confidence that it has been done and saved. You depend on seeing its status on the main layout window where it will be obvious which orientation it is in.   If you doubt it in any way then delete it from the Project list and re-import from your PCB design package and reprocess the file through the various steps.

Milling the Board

The following notes assume that the board is double sided, you have a symmetrical mirror on the BOTTOM file and you have at least one tooling hole that has been positioned and mirrored with consideration to the BORDER file outline.

1   In order to protect your milling table you must first clamp a piece of sacrificial material to the milling table.   I use MDF as this tends to be quite uniformly flat. It needs to be 10mm minimum to give rigid mounting of the tooling pins. This board is clamped independently of the PCB material so the fixings must be outside the area that will be occupied by the PCB material.

2   Cut a piece of PCB blank which must be oversize on the actual board geometry and make allowances for the tooling hole positions. Mark on it two diagonal lines to indicate a rough central position. Fix the PCB blank in place on top of the sacrificial board such that the milling machine spindle is at X0,Y0 and closely sat over the centre of the PCB material indicated by the crossing of the diagonals. Clamp the PCB material in place. This can be hard clamping or could just be using masking tape so long as you are confident it can't move and is held flat to the sacrificial board.

3   Zero the mill spindle Z0 to the top of the PCB material. You could use a Haimer or active probe or you can zero each tool in turn using a thin sheet of paper under the tool end point.

4   Load the Alignment Drills.nc file into the milling machine.

5   Load the milling tool or drill specified for the tooling hole size and re-check that the tool measures to Z0 when just touching the top surface of the PCB blank.

6   Check the cut Z reflects the depth of the PCB blank and 80% of the depth of the sacrificial material.

7   Run the Alignment Drills.nc.

8   This will create the two tooling holes or some other even number of holes depending on how many tooling hole positions you have chosen.

9   Dust off the material and insert tooling pins into the tooling holes. The PCB is now rigidly located on the sacrificial material which in turn is rigidly fastened on the milling table.   The PCB will need some extra fastening to make it as flat as possible on the sacrificial material.

10   Load the TOP.nc GCode into the mill and install the chosen milling tool. Check Z0 to the top of the PCB blank. Run TOP.nc

11   Repeat this for the DRILL.nc file with the correct tool loaded and after having checked Z0.

12   When the above operations have been run, carefully lift the PCB blank from the tooling pins and turn it over either in X or Y. This will depend on the Mirror axis you have chosen. Fit the board back over the tooling pins and clamp it in place. If you have followed the symmetrical mirroring instructions the PCB blank will sit in the same position over the tooling pins but now be bottom side upwards.

13   Run the BOTTOM.nc GCode file to cut the bottom side artwork profile after having rechecked the tool Z0.

14   Change the tool to a milling cutter as chosen in the Board Cutout section, zero the tool to the Z0.   Remember that the cutter is now cutting just through the PCB material so make sure the Z depth of cut is correct. Run the PROFILE.nc. This will leave tabs to keep the board fixed in place until the milling has finished.

15   This completes the board and it can now be removed from the milling table and broken out of the PCB blank reading to use.

Default Settings and CNC Edits

FlatCAM has the ability to save a set of preferred settings. As already mentioned, the Preferences section runs to 8 tabs. It is somewhat daunting to look at so I have just done screen shots of my settings. I should probably recommend that first of all you investigate the settings on these tabs and enter values that will suit your processing going forward. This will speed up the processing of each file.

Things to note : -

General – set Units

Gerber – set number of passes and standard milling tool diameter (0.127mm=5thou)

Excellon – set drill depth and default drill size. Fast retract in Adv Options.

Geometry – set cut depth (0.127mm=5thou) and spindle speed.

CNC Job – allows you to enter fixed header and footer GCode to your output files. Very useful.

Tools – (Huge tab !) Set your default Cutout Tool and 2 Sided Tool.

Tools 2 – nothing of major importance

Utilities – nothing of major importance

Once you save your Preferences these will load with every new job you process.   Note that FlatCAM updates are very frequent and you will find tweaks and mods continually being added. I find it best to stick with a version that does what I need and not be tempted to upgrade.   There is a very good forum on the FlatCAM website for watching issues and problems.

Milling Cutters

You will have your own opinions on what is best to use to cut the PCB geometry.

V bit cutters are the lowest cost and are readily available from overseas sources. They can be fragile if you are using a narrow tip (5 to 10 thou) with a narrow V angle (10 degrees). If you choose to use a narrow tip with a wide angle (45 degrees for instance) then small variations in board clamping flatness will lead to a wider and deeper cut into the PCB copper and substrate. Depending on the artwork this may or may not be a problem.

A simple exaggerated sketch of the problem is below.
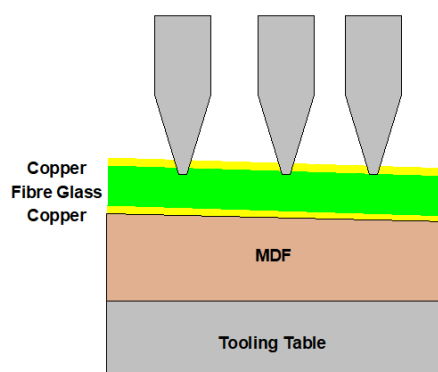
The three V tools should be regarded as being at the same height and the board bowing up more on each one. Underneath the board is a sacrificial material such as MDF so that through drilling does not damage the tooling table or the drilling tools. It can be seen that the width of cut would change with the board flatness with respect to the tool tip.

The ideal cutters to use are these with a short parallel section from the cutting edge before the taper section commences. This eliminates width of cut variations but will not improve depth of cut changes. Cutting too deep into the PCB substrate material will lead to more wear on the cutter than would be the case if just cutting copper.

There are hybrids of both types available and I have found that the US supplier *Think and Tinker* have some very good tooling for PCB routing.

<u>**Conclusion**</u>

This write up is long and will need a number of readings and experimentation to understand the process. There is repetition in the process which you will soon realise means a simple click to accept action as you go through the various stages and you will get quicker as a result.

This is greatly improved if you spend time getting the Preferences set up to your satisfaction.

I have spent many hours fine tuning my FlatCAM process and am now able to produce repeatable very fine artworks. Here is a micro-USB pad array as an example.
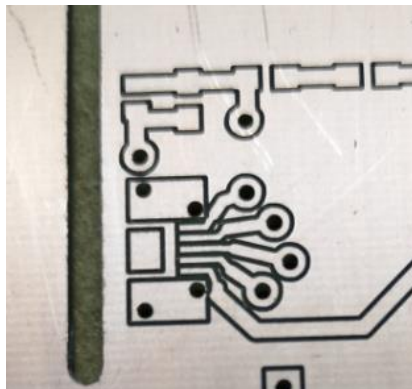
I hope this write up helps you achieve the similar results. If it has been a help to you then please leave a comment in my blog www.altrish.co.uk or via email on subs999@altrish.co.uk

Appendix A - Clamping the PCB blank and Coolant

This is an 'experiences download' for you to cherry pick what suits you best. There are various methods of clamping and the aim of each is to try to clamp the PCB blank as flat as possible on the tooling table. As already mentioned any variations in the surface flatness will create a variation in the width of cut and depth of cut into the PCB copper surface and underlying substrate. This is the most critical aspect of the process. Note that single sided PCB material is much more difficult to hold flat than double sided. This is due to the asymmetric stress created by the one sided copper lamination process. This leaves the board with a pronounced bow/set.

Edging Strips or Fixed Frames

Masking tape, double sided tape and some form of edging strips from metal stock are the simplest port of call for holding the PCB blank. The downside is that as these strips clamp the edge, they can exert a downward pressure on the PCB edge and this in turn leads to an upward bow of the centre of the PCB blank. This can be improved by having foam rubber strips on the edging strips to decouple the clamping action.

If you have a matrix type tooling plate on your mill table you can have an open frame that matches the matrix and this can have foam decoupling also. The frame should clamp down hard onto the table and have the correct spacing for the sacrificial board and the PCB thickness and an associated gap for the foam pressure pads.
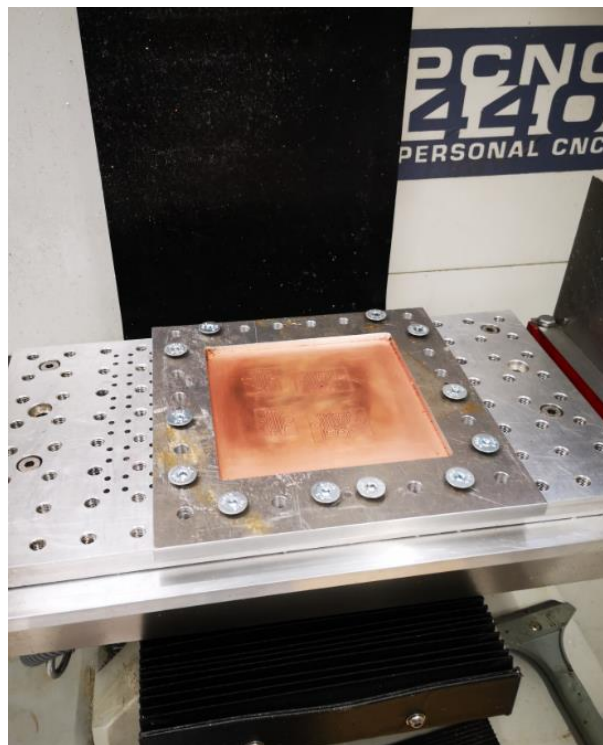
The following image shows my modifications to my original tooling plate frame. As an aside, if you use a tooling plate make sure the Z movement heights allow for the depth of the frame and any screw heads.
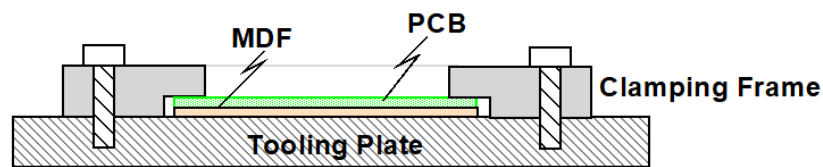
Original PCB/MDF sandwich
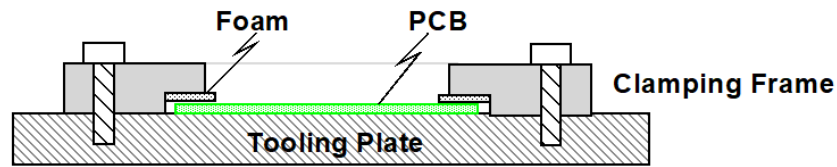
New PCB with foam rubber sandwich

With a clamping frame it is essential to make sure the mirror process is followed correctly to get a symmetrical flip or the frame will have to be moved to match the offset. Clearly the board blank must also be a tight fit in the frame if this is the only reference. Tooling pins are therefore still recommended.

Floating Foot

It is possible to buy or make a spring-loaded floating foot with ball bearing pressure points. This sits on the spindle and creates downwards pressure on the PCB blank around the area that the tool is cutting. I have 3D printed such a device for use on my CNCEST3040 mill and works very well. It does need an appreciable amount of blank (wasted) PCB material beyond and bordering the artwork milling area. It has ball transfer bearings on the lower surface which combined with the spring tension ensures the PCB is always pushed down flat around the area of the board being cut.

I have yet to conceive a version of this for my Tormach 440 spindle.

Software Compensation

There are various sources of software that will create a compensating adjustment of the Z cutting height.   The software requires a preliminary profile scan of the variations in height of the clamped PCB material. This is done before the actual milling takes place and the profile variation file is added as a compensation to the actual cutting Z height defined in the Gerber file.   I am told this can work very well but does extend the manufacturing process timescale and if care is not taken it could potentially cause problems with side 2 alignments.

Vacuum Table

Currently this is my favourite method. I modelled and machined an aluminium vacuum table using Fusion 360.   This is driven from a domestic vacuum cleaner acting as the suction source. The table has a working area of 160mm x 100mm and is made from 15mm cast aluminium. It has a 10mm x 10mm matrix of 6.8mm holes with 1mm diameter suction holes at the bottom of each. In operation it works very well with the PCB seen to 'jump' flat when the vacuum is turned on.

Here is the Fusion model and the actual finished table. This is showing the corner tooling pins and the central indexing mushroom tool mounted in the table and the secondary mushroom shaped tooling pins.
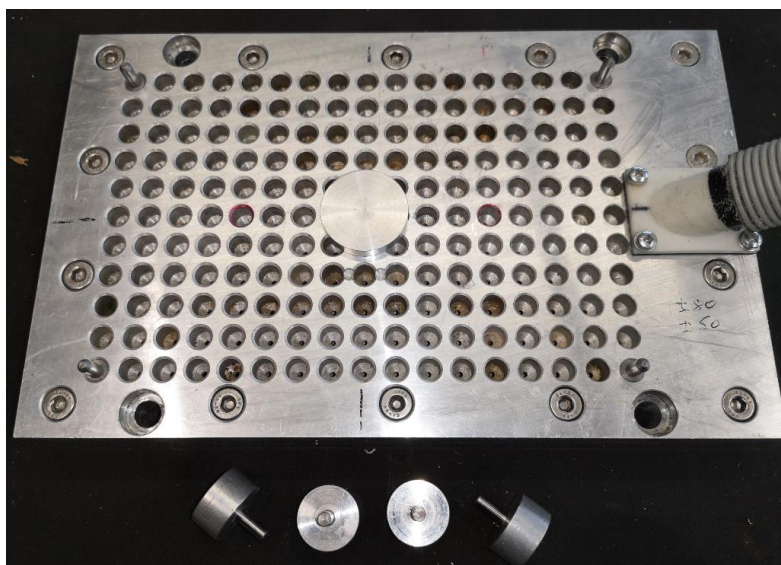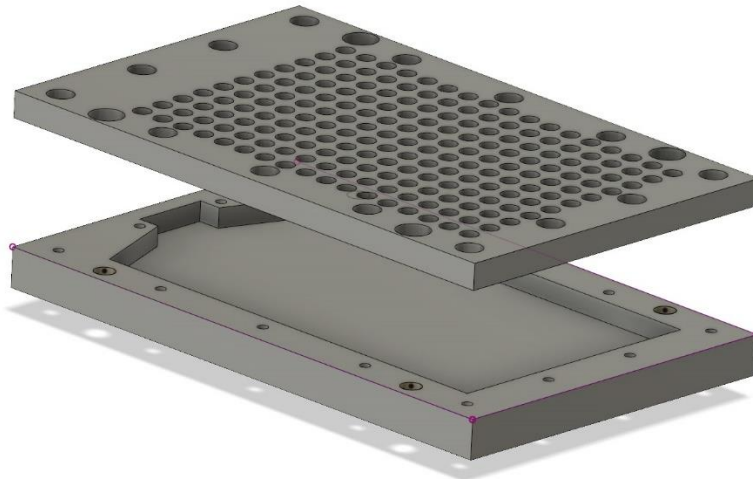
My tooling pins have a 6.8mm shank to fit into the vacuum plate matrix holes with 4mm protruding top section that engage with the sacrificial MDF layer and into the PCB. Despite what you might think, MDF is very transparent to air circulation.   The vacuum will therefore suck the PCB material down onto the vacuum table through the MDF. I use 3mm MDF.

My operations with the vacuum table can take two routes.

I can follow the FlatCAM process and have a sacrificial plate held in place with tooling pins in the plate corner holes with separate tooling holes in the PCB material that sit central in the vacuum holes and within the 100mm x 160mm operational area of the plate. These pin locations are defined in FlatCAM as I have described. This was how I conceived it would work. I then had one or two accidents with this method where I did not set the tooling hole location in FlatCAM to correctly coincide with the vacuum holes in the plate … much cursing.

I now depart from locating the tooling holes in FlatCAM. Instead, I have a standalone CNC program that is independent of FlatCAM that will drill 4 off x 4mm tooling holes in the PCB material and the MDF. These are located at the four extreme corner holes in the vacuum table hole matrix.   These sit centrally within the area of the 6.8mm vacuum hole but are drilled just to the depth of the PCB material and the MDF. The location of these holes is referenced to the central hole in the vacuum plate such as to be +/- 50mm in Y and +/-80mm in X.

This referencing needs to be accurate if the vacuum plate is not going to be damaged when drilling the tooling holes. To achieve this, I have a 'mushroom' that has a 6.8mm shank that sits in the vacuum table central hole. The top cap of the mushroom, which is 25mm diameter, is probed in X and Y to set the exact location of the centre hole.

Let's review this.   Four holes at 4mm diameter passing through the PCB and the MDF that are located in the most extreme holes in the vacuum table grid of holes. The tooling pins have a 6.8mm shank and a 4mm post.   This means the PCB and the MDF are locked together on the same pins.

If I lift the PCB up off the vacuum plate and the tooling pins, I can flip it over in a horizontal or vertical plane. In the horizontal flip the centre line of holes in the vacuum table are always going to be at the centre of the flip (the hinging line if you like to call it).   Similarly, a flip on the vertical line will also leave the hinging line in the strip of centrally placed vertical holes.

Providing my PCB has been exported with 0,0 as the centre point of the Profile then I can place the PCB centre point on the vacuum plate on any one of the central horizontal or vertical lines of holes.

The end result of this is I can pre-cut standard size pieces of PCB and MDF that completely cover the vacuum plate holes and have the standard 4mm holes already drilled at +/-80mm, +/-50mm from the centre point.

I can now 'move' the PCB geometry to have its central X0Y0 at any position on the horizontal or vertical centre line of holes. This makes the unused parts of the PCB reusable given appropriate geometry of a new board.

My vacuum table is very tolerant of uncovered holes not leading to a loss of vacuum. If the previous board milled has left a large opening in the PCB blank, I simply tape over any such apertures to regain full vacuum strength.

With regards to referencing, I only need to initially reference the centre hole on the vacuum table on the mill. This becomes 0,0 as the initial working coordinates. You could regard this as G54. I can then use the machine readout to step along left/right or up/down in 10mm steps along the cross axis of central holes to choose a new location for the next board's 0,0 position. Once chosen I then re-reference the spindle. I could be really clever and call the new position G55 but there might be something in the exported G code from FlatCAM that would override this back to G54 and disaster would strike.

If I do not use this method but instead have the PCB material separately clamped on the MDF then my tooling pins are also mushroom shaped. As I am only using 3mm MDF there is the potential for simple pins to sway and not remain vertical with respect to the PCB surface. This would lead to alignment errors between top and bottom artwork milling. By having mushroom shaped pins with a top hat section, they can be pushed down flat onto the PCB surface and maintain the vertical alignment.

Here are two images of the two setups. The first image shows the PCB tooling pins located via FlatCAM and independently positioned from the MDF tooling pins. The second image shows the PCB and MDF both using the same tooling pins where the tooling pin location is done via CNC and not FlatCAM.
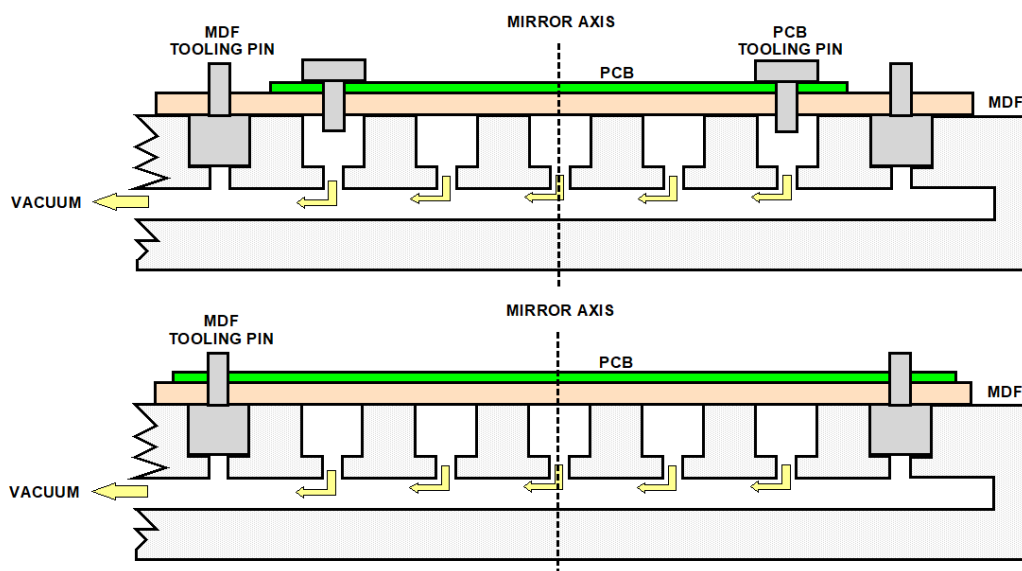
Coolant

My Tormach 440 has a FogBuster mist coolant system. Having this running with a very fine mist spray appears to help the milling process. The coolant must help lubricate the cutting tip and the air blast removes the swarf. The coolant dampens the swarf dust. The coolant also flows into the geometry cut traces and as a result these appear darker during the milling process and are more obvious to inspect as the milling progresses. Note however that the coolant will be absorbed by the MDF and cause it to swell. This will degrade the flatness of the MDF so it can only be used for one PCB run before disposal and certainly not left overnight to continue a run next day. An alternative manually applied lubricant could be a diluted washing up liquid dabbed on with a paint brush.