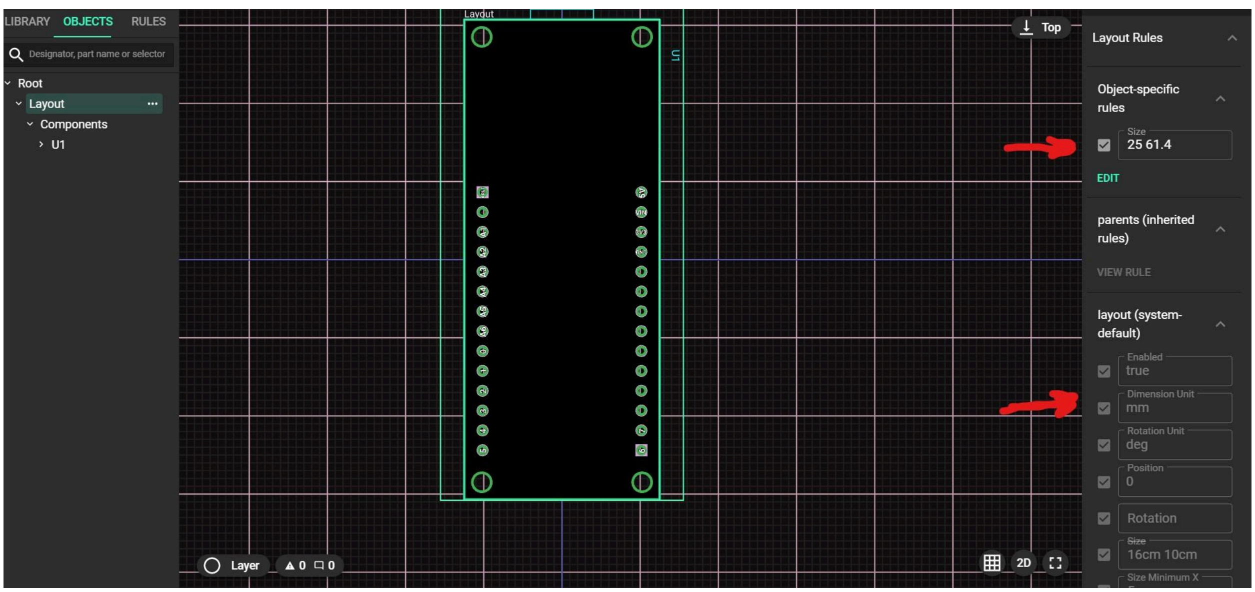
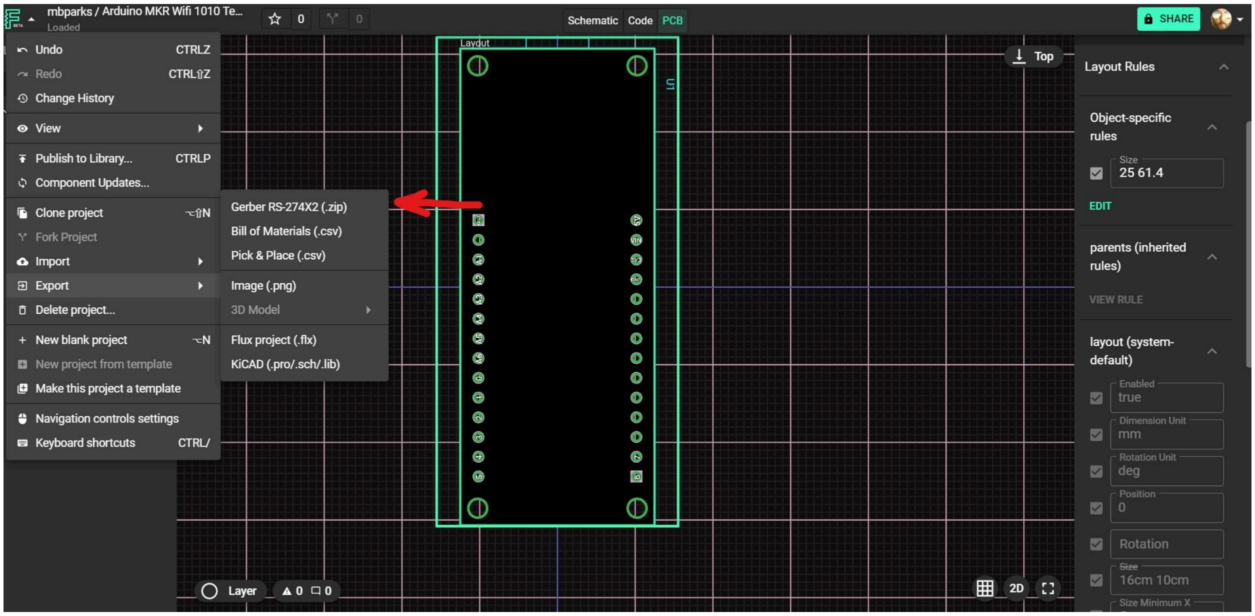
From *Flux.ai* to *CopperCAM* to *Makera Carvera* in 18 Easy Steps!

WARNING: These steps may or may not work for you based on your layout, workflow, or countless other variables. Consider this a template that you will need to tweak for your own purposes.

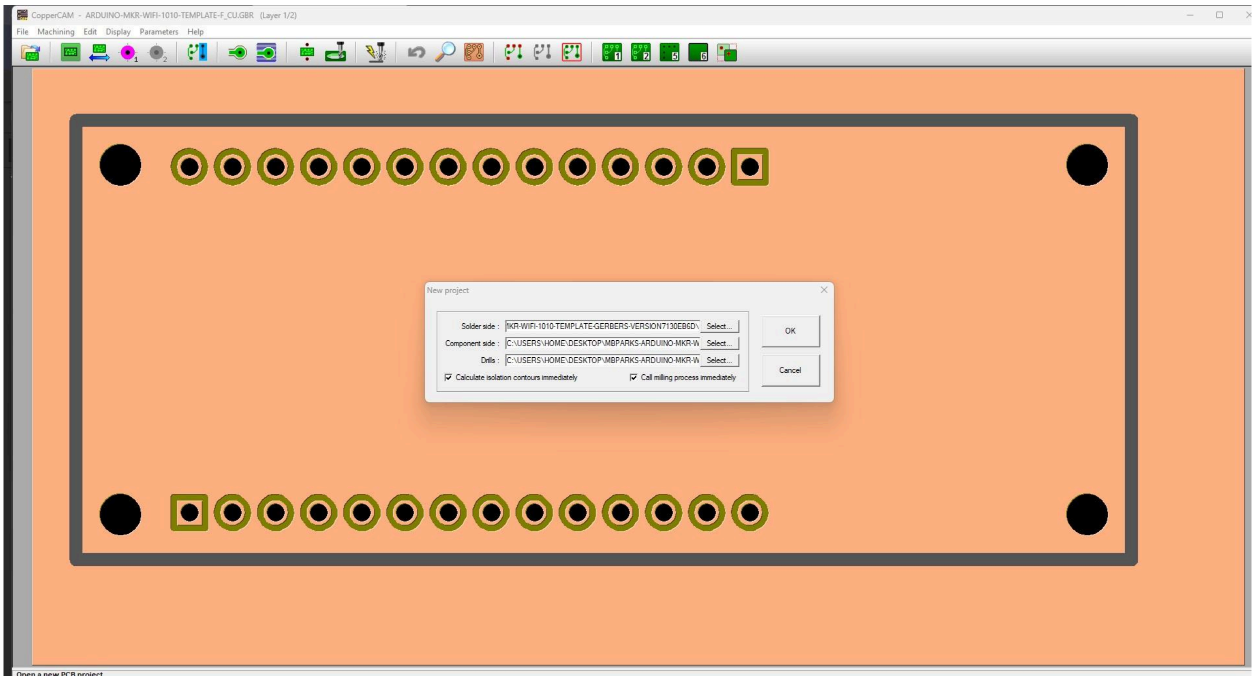
1) **Make sure your circuit board design in Flux.ai has "Size" and "Dimension Unit" attributes. Millimeters (mm) is a more universal unit measure, so go with that.**



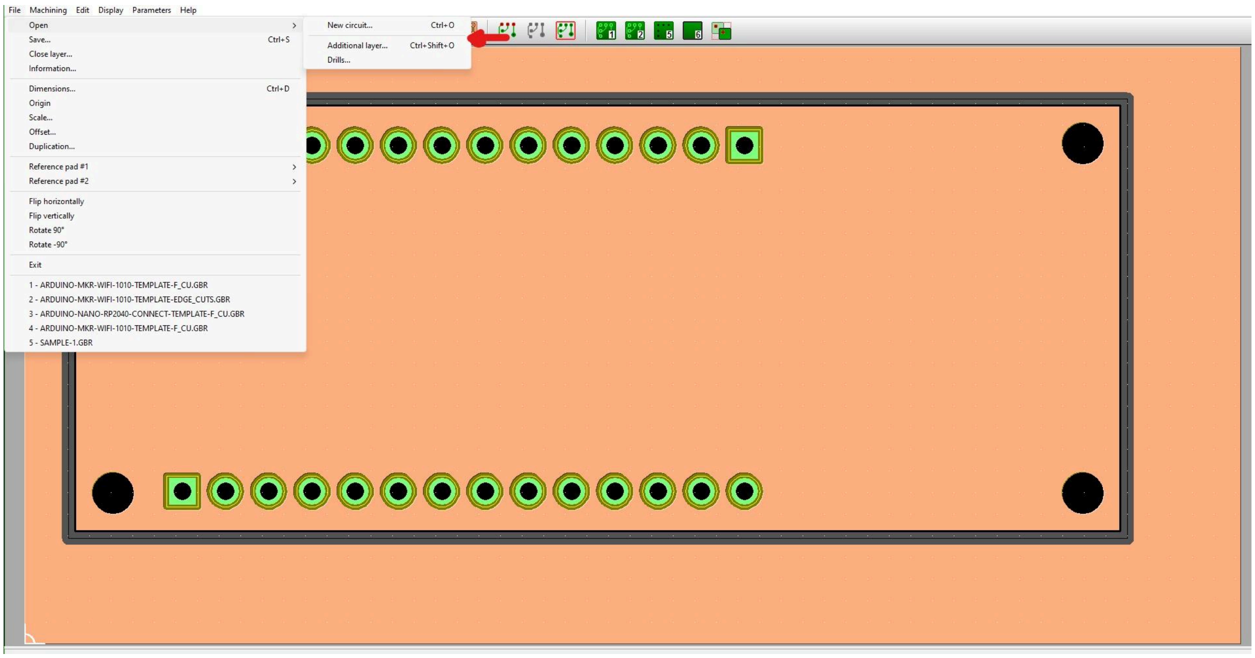
2) **Export your design from the Flux Menu>Export>Gerber RS-274X2(.zip) and download to your computer.**
    a) **Extract the ZIP file.**



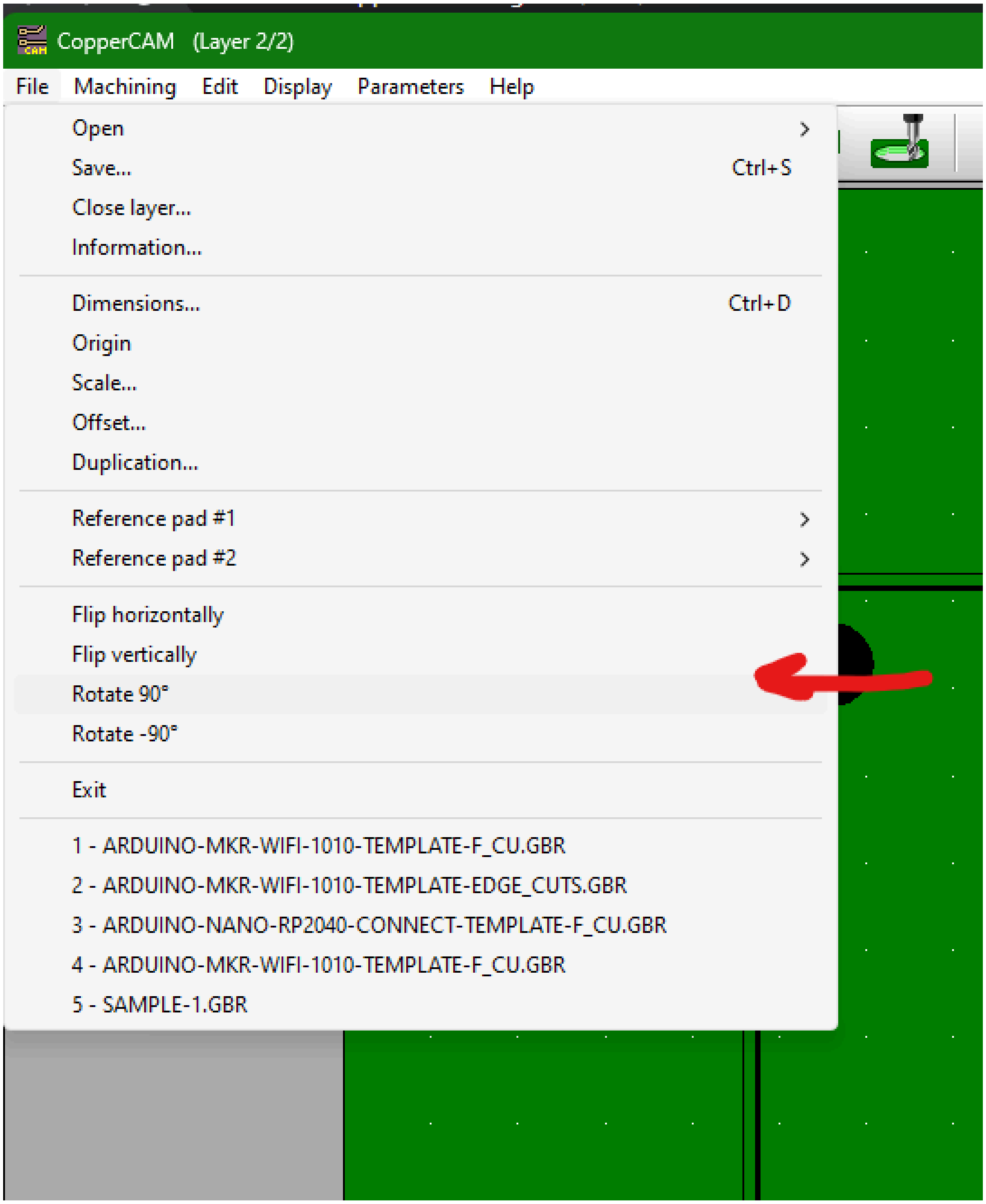
3) Open up CopperCam. Select File>Open>New Circuit. Navigate to the folder extracted from the ZIP file.
    a) Select the Solder Side, Component Side, and Drill file (.drl) as appropriate.
    b) The solder side and components side will likely be your copper files (end is *-f_cu.gbr* and *-b_cu.gbr*)



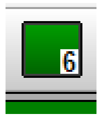
4) If you wish to have the Carvera cut the outline of the board, we need to add an additional layer.
    a) Click on File>Additional Layer
    b) Select the file ends with *-edge_cuts.gbr*.



5) Rotate the board, from the toolboard select File>Rotate 90



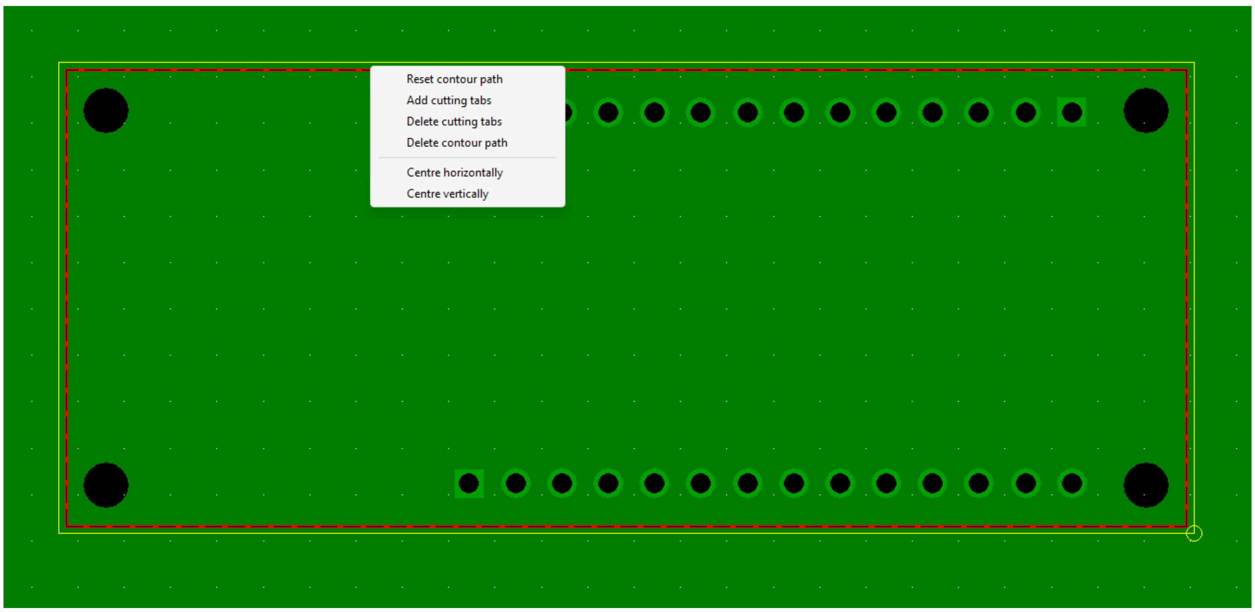
6) Align the cutout to the center of the layout.

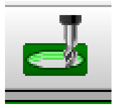
a) Press  on the toolbar

b) Right click the outline

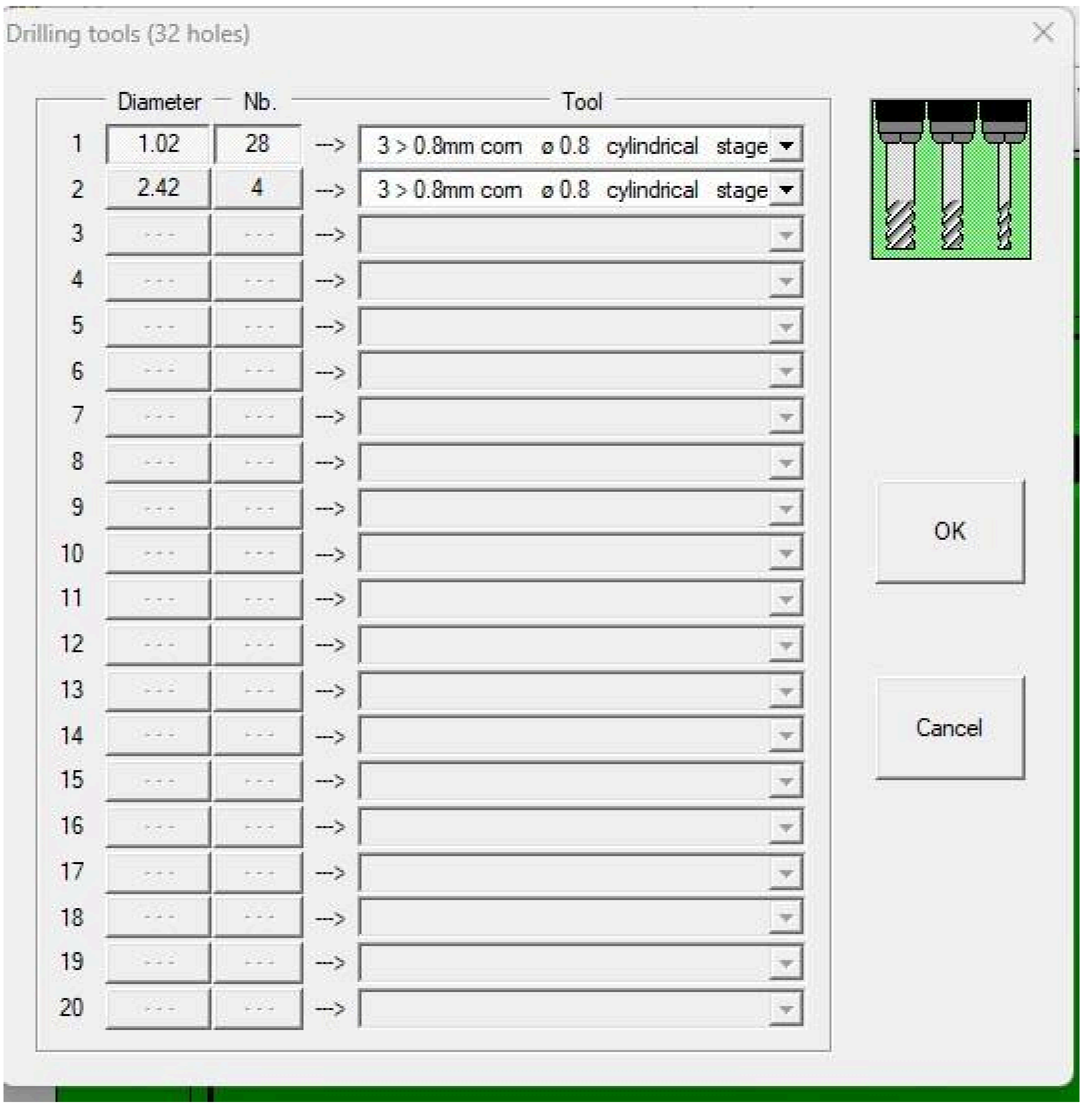
c) Select Centre Horizontally

d) Select Centre Vertically

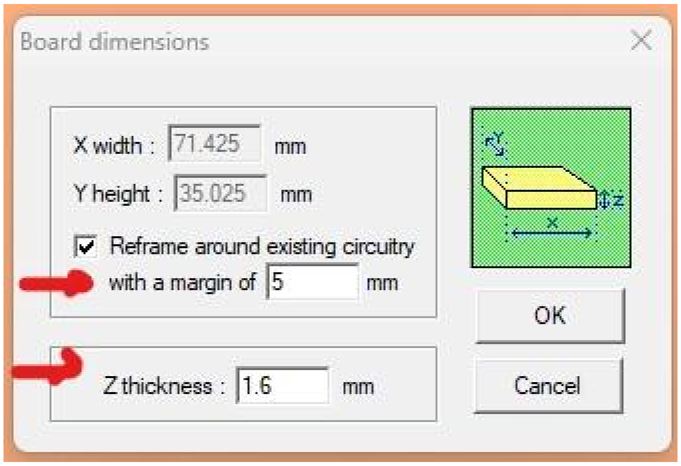


7) **Verify the holes to be drilled by selecting this icon from the toolbar:** 

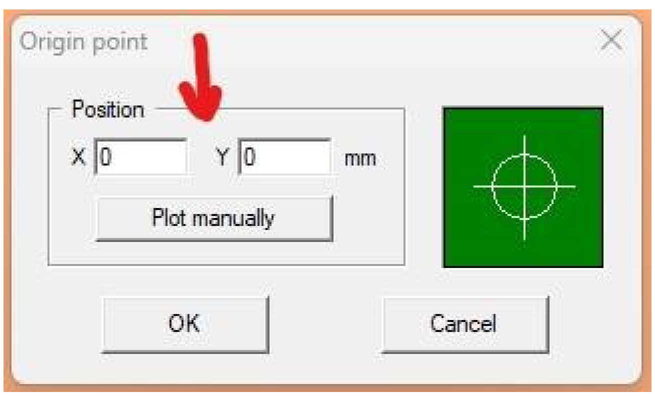
   a) **Hit cancel on any warning pops-up about hole vs drill size.**

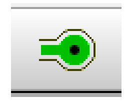


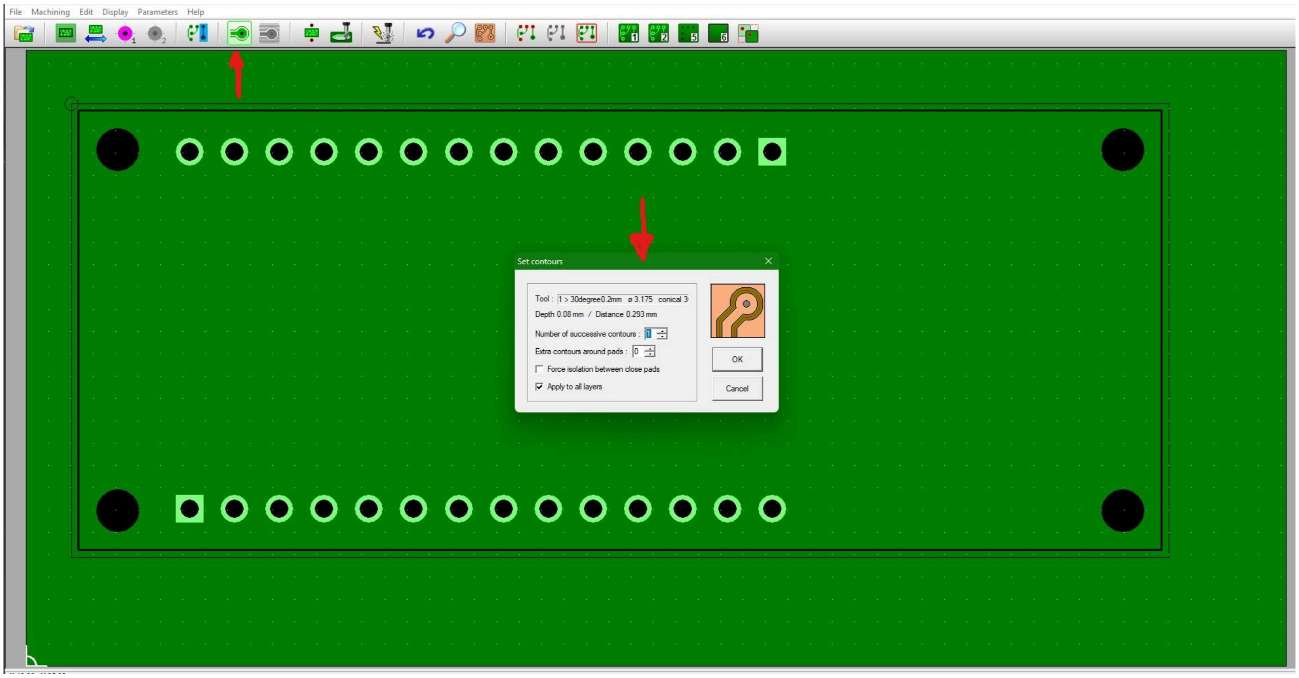
8) To give the machine some clearances whole milling, let's add some margins.
    a) Click on File>Dimensions and make the following changes.



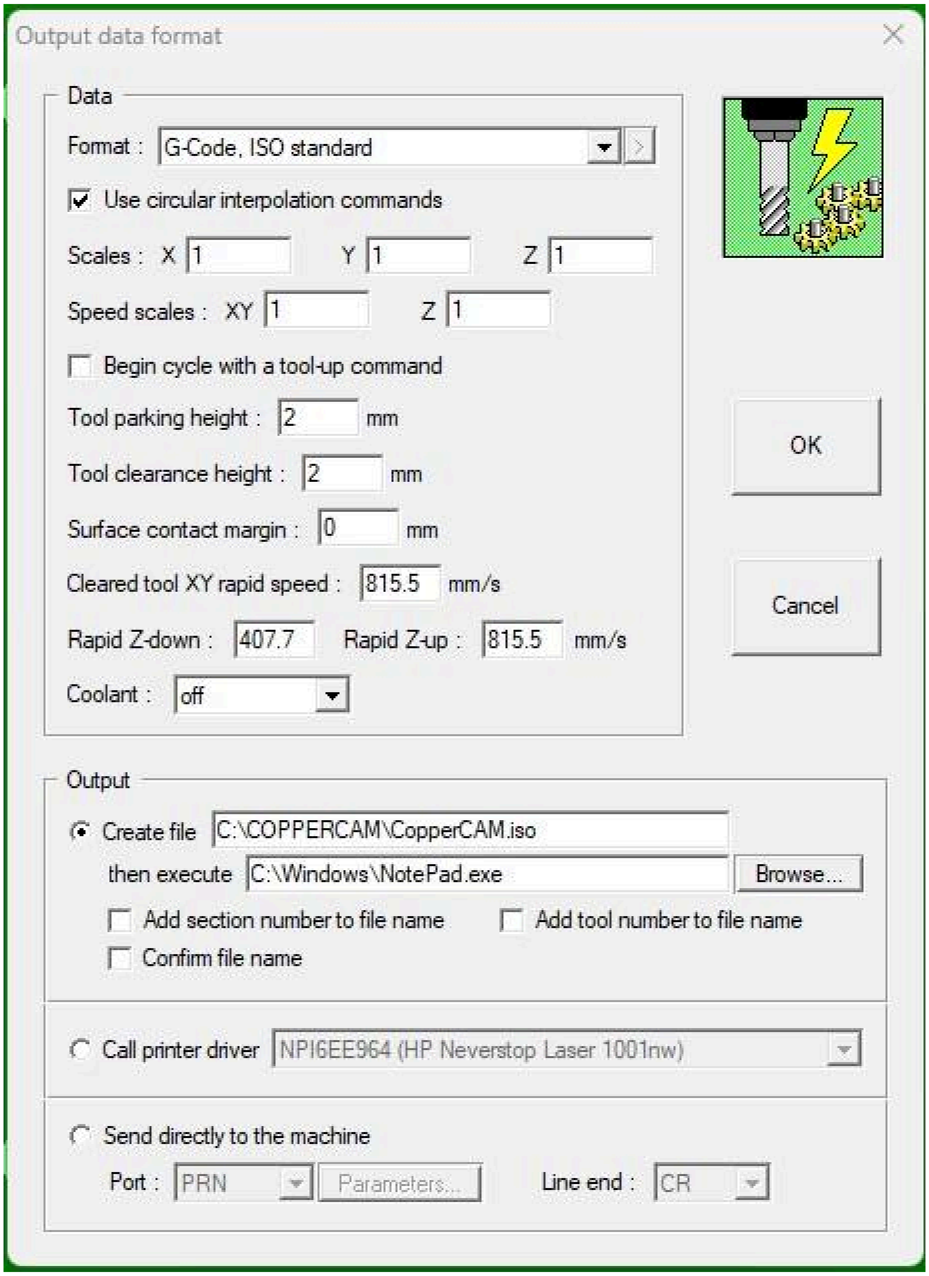
9) Let's set the origin to the bottom-left of the board.
    a) Click on File>Origin and make the following changes.



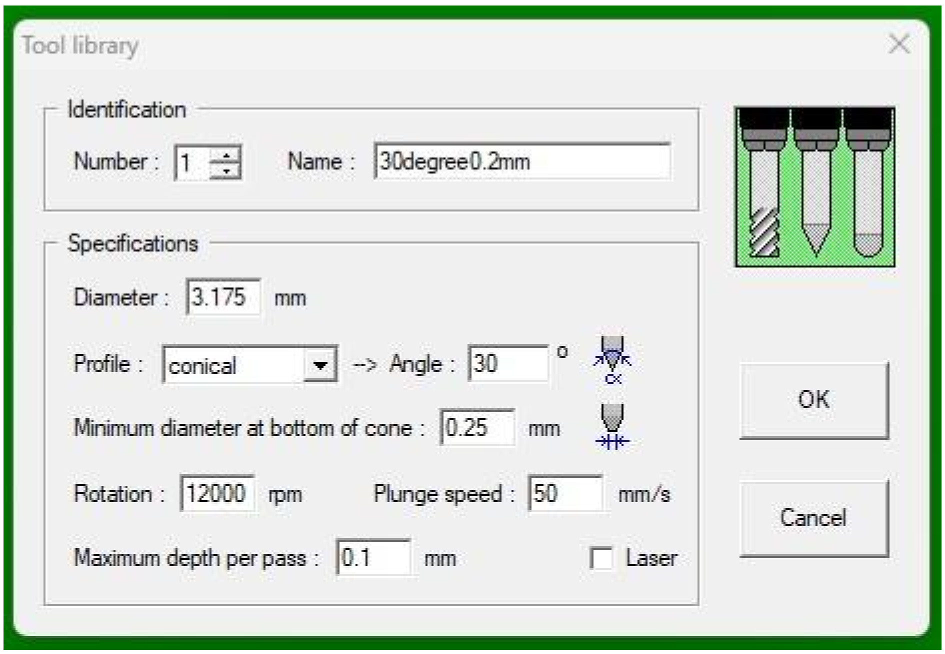
10) Set the contours of the milling by selecting this icon: 

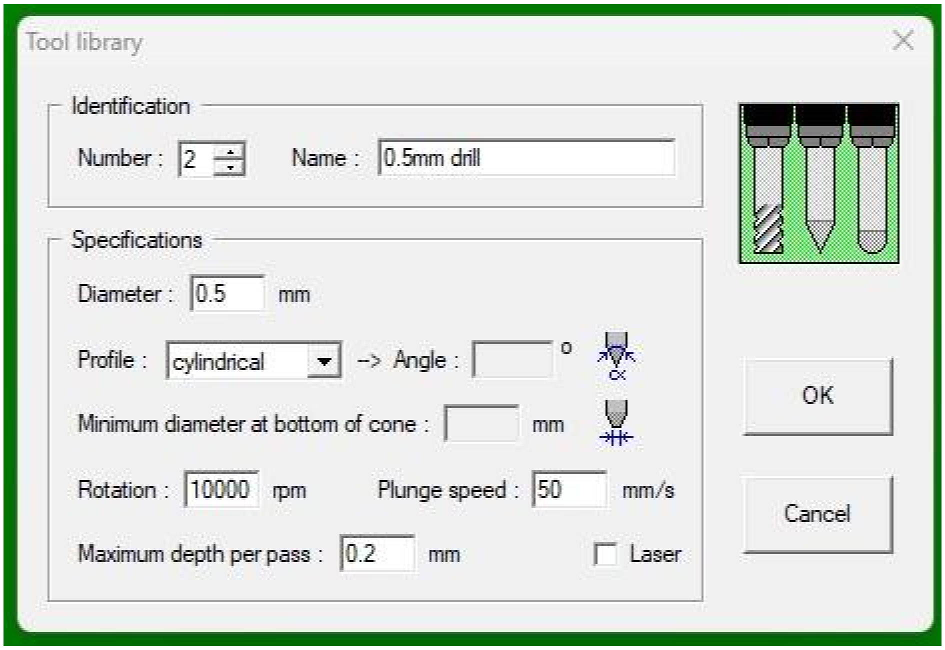


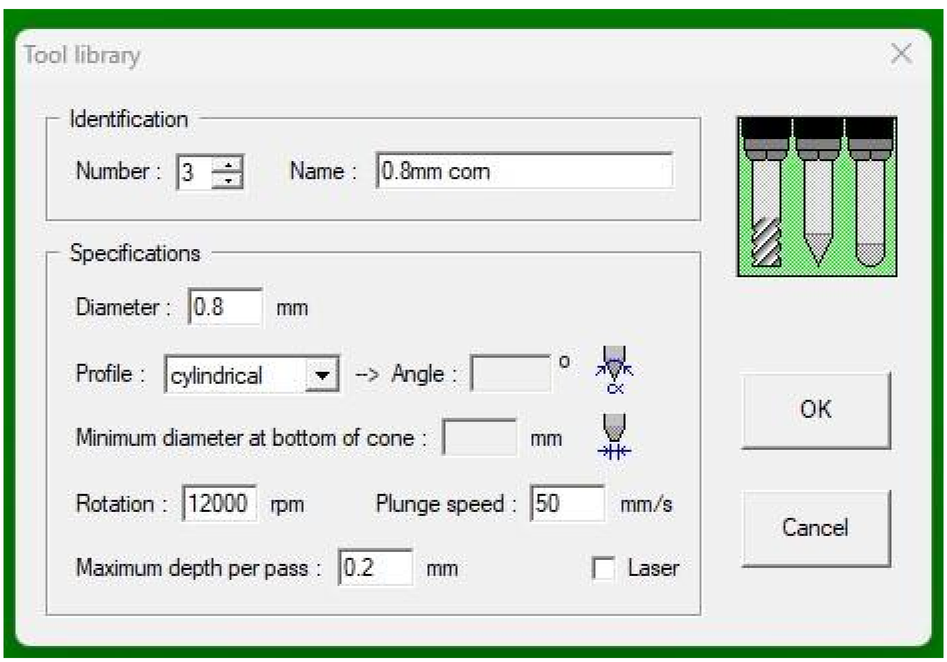
11) Now let's set the output file format. From the toolbar, select Parameters>Output Data Format. Make the following changes:

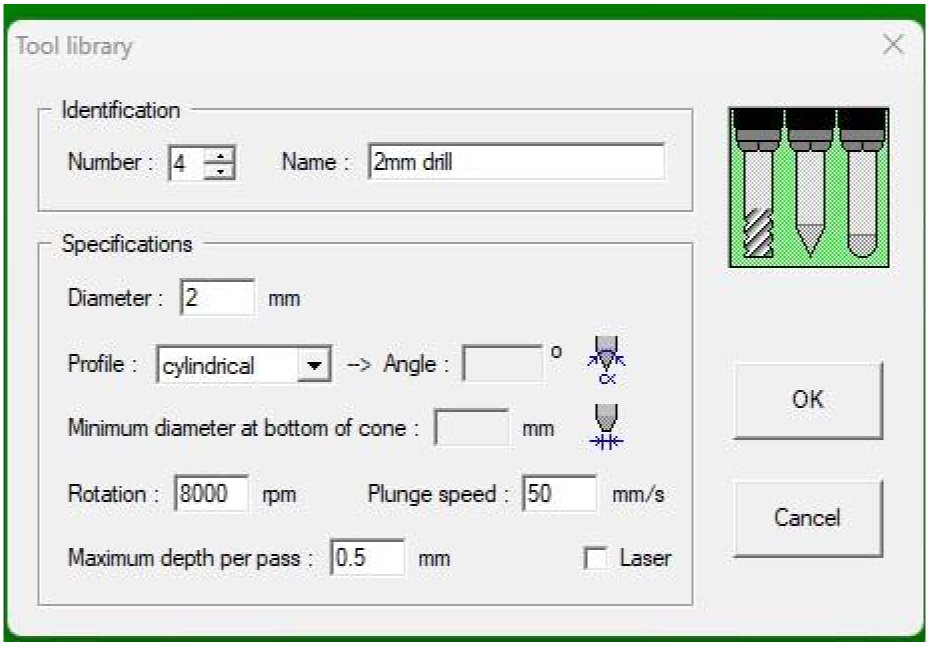


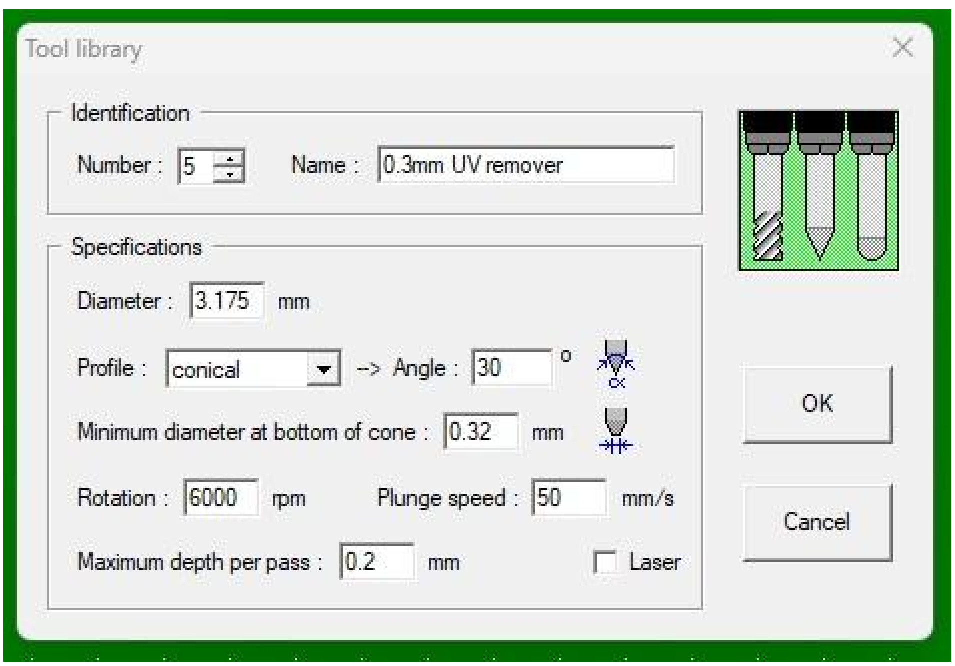
12) Let's set up our milling bits. For the Carvera, we have 5 milling bits to set up.
   a) To do so, from the toolbar, select Parameters>Tool Library.
   b) Walkthrough the five milling bits using the Number selection box.
   c) Set up each of the 5 bits with the following parameters.




Tool library
Identification
Number : 2
Name : 0.5mm drill
Specifications
Diameter : 0.5 mm
Profile : cylindrical
--> Angle : °
Minimum diameter at bottom of cone : mm
Rotation : 10000 rpm
Plunge speed : 50 mm/s
Maximum depth per pass : 0.2 mm
Laser
OK
Cancel


Tool library
Identification
Number : 3
Name : 0.8mm corn
Specifications
Diameter : 0.8 mm
Profile : cylindrical
--> Angle : °
Minimum diameter at bottom of cone : mm
Rotation : 12000 rpm
Plunge speed : 50 mm/s
Maximum depth per pass : 0.2 mm
Laser
OK
Cancel

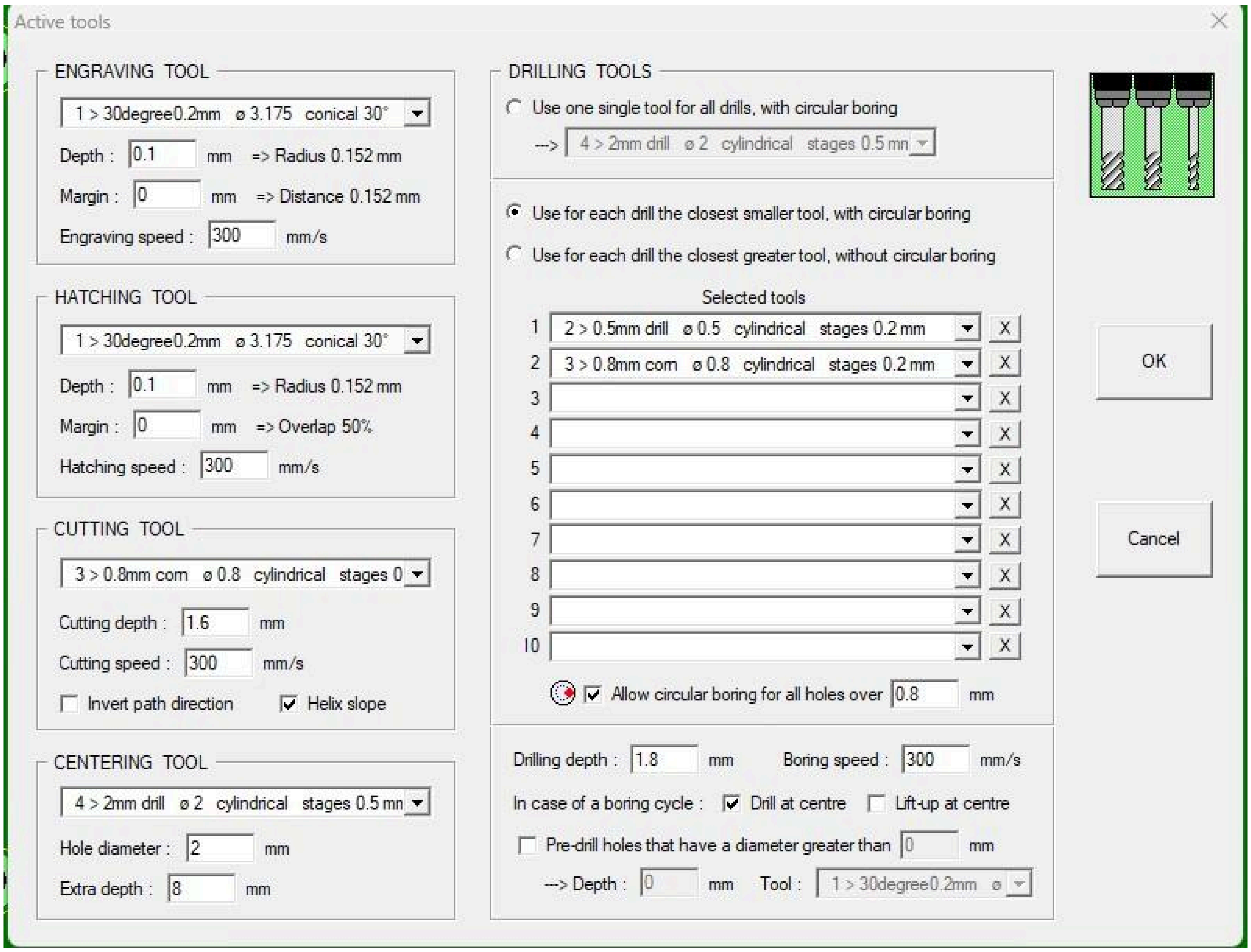

Tool library
Identification
Number : 4
Name : 2mm drill
Specifications
Diameter : 2 mm
Profile : cylindrical
--> Angle : o
Minimum diameter at bottom of cone : mm
Rotation : 8000 rpm
Plunge speed : 50 mm/s
Maximum depth per pass : 0.5 mm
Laser
OK
Cancel


Tool library
Identification
Number : 5
Name : 0.3mm UV remover
Specifications
Diameter : 3.175 mm
Profile : conical
--> Angle : 30 o
Minimum diameter at bottom of cone : 0.32 mm
Rotation : 6000 rpm
Plunge speed : 50 mm/s
Maximum depth per pass : 0.2 mm
Laser
OK
Cancel

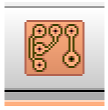
13) **With all the milling bits now in the library. Let's assign the different bits to the different passes.**

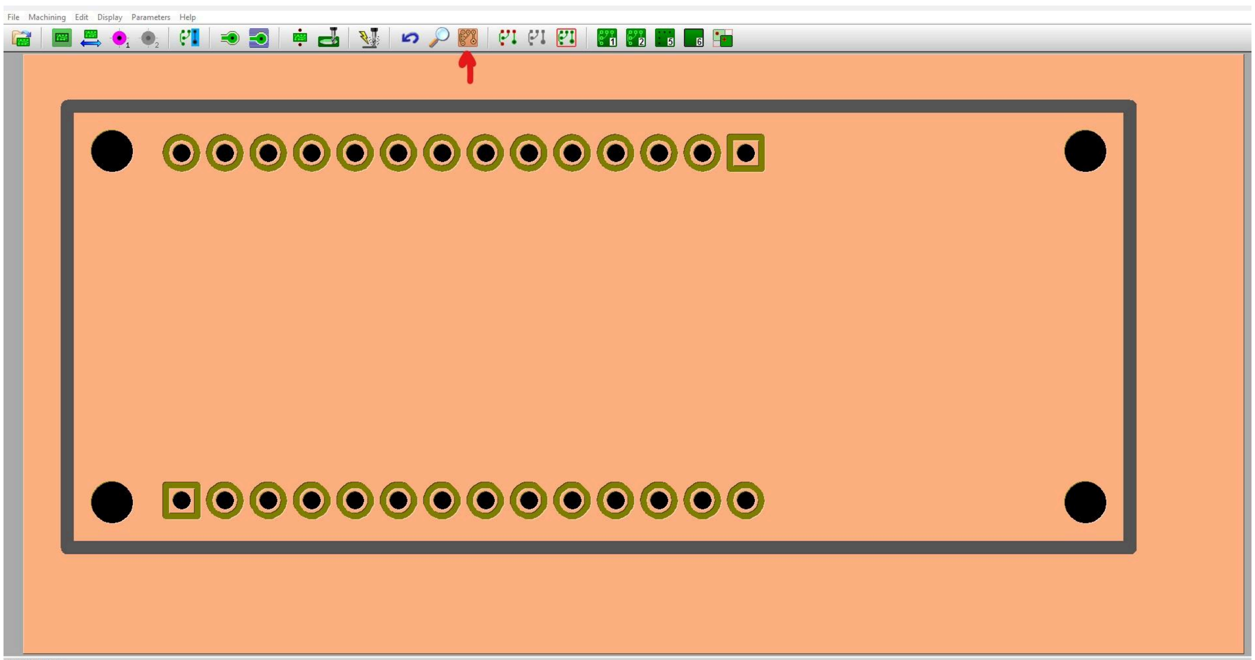
a) **From the toolbar, select Parameters>Selected Tools**

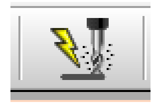
b) **Make the following changes.**



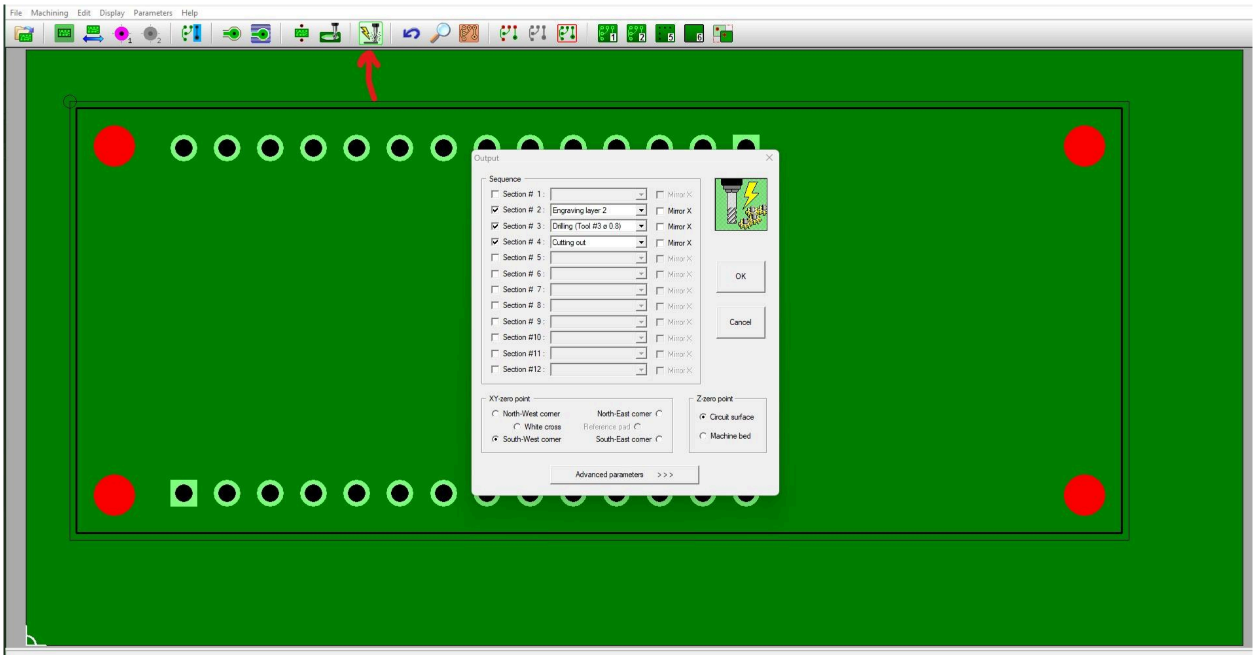
14) Verify that everything looks good by clicking on this icon:



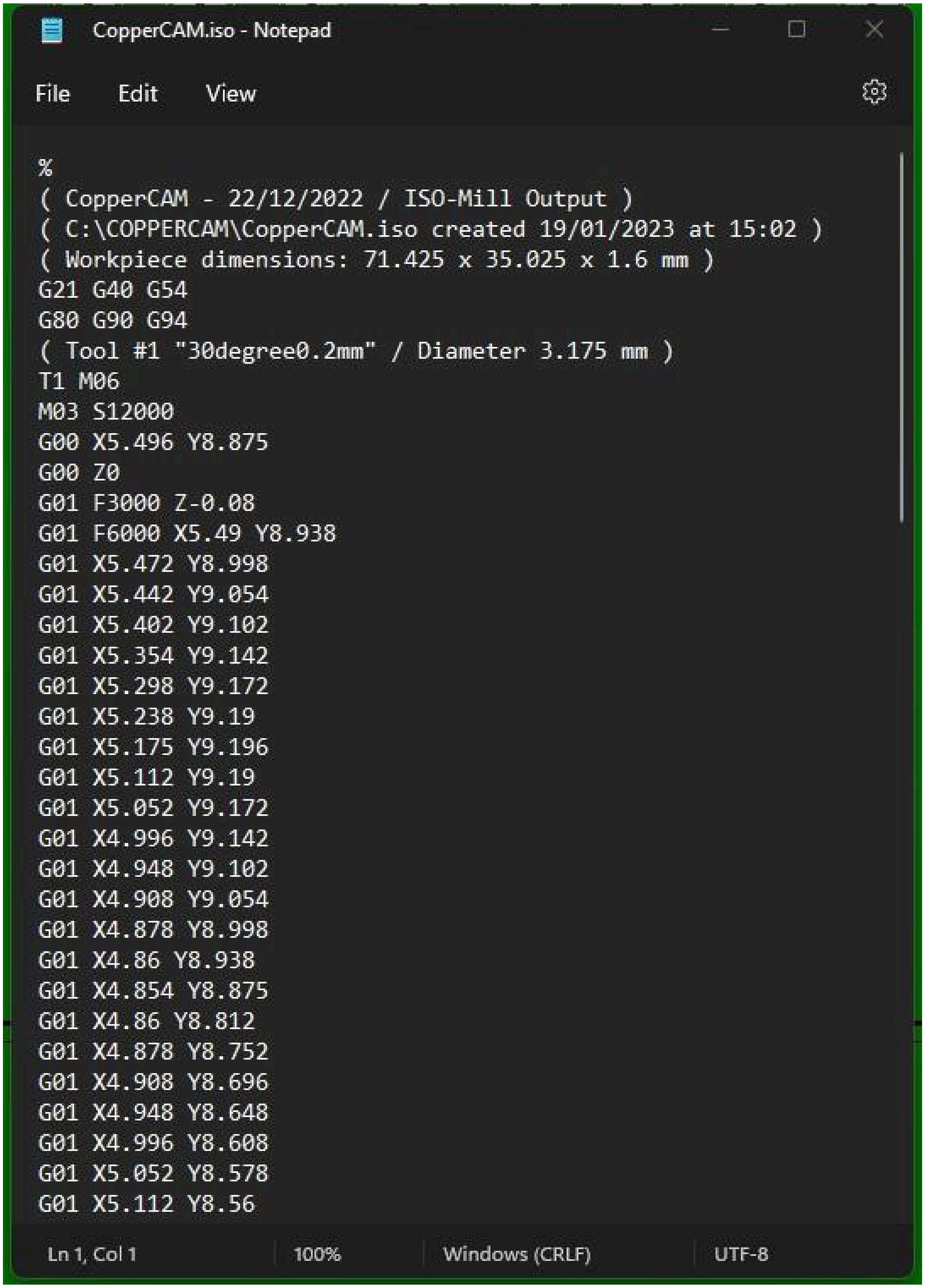


15) Generate the G-Code by selecting this icon: 

a) Verify that the correct tool is assigned to each section.



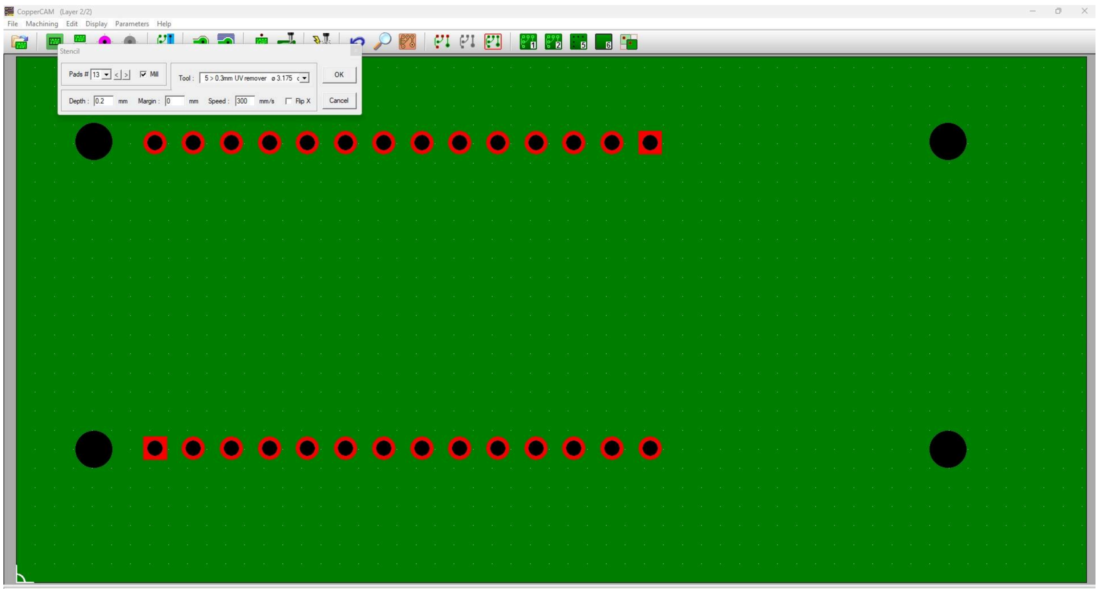
16) **Review the G-Code and save the file with the *.nc* filetype.**

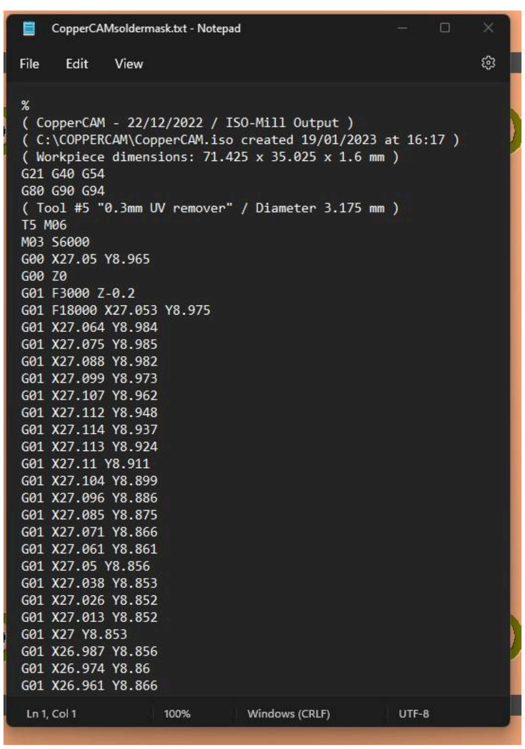


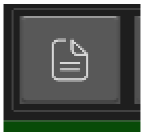

```
%
( CopperCAM - 22/12/2022 / ISO-Mill Output )
( C:\COPPERCAM\CopperCAM.iso created 19/01/2023 at 15:02 )
( Workpiece dimensions: 71.425 x 35.025 x 1.6 mm )
G21 G40 G54
G80 G90 G94
( Tool #1 "30degree0.2mm" / Diameter 3.175 mm )
T1 M06
M03 S12000
G00 X5.496 Y8.875
G00 Z0
G01 F3000 Z-0.08
G01 F6000 X5.49 Y8.938
G01 X5.472 Y8.998
G01 X5.442 Y9.054
G01 X5.402 Y9.102
G01 X5.354 Y9.142
G01 X5.298 Y9.172
G01 X5.238 Y9.19
G01 X5.175 Y9.196
G01 X5.112 Y9.19
G01 X5.052 Y9.172
G01 X4.996 Y9.142
G01 X4.948 Y9.102
G01 X4.908 Y9.054
G01 X4.878 Y8.998
G01 X4.86 Y8.938
G01 X4.854 Y8.875
G01 X4.86 Y8.812
G01 X4.878 Y8.752
G01 X4.908 Y8.696
G01 X4.948 Y8.648
G01 X4.996 Y8.608
G01 X5.052 Y8.578
G01 X5.112 Y8.56
```

17) If you wish to use the UV solder mask, it is necessary to generate a toolpath for the UV removal bit.

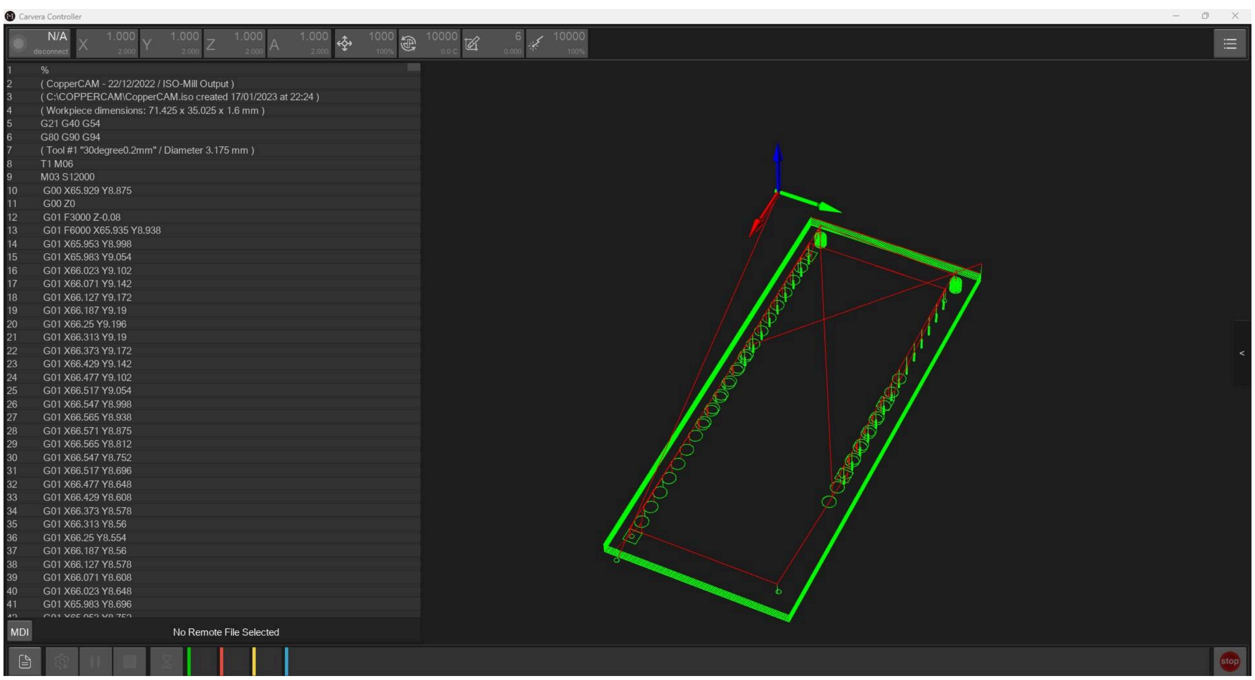
a) **To do so, from the toolbar select Machining>Mill Coating on Pads**
b) **Make the changes as shown below.**
c) **Hit OK to generate the G-Code and save a *.nc* file.**

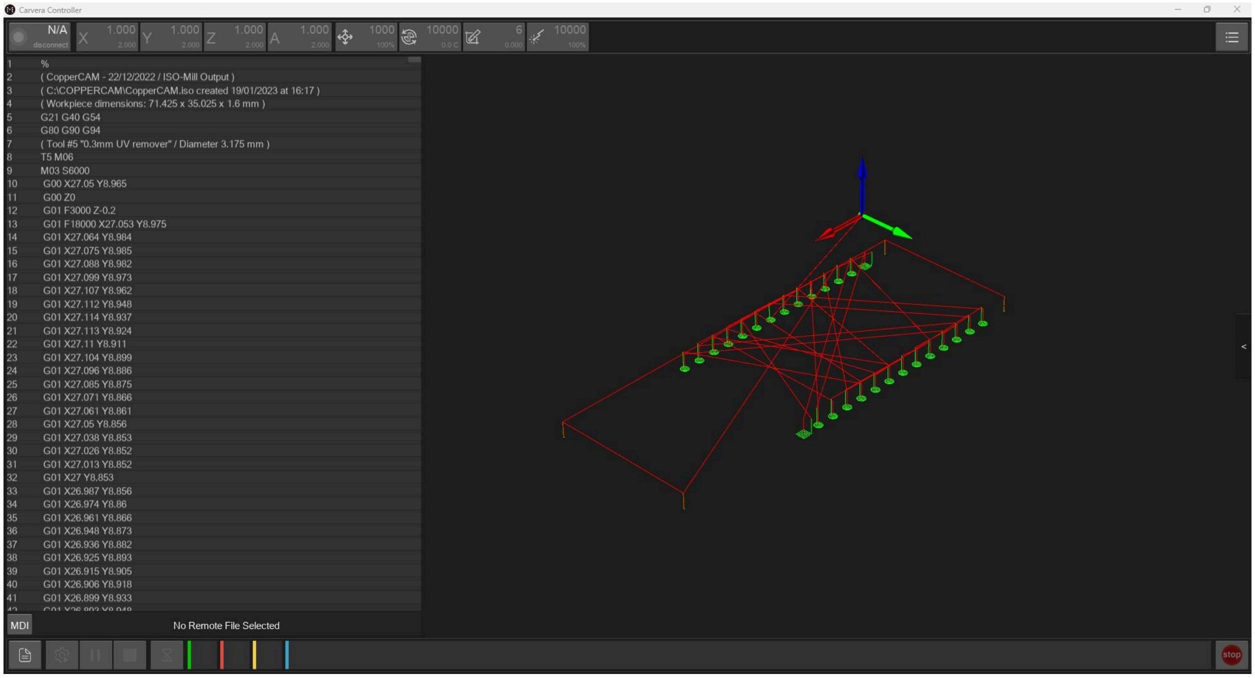




18) **Inside of Makera Controller app, select the file upload by pressing this icon:** 

a) **Navigate to the .nc files**
b) **Choose the file and press Select.**
c) **The milling should proceed in the following sequence:**
   i) **Mill pads and traces**
   ii) **Apply UV soldermask and cure.**
   iii) **Use the UV removal bit to remove the soldermask from the pads**
   iv) **Drill the holes**
   v) **Cut out the circuit board.**





Other PCB Design Rules of Thumb:

- **Leave 50 mils from board edge to a trace.**
- **Leave 100 mils from board edge to component.**
- **Make traces 10 mils wide per amp (e.g. 2A, make trace 20mils)**
- **Default 20 mil for signals and 30-40 mil for power traces.**
- **If milling circuit board, consider setting minimum width for traces at 40 mil (or 0.04”) to 70 mils (0.07”)**
- **1 via in parallel per amp**
- **Try to avoid right angle turns on traces, especially for high speed signals. 45* is ideal**
- **Don’t have ground plane near antennas**
- **PCB rivets: 0.4mm vias, 1.0mm PTH component**
- **IC footprints need to share the same alignment for the pick and place machine, pin 1 to top-left for example**

- **Make sure decoupling capacitors are placed close to the pins they are meant for**
- **Connectors should be placed on edges**
- **If you are using a two-layer circuit board, route one layer horizontally and the other vertically.**
- **Pitch between traces, make sure that there is a minimum gap of 0.007″ between items, 0.010″ (10 mils) is better.**
- **Make sure snap-to-grid is turned on. Usually, a value of 0.050″ for the snap grid is best for this job.**
- **Heat-sensitive electrolytic capacitors must be kept away from heat-generating diodes, resistors, and inductors.**
- **To avoid creating a nightmare of multiple ground connections and voltage drops, include a dedicated ground plane in your printed circuit board design. The ground plane could be a large layer of copper or, better yet, an entire plane on a multilayer board. With the ground plane in place, all you have to do is connect the components that require grounding to the plane via vias.**
- **To channel heat away from a component, simply route vias beneath it. These vias will effectively channel away unwanted heat from the component.**
- **Leave space between traces and mounting holes**
- **Position polarized parts (i.e. diodes, and electrolytic caps) with the positive leads all having the same orientation. Also use a square pad to mark the positive leads of these components.**
- **You will save a lot of time by leaving generous space between ICs for traces. Frequently the beginner runs out of room when routing traces. Leave 0.350″ – 0.500″ between ICs, for large ICs allow even more.**
- **After the components are placed, the next step is to lay the power and ground traces. Large traces feeding from a single rail are used for the positive supply. To avoid creating a nightmare of multiple ground connections and voltage drops, include a dedicated ground plane in your printed circuit board design. The ground plane could be a large layer of copper or, better yet, an**

entire plane on a multilayer board. With the ground plane in place, all you have to do is connect the components that require grounding to the plane via vias.