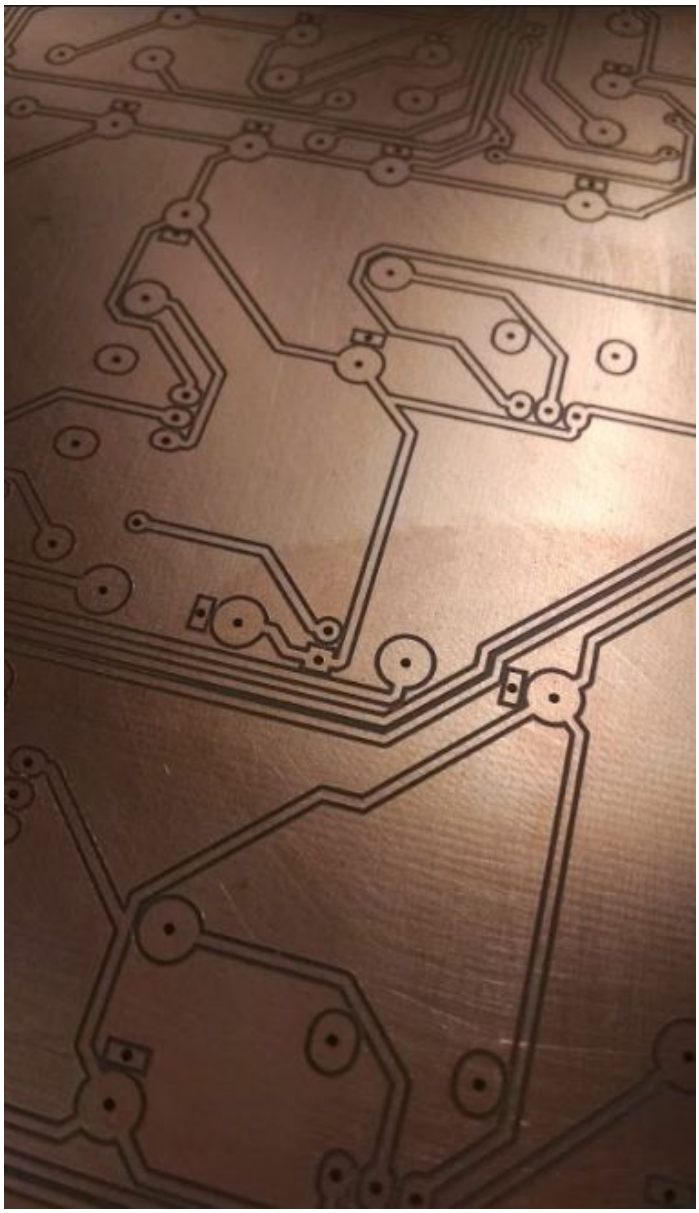
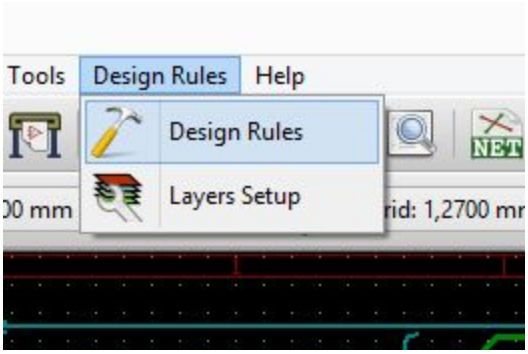
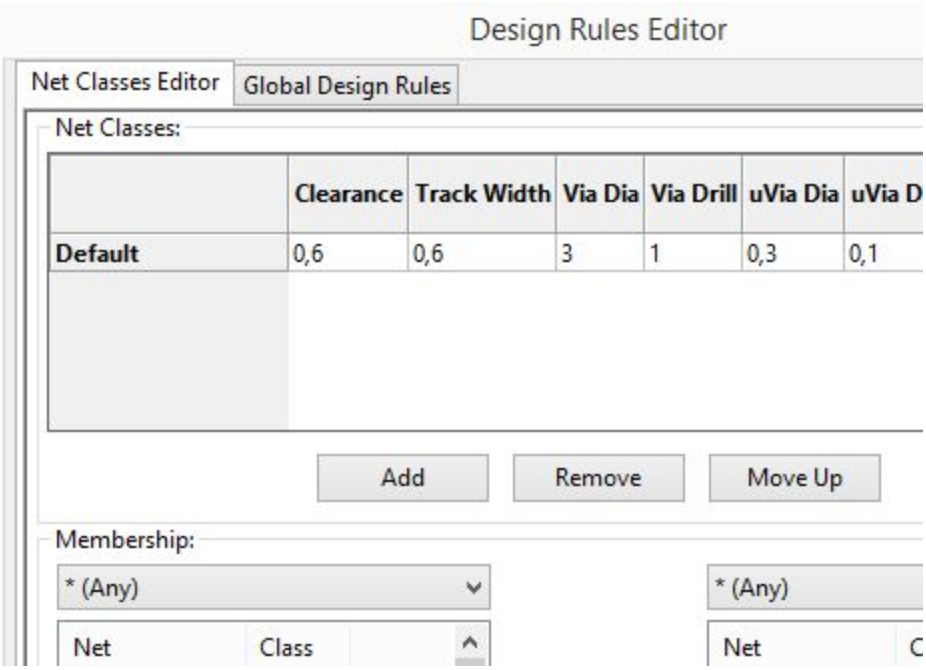
Single Layer PCB routing and drilling using KiCad and Alphacam



Makerlab
May 2017

Before starting any cutting, make sure your design will work with the CNC:
Check the design rules in the top kicad menu bar.
Design rules: I've used 0.6mm clearance and 0.6mm track width and gotten great results.



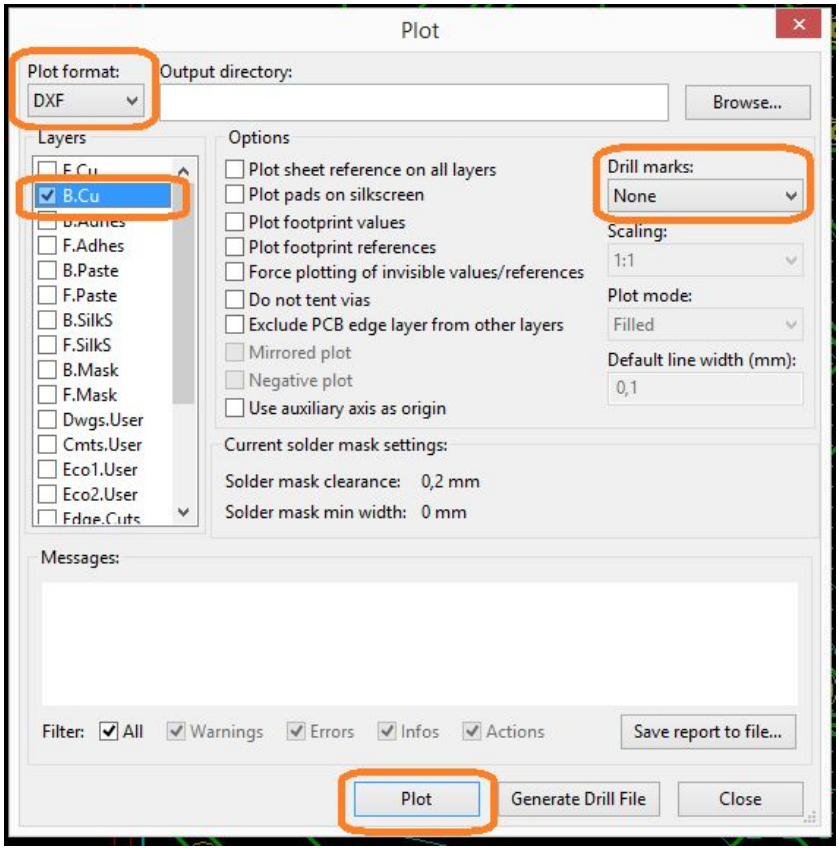


Avoid ground planes / copper fills. They don't follow the design rules and make routing much more complicated (in my experience).

-Run design check in KiCad
Never hurts to run a quick design check before sending your pcb off from KiCad.

We're going to split our operation into two steps. First we'll do the cutting of traces, then we'll drill all the holes.

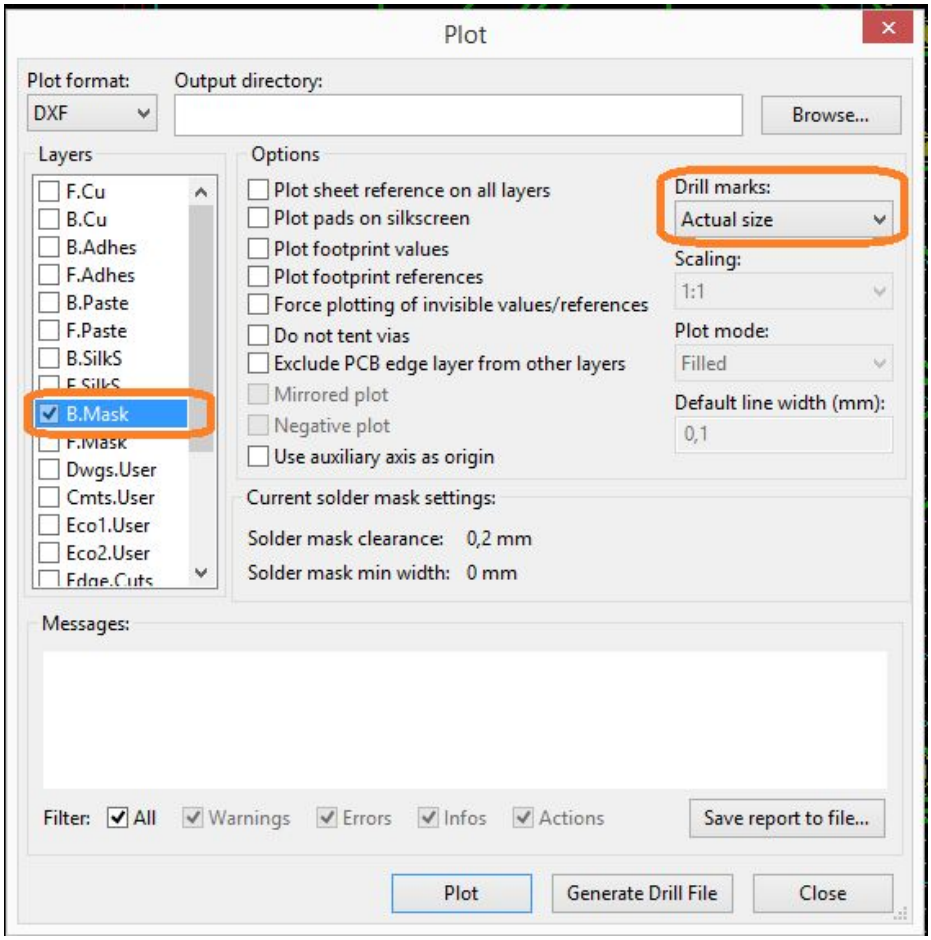
File - Plot - DXF
Exporting as a DXF will keep all measurements exact. The problem with exporting as a pdf or any raster formats is that you'll have to carefully measure and scale later - dxf avoids all that.



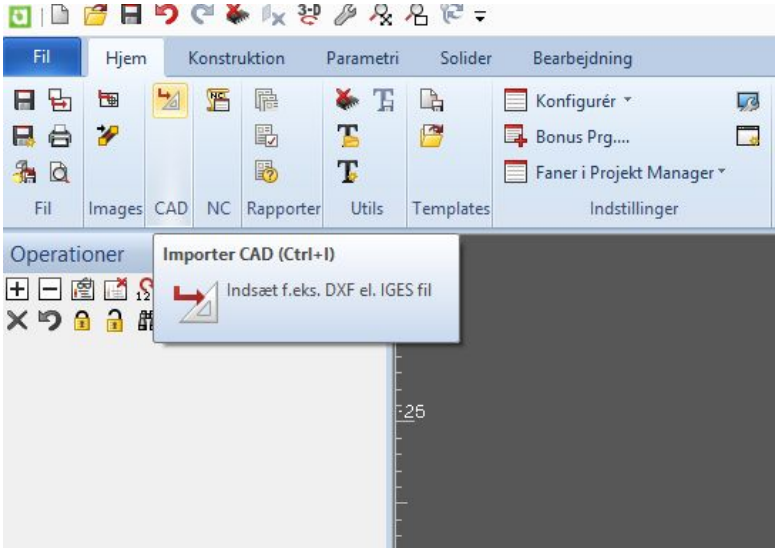
Export bottom copper layer as dxf. Drill marks - none.

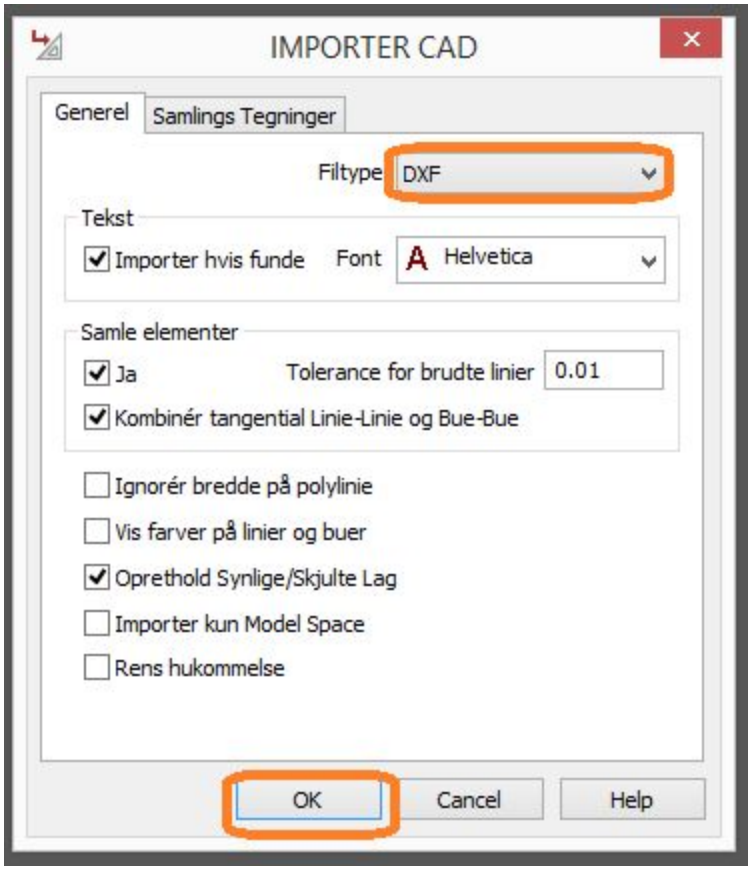
Next, export holes:
Export extra layer (i used ‘mask’ layer that was pretty much empty) as dxf with drill holes - actual size.[1]

[1]Using vias? I use them as jumpers (instead of cutting double-layer boards). I’m getting them imported in the copper layer. I need to manually select them and move them out of the board area first. Note that they import to alphacam as perfect circles.

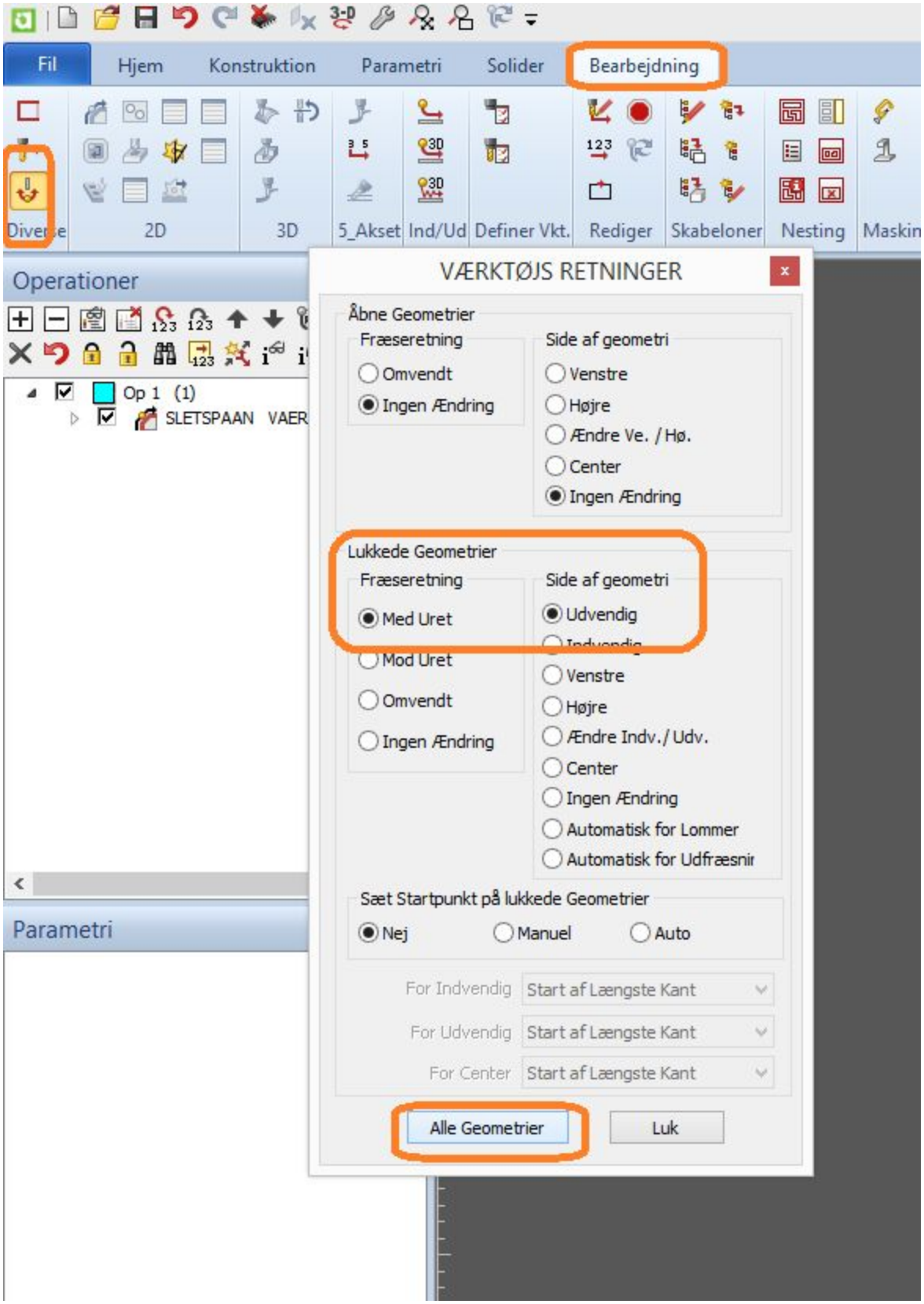


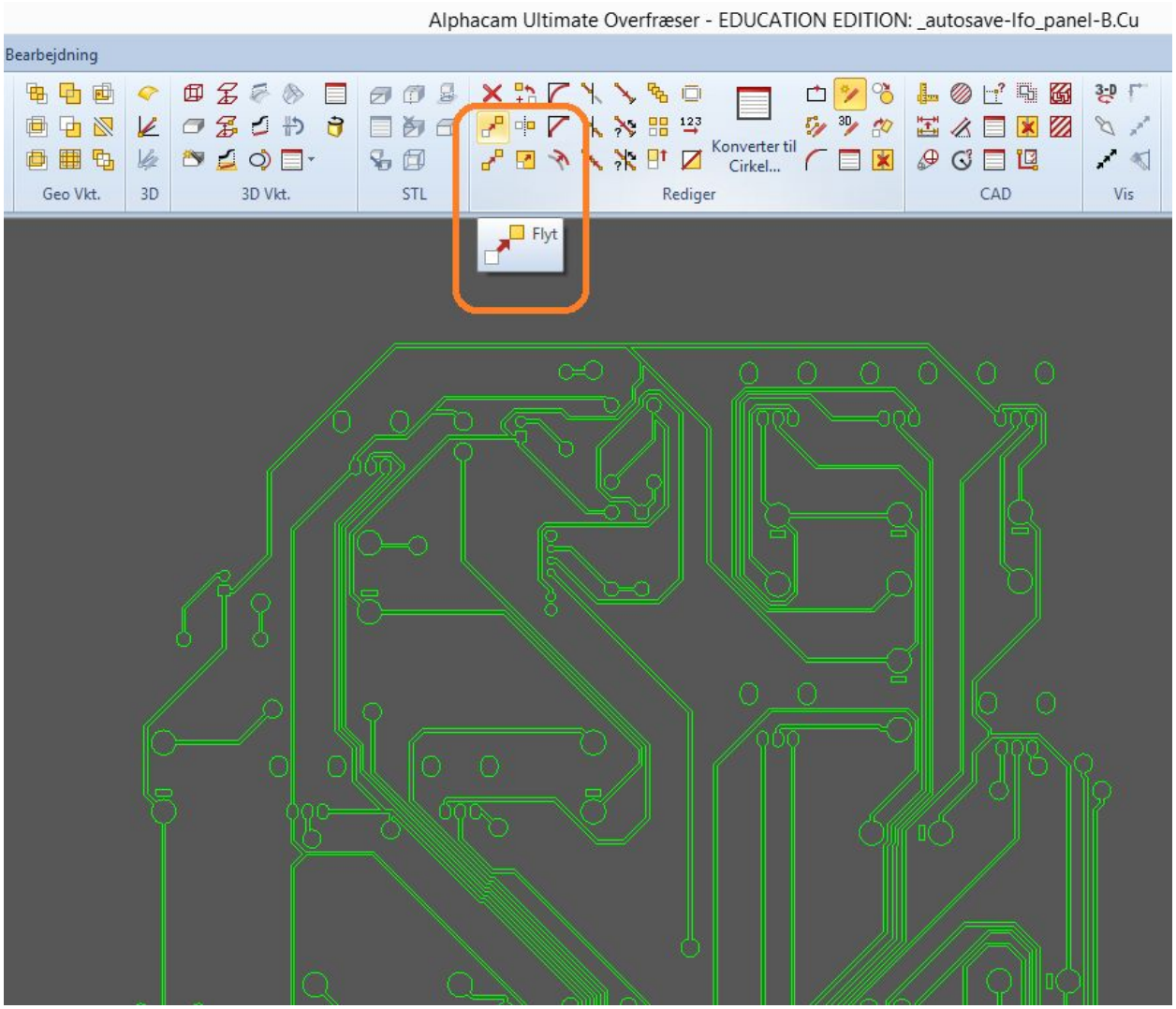
In a new alphacam file, go to Hjem - CAD - Importer CAD. In the type menu, select DXF and your first file.



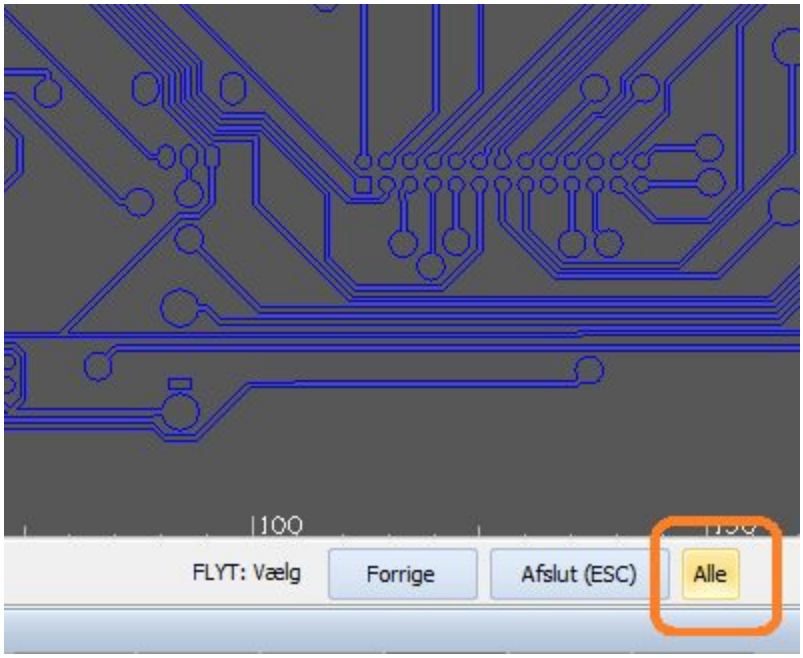


Our smallest traces in the DXF should be 0.6mm. We're going to want the cutting tool to follow the outside of our lines instead of the center of the lines to keep these measurements.

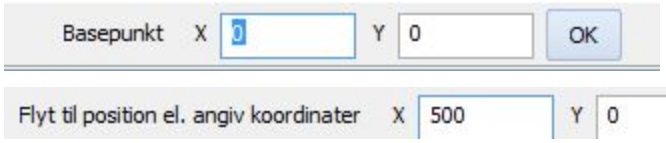

Fil
Hjem
Konstruktion
Parametri
Solider
Bearbejdning
Diverse
2D
3D
5_Akset
Ind/Ud
Definer Vkt.
Rediger
Skabeloner
Nesting
Maskin
Operationer
Op 1 (1)
SLETSPAAN VAER
Parametri
VÆRKTØJS RETNINGER
Åbne Geometrier
Fræseretning
Omvendt
Ingen Ændring
Side af geometri
Venstre
Højre
Ændre Ve. / Hø.
Center
Ingen Ændring
Lukkede Geometrier
Fræseretning
Med Uret
Mod Uret
Omvendt
Ingen Ændring
Side af geometri
Udvendig
Indvendig
Venstre
Højre
Ændre Indv./ Udv.
Center
Ingen Ændring
Automatisk for Lommer
Automatisk for Udfræsnir
Sæt Startpunkt på lukkede Geometrier
Nej
Manuel
Auto
For Indvendig
Start af Længste Kant
For Udvendig
Start af Længste Kant
For Center
Start af Længste Kant
Alle Geometrier
Luk



Move this to the side so we can import the holes while keeping them separate.



Flyt - select all - 0 to 500 X, for example.



Repeat the import steps for the second file. (note - if you try to drag-and-drop the second file, it will open it as a new file rather than adding it)

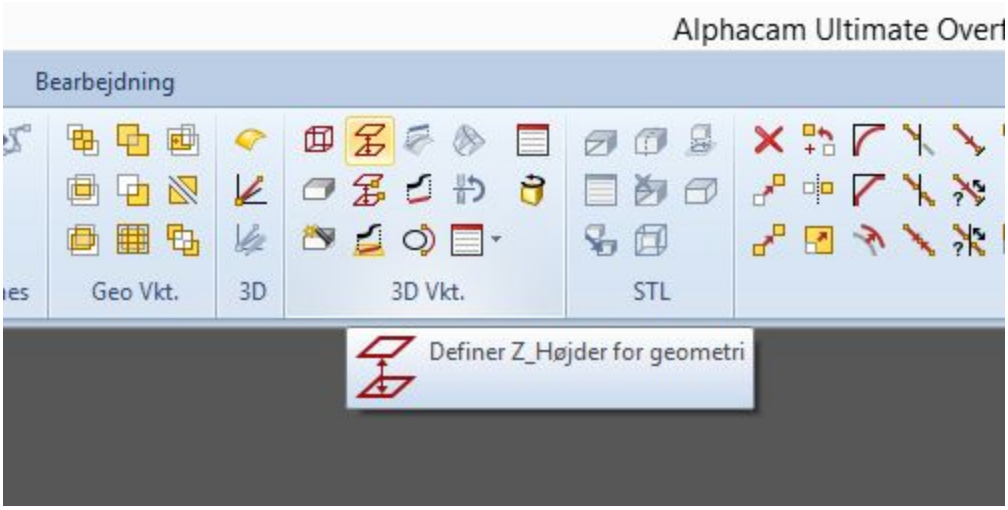
Now we have our two operations separate and can deal with them individually, then combine them.

When alphacam imports the drill holes from a DXF, they're not perfect circles. Alphacam needs circles for the drill function, so use alphacam function "convert to circles" on all holes.

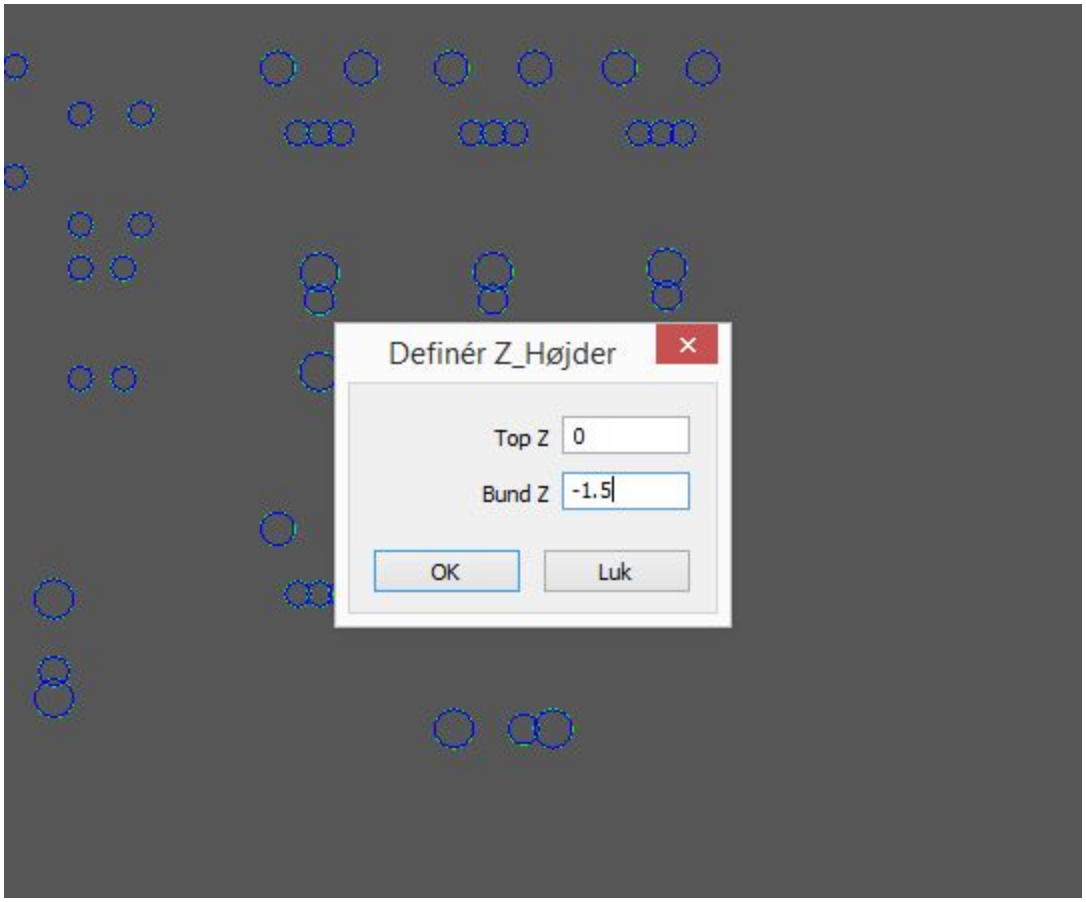
-Note that if you have oval holes, it seems to do a good job on selecting the center point.

Tolerance of 99 works fine.

Set the depth for all holes. Use “Define Z-Height” operation and set this number as the thickness of the PCB, as a negative number, in millimeters.

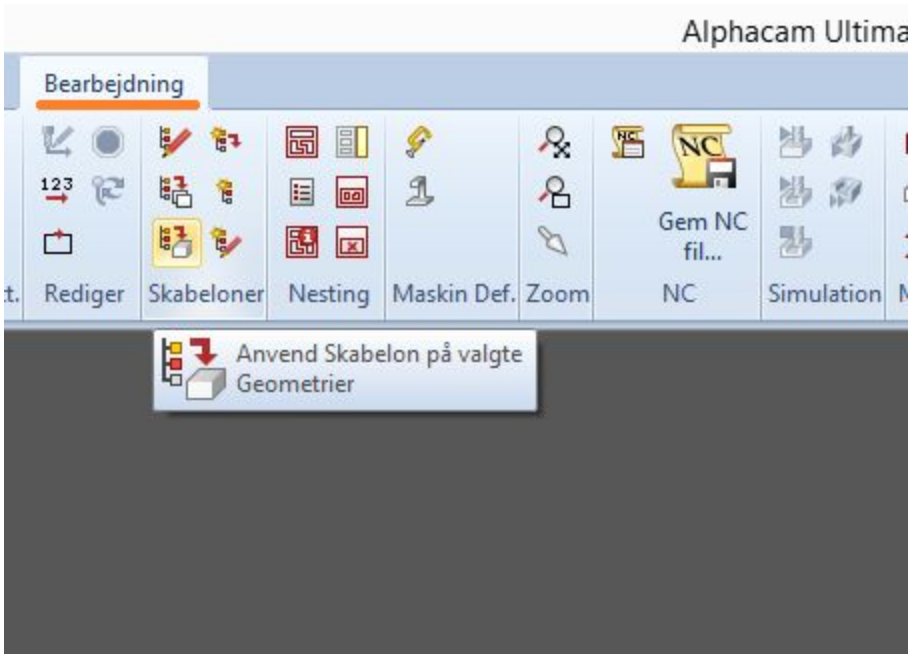


Draw a box around all your holes and right click.



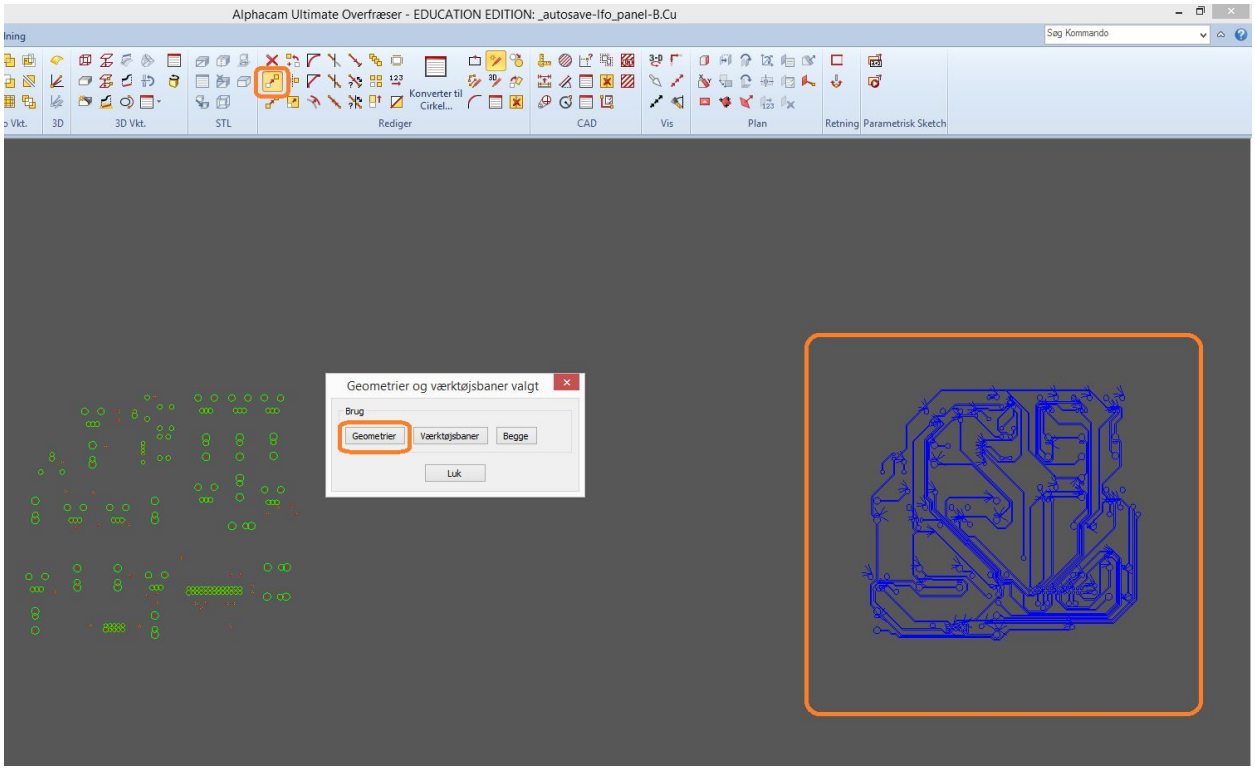
Example for 1.5mm thick PCB.

I’ve set up templates (skabeloner) for PCB trace routing and hole drilling after some trial-and-error. These are the settings that work the best for me and I’ve found them to be reliable.



Pick skabelon “PCB routing”, select all the traces, and right click to confirm selection. The proper toolpaths should be generated and visible as blue lines.

Use skabelon “PCB drilling” along with the desired drill hole size for holes - understanding that some components need different hole sizes. Select the desired holes and right click to confirm.

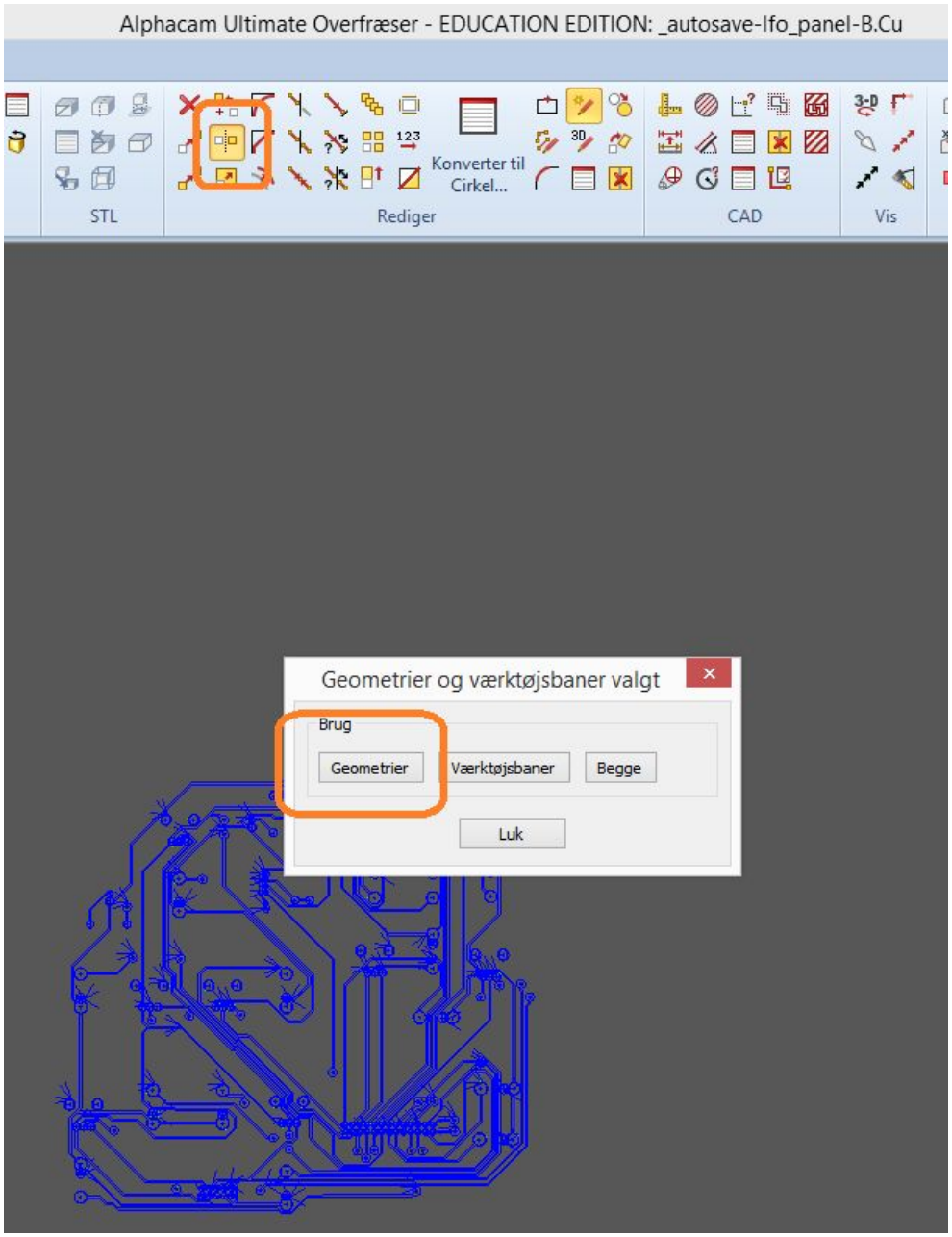


Here is my file, with traces on the right and holes on the left, with all the toolpaths generated. Using the “Flyt” command (move), I’ll select all my trace geometries and move them back using the coordinates at the bottom (remember we moved them exactly 500 before - we can reverse this by putting -500 as the destination). The toolpaths are linked to these geometries.

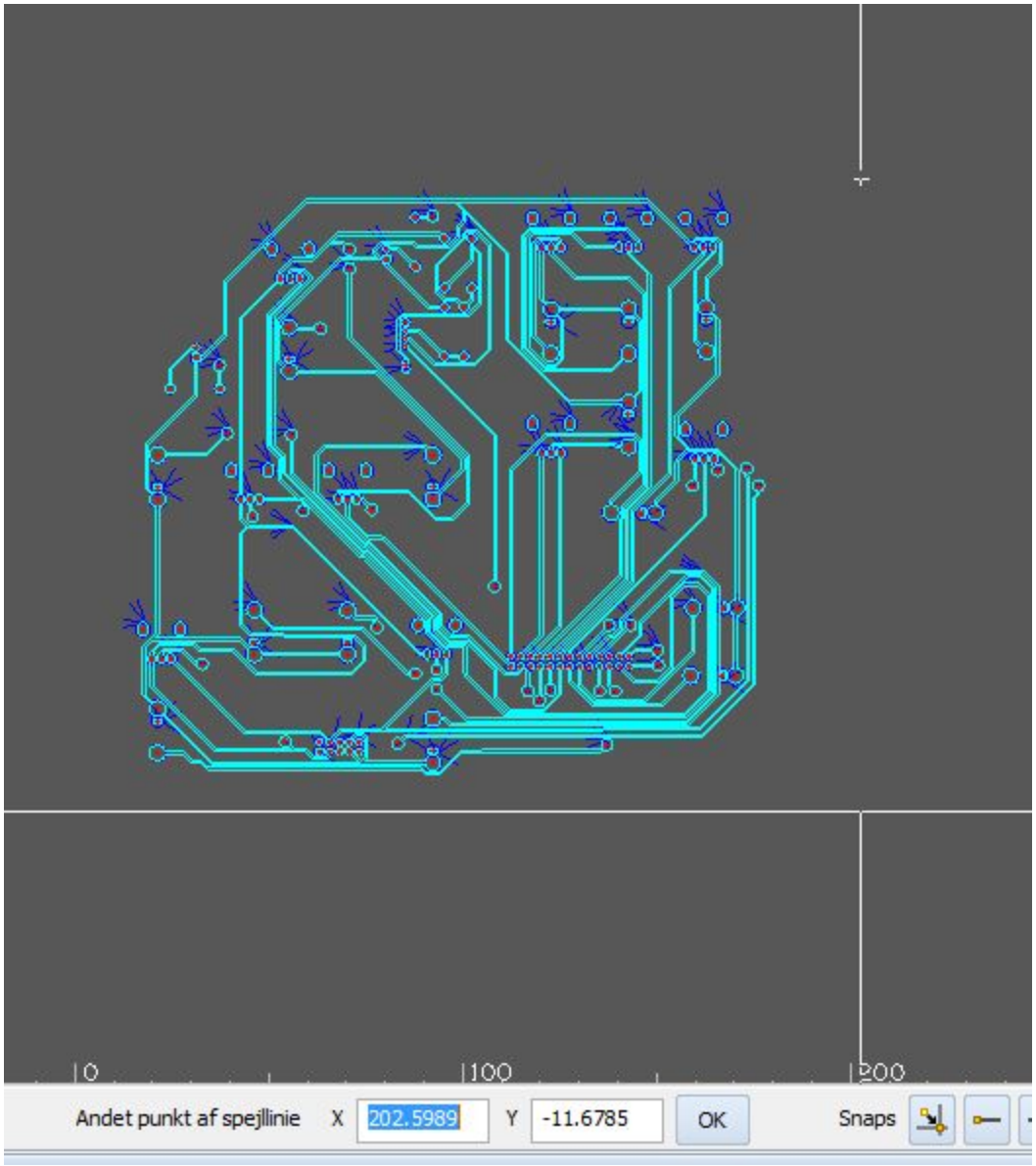
Update the toolpaths using the icon at the top of the screen.

MIRROR

KiCad exports the bottom layer un-mirrored. You need to manually do this for both the traces and the holes. Very important! Make sure your pins of any components match up after cutting the -bottom- of the board.



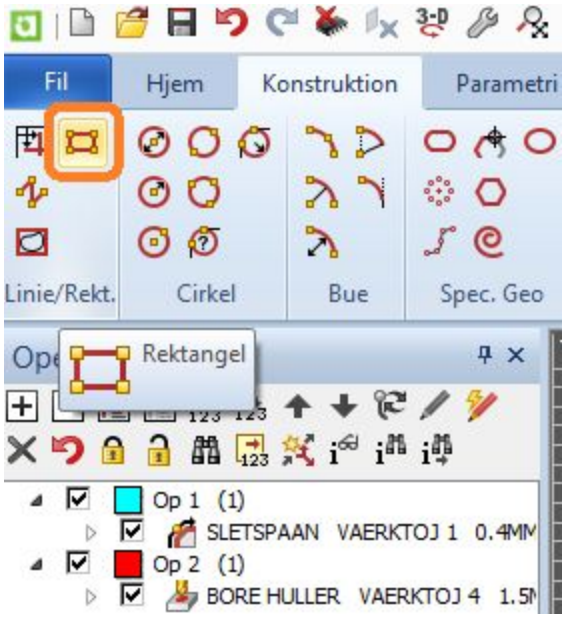
Spejl - select all - use geometries



To pick a mirror line, you can click a point next to the PCB area, then press F3 to lock axes and click again at a point underneath.
Dialog box will ask to keep original lines - click “No”.

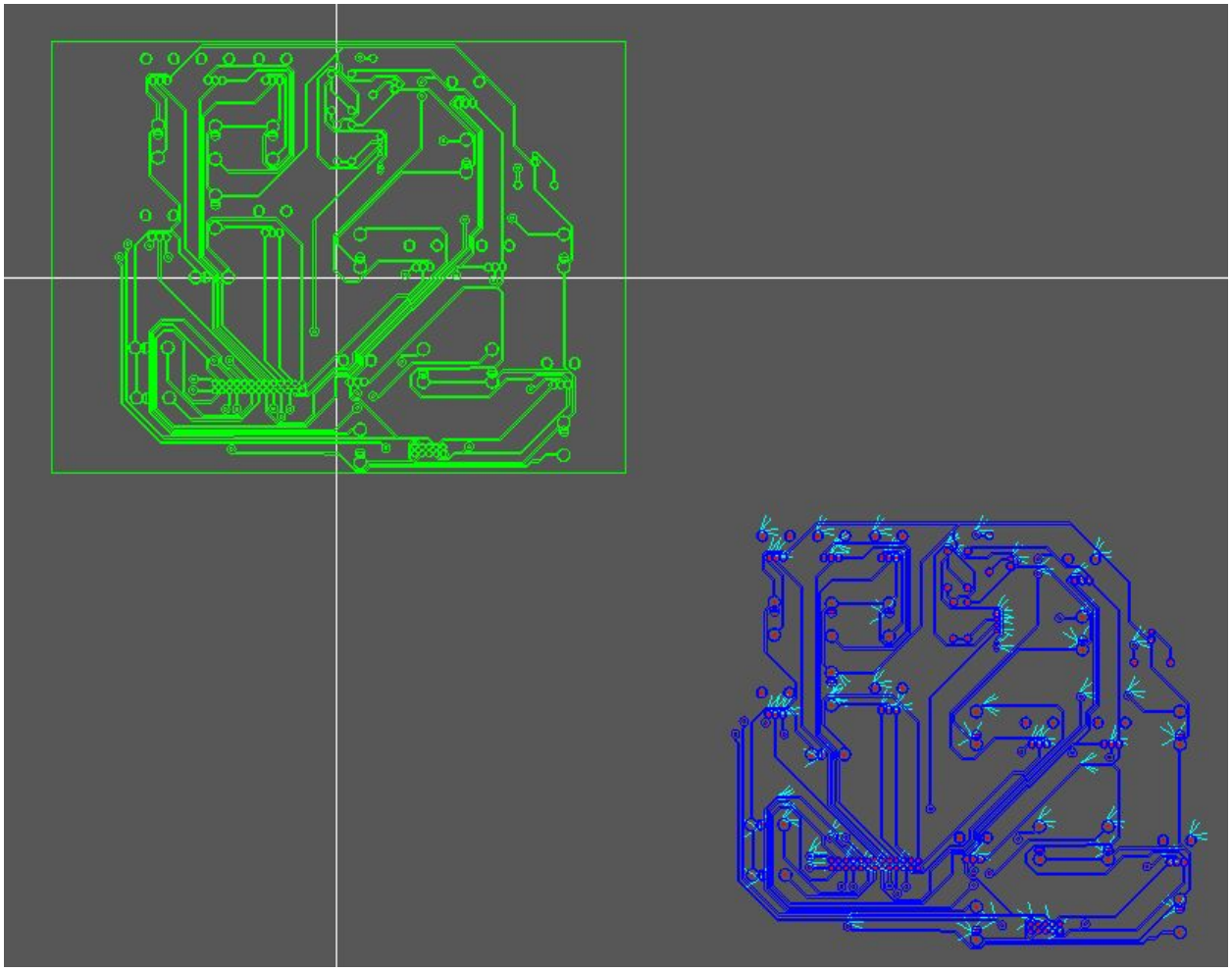
Update again to check out how it looks.

Lets draw our PCB area with the rectangle tool.

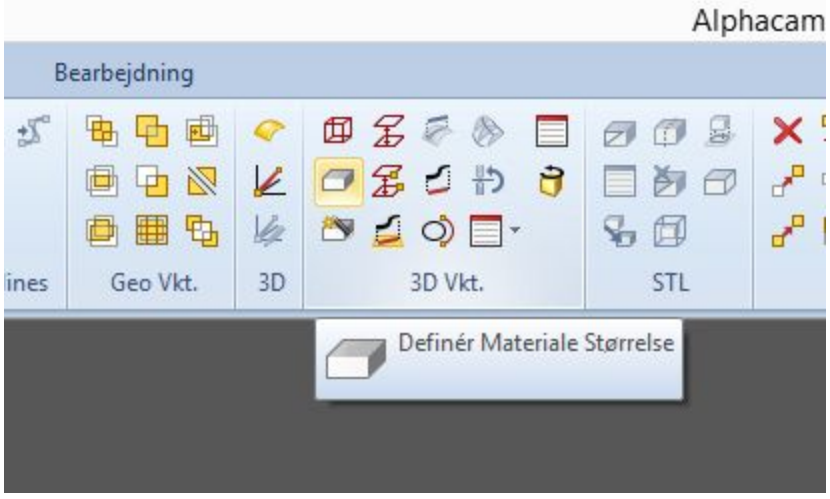


Put the coordinates in the box at the bottom.

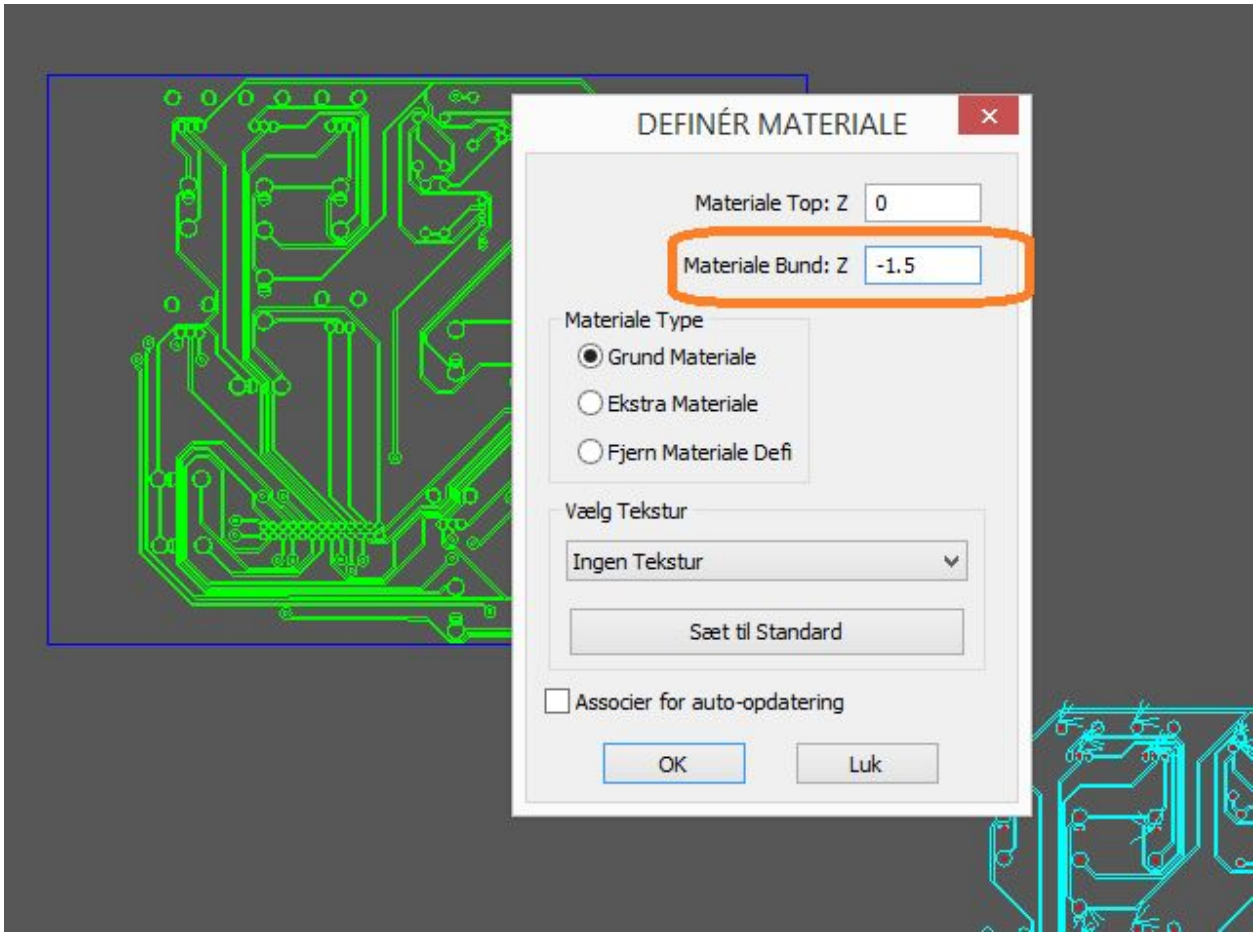
My PCB is 20cm x 15cm, so i'll use 0,0 as the first corner and 200,150 as the second corner.



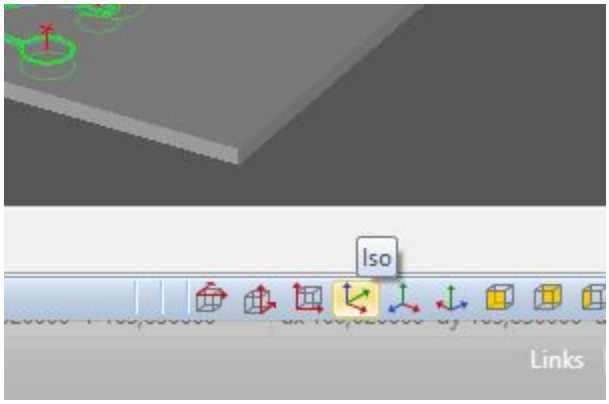
Using the “Flyt” operation, pick all traces and hole geometries (not toolpaths) and move them into the PCB area. You may have to hit F3 to turn off axis lock.



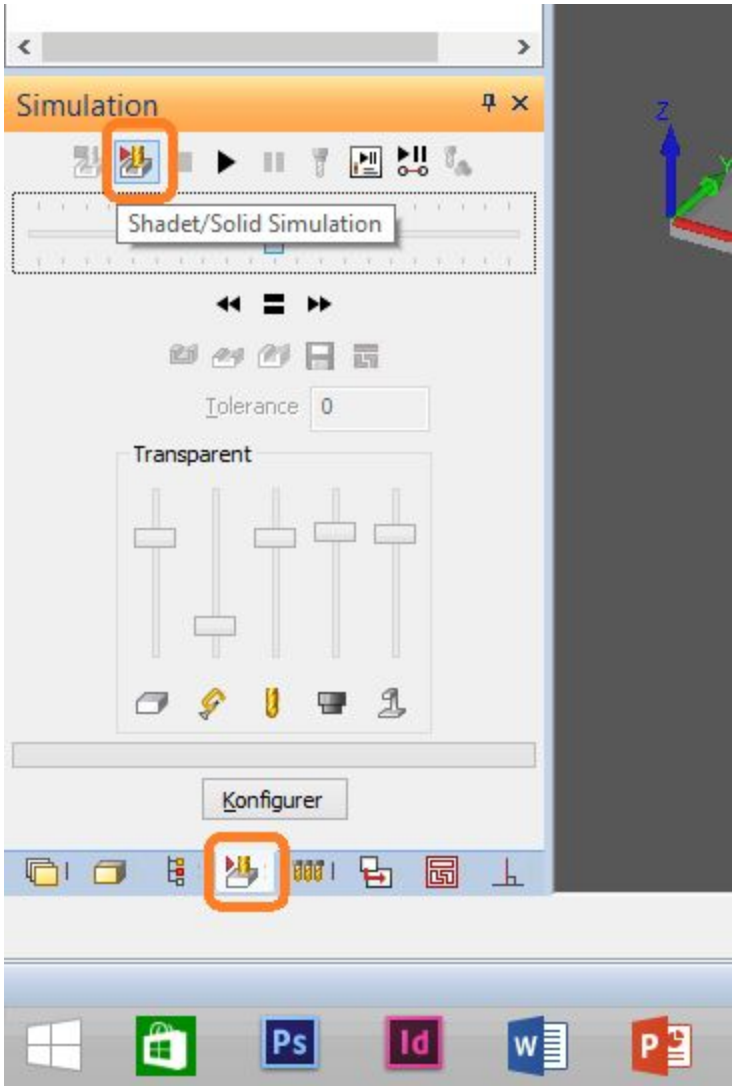
Define the thickness of the PCB area.
Select the rectangle you just drew.



Update again.

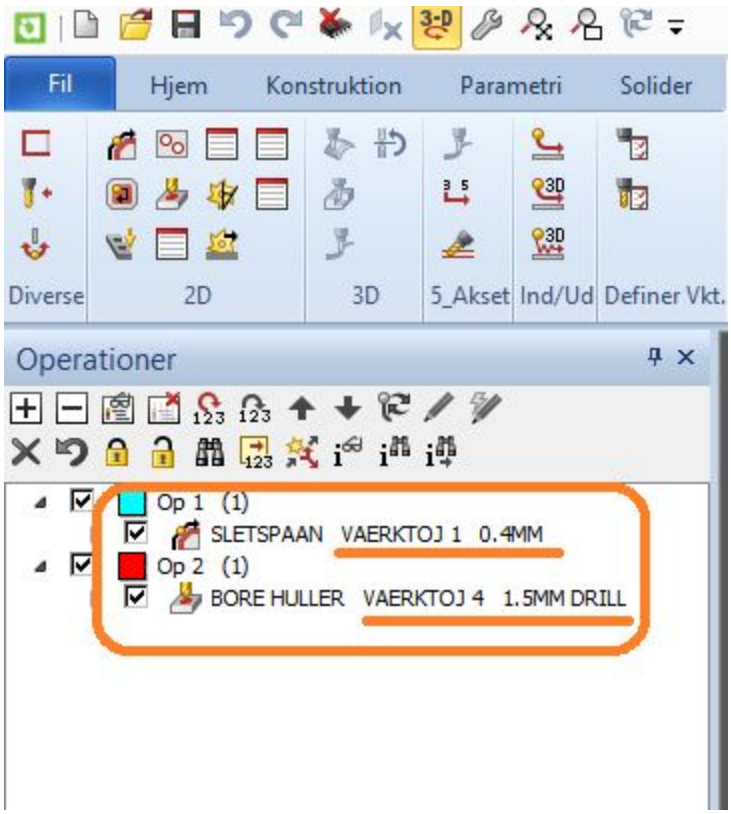


In the bottom toolbar, click Iso.



In the left menu bar, select “Simulation”, then “Solid simulation”. Hit Play and drag the bar all the way to the right to speed it up.

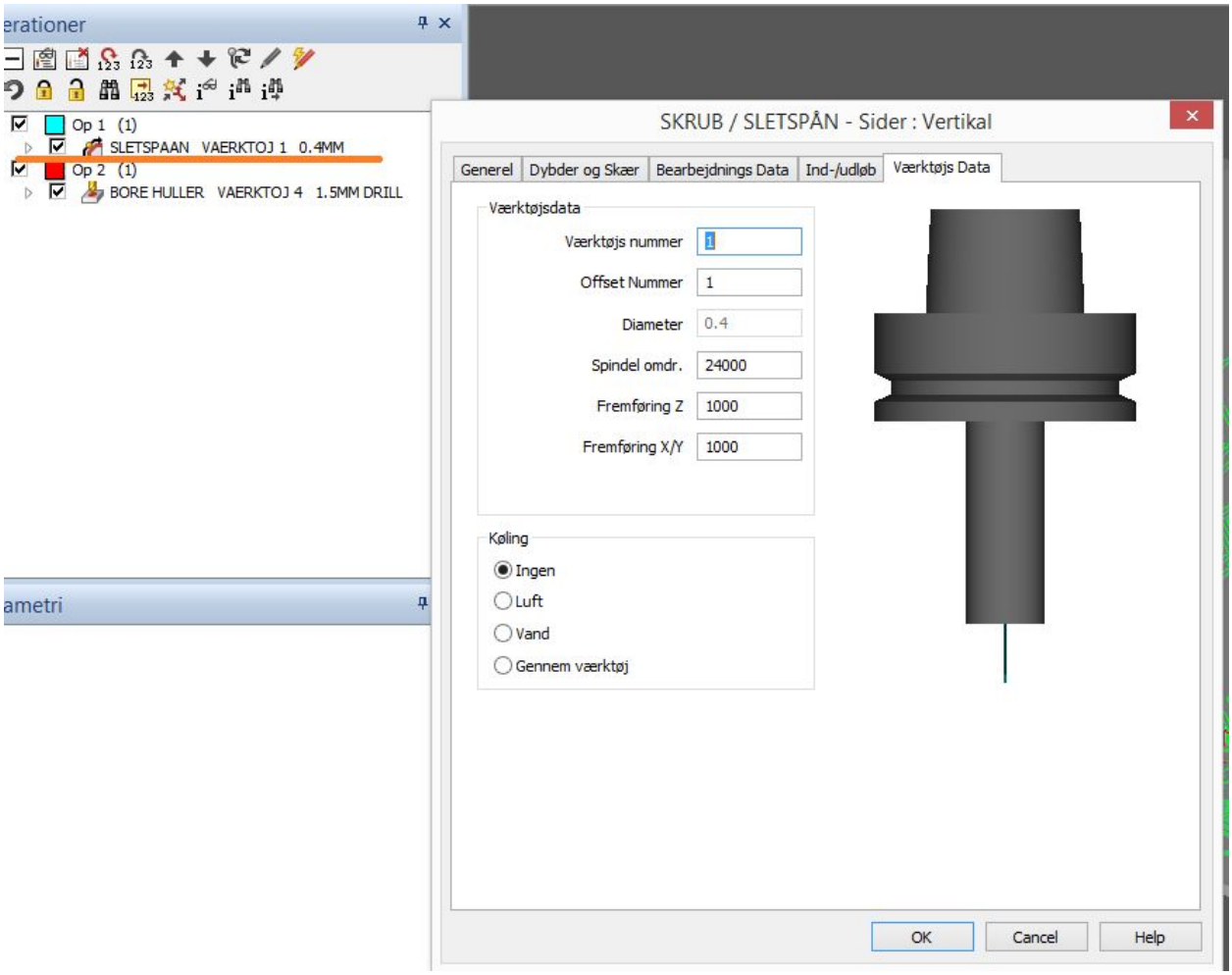
If this is looking good, we’re almost ready to send it to the machine. Make sure there are holes everywhere they should be. Toggle the view back to “Top” in the bottom toolbar and carefully check that everything looks good.



On the left side of the screen, all of the operations should be listed. I only need one drill size, so I'm only using two tools - a 0.4mm cutting tool and a 1.5mm drill.

- Quick note: it's important you don't try and use a drill to cut the traces, or vice versa! Even if both tools are the same size, keep the operations separate. End mills, like we have on the tool shelf, are designed to cut from the sides - drills are designed to plunge -

In our operations menu we can see that the cutting operation is done by Værktøj 1 (Tool 1), and drilling is done by Værktøj 4. That means we should load our tools into positions 1 and 4 in the tool changer.

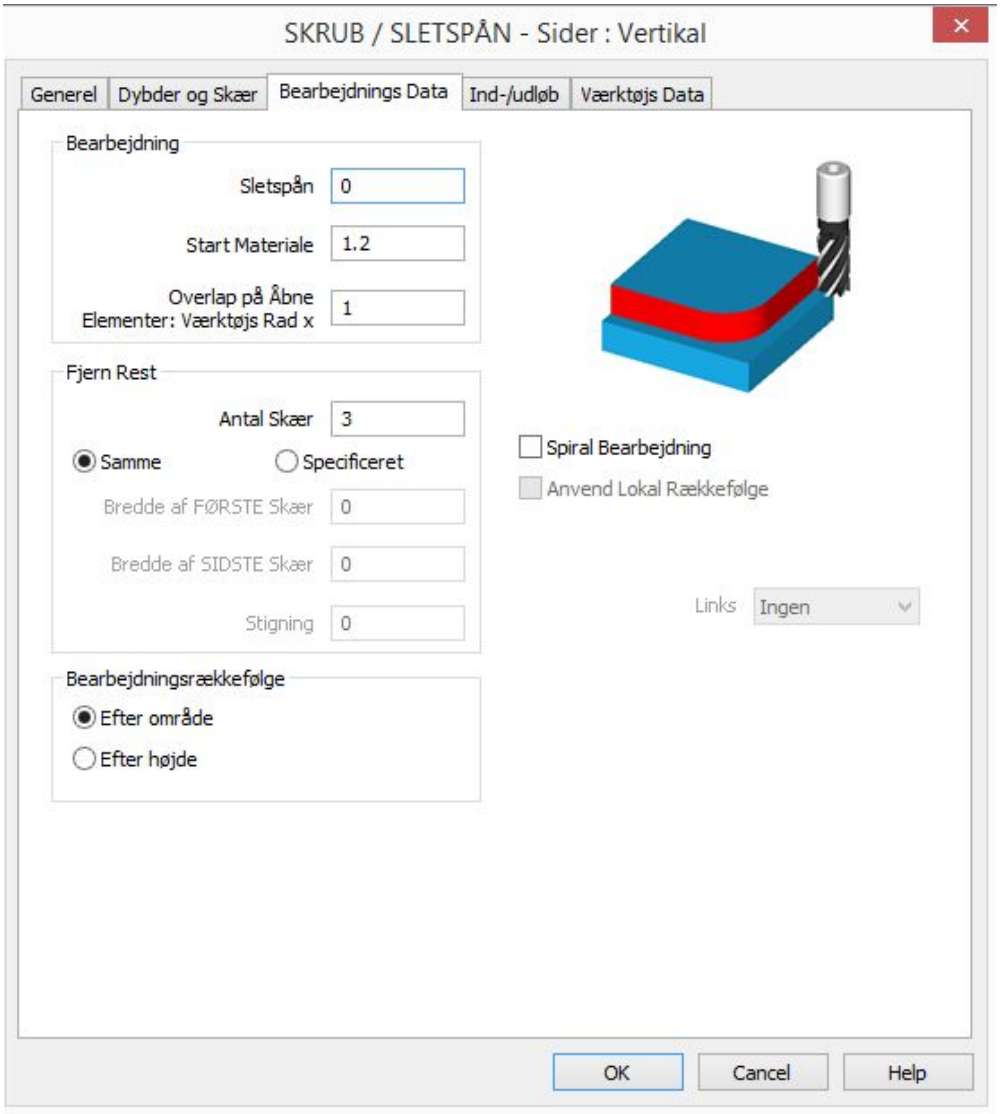


Note: if you need to use different places, right click the operation, select “Redigér”, then change the tool number in the last tab.

Extra cutting

The default setting in the skabelon is to cut the traces with one pass. This has been fine in my testing. Here's how to cut extra space for more cleared area in-between the copper traces to play it extra-safe.

Careful - this does not work well around areas with a fraction of a mm of copper!



The edit menu of the skabelon (right click the operation - redigér - bearbejdings data)

Sletspån - extra depth to cut traces (use negative number in millimeters)

Start Materiale - total amount of material to remove from either side of a trace

Antal Skær - number of passes to remove extra material

In this example, I'm using the 0.4mm endmill to route the traces. I'm going to do it in three successive passes to give a bigger gap in-between the traces; instead of the 'negative' space in-between the copper traces being 0.4mm, it's now 1.2mm. This is done in 3 passes.

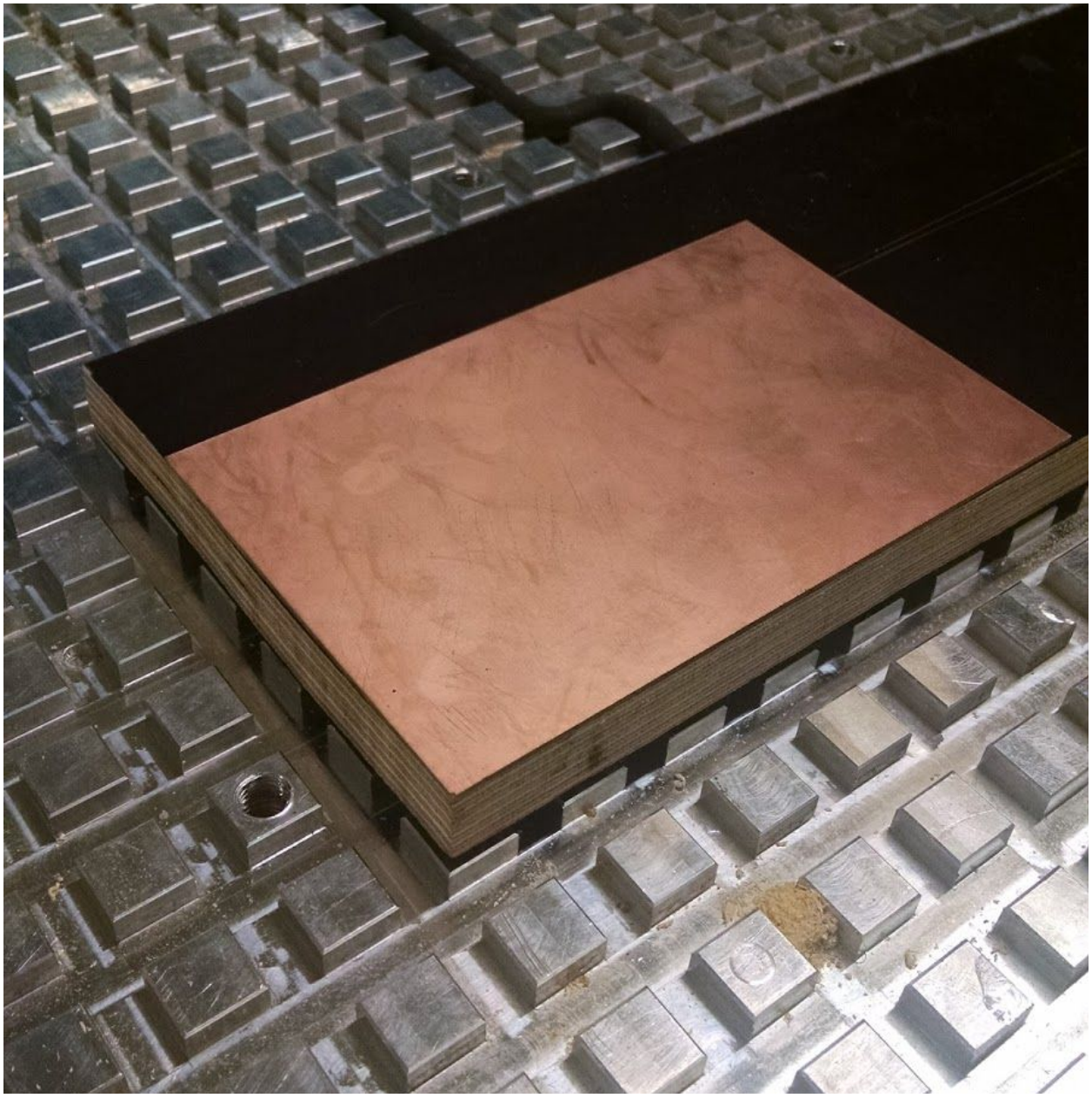
Picture with and without.

Finally, follow the CNC machine tutorial on github (here) for the final steps. Extra things to note:

Our CNC holds down materials using a vacuum table. This means we can avoid clamps, and can make sure our material is totally flat. Because our PCB routing preset mills off 0.2mm of copper, it's highly important that our surface is totally flat. If we're going to be drilling through our board, we need some material underneath our board so we don't drill into the metal table.

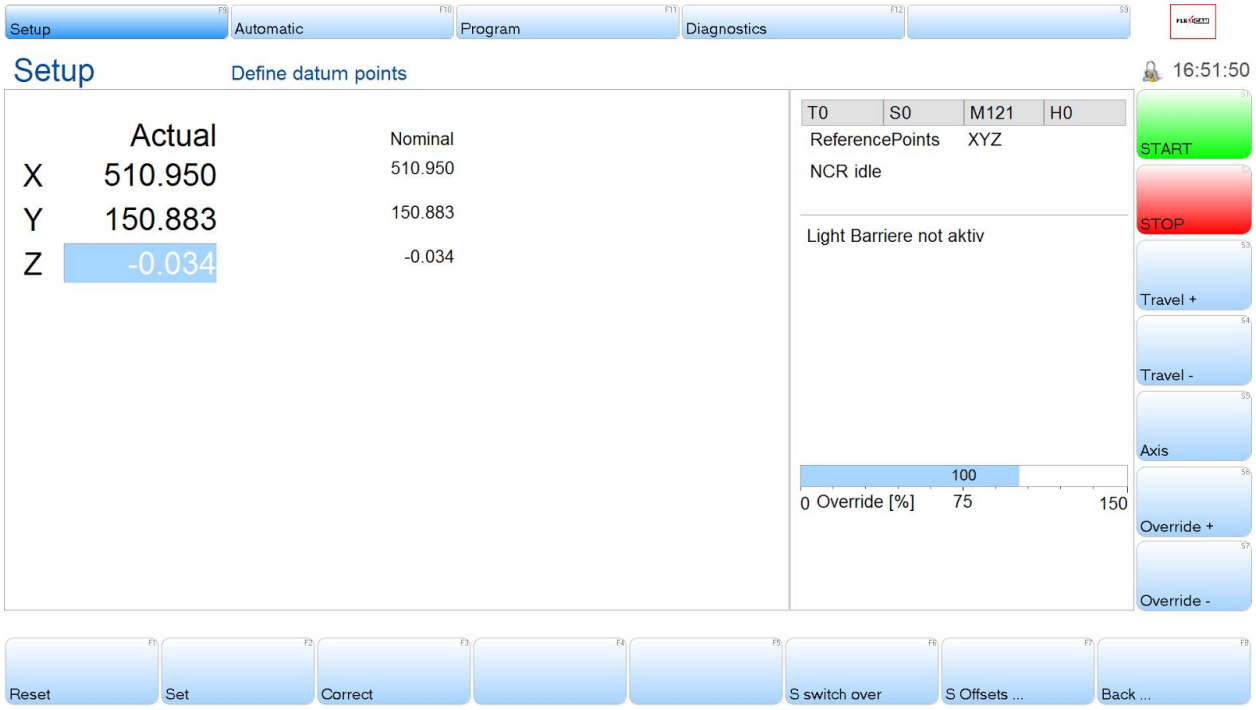
The CNC table has two holes that pull air through. Find piece of flat wood at least 54 cm long that will cover both vacuum holes. Holding it from both sides means the wood won't rotate on the table at all. Lay out the rubber perimeter to seal the vacuum (details are in the machine tutorial).

Thick pieces of laminated wood are perfect because their thickness is usually *very* even. Put lengths of double-sided tape on the non-copper side of the PCB, covering as much as possible, and stick it to the wood. If one end of the wood is cut at a perfect right angle, you can use it as guides to make sure the PCB is straight.



Here’s the copper PCB stuck to the board. I’m using the metal squares underneath the wood to make sure the wooden piece is straight, and the sides of the wood to make sure the copper plate is straight.

When you set the zero points, it’s useful to set X and Y as usual, but set Z in the middle of the board. If the PCB isn’t stuck down totally flat, the edges might be a little higher than the middle. Setting Z0 in the middle of the board can help your results.



Once the Z0 is set, you can check flatness by lowering the tool to the surface on different points of the board, then monitoring what the Z height is in the NC software. Remember that the routing path cuts down 0.2 mm. This deflection shows 0.03mm, which is fine.

Happy milling!