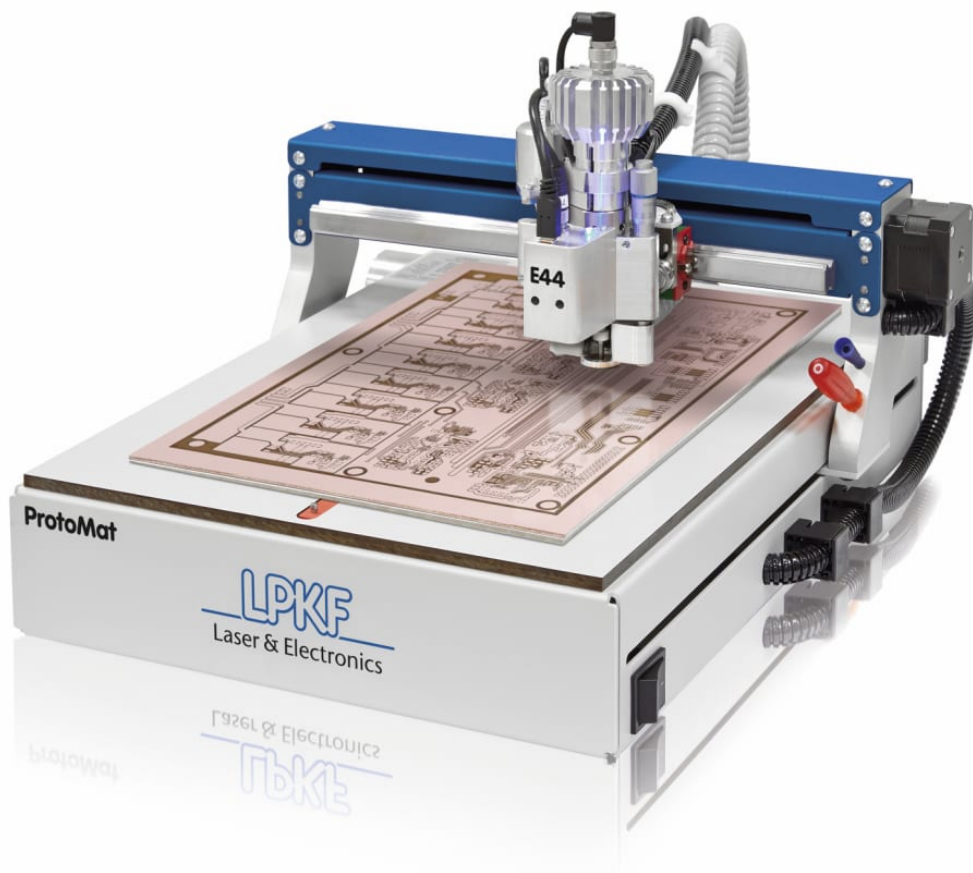# LPKF E44 PCB Mill Getting Started Guide
Student Workshop
KTH

Simon Carlsson, Emil Bergvall

April 6, 2020

1   Introduction

This guide will inform you on how to use the PCB mill located in the student workshop. The mill that we have is a LPKF E44 mill. It can machine single and double sided PCB's quickly and with good quality. However, it is no replacement for proper factory made PCB's since it does not plate vias and holes and it does not apply any soldermask. But, when you want a PCB made cheaply and extremely quickly (usually 30 min) then this process is unmatched.
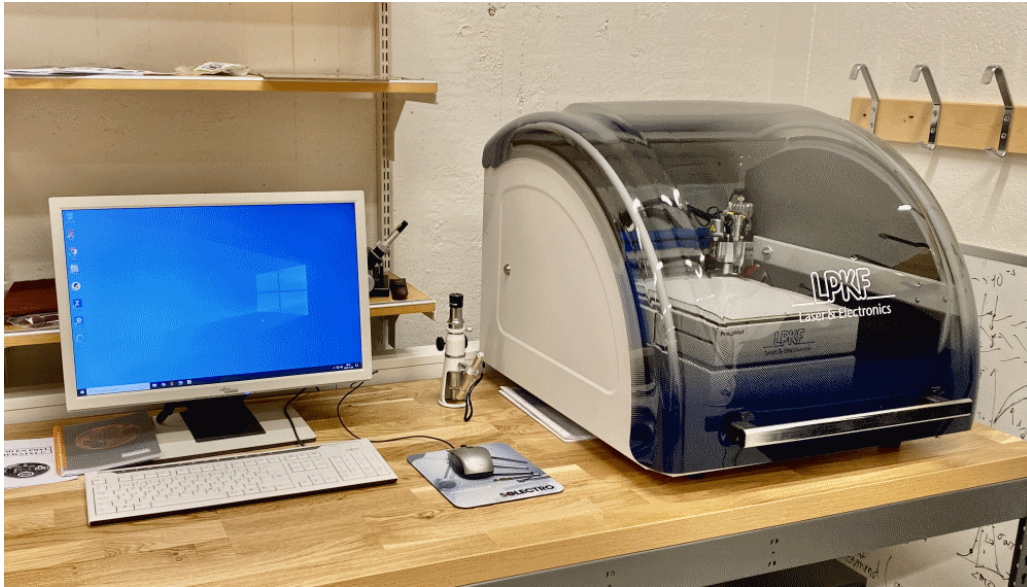

Figure 1: The E44 mill and PC in the Student Workshop.

2   Preparing your board

The first step in creating a printed circuit board (PCB) is the design. This can be done in many different softwares such as OrCad, Altium designer, kiCad or Eagle. We will consider **Autodesk Eagle** for this example, however, the idea is the same for all software.

This will not be a tutorial on how to use Eagle, however, there are some steps and considerations that are unique to the PCB milling process.

At first you will start by creating a schematic. Here you select the components you wish to use and draw the appropriate connections between them. Here, it might be a good idea to use as many **surface mount** components as possible to minimize the number of drills used in the milling process. Figure 2 shows the "add part" menu of Eagle. Note the top right image in red that shows the SMD footprint. There are thousands of different sizes for components. Try and use popular footprints that are easy to find and solder. Some components have a "popularity" meter that can help you find footprints that are commonly used.
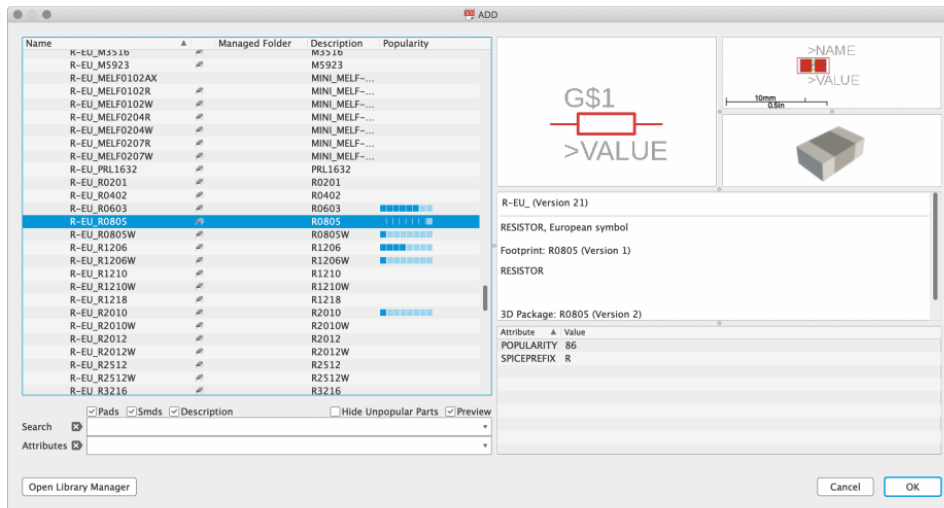
Figure 2: Picking an 0805 resistor from the build in RCL library.

When your schematic is finished you can start with the layout of the board. Firstly you will need to load proper design rules. This is done by clicking the **DRC** button. Load the **Student workshop 2 layer template** and click apply (note that for 1 layer boards you still use the 2 layer template). It should look like the image in figure 3.
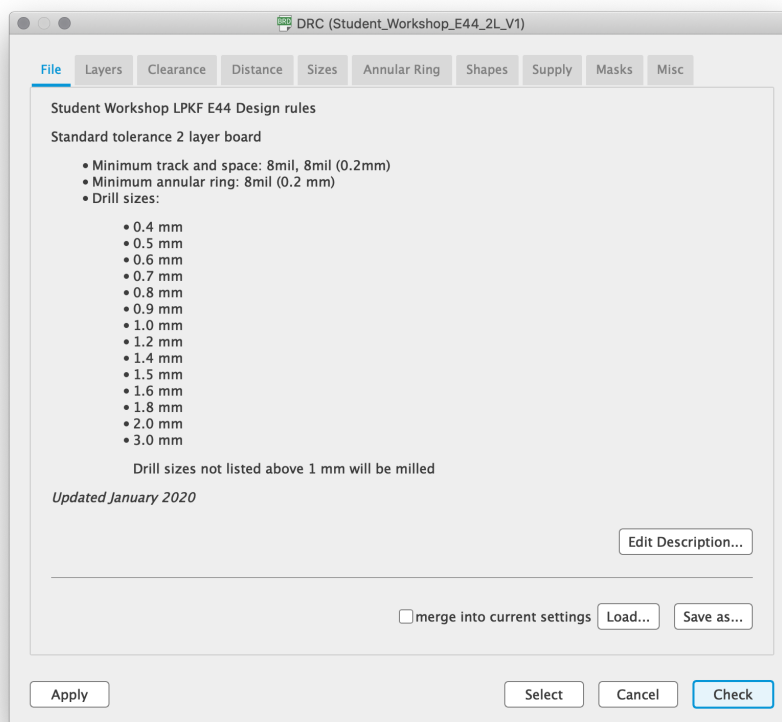

Figure 3: Eagle design rules checker with student workshop 2 layer template loaded.

You can now start routing your board. Try and use thick tracks and spacing, **0.3 mm** and above whenever possible. Also try and visualize on which side you will position your through

hole components (if you have any). Connections needs to be positioned on the opposite side of the components, otherwise soldering these might be tricky. This is illustrated in figure 4 and 5.
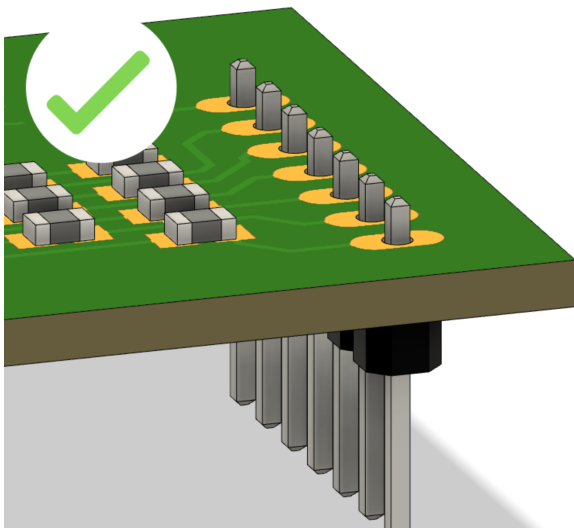

Figure 4: A good example on how to connect through hole components.
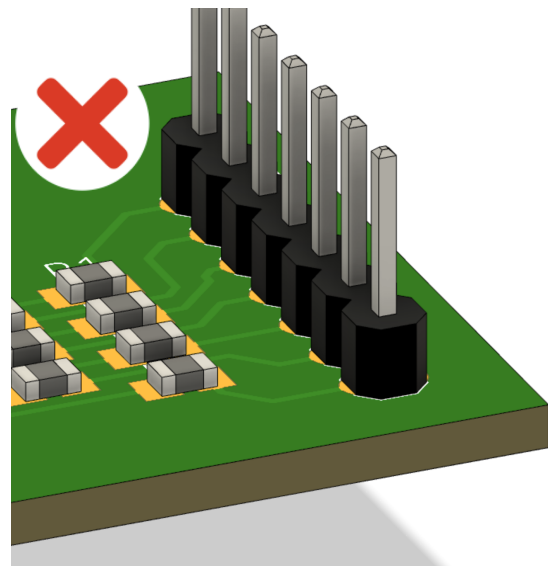

Figure 5: A bad example on how to connect through hole components.

For 1 layer boards, route all your tracks on the **bottom** side (Layer 16). If you forget to do this, don't worry, we can fix that later.

When your design is finished, click the DRC button and with the correct template loaded, click "Check", This will show you any warnings or errors with your design. An error could be that your traces are too close to each other, and a warning could be that you have tracks that lead nowhere. Errors needs to be fixed. However, warnings can sometimes be ignored. Figure 6 shows an acceptable DRC completion.
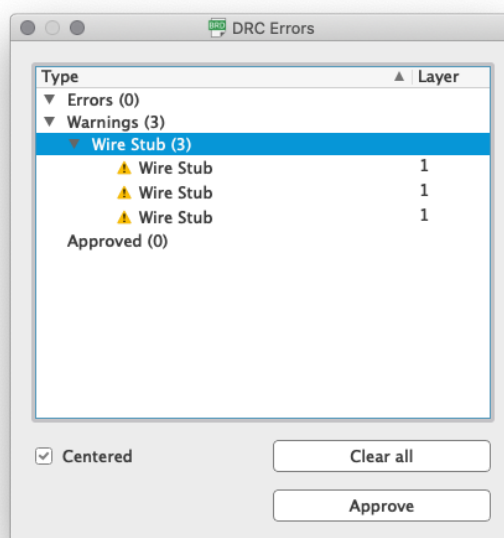

Figure 6: DRC results window showing some warnings but no errors, this is usually fine.

Now, click on **CAM Processor** and the Gerber output window will appear. Go to the top menu and click **Open CAM file...** and select the 1 layer or 2 layer template. The window should look as in figure 7. For 1 layer boards, make sure that your design is visible in the Copper layer. Otherwise you can change it according to the description in the window. When everything looks good, click the **Process Job** button and Eagle will generate the Gerbers. Put them on a USB drive and prepare them for the mill.
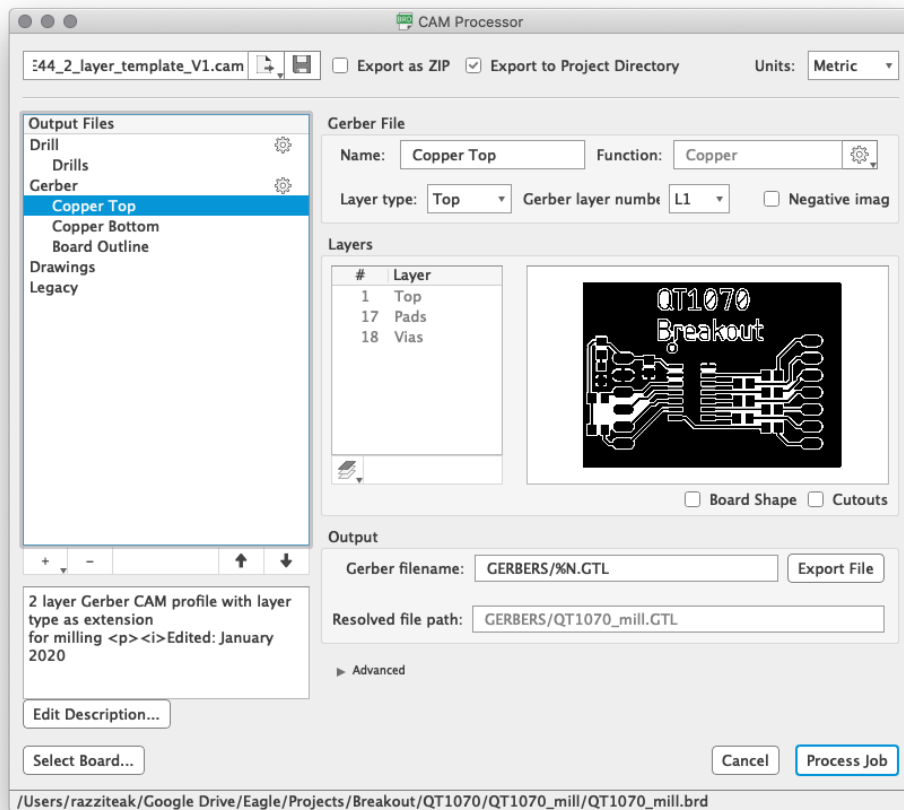

Figure 7: CAM output window with the 2 layer template loaded.
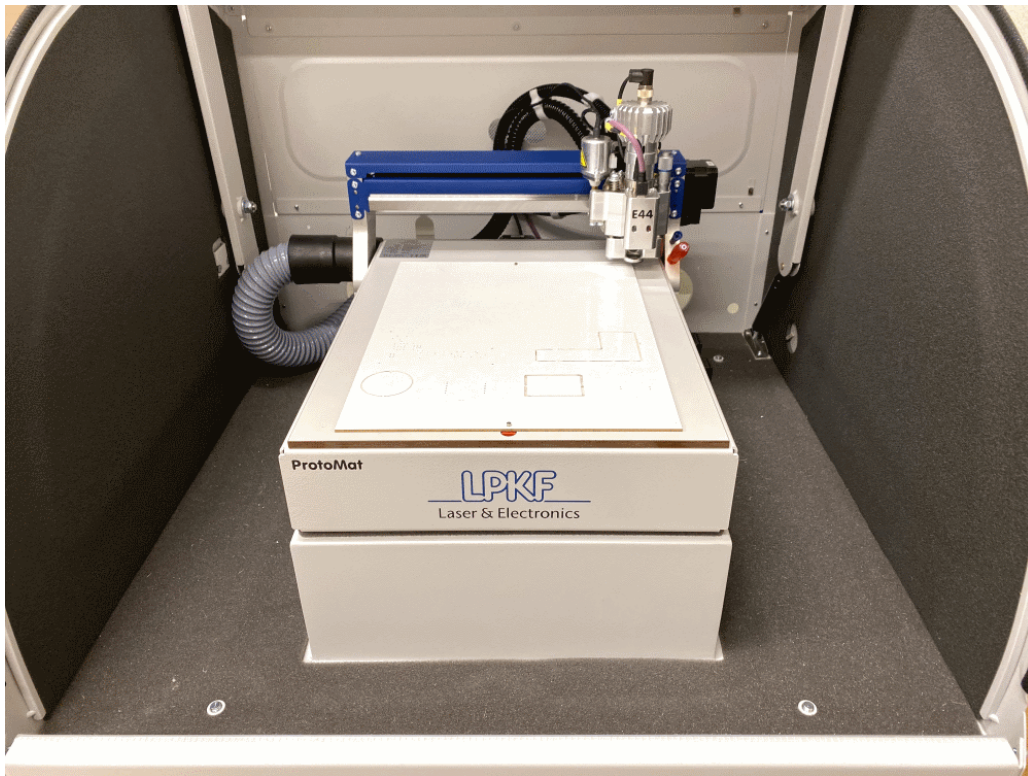
3 Setting up the mill

Figure 8: Mill with opened lid.

To start the mill, open the lid and press the power switch on the front right side of the mill. Start the Circuit Pro PC software. The machine will initiate, this takes a few minutes. In the meantime, locate the the tools that you will be needing:

- USB drive

- Tweezers

- Box of mills

- Masking tape

- Blue tool holder, located on the mill (right side)

- Orange screwdriver, located on the mill (right side)

4 Loading material

A laminate is needed to create a board, choose laminate according to the PCB design you are making. If your design is double sided then you need a material with laminate on both sides. Otherwise single sided laminate should be sufficient. You can also choose the thickness of the laminate and the plating thickness. Normally, the student workshop stocks 1.5 mm thick laminate with 35 μm plating thickness.

To minimize waste, it is always a good idea to find space on an already milled board. Usually, there will be enough space on at least one of these for your board. These are located next to the machine. Consider figure 9, try and find a space that is sufficiently large for your board and that is not creating a lot of dead space. Keep an eye out for other marks that may interfere with your board (such as the milling marks on the right side of this board). In the case of double sided boards, also inspect the back for imperfections.
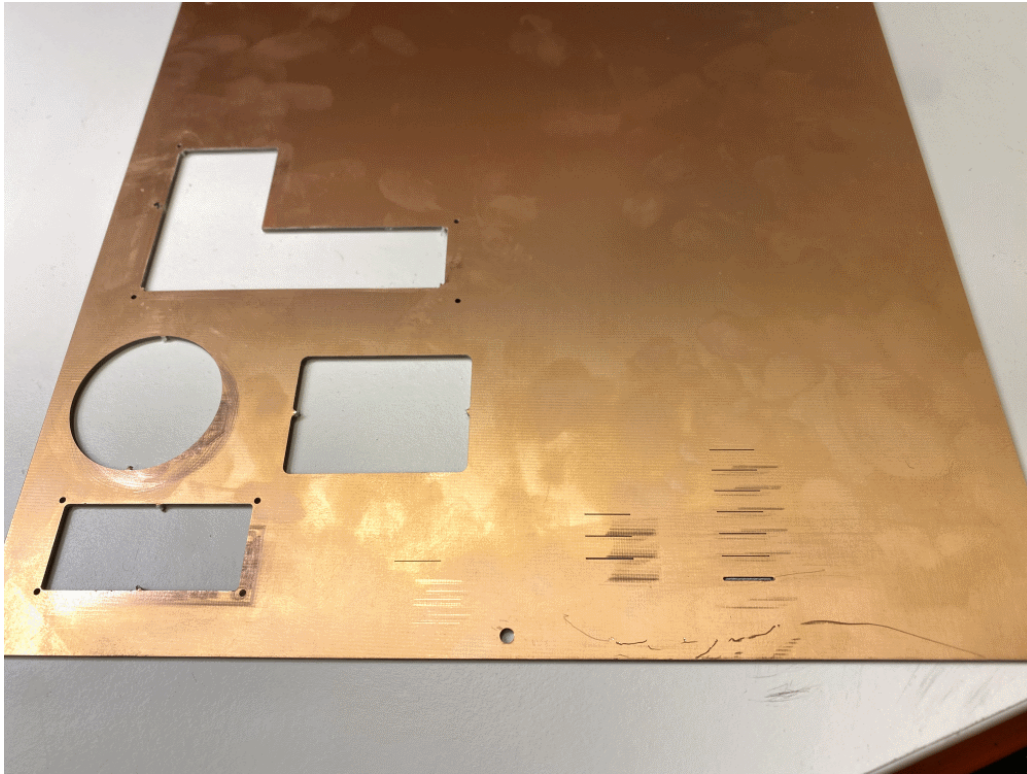

Figure 9: A used 1.5 mm thick board. Note the used section to the left, and milling marks on the right.

To load material, make sure the mill head is in the right, furthest back corner. Take the desired laminate, and place it with the two pins attached to the holes in the material.

In order to secure the board, place a few pieces of tape in the corners of the board, make sure that these don't interfere with your design placement.

5   Starting a job

Open the LPKF software, chose "Templates" in the starting menu. The software opens showing old saved, projects. In the "Templates" section you will need to chose what template you will be using for your design. If you have a single sided design, either chose "**SingleSided_Top**" or "**SingleSided_Bottom**" depending on what suits your design choices the best. If you have a double sided design, chose "**DoubleSided_NoTHP**" for standard template.

When you are ready to print your design, transfer your files to the computer next to the PCB mill. An easy way of doing this is with a USB-stick. Once you have the files, navigate to the file menu and choose "Import" alternatively press "Ctrl + I" as illustrated in figure 10.
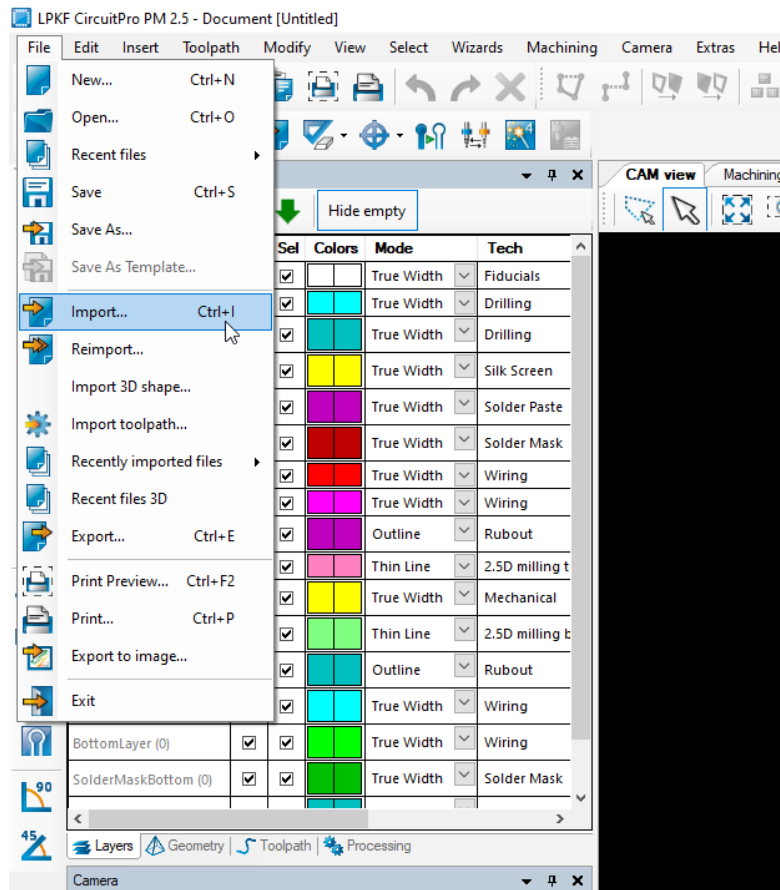
Figure 10: Import your GERBER-files here.

A file explore menu should now have opened up. Navigate to where your stored files are, mark one or several of them and import them. For a single sided board you will need your drill file, Copper file and a outline. If the board is double sided you will also need to specify a file for your bottom layer and top layer. Once you have highlighted your needed files, press "open" and you will be in the import menu illustrated in figure 11. Before you klick "OK" you need to set the "Layer/Template" column. Select what kind of layer/template the selected file is. Beneath is a summary of what file belongs to what template.

Description	Import selection	Gerber postfix
Board top layer	TopLayer	.GTL
Board bottom layer	BottomLayer	.GBL
Board outline	BoardOutline	.GKO
Board drill holes	DrillPlated	.TXT

The file names you have will differ depending on what program you where using. You can check this when compiling your files in your CAD program.
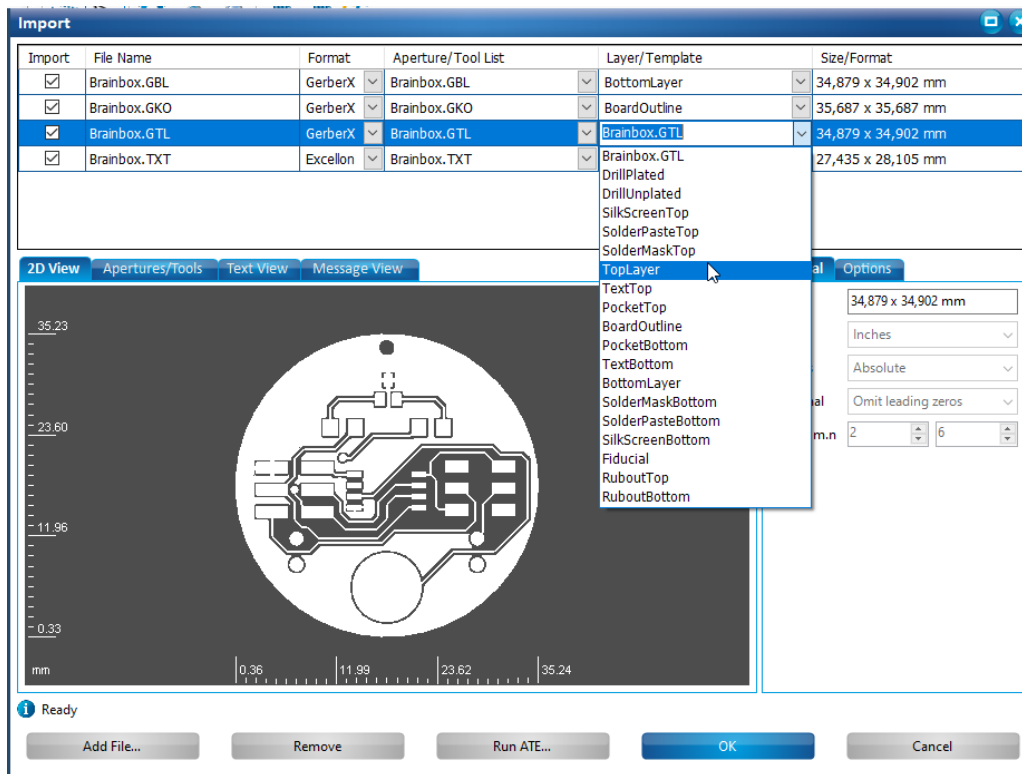
Figure 11: Inside the import menu after the needed files have been imported from the USB stick

After you have imported all the needed files and assigned the right templates, simply press "OK" in the "Import" menu window. Next step will be choosing the right isolation, contour routing and if needed fiducials.

5.1   Creating fiducials (optional)

Fiducials are only needed for double sided boards and usually you can do without them. However, they can make your vias lign up better if you have a very complex design. They help with calibrating the PCB mill after flipping the laminate to mill the other side of the design. To create fiducials you press the button with a circle with a cross over.

This button can be found on the left hand side of the screen when in CAM view. A small window will then pop up named "Create fiducial". Place four fiducials in the corners of your PCB design in the CAM view window, see figure 12. The easiest way of placing them are by using the mouse and clicking where you wish for the fiducials to be. Four fiducials are optimal for precision and more are usually not needed. When the fiducials are placed, simply press the "close" button in the "Create fiducial" window.
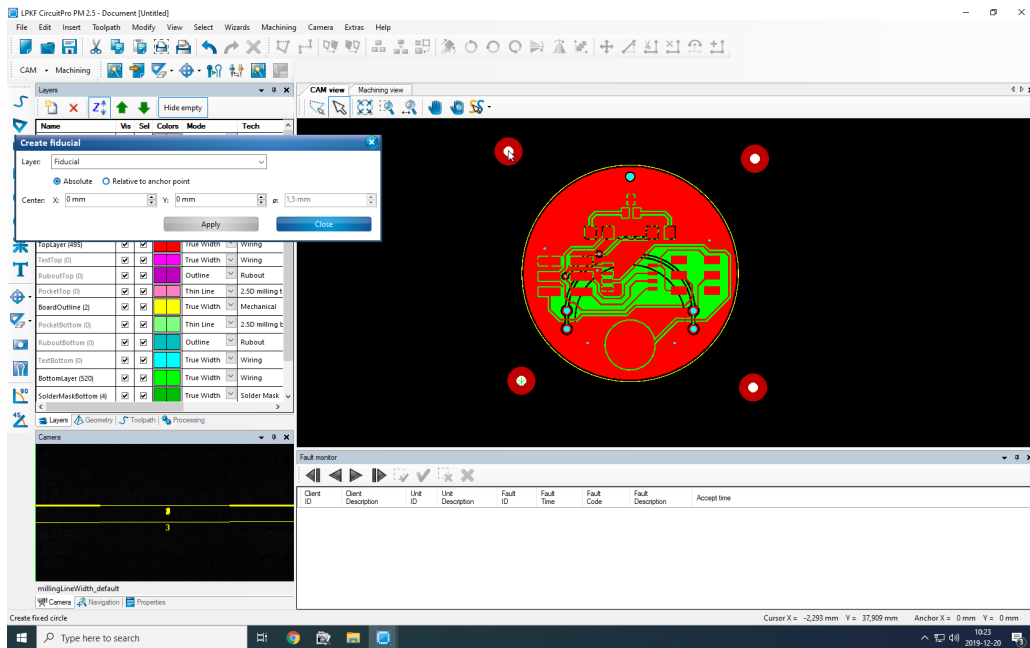
Figure 12: Four fiducials placed in each corner of the PCB design for optimal precision

5.2 Creating isolation

The next step is creating your isolation. This is the main milling toolpath, what actually creates your PCB. Press the button for generating isolation, it is located above the "Layers" window and looks like two pins with a little green arrow between them.

Now you need to let the machine know what thickness of laminate your chosen material has this will almast always be **35 μm**. The window that popped up should be named "Isolate" as in figure 13. Using the nice big arrows to the left, adjust the "isolation method" to achieve the desired effect. You have four options here;

1. Basic

2. Basic, pads double

3. Partial rubout

4. Complete rubout

Option number 2 is recommended for easy soldering and all of the options are explained in the description window in the isolation menu. If you are choosing option 3 you will need to define on your PCB what areas to rub out.
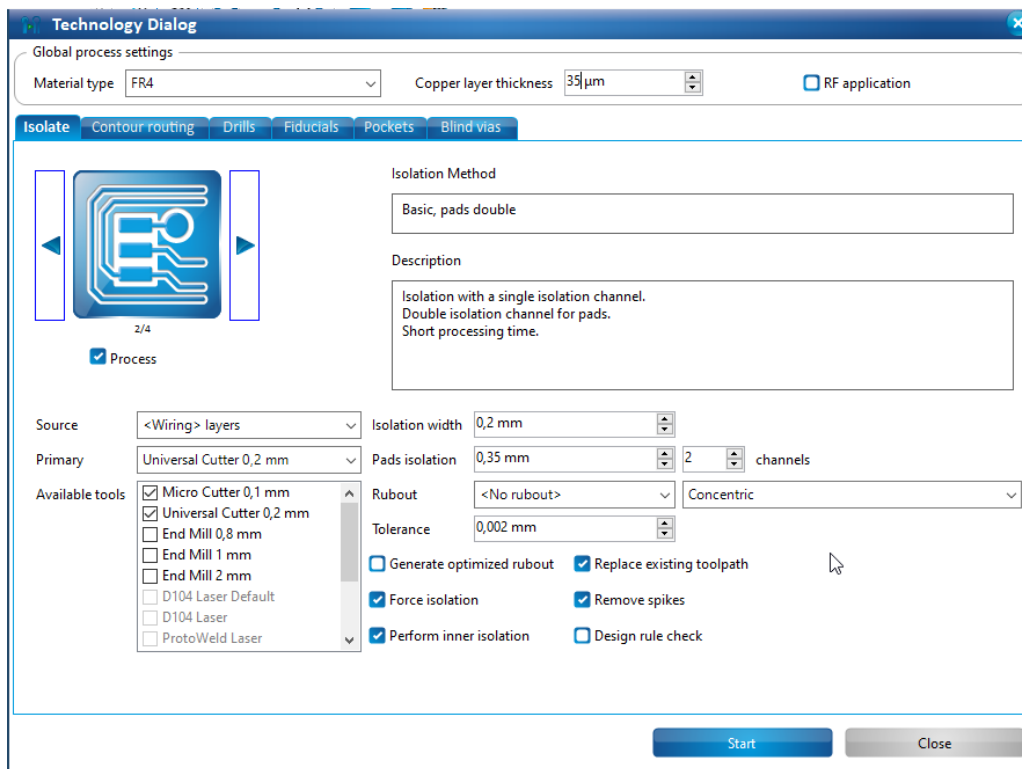
Figure 13: How the window for creating isolation looks. In the upper middle part you will find the Copper layer thickness box.

There are more settings on this page, however, 99 % of the time, we don't need to change these. However, it can be useful to de-select some tools that you don't wish the machine to use, like the 0.1 mm micro cutter, which is not necessary if you used 0.2 mm or above track spacing in your design.

5.3   Creating contour

The countour step will be defining how your outline will be handled. The contour will mill the outline of your PCB excluding anchor points. You have six options in the amount of anchor points and placement, chosing them is similar to how you chose different isolation options. You may also chose if you wish for the contour to be milled outside or inside your outline and which tool to use. The standard tool is the Contour Router 2mm but the Contour Router 1mm is also viable. The outside option is usually recommended as it will not run the risk of cutting parts of your PCB design out.
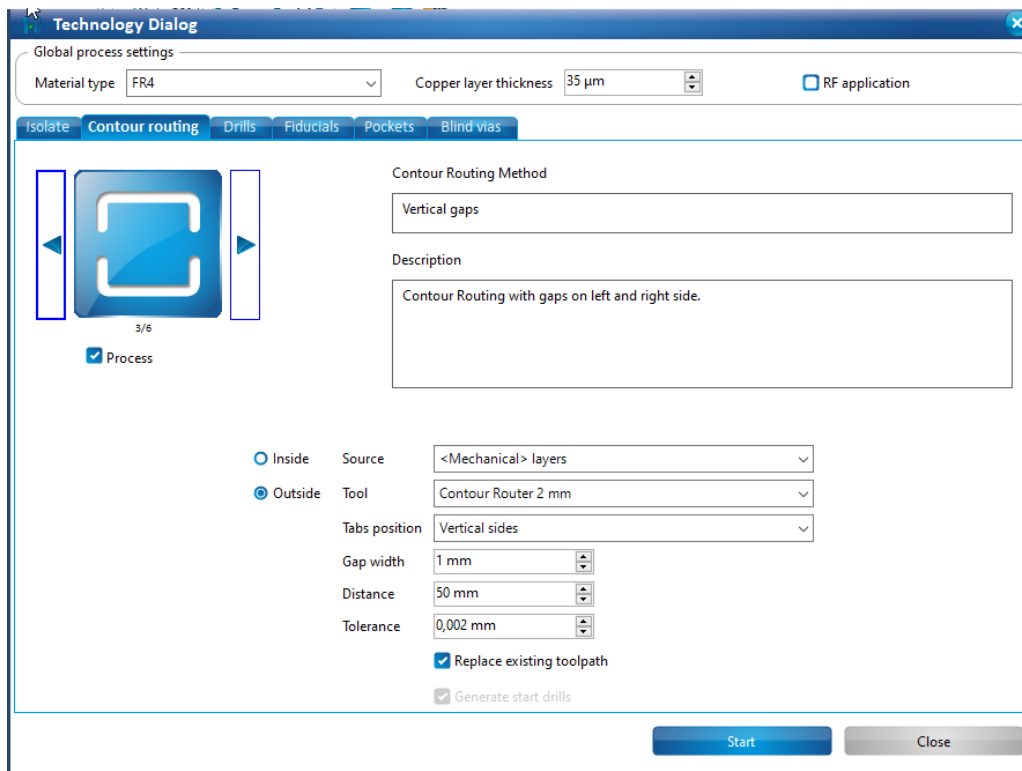
Figure 14: How the window for choosing what contour you will be using looks.

If you are happy with your design choices you may press "START" which will take you to a summary page as illustrated in figure 15. After you have pressed start you will first get an overview over your design choices and the tools that are needed to complete the PCB. Check so that the tools listed all exist in the student workshop. If diameters do not exist but are still needed you can usually chose the closest fitting tool. An example of this would be a spiral drill 1,3 mm which does not exist. You can then use a spiral drill 1,4 mm for an example if your PCB design allows for it. It might also be a good idea to save this file, if the computer craches.
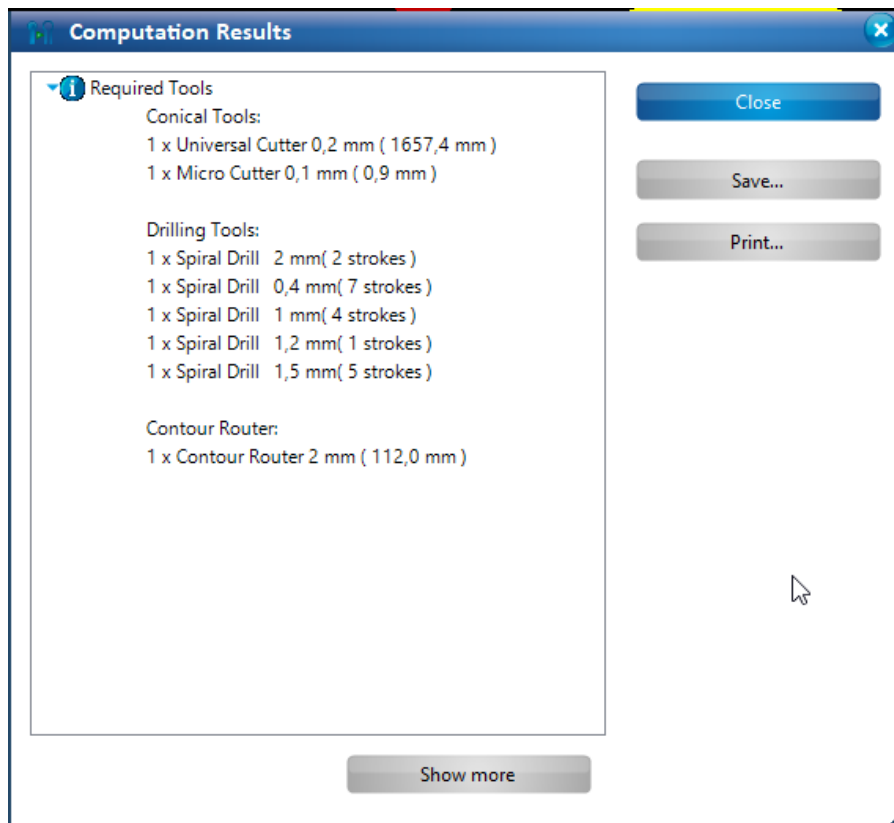
12

Figure 15: Results of the previous steps, with potential warnings about features not used and tools not found.

6 Placing the board

The next and crucial step is placement. Most of the time your board will be smaller than the full laminate that is provided, and a single laminate may be used several times before it will be thrown away. For this it is important to place your board somewhere that the least amount of material is wasted.

If you haven't done so already you can enter the machining view by clicking the button illustrated in figure 16.
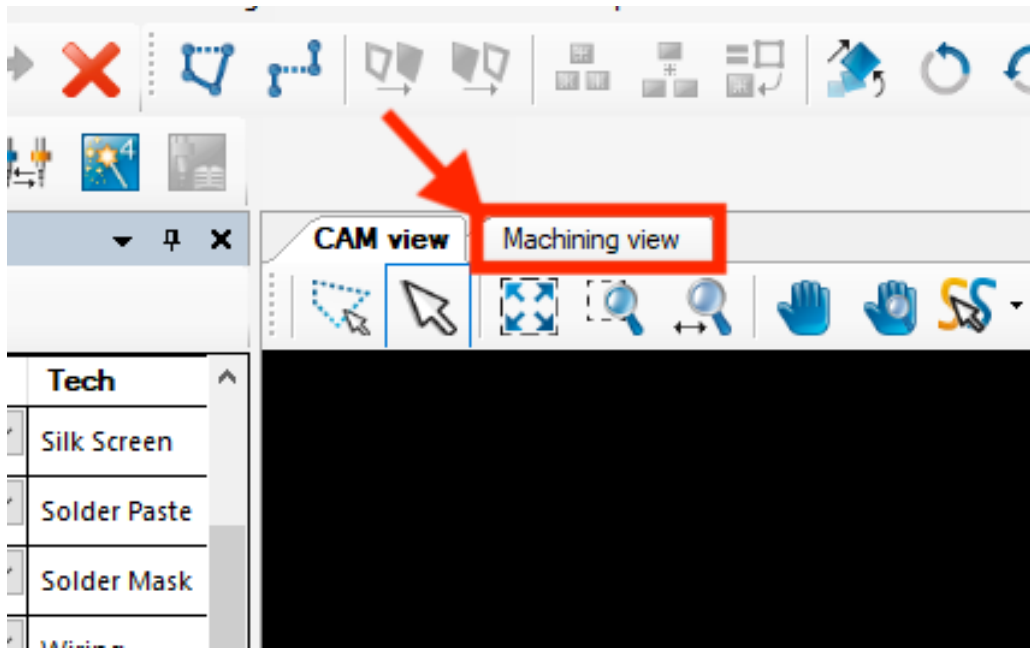
Figure 16: This is how you move into the machining view.

You will now be greeted with the machining view, consisting of the board and a control panel as illustrated in figure 17.
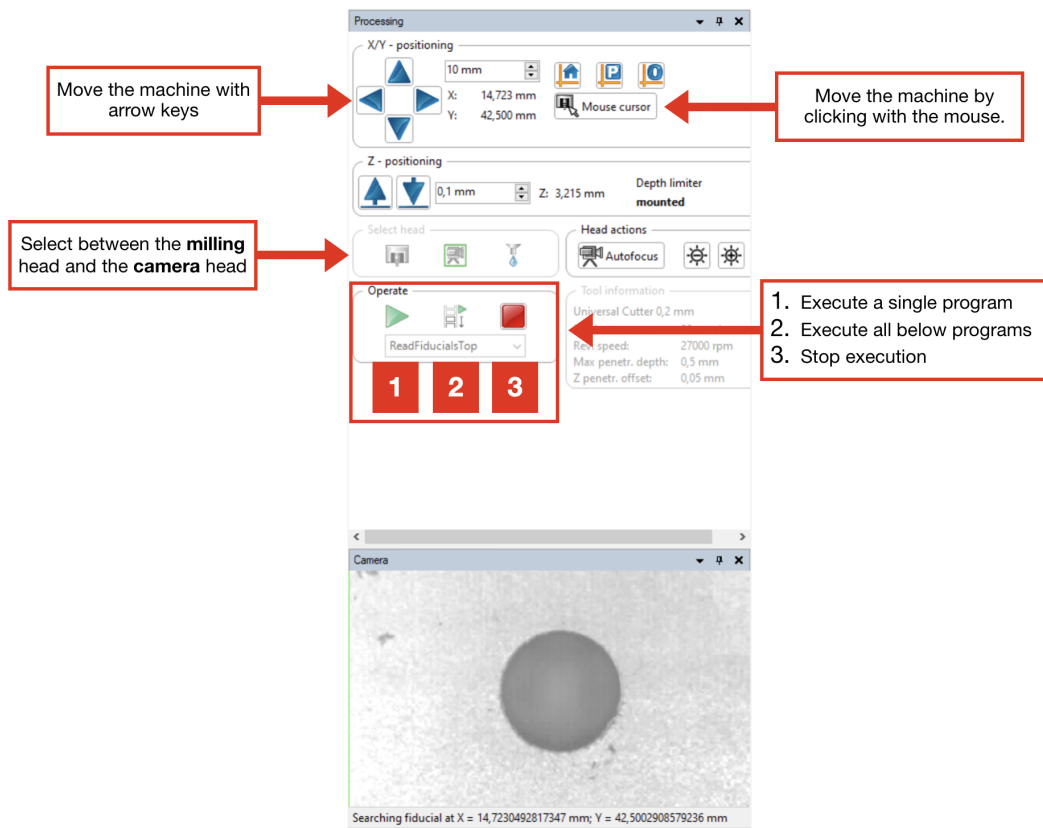

Figure 17: The control panel for the machining view. The machine can be moved around with the arrow keys or with the mouse by pressing "mouse cursor" and clicking in the machine space. The relative position can be according to the camera or the milling tool. A program is executed with the operate controls.

To begin placing your board go to "Machining" -> "Placement". A placement dialog box will now appear, illustrated in figure 18.
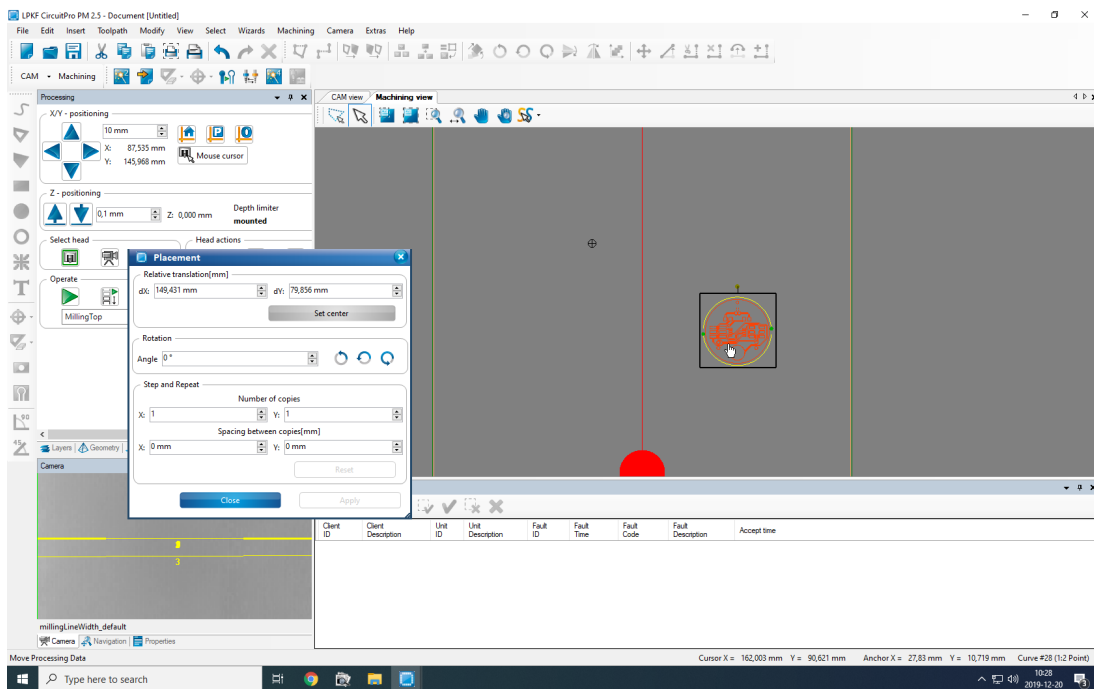

Figure 18: .

You can now use the mouse pointer to place the board on screen to approximately where you want the board.

From here you can move the machine over and make sure that you are clear of any used spaces. **WARNING! Keep away from the red circles top and bottom, do not place anything inside these, if you do you will crash the machine!**

When you're happy with the placement you can click the green arrow to start milling the board.

The machine will take you through all previous steps again to make you confirm these (in case there was any mistakes).

7   Changing tools

For the PCB mill in the student workshop you have a set of tools available for use. These tools are stored in a toolbox, figure 19. The toolbox is stored next to the PCB mill, inside the plastic protection shell. Several different sizes and types are available for use, the computer will tell you what kind of tool to use with a notice during the milling process.

Colour	Name
Orange	Universal Cutters
Pink	End mills
Green	Spiral Drills
Yellow	Contour Routers

Figure 19: The tools available for use for the PCB mill in the student workshop

Changing tools will have to be done during the PCB milling process. The machine will announce when it is time to change tools with a notice on the computer screen. Changing tools is a simple task needing two tools, a screwdriver and a plastic cap. These can be found on the right side of the PCB mill.

Put the blue plastic cap over the tool and gently unscrew the screw holding the tool in place, figure 20. The screw needs only to be gently released and the tool should drop down by itself. If it does not, gently apply some pressure with the tip of the screwdriver on the ring sitting around the tool.

Next you will need to mount the new tool in the mill head. Put the selected tool in the blue plastic cap and insert it into the mill head. Use the screwdriver to screw in the screw sitting in the PCB mill. **WARNING! The screw should only be fastened with a small amount of force!** Once the new tool has been installed properly it should look like figure 21.
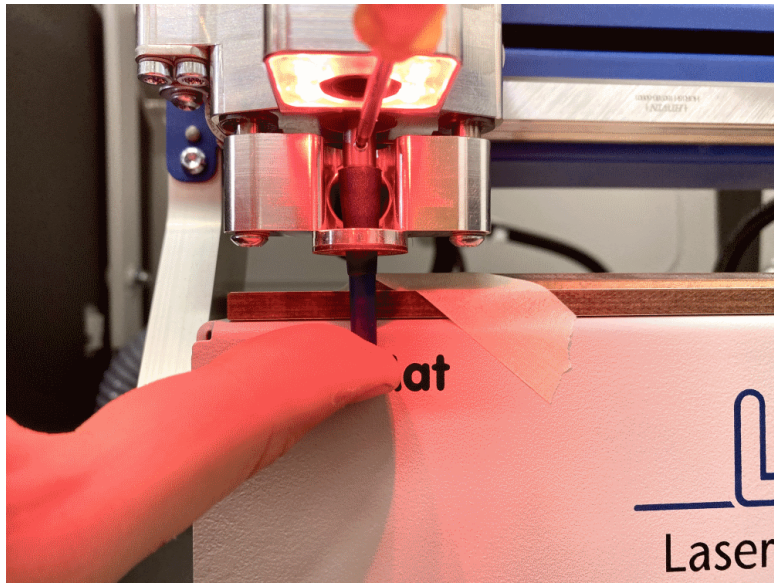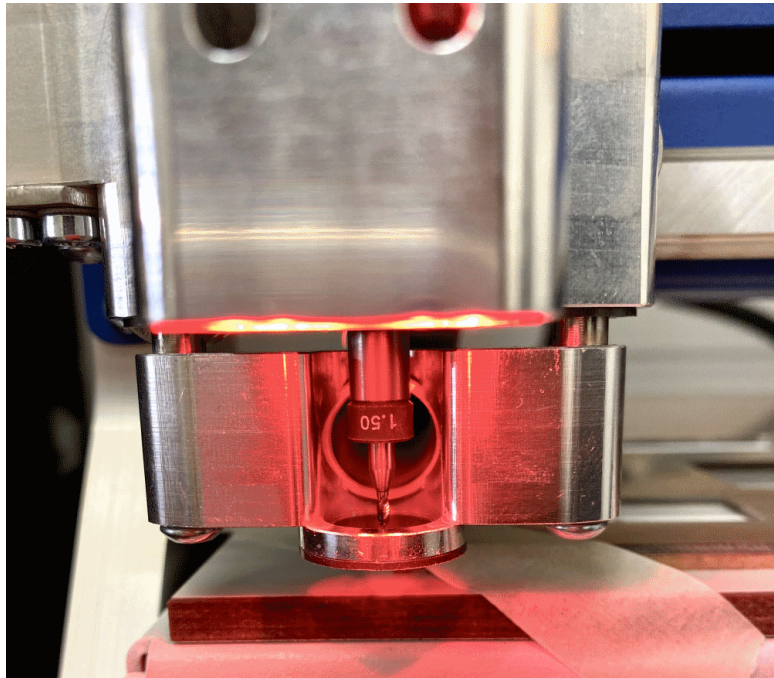
Figure 20: How it looks whilst changing tools

Figure 21: Correctly installed tool

8   Mid-time inspection

When all operations on one side are completed, you can inspect the milling quality by placing the hand-microscope over the board. An image of what a good board looks like is shown in figure 22 and 23. If something looks oddly bad, contact the workshop responsible and set aside the tool that made the operation.
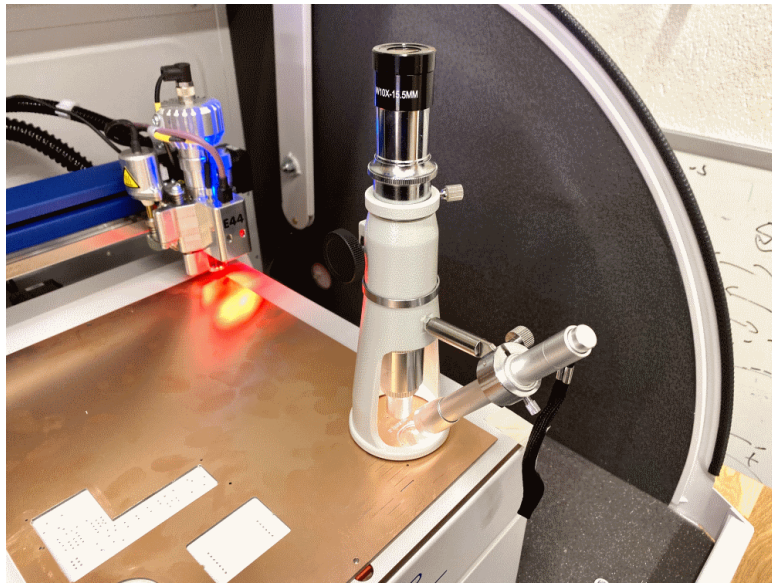
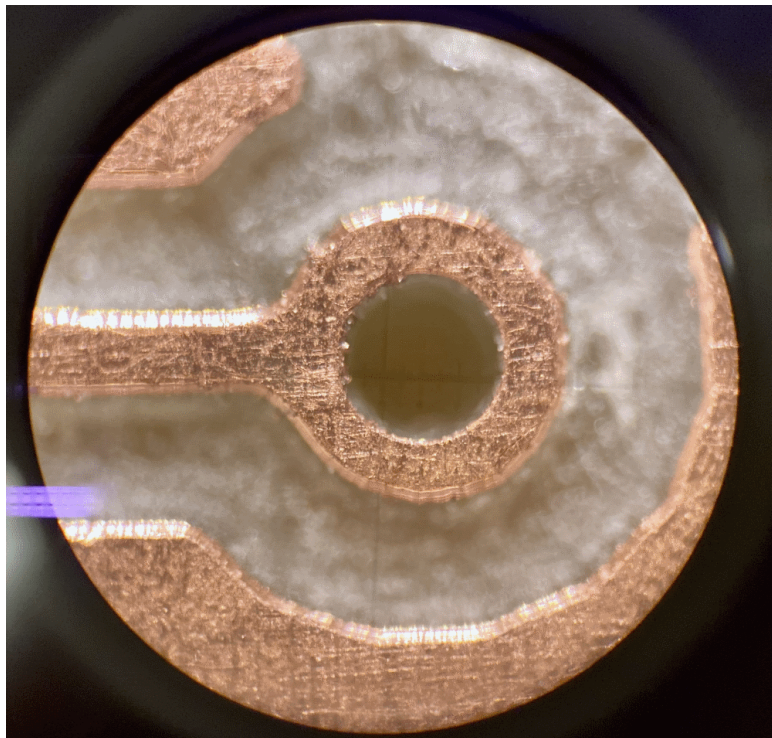Figure 22: How to place the microscope on the board.

Figure 23: Example of a very good milled trace and via.

9 Flipping the board (2 sided only)

If you are doing a single sided PCB design you can ignore this step.

If you are doing a double sided design, during the milling process you will have to flip your laminate to mill the other side. You should only flip the laminate once the program tells you it is time for the laminate to be flipped. The laminate should be flipped around the long side of

the laminate so as the same holes on the laminate are placed on the same centering pegs of the machine, see figure 24.
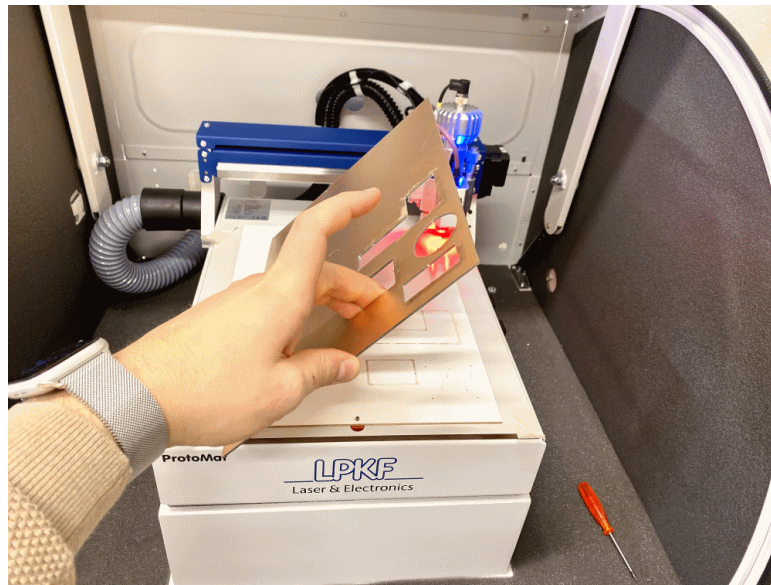

Figure 24: Flipping the board around the correct, long side of the laminate .

10   Removing the board

The last step of your PCB creation will be to remove the finished product from the laminate. First remove the laminate from the PCB mill. Then using side-cutters, cut the support material holding your design to the laminate.

11   Post processing

After your PCB is finished, a small finishing touch will ensure the best result. Use some sandpaper to remove any blemishes left from the machine during the production. To ensure that the laminate won't oxidize you can spray some clear protective lacquer on your board before soldering, an illustration can be seen in figure 25. You can solder straight on top of the lacquer, it's simply a protective coat that keeps the board from degrading.

Figure 25: Electrolube, soldarable protective lacquer.

12 Soldering vias (2 sided only)

The milling process does not create through hole vias. Ie, there won't be any connection between the top layer and bottom layer. The easiest way of achieving electrical connection between these is to solder a small thin wire through all the vias on the board.

13 Admire your board

Take a few minutes to inspect and admire your board.