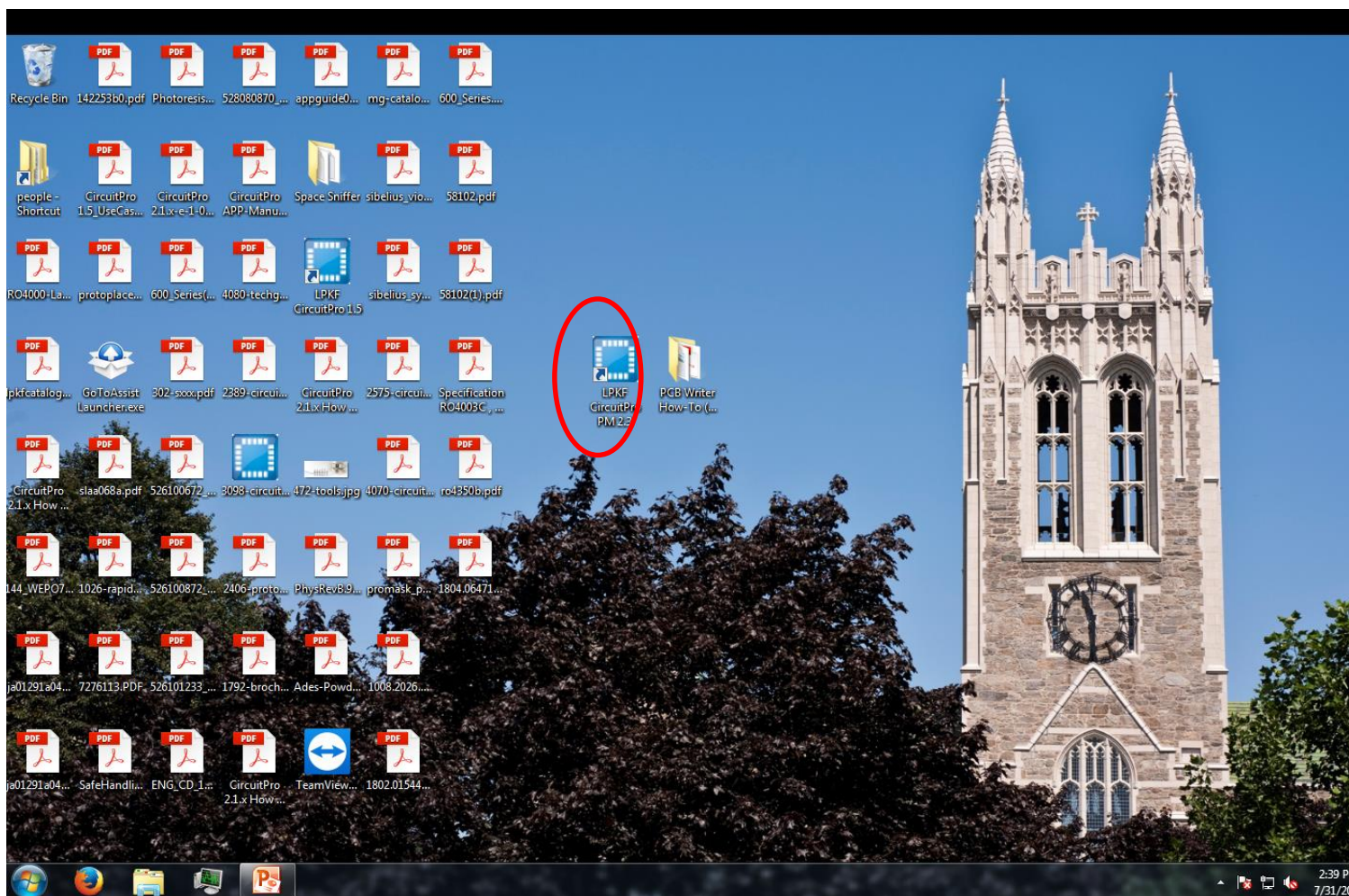


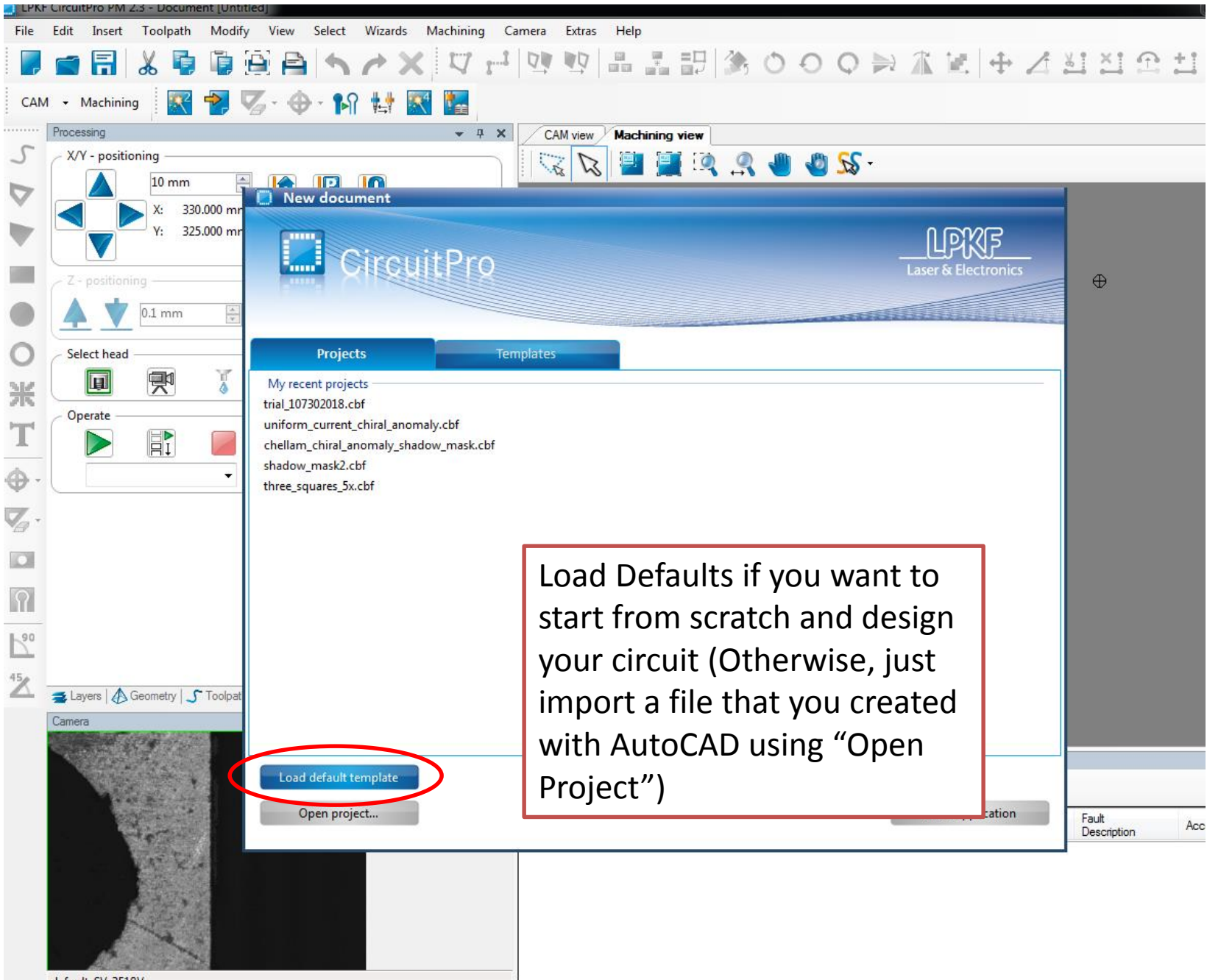
PCB writer

Model S-63



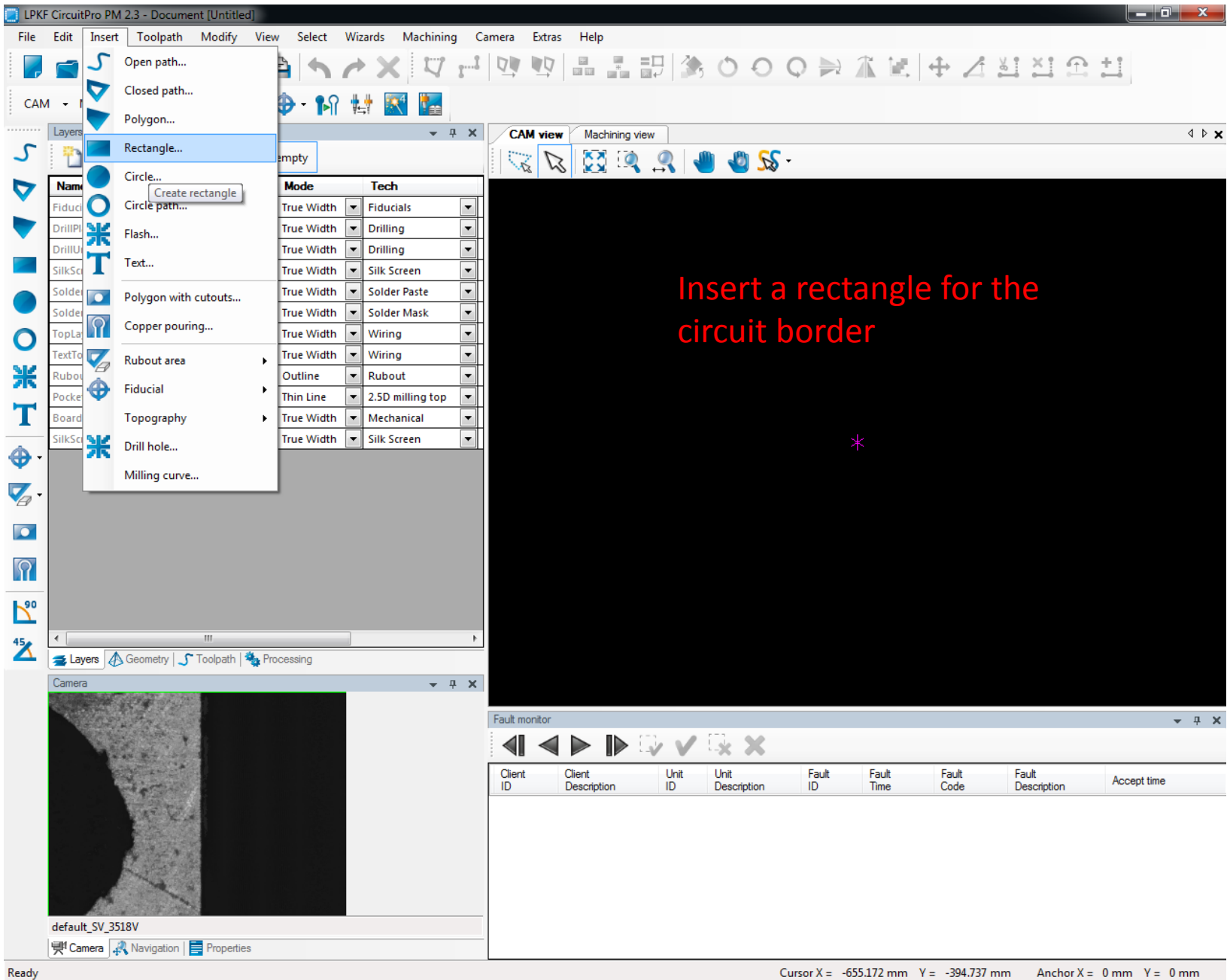
- Use a dust mask when opening the cover
- Open the cover and turn on the system →
- Double click on **LPKF circuitpro 2.3**
- Do not run the other version of the program (1.5)

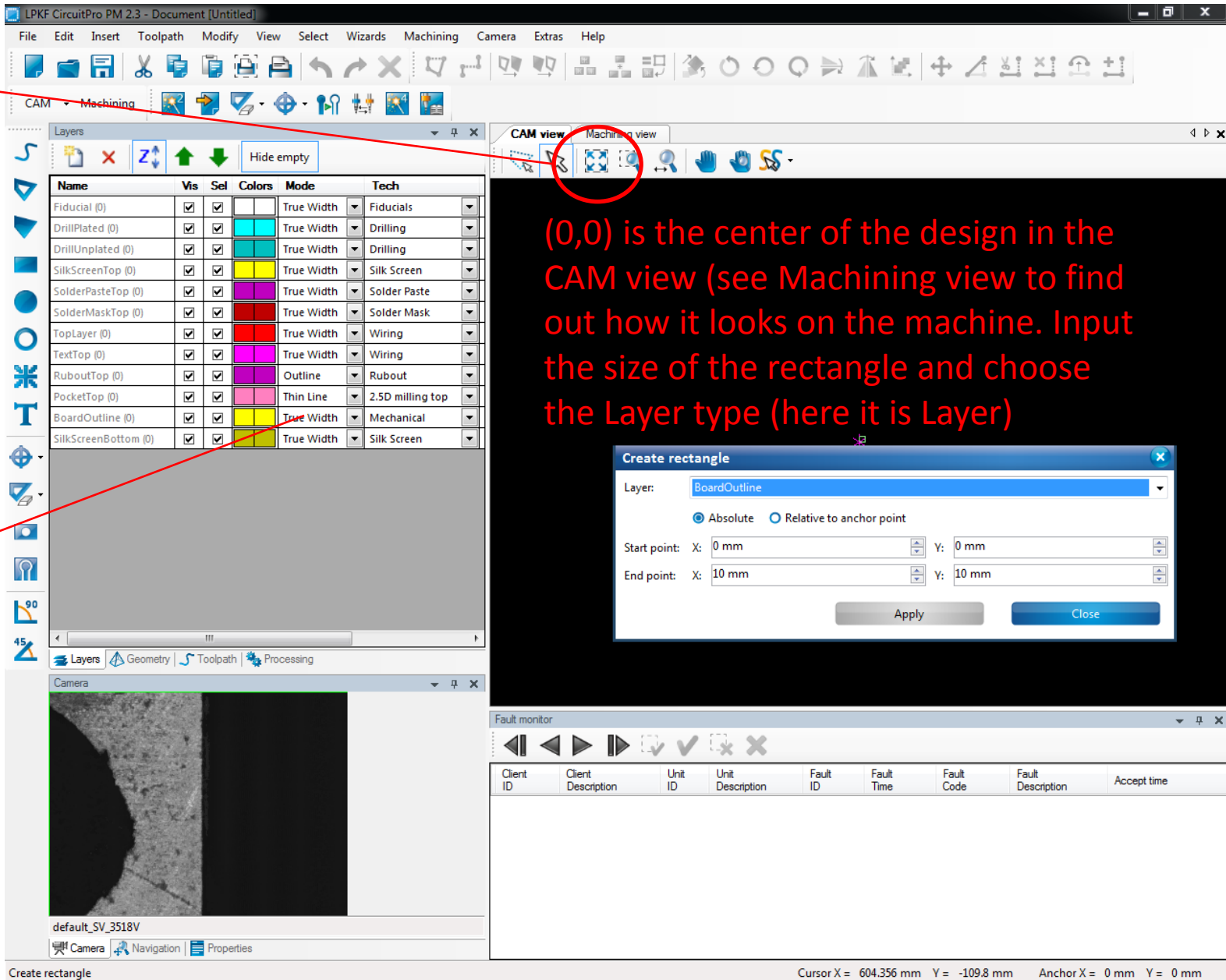




Load Defaults if you want to start from scratch and design your circuit (Otherwise, just import a file that you created with AutoCAD using "Open Project")

Load default template





To magnify

Choose Outline mode from dropdown menu here

(0,0) is the center of the design in the CAM view (see Machining view to find out how it looks mask on the machine. Input the size of the rectangle and choose the Layer type (here it is Layer)

Create rectangle

Layer: BoardOutline

Absolute Relative to anchor point

Start point: X: 0 mm Y: 0 mm

End point: X: 10 mm Y: 10 mm

Apply Close

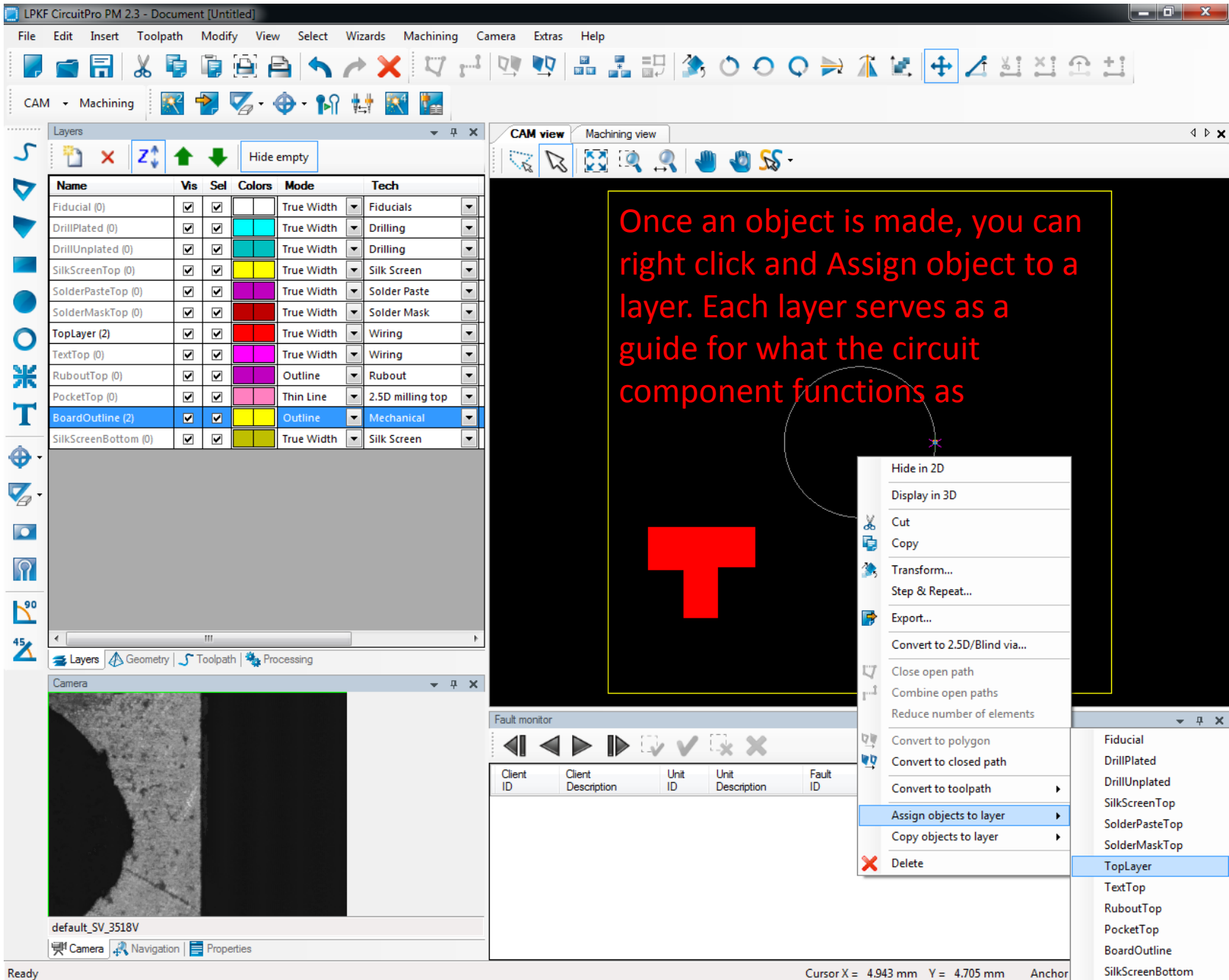
From Modify choose Step & Repeat to duplicate objects or use this

The screenshot displays the LPKF CircuitPro PM 2.3 software interface. The main window shows a CAM view of a PCB layout. A red circle highlights the 'Step & Repeat' icon in the top toolbar. A dialog box titled 'Step & Repeat' is open, showing the following settings:

- Repetition X: 1
- Repetition Y: 1
- Distance X: 2.134 mm
- Distance Y: 1.791 mm
- Combine to flash, list

The dialog box has 'Apply' and 'Close' buttons. Below the dialog box, a small preview shows a duplicated object. The bottom status bar displays the following information:

Repeating of geometrical objects in X, Y directions Cursor X = 10.488 mm Y = 9.088 mm Anchor X = 2.18 mm Y = 2.532 mm Filled area #1 (4:10 Point)



Create toolpath, Material Type is FR4 (copper and plastic) and the copper thickness is 64 um (make sure to change it correctly)

LPKF CircuitPro PM 2.3 - Document [C:\Users\higg150admin\Documents\people\Tafti Lab\Nikhil\Fazel_test_01.cbf]

File Edit Insert Toolpath Modify View Select Wizards Machining Camera Extras Help

CAM Machining

Layers

Name	Vis	Set	Colors	Mode	Tech
Fiducial (0)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>			
DrillPlated (0)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>			
DrillUnplated (0)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>			
SilkScreenTop (0)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>			
SolderPasteTop (0)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>			
SolderMaskTop (0)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>			
TopLayer (4)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>			
TextTop (0)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>			
RuboutTop (0)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>			
PocketTop (0)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>			
BoardOutline (1)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>			
SilkScreenBottom (0)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>			

Technology Dialog

Global process settings

Material type: FR4 Copper thickness: 18 µm RF application

Insulate

Insulation Method: Basic

Description: Insulation with a single insulation channel. Shortest processing time.

Contour Routing

Contour Routing Method: Corner gap

Description: Contour Routing with one gap in each corner.

Convert to Toolpath

Drills

Fiducials

Pockets

Start

Close

For Contour Routing, make sure to NOT choose the **No gaps** option

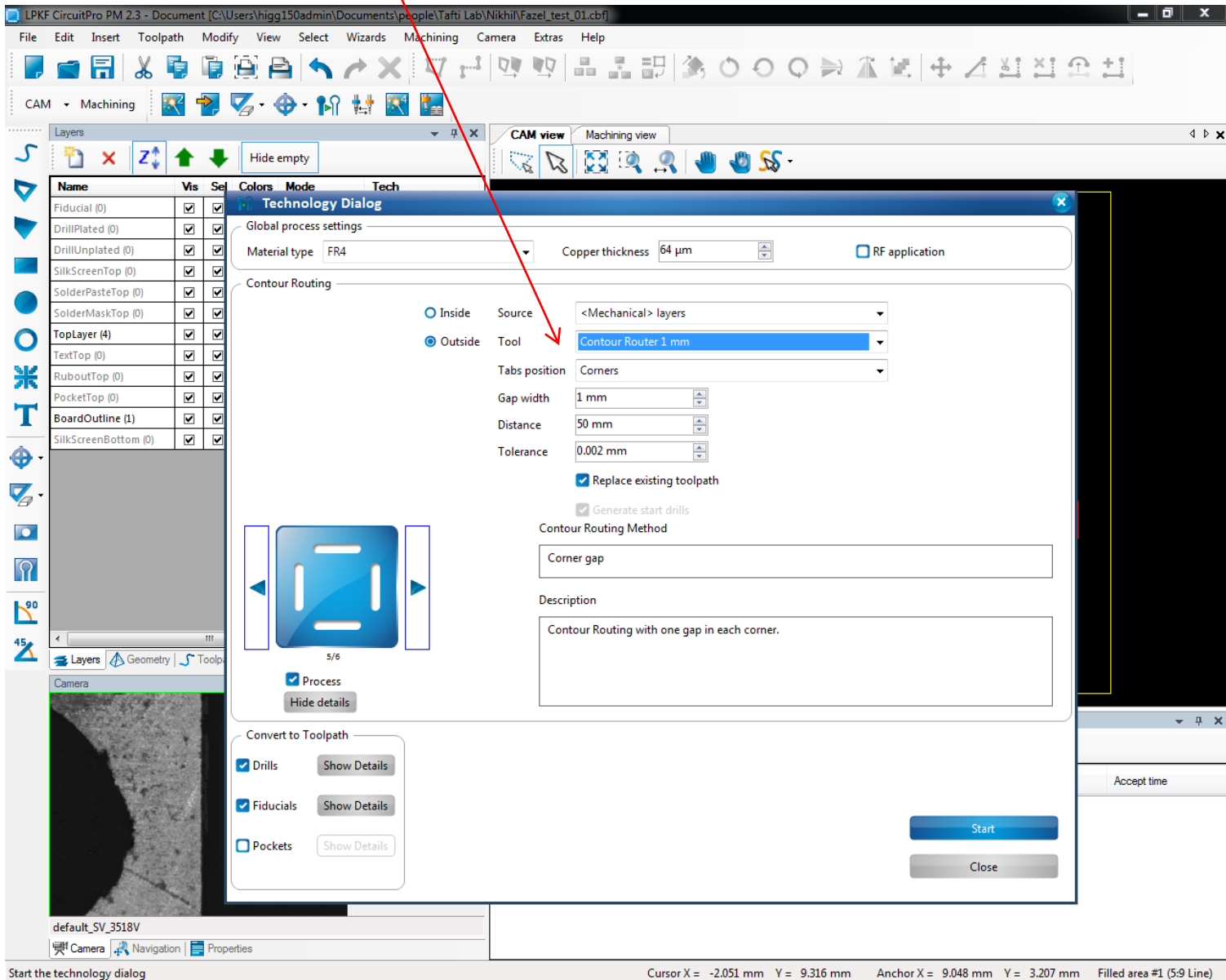
default_SV_3518V

Camera Navigation Properties

Start the technology dialog

Cursor X = -2.051 mm Y = 9.316 mm Anchor X = 9.048 mm Y = 3.207 mm Filled area #1 (5:9 Line)

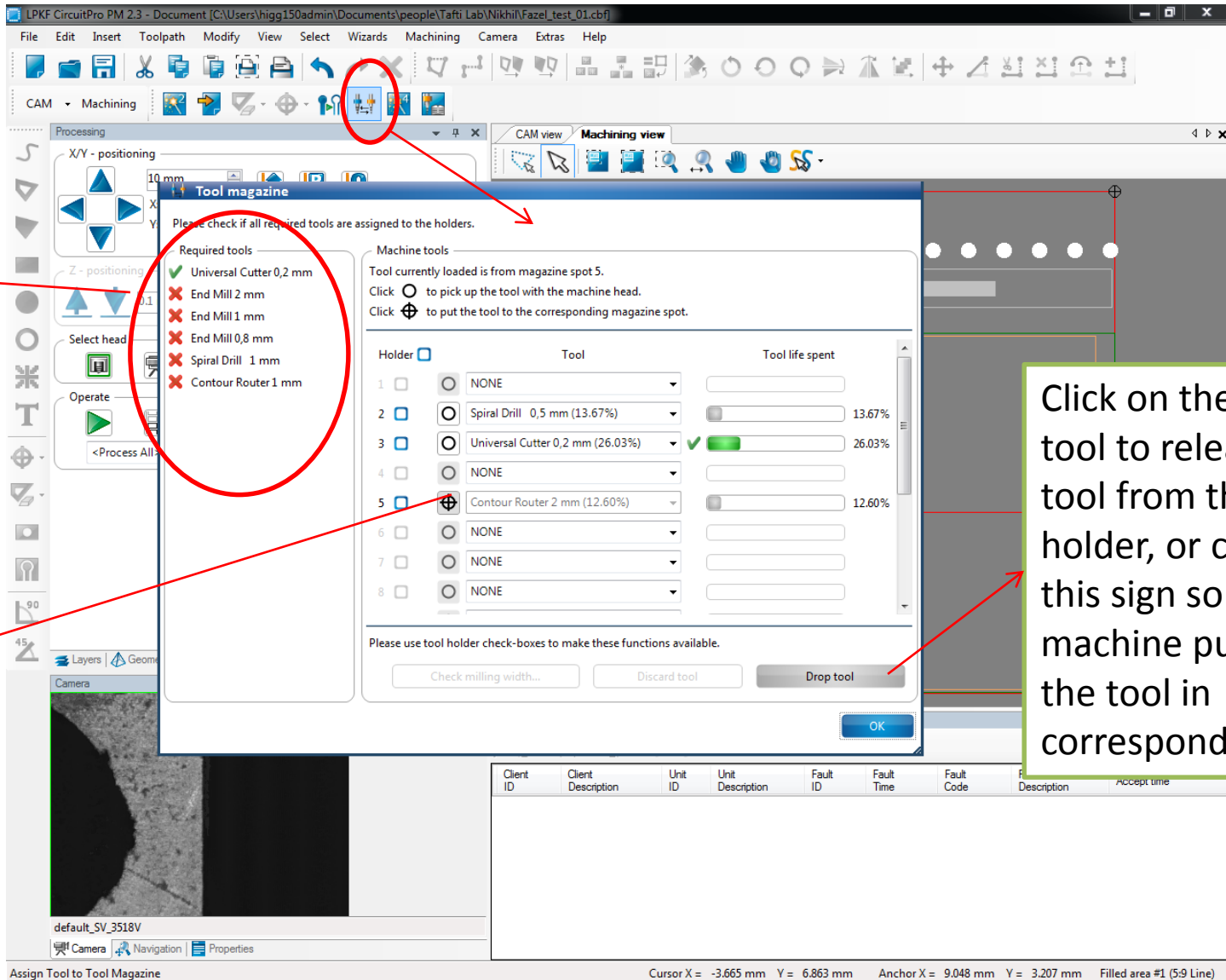
If the design is small (< 10 mm) go to details menu of the Contour Routing and choose a smaller Tool size



Make sure to save your design in “Tafti Lab” folder under your name. Then click on the Tool magazine to open its dialogue box

You need all of these tools

This tool is already inside the tool holder



Click on the drop tool to release the tool from the holder, or click on this sign so the machine puts back the tool in corresponding spot

Teach tool-holder positions by adjusting the last and the first spots

The screenshot displays the LPKF CircuitPro PM 2.3 software interface. The 'Machining' menu is open, and the 'Teach tool-holder positions...' option is highlighted. A red arrow points to the 'XY - positioning' section in the 'Processing' panel, which shows X: 330.000 mm and Y: 325.000 mm. The 'Z - positioning' section shows Z: 0.000 mm. The 'Camera' window shows a live feed of the workpiece. The 'Fault monitor' table is empty.

Processing Panel:

- XY - positioning: 10 mm, X: 330.000 mm, Y: 325.000 mm, Mouse cursor
- Z - positioning: 0.1 mm, Z: 0.000 mm, Depth limit mounted
- Select head: [Icons]
- Head actions: [Icons]
- Operate: [Icons], ContourRouting
- Tool information: No tool inform

Machining Menu:

- Process all
- Stop processing
- Add production phase
- Placement...
- Fiducials...
- Measure milling width...
- Alignment...
- Drill reference holes...
- Step loss check wizard
- Service...
- Determine camera head offset...
- Teach tool-holder positions...**
- Connect...

Camera Window: default_SV_3518V

Fault monitor Table:

Client ID	Client Description	Unit ID	Unit Description	Fault ID	Fault Time	Fault Code	Fault Description	Accept time
-----------	--------------------	---------	------------------	----------	------------	------------	-------------------	-------------

Status Bar: Ready, Cursor X = -59.993 mm, Y = 55.047 mm, Anchor X = -0.499 mm, Y = 9.392 mm, Drill #2

From Wizards use Board Production Wizard and follow through.

The screenshot displays the LPKF CircuitPro PM 2.3 software interface. The 'Wizards' menu is open, showing the following options:

- Equipment configuration wizard...
- Process planning wizard...
- Board Production Wizard...
- Start the Board Production Wizard
- Dispense process wizard...

The main workspace shows a 'Machining view' of a PCB layout with a red bounding box and a small tool icon in the center. The left sidebar contains various tool icons, including positioning, head selection, and tool information. The bottom status bar shows the following coordinates: Cursor X = -70.521 mm, Y = 120.695 mm, Anchor X = -0.499 mm, Y = 9.392 mm, Drill #2.

Ready

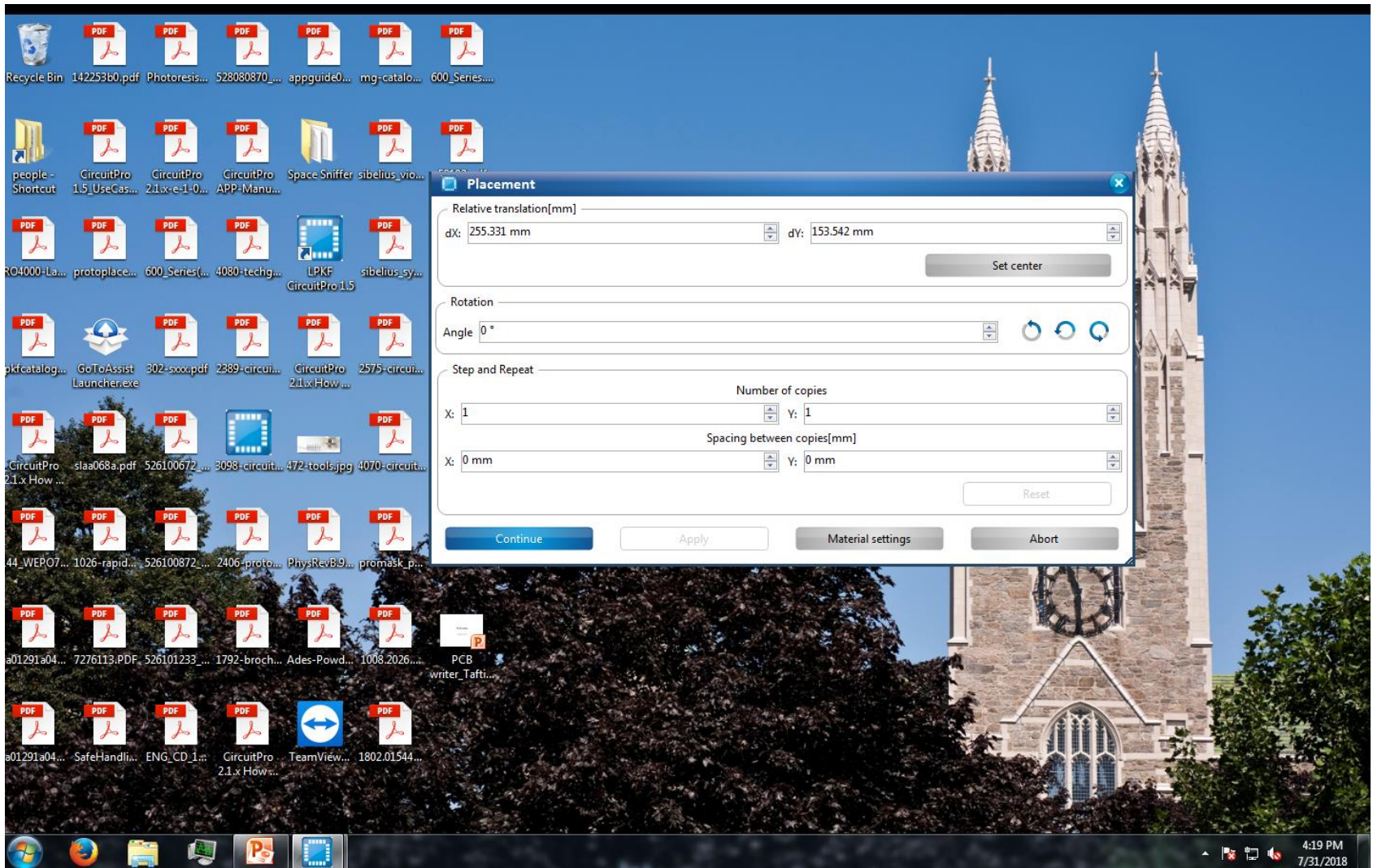
Make sure to turn on the vacuum to fix the board in place and prevent dust accumulation during production.

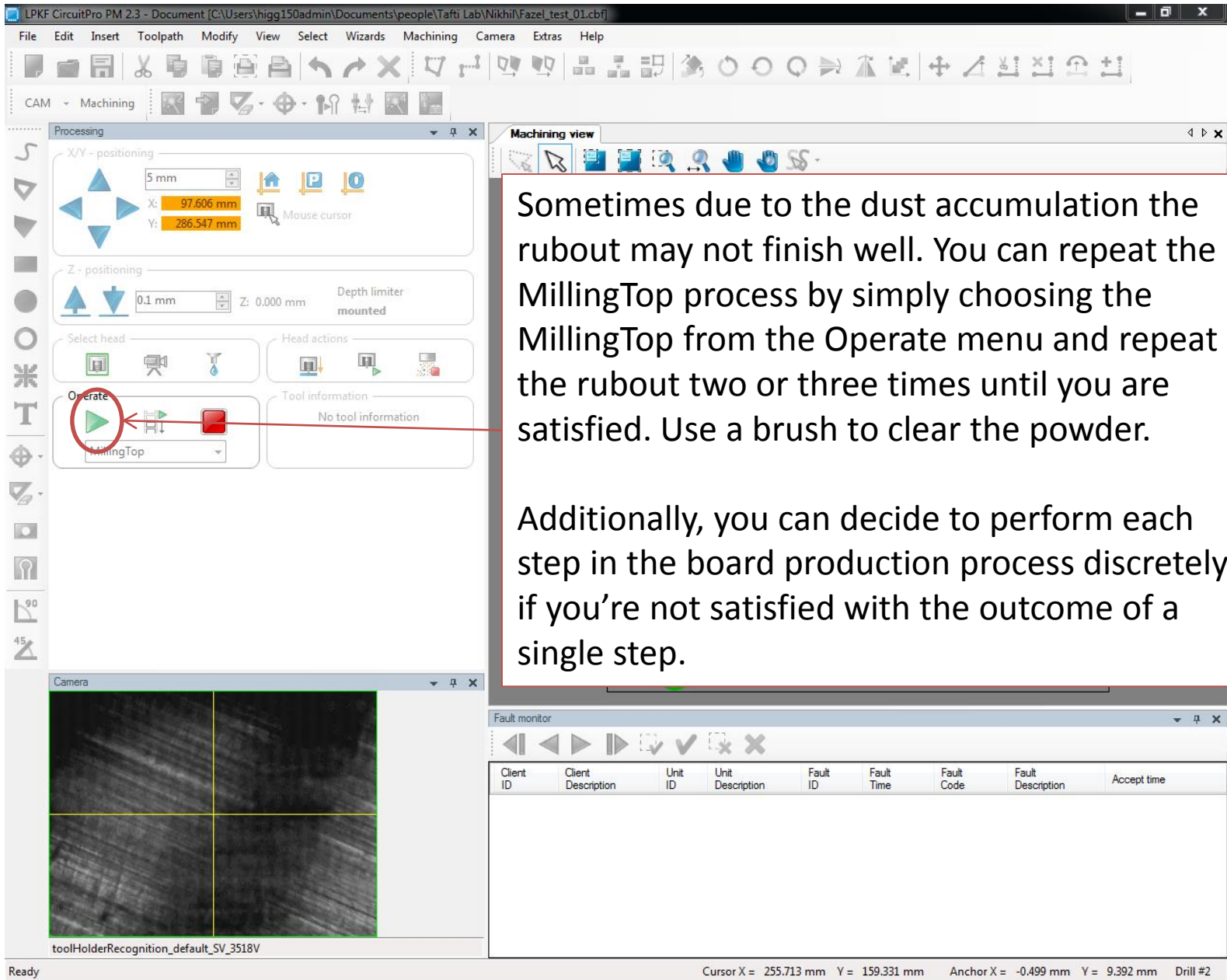
The pump is under the machine.

Make sure the board is firmly stuck to the stage with masking tape.



The camera view is offset from the tool holder position (i.e. you cannot trust the camera view when choosing your material placement). You need to visually inspect the board and place (drag and drop) the tool holder on a good spot.





Sometimes due to the dust accumulation the rubout may not finish well. You can repeat the MillingTop process by simply choosing the MillingTop from the Operate menu and repeat the rubout two or three times until you are satisfied. Use a brush to clear the powder.

Additionally, you can decide to perform each step in the board production process discretely if you're not satisfied with the outcome of a single step.



Once you are done with the board production, use a flathead screwdriver or an old drill bit (in the “container” cup next to the PCB writer) to pry off the edges of your board. Alternatively you can remove the tape and then pop off the board using your fingers and a little bit of pressure but that might take a little longer and more steps...

You might then want to manually sand the edges using sandpaper to remove any of the sharp, deformed corners that result from popping out the circuit board from the substrate.

Then, go to Machining > Connect > Disconnect, to disconnect the machine. Once the machine has disconnected you can turn off the machine and quit the software. Don't forget to turn off the vacuum as well.