makerlab

CNC

Using the FlexiCam CNC and HMI Software

Guldbergsgade 29N, P0
E: makerlab@kea.dk
T: +46 46 03 90

This grey box is the NC controller. Let's start by turning the red switch to the ON position, then press the green button. You should hear a light thump sound that confirms the machine is on.
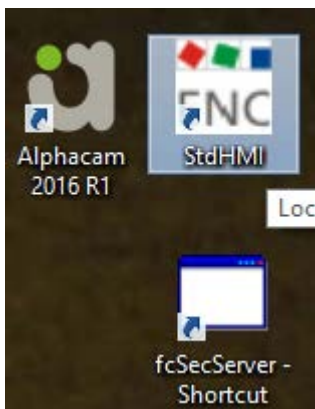
Remember to turn the compressor on before you start your job. 3 steps: start it with the green button on front, turn on cooling with the green switch on the side, and turn the red valve to let out the compressed air.
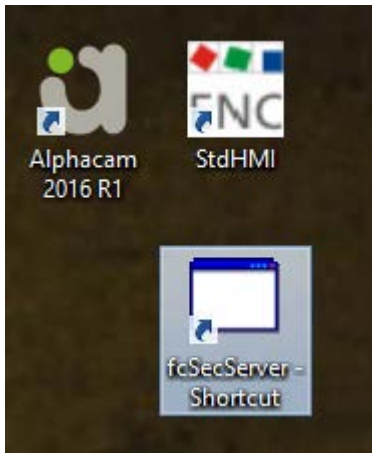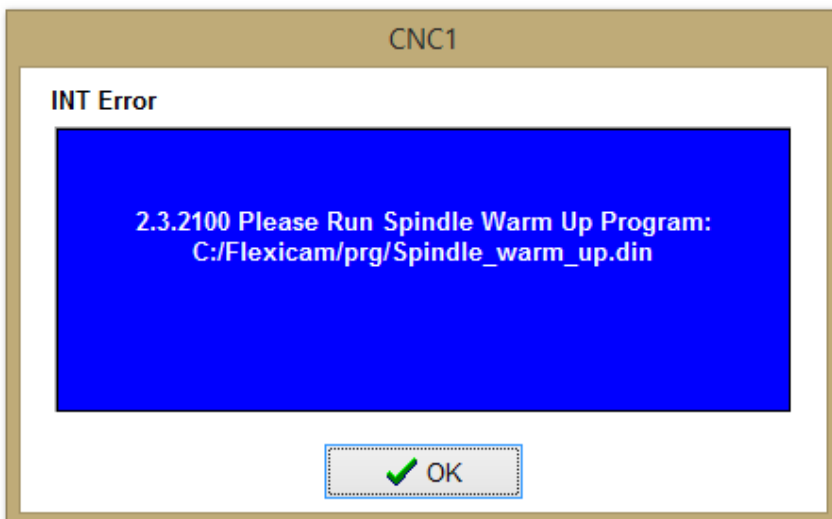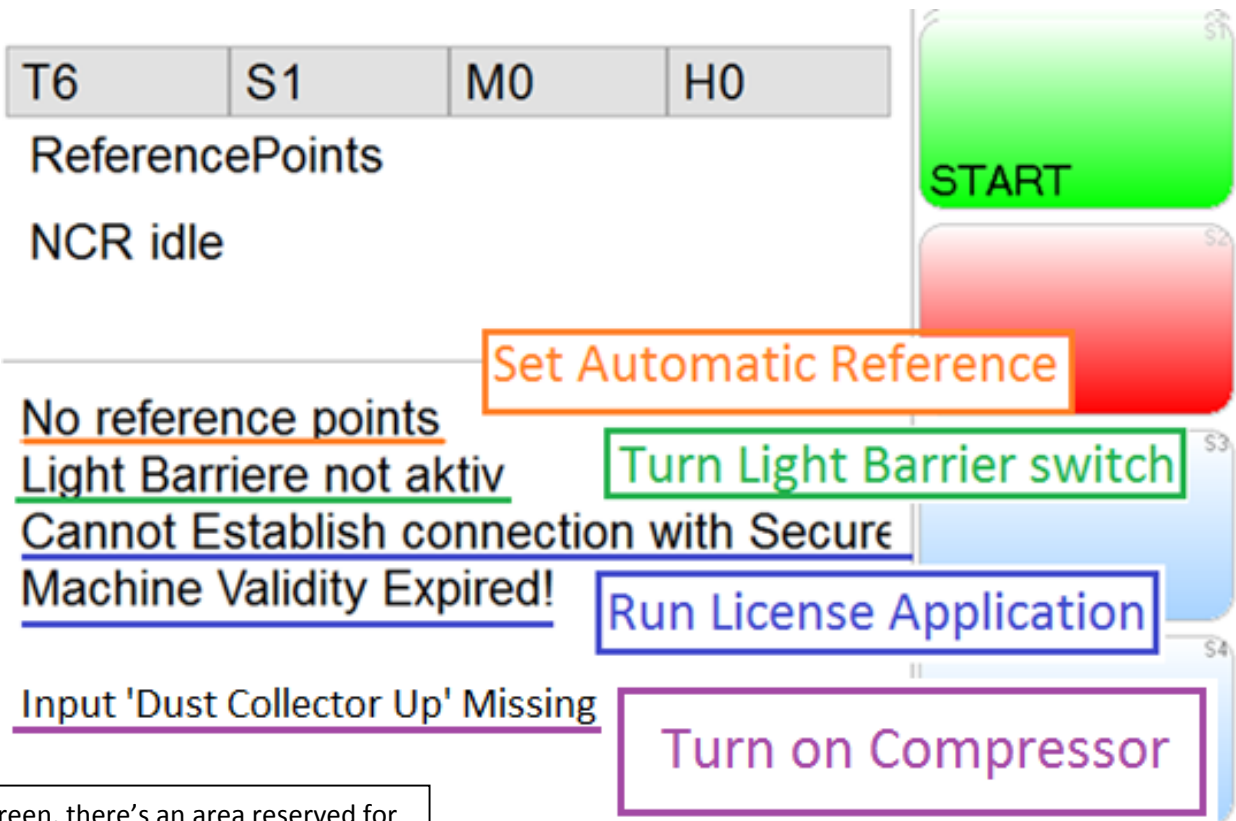
Firstly, run the fcSecServer application on the desktop. No other action is required, just let it run in the background. This connects to the FlexiCAM server and makes sure the license is up to date.

Next, open the StdHMI icon from the desktop to open the Flexicam CNC controller software.

CNC1

INT Error

2.3.2100 Please Run Spindle Warm Up Program:
C:/Flexicam/prg/Spindle_warm_up.din

✔ OK

This warning might show up. Hit OK, it's fine.

T6	S1	M0	H0

ReferencePoints

NCR idle

START

Set Automatic Reference

No reference points
Light Barriere not aktiv
Cannot Establish connection with Secure
Machine Validity Expired!

Turn Light Barrier switch

Run License Application

Input 'Dust Collector Up' Missing

Turn on Compressor

On the main screen, there's an area reserved for displaying warnings, many of which will stop the machine from being able to run programs.

The picture shows common problems that might show up when first booting the software. Here's how to fix them:

CNC1

ZST Error

2.9.1108 PLC reports emergency stop

✓ OK
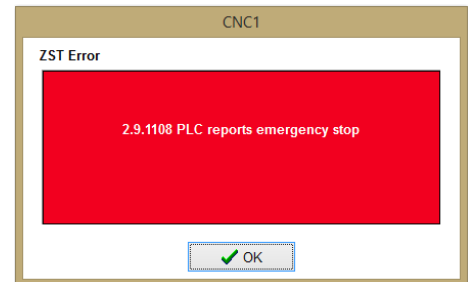
Note: if a popup says the emergency stop has been reported, it's probably the green button on the NC that wasn't pressed.

Alphacam 2016 R1

StdHMI

fcSecServer - Shortcut
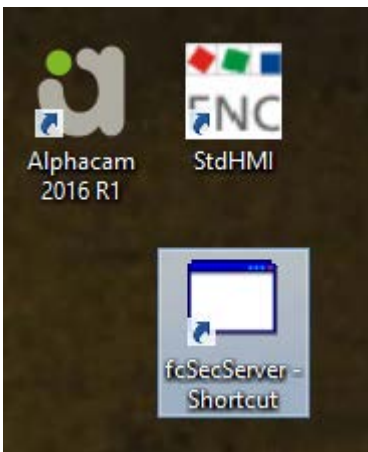
1. Run License Application

If you opened the fcSecServer application, you may just have to wait a few seconds for this warning to disappear. If you didn't, go to the desktop and run fcSecServer.

2. Turn Light Barrier Switch

The yellow poles around the CNC make up a light barrier. A laser is reflected between them – if anything passes in-between the yellow poles, the machine will shut off immediately. This is a safety feature to ensure nobody gets too close to the machine while any parts are moving.

This switch resets the barrier. Turn it back and forth – once in the clockwise position, once counter-clockwise – to activate the barrier. The light barrier warning in the HMI will disappear once the barrier is active.

Note: most people using the workshop don't know about this feature and often will walk through the barriers or put materials through, breaking the light barrier during a job. It's worth talking to whoever is in the shop and telling them about this so you save time and embarrassment.

You can resume the CNC at any time after the light barrier is broken during a job. Just flip the switch and press "START" in the HMI software.
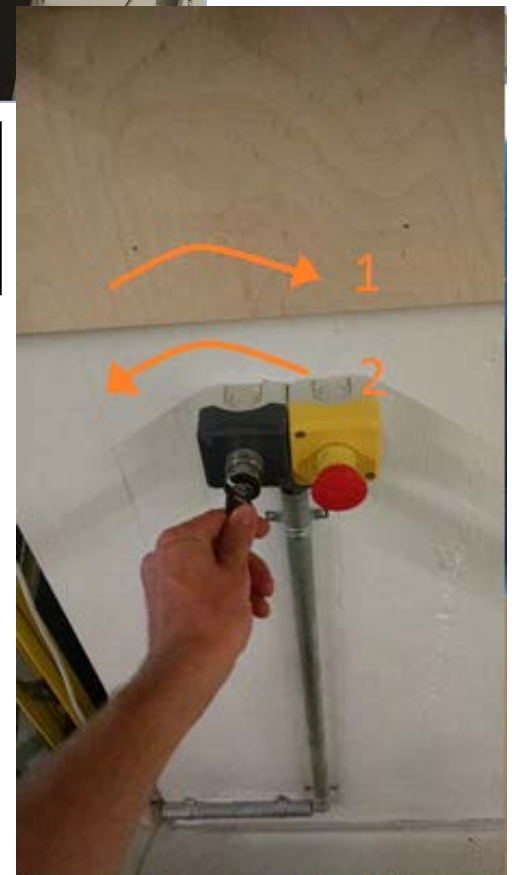
3. Set Automatic Reference

When the CNC is turned on, it needs to run a quick check to make sure what exact coordinates the head is at. This step will move the CNC head to the corner closest to the desk – a zero point that will calibrate the machine coordinates.

Important: Make sure nothing is on or around the table! The shiny metal surface on the left will fold out and dump off anything that's sitting there, and the drill head will drag into anything in its way. Clear off the table of any rulers, scrap materials, etc.

Setup	Automatic	Program	Diagnostics		

Setup

15:28:12

	Actual	Nominal		T6	S1	M0	H0
X	-12.404	-12.404		ReferencePoints			
Y	-116.444	-116.444		NCR idle			
Z	179.131	179.131					

No reference points
Light Barriere not aktiv

START

STOP

100
0 Override [%]   75   150

Override +

Override -

Reference ...	Manual travel ...	Zero points ...	Tool management ...		Documentation / Help	Input	Exit

After selecting the Setup tab up top, go to "Reference" in the bottom menu.

Automatic	Manual

Select "Automatic" in the bottom menu, then if there are no remaining warnings, press "START" to bring the head to the origin.

The CNC will now have reference points.

The NC controller has built-in memory that it uses to store the build file. Let's load it in.
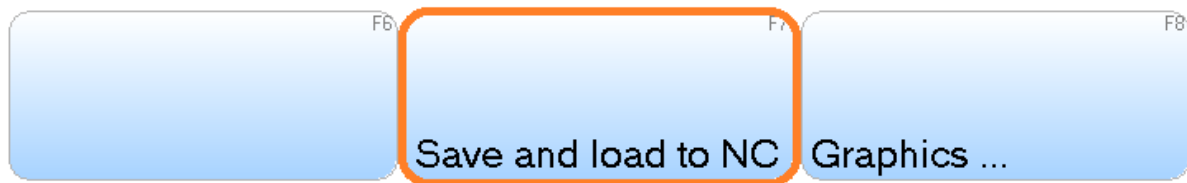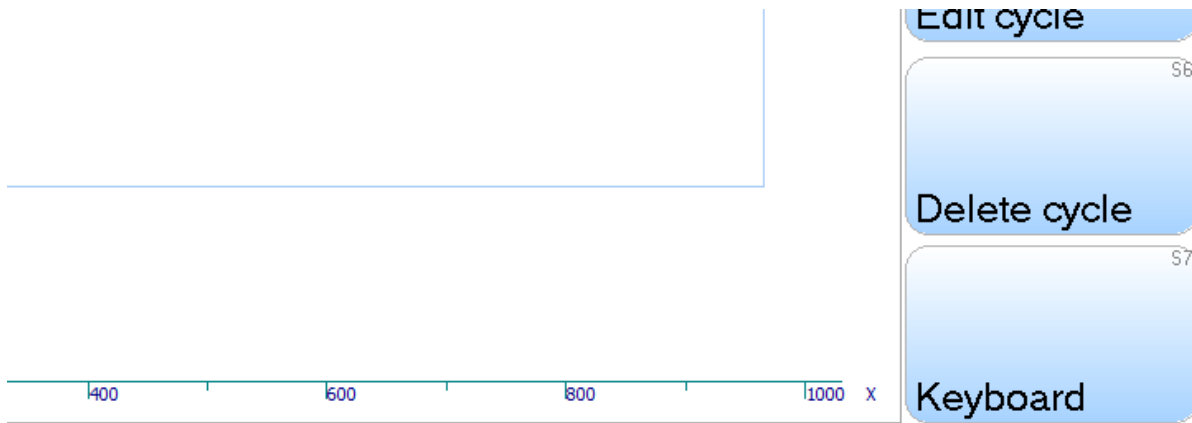
Note: In most cases, the file will load just fine. In certain cases, especially for very large and complex 3D surfaces, additional steps will need to be taken. We'll get to that later.
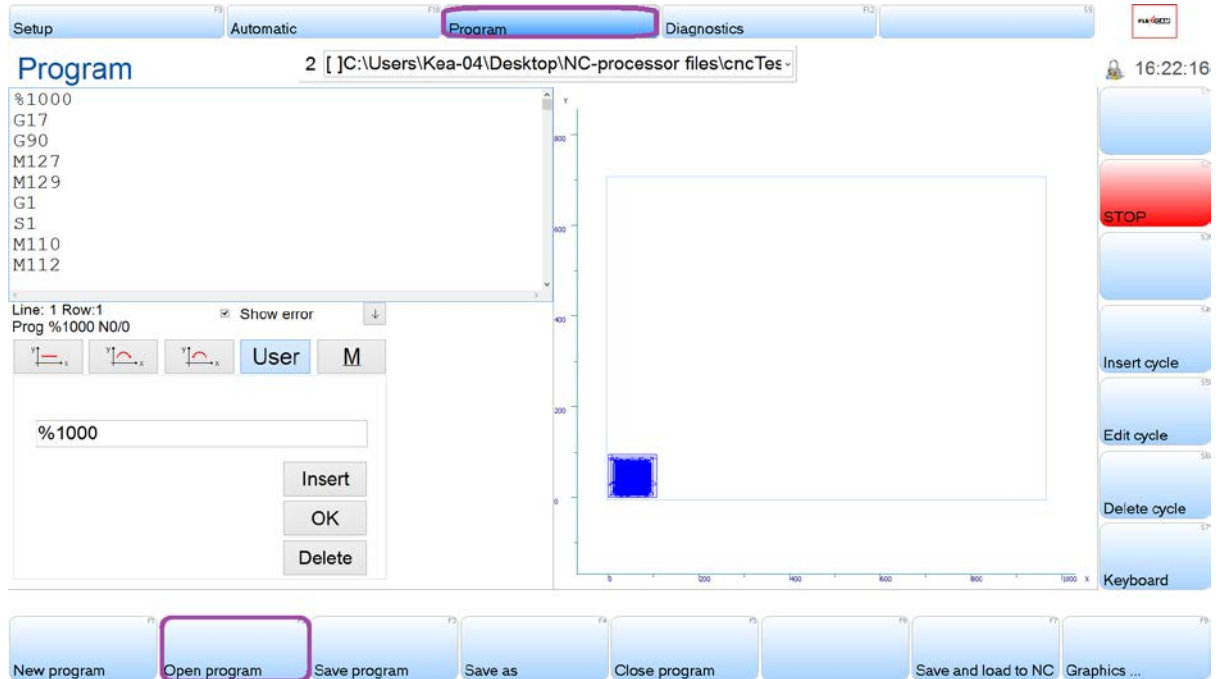
Under the "Program" tab up top, select "open program" in the bottom menu.
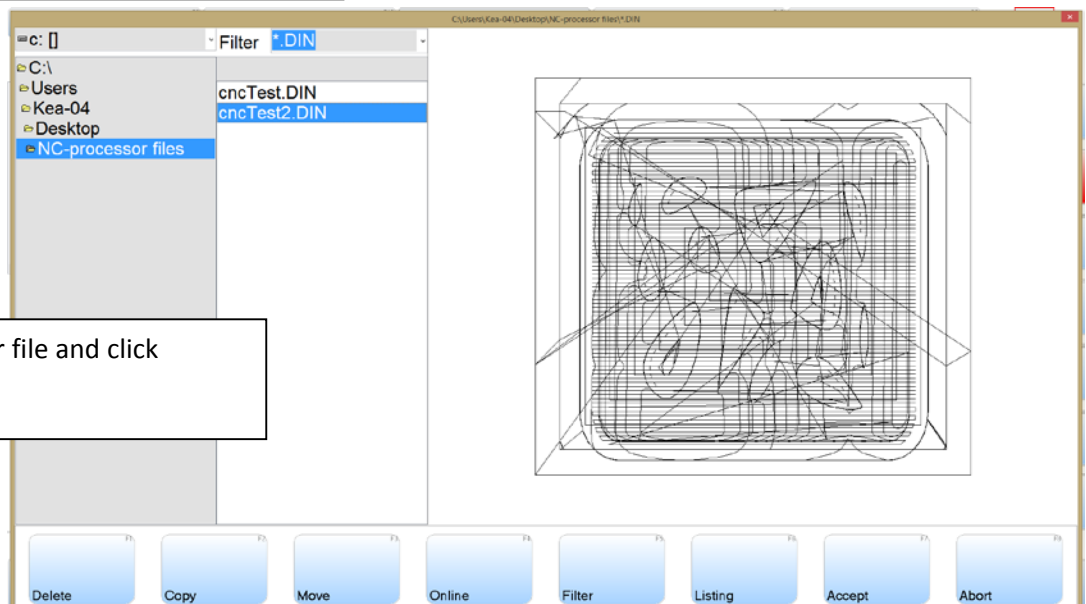

Navigate to your file and click "Accept".

Edit cycle

S6

Delete cycle

S7

Keyboard

F6

Save and load to NC

F7

Graphics ...

F8

Important!

Hit 'Save and load to NC' if the preview looks good. This step is easy to forget, so pay close attention.

600

400

200

0

0   200   400   600   800   1000   X

This is the preview window now that the file has been loaded.

We need to set the zero point on our material now. If you imagine this preview window as the CNC cutting area, our zero point is going to be on the bottom left of the top plane of our material.

Side note: "Online Program" function
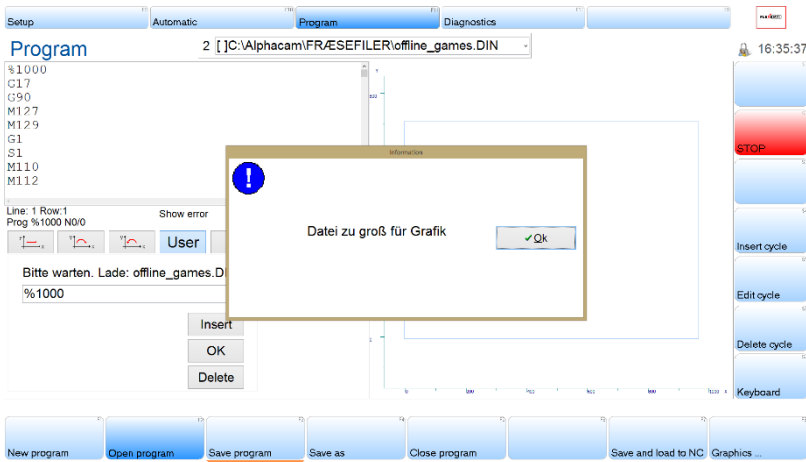
In some cases, especially when working with files with complex 3D surfaces, the file will be too big to open.

To get around this, select "Automatic", then "Online Program",
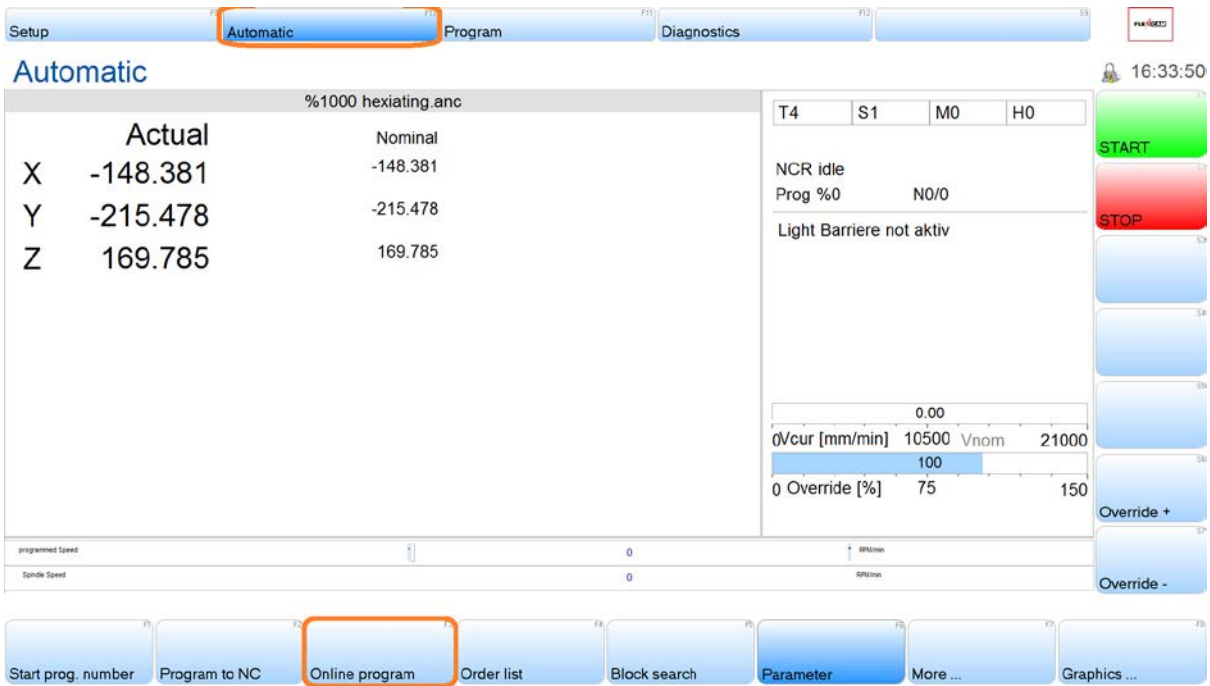
72% [4910 Kbytes] W A I T !

The NC has a limited amount of memory. The "Online Program" function loads in only a portion of the file at a time; when the machine has finished with that portion, the software will load the rest in.

When you're putting a new tool in the rack, make sure the machine isn't holding a tool that belongs in that spot.

Once you're sure the spot is free, or the machine isn't holding a tool, slide in the tool holder manually into a spot on the rack and make sure it matches the position in the software.
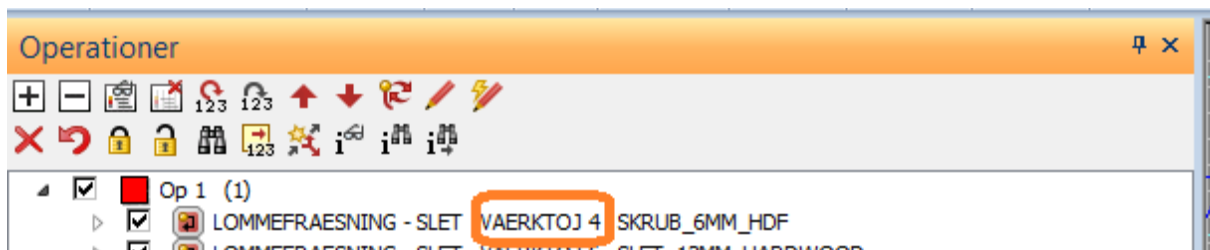
As an example – the first tool on my list of operations in Alphacam is the 6mm HDF roughing endmill, and it's set to tool #4.

Making sure the machine isn't currently holding a tool in position 4, I'll slide the tool into tool holder 4, endmill pointing down, with the top of the chuck at the same level as the top surface of the white tool holder.

You can manually load and unload tools into the machine.

To remove a tool, hold it with one hand and press the green button above. It's pneumatically released. Take a moment to enjoy the sound.

The same principle loads the tool back in – hold the tool in place with one hand and press the button.

To avoid confusing the machine, it's best to load the tool in for the first time using the white tool holders.

100

0 Override [%]　　　75　　　　　　　　　150

S6

Override +

S7

Override -

Documentation /
Help

Input

Exit
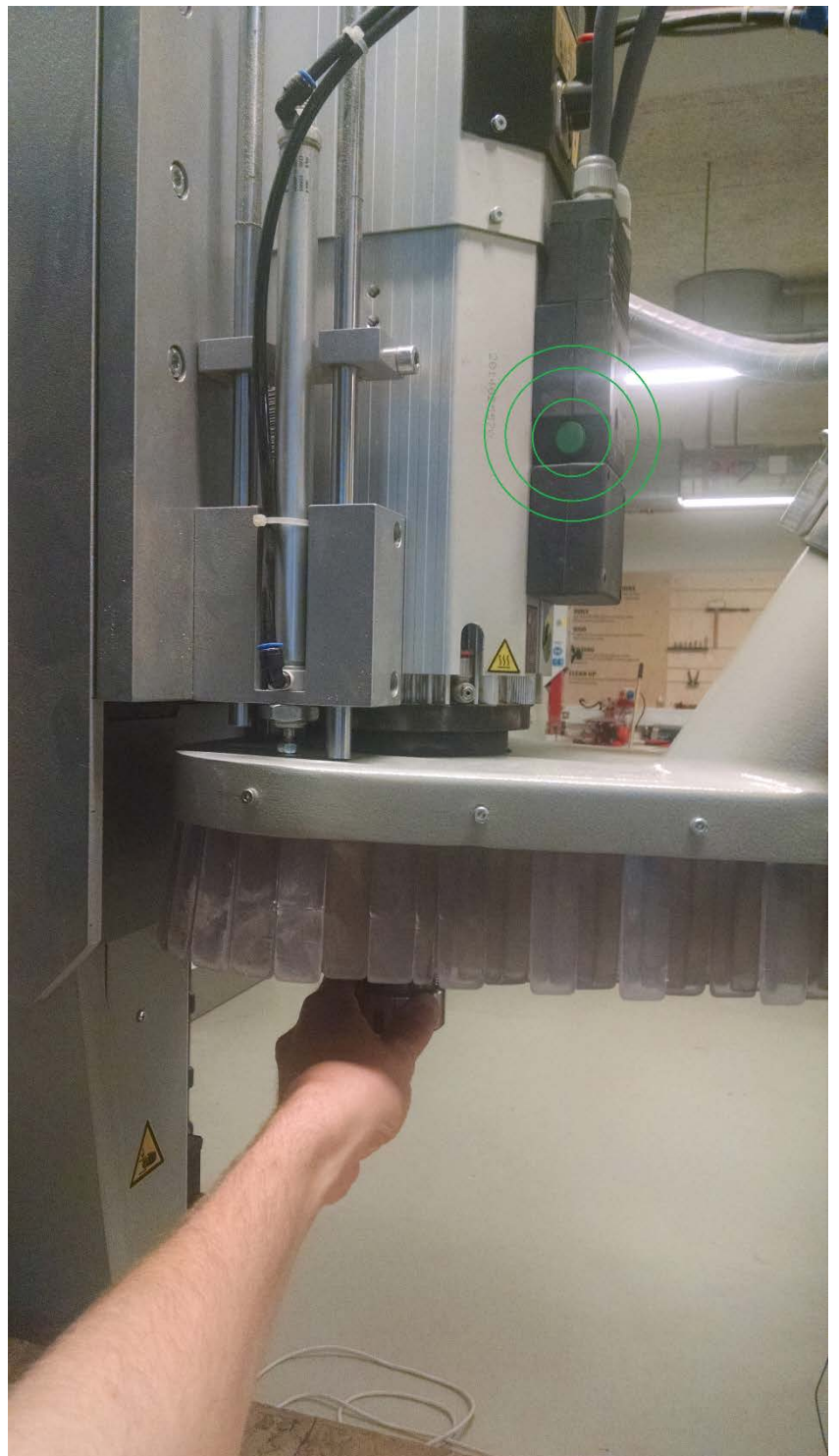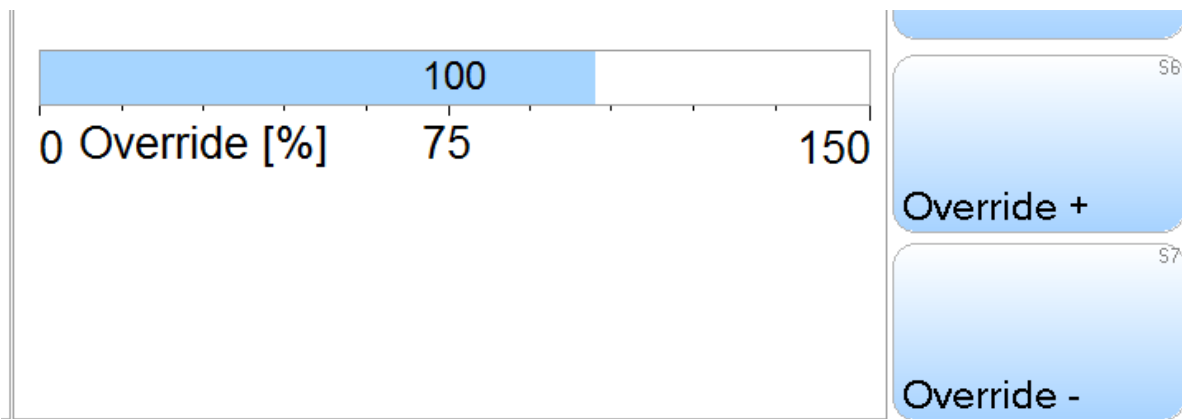
Once your tools are in the rack, grab them using the Input command M6 Tx, where x is the position in the rack the tool is (for example, M6 T3 for a tool in holder 3).

Once it's in the holder, we need to let the machine know how long the tool is. Use the input command M7 to automatically measure the tool. This might take a minute as the machine slowly lowers the bit onto a touch plate to tell exactly where the bottom is.

Make sure to do the M7 command on all of the tools that are loaded in the tool holders. – first M6 Tx, then M7. The machine will first automatically place the tool it's holding in the proper tool holder, then pick up the one you've specified.

Also make sure to measure the tools **before** you set the zero point – the CNC needs to know how long the tool is first.

Most jobs will use the vacuum table to hold the material down, which is the easiest solution on our CNC.

We use a thin "sacrificial board" as a surface that covers the metal vacuum table. The HDF we use as our sacrificial board has enough tiny holes in it that when we pull a strong vacuum on the underside of it, whatever material is on top will stick down flat. This way if a cutting bit plunges a little too deep or if it's cutting all the way though a material, the bit won't grind the metal table.

It's best to choose a smooth, flat material that will stick to the vacuum table.
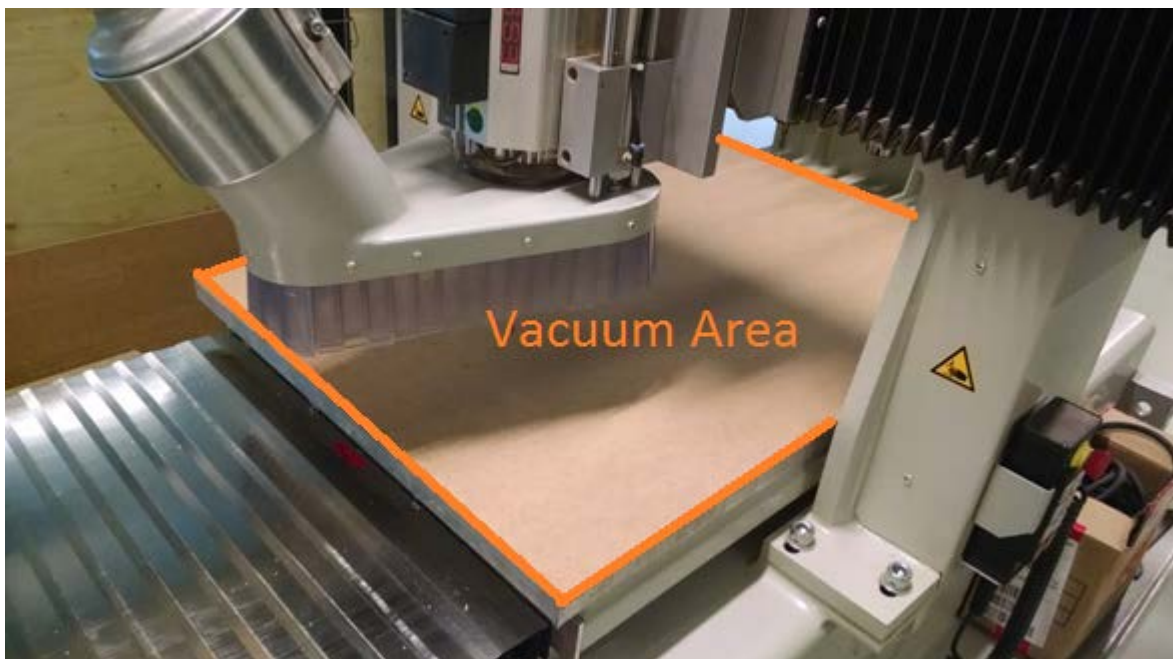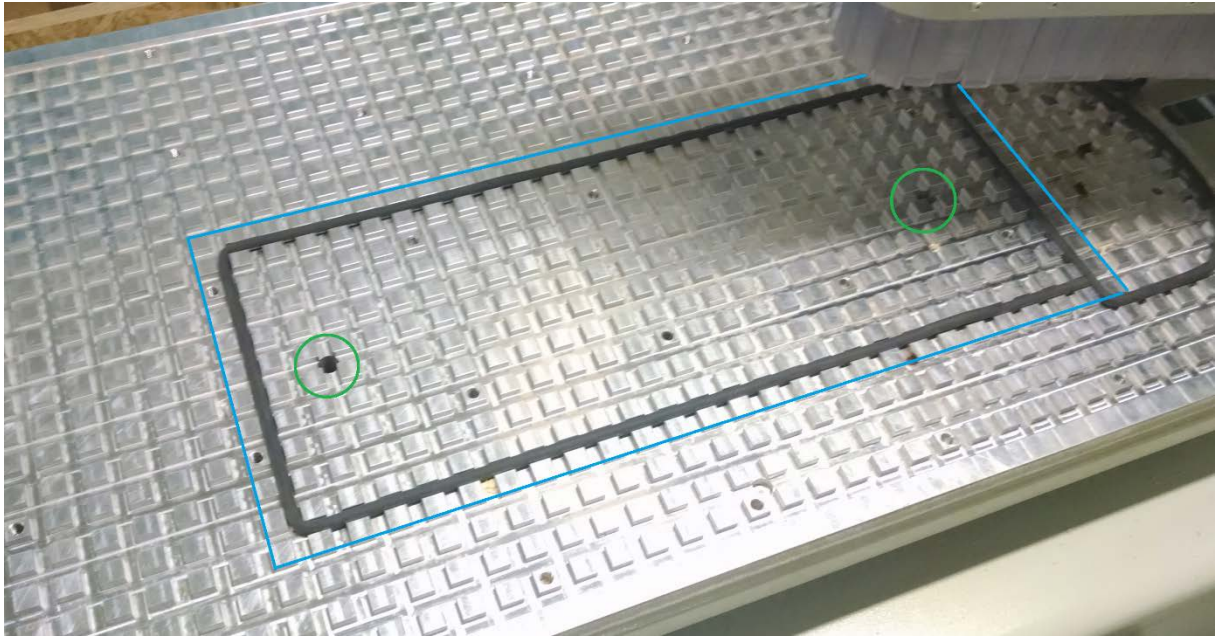
The vacuum is pulled through two holes in the metal table, shown here in green. The border material keeps an air-tight seal.

If you don't need to use the full table size, wrap the black border around so it makes a full loop under your material. The vacuum pull will be weaker if it pulls on a lot of empty space where there's no material.

Make sure that the end is fully flat and air-tight up against another border.

If you choose to only use one of the holes, remember to block up the other one!

The vacuum will be pulled through both, and your material won't be held tightly if all the pressure is leaking through an open hole.
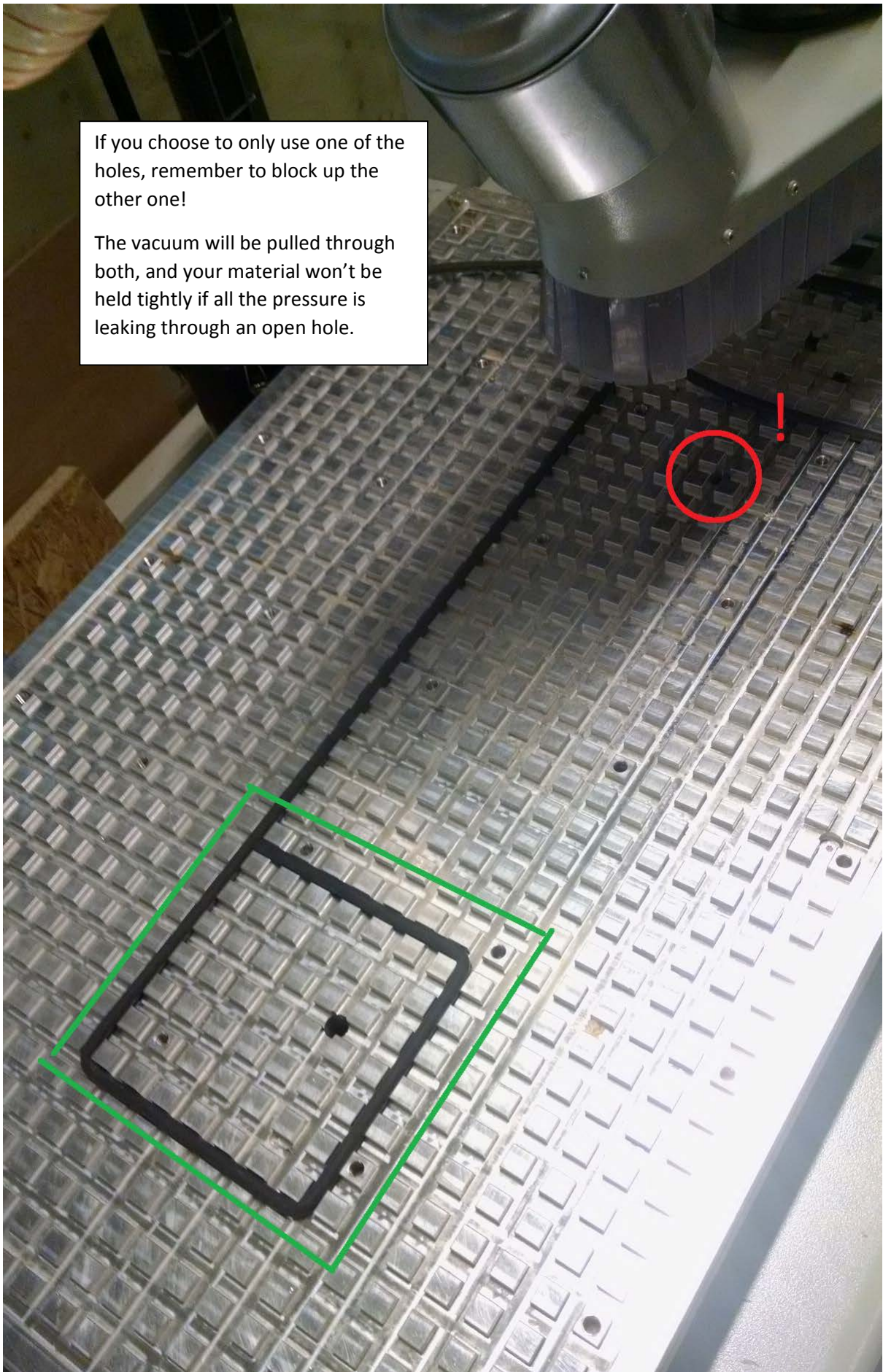
In this example, the milling doesn't go all the way through the material – there's still solid material between the bottom of the cut and the metal table.

We don't really need the sacrificial board in this situation. By putting this small block right on the black border material, we can be sure it's not going to move at all. We'll block up the other vacuum hole.

Note: if you're not sure, always use the sacrificial board.
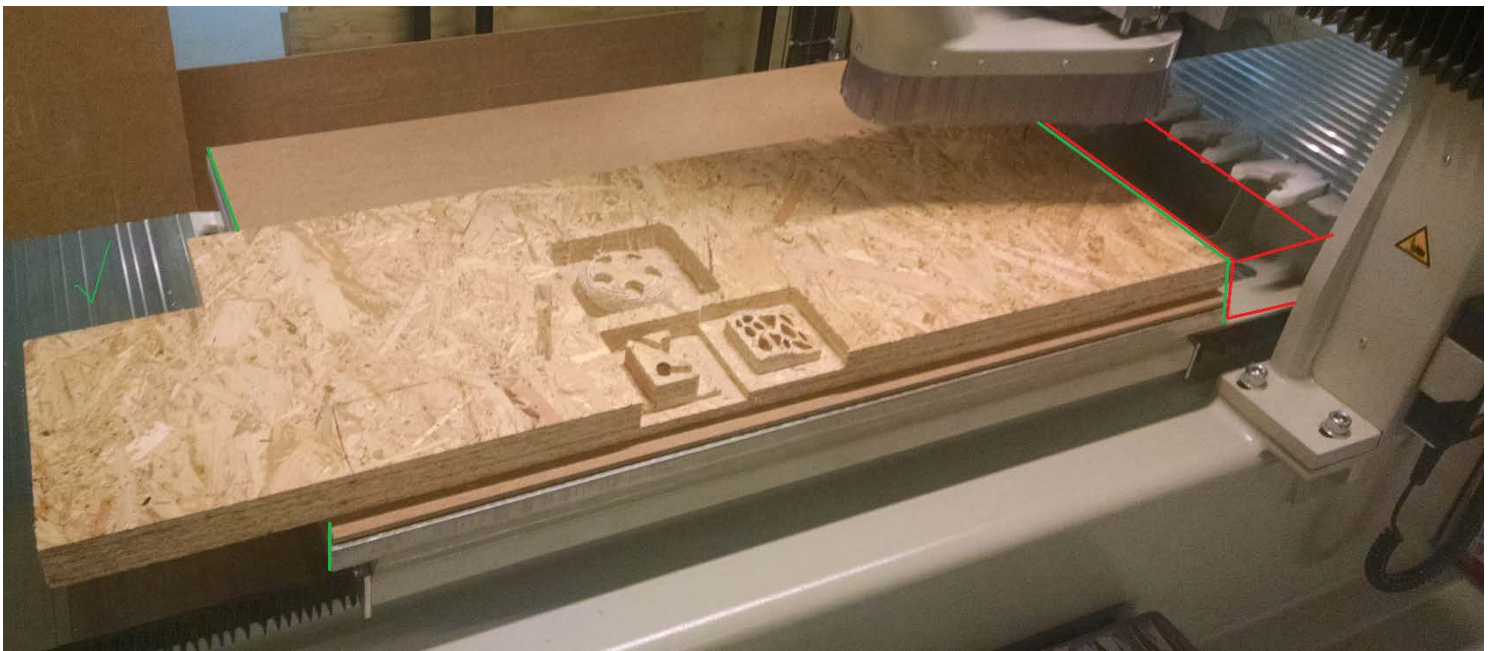
Here we've got our material laid flat on the vacuum table with the border material fit to size under the sacrificial board.

Note: the material extends out off the table to the left. This is beyond the area that the machine can mill, but that's fine.

Make sure any excess material does not extend to the right – the CNC head needs this area to swap tools. Any material in that area might damage the tool or the head.

This pedal activates the vacuum pump.

Once your material is in place with the border material correctly set, hit the pedal to test that it's held in place firmly. The air compressor needs to be on, and the HMI and the license software both need to be running for the pedal to work.

Listen for a loud hissing sound coming from under your material– if you hear one, it probably means there's a break in the air seal. Try re-laying the border material and making sure the corners are definitely tight.

If you see the material firmly held down and not shifting, use some muscle and really try to move it. If you can easily shift the material around on the table, so can the CNC bit. If it really seems firm and tight on the table, you're good to go.
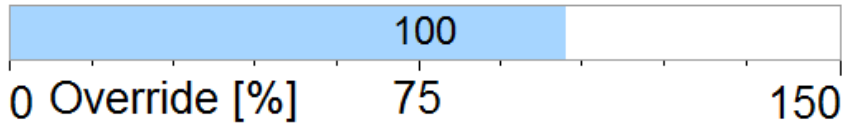
Milling Area
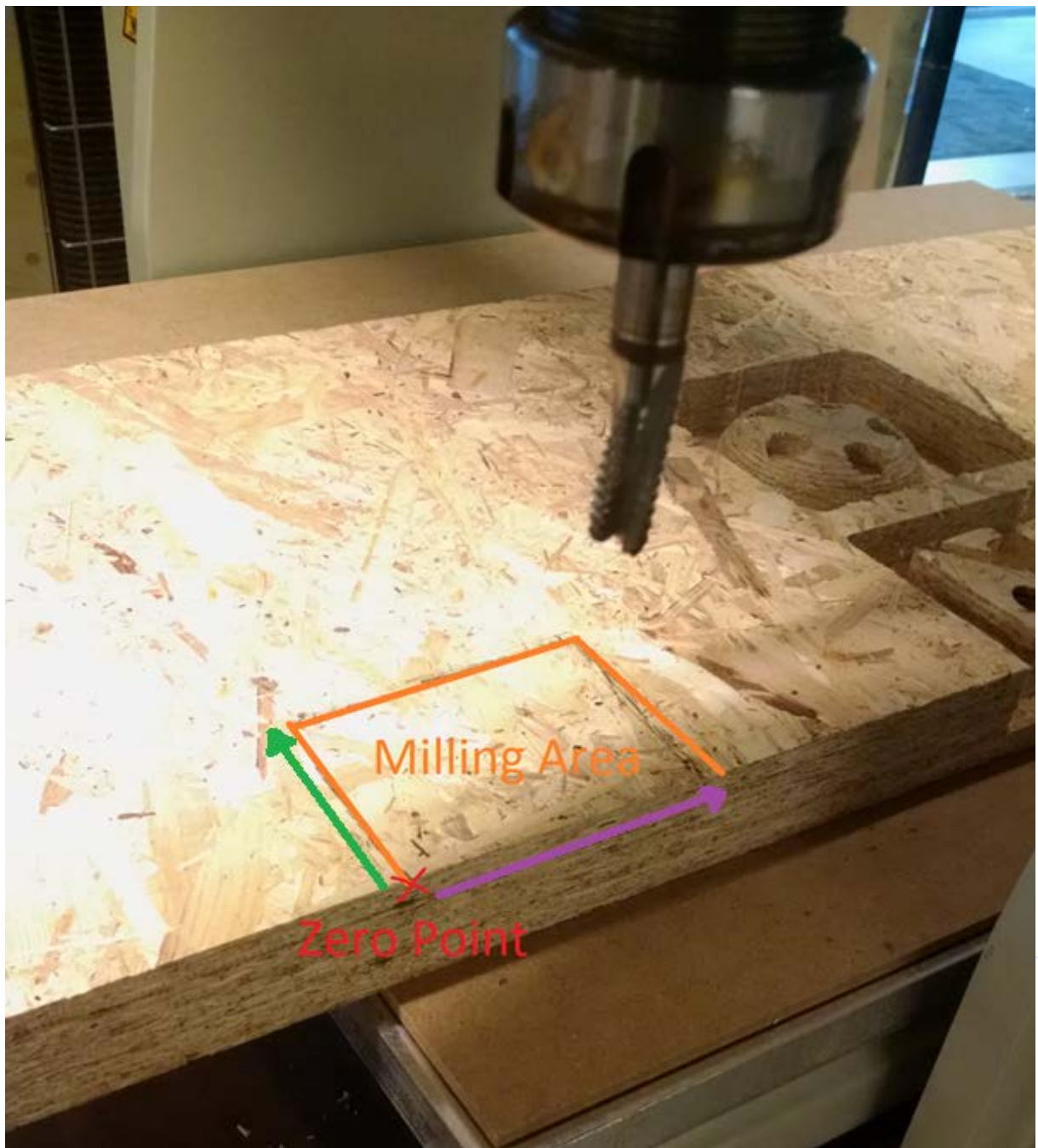
Zero Point

Our file's zero point is the bottom left of the shape. Let's set our zero point right on the material's corner to save room.
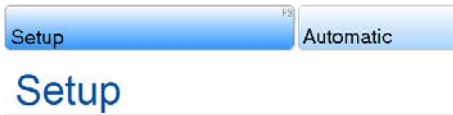
We're going to touch the bit exactly where we want the zero point to be.

First, look at the bit that's currently in the tool head – we can be more precise with a thinner bit. Let's swap out the tool with a smaller one.

0, 0

	100			
0 Override [%]	75		150	

Override +

Override -

Documentation / Help

Input

Exit

Setup | Automatic

Setup

Actual

M6 T2

ESC	1	2	3	
Tab	Q	W	E	

...then put in the command to swap out the tool.

The Input menu is where you can manually type in commands to control the hardware of the machine. Note: you can see a full list of these commands by clicking on "Documentation", then "Machine Functions" in the bottom menu under "Setup".

M6 is the tool-swapping command – T2 tells it which tool to grab. It will first return the tool that's in the holder back to its' original position, then will pick up the new one at position 2.

Check for error messages before running this command– the light barrier might be broken, for example – the machine needs all error messages clear before it will tool change.

So now we have a smaller bit in the tool head.

Toggling between the axes, click the "Travel" icons to move the head near the desired zero point.

This is close enough.

Note: The vacuum pump needs to be on when you're setting your zero point; you don't want your material moving at all after the point is set!

Using the manual controller, hold down the "Slow" button in the bottom left, then use the X, Y and Z +/- keys to move the head finely.

If you let go of the "Slow" button, the machine will move in one-step increments; practically too small to see.

This is really close to the zero point, but it's not fully touching. I got to this point while holding the "Slow" button, so now I'm going to release the button and tap the Z- key a few times until the bit is just touching the surface.

Note: Some people put a very thin piece of paper down in-between the bit and the material during this step. Every time you hit the Z- key, you move the piece of paper a little. When the paper stops moving, you know the bit is close enough to get started.

Looks good.

Here you can see why a 20mm-thick bit would make this step less precise.

Once you're satisfied, hold the "Shift" key, then press X0, Y0, Z0 to set the zero point.

When you press Z0, the head will pop up 1 cm to give a little safety space.

Part 5: Start the Job

Let's run through a quick checklist.

- The file has been loaded and you've clicked "Save and Send to NC"
- The zero point has been set
- The correct tools are in the correct positions of the tool holder
- The vacuum is on and the material will not move
- You're wearing personal protective equipment (ear muffs, goggles, no loose sleeves or sandals) and you've told the people around you you're about to start the machine (they need to be wearing protective gear)
- The light barrier has been set

If all of the above are looking good and safe, go ahead and hit "Start" in the main menu of the HMI. Good luck!
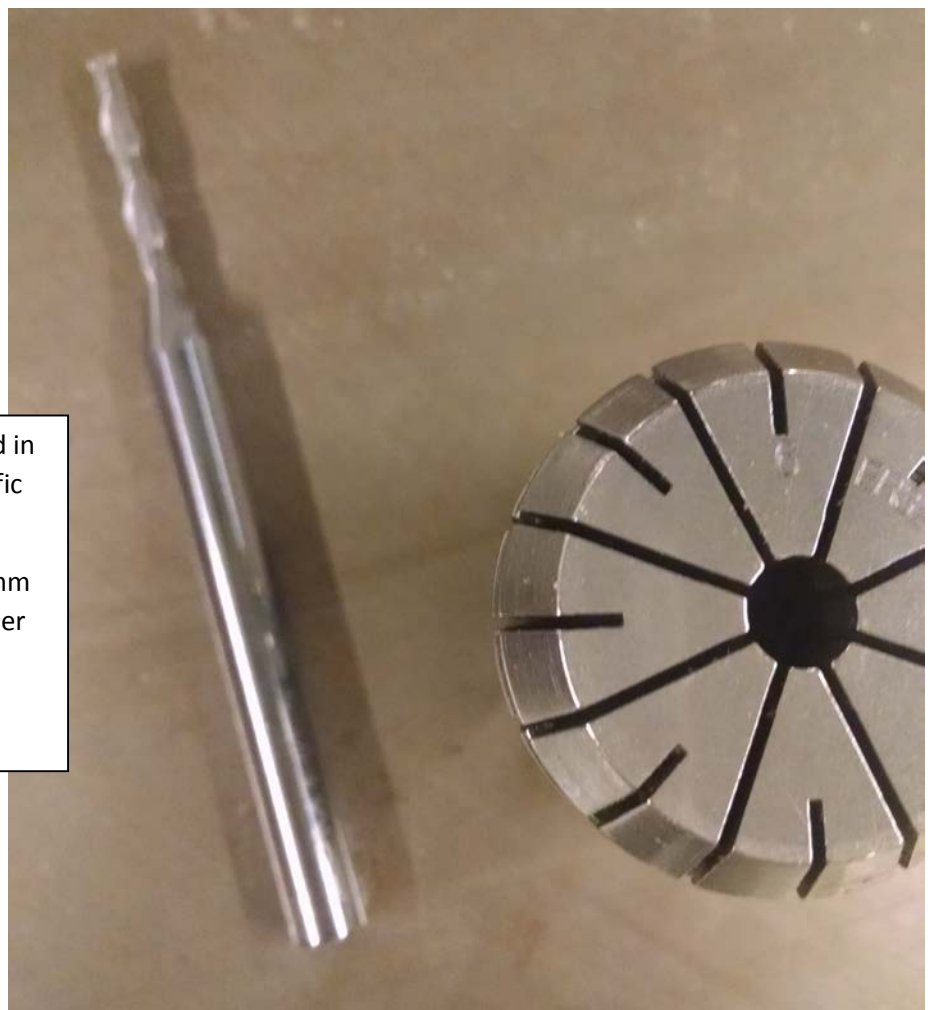
Part 4: More Tool Options

On the tool rack, there are assorted tool holders in the small jars with yellow and red lids.

Each one has a number engraved in the top of the metal and is specific to a certain thickness of bit.

The bit I need to load in has a 6mm thick shaft, and I've found a holder that's rated for 6mm.

Snap on a round chuck onto the top. This should fit over the top and pops in place…

…then screw the tool holder onto the top.

On the side of the CNC, there's a little spot to loosen and tighten the tool holders. If the pin is in the side, pull it out.

Fit the tool into the hole and rotate it until it fits in tightly and does not wiggle.

Once it's firmly in the hole, push the pin in the hole to lock it into place. The tool holder should now be completely pinned in.

Fit the black wrench over the tool holder and tighten it fully.