STEPCRAFT M700

USERS GUIDE

ENGLISH

v. 1.1

1.1

Make sure the machine has power. If not, check DK-plug to the wall.

When the machine is powered the lights 'power' and 'aktiv' lights up on the back of the machine.

1.2

Check if the spindle is holding any tools from previous operations. Release the tool by pressing the green button at the back end of the machine.

*Dont let the tool fall directly on to the bed of the machine, make sure you grab it before pressing the release button!

1.3

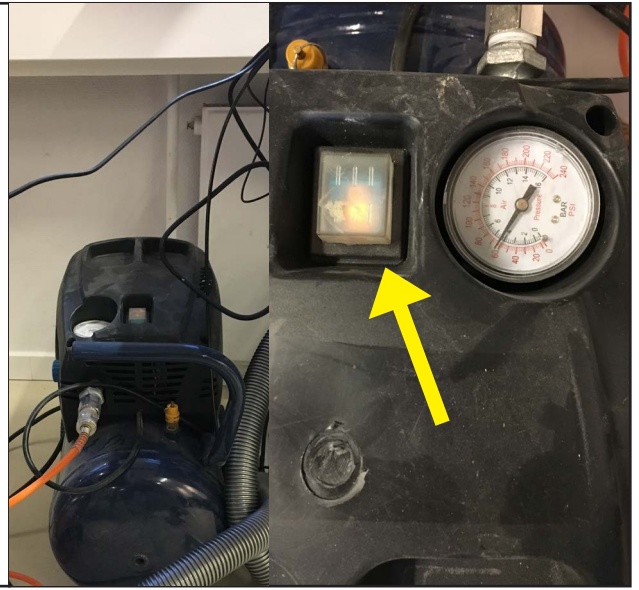Empty out the toolholder of any tools from previous operations.
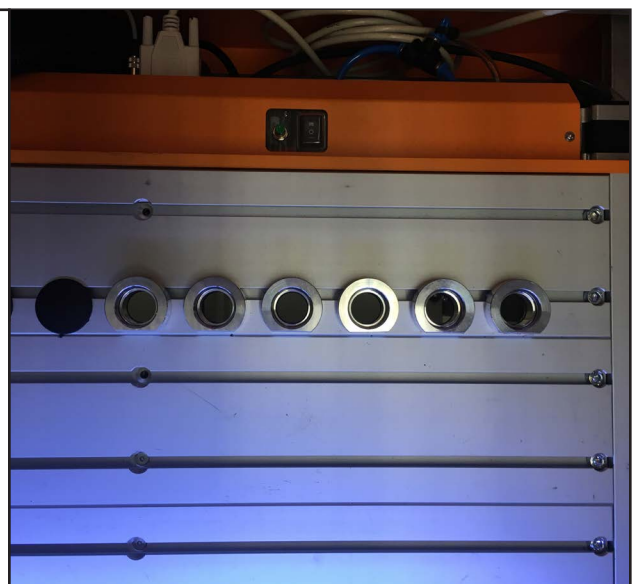
1.4

Check that the compressor is on and has compressed air in it.

If not check the powerplug, and the powerbutton next to the pressure indicator.

1.5

Make sure the airvalve located on the hose running from the compressor to the end of the CNC is partly open by at least 50%
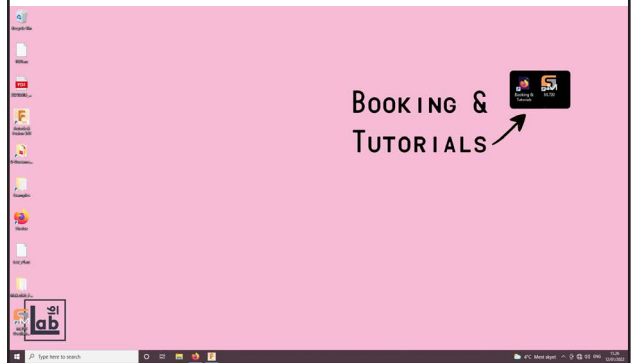
2.1

Open UCCNC M700 file on the desktop.

* Make sure to open 'M.700' program, and not the 'M.700 Oscillating Knife'

2.2

Place your desired tools accordingly to the manufacturing file you are running.

1-6 from the machines bottom, left to right.

Push the tools all the way to the toolneck hits the holder.

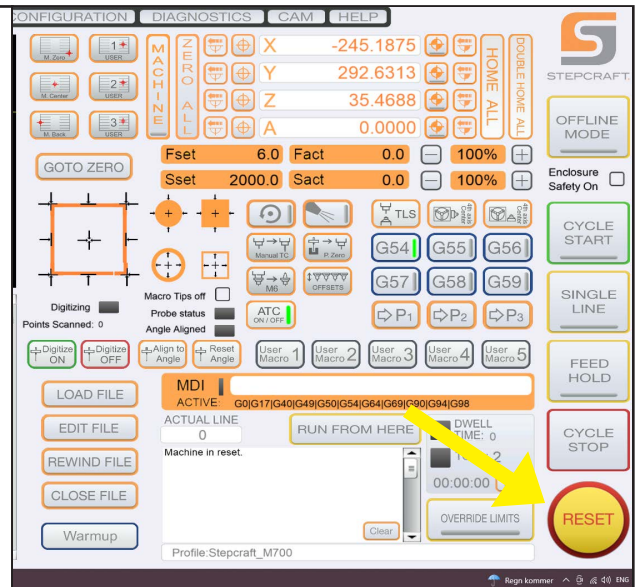## 2.3

Upon opening UCCNC the 'RESET' button in the user interface will be blinking red and yellow.

Press the 'RESET' button and it stops blinking and stays red.

If this doesn't work its probably because the physical emergency stop is pushed in. Release it and try again.

2.4

Note what tool UCCNC thinks is in the spindle. Even though you manually removed the tool, UCCNC will state its still holding said tool.

Manually change the tool to the right one. Find the corresponding tool in the tool holder, place the tool by pressing the green button on the CNC while pressing the tool firmly upwards in to the spindle until the tool settles securely in the spindle.
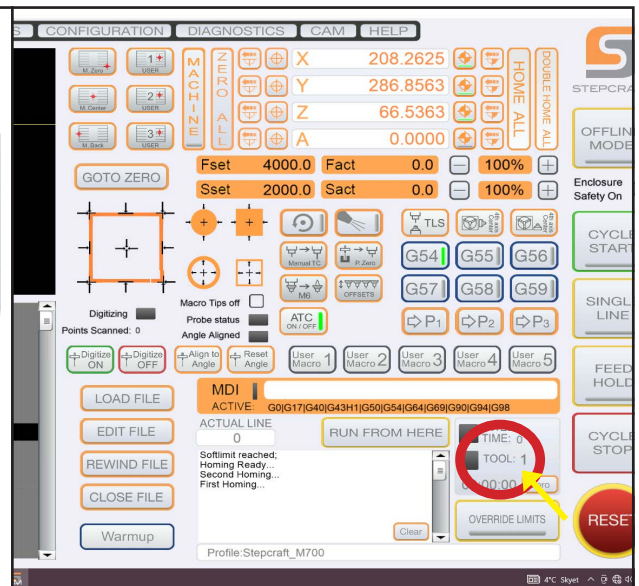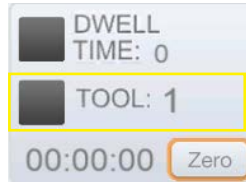
2.5

If UCCNC states that tool 1 is in the spindle, the machine should hold the corresponding tool like shown in the picture.

If UCCNC states its holding tool 2, it should hold the tool in tool slot 2, and so forth.
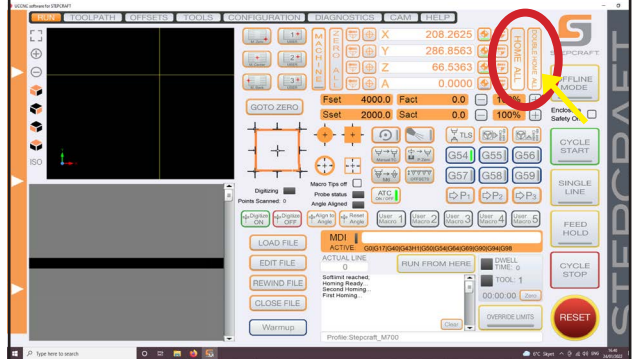

TOOL 1

Lab 16
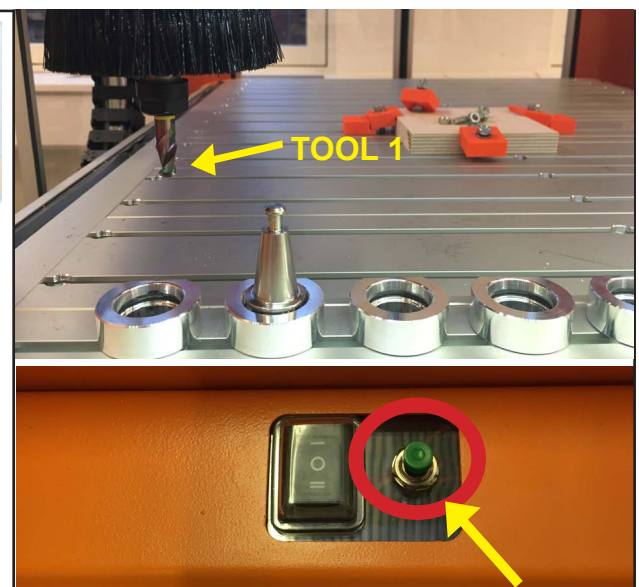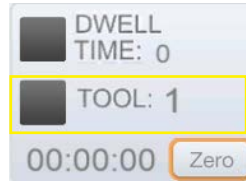
2.6

In UCCNC

Home all of the machines axes by pushing the 'home all' button. For highest possible precision press double home after the initial procedure, this will home the machine and re-home it with smaller increments in the steps.
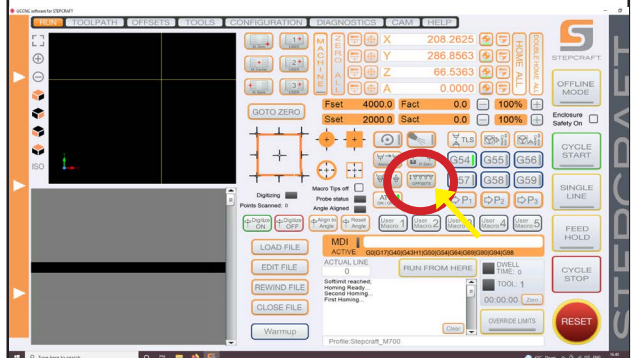

2.6

Make sure you have tool 1 in the spindle, if not type 'M6T1' in the MDI entry. This command instructs the machine to pick up tool 1 from the tool magazine.
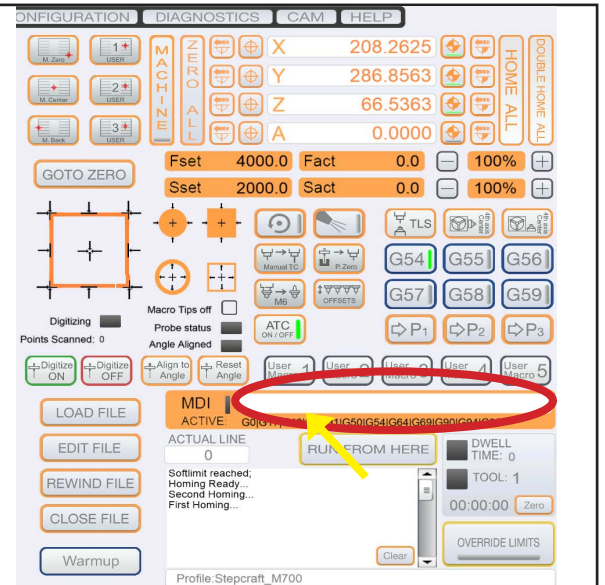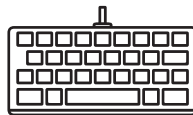
2.7

Press the 'OFFSETS' button in UCCNC the initiate tool measuring sequence.

2.8

The message shown in the picture to the right while pop up. Click 'OK'.

*Note that this will overwrite the current tool length offsets.

*Trough the whole of this procedure, take your time to read and understand the pop-up messages!

2.9

Enter the number of tools used in your current worksetup, from 1-6 depending on individual setup.

Then click 'OK'

This tells the software how many tools we want to measure.

2.10

Check if the typed in number of tools is corresponding with the actual number of tools in the tool magazine.

If this is true to the typed in number, press 'OK' - otherwise, cancel and start over.

The machine will now measure each tool individual tool on the tool length sensor.
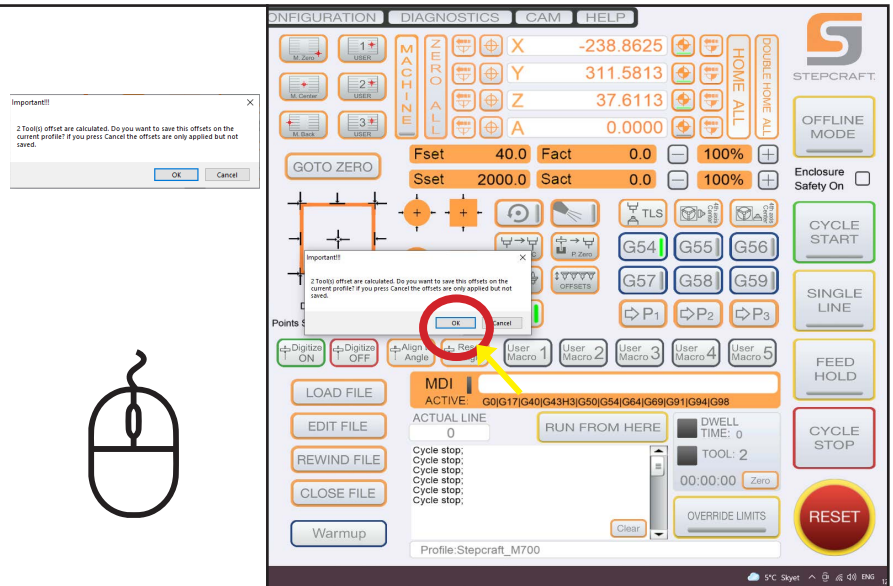
2.11

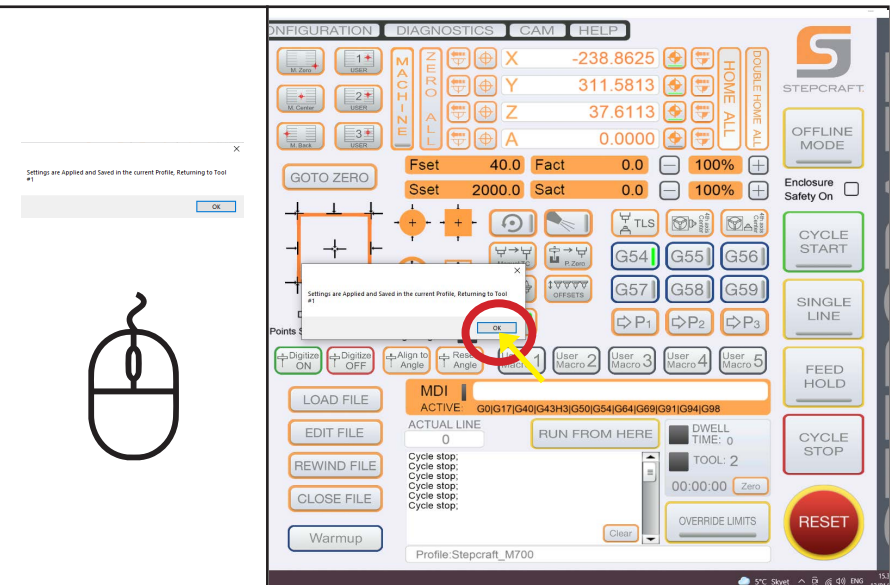After measuring all of the tools the message shown to the right will pop up.

Press 'OK'.

2.12

UCCNC will show the message shown in the picture to the right.

Press 'OK' - The CNC will return to the tool magazine and pick up tool 1.

You have now measured all the tools in the magazine.
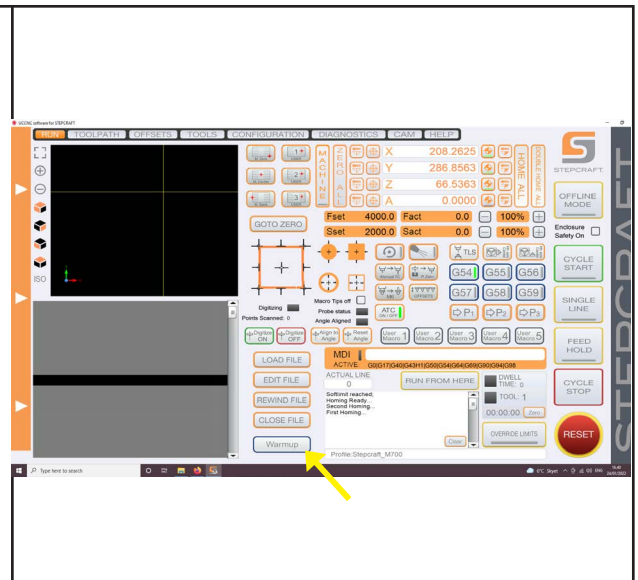

2.13

Make sure the the spindle can travel freely in the X and Y axis, then run the warmup sequence by pressing the 'Warmup' button in the bottom left corner of the UCCNC interface. If you made sure the X and Y axis are free, press ok on the two pop-up messages. This will initiate the warmup program, running the spindle along the X and Y axis a few times.

Warmup

3: Setting the 0-point

3.1

Setting the zero-point is key for a succesful operation.

Start by picking up a flat endmill, or a peak-point tool. These tools are the easiest to define the zero-point with.

Pick up the tool with the M6T-number of tool, pick up command. (Ex. M6T2 for tool 2)

3.2

Find your physical zero-point according on your stock to the zero-point defined in your CAD-file with the specified tool. Mark it with a pointed pencil

This often being either an upper corner or centerpoint of you stockmaterial. Make sure your defined axis/origin directions match those of the CNC.
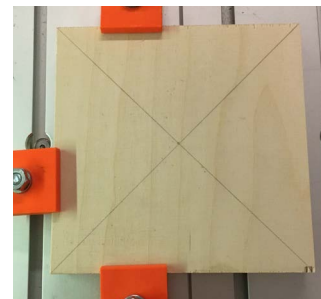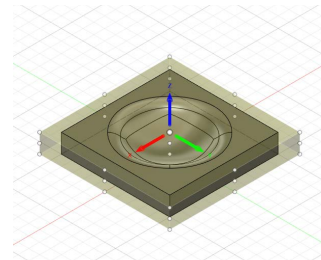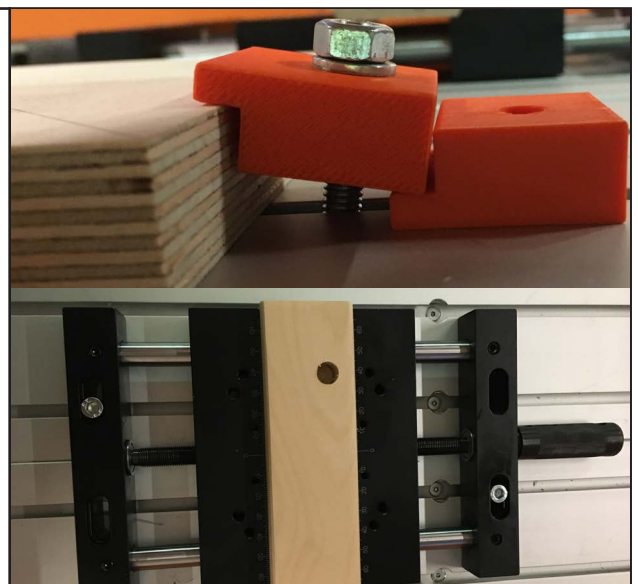

3.3

Make sure the workpiece is secured with clams, or secured in the vise.

Make sure the vise is also secured in the T-slot table.

*You shouldn't be able to move the stock/vise by pulling it by hand.

3.4 Use the 'JOG' function in UCCNC to pinpoint the physical zero-point and hover directly above it. Do it by clicking axis arrows with the mouse, or use arrow keys.

Start by jogging at a max. of Jog feed 30% to get within eyeball measurement of the point. Lower the Jog feed substantially (I.e single digits) when closing in on the zero-point

Jog feed %

− 30 +
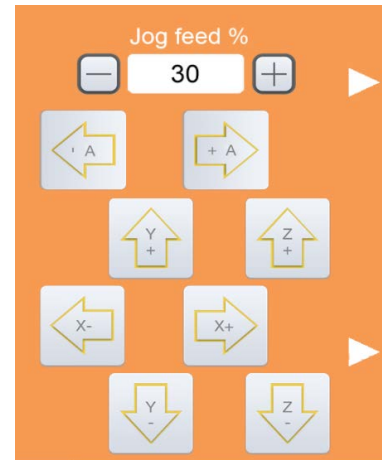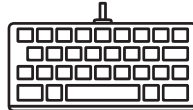
3.5

With a single digit Jog feed % move in on your point on all axes. The last axis you set is the Z-axis.

When X and Y are as close to your point as possible, place a piece of thin paper under your tool. Jog the Z axis at 1% Jog feed until the paper is clenched between the tool and your stock.
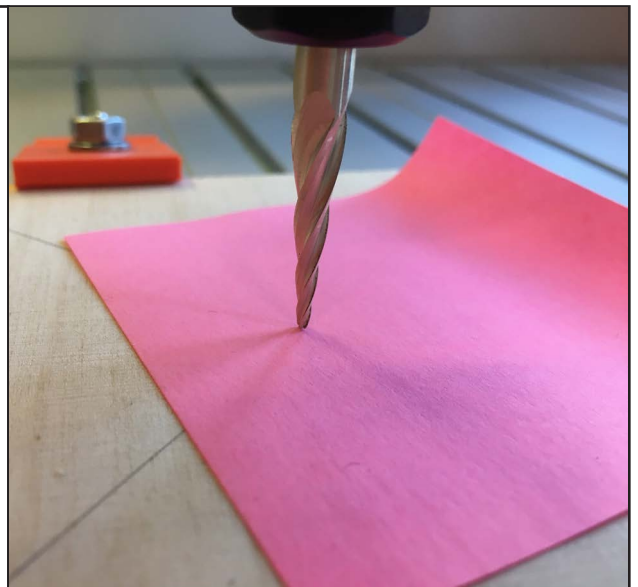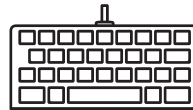
3.5.2

As an alternative to the method above, use your cameralight to shine light in between the zero-point and the tool while jogging towards it, until the light breaks.

Or use the camera to zoom close in on the point and identify when the two touch by sight.
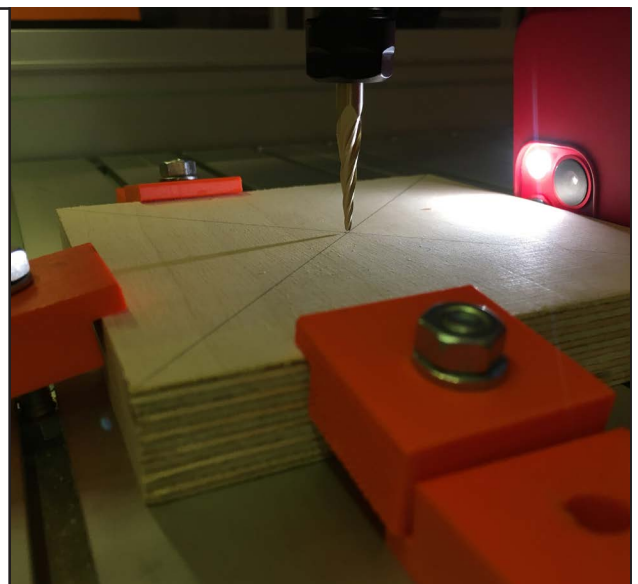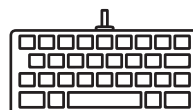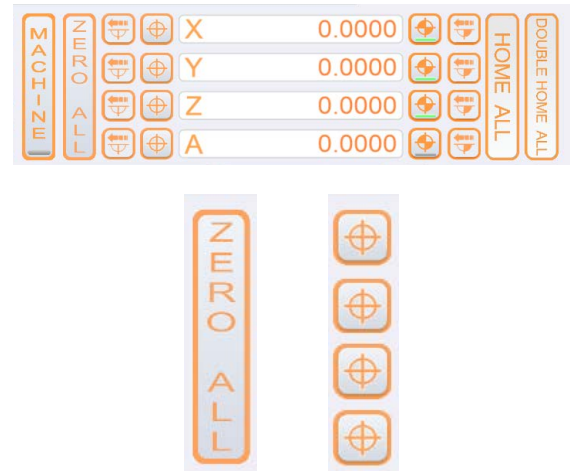
3.6

When all 3 axis are closed in on your zero-point go to UCCNC and press the 'ZERO ALL' button. This sets all 3 axis zero-points as you just determined them.
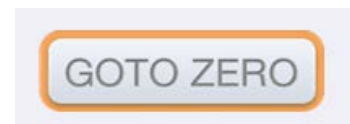
Its possible to set each axis zero-point one by one by clicking the 'target' icon according to each axis row in UCCNC

3.7

If all axis rows shows o.oooo your zero-point is now set.

Try jogging a bit away in all axis and press 'GOTO ZERO' - this should make your tool return to your exacts set zero-point.
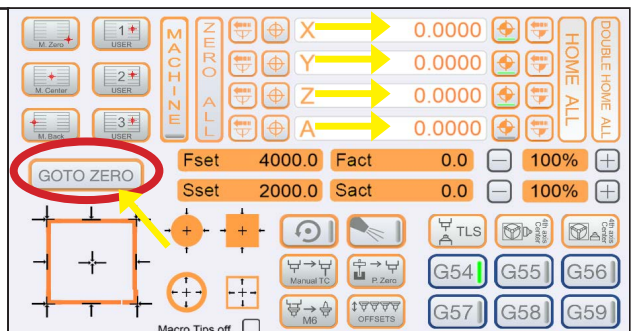
3.8

If the zero-point checks out when pressing 'GOTO ZERO' - you are all set to move along with your operation.

For good measure, Jog a bit away from your zero-point.

4: Loading your G-code to UCCNC

4.1

Press the 'LOAD FILE' to load your postprocessed G-code file.
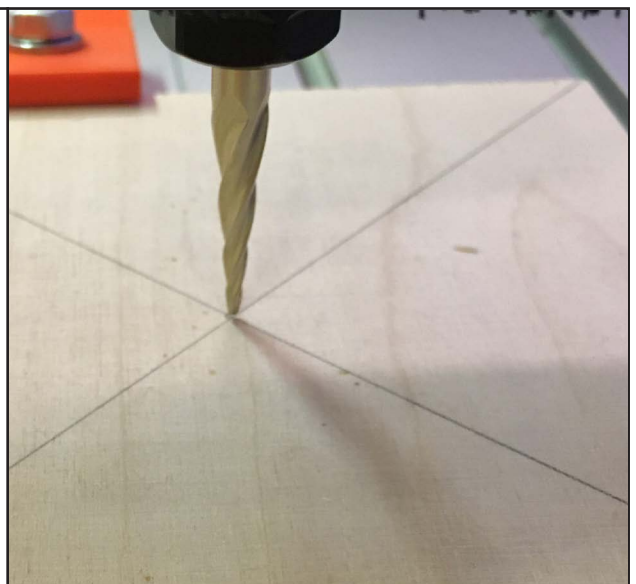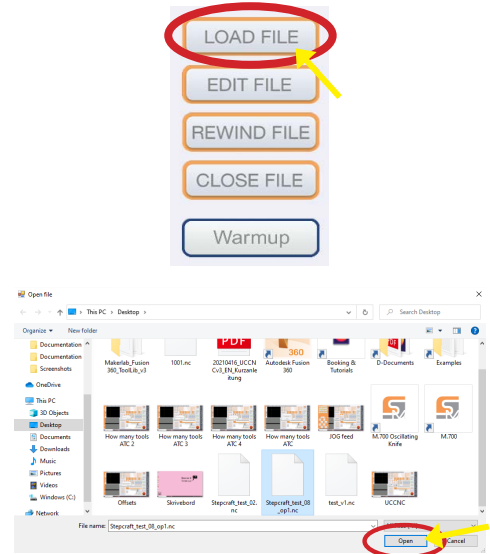
Choose your file and press 'Open'
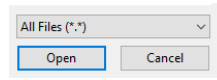
4.2

Your toolpaths and G-Code will now be shown in the left side of the screen.

Check the visual representation of your operation to see if somethings of.

Pan around the visuals of your toolpath by clicking and holding right mousebutton within the window.

4.3

You are now ready to run your operation.

Do a quick check on toolsetup, stockfixation and your zero-point.

Is it all good?
Press cycle start,

5.1.1

*Read all of the steps below!

Pausing the operation

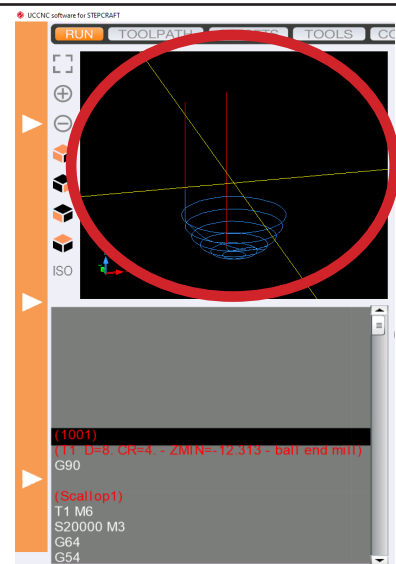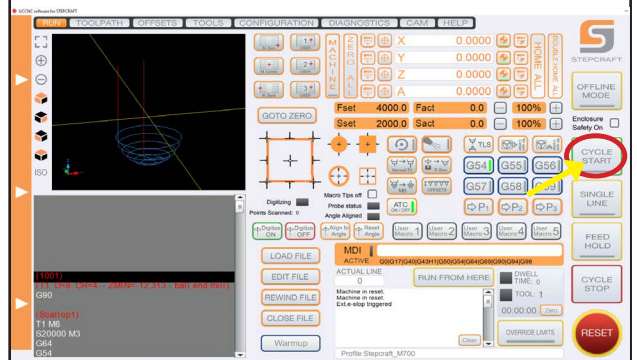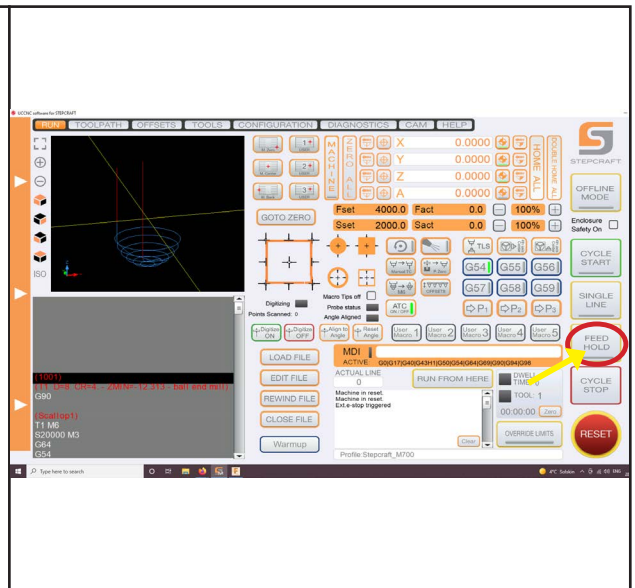To pause the operation you need to press two seperate buttons in the UCCNC interface.

1. Feedhold button

Holds the feedrate in your current operation. Note that the spindlespeed will remain unchanged!

5.1.2

Pausing the operation

Immediatly after pressing the feedhold button, press the 'spindlehold' button.

2. Spindlehold button

This will stop the spindle from running and make sure not to make friction burns on your part.
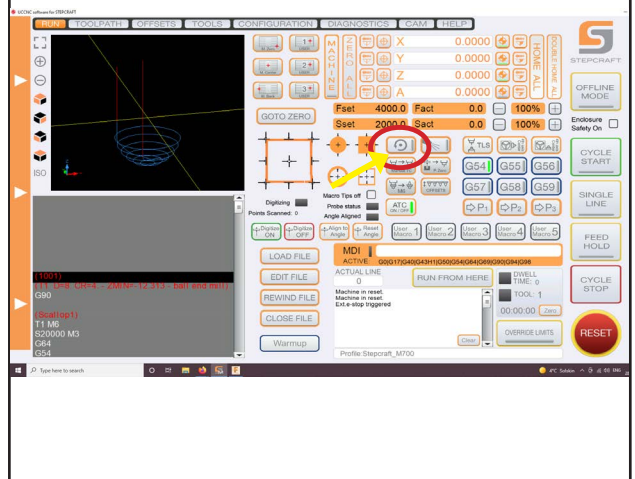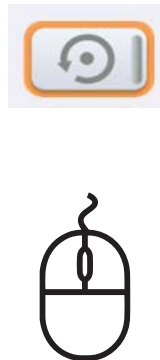
5.2.1

Overwriting feeds & speeds

Sometimes it can be benefitial to overwrite the set feeds and speeds of your operation in realtime. This can be to asses an operation when starting it out, or to optimize the chipload realtime based on experince of running the machine.

5.2.2

Overwriting feeds & speeds

The reader shown on the right shows your respectively set feeds and speeds, and the actual feeds and speeds while running your milling operation.

'Fset' and 'Sset' shows the predetermined feeds and speed, and the 'Fact' and 'Sact' shows the speeds you are actually running.
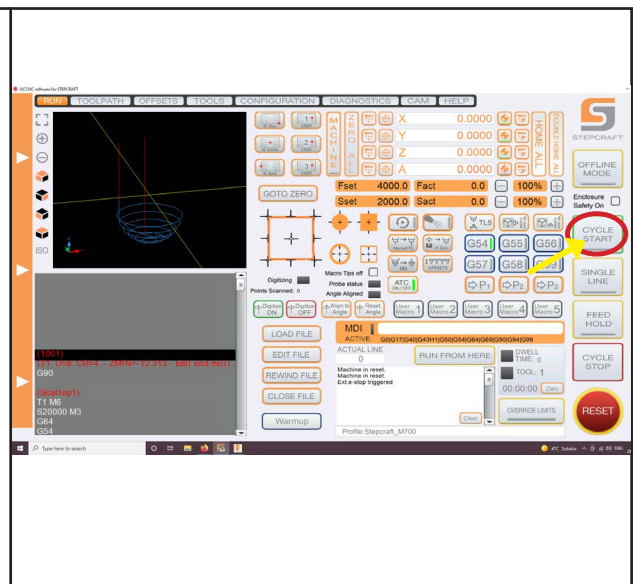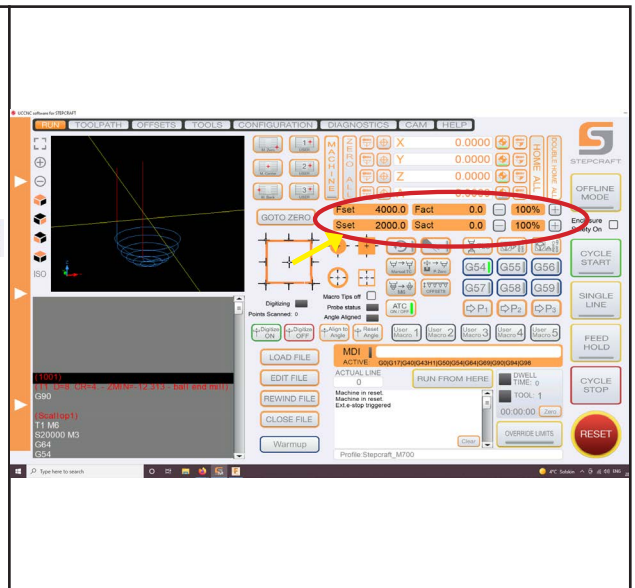
5.2.3

Overwriting feeds & speeds

By respectively pressing the plus and the minus on the far right of the feeds & speeds monitor you can manipulate the speeds by increments of 10% at a time.

By aware that these speeds are corrected by %, and thereby are relative to your predetermined feeds and speeds.
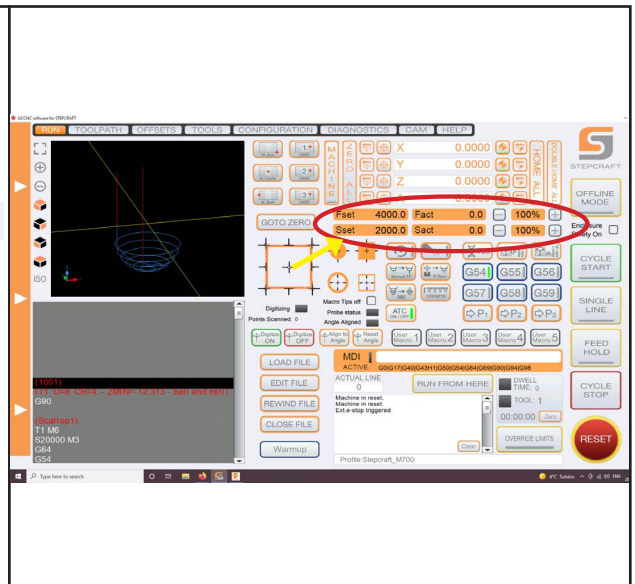
5.3.1

*Read all of the steps below!
 Restarting after 'cyclestop'

If you have to stop the operation by pressing cycle stop, or the emergency stop you will have to restart your operation, and let the milling operation run 'air milling' (running the toolpath without being in contact with the stock) until you reach the point where you stopped the operation.
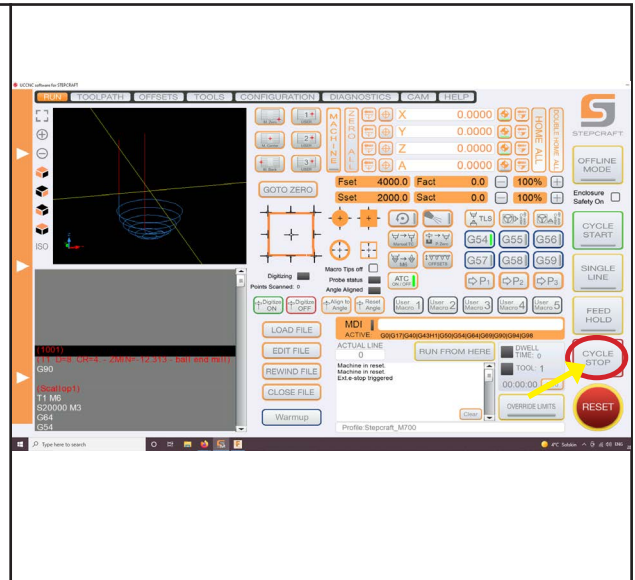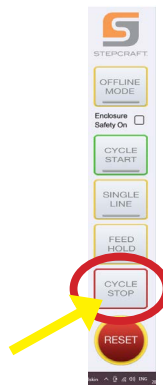
5.3.2

Restarting after 'cyclestop'

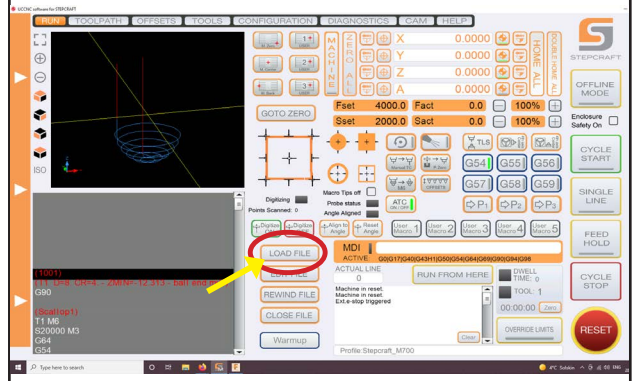To restart the operation after cycle- or emergency stop, press load file and run the file from the top.

To make sure your setup is still correct, follow the 'setting your zero-point' and ' loading your G-code to UCCNC'
steps 3.7 - 4.3
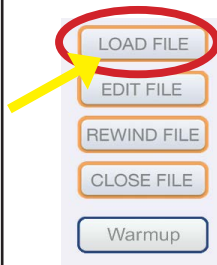
5.3.3

Restarting after 'cyclestop'

DO NOT press 'REWIND FILE' as this can cause stock collision when the tool returns to its G-code position.

When forced to press 'CYCLE STOP' or emergency stop, always reload your operation.
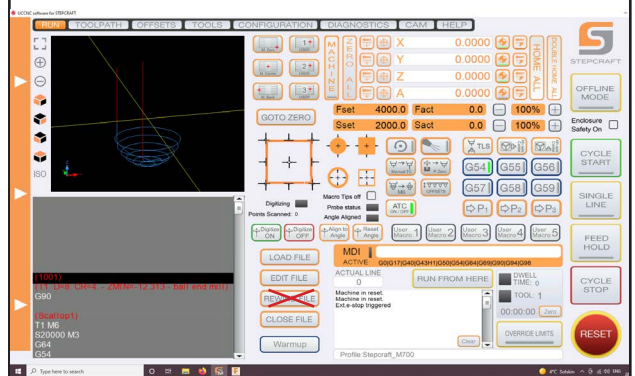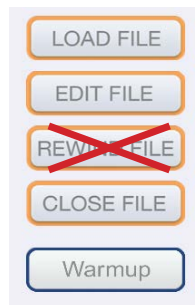
5.4
Removing dust mid-operation

The dust extraction is not optimal as of now, so when doing longer operations you can required to remove some of the excess dust from the CNC-bed and your stock. Follow the pause operation steps
(5.1.1 - 5.1.2) and remove dust by hand with the workshop vaccumcleaner.

6.1

Drive the spindel up and away from your stock and part using the Jogfeed menu.

Make sure you avoid collisions.

Jog feed %

⊟ 30 ⊞ ▶

' A   + A

Y +   Z +

X-   X+

Y -   Z -

▶
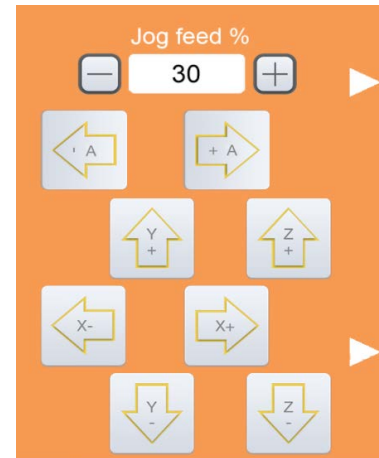
6.2

Remove all the tools including the one in the spindle, and put back at their right place in the toolholder.

Picture to come

5.4

Clean and vaccum the whole CNC and the surroundings with the shop vacuum cleaner.

Please leave the works area as you would expect to find it - ready to use for the next person

7.1

'Ext. e-stop triggered'

This error message occurs when the physical limit switch is triggered. This can be caused be three different issues.

The limit switch system consists of three different physical limit switches, designed to stop the machine from running beyond its X,Y and Z axis limit.

UCCNC software: Message

The reset logic of the software is active!
Remove the reset condition with pressing the blinking reset button on the screen.
If the reset condition cannot be removed can be due to an external e-stop button is pressed in or a limit switch on the machine tool is active.
Please check the Diagnostics page for the input signals condition.

OK

7.2

Limit switch issue #1

The emergency stop button is engaged.

Check if the emergency stop button is engaged, if so turn it right until it resets. Press the 'RESET' button in UCCNC to see if the issue is resolved.
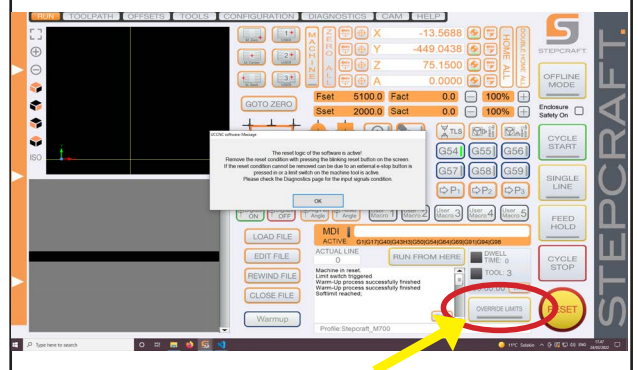
7.3

Limit switch issue #2

One or more axis are engaging with the physical limit switch.

Press the 'Override Limits' button next to the reset button, then toogle the opposite direction of the axis that triggers the limitswitch via the 'Jog feed' dropout menu.
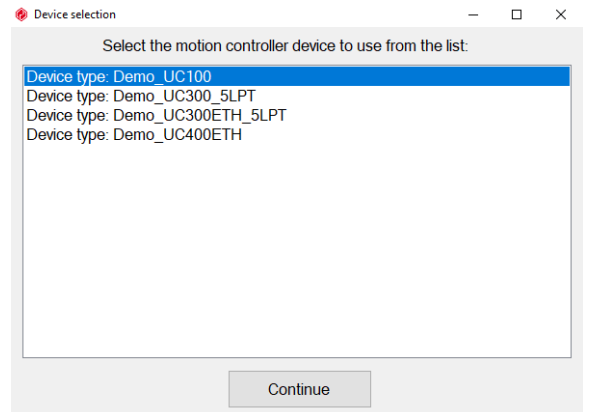
7: Error, problems and solutions

7.2.1

'Startup software issues'

Upon launching UCCNC rarely it will show the message displayed in the picture to the right.

This is caused by a loose connection in the USB-cable running from the CNC to the computer, keeping the software from comfirming the UCCNC license with the machine.
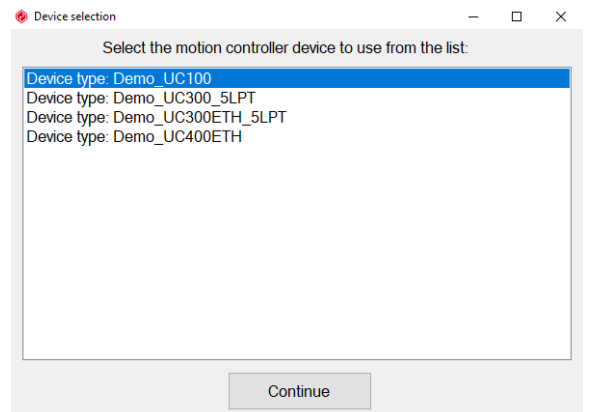
7.2.1

'Startup software issues'

When this issue occurs the software will only run trial versions.

Close down the UCCNC pop-up message and check the USB cable connection. Reopen the UCCNC 'M.700' program, if the issue is resolved UCCNC should open without showing the error message.

7.3.1
Other issues?

Upon encountering other issues than the one listed in this manual, contact working personal or screendump the error message and describe the issue and email it to:

SIBC@kea.dk