Section 2 – Familiarise yourself with UCCNC

4.0 ROBOTICS

IMAGINATION TO REALITY

CNCdrive
motion controls

BG Precision

Version 1.0

April 2022
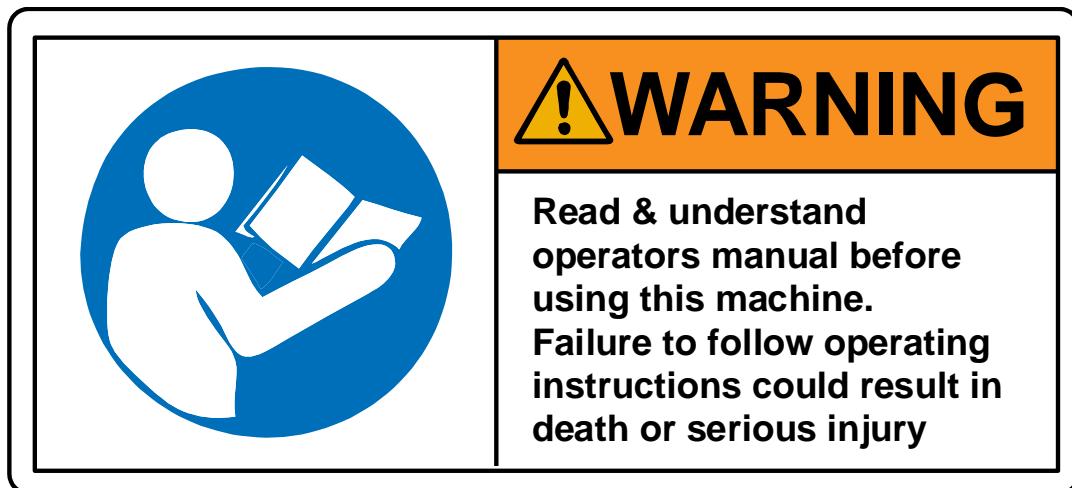
WARNING

Read & understand operators manual before using this machine. Failure to follow operating instructions could result in death or serious injury

Before using/turning on the machine, the device should be carefully checked to make sure all connections are secure and the device is technically sound.

bg precision

⚠️

Ensure You understand the safety considerations of a machine provided in the open configuration without a safety enclsoure

⚠️

Do NOT Interfere with the machine when under CNC control

⚠️

NEVER LEAVE THE MACHINE WORKING UNATTENDED

TABLE OF CONTENTS

1 INTRODUCTION

Thank you for purchasing your CNC system from BG Precision. This manual is a general introduction to the UCCNC Software provided with all BG Precision CNC Systems and is not specific to any one system in our range. Supplementary material will be provided specific to the machine make and model of which you have purchased.

Please ensure you read all the operational manuals for this CNC machine prior to attempting to use the system. Through-out this manual there are references to "A Trained Operator" or "Trained and Experienced personnel". These are defined as follows:

All persons that uses, or comes into contact with, the CNC router system MUST:

- understand what a CNC router is and can do
- read and understood the content of this user manual prior to using the system
- be able to exercise control of the router system at all times
- follow all the guidelines presented including the use of appropriate PPE
- seek further instruction if anything is unclear
- be sure that you have understood these instructions completely

Responsibility of use or misuse belongs to the end user. BG Precision PTY LTD and its affiliates accept no responsibility for use or misuse by the user. If you may not be able to use this product properly, we recommend that you do not begin use or cease use immediately.

This manual was not intended to cover every facet of machine operation. This manual serves to provide the information needed to safely operate and maintain the CNC router system. This manual has been designed to be used as an instruction tool as well as a reference tool for everyday work. Step by step instructions are provided where possible to help all levels of users understand the machine.

NOTE: Important aspects of machine use and best practice are highlighted and should be adopted where possible to maximise the machine tool life and performance. It is VERY IMPORTANT that all personnel read and understand the safety chapter BEFORE operating the machine. All Warning and Caution notices must be noted before interacting with the machine. Please refer to STEP 1 – Introduction to CNC for all safety considerations.

If there are any further questions or if anything is not clear, please contact us at info@bgprecision.com.au

2 UCCNC Overview

The UCCNC is a machine control software. It uses external hardware to generate signals to produce coordinated motion on up to 6 machine axis.

This software connects to the external motion controller via ethernet connection of a personal computer (PC) and via a software application interface (API) which is built into the software.

2.1 Hardware requirements

This software requires Microsoft Windows XP, 7 , 8, 8.1 or 10 Operating System running on an x86 or x64 desktop or laptop or tablet computer.

The minimal hardware requirements for the computer are as follows:

- CPU frequency: minimum 1.8 GHz (duo or dual core is recommended.)
- Graphics card: OpenGL 1.3 or higher compatible
- RAM: minimum 1GB for XP and 2GB for all other supported OS
- Hard drive space: minimum 16GB

The above are the recommended minimal values, however the software may run on PCs with lower resources, but we do not advice to do so. If large G-code files with the hundreds of thousands or millions of code lines count are run then the requirements may be higher. We suggest to always try the software even in demo mode on the planned to be used computer with the largest and most complex planned to be run g-code files to see the performance and to see if the computer fits the software requirements.

2.2 Software requirements

The software targets the .NET framework 4.0. The .NET framework 4.0 is Microsoft's runtime environment for applications (like the UCCNC) developed in Visual Studio. Starting from .NET 4.0 the frameworks are backwards compatible which means that installing a higher .NET framework version than 4.0 (for example .NET 4.5) will also run the UCCNC software. Window may prompt you to download the .NET Framework.

2.3 Licensing

The software requires a license key to fully function. One license key is valid and can be used with one device only. The motion controller device's serial number and the license key's serial number must match, the license key with one serial number and the motion controller device with a different serial number will not work together. You can have multiple license keys on the one control PC but can only control one device at a time.

3 UCCNC INSTALLATION

This section can be completed independent and prior to connecting to the CNC machine.

The installer files (Figure 1) are all on your USB key or the link sent to you via email. Do not rush the installation. Open the USB Key provided and open the "*UCCNC Set up Folder*" contained within.
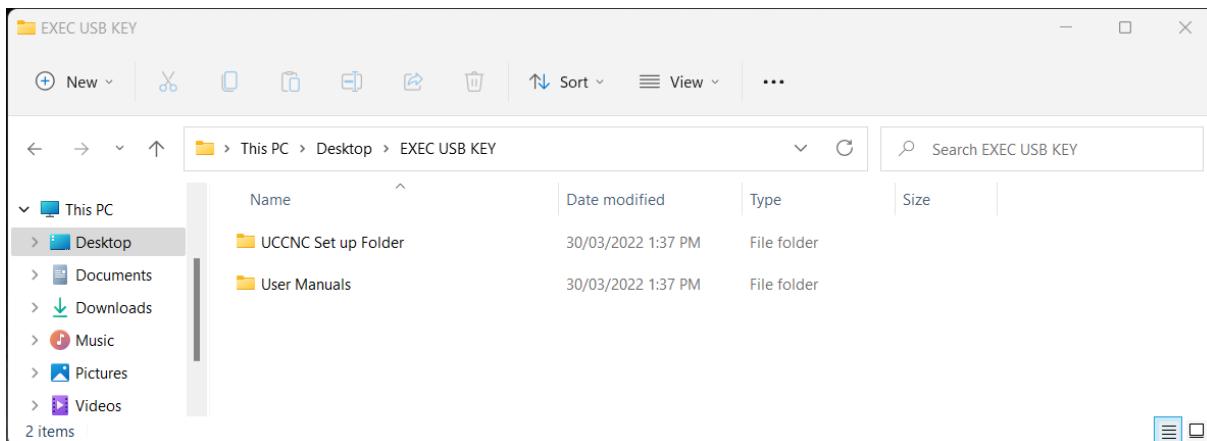

Figure 1: UCCNC Set-up files

The below instructions are written for Windows 10/11 of which can also be found in your installation folder under Step 1 as per Figure 2. If you have Window 7 you will find supplementary instructions in this folder also to guide you through.
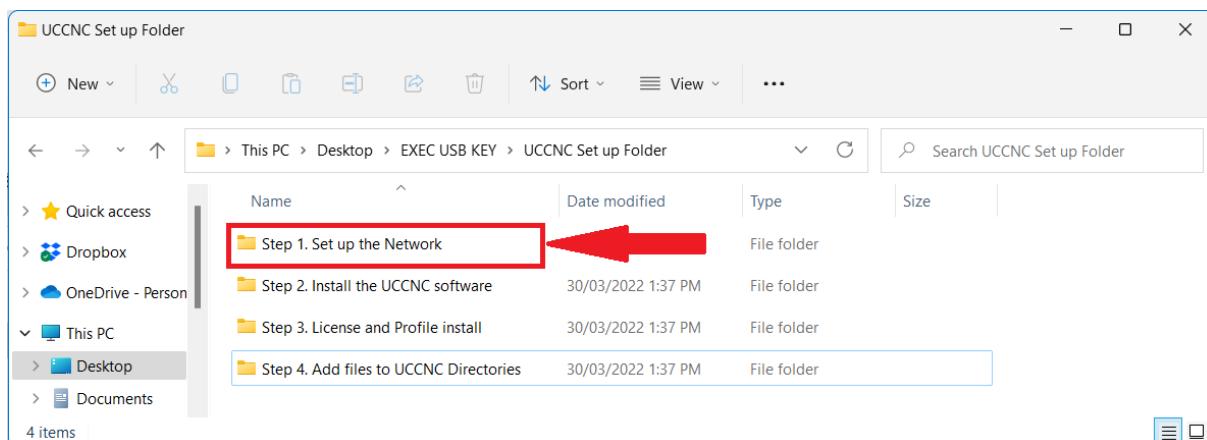

Figure 2: UCCNC Set-up, Step 1

Your UCCNC control module in the CNC controller is an Ethernet controller. Ethernet control is the most robust control for CNC applications. The controller requires a fixed IP address to be set on the local PC connected to the machine. The following instructions is based on direct connection to a LAN network card (Local Area Network) using a RJ45 Ethernet cable. For the CNC Ethernet controller, no USB drivers are needed.

We highly recommend you DO NOT use a USB to Ethernet Dongle/Adapter as USB interface is not as fast and stable as Ethernet for CNC control.

3.1 STEP 1 SET UP THE NETWORK

1- In Windows right click the Start menu (windows icon) and click on the search icon. See Figure 3
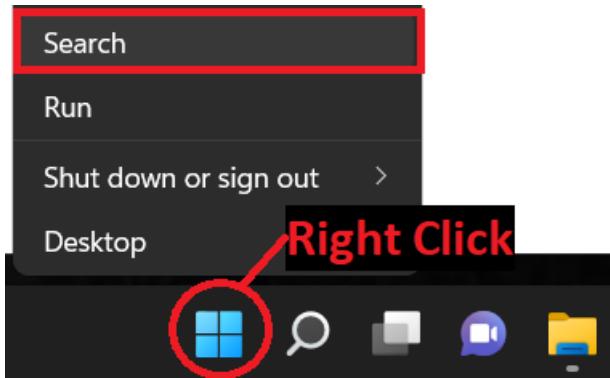
Figure 3: Windows search console

2- Type in the search "network connections" and either select when it is displayed, or press enter. See Figure 4.
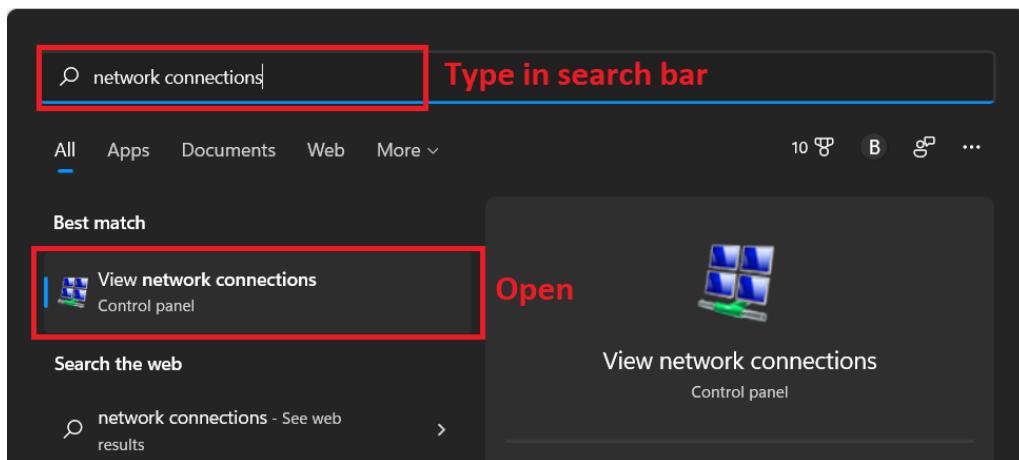

Figure 4: View Network Connections

This will open a Network connections window. By default, the network will have a name local connection or Ethernet or similar, please identify it. We are going to rename the connection by right clicking on the "Local Area Connection" and selecting "Rename". See Figure 5.
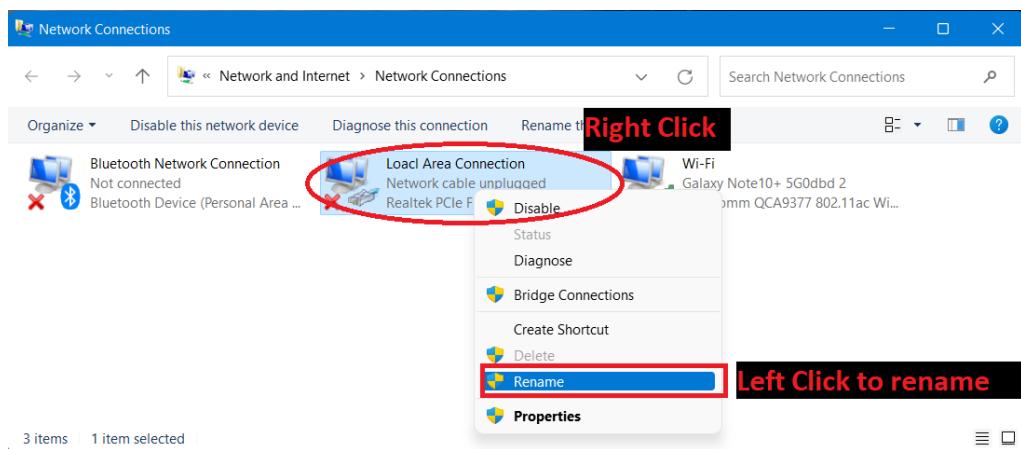
Figure 5: Rename this connection

This is just a name to allow you or any PC support personal to see the connection has been renamed and edited from default settings. You can call it anything to easily identify it like "CNC" or "EXEC" or "CNC Machine". The name does not affect the function of the machine. See Figure 6

3- After finding and optionally renaming the connection it is time to setup the network parameters. To setup the connection with direct cable connection to your CNC machine, right click on the renamed connection and on the pop-up window left click the Properties button. Please note that for this action to work the user must have administrator rights in the Windows account. See Figure 6.
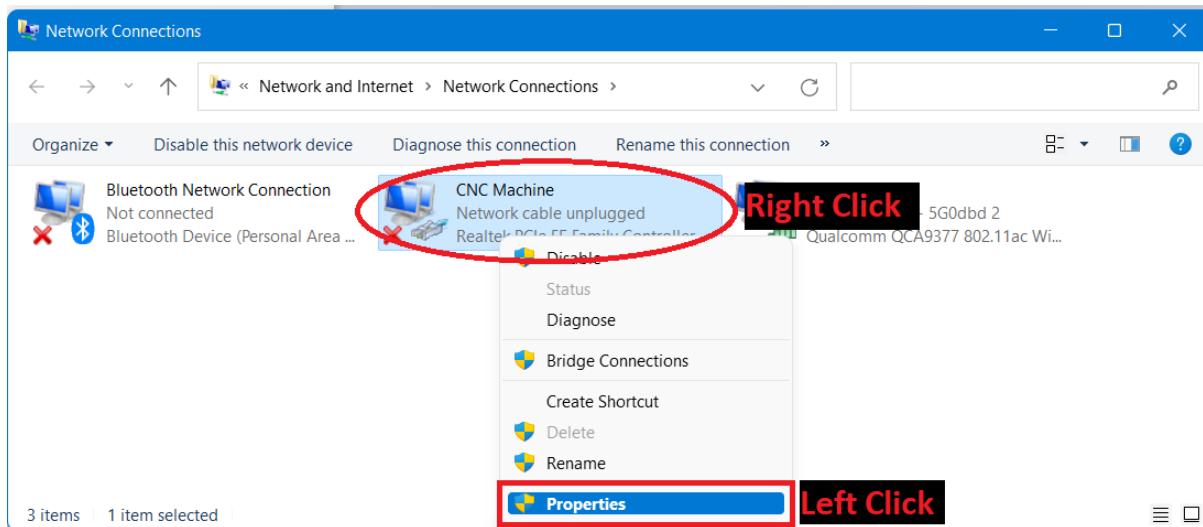

Figure 6: Network Properties

4- On the popup window find and select the 'Internet Protocol Version 4(TCP/IPv4)' row and click the Properties button. See Figure 7.
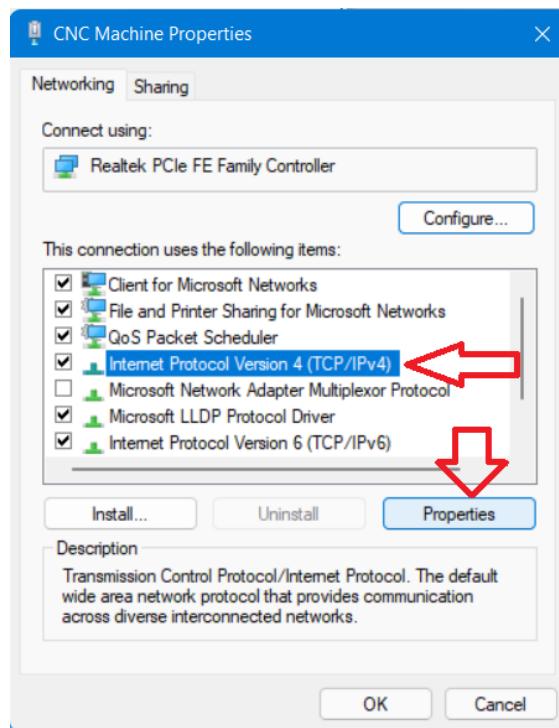
Figure 7: IPv4 Properties

5- On the pop up Internet Protocol Version 4((TCP/IPv4) properties window select the "Use the following IP address" option and fill in the IP address of the CNC Controller and set the Subnet mask as shown. The default IP address of the is 10.10.10.10 (this can be changed, refer to the CNC Controller manual for more details) and set the subnet mask to 255.255.255.0 value. Figure 8 shows the IPV4 network IP address that needs to be set.
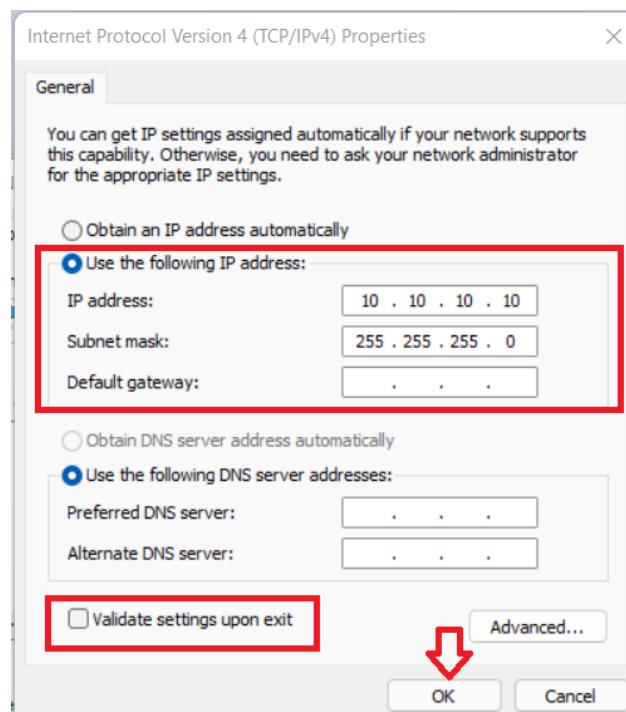

Figure 8: Network settings that need to be set prior to running the machine

After filling the values, you do not need to check the 'Validate settings upon exit' checkbox. Finally press OK to all the pop-up windows to exit the setup. Wait a few seconds to let Windows to update the settings in the LAN card.

6- Upon clicking OK and closing the properties menu, Windows may run a trouble shooter and look for any issues. If the settings were entered correctly, it may still tell you "Troubleshooting couldn't identify the problem" as we may not be physically connected to the machine yet so do not be concerned. For now just close this message.
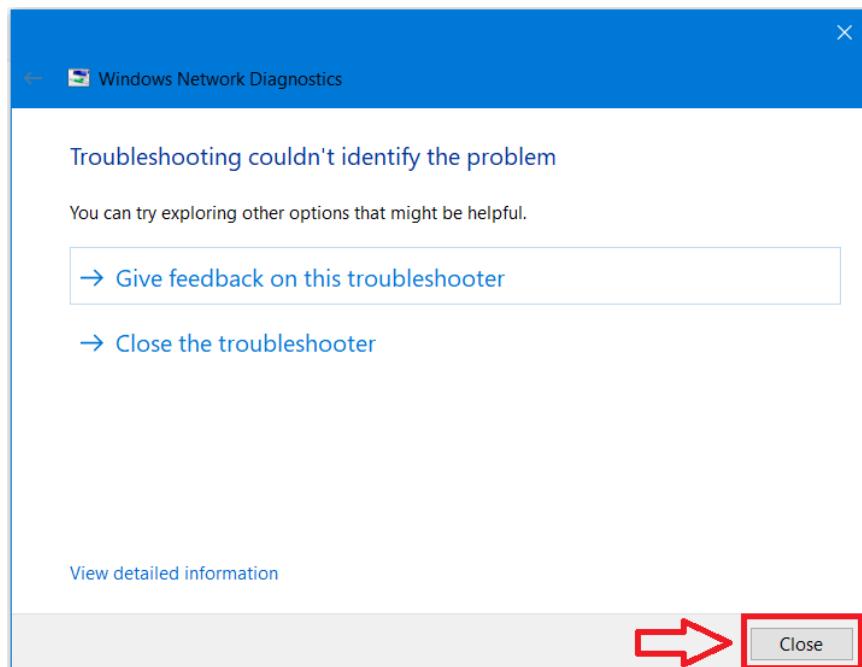

Figure 9: Window Network Diagnostic

3.2 STEP 2 INSTALL THE UCCNC SOFTWARE

Once you have the module successfully connecting to the control computer you can now install the UCCNC software.

Open up STEP 2 from the installation folder and double click the installer Please refer to the _Installing UCCNC Software_ PDF when installing. Depending on the PC certain steps during the installation (like USB drivers) can take some time.

1. Enter the installation folder
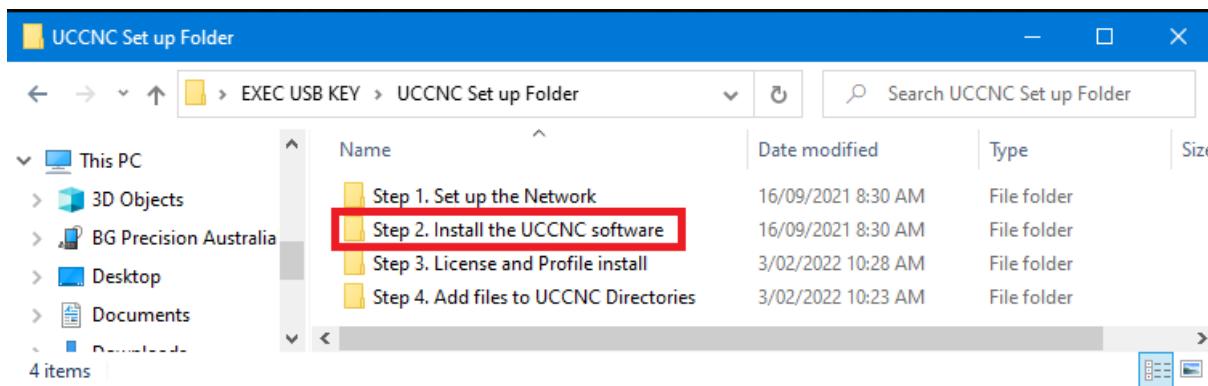
Figure 10: UCCNC Set-up, Step 2

2. Run the installer (note the version in this manual may differ from the version in your installer)
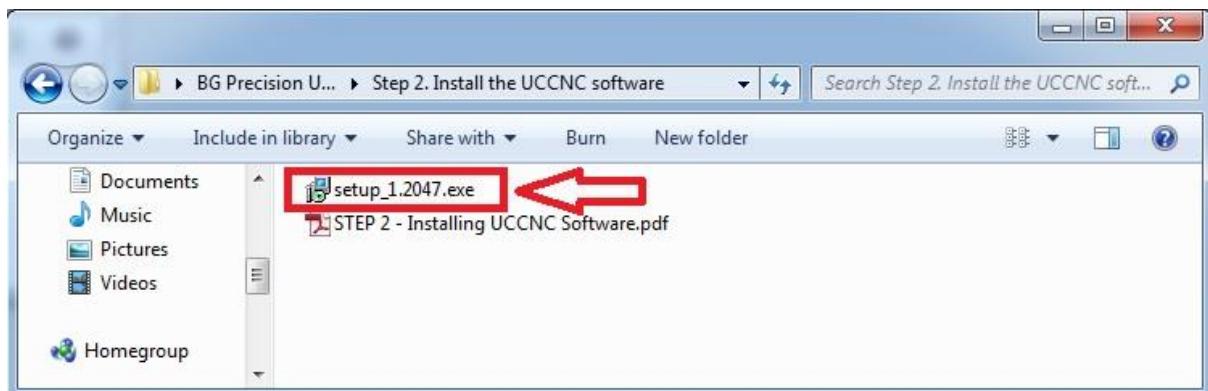
Figure 11: UCCNC Set-up exe

NOTE!!!!! The VERSION OF THE SOFTWARE MIGHT BE DIFFERENT IN YOUR INSTALLER

The latest version of the installer is always available provided where applicable

3.3 STEP 3 UCCNC LICENSE AND PROFILE INSTALLER

The next step is to install the license for your UCCNC controller and your specific machine parameters called the machine profile for the CNC System you have purchased.

1. Double click on Step 3. To open the profile and license installer
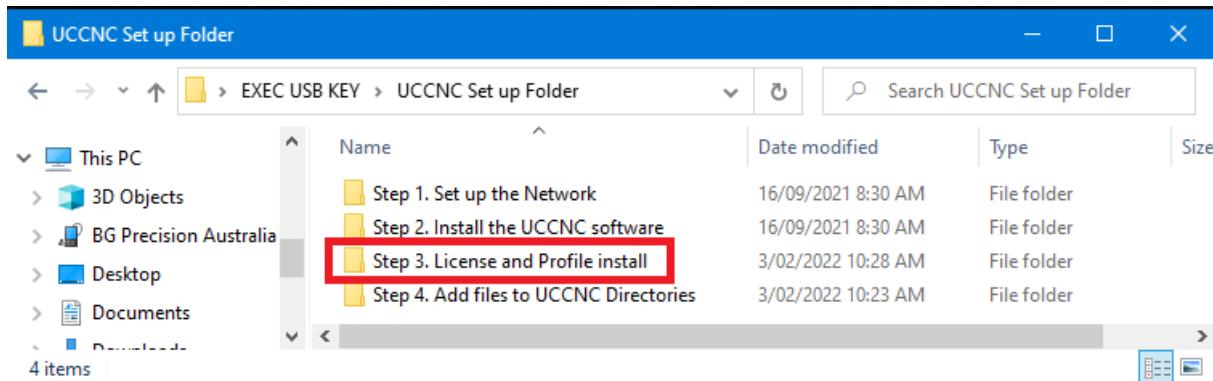

Figure 12: UCCNC Set-up, Step 3

2. Click on the profile folder you wish to install for your machine. You may have more than one applicable to your purchase.
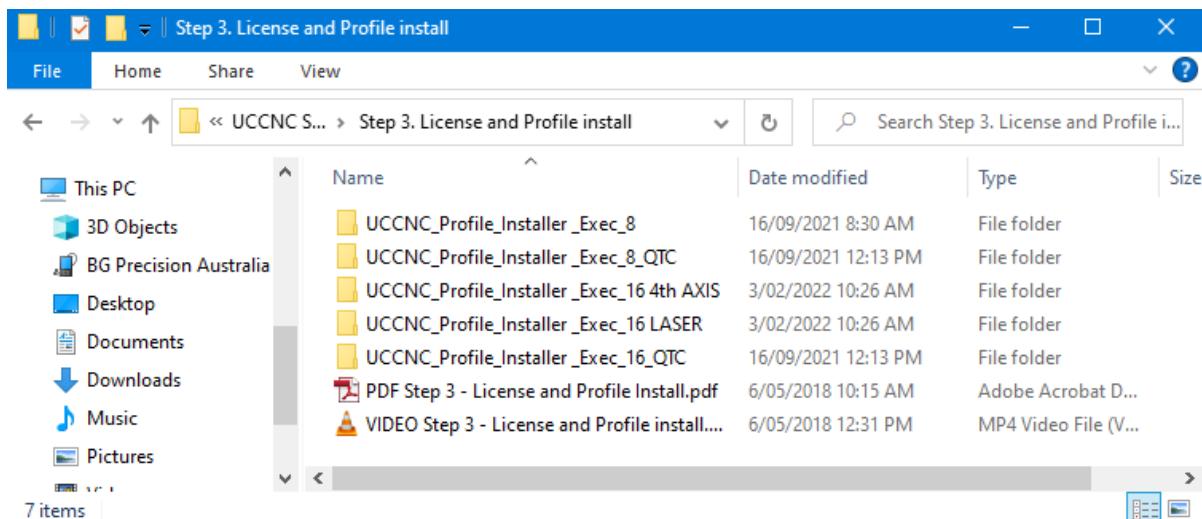

Figure 13: UCCNC profiles for selection

3. Once opened the profile installer folder, click the UCCNC_profile installer run me.
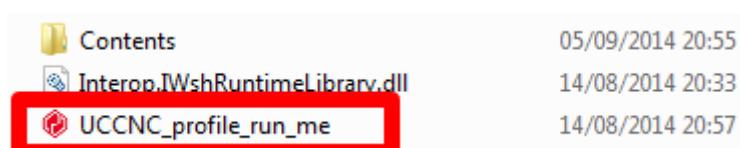
Figure 14: UCCNC Profile Run Me

When complete a pop up window will display the following:

Figure 15: Installation of the profile finished!

4. There should now be a desktop shortcut for your CNC machine. All of the settings are now loaded and your machine is ready for full operation.

- NOTE!!!! When UCCNC installs for the first time there is a default shortcut always loaded to the desktop.

- VERY IMPORTANT you either delete the generic shortcut or understand that it won't be the correct profile.

- The name of your machine profile such correspond with the machine you purchased and may differ from what you see below.
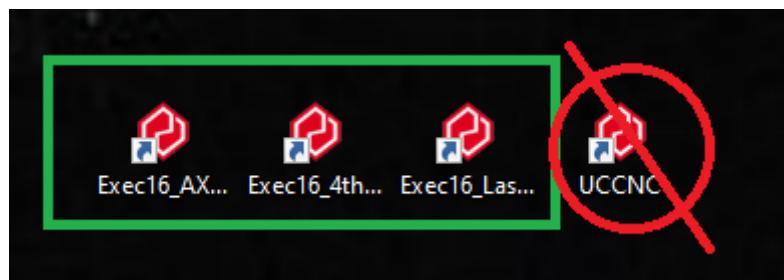
Figure 16: Desktop Shortcuts

3.4 STEP 4 ADD FILES TO UCCNC DIRECTORIES

This step is not applicable to all CNC machine purchases. If step 4 is not on your USB key, please skip to the next section of this document.
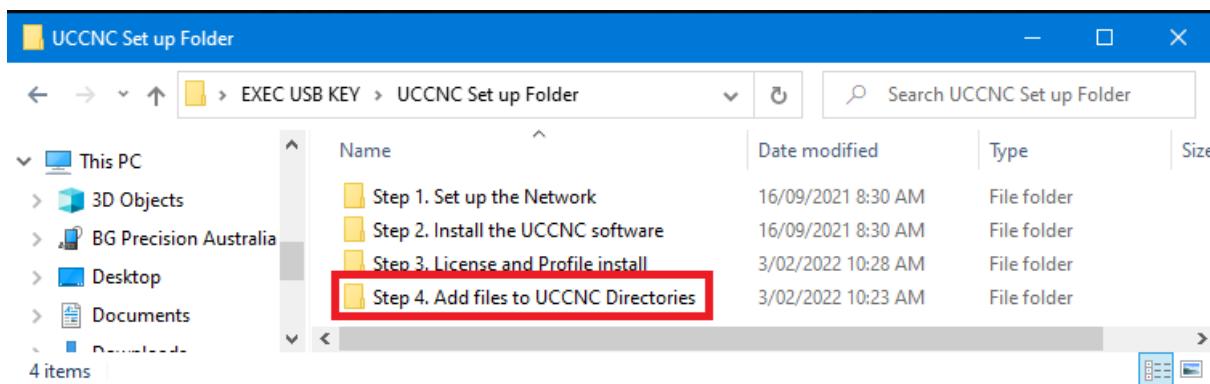
Figure 17: UCCNC Set-up, Step 4

Open the Step 4 folder and you will note an Images and Screen subfolder
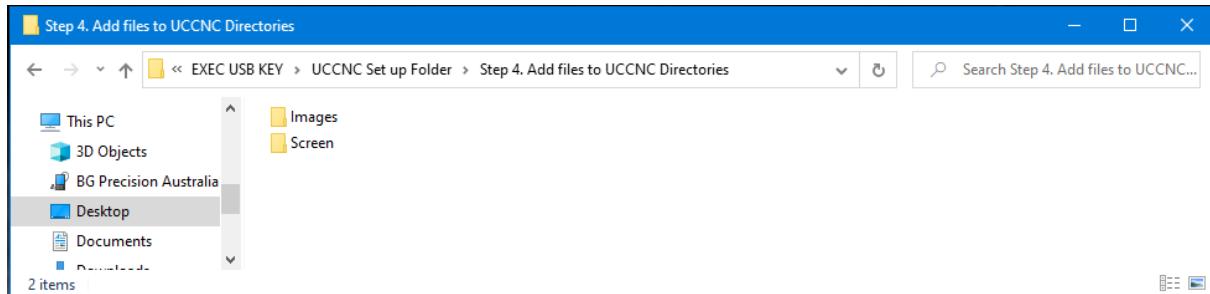
Figure 18: Add images and screen files

Lets start with the "Images".

- Open the folder to view the contents
- Select all (Ctrl + A) and copy the files (Ctrl + C)
- Navigate to "C:\UCCNC\Flashscreen\BMP\Defaultscreenset"
- Right click and click paste (or Ctrl + P)
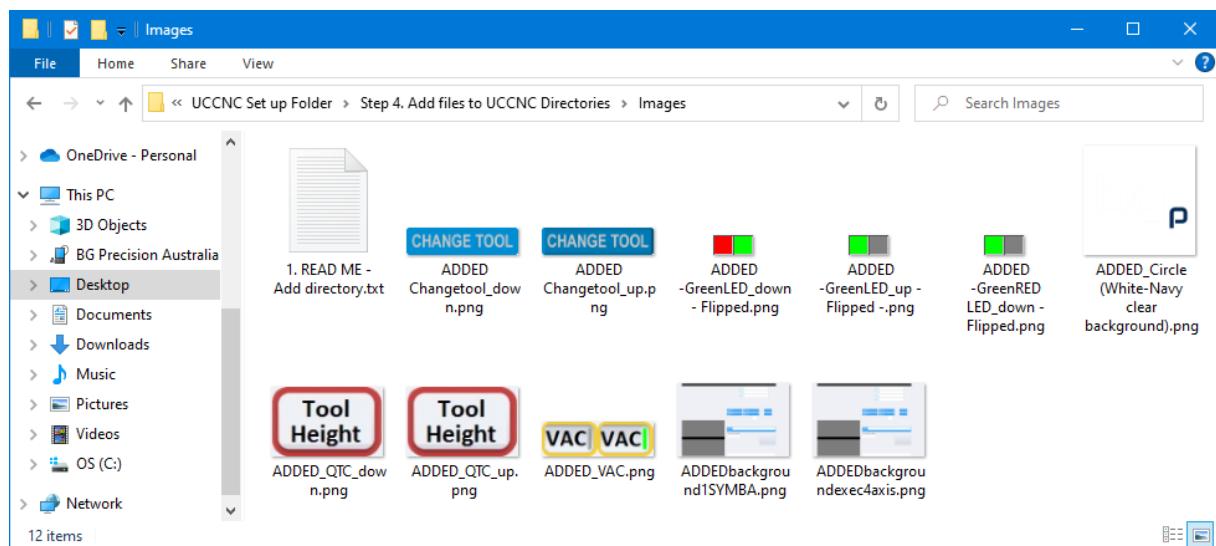
Figure 19: Images Files

Repeat for "Screens".

- Open the folder to view the contents
- Select all (Ctrl + A) and copy the files (Ctrl + C)
- Navigate to "C:\UCCNC\Screens
- Right click and click paste (or Ctrl + P)
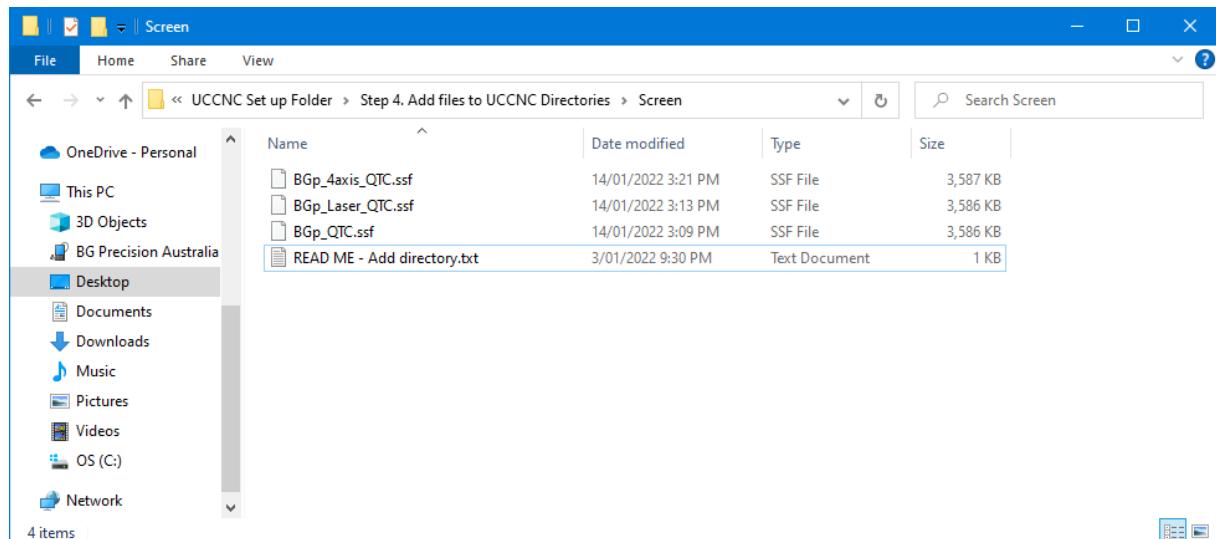
Figure 20:Screen Files

4 BASIC MACHINE CONTROL WITH UCCNC

- Make sure your CNC machine is powered on

- Make sure your Ethernet connection on front of control box is connected to the PC

- Make sure the emergency stop button on the front of the machine is released

- Make sure you have carried out all "setup operations from Step 1. Step 2. And Step 3 _Exactly._

- Double click on the icon on your desktop that the profile installer created in Step 3. This should be "machine Model here" on the desktop (as shown in Figure 21)
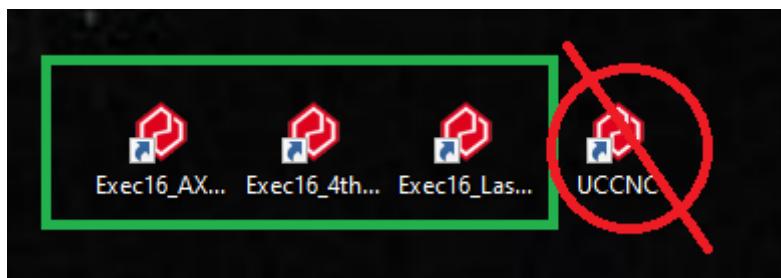
Figure 21: The UCCNC shortcut you should now have on the desktop

- The software should open and go straight to splash screen shown in Figure 22

Figure 22: UCCNC splash screen

- Note if you have installed the license file correctly you should not see any window pop up regarding the license when you open the program. Before you try to open the finally installed and validated UCCNC – make sure there are currently no open UCCNC windows.   When you have checked that there are currently no open UCCNC windows then attempt to open the UCCNC to test the installation has been successful.

Your UCCNC Profile for the CNC machine may come with a **custom UCCNC screen set**. Below describes some of the **Default screen tabs** and functions which may differ with slightly with custom screen files.
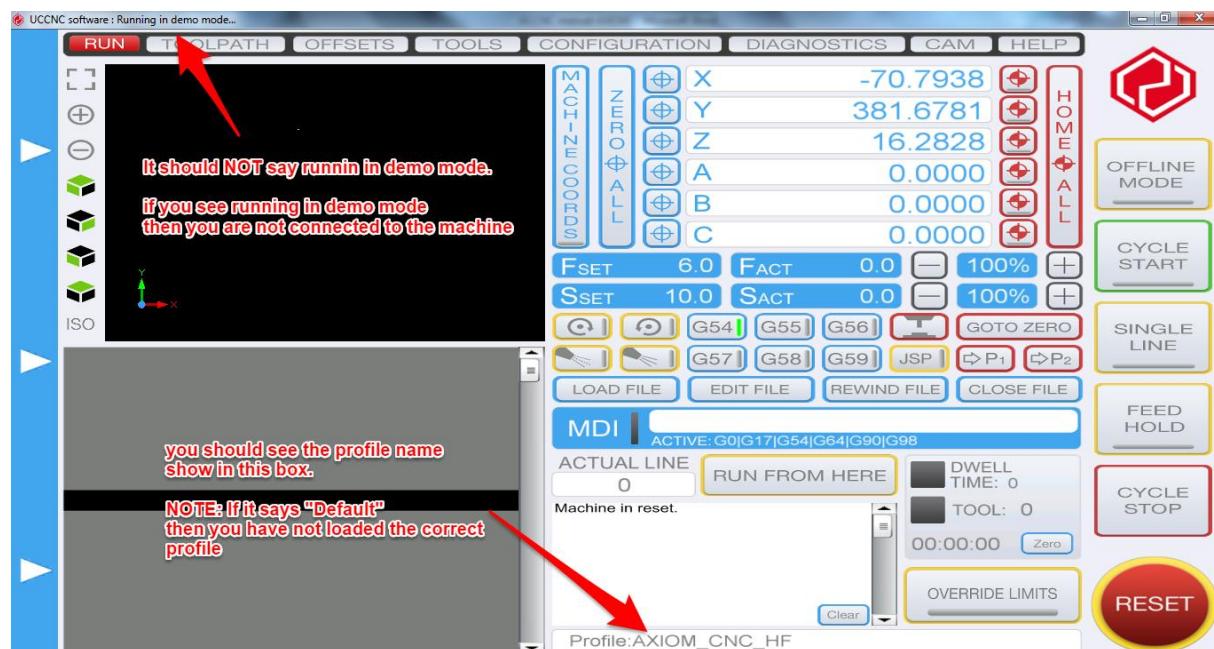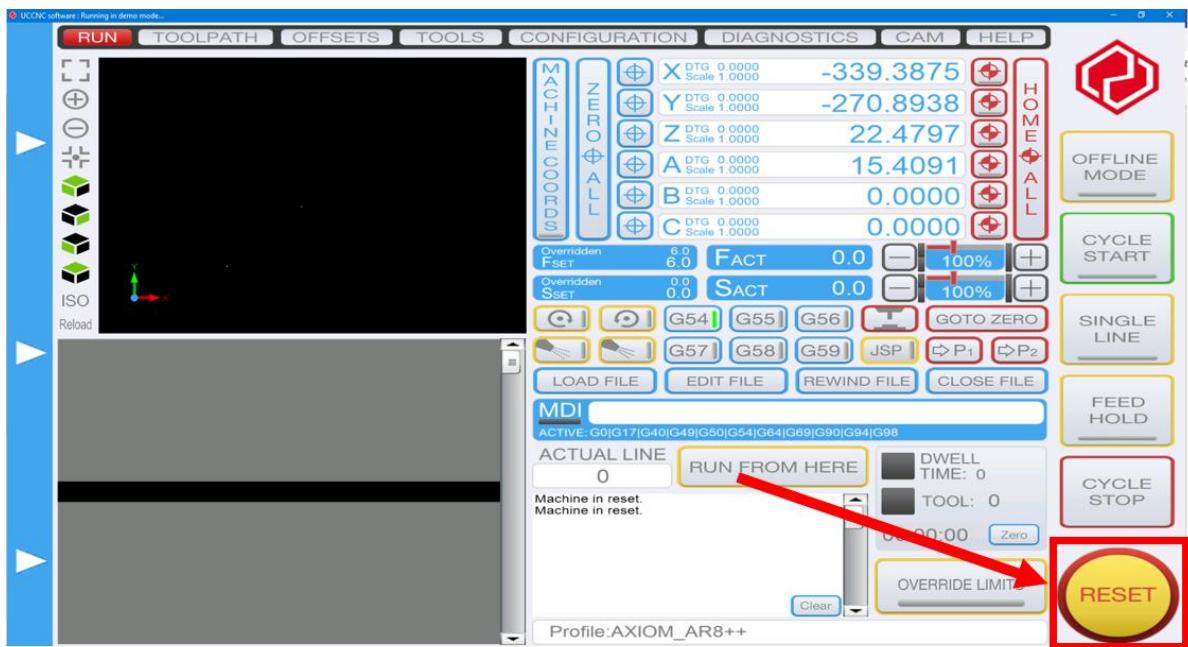
Figure 23: UCCNC main screen summary

- Figure 23 shows the main screen once you open the UCCNC controller.

    o NOTE – top left hand side. You should NOT see the text "Running in Demo mode" is should just say UCCNC software after the icon.

    o NOTE – In the profile summary box (next to red reset button) you should see the text "(machine model here)" showing you have loaded the correct profile.

- Hit the reset button (it should be flashing yellow and red and is shown in Figure 24). Once clicked it should remain solid red in colour. You should now be able to control the machine with your UCCNC software

NOTE – Whenever you are not directly interacting / controlling the machine you must click the reset button, so the **button flashes red and yellow**. This is a safety procedure to make sure you don't accidentally turn on the spindle system.

Figure 24: Reset button on UCCNC front panel

The UCCNC lets you jog the machine with the Keyboard in **Rapid** and **Slow** jog modes. The directions of movement are assigned as follows

- Right arrow key – jog X right / positive

- Left arrow key – jog X left / negative

- Up arrow key – Jog Y up / positive

- Down arrow key – Jog Y down / negative

- Page up – Z axis up / positive

- Page down – Z axis down / negative

o Holding the "**Shift**" key while pressing the directions of movement keys will move the machine in rapid mode.

o Releasing the "**Shift**" key will result in the slow jog motion
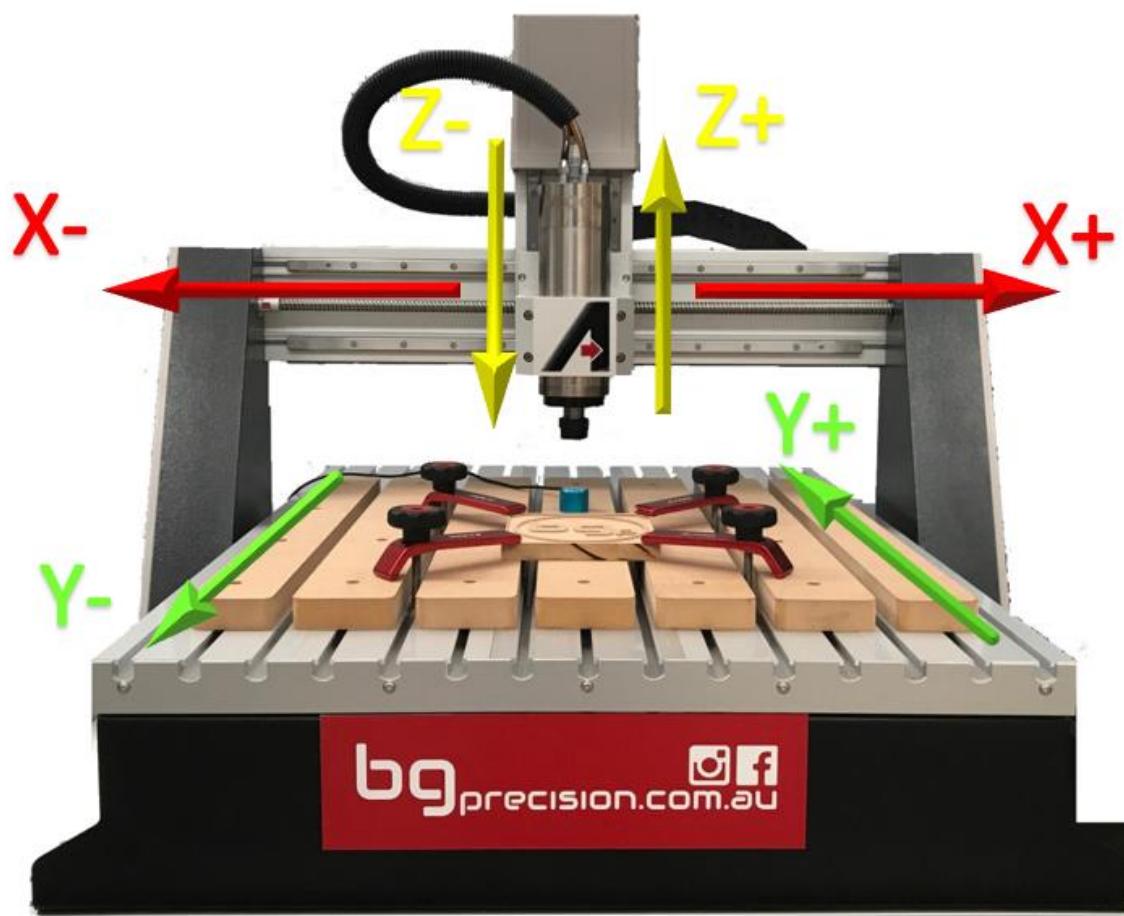
Figure 25: X - Y - Z Axis explained

Z positive move

Z negative move

Hold for Rapid Move

Y positive move

X positive move
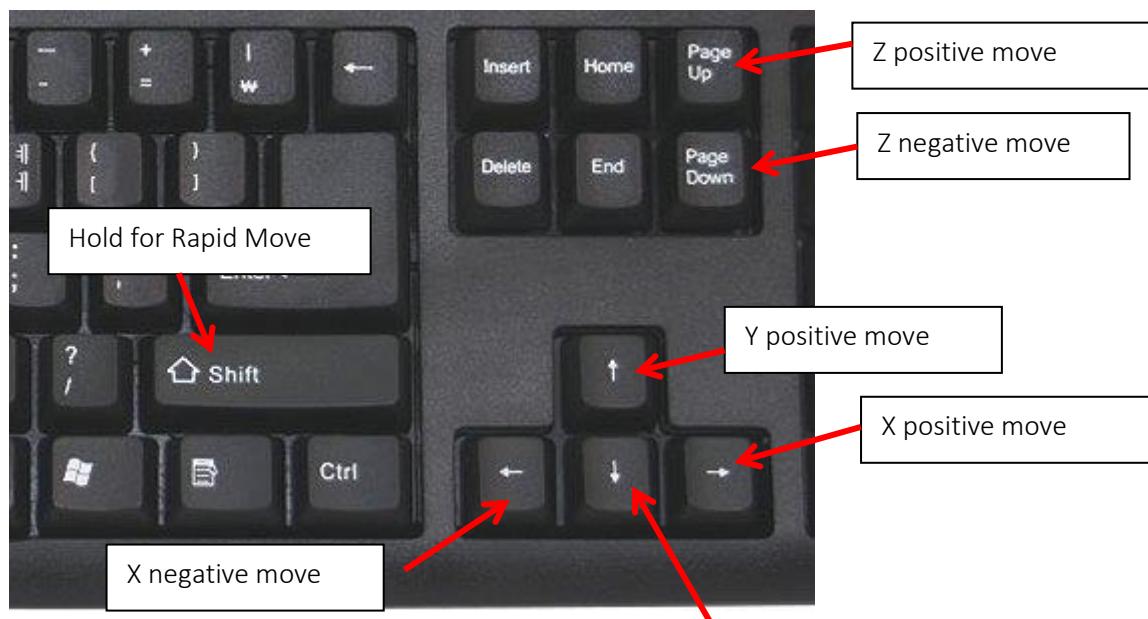
X negative move

Figure 26: UCCNC Keyboard commands

Y negative move

You can also control your machines movements via the UCCNC software interface on your PC. Hover your mouse over the left-hand side of the UCCNC interface for the movement controls menu to pop out. See Figure 27. For more details on the Jog Panel see section 0

Jogging the Machine.
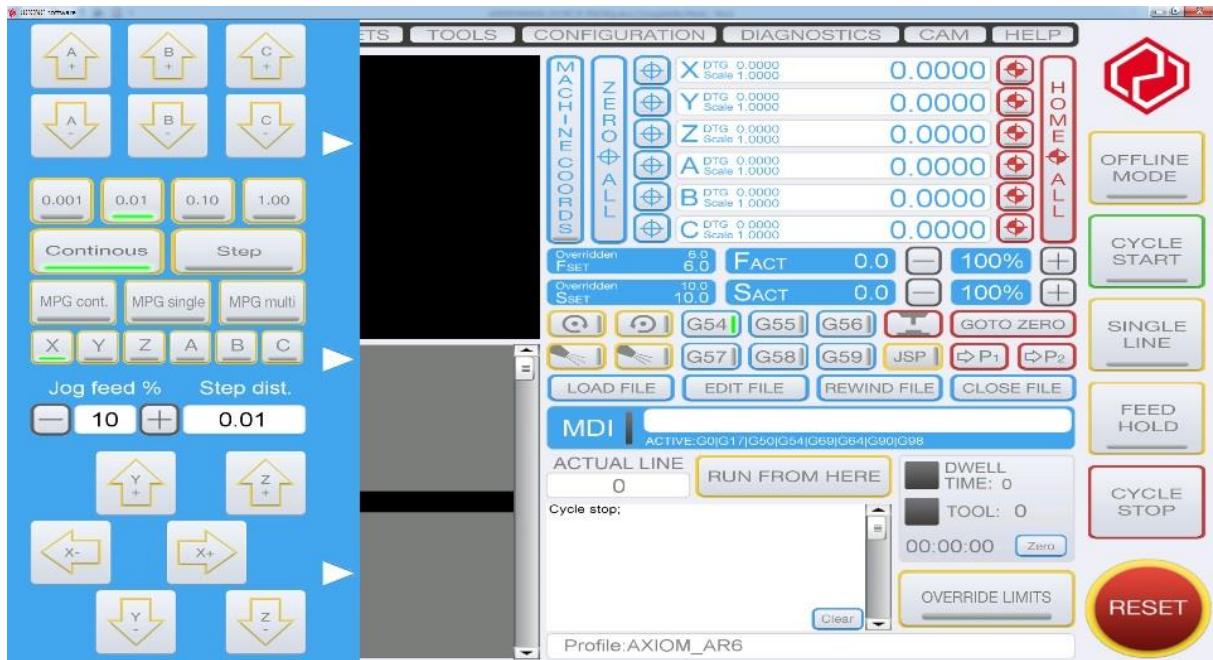
Figure 27: UCCNC screen interface movement controls

4.1 Machine Homing and limit switches

You must reference the machine each time the machine gets turned on to tell the software where the edges of the machine movement are. This can be done by using the "Home All" Command as you can see in Figure 28. The machine will move each axis to a single extremity until is hits that axis limit switch. It will then move away until the switch releases setting the machine home position. This is how the software knows where the machine is relative to the limit switches.

Nearly all machines will home the Z axis first which clears any obstacles on the table, then the X and Y axis thereafter. This is useful to take a mental note of.

Crashing the machine will not do damage provided you press the ESC key on your keyboard or emergency stop button shortly after. The stepper motors will simply "buzz" and the machine will not move.

Soft limits are software defined limits that can be enabled to prevent your machine from moving outside of its zone of movement (and crashing) including activating a limit switch. In order for the soft limits to work you have to make sure you have loaded the correct profile for your machine as the values are related to the geometry of the machine set in the machine profile. **For the softlimits to work properly you must home the machine every time you turn the machine on.** For more information on SOFTLIMITS please refer to section 4.3 Softlimits
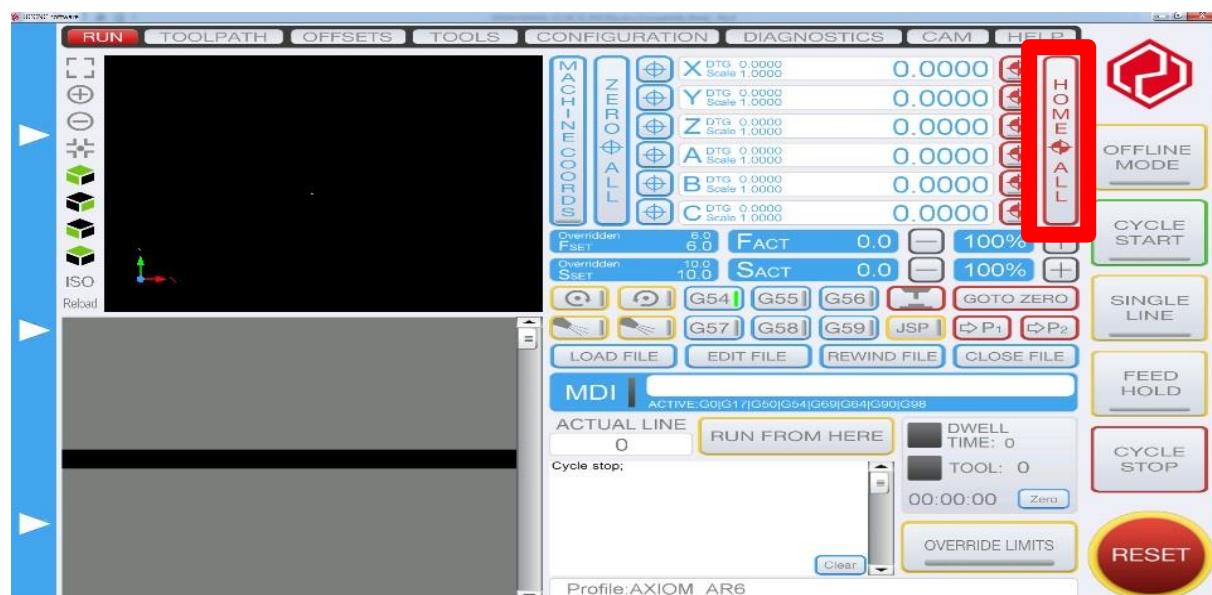

Figure 28: UCCNC Home All Command

4.2   LIMIT SWITCH TRIGGERED

Occasionally it is possible to jog the machine to the end of the machine travel such that a limit switch will be triggered. The software will now prevent you from controlling the machine. You will not be able to reset the software when the limit is triggered.

To determine if a limit is active you can click on the diagnotics tab on the top of UCCNC and refer to the I/O (Input/Output) function monitor to see the status of the functions of your machine. In the example in Figure 29 we can see the "Y limit –" is active. To remove this hard limit signal, we will have to move our CNC machine in a Y + direction.

It is very important you check what limit is active to determine which way to move.
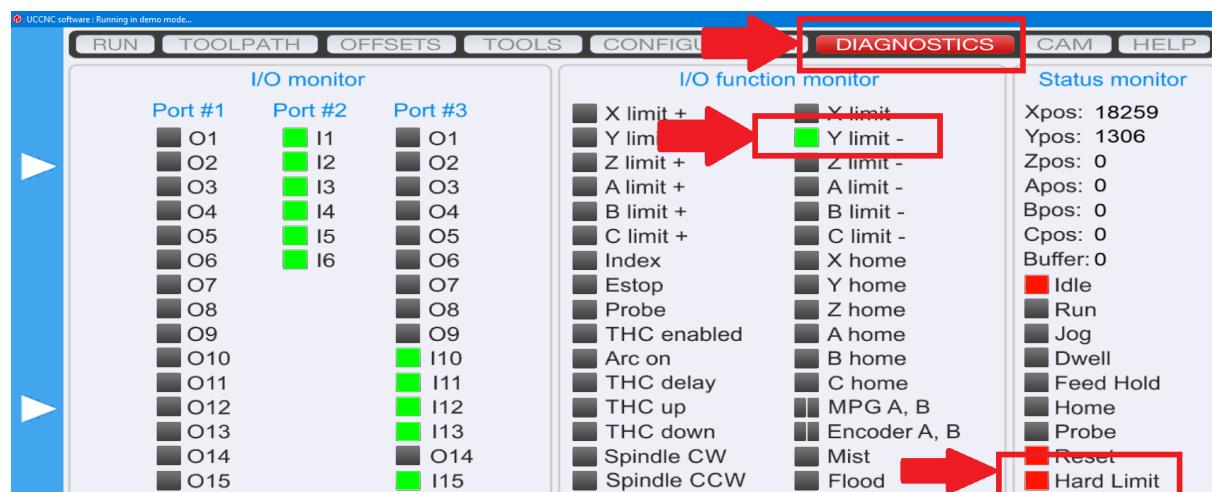
Figure 29: UCCNC Diagnotics

To do this, you must press the "OVERRIDE LIMITS" button in order to reset the machine. This can be done by using the "OVERRIDE LIMITS" command as you can see in Figure 30. Once the "OVERRIDE LIMITS" button is pressed you can then click the reset button and regain control of the machine. Make sure you jog/move the machine AWAY from the limit switch. You are now back in the zone of operation for the machine and can continue.
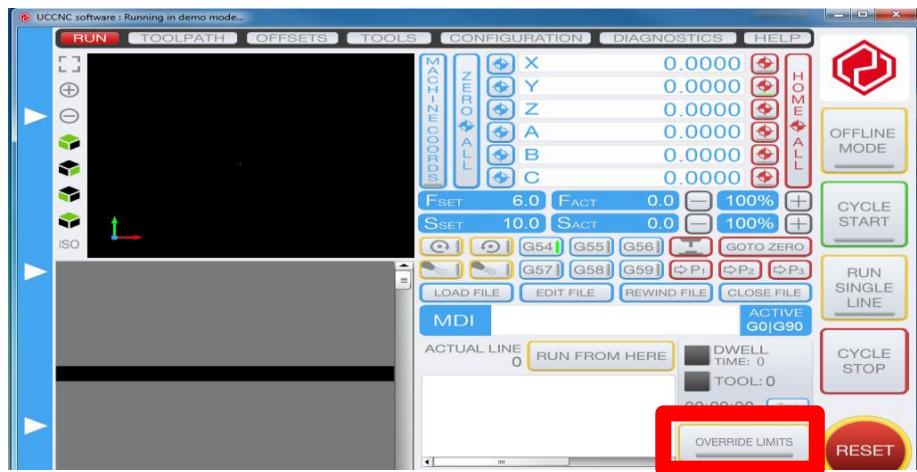
Figure 30: UCCNC Override Limits Command

4.3 SOFTLIMITS

The soft limits are designed to restrict the machines motion to the machines working envelope via the UCCNC software. The area the envelope encapsulates is dependent on the user running a homing cycle on start-up. Otherwise, this area could still me in the next factory.

Soft limits places software limits on movement availability. These limits are imposed by the maximum workspace boundaries based on machine profile settings in UCCNC and the location of HOME.

To get a better understanding between Home Position, Job position, Soft-limit area, Workpiece Area refer to Figure 31: Machine layout illustrated below.
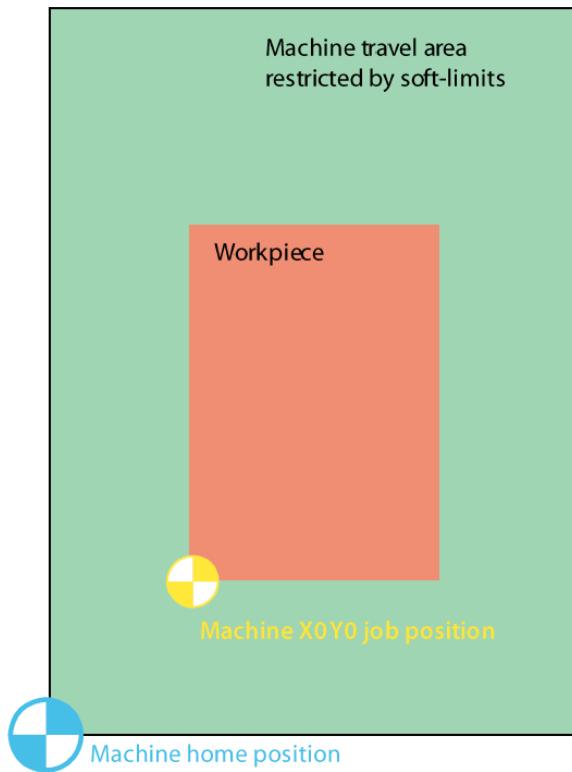
Front of the machine
Figure 31: Machine layout illustrated

Note there are two co-ordinate systems as per Figure 31: Machine layout illustrated

1) The <u>machine co-ordinate system</u> (marked green) where all the background machine co-ordinates and soft limits are stored and located

2) The <u>workpiece co-ordinate system</u> (marked orange) where you spend most of your time. This is the system that allows you to setup and zero our work in the machine bed. This is the default system of number you will see displayed on UCCNC also know as the DRO.

NOTE – if you setup a workpiece that is outside of the machine software limits – you should get an error telling you the job cannot be run.

Figure 32 shows the DRO and the machine co-ordinate button
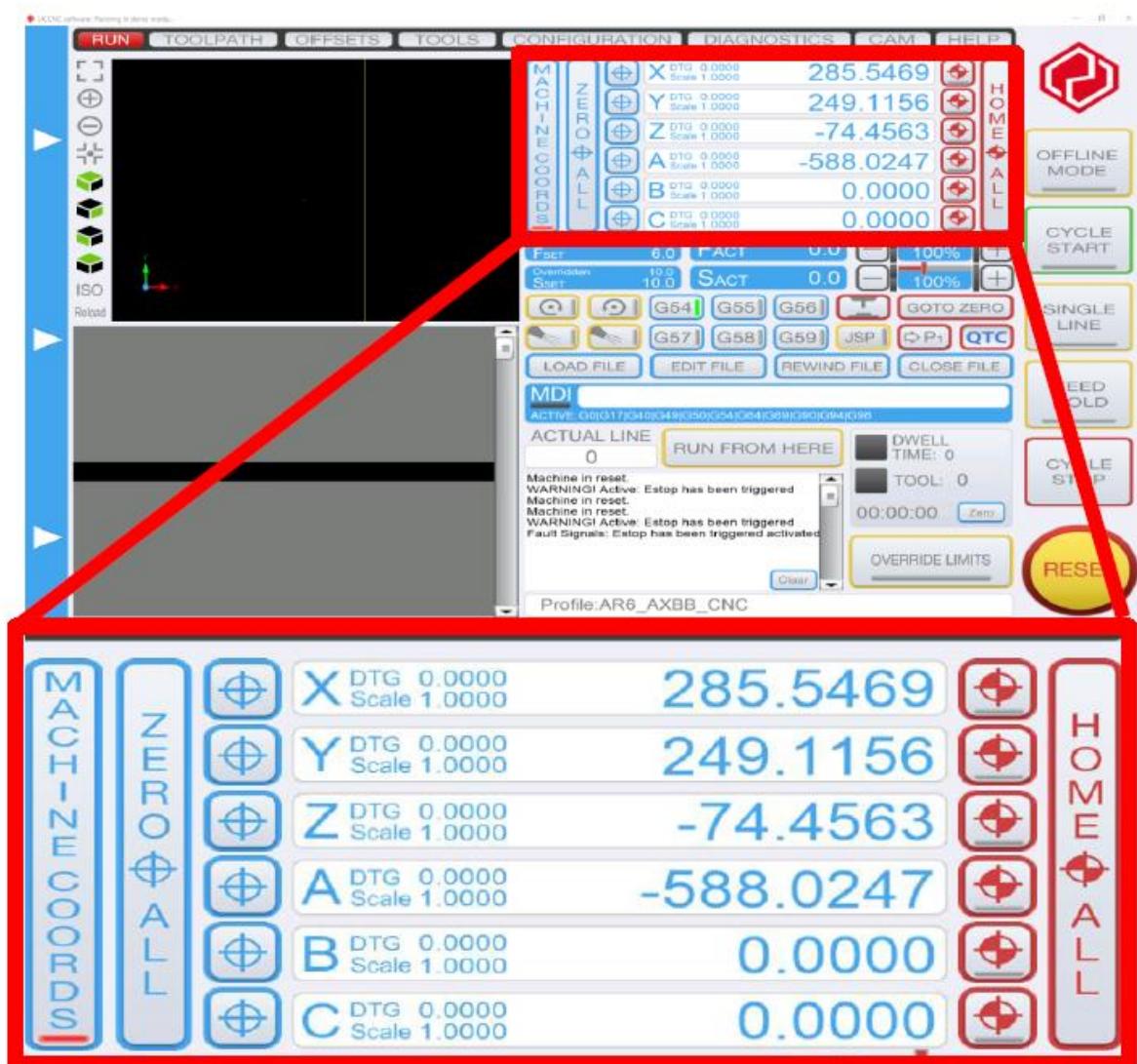
Figure 32 Note when the machine coordinates button is red you are in machine coordinates

When in machine co-ordinates you are in the machine reference system. This is only necessary to activate for advanced users only. On an all our CNCs after the home all routing the machine co-ordinates should read 0,0,0

NOTE be sure to go back to the workpiece co-ordinates when you are finished checking the machine co-ordinates.

You do not need to start your job from the machine home position. You can set your job start position anywhere within the machines working envelope.

There are only two messages in UCCNC related to softlimits.

1) A machine axis position software limit was reached. The axis was now stopped.
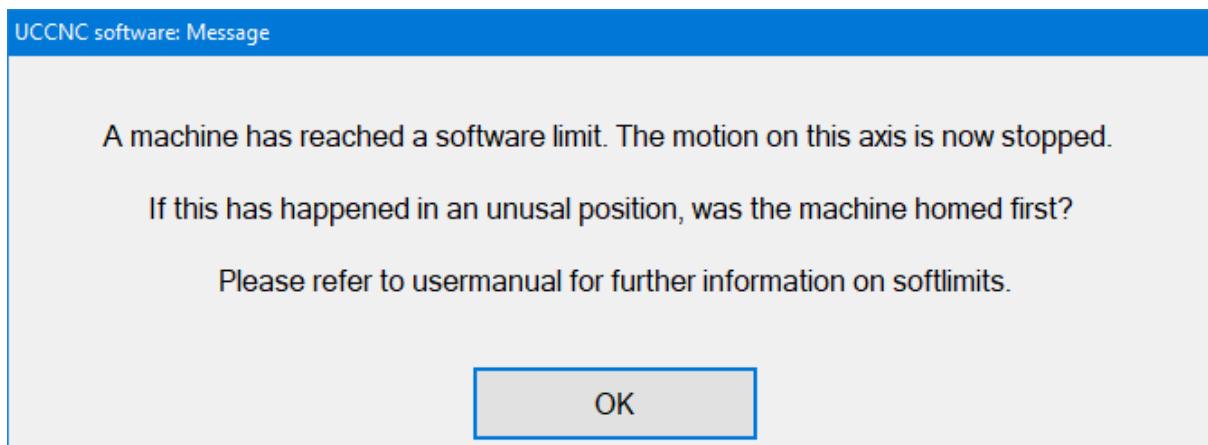

Figure 33: Softlimits message 1

If machine axis position software limit was reached, this implies that you have tried to manually move the CNC machine outside the softlimits area. If you believe this has happened in a strange place such as the middle of travel, have you "Home All" on start-up of the software? Another reason this may appear in an unexpected location would be if the incorrect machine profile was loaded.

2) The current job workspace is out of the set software limits.
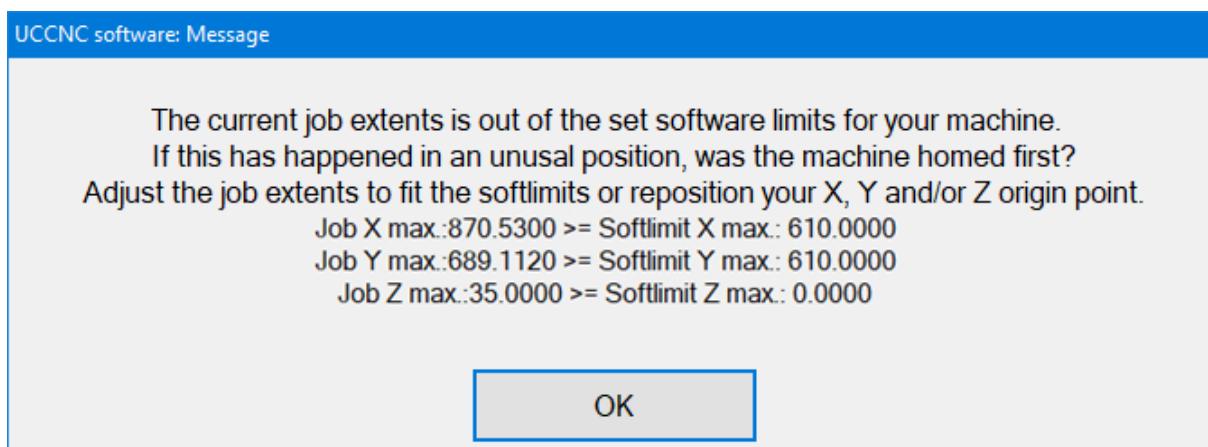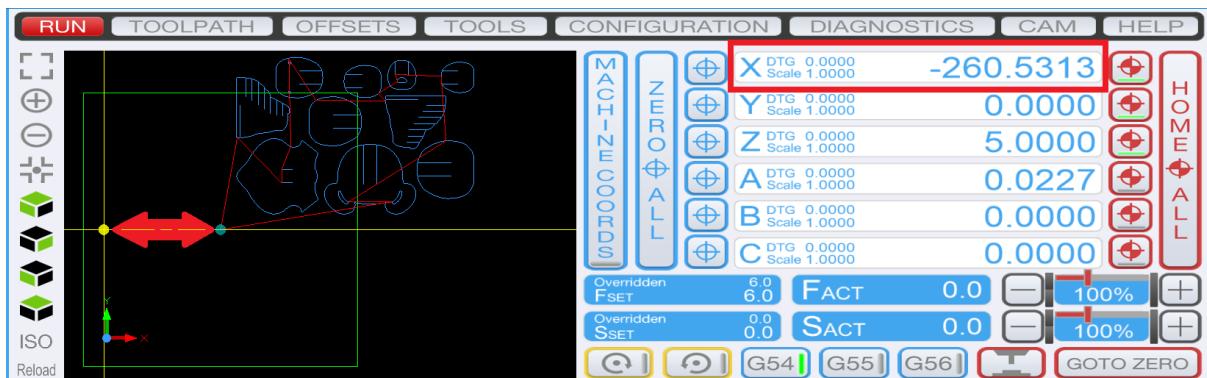
Figure 34: Softlimits message 2
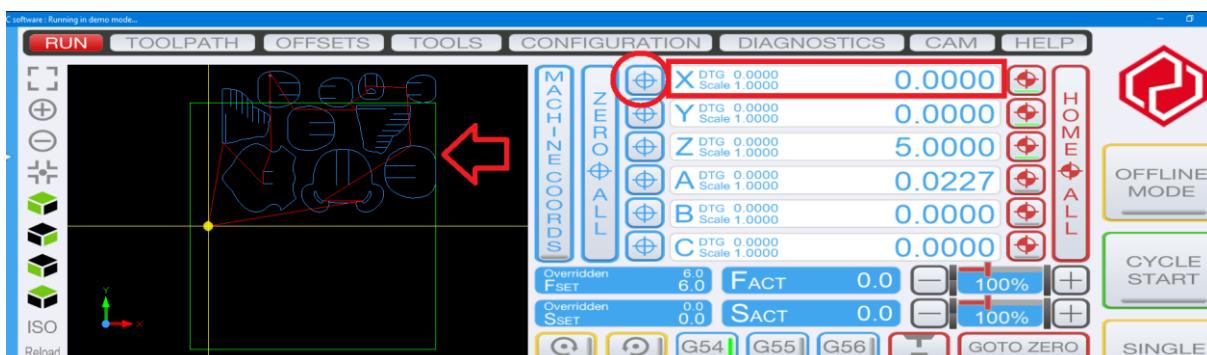
This message normally displays as a warning after you have hit "start Cycle". If the current job workspace is outside of the set software limits there can be two reasons for this:

i) The G-code file loaded is larger than the machines working envelope (softlimit area)

ii) You have set an axis origin in the wrong place and if adjusted it may work assuming the G-code file can fit the machines working envelope. This point can also be related to incorrect job set-up during your CAM process on your design software.

In Figure 32 it clearly states what is wrong with your file you have loaded and user set-up of the machine job position. This also assumes you have created a file that will fit within the machines working envelope. The job x max is 870.53mm. This is the widest point of the job file from the machine home position in the x axis. The softlimit X max for this particular machine is 610mm. This suggest that my user defined machine job position of the x axis needs to move back a value greater than 260.53 (870.53 - 610 = 260.53) and the x axis zerod in this location to make it fit.

After you reset you working position. You can see from the toolpath preview your job will now fit in the softlimit area:

The Z axis is not as easy to see, but common mistakes are incorrectly setting your job Z position in the incorrect position. Review your CAM job setting in your design software and determine if you have indeed set the material surface or the machine bed.

Second to that, commonly overlooked is the clearance values set in your CAM job settings. You may only be cutting 5mm deep but if your job clearance is 500mm and your z travel is only 100mm then your job z max will be exceeded. Review your setting in your CAM software.

Final common factor is the tool is too large or there is too much tool stick out for the available travel of your machine.

By default softlimits and softlimits file precheck should be activated on your profile as per Figure 35
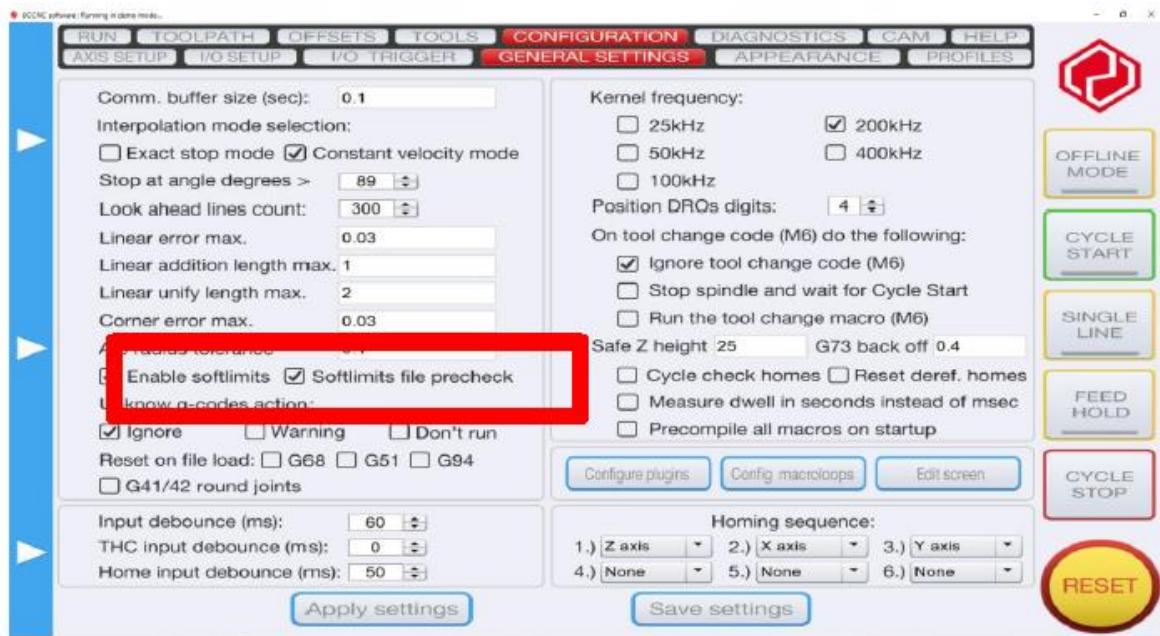
Figure 35 Enable and Disable softlimits

NOTE we do not recommend turning them off…

However, if you have a limit switch failure or other reason for homing failure – they can be turned off to keep machine going so as not to stop production

We recommend you call us before turning the soft limits off.

4.4   SPINDLE SYSTEMS AND SETTING SPINDLE RPM

DANGER Before you change the tool in the spindle and before you handle any of the spindle components in any way you must de-activate the spindle system to avoid accidental spindle start-up during handling.

The first stage in disabling the spindle system is to hit the reset button in UCCNC. This will de-activate the charge pump signal and will disengage a safety relay in the machine control electronics preventing unexpected start-up of the machine spindle.
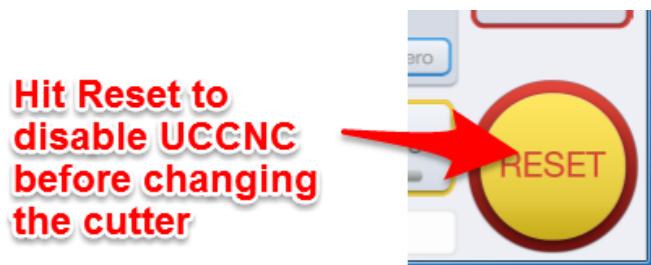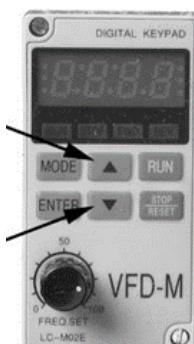

Figure 36: Reset to disable spindle relay

Further details on how to change tools and use the collet system are outlined in subsequent section.

All our CNC machines are equipped with a VFD invertor for spindle RPM control. This can be found on the front of the control box.

On all our CNC machines the factory setting for spindle speed control is determined via UCCNC software or G-Gode (outputted by Vectric) only.

The buttons and the dial on the VFD, are intended to be used for programming only and cannot be used to set the spindle speeds.

The display shows the RPM in 60Hz frequency. This means that while the RPM is 0-24,000, the display will show 0-400. Each 100-displayed is equal to 6000RPM.

Delta VFD - RPM Values	
Frequency	RPM
250	15,000
267	16,000
283	17,000
300	18,000
317	19,000
333	20,000
350	21,000
367	22,000
383	23,000
400	24,000
Frequency equals desired RPM/60	

Figure 37: VFD RPM Values

To set the spindle speed manually in revolutions per minute (rpm) you can type "S" for Spindle speed and the value you want to set the speed in RPM into the MDI (Manual Data Input) on the UCCNC faceplate.

For example, type S6000 and press enter to set the Spindle RPM to 6000

The spindle will turn at that speed when it has been programmed to start turning. It is OK to program an S word whether the spindle is turning or not. If the speed override switch is enabled and not set at 100%, the speed will be different from what is programmed. It is OK to program S0; the spindle will not turn if that is done. It is an error if:

• the S number is negative.

Note that the Spindle speed can be override in the 1-300% range using the + and – buttons in the spindle speed override DRO.

To manually start the spindle, select the spindle On/Off Button as shown in Figure 38.
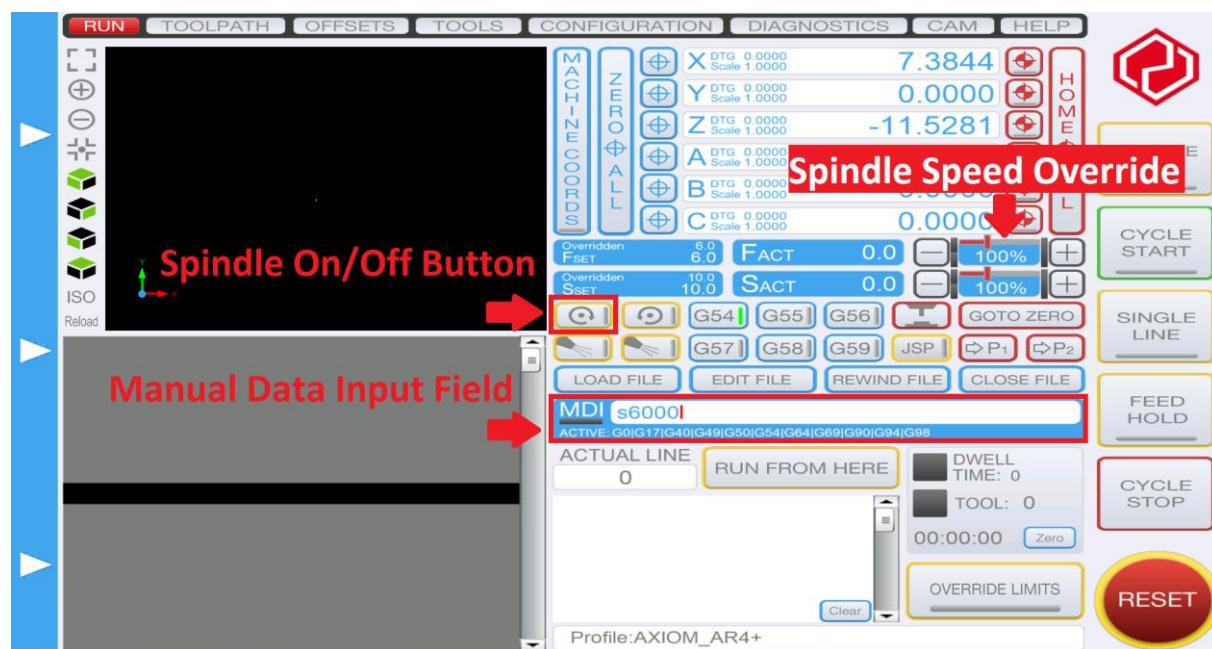
Figure 38: UCCNC Spindle Control

5 UCCNC ONSCREEN INTERFACE

Your UCCNC Profile for the CNC machine comes with Custom UCCNC screen set. Below describes some of the Default screen tabs and functions which may differ with slightly with custom screen files. This default screen set file contains components and graphics for a 6-axis machine controller. Print screens of this screen set are shown in this documentation. For more information on how to edit and customise the default screens please refer to the UCCNC user manual provided in your installation folders.

5.1 UCCNC PAGE TABS

The TAB pages on the default screen set are as follows:

5.1.1 RUN
This page is the main page of the screen, it contains the buttons to load, edit, run, close a G-code file. It also contains buttons to switch the spindle on/off, select the offset coordinate system. It contains a 3D toolpath viewer and a G-code viewer. On the top of the screen the 6 axis position DROs and actual and set feedrate, spindle set rotational speed and spindle actual rotational speed DROs are taking place. An MDI (manual data input field) is placed on the middle of the screen, this component allows a manual G-code input via the keyboard.

5.1.2 TOOLPATH
This page is to get a clearer view of the toolpath loaded into software. The page contains a large, high resolution 3D toolpath viewer and buttons to navigate, zoom and to have different viewing angles of the toolpath.

5.1.3 OFFSETS
This page contains the offset coordinate system parameters. The offset coordinate systems are on 6 sub TAB pages and are G54, G55, G56, G57, G58, G59 respectively. The actual offset coordinate system can be selected on the Run page. The offset values on the selected offset system is applied to the coordinate DROs of the machine and the name of the actual offset system is indicated in a label on this screen for example "Active fixture: G54" means that the G54 coordinate system is selected. Currently work offset is available for all the 6 axis and in addition a tool offset is available for the Z-axis. The offset numeric values are one by one all editable on the screen.

The current position which is the actual position of the machine can be offset with a single button press. Also the work offset can be cleared with one button and the tool offset can be cleared in a similar way with a single button press.

5.1.4 TOOLS (ONLY APPLICABLE TO QTC AND ATC CONFIGERATIONS)

This page contains the tool offsets for the Z-axis, because as previously described tool offsets are currently only available for the Z-axis only. There are in total 96 pieces of tool offsets are available marked Tool#1 to Tool#96 on the screen. All tools can have its own tool length offset value. All tools numeric values are editable on the screen and the values can be saved to the profile file.

5.1.5 CONFIGURATION

This page is the most complex tab page, it contains several sub TAB pages and this page has all the software configuration parameters. We do not recommend you change any setting here. Your machine specific profile has been designed and tested to ensure your machine works how it was intended to.

5.1.6 DIAGNOSTICS TAB PAGE

This page shows data and feedback about the current job and the machine properties, like I/O and functions logic states. This page can be very useful when problem solving. We recommend you always check the current job properties prior to starting any job and ensure they are within your machining limits.

5.1.7 HELP TAP PAGE

This page lists the supported G and M and other codes with basic descriptions. Also, the motion controller device parameters and the license key validity can be read here.

5.2 OVERRIDING THE FEEDRATE AND THE SPINDLE SPEED

The programmed feedrate and spindle speed can be overridden any time, even when a code is executed and when a motion is in progress. Both properties can be overridden in the range of 0- 300% of the programmed value. The override feature is useful for example if the programmed feedrate is too slow or too fast and need to be adjusted, but the operator does not want to bother with regenerating the code with new feedrate values. The same is true for the spindle speed override, if the spindle speed is controlled from software then the programmed speed can be overridden on the fly which makes the system more comfortable for the operator and more productive saving lots of additional work and time.

To override the feedrate use the + and − buttons next to the feedrate and the spindle speed DROs. Pressing the plus button increases the value with +10% if the value is equal or above 10% and it increases +1% if the actual value is below 10%. Pressing the minus button decreases the value with 10% in case the current value is above 10% and decreases the value with 1% if the actual value is equal or below 10%.

If the feedrate override is set to 0% then the motion stops and stalls till the DRO value is changed to any higher value.
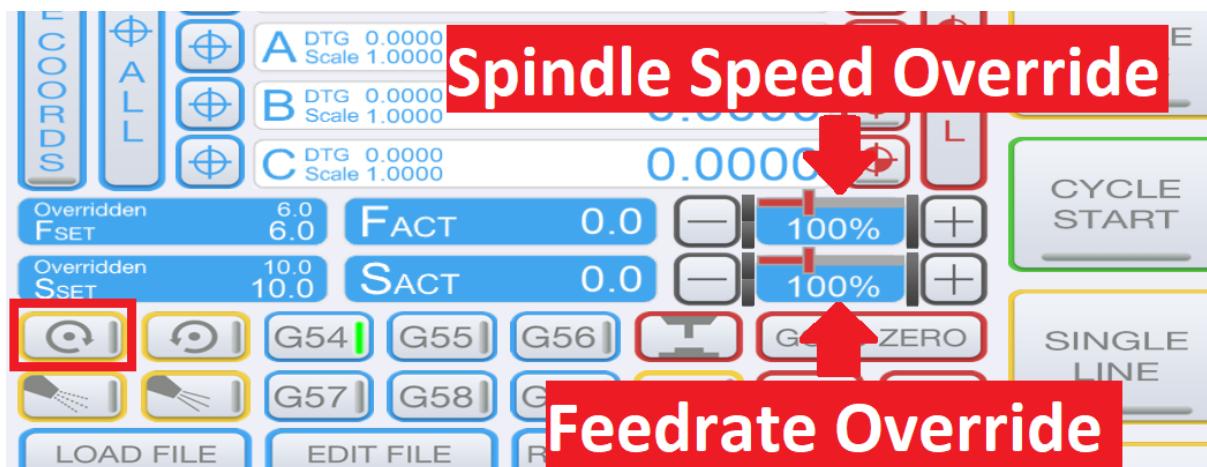

Figure 39: UCCNC Feedrate and Spindle Speed

5.3 JOGGING THE MACHINE

As well as using the keyboard commands as highlighted Section 4 above you can use an onscreen jog controller. The jog controller is used to move the machine manually by pressing screen buttons. Also, the manual pulse generator and the associated buttons can be found on this screen.

The jog panel is located on the very left side of the screen and on start-up this screen is hidden, only its right-side border is shown. Touching the panel's border with the mouse pointer makes the jog panel to pop and appear on the screen. The panel is not available when a motion is in progress, when the machine controller is busy executing commands.

There are + and − jog buttons on the jog panel, pressing these buttons all axis can be jogged to the negative and to the positive directions. The Jog feed sets the feedrate of the jog movements, the value is defined in percentage of the set maximum (G0) feedrate of the axis.

The mode of the jogging can be selected with buttons on the screen, there are continuous and stepping modes.

In continuous jogging mode the machine axis jogs while the jog button of the axis is being pressed and the jog finishes when the button gets released.

In step jogging mode the axis moves the selected distance for every jog button press. Currently there are 0.001, 0.01, 0.1 and 1 unit lengths are selectable.
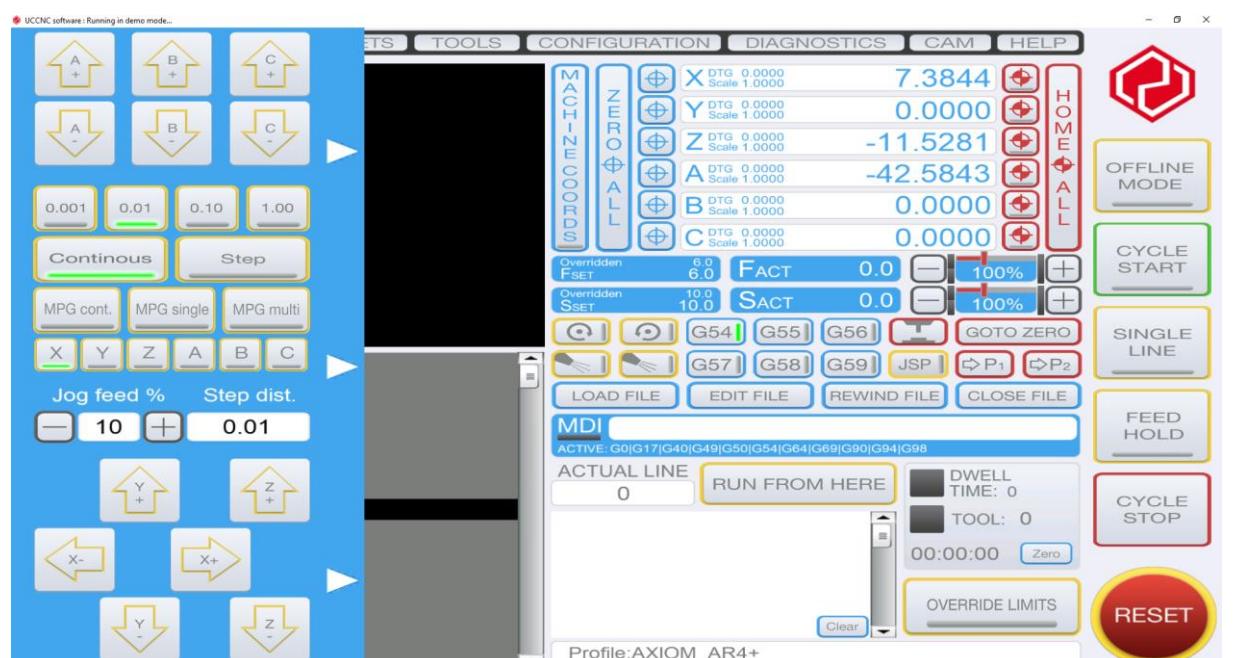

Figure 40: UCCNC Jog Controller

6 QTC UCCNC INTERFACE

For the Executive or Symbiosis CNC with QTC we have made a customer interface to display the relevant information to help you configure and operate your QTC Spindle
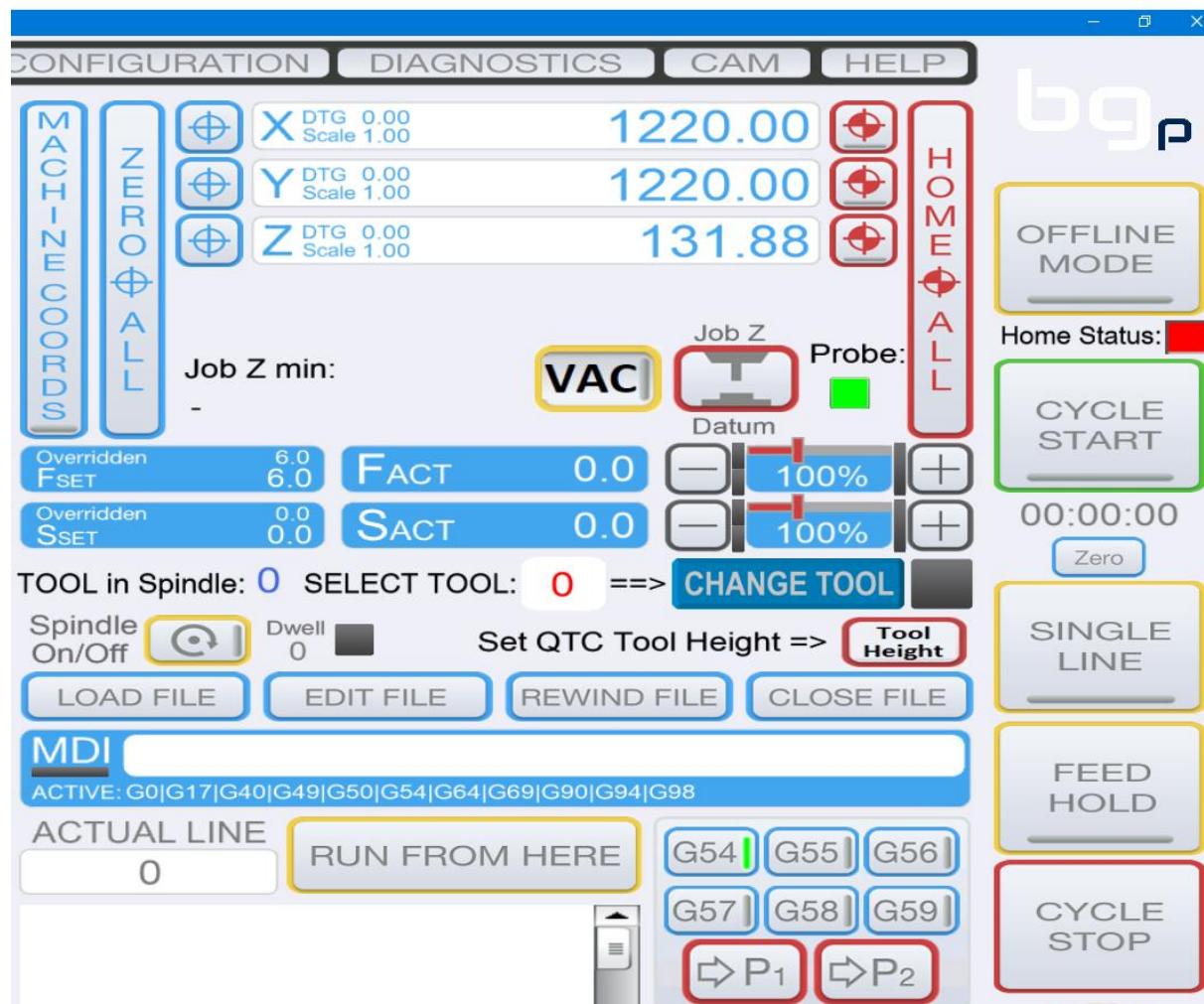

Figure 41: EXEC 16 QTC UCCNC Interface

Job Z min: -	When you load a file in UCCNC this will display the minimum Z depth of this file. This is a working coordinate not a machine coordinate and relies on the user setting the Job Z datum
VAC	This Clickable button will turn on and off the Vacuum Table. There are a few ways to turn on the vacuum table on your Exec 16. You can turn the switch on the hand held controller to override any automation. You can use this VAC button on the screen. Alternatively you can use the M code "M20005" in the MDI or include it in your G-code

Job Z Datum	This clickable button is how you set your Job Z datum for your job. This can be on the job surface or machine table for example as pre-set in your CAM. It requires you to plug in the Auto Touch probe which is located on the rear z axis. This has a Pre-Touch Function built in to ensure the probe is functioning prior to operation.
Probe:	This LED indicator will be lite green when the probe is active and has useful continuity.
Home Status:	It is imperative you Home a Servo controlled machine **EVERYTIME**: 1) you start UCCNC 2) you place UCCNC in "Reset" 3) after you disengage an Estop. Both the individual Axis home buttons and a separate LED shows Home status with a green or Red LED
TOOL in Spindle: 0	The "TOOL in Spindle" indicates what tool number is active in the UCCNC software. It is vital the this number match what toolholder is actually in the spindle which will insure the correct tool offset is applied.
SELECT TOOL: 1	The "SELECT TOOL" is a data field that the user can use to enter in a tool number to commence a Quick Tool Change. Enter in the number "1" for example and press enter. Nothing will happen until you execute a "Change Tool"
CHANGE TOOL	This is an automated sequence that will move the machine to a predefined location without warning (once clicked) for the user to execute a Quick Tool Change. It relies on the user entering a tool number into "Select Tool" and the number being different than "TOOL in Spindle". There is a LED that will illuminate Red when the "Change Tool" Sequence is being executed.
Spindle On/Off	You Gcode will automatically turn your spindle on upon cycle start assuming you have defined a spindle speed greater than zero. This clickable button allows you to manually turn on or off the spindle if needed.
Dwell 0	Dwell is a pause of motion where the time has been predefined to allow the spindle accelerate or decelerate to the pre-defined speed.
Set QTC Tool Height => Tool Height	This clickable button allows you to set each individual tool height offset. You must load the correct tool in the spindle prior to running this function for the offset to be saved to the tool offset table. DO NOT mistake this for "Job Z Datum"

	P1 and P2 are predefined locations written into the M200 and M201 macros respectively. P1 has been set to the front left of table and P2 to back right.

7 ATC UCCNC Interface

TO BE ADDED...