IMAGINATION TO REALITY

i2R CNC Router User's Manual

bg precision

CNCdrive motion controls

BG Precision

Version 2.0

July 2025

Agent and Appointed Reseller for Australia and New Zealand

BG Precision PTY LTD

Unit 1/82 Brunel Road

SEAFORD

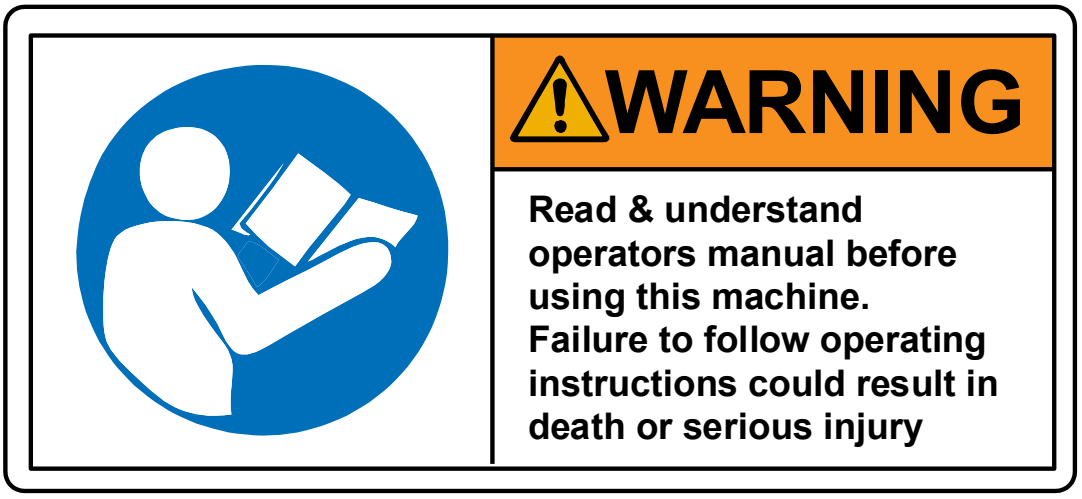
3198 VIC

Australia

Mob: +61 (0)405685515

Email: info@i2rcnc.com.au info@bgprecision.com.au

Website: www.i2rcnc.com.au www.bgprecision.com.au



Before using/powering on the machine, the device should be carefully checked to make sure all connections are secure, and the device is technically sound as highlighted in this user manual.

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

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 i2R CNC AUSTRALIA / BG Precision PTY LTD. This manual is a general introduction to CNC and specifically how to use the i2R CNC with UCCNC.

Firstly, please inspect the machine and all components after delivery has been received. Please check and ensure all aspects of the machine and associated extra items are in good condition and there is no evidence of damage or wear to any components due to the shipping process.

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

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

- understand what a CNC router/mill/laser is and can do
- read and understood the content of this user manual prior to using the system
- be able to always exercise control of the CNC system
- 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. I2R CNC AUSTRALIA / 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 must 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 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.

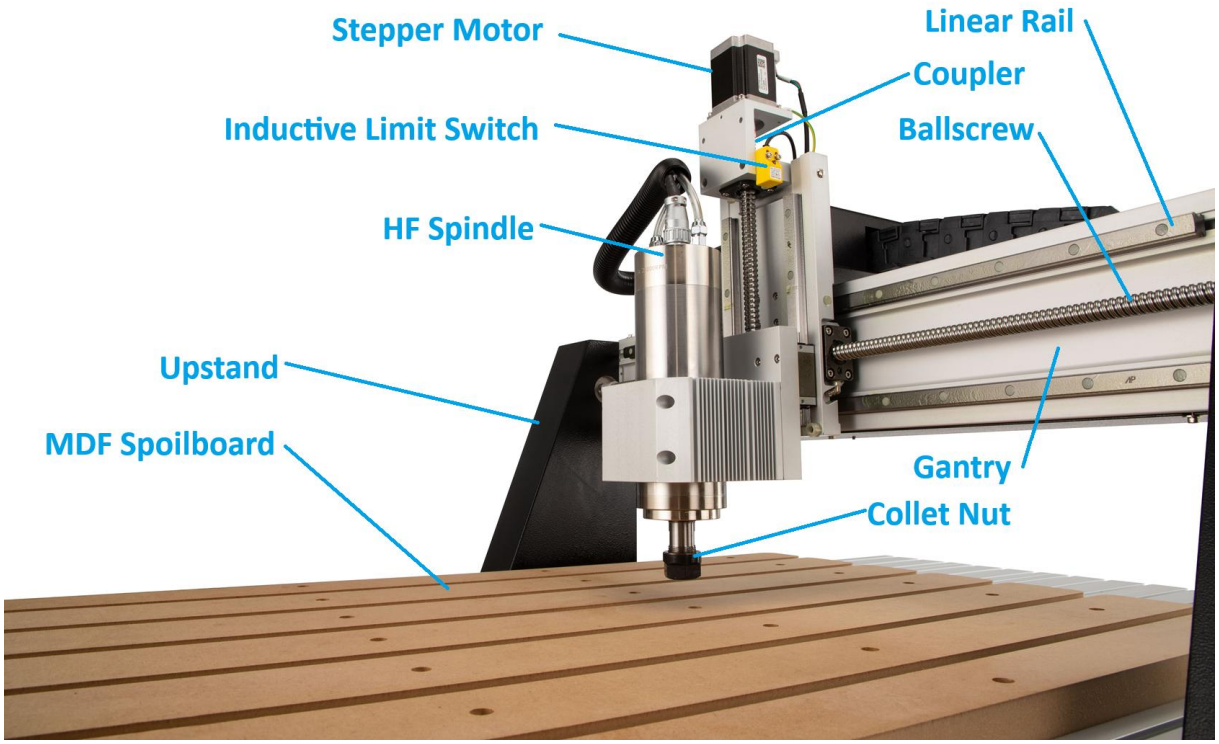
If there are any further questions after reading and understanding the manual, or if anything is not clear, please contact us via email, at info@i2rcnc.com.au

2 CNC Machine Overview

Broadly speaking a "CNC Router" is a computer-controlled machine that has a spindle mounted on it that holds a cutting tool (router bit). It is typically set up with 3 directions of movement referred to as the X, Y and Z axis. The position of the router is determined by a computer telling the motors mounted on each axis how much to move in each direction.

Using this method of positioning, any location within the machines work area (envelope) can be defined and the router can be moved within that space. As the machine is driven by a computer telling it where to move, the operator uses a software program to draw the shapes they want to cut and create the path that the machine will follow.

2.1 COMPONENT IDENTIFICATION



Component	Function
CNC Controller	The controller is the white control box that has the electronics necessary to move and manipulate the machine position and spindle speed. We utilize a Windows PC based controller called UCCNC. All our CNC machines come with the controller and UCCNC control software.
CNC Router	A computer-controlled machine for subtractive machining of non-ferrous materials such as wood, plastics, brass and aluminium
HF Spindle	All our machines are equipped with a High Frequency (HF) Spindle which has better power and torque characteristics than a single-phase fixed frequency spindle such as a trim router or drill. There is a frequency invertor inside the CNC Controller that controls the spindle speed by varying the frequency from 0-400hz. This in turn controls the speed from 0-24,000 RPM.
Collet Nut	CNC Spindle use a collet and nut system for tool holding. This allows you to produce a high clamping force for tool holding with very accurate, self centering alignment. The i2R CNC machines use an ER20 Collet system which is the series size and they come in a range from 1-13mm to suit each size shank of your tooling.
Tslot Table and Sacrificial layer	Your work is usually placed on the bed of the CNC. Most of our machines come with a T-Slot table which allows for multiple clamping methods such as T-slot clamps and T-nut inserts. It's a good idea to always use a sacrificial layer under the work piece you are machining. This is especially important when you are

	milling all the way through the material. 3-4-6mm MDF often works well. It saves you blemishing the T-slot Aluminium table. Often if you use a thick layer you can use a drill and screws to fix your work. Crude but functional.
Gantry	The gantry is what supports the spindle above the material and allows the CNC machine to traverse in the X direction.
Upstands	The Upstands support the gantry and are normally connected under the table via an undercarriage allowing the CNC machine to traverse in the Y direction
Linear rail	The profiled linear rails are one of the best linear motion systems for machine accuracy. The coefficient of friction on linear guides is only a small fraction of what is created by traditional round guides and can take loads in all directions. With these features, our CNC machines achieve the highest level of system accuracy and greatly enhanced moving accuracy to allow quality surface finish when machining woods, plastics and alloys.
Ballscrew	Ball screws offer a high level of mechanical efficiency, can carry remarkably heavy loads and produce minimal friction. This is because they are designed with a nut in which ball bearings circulate. The ball bearings create a smooth gliding surface for the screw, thereby reducing friction and subsequently increasing the ball screw's lifespan. Motion on every axis of our CNC machines is controlled by high-precision ball screws. Their ability to create low internal friction, while withstanding high thrust loads at high rates of speed, is paramount to the accuracy of the CNC system
Coupler	This is a coupling between the motor shaft and the ballscrew shaft. Care should be taken to ensure this is tight and not slipping.
Inductive Limit Switch	Normally only situated on one end of each axis is an inductive limit switch used to find the "home position" which in turn allows the software to define the "softlimits" which is the extent of travel of the CNC machine. An inductive switch has no mechanical parts which means it will last much longer than a mechanical switch and maintain its accuracy. It requires a metallic striker plate to pass in front of it to activate it.

2.2 KEY AREAS OF KNOWLEDGE

As with any subject, the more time you invest in learning about CNC and the related technologies, the more you will get from it. To achieve the best results, there are a few key areas which you should concentrate on:

1. Computer skills

One requirement common to all aspects of CNC work is how to use a computer to perform basic tasks. You will be working with computers and computer programs during almost all the steps of the process as you design your parts and need to understand basic operations such as starting and stopping programs, saving, copying, and deleting files, finding files stored on your computer and installing programs and updates.

Your CNC machine is also run by a computer, this may be a standalone PC or a dedicated Control Box with PC built in.

By purchasing a CNC Machine, it is assumed that a basic knowledge of computers and the Windows operating system exists. If you don't feel comfortable with your current computer skills or are new to running a PC then it would be well worth taking a basic course or buying a general guide to working with your PC.

2. Design & Toolpath Software

Before you can cut anything with a CNC, you need to first create the design layout that the machine is going to follow to cut the parts. The software you choose will play a significant role in successfully creating projects with your CNC. Simply put, the design software will allow you to transform "pencil and paper" ideas to a set of instructions used to run the machine. When done correctly, the end result will be a physical product you can touch and hold, that has value and purpose and delivers a great sense of achievement.

3. Operating and Maintaining your CNC Machine

If you currently own or use a CNC machine, you already know how important it is to keep it properly maintained and adjusted, to know and understand its limitations and how to set it up correctly to run a job.

If you don't own a machine yet, then it's important to spend time thinking about what you want your machine to be able to produce, as this can eliminate a lot of future frustration. Cost will always be an important factor but realize that you need to balance that with capabilities, because nothing can be

more expensive than a machine that cannot do what you need. For example, if you want to cut large sheet goods then a desktop model will probably not be your best choice. However, if you only have room for a small machine this may be your only option, and you need to understand its limitations on how large a part it can cut. Only you can determine what this balance will be for your situation and budget.

Some important considerations when researching the purchase of a machine or when looking at building one yourself include size, speed and accuracy and the technical support offered both before and after the purchase. As with software, the importance of a company's reputation, support, and an active website and/or forum cannot be understated.

Every CNC machine needs software to directly drive its movement; this is commonly referred to as the 'Control Software'. All machines sold by i2R CNC Australia utilise a Windows based control software called UCCNC.

4. Knowledge of Materials and Tooling

The user is responsible for defining the tool parameters in your CAD/CAM Software such as Fusion360 or Vectric. When setting up tools, one standard setting does not cover all toolpath operations to efficiently machine/mill all materials. The geometry you will need to input into the tool setup screen are: number of flutes, diameters and/or Angles and this information should be on the tool label.

When it comes to obtaining the best possible results, you cannot forget the material you are working with, nor the tool you are using to cut it. The type of material will factor into every stage of the Project – from initial concept through final finishing.

The common materials people using CNC Routers work with include; wood, plastics, dense foam board and softer (non-ferrous) metals (brass, aluminium, etc.). If you are not already familiar with the type of material you want to use, there are many sources of information that can help you.

Typical questions you must answer for the type of material include proper tool (bit) selection, how fast you can move that tool through that material (Feed Rate and Plunge Rate), how much material you can remove at one time (Pass Depth and Cut Depth) and how fast the bit should be rotating (Spindle or Router speed). Typically, suppliers of tooling offer information on the correct settings for the router bits they sell.

I recommend you read these two pages:

https://docs.vectric.com/docs/V10.5/VCarveDesktop/ENU/Help/form/Tool%20Database/index.html

https://www.cnccookbook.com/feeds-speeds/

3 SAFETY

The CNC Machine is an electrical appliance and a precision machine. Protect yourself and your investment by reading and understanding the entire owner's manual before attempting assembly or operation. Read and understand the warnings posted on the machine and in this manual. Failure to comply with all the warnings may cause serious personal injury or costly damage to your CNC machine.

This CNC machine is designed and intended for use by properly trained and experienced personnel only. If you are not familiar with the proper and safe operation of a CNC machine, do not use the CNC Machine until proper training and knowledge have been obtained.

Your CNC machine is intended for cutting wood, acrylics, wood-fibre composites, certain plastics and non-ferrous metals. Do not use this machine for other than its intended use. If used for other purposes, i2R Australia/BG precision PTY LTD disclaims any real or implied warranty and holds itself harmless from any injury that may result from that use.

3.1 MEANING OF THE RELATED SYMBOLS

DANGER	Symbols and texts, that are marked with the addition "DANGER", warn against a specific threatening/dangerous hazard (serious injury, long term damage, death) **Unconditional attention must be given to these references!**
WARNING	Symbols and texts, that are marked with the addition "WARNING, "warn against a possibly threatening danger (serious injury, long term damage, death) **Unconditional attention must be given to these references!**
CAUTION	Symbols and texts, that are marked with the addition "CAUTION, "warn against a possibly threatening danger (possible injuries, risk of damages) **Unconditional attention must be given to these references!**
	Where the manual refers with this symbol (shown left) it contains very important references. Compliance is unconditional in order to avoid damage to people and property. **Unconditional attention must be given to these references!**

DANGER through electric shock.

DANGER through mechanical movements – danger of crushing and pinching

DANGER through uncontrolled movement / crashing and heavy loads.

DANGER spindle cutting edge

DANGER through trip hazard

READ THE MANUAL – and always have the user-manual to hand near the machining area.

LOOSE CLOTHING – Never wear loose clothing of jewellery near machine.

WARNING – Ear protection must be worn

WARNING – Face mast must be worn

WARNING – Eye Protection must be worn

3.2 Workshop Environment

1. Ensure that the floor can bear the weight of the machine and work pieces mounted on it.
2. Keep the floor around the machine clean and free of scrap material, oil and grease.
3. Do not lean lumber or other heavy materials against the gantry, guide rails or table.
4. Support the weight of the dust hose attached to the dust shoe accessory to prevent the weight of the hose from dislodging the dust shoe. Ensure that there is enough slack in the dust collection hose to allow the spindle to cover the entire work area.
5. Locate the CNC Machine away from overhead pipes and plumbing fixtures to prevent condensation from dripping on to the spoil boards and control system components.

6. Locate the CNC Machine away from sinks, faucets or other water supplies or storage to prevent splash-out that can damage the spoil boards and control system components.
7. Provide adequate room between the CNC Machine and other machines in the shop to reduce the chance of accidental jarring when transporting lumber or other heavy materials through the shop and while materials are being worked on other machines.

8. Ensure adequate space between machines to allow for the possibility that the work piece will extend over the end of the CNC Machine table.
9. Store cutting tools in a dry location and prevent contact to preserve the cutting edges.

3.3 Health and Safety

1. Always wear approved personal health and safety equipment as indicated for the materials and type of operations that will be performed. These should include a dust mask, hearing protection, safety clothing, and safety glasses/face shield. Do not rely on prescription or over-the-counter eyeglasses; they are *not* safety glasses. Wear ear protectors (plugs or muffs) even during short periods of operation.

2. Before operating this machine, remove any hand, wrist, and neck jewellery and roll sleeves up past the elbows. Be sure to not wear loose clothing which may become caught in the machine and confine long hair.

    a. Non-slip footwear or anti-skid floor strips are recommended.
    b. Wear protective hair covering to contain long hair.
3. Use a dust mask or other safeguards to avoid inhaling dust generated from material being cut. Install dust collection equipment consistent with shop ventilation practices. Remove dust and debris from the floor frequently to prevent slipping. Drilling, sawing, sanding or machining certain materials generates dust and other substances known to cause cancer. Certain materials also emit chemicals and must be dealt with accordingly.

4. In addition to other health hazards, dust from wood and other materials is flammable. Do not operate welding, wood burning, smelting, soldering or other high-heat tools on the CNC Machine table or vicinity.
5. Do not operate this machine while tired or under the influence of drugs, alcohol or any medication.

3.4 ELECTRICAL

6. Make certain the switch is in the OFF position before connecting the machine to the power supply.

7. Check all cords before each use. If damaged, replace immediately. Never use a tool with a damaged cord as it may result in serious injury or cause an electrical shock.
8. Make certain the machine is properly grounded and the circuit is protected with a fuse or circuit breaker in accordance with local codes. Install a separate circuit if necessary, to limit power loss when multiple machines in your shop are operating simultaneously. If necessary, place a cover on the outlet to prevent accidental disconnection.
9. Make all machine adjustments or maintenance with the machine unplugged from the power source.
10. Follow effective lockout procedures to reduce the risk from high voltage wires and components

and prevent intentional bypassing of safety controls and accidental operation.

11. Don't use in a damp or wet location, or expose to rain, fog or snow.
12. Keep the electrical cord away from sharp edges, heat or moving parts, and do not store materials on top of it. Position the cord so it will not become a tripping hazard.
13. While the use of an extension cord is discouraged, it is recognized that the wiring layout of the shop may not allow the placement of the CNC Machine directly next to an outlet. If it is necessary to use an extension cord, make sure the extension cord is in good condition, uncoiled fully, heavy enough to carry the required current, and installed to prevent a tripping hazard. An undersized cord will cause a reduction in voltage resulting in loss of power and overheating which may result in fire or electrical shock.

3.5 Other Safeguards

14. Remove and store adjusting keys and wrenches when finished with them and before turning on the power. If necessary, for visibility, apply safety markings to adjusting wrenches and keys.
15. Install safety guards consistent with general shop safety practices. Keep safety guards in place at all times when the machine is in use. If removed for maintenance purposes, use extreme caution and replace the guards immediately after completion of maintenance.

16. Check damaged parts immediately. Before further use of the machine, a guard or other part that is damaged should be carefully checked to determine that it will operate properly and perform its intended function.
17. Keep visitors a safe distance from the work area. Keep children away.
18. Control liquids in your shop to limit the possibility of spillage that can damage the CNC which can cause damage or personal injury from electric shock or fire. Be careful with storage and use of cleaning fluids, finishes and solvents. Never use the table to apply or dry finishes.

3.6 Maintenance

19. During any inspection or maintenance activity always ensure the machine is powered off and disconnected from the power supply.
20. Establish a weekly and monthly maintenance checklist and follow it diligently.
21. Routine maintenance should include periodic checks for alignment of moving parts, looseness or binding of moving parts, worn or bare wires, breakage of parts, skewed mounting and any other conditions that may affect its operation or cause injury. Analyse breakage or damage to

determine the cause and take appropriate remedial action.

22. Do not operate the CNC Machine if a component of the control system is damaged. It should be properly repaired or replaced before use.
23. Follow instructions for lubricating and changing accessories.
24. Store maintenance tools and supplies nearby, consistent with shop maintenance practices and resources.
25. Only use identical replacement parts, use of any other parts may create a hazard or cause product damage
26. All repairs whether electrical or mechanical, should be done by a qualified person.
27. NEVER manually force movement of the machine by hand even when machine is off.

3.7 Operational Practice

28. Never leave the machine running unattended. Always be in close reach of the emergency stop button.
29. Turn the power off and do not leave the machine until it comes to a complete stop.
30. Always stay alert! Do not allow familiarity (gained from frequent use) to cause a careless mistake. ALWAYS REMEMBER a careless fraction of a second is sufficient to inflict serious injury.
31. Stay alert and exercise control. Watch what you are doing and use common sense. DO NOT operate tool when you are tired. DO NOT rush.
32. Avoid pinch points and entanglement hazards. Keep hands and clothing away from the ball screws, thrust bearings, gantry, guide rails and rotating cutting tool while in operation.

33. Use the right tool at the correct speed and feed rate. Do not force a tool or attachment to do a job for which it was not designed. The right tool will do the job better and more safely.

34. Do not touch a cutting tool immediately after use. It will be hot and may cause skin burns. Exercise caution when handling the collet and spindle nut if the cutting tool is hot. Keep a heavy glove or oven mitt on hand for the purpose.
35. Do not lay a hot cutting tool on its side. Create a rack for cooling off hot cutting tools.
36. Use recommended accessories; improper accessories may be hazardous.
37. Do not use dull, gummy, or damaged cutting tools. Keep bits and other cutting tools clean and sharp for best and safest performance.

38. Turn off the machine before cleaning. Use a vacuum, brush or compressed air to remove chips or debris. Do not use hands.
39. Do not climb or stand on the machine. Serious personal injury and costly damage could occur if the machine tips over or the gantry is dislodged.
40. Remove loose items and unnecessary work pieces from the table before starting the machine.
41. Plan tool paths to make multiple passes rather than to take off a large amount of material at one time. This will reduce mechanical stress and heat on cutting tools.
42. Always secure a work piece to the spoil board using clamps, vacuum, or double-sided tape. If the work piece is mounted in a jig ensure that the fixture is securely held to the table. Never hold a work piece down by hand while operating.
43. Inspect the material of your work piece to detect any defects that may result in ejection of large pieces of scrap.
44. Make sure the work piece is free from nails, hardware, or other foreign objects.
45. After installing a cutting tool, make sure the collet is securely tightened. An unsecured cutting tool may fly loose from the collet and cause injury. Be sure that the adjusting wrenches have been removed and are secured before turning on the power.

3.8 Grounding Instructions:

This tool should be connected to a grounded metal permanent wiring system, or to a system having an equipment-grounding conductor. The CNC Machine control system assumes the ground pin on the AC controller box connection is connected to a grounded conductor and the control box is correctly plugged into the CNC machine table.

3.9 Storage

In the case where the machines are stored and not used for extended periods of time (greater than 3 days) the machine must be placed in an atmosphere free from moisture and from excessive changes in temperature. The slides (more details in section 10) must be kept clean and lubricated with light "3 in 1" oil. The ballscrews (more details in section 10) must be lubricated with a grease gun using the greasing nipple.

4 LIABILITY AND GUARANTEE

All statements in these operating instructions serve the certain and undisturbed business of the CNC Machine.

The operating instructions are an important component of the machine, its safe use, and the long term reliability of the CNC Machine. These instructions must be read and studied carefully by ANY operator of the machine system before use.

Improper handling of the machine can lead to serious injury to the operator(s) and severe damage to the machine. The machine manufacturer and provider are not liable (except in the case of due negligence) for damage and injury due to improper handling and use of the CNC machine.

We the manufacturers and appointed agents reserve the right to make variations to the frame and components in the event of future developments.

5 WARRANTY

The machine system is guaranteed with a 12 month manufacturers warranty from the date of delivery on site at the customer location. If you believe a part of the machine is defective under warranty, please contact the dealer you purchased from. If the machine was not purchased direct from i2R Australia or but an affiliated dealer, you must contact the dealer from whom you purchased the machine not i2R CNC Australia.

The warranty covers all hardware aspects of the machine within the 12 month warranty period ***provided the machine operation and use meets the following requirements:***

- The machine is used by *trained operators only*. The use of the machine by untrained persons will **immediately void** the warranty as well as putting the machine and the untrained operator in serious risk of injury or damage.
- The machine is used within the application scope recommended/discussed. This will have been discussed with the customer during the enquiry. Machining of hard materials e.g. stainless steel or other hard metals will put the machine under excessive stress and load and will void the warranty. Section 6 outlines the scope of work that can be carried out on the CNC machine.
- The machine is maintained in the same state and configuration as when the machine was delivered / installed. Modifications effecting the safe operation of the machine will void the warranty.
- The environment where the machine is kept does not expose the machine to excessive moisture or thermal changes.
- The AC electrical supply to the machine is reliable and free from excessive noise – if necessary, a dedicated MCB/RCD switch may be required.
- The AC electrical system has a reliable and professionally installed earth connection.
- I2R Australia must be notified in the case where a mist coolant system is to be run on the machine. Special precautions are needed, and the machine configuration will be different if mist coolant is to be used
- The machine is kept clean, free from build-up of materials on moving parts especially the linear slides, and generally in good condition. Excessive build-up of material around the spindle and the linear slide mechanisms of the CNC Machine frame can cause excessive loads in the drive system and damage to the machine drives.

- The maintenance of the ball screws is carried out periodically to ensure free unrestricted movement
- The linear slides are periodically cleaned, inspected and lubricated

- The cables to the machine, the power and communications are managed and protected. Damaged cables can cause short circuits and can damage the machine's sensitive electronics. Machine damage due to cable damage voids the warranty terms.
- Damage or failure of the machine, or part of the machine is reported to i2R Australia immediately upon discovery of a problem.
- The machine is not modified from the factory configuration. This includes; mechanical configuration, electrical configuration and software configuration

5.1 WARRANTY REGARDING THE CONTROL PC

In the case ***where i2R Australia provide a PC for the CNC Machine operation*** the Guarantee on the control PC will be in-line with the CNC Machine system.

In the case ***where the customers control PC is used*** i2R Australia can take no responsibility for loss of functionality caused by subsequent failure of the customer's PC. In the case were i2R CNC Australia installs the machine on site, i2R CNC Australia will test the control PC to ensure it is suitable for controlling the CNC Machine system.

We can provide remote login and reconfiguration if the control PC goes down or has an issue. We recommend you purchase a fresh, new control PC prior to the machine arrival and use this as the dedicated machine for the CNC system. A moderate spec Windows PC is sufficient for controlling the machine. Any extra/additional CAD packages you wish to run may require you to upgrade to a higher spec windows system.

A Laptop is fit for purpose and often the preferred unit to control the system – you as a machine operator need to consider how to manage the use of the control PC as it is important not to let the swarf and chips generated during the cutting process interfere with the PC's – for example - the keyboard etc.

6 Recommended Use

6.1 Typical Application Materials

The CNC Machine systems are suitable for milling, boring, engraving the following materials:

- Wood.
- Plastics.
- Aluminium and other NE metals (under certain very specific conditions)
- Fibre composites e.g. GFK or CFK (with specific attention to the dust generated from cutting)
- Foam (with specific attention to the dust generated from cutting)

The above list of materials is appropriate if the following conditions are met

- Appropriate tooling is used for the job requirement
- The tools have been installed correctly in the collet mounting system in the provided spindle. Never use a tool outside of its operating parameters.
- The spindle speed and feed rate are appropriate for the job
- In the case where you are machining aluminium some extra care is required to maintain good chip development. If necessary, a cold airline will improve chip generation.
- Coolant may be required in some cases where aluminium is the main application. In the case where coolant is used the operator must pay close attention to manage the coolant run off.
- In all cases – especially where foam or composites are machined – provision of adequate extraction systems to eliminate hazardous swarf, dust and particulates must be implemented. There must be systems in place to stop dust ingress into the machine's running gear and spindle bearings.

6.2 Special note on Application materials

If the CNC Machine system is used on materials outside of the scope of recommended use, then the warranty does not cover any issues caused by this incorrect machine usage.

7 SPECIAL NOTES ON CLAMPING

WARNING: Very important the work piece is clamped securely.

7.1 SHEET CLAMPING USING SUPPLIED CLAMPS WITH THE MACHINE

The T-slot bed provides several methods for clamping. The image below shows the typical method for clamping sheet work in small sizes to the T-slot table. The clamps can be used to clamp a variety of thicknesses up to 18mm MDF. There are other options available- please contact us for more information. It is very important the work is securely clamped as if the work hops during use and becomes loose – it is extremely dangerous.

Figure 1: Clamping methods – Sheet Clamping

7.2 CLAMP SETS FOR LARGER WORK PIECES

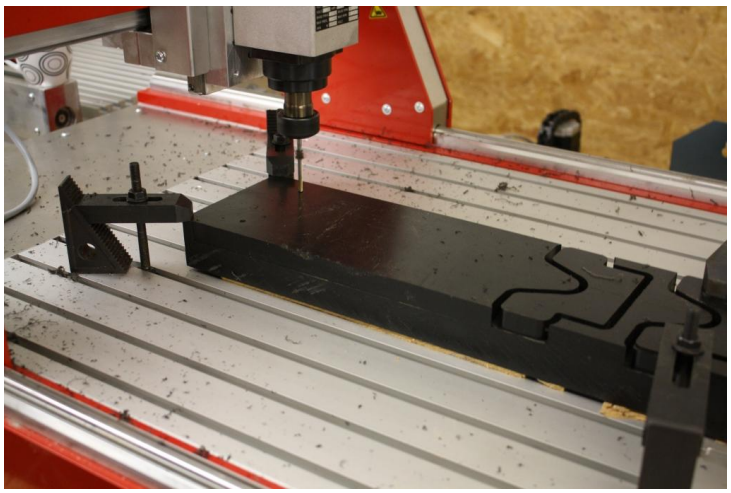


Figure 2: Clamping methods - Clamp Sets

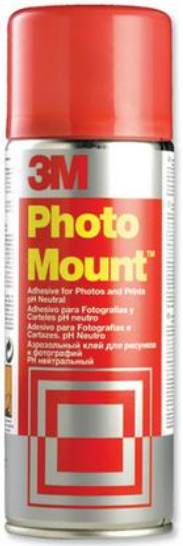
Another method of clamping is shown here in the image on the left using triangular brackets and fixing cantilever arms. These can work very effectively for larger pieces like the acetal part shown here in the image on the left. The advantage of this configuration is larger work pieces can be clamped securely. However you have to be careful as the larger the clamps you

use, the more you run the risk of crashing into the clamps during use. Always double check before you run your toolpath that the machine will not move itself to crash into the clamps.

7.3 SACRIFICIAL LAYERS

It's a good idea to always use a sacrificial layer under the work piece you are machining. This is especially important when you are milling all the way through the material. 3-4-6mm MDF often works well. It saves you blemishing the T-slot Aluminium table. Often if you use a thick layer you can use a drill and screws to fix your work. Crude but functional.

7.4 SPRAY ADHESIVE



Spray adhesive, especially 3M photo mount spray is a very useful additional method of clamping. For example a piece of MDF can be clamped to a sacrificial layer of MDF with just the spray adhesive. You have to play with trial and error to find the optimal machining conditions as the spray adhesive is not as strong as the other clamping methods – but can still be VERY effective. For example, when machining letters out of MDF or plywood the addition of spray adhesive as well as the clamps can help prevent the cutout shapes from "hopping" on the bed. For larger pieces this is especially helpful. The reason for this is that is makes sure that the work piece sits flush to the top surface on all areas. Sometimes when clamping larger sheets – the middle of the sheet can rise slightly when clamped at the edges. The spray adhesive does a great job of keeping everything flat and true. Light sanding afterwards gets rid of the adhesive. Trial and error is the best method of approach.

7.5 CLAMPING CONSIDERATIONS

It's very important that whatever clamping approach you use that you proceed with caution. This is especially important when you experiment with new methods. Work coming loose when you are machining, is extremely dangerous.

8 ELECTRICAL CONNECTIONS

All Electrical connections must be performed by a qualified electrical and follow any local codes and ordinances. Failure to comply may result in serious injury.

Electrical connections that are improperly installed or are outside operational specifications may cause machine damage and void any warranties that are in place.

Figure 3: Industrial Connections

All our CNC machines are rated for 220V +/-10% having an operational range of 200-240V unless otherwise stated. The machines come pre-wired with a 10 AMP Australian 3 pin plug for use on a circuit with a grounded outlet as pictured. It is recommended that these machines be connected to a dedicated 10-amp circuit.

If the machine must be reconnected for use with a different type of electric circuit, the connection must be done by qualified person(s) and must comply with all local codes and ordinances.

As part of initial setup of the CNC machine, the industrial connections from the control box both must be connected to back of machine table as per figure 3. Failure to do so may result in a machine not properly grounded.

8.1 GROUNDING INSTRUCTIONS:

CNC machines must be grounded. This grounding provides a path of least resistance for electrical current, which during a malfunction will reduce the risk of electrical shock.

All CNC machines are equipped with an electrical cord with grounding conductor and plug. The plug must be used with a matching outlet that is properly installed and grounded in accordance with local codes and ordinances.

These plugs must not be modified, if a matching outlet is need, one must be installed by a qualified electrician.

Improper installation may result in electrical shock.

If grounding instructions are not completely understood or if in doubt as to whether the machine is properly grounded, a qualified electrician should be consulted.

8.2 EXTENSION LEADS:

The use of extension cords should be discouraged. It is recommended to place the machines as near to the power source as possible.

If an extension cord is necessary, make sure any cord used is in good conditions. Worn or damaged cords should be replaced immediately. 'Daisy-chaining' extension cords should not done.

When using an extension cord, be sure to use one that is heavy enough to carry the required current and use only 3-wire extension cords that feature the correct 3-prong grounding plugs and 3-pole receptacles.

An undersized cord will cause a drop-in line voltage resulting in loss of power, overheating and runs the risk of fire.

9 POWER MANAGEMENT OF LAPTOP/PC

It is very important that you disable all screen savers and any temporary shutdown present on your window system. Automatic restarts, updates, screen savers, power save modes etc can all cause the USB/Ethernet communication to be lost between your machine and control PC. This usually results in lost work.

As the figure show – disable the **Display Dim**, **Turn off Display** and **Sleep functions** when the laptop is plugged in.

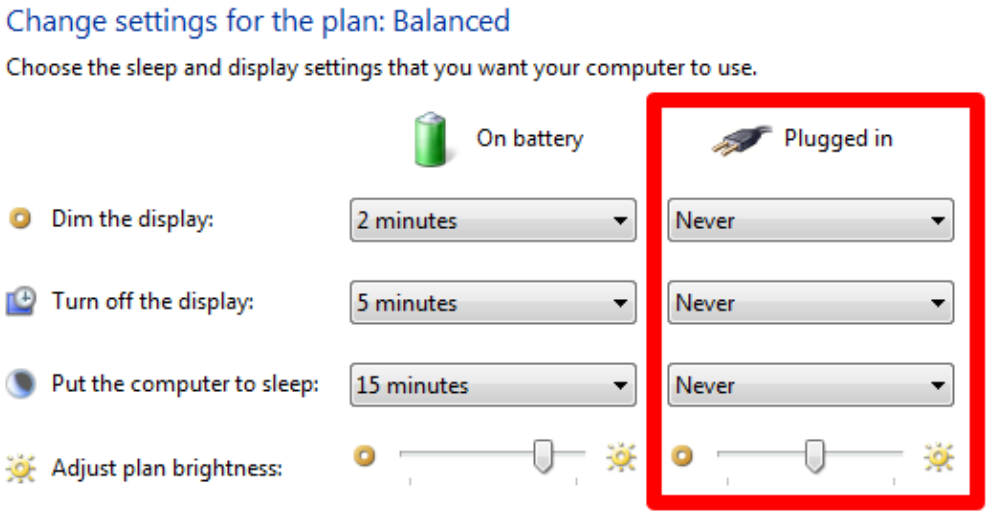


Figure 4: Power Management of Laptop/PC

10 MAINTENANCE

10.1 GENERAL MAINTENANCE

Best practice with machine tools is to keep the machines in a dry, clean, free from dirt and debris environment.

To ensure proper machine operation, it is regularly recommended that the prismatic guides and rolled ball-screws on all axes are wiped clean with a rag to show a little shine by cleaning away the shop grime. You should then lubricate with a light machine oil lubricant (such as 3-IN-ONE oil). Figure 6 shows the X axis rails and ballscrew. The Z and Y axes have the same components. Failure to do this can result in poor performance and in extreme cases, cause surface corrosion or pitting of the hardened steel linear rails. We do not recommend you try to remove surface corrosion if it gets to this stage as this may further reduce the performance of the linear guides.

Figure 5: Recommended lubricating oil

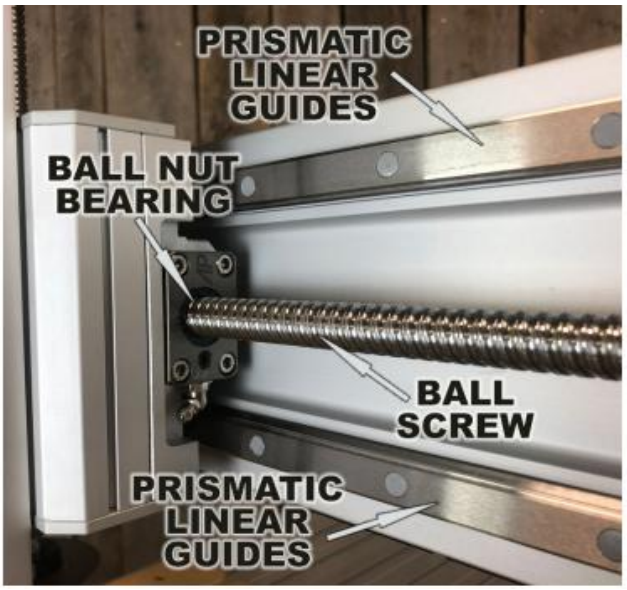


Figure 6: Weekly (best case daily) cleaning and application of light oil externally to screws and rails with cloth and light oil.

The ballscrews also have grease nipples on them. Every 1000 hours or every year these can be packed with suitable grease for optimal operation – See Figure 7 for the recommended grease specification.

Figure 7: Recommended grease for the ballscrews when greasing via the grease nipple

10.2 Routine Checks: (Performed Daily with machine use)

1. Wipe the ball-screws and prismatic guides clean and dust free.
2. Apply light machine oil to ball-screws and prismatic guides using a clean lint free cloth. Move the machine through travel limits to properly disperse lubricant using UCCNC.
3. Check cutting tool edges for chips and/or dullness.
4. Generally, inspect the machine for any damage.
5. Verify that the Spindle and Collet Nut threads are clean of debris and undamaged.
6. Check coolant levels in reservoir.
7. Verify that the dust extractor is free of blockages.
8. Check that all electrical connectors are fitted correctly and are not loose.

10.3 Collet Maintenance

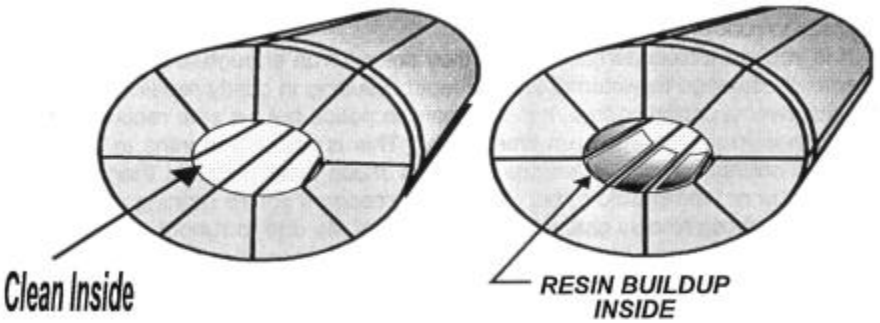
Heat is the biggest enemy of a cutting tool and the initial transfer of heat is from the tool to the collet. Collets are manufactured from spring steel and over a period of time, heat and usage causes them to lose elasticity. This hardening process makes tightening of the collet more difficult thus causing uneven gripping and ultimately tool runout.

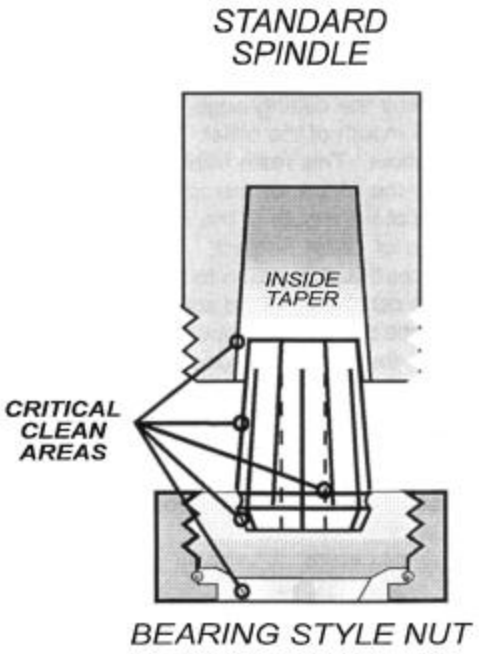
It is important to understand when hardened collets are not replaced; over tightening will eventually damage the internal spindle taper resulting in costly repairs. This process occurs gradually over a period of time and is difficult to diagnose. A practical recommendation for collet life is in the 400-600 run time hours. This is about 3 months in a normal two-shift operation. If collets are not changed, they will eventually become brittle enough to crack or break and permanently damage the spindle. Preventative maintenance is much cheaper than this costly alternative. Timely collet replacement is important, but cleaning the collet, along with the collet nut, toolholder taper, and inside spindle taper each time the tools are changed is equally important.

As material is routed, whether it be wood, plastic, aluminium, or man-made board, the chips carry many resins migrating up the slits in the collet and depositing onto the inside of the collet ears (usually nearest the mouth of the collet). The resin acts as pressure points gripping the tool tighter at the mouth of the collet. These pressure points often distort the grip on the tool creating runout. This resin heats up as the tool does and actually transfers onto the shank of the tool almost adhering the tool into the collet. Many times, the tell-tale sign of this transfer is brown marks at the mouth of the collet contact on the shank. These marks are a strong signal of collet neglect and the necessity to institute a collet maintenance procedure.

To prevent this problem, the resin must be removed from all surfaces using a non-abrasive brush. Make sure that all surfaces including outside and inside collet and inside spindle taper are thoroughly clean and dry before reassembling.

Also, the collet nut should be cleaned of resin and chip buildup and regularly replaced to ensure the integrity of the whole collet system. It is important to point out that simply blowing out the collets or soaking them overnight in a thinner does not rid collets of resin buildup. In fact, the latter procedure can prove to be hazardous. Do not use a petroleum-based lubricant for cleaning, as it will only act as a magnet for all the dirt and dust by the residue it leaves behind.


Clean Inside
RESIN BUILDUP
INSIDE


STANDARD
SPINDLE
INSIDE
TAPER
CRITICAL
CLEAN
AREAS
BEARING STYLE NUT

10.4 Air Cooled Spindles ONLY

If your CNC routers come with an air-cooled spindle, keep the cooling fan free from any blockages so that nothing is disrupting airflow. Be sure to wipe down the area around the fan on the top of the spindle encasement to remove any dust after every use. You can also use a vacuum or air compressor to remove dust from the local area.

10.5 Water Cooled Spindle ONLY - Cooling System Flush/Refill:

On CNC routers with the liquid cooled spindle, every 4-6 months the electro spindle cooling system should be flushed and refilled to guarantee that the coolant is fresh and able to perform optimally.

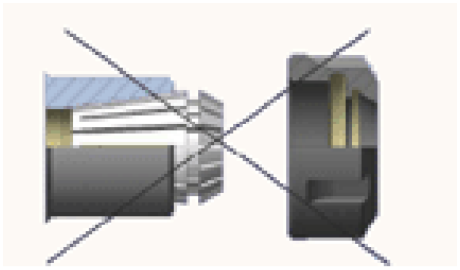
Refer to the document titled: i2R-A Cooling System Flush-Refill

Figure 8: Cooling system is under cover on back of bridge

11 How to use Collet Systems

All our HF (High Frequency) spindles (MTC/QTC/ATC) use collets to hold the cutting tool.

- It is very important that you use the correct collet size for the diameter of the shank diameter of the tool.
- It is very important that you use the correct collet family to suit the spindle or toolholder on QTC/ATC spindles. For example, ER16, ER20, ER25 or ER32
- To ensure you spindle and cutting tools live a long life, refer to section 1 for suggestions on collet maintenance.
- It is imperative you look after your collets and understand that they should be replaced on a regular basis.
- The tool is only inserted into the collet after the collet is seated correctly into the collet nut.



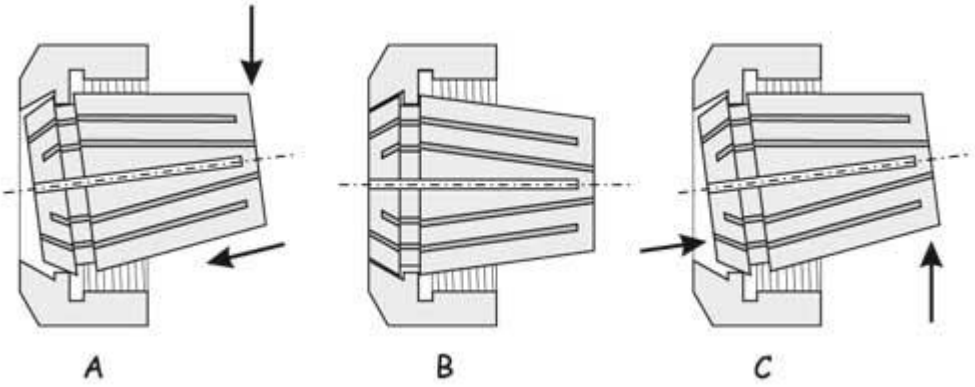
- Never use a tool beyond its recommended RPM.
- Always be prepared for the unexpected.

Figure 9: HF ER20 Collets

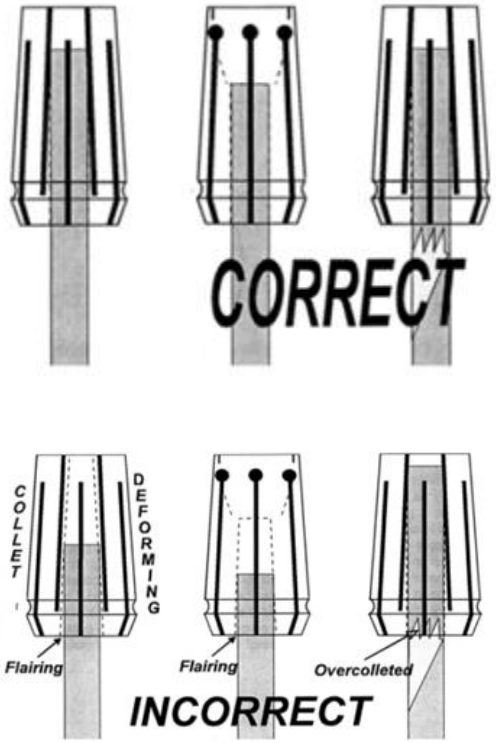
11.1 Mounting a Collet and Tool

1) Choose the correct collet for your tool. Always stick to the cutter spec. If using a 3.175mm (1/8th inch) cutter, then use a 1/8th inch collet. Using the correct collet for your cutter will make tool mounting easier and safer.
2) Make sure the collet and collet nut are clean and debris free
3) Insert the collet into the collet nut until the collet seats. To do this, insert it on an angle, turn slightly and push it into the nut until it clicks into place on the flange. You should hear a "click".

NOTE: if you mount a collet incorrectly into the spindle it will damage the collet and worse the spindle.



4) Mount the collet nut (now with seated collet) into the spindle head fixture and loosely tighten by hand only – just so the collet is on the threads of the spindle. If using a QTC/ATC toolholder the collet gets mounted into the toolholder not the spindle.
5) Insert the tool you wish to use. The tool is only inserted into the collet after the collet is seated correctly into the collet nut. (NOTE: you should have pre- selected your collet size for your tool choice)
6) Mount the tool so that you have enough tool stick out to carry out your machining operation. Be careful not under collet causing flairing or over collet resulting in deforming the collet as it tries to squeeze onto the flutes of the cutting tool.



7) The collet nut can now be used to tighten the collet into the spindle head/toolholder securing the cutter.

   a) In MTC spindles, the HF spindle needs a special collet spanner to hold the shaft of the spindle from turning while you tighten the collet nut with a separate spanner
   b) For QTC or ATC spindle refer to section **Error! Reference source not found.** for tightening the collet nut

   NOTE: in both cases for MTC/QTC/ATC you need only use the strength of your wrists to tighten the collet nut and care should be taken not to overtighten by using all your upper body strength

Be careful as you do this as you can slip and damage the cutter or cut/injure yourself. You will be exerting a tightening force with the locking spanner around and near a sharp cutter. So be careful!

8) Check the cutter is seated in the spindle and collet correctly by eye after you have installed the cutter and tightened the collet.
9) Take extra care when loading a tool and ensure the tool is mounted correctly prior to running a tool path and turning on the spindle

11.2 Removing a Collet and Tool

10) To remove the cutter, slacken and undo the nut until resistance is felt. Most spindles have a double locking mechanism on the spindle thread. Then, using a collet wrench, further undo the nut until the collet is released from the chuck body. Removal of the collet from the nut is the reverse of the mounting procedure.

12 THE BASICS OF CUTTING TOOLS

We only recommend using Solid Carbide or Carbide Tipped bits in our CNC machines due to the high RPM and material removal rates our CNC machines can reach. Please ensure the bits you use are rated to the RPM available from the spindle. Not all endmills or handheld router bits are suitable for CNC router. The advantage of carbide over other alternatives is long tool life thanks to its hardness with well-matched rigidity and finish. Some basic terms to start off are outlined below:

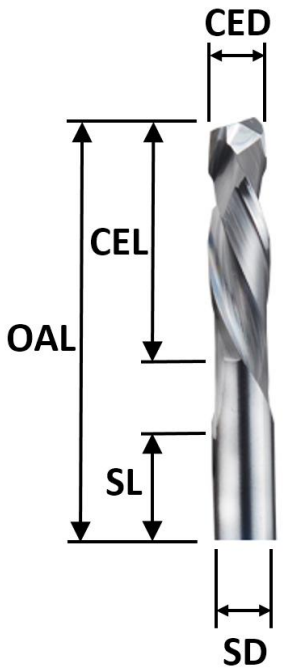


Figure 10: Basic cutting tool Terms

Common dimensions of the tools you need to be aware of are:

CED – Cutting Edge Diameter

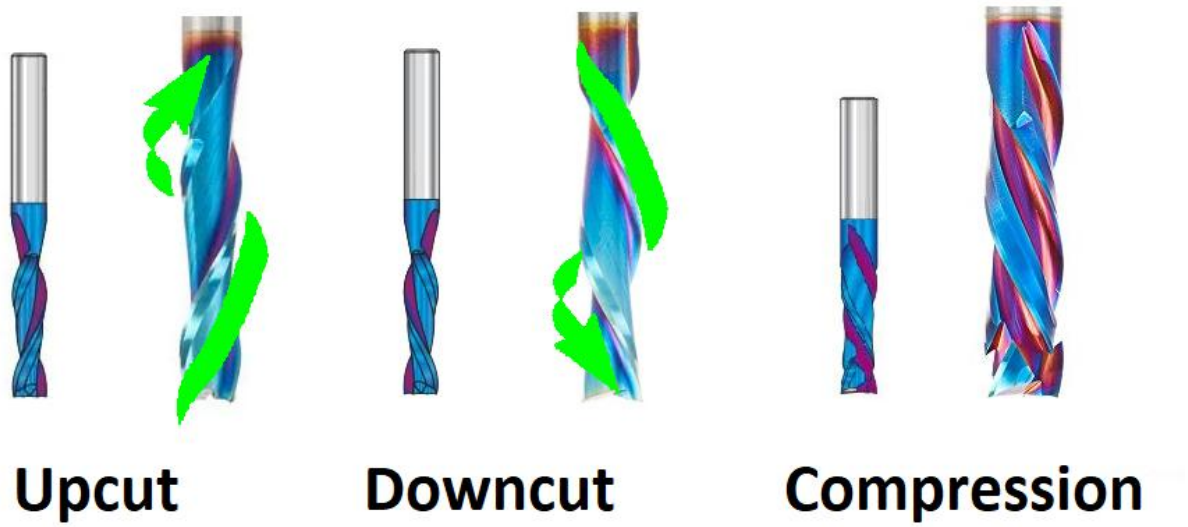
CEL – Cutting Edge Length

OAL – Over All Length

SL – Shank Length

SD – Shank Diameter

In choosing a cutting tool for any application, always select one with the shortest cutting edges and the shortest overall length that will reach the required cut depth. Excessive length intensifies deflection and vibration, which degrade cut quality and lead to tool breakage



The spiral edges on your cutting tools are known as flutes. The sole purpose of the flutes are to clear the cut chips while minimising friction on the workpiece. Nearly all tools are design for clockwise rotation and your spindle is set to spin in a clockwise direction only. Depending on the direction of spiral of the flute, it will determine in which direction your chips will go. If the spiral of the flute is spiralling up when the tool spins clockwise, this is known as Upcut. When the spiral of the flute is spiralling down when the tool spins clockwise, this is known as Downcut. It is worth noting that this spiral rotation will exert a force on your workpiece.

In choosing a cutting tool for any application, always select one with the shortest cutting edges and the shortest overall length that will reach the required cut depth. Excessive length intensifies deflection and vibration, which degrade cut quality and lead to tool breakage

12.1 Upcut:

The Up-Cut Router Bit yields an especially clean & accurate cut, while effectively clearing chips from the cut. The "up-cut" shears from the bottom up, pulling chips from the bottom up, thus allowing deeper penetration with less stress on the tool.

The advantage of an Upcut bit is it will eject and force all the chips up and out of the cut. Making them very efficient at clearing chips. There are two major drawbacks of this to consider especially when working with timber. It had the tendency to cause tear-out of the material at the surface on the top edge and also wants to lift the material off the bed

Applications:

- Cutting inlay pockets and shaping components

- Shaping / routing hardwoods and softwoods
- General purpose soft media machining

Excellent for Cutting:

- Aluminium
- Wood

12.2 DOWNCUT:

The 'Down-Cut' cuts from the surface down leaving a smooth edge at the surface. Due to the downward flute geometry this router bit provides better work hold down, reduces material tear out on the surface and can reduce vibration when cutting thinner sheets. Down-Cut router bits are excellent for creating grooves, super effective on surface operations and dado cuts in plywood and composite materials.

The advantage of the Downcut bit, since the spiral of the flute is down, is that the material is forced back onto the bed and leaves a much cleaner top edge. The disadvantage is the chips are being ejected in a downward direct back into the cut reducing dramatically the chip clearance capabilities of the tool. There is also a tendency to tear-out the bottom edge.

For best results you can consider using a downcut tool for the first initial cut deep enough to cut into the surface of the material but shallow enough to still promote chip ejection. Then you could change to an Upcut bit to finish cutting through the material with a more aggressive pass depth to save time and noting the upcut will produce a better finish on the bottom edge. Do not forget the upcut does tend to lift your material so ensure its well clamped.

Applications:

- Cutting inlay pockets and shaping components
- Shaping / routing hardwoods and softwoods
- General purpose soft media machining

Excellent for Cutting:

- Laminate
- Melamine
- Melamine Particle Board
- MDF
- Veneered Plywood
- Wood

12.3 Compression Cutter

Compression end mills combine up-cut and down-cut geometry which provide a superior cut at greater depth than traditional upcut or downcut router bit. These bits are designed to leave clean edges on the top and bottom faces of any material. The compression bit's unique design pulls chips upward at the bottom of the material and downward at the top face, producing chip-free surfaces that are perfect for cabinet and furniture parts.

The bottom section of the compression cutter is ground like an upcut and then there is a turning point, usually at a height the same as diameter of the tool, where the cutter is ground like a downcut. This results when used correctly a clean top edge and clean bottom edge. For this to work you must ensure your first pass depth is greater than the changing point between upcut and downcut.

Applications:

- Profile cutting
- Cabinet making
- Shaping / routing hardwoods and softwoods

Excellent for Cutting:

- Laminate
- Melamine
- Melamine Particle Board
- MDF
- Veneered Plywood
- Wood

12.4 SINGLE OR OFLUTE

Single Flute or OFlute have the ability for large chip removal, with single flute, up-cut design. The large, polished flute enables a very good chip clearance, less chance for chip re-welding, a superior surface finish and a longer tool life.

With a high rake angle and relatively low helix, O-flute cutters manage to cut acrylic, polycarbonate, ABS, PVC and a host of other sheet polymers at very high chiploads without raising a burr or fracturing the edges of the kerf. Not only have we found these bits good for high precision cutting of all plastics, but they are very suited to cut Aluminium and Aluminium Composite Material also.

13 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 axes. 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.

13.1 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
- Available Hard drive space: minimum 16GB
- Ethernet port(CAT5 or higher)

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.

13.2 SOFTWARE REQUIREMENTS

The software targets the .NET framework 4.0 which 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.

13.3 LICENSING

The software requires a license key to fully function with the CNC machine. One license key per CNC machine is required but can be used with multiple control PCs. 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.

14 UCCNC INSTALLATION

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

The installer files (Figure 11) 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.

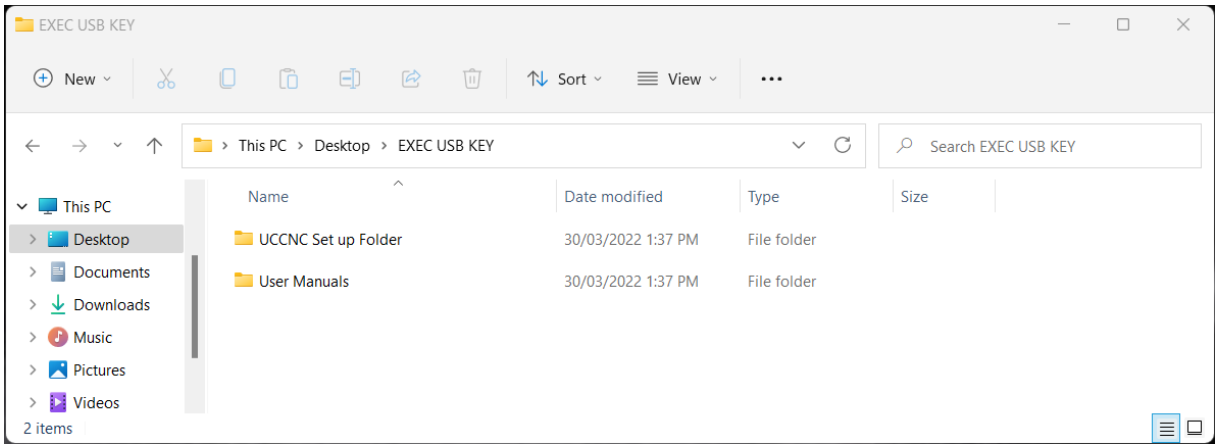


Figure 11: UCCNC Set-up files

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 12. If you have Window 7 you will find supplementary instructions in this folder also to guide you through.

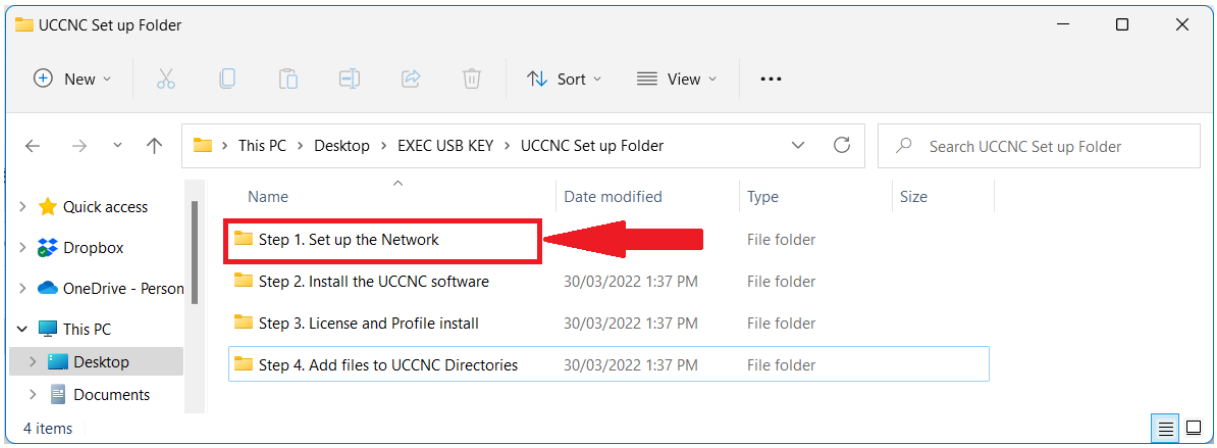


Figure 12: UCCNC Set-up, Step 1

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.

14.1 STEP 1 SET UP THE NETWORK

1- In Windows right click the Start menu (windows icon) and click on the search icon. See Figure 13

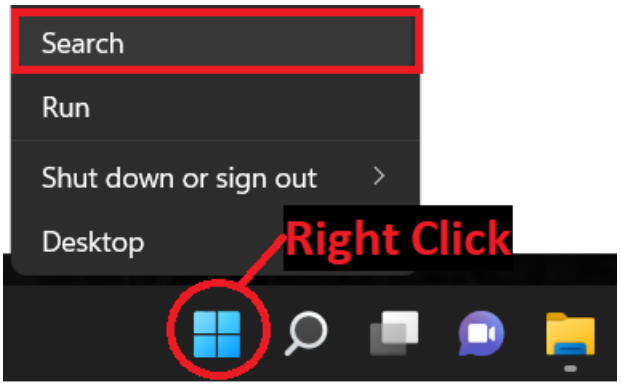


Figure 13: Windows search console

2- Type in the search "network connections" and either select when it is displayed, or press enter. See Figure 14.

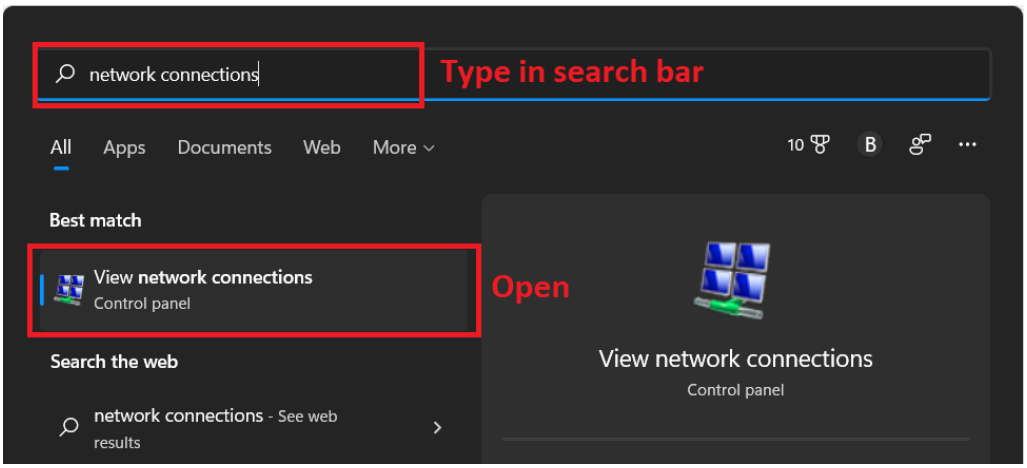


Figure 14: View Network Connections

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 15.

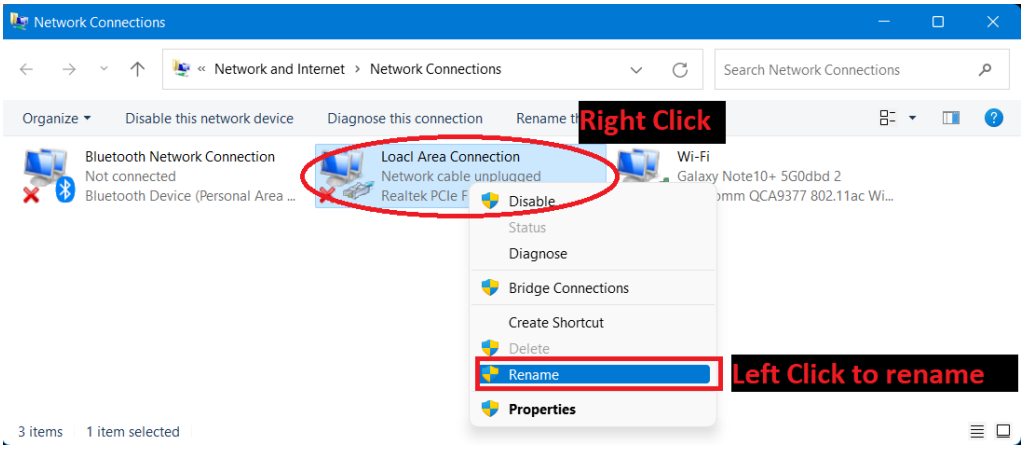


Figure 15: Rename this connection

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 "i2R" or "CNC Machine". The name does not affect the function of the machine. See Figure 16

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 16.

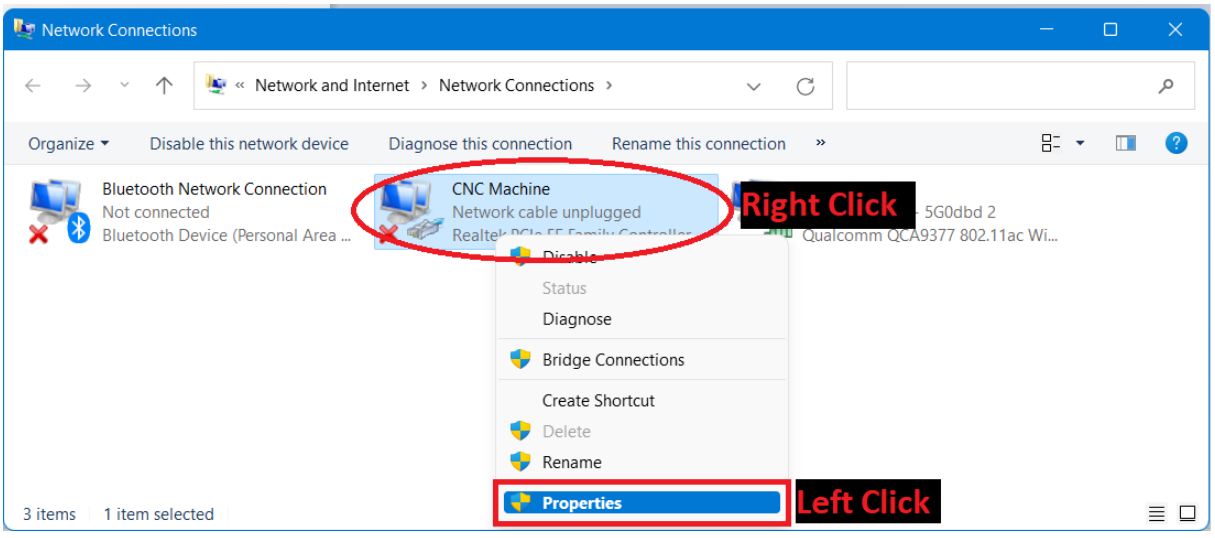


Figure 16: Network Properties

4- On the popup window find and select the 'Internet Protocol Version 4(TCP/IPv4)' row and click the Properties button. See Figure 17.

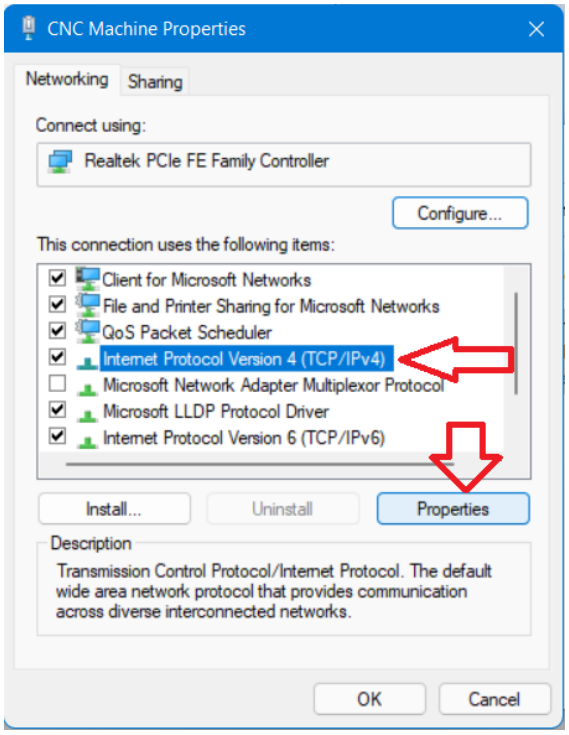


Figure 17: IPv4 Properties

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 18 shows the IPV4 network IP address that needs to be set.

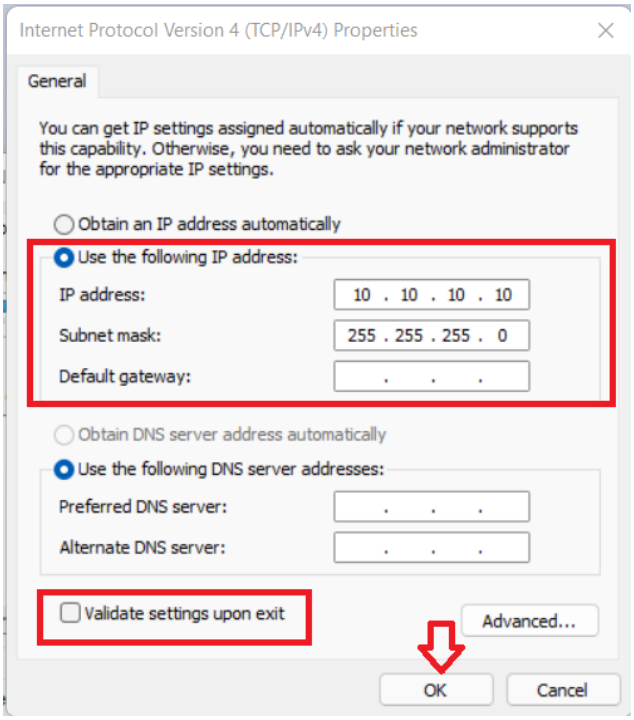


Figure 18: Network settings that need to be set prior to running the machine

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.

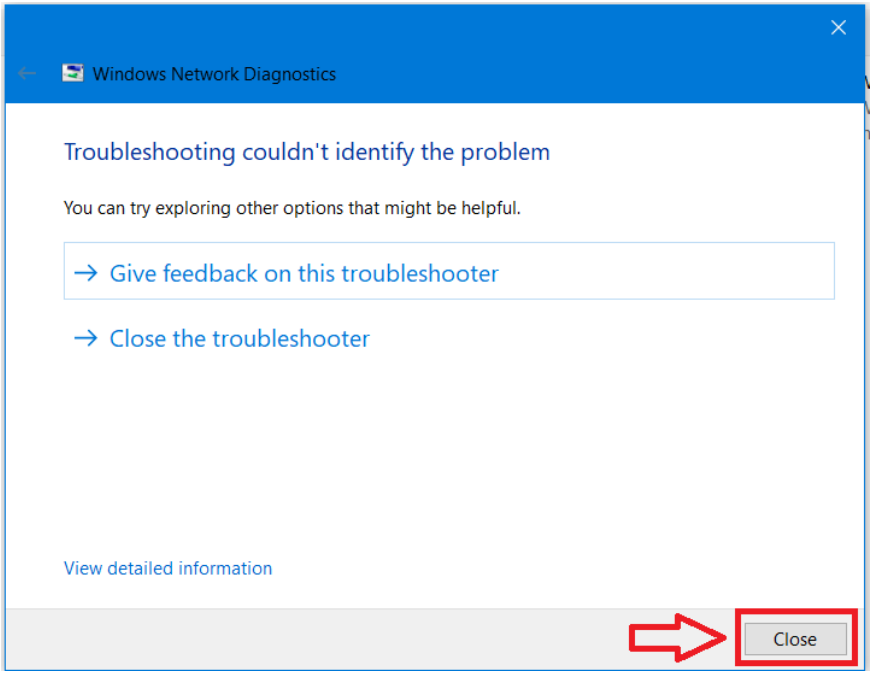


Figure 19: Window Network Diagnostic

14.2 STEP 2 INSTALL THE UCCNC SOFTWARE

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

Open 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

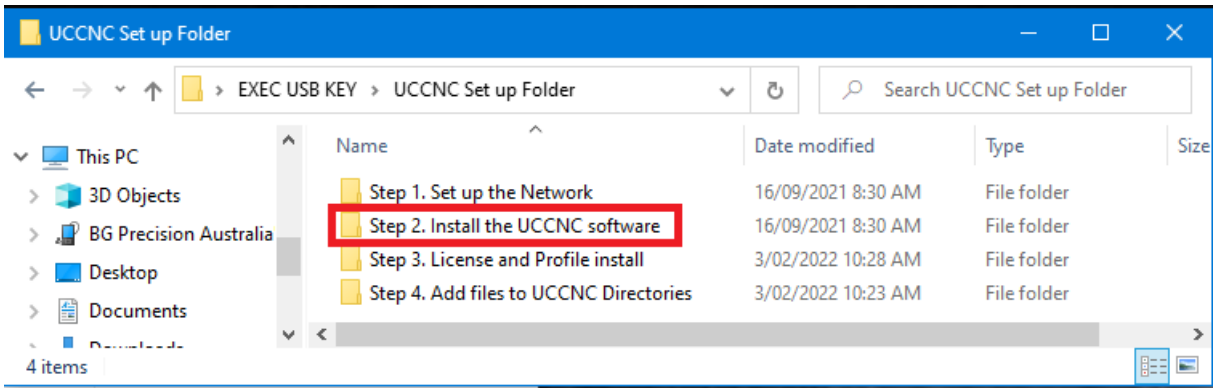


Figure 20: UCCNC Set-up, Step 2

2) Run the installer (note the version in this manual may differ from the version in your installer)

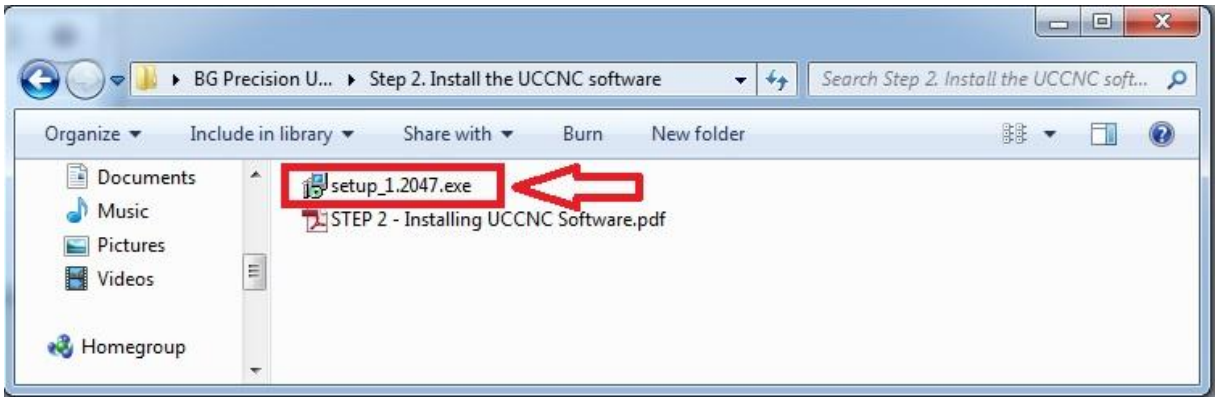


Figure 21: UCCNC Set-up exe

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

The latest version of the installer is always provided where applicable

14.3 STEP 3 UCCNC LICENSE AND PROFILE INSTALLER

The next step is to install the license for your UCCNC controller and your machine specific 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

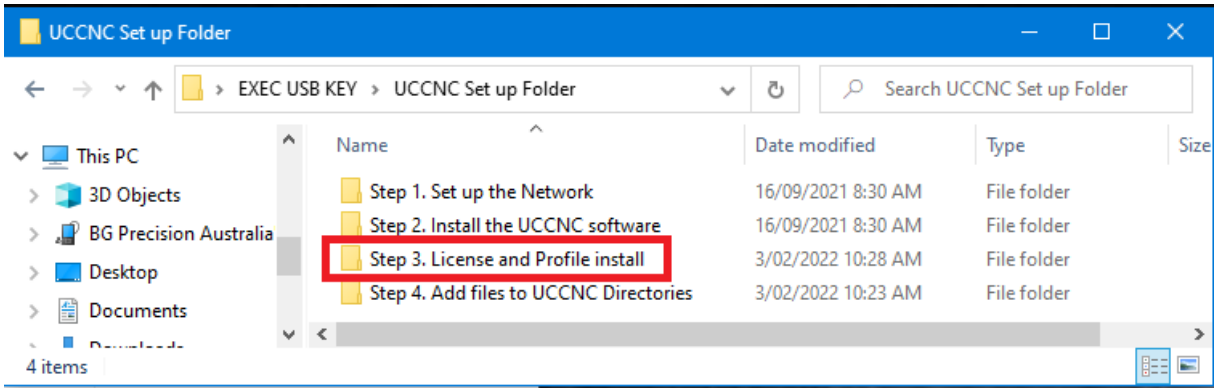


Figure 22: UCCNC Set-up, Step 3

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

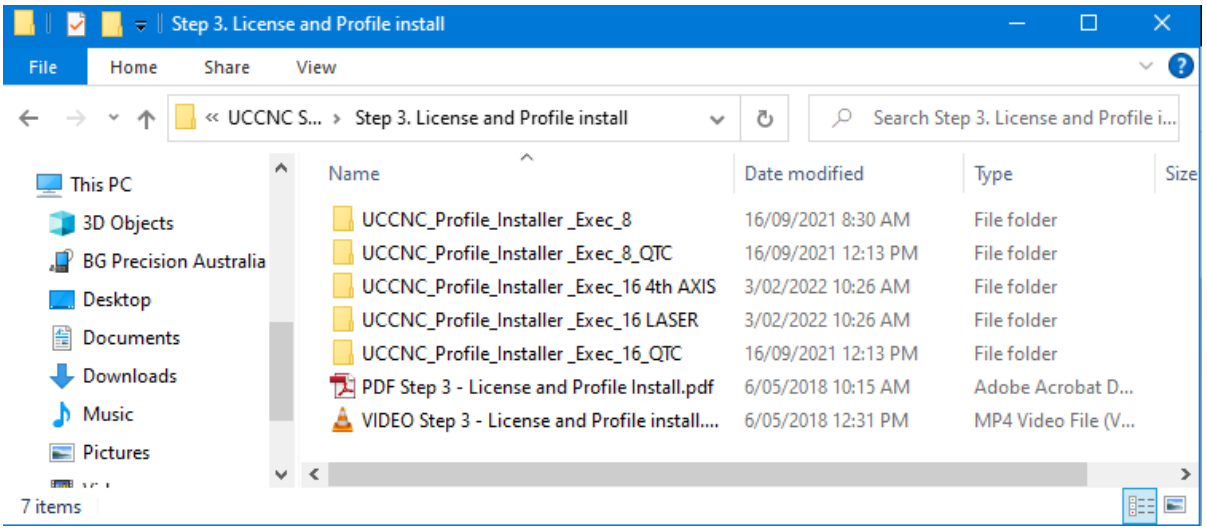


Figure 23: UCCNC profiles for selection

3) Once opened the profile installer folder, click the UCCNC_profile installer run me.

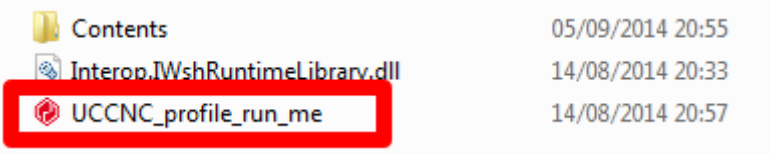


Figure 24: UCCNC Profile Run Me

4) When complete a pop up window will display the following:

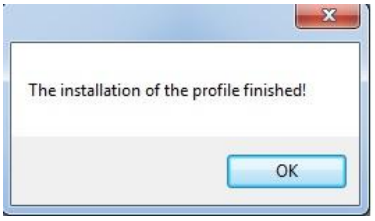


Figure 25: Installation of the profile finished!

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 must correspond with the machine you purchased and may differ from what you see below.

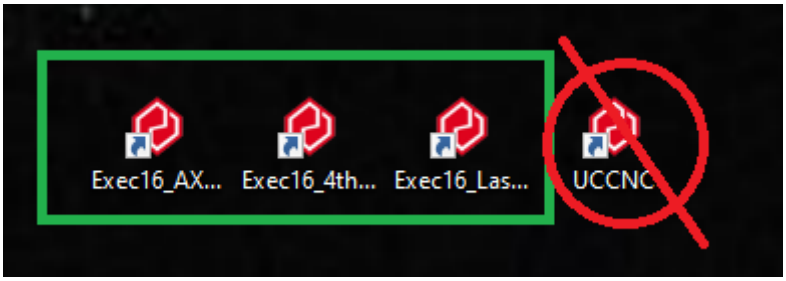


Figure 26: Desktop Shortcuts

15UCCNC 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.

15.1 UCCNC PAGE TABS

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

15.1.1 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.

15.1.2 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.

15.1.3 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.

15.1.4 Tools (Only Applicable to QTC and ATC Configurations)

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.

15.1.5 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.

15.1.6 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.

15.1.7 Help tab 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.

15.2 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 27)

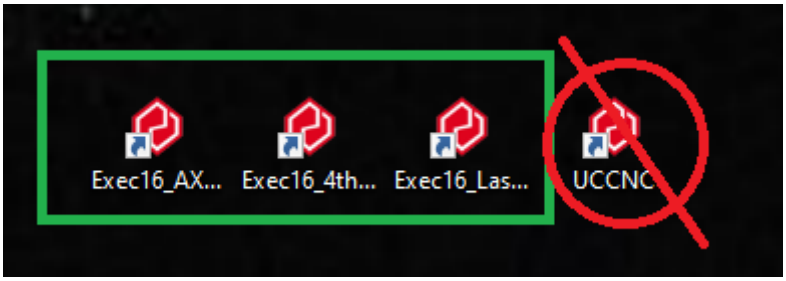


Figure 27: The UCCNC shortcut you should now have on the desktop

- The software should open and go straight to splash screen shown in Figure 28

Figure 28: 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.

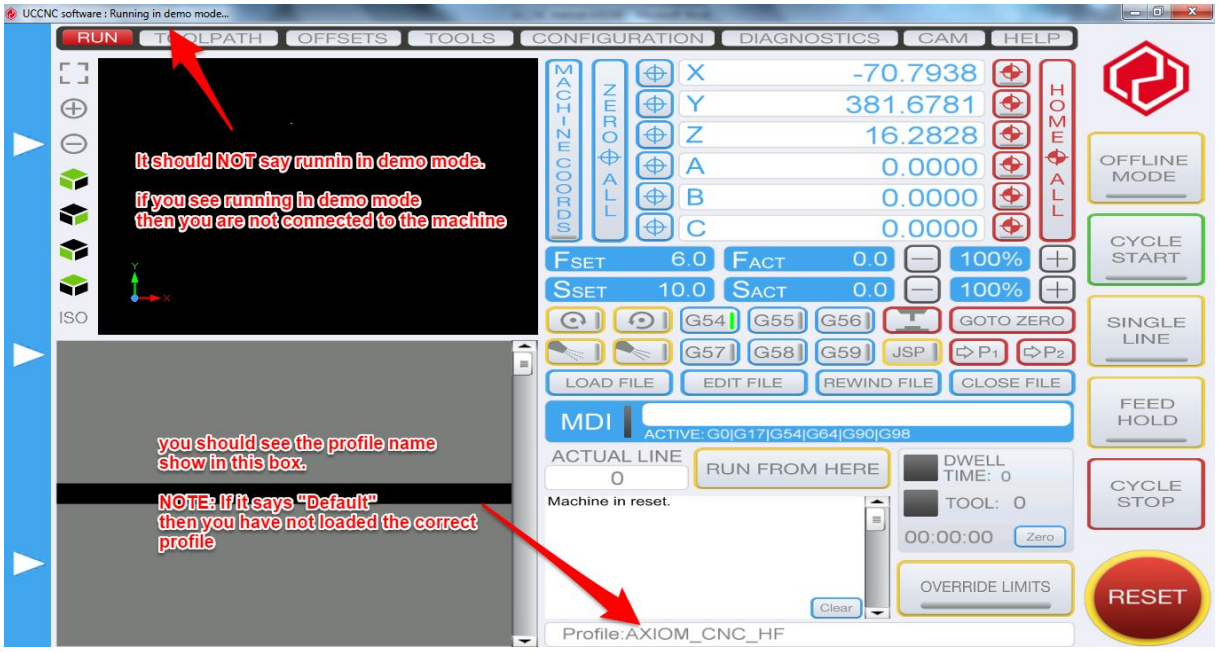


Figure 29: UCCNC main screen summary

- Figure 29 shows the main screen once you open the UCCNC controller.
    - NOTE – top left hand side. You should NOT see the text **"Running in Demo mode"** is should just say UCCNC software after the icon.
    - 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 30). 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.

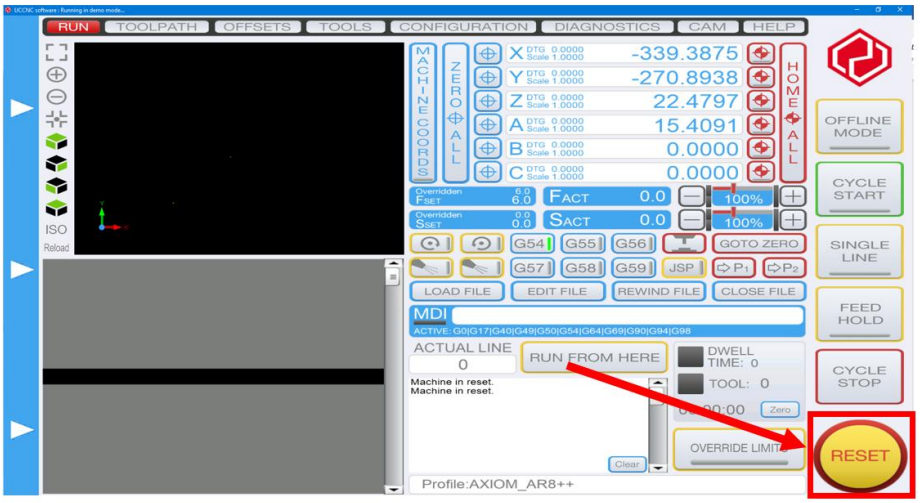


Figure 30: Reset button on UCCNC front panel

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

  - Holding the "**Shift**" key while pressing the directions of movement keys will move the machine in rapid mode.
  - Releasing the **"Shift**" key will result in the slow jog motion

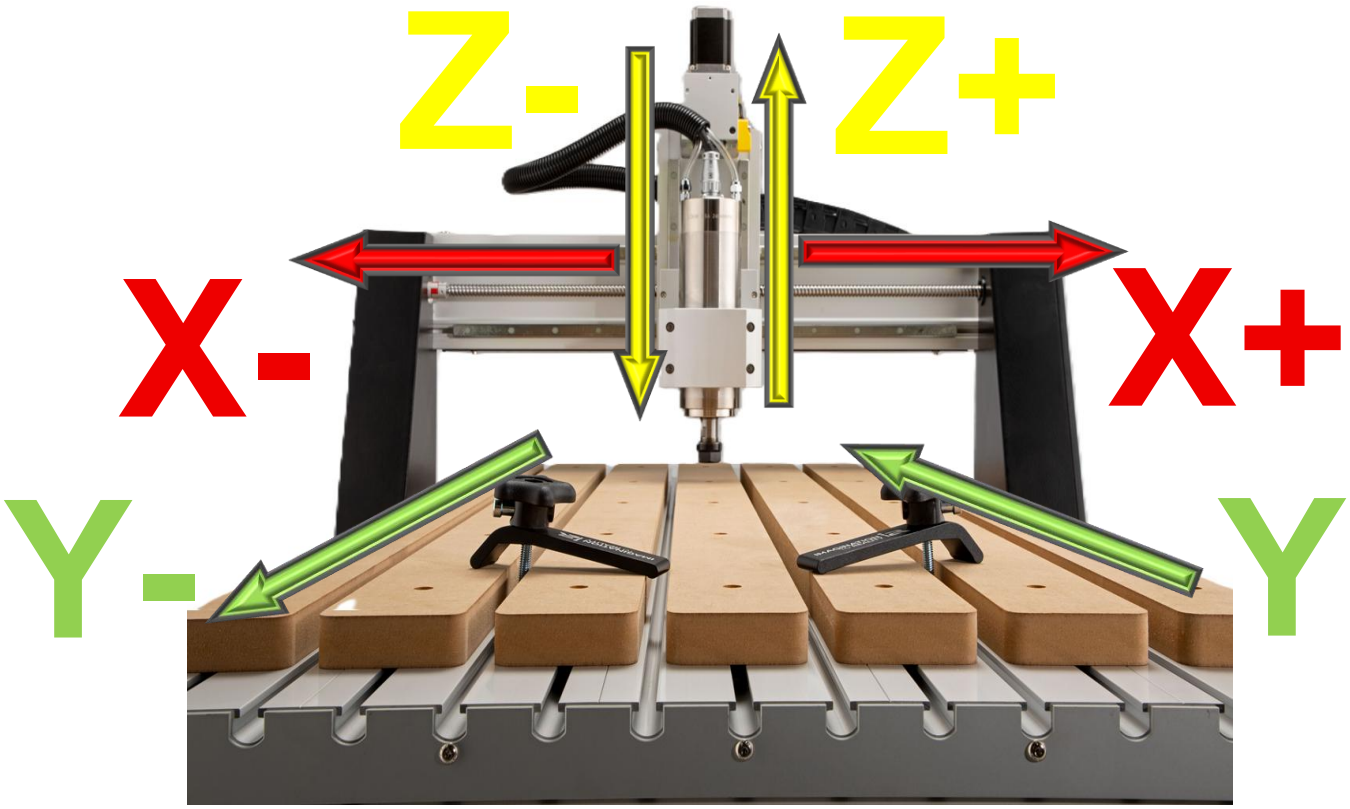


Figure 31: X - Y - Z Axis explained

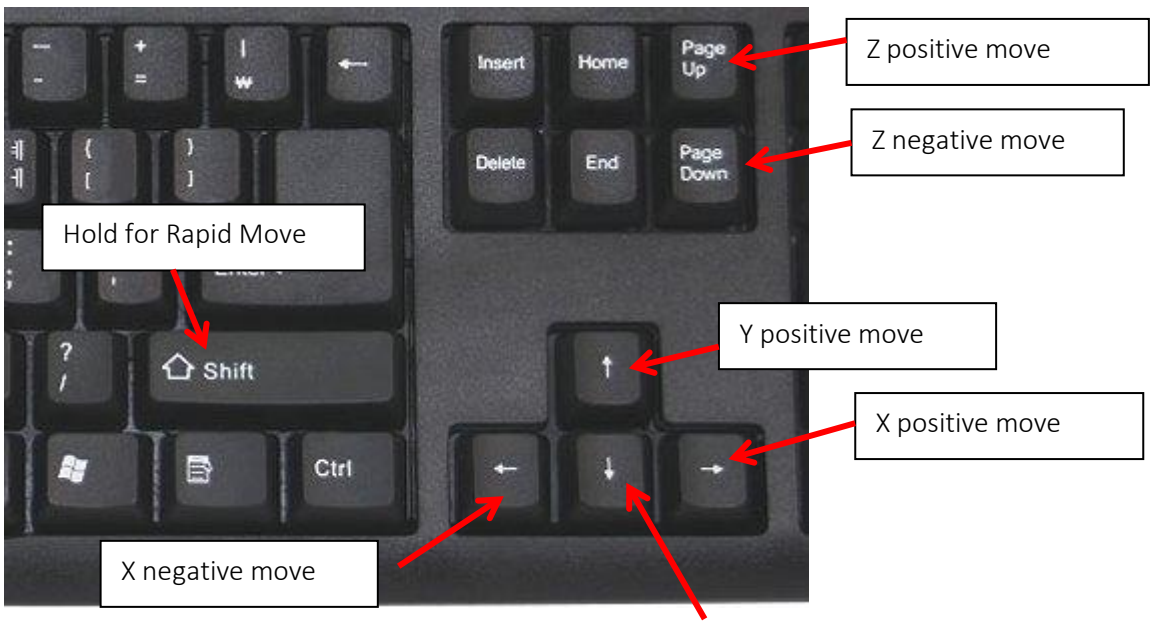


Figure 32: UCCNC Keyboard commands

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 33. For more details on the Jog Panel see section 15.4 Jogging the Machine

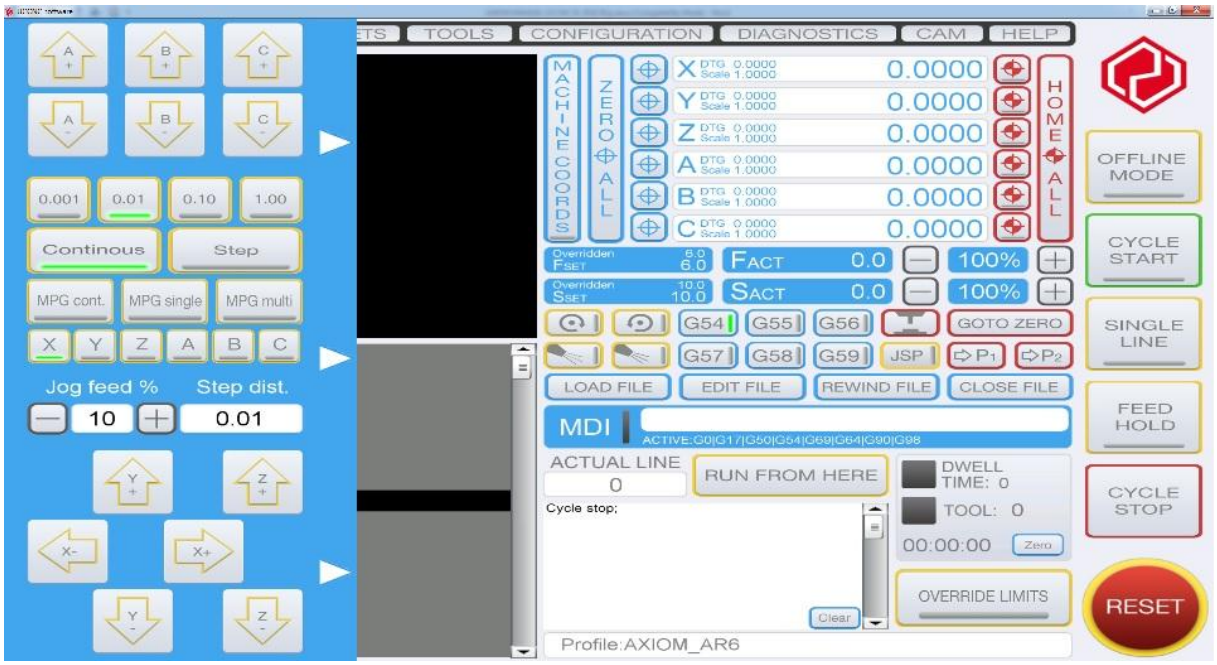


Figure 33: UCCNC screen interface movement controls

15.3 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 34. The machine will move each axis to a single extremity until it 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 15.6 Softlimits

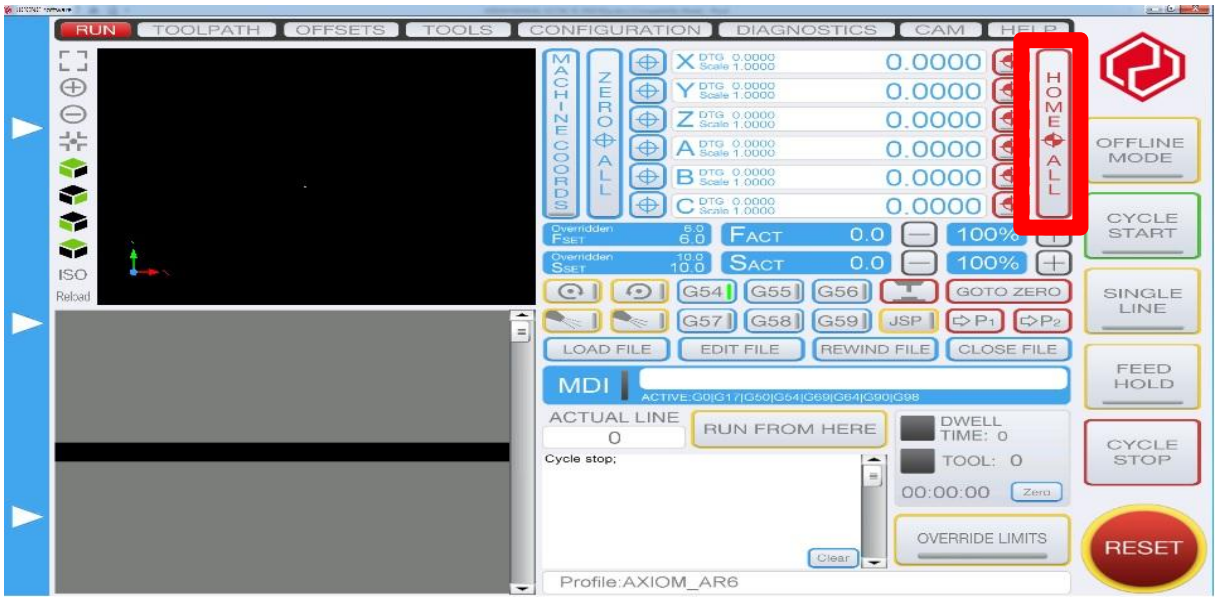


Figure 34: UCCNC Home All Command

15.4 JOGGING THE MACHINE

As well as using the keyboard commands as highlighted Section 15.2 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 axes can be jogged to the negative and 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 selectable.

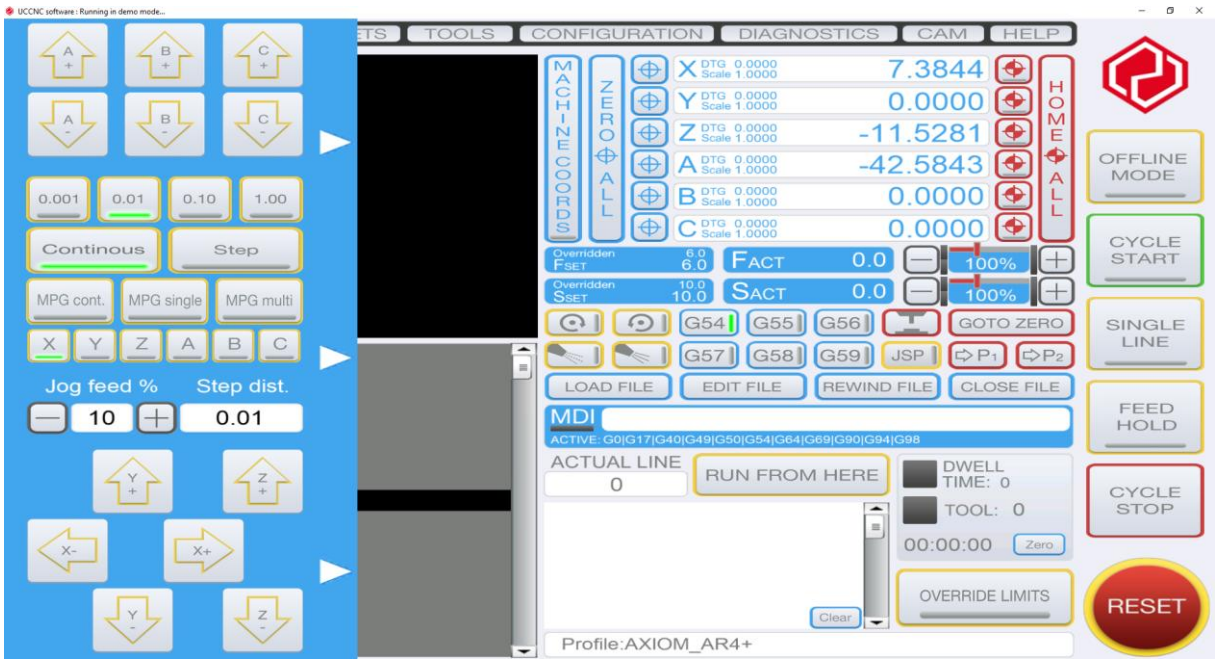


Figure 35: UCCNC Jog Controller

15.5 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 diagnostics 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 36 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.

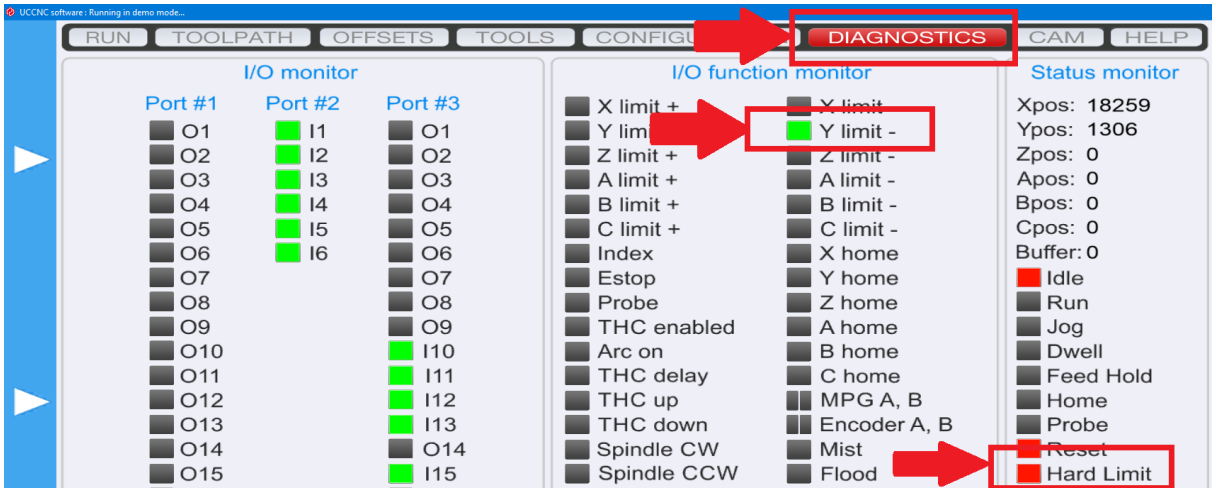


Figure 36: UCCNC Diagnostics

To do this, you must press the "OVERRIDE LIMITS" button to reset the machine. This can be done by using the "OVERRIDE LIMITS" command as you can see in Figure 37. 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.

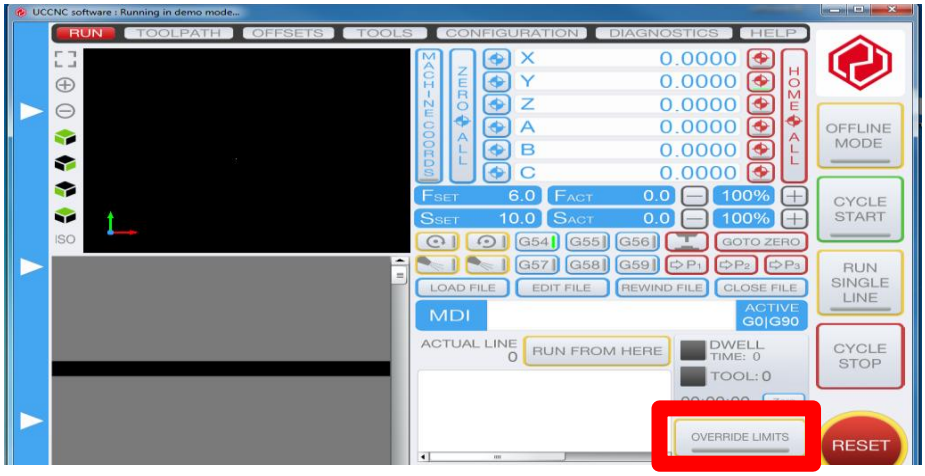


Figure 37: UCCNC Override Limits Command

15.6 SOFTLIMITS

The soft limits are designed to restrict the machines motion to the machines working envelope via the UCCNC software. These limits are imposed by the maximum workspace boundaries based on machine profile settings in UCCNC and the location of HOME. 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.

To get a better understanding between Home Position, Job position, Soft-limit area, Workpiece Area refer to Figure 38: Machine layout illustrated below.

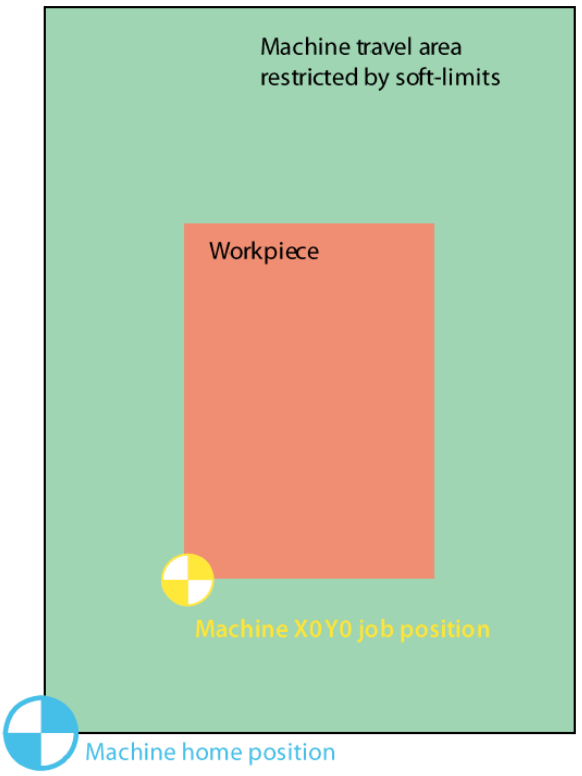


Figure 38: Machine layout illustrated

Note there are two co-ordinate systems as per Figure 38: Machine layout illustrated

1) The machine co-ordinate system (marked green) where all the background machine co-ordinates and soft limits are stored and located
2) The workpiece co-ordinate system (marked orange) where you spend most of your time. This is the system that allows you to setup and zero your 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 39 shows the DRO and the machine co-ordinate button

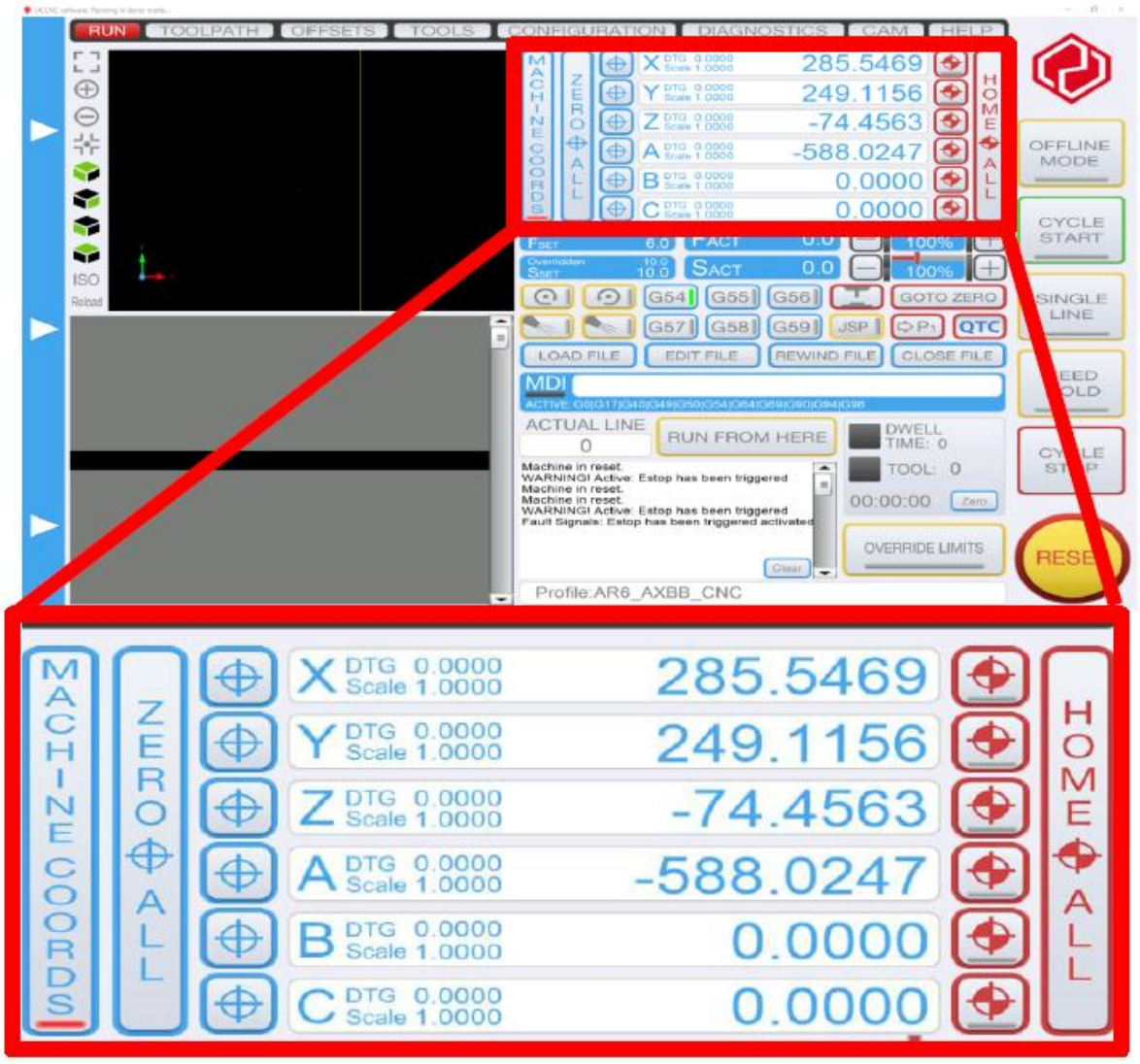


Figure 39 Note when the machine coordinates button is red you are in machine coordinates

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.

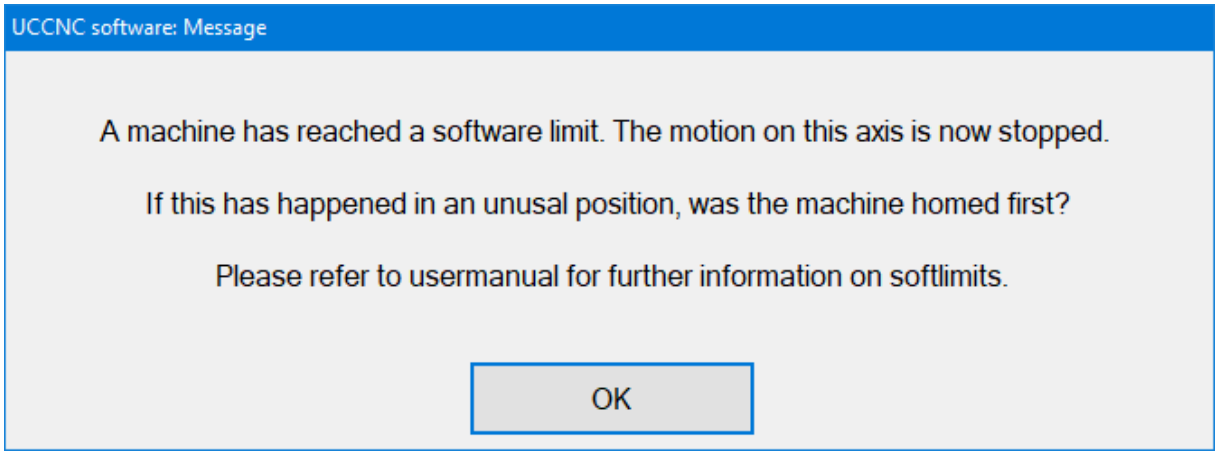


Figure 40: Softlimits message 1

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 carried-out "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.

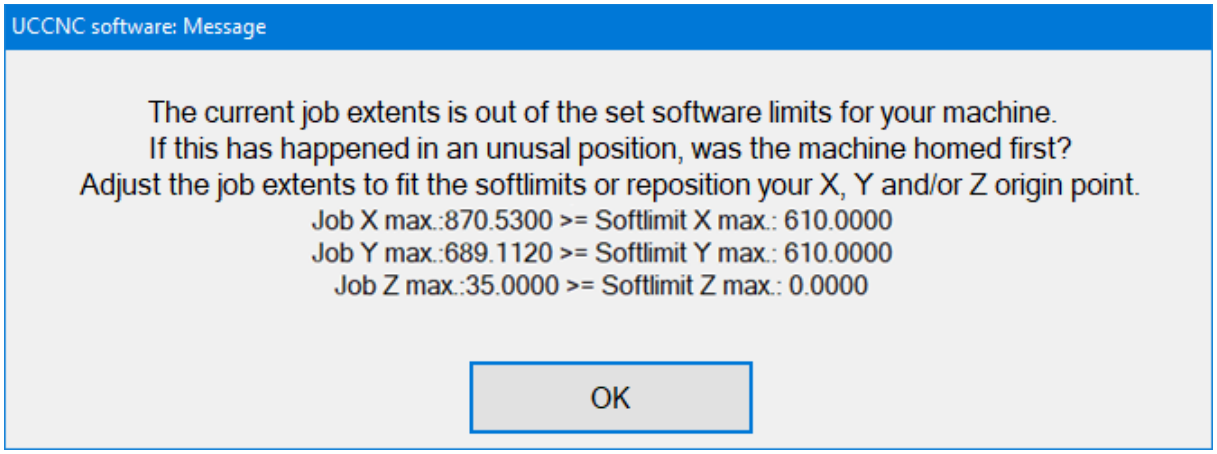


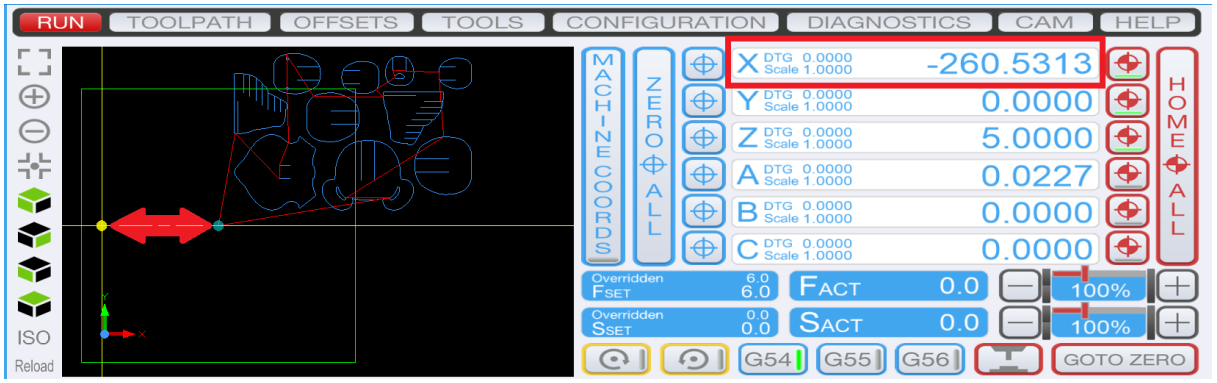
Figure 41: Softlimits message 2

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:

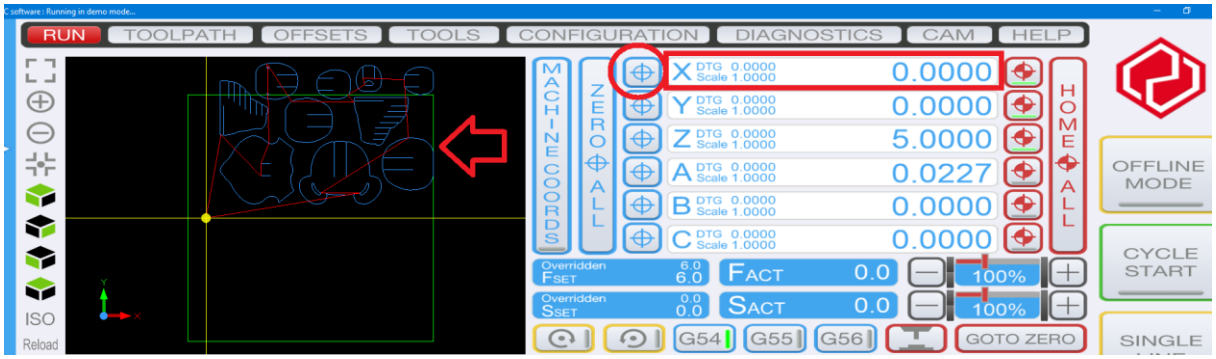
i) The G-code file loaded is larger than the machines working envelope (softlimit area)

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 41 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 zeroed in this location to make it fit.



After you reset your 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 42**

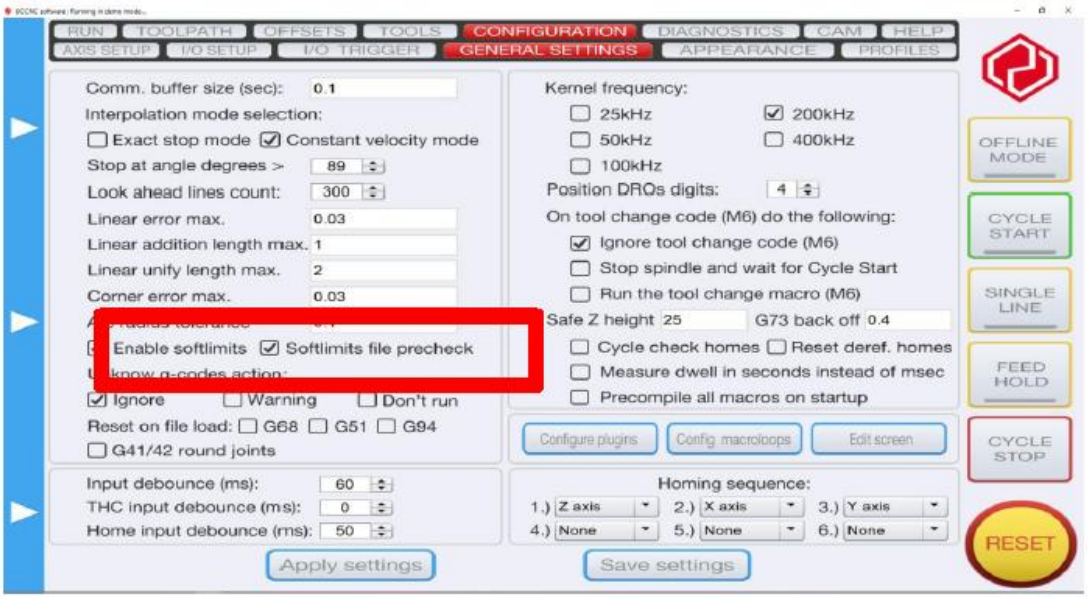


Figure 42 Enable and Disable softlimits

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.

15.7 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.

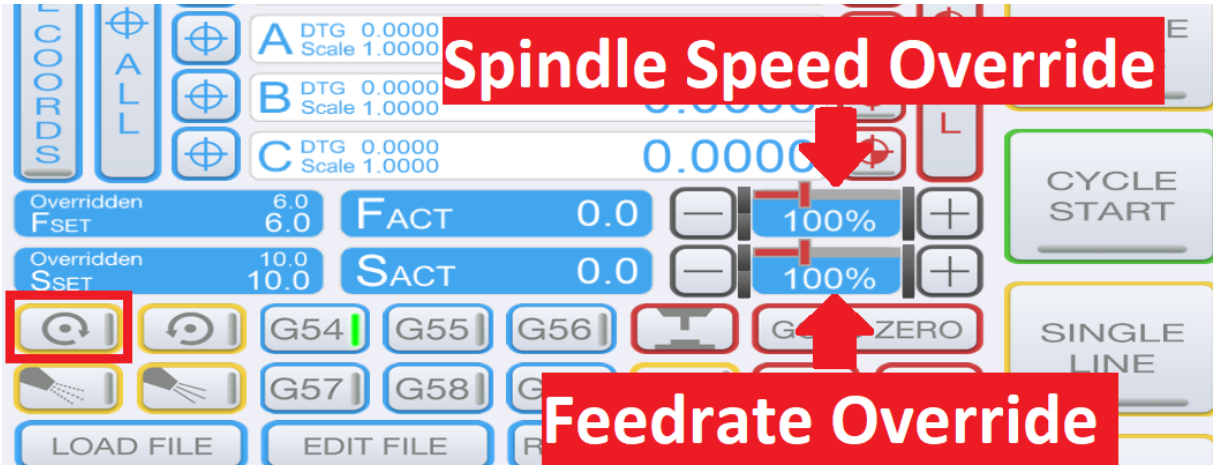


Figure 43: UCCNC Feedrate and Spindle Speed

15.8 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 44: 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 45: 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 46.

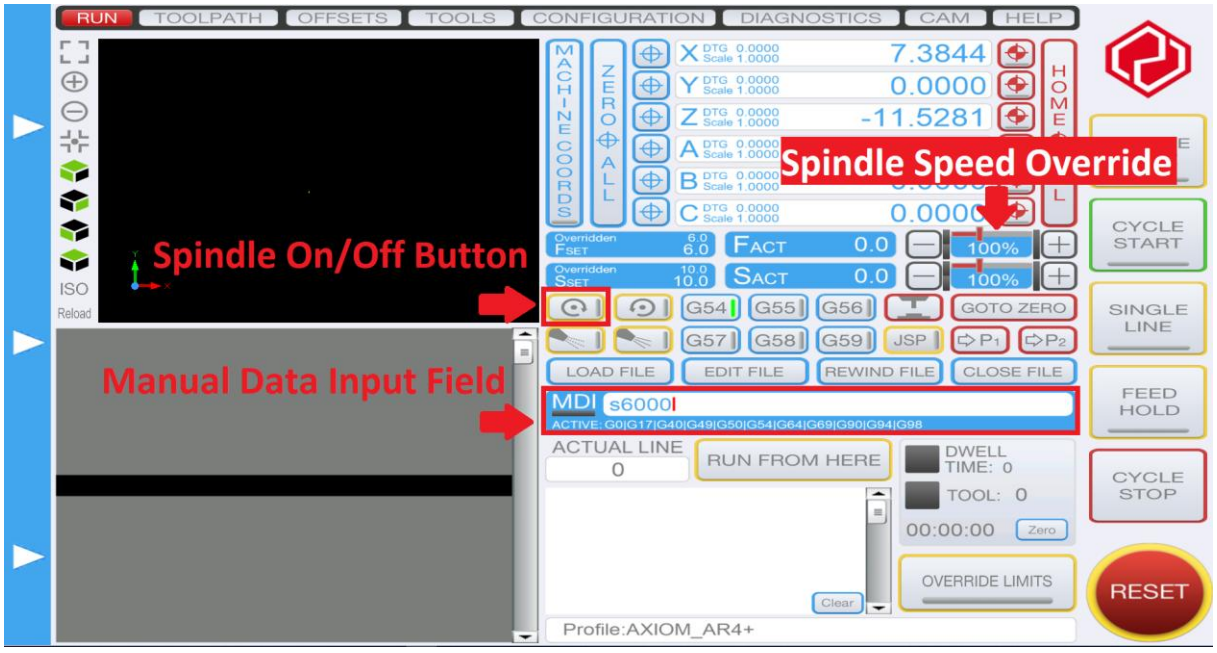


Figure 46: UCCNC Spindle Control

16 Detailed Parts Machining

This section goes into further detail on the provided 10 step cheat sheet.

16.1 Home the Machine

You must reference the machine each time the machine gets turned on or you start a fresh version of UCCNC to tell the software where the edges of the machine movement are each time. This can be done by using the "Home All" Command as you can see in Figure 47.

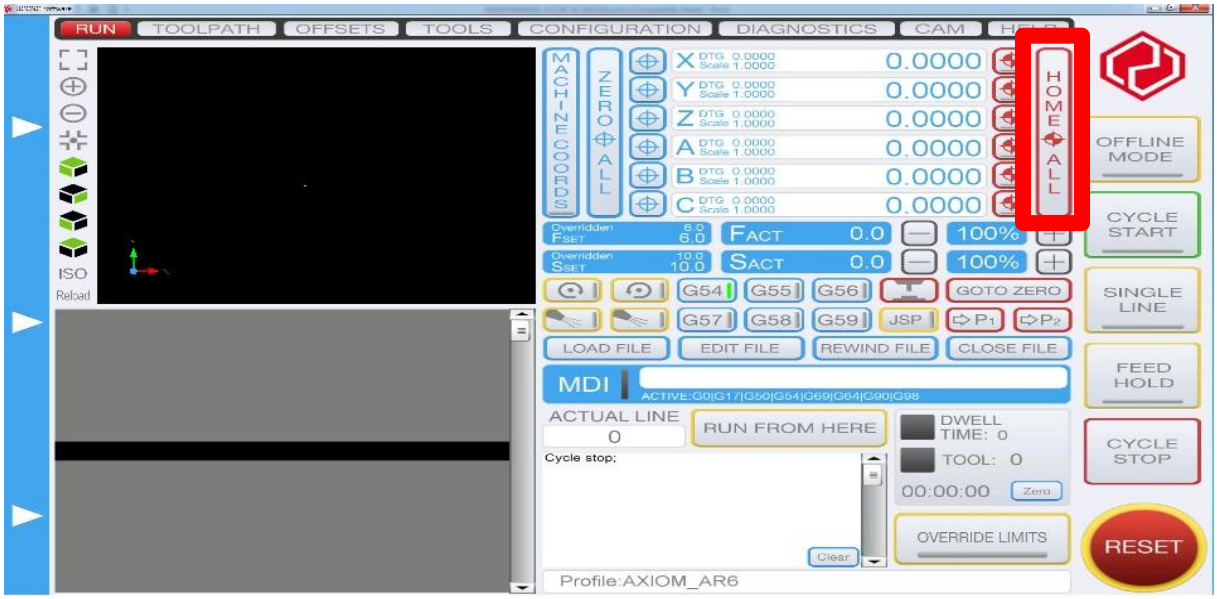


Figure 47: UCCNC Home All Command

This function aligns the software limits (or softlimits) for your machine with the actual working boundary on your table. For the softlimits to work properly you must home the machine every time you turn the machine on. Soft limits are explained in Section 2 in greater detail

16.2 LOAD YOUR GCODE FILE

The G-code file is generated from Cut2D/Vcarve and from there loaded into the UCCNC software

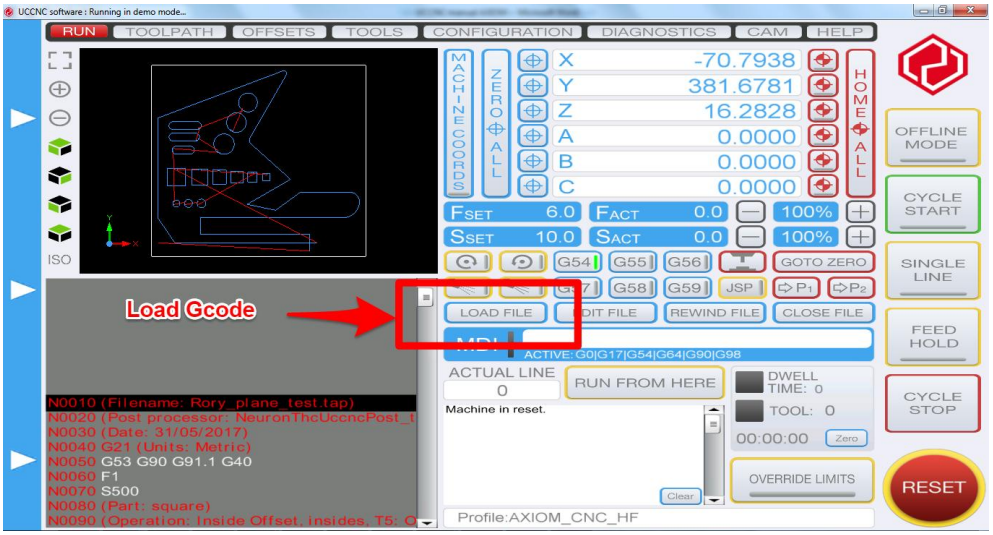


Figure 48: Load Gcode into UCCNC

16.3 CLAMP YOUR WORK PIECE TO THE MACHINE TABLE

WARNING: Very important the work piece is clamped securely. Refer Section 0 for more details

Figure 49: Clamping methods

16.4 LOAD YOUR TOOL AND SPECIAL NOTES ON COLLETS

Before loading any tool or touching the spindle refer to Figure 50 and press the reset button BEFORE attempting to change the tool.

Figure 50: Reset to disable spindle relay

DO NOT TOUCH the spindle unless the reset button is flashing red and yellow on UCCNC control interface on your PC.

The HF spindle use collets to load the cutters into the spindles. The HF spindle uses an ER20 collet system. You must disable the spindle system before you handle the collet mounting system by pressing the reset button on the UCCNC front panel.

Figure 51: HF ER20 Collets

The procedure for mounting the collets is important

11) Choose the correct collet for your tool. Always stick to the cutter spec. If using a 3.175mm (1/8th inch) cutter then use a 1/8th inch collet. Using the correct collet for your cutter will make tool mounting easier and safer.
12) Make sure the collet and collet nut are clean and debris free
13) Insert the collet into the collet nut until the collet seats. You should hear a "click". NOTE: if you mount a collet incorrectly into the spindle it will damage the collet.
14) Mount the collet nut (now with seated collet) into the spindle head fixture and loosely tighten by hand only – just so the collet is on the threads of the spindle.
15) Insert the tool you wish to use (NOTE: you should have pre- selected your collet size for your tool choice)
16) Mount the tool so that you have enough tool stick out to carry out your machining operation.
17) You must now lock the spindle head in order to tighten the collet nut
    a) On the HF spindle you must use the spanner to lock the head

18) The collet nut can now be used to tighten the collet into the spindle head securing the cutter. The HF spindle needs a special ER20 collet spanner. (NOTE: be careful as you do this as you can slip and damage the cutter or cut/injure yourself. You will be exerting a tightening force with the locking spanner around and near a sharp cutter. So be careful!)

19) Check the cutter is seated in the spindle and collet correctly by eye after you have installed the cutter and tightened the collet.
20) Carry out steps in reverse to remove the cutter.
21) Never use mismatched tool sizes for collets. Always use the correct collet for the correct tool
22) Never us a tool beyond its recommended RPM.
23) Always be prepared for the unexpected.
24) Take extra care when loading a tool and ensure the tool is mounted correctly prior to running a tool path and turning on the spindle.

16.5 SET X AND Y WORKING ORIGIN

Jog the machine such that the centre of the spindle is in line with the defined origin position you have set in the "Job Setup" in Vectric.

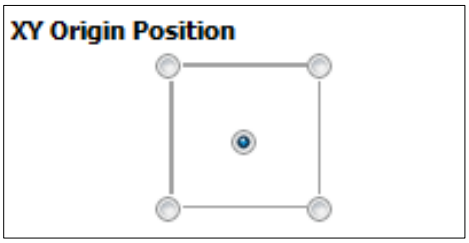


Figure 52: Vectric XY Origin

When in position you can select the set axis origin button for the applicable axis as below:

Figure 53: UCCNC Set workpiece origin button

16.6 SET Z WORKING ORIGIN

To set the z working origin, plug in the Touch off puck provided with your machine into the back of the left side upright. Place the puck either on the material surface or the table surface as per your defined origin position you have set in the "Job Setup" in Vectric.

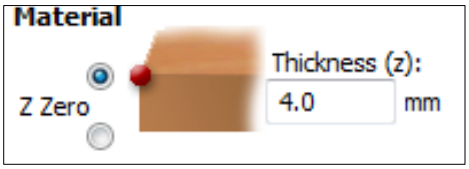


Figure 54: Vectric Z Zero Position

Jog the machine over the puck with your selected tool in place and run the auto touch off command ensuring you keep clear of machine at this point:

Figure 55: UCCNC AUTO Touch off button

The machine will automatically reduce the Z height until it touches the puck. When contact is made the machine will retract 10mm and set the Z working coordinate to 35. This takes into account that the puck is 25mm. The tip of your tool should now be 35mm from your required z origin position.

Please note this is an automatic operation and the machine will not stop until the cutting tool makes contact with the puck. To abort the operation hit "Cycle Stop" or the "Reset" button.

16.7 VERIFY THE WORKING AREA IS SUFFICIENT FOR MACHINING

With the live 3D toolpath viewer on UCCNC which can be found on the run tab or toolpath tab, you can see the current spindle location (yellow dot) with respect to the toolpath (blue line) you are about to cut.

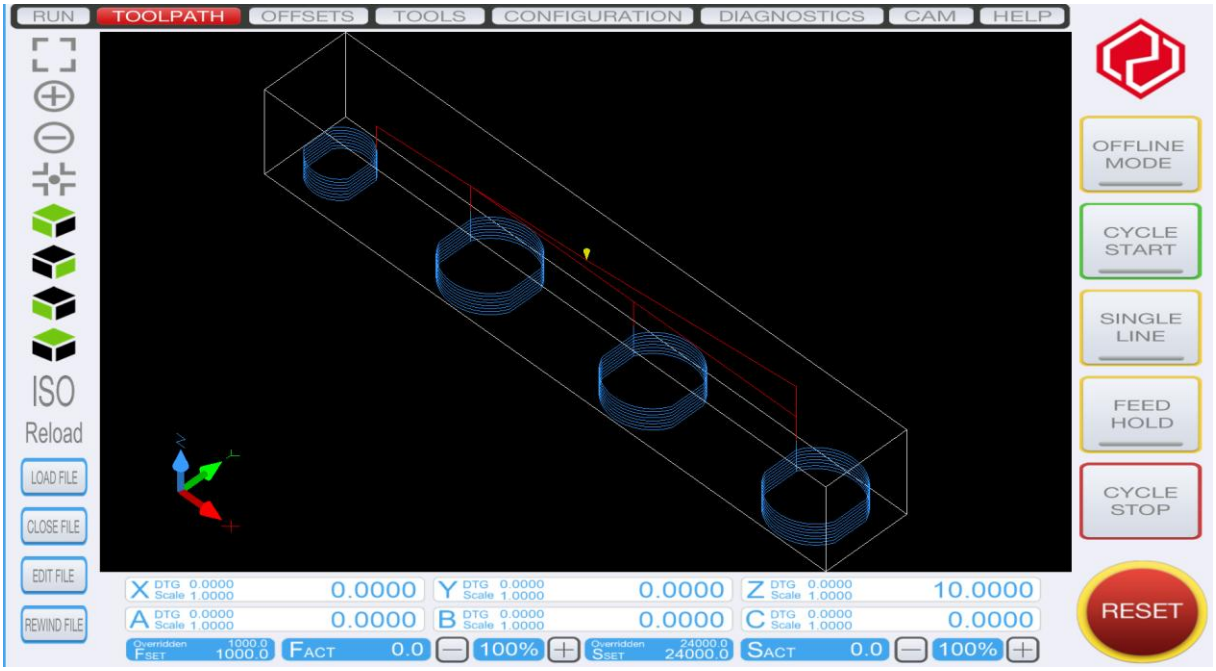


Figure 56: UCCNC Toolpath view

The red lines are rapid movements where the machine will move full speed vs the blue lines picking up your set feedrate for machining. The white box is a boundary created by UCCNC and does not represent any part of the Vectric CAD/CAM setup. It signifies there is no movements outside this boundary.

Prior to starting your cycle, this is an opportunity to manually jog your machine within this boundary to ensure the following:

- Verify the working area is sufficient for machining
- Ensure there are no clamps or fixings in the boundary area
- Ensure the machine can travel within the boundary and the work doesn't extend beyond the size of your machine
- Ensure the Material is large enough to be machined

16.8 Reduce Feed rate %

Prior to starting the cycle you can override and reduce the feed rate set in your G-Code by clicking the "-" button on the Fset line.

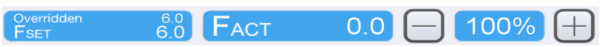


Figure 57: UCCNC Fset Override

This will slow the machine down giving you more reaction time if your set-up in incorrect for machining.

16.9 Start cycle

Figure 58: UCCNC Cycle start

- Verify the working area is sufficient for machining
- Start Cycle
- Observe and listen

16.10 Increase Feed Rate %

After you have observed the start of your cycle and believe you have set-up correctly, and the machine is performing appropriately, you can increase the feed rate to what is set in your G-Code by clicking the "+" button on the Fset line. Observing the chips, the cut surfaces and listening to the machining operation provide clues to efficient and effective cutting. While you can continue machining and experimenting with feeds and speeds on the fly, it will worth making a note of the actual values that are proving effective, so that the G-code in the CAM may be appropriately modified for future iterations of the same toolpath.

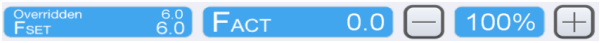


Figure 59: UCCNC Fset Override