THE NEW GENERATION

LC-Plus

CNC MACHINE MANUAL

Techno CNC Systems



TECHNOCNC.COM

29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

TABLE OF CONTENTS

1 ***Safety*** 3

2 ***Guidelines***
- Machine Guidelines 5-8
- Materials Suitable for Cutting 9

3
- Machine Site Preperation 11-15
- Control Cabinet Functions 16-22
- Tooling and Colleting 23-28
- Control Interface Cheat Sheet 29-40
- Start up and Homing the Machine 41-43

4 ***Machine Operations***
- Motion Control and Jog Menu 44-47
- Tool Changes and Measuring 48-52
- Setting a WCS Origin 53-58
- Loading a G-Code File 59-61
- Advanced Controls 62-64

5 ***Basic Troubleshooting*** 65-69

6 ***Machine Warranty*** 70-74

CONTACT US TECHNOCNC.COM 29 Trade Zone Drive Ronkonkoma, NY 11779 631-648-7481

SAFETY INSTRUCTIONS

READ THESE INSTRUCTIONS THOROUGHLY BEFORE OPERATING MACHINE. DO NOT OPERATE MACHINE IF YOU ARE UNFAMILIAR WITH THESE SAFE OPERATING INSTRUCTIONS. DO NOT OPERATE MACHINE WITHOUT KNOWING THE EMERGENCY STOP SWITCH IS LOCATED.

WARNING: IMPROPER OR UNSAFE OPERATION OF MACHINE WILL RESULT IN PERSONAL INJURY AND/OR DAMAGE TO MACHINE AND EQUIPMENT.

1. Keep fingers, hands, and all other objects away from machine while power is on.
2. Disconnect power to all system components when not in use, when changing accessories, and before servicing.
3. Do not loosen, remove, or adjust machine parts or cables while power is on.
4. Exercise care with machine controls and around keyboard to avoid unintentional starting.
5. Make sure voltage supplied is appropriate to specifications of components.
6. Machines must be plugged into three-pronged grounded outlets. Do not remove the grounding plug or connect into an ungrounded extension cord.
7. Keep cables and cords away from heat, oil, and sharp edges. Do not overstretch or run them under other objects or over work surfaces.
8. Use proper fixtures and clamps to secure work. Never use hands to secure work.
9. Do not attempt to exceed limits of machine.
10. Do not attempt to use machine for purposes other than what is intended.
11. Use machine only in clean, well-lit areas free from flammable liquids and excessive moisture.
12. Stay alert at all times when operating the machine.
13. Always wear safety goggles and dust mask.
14. Do not wear loose-fitting clothing or jewelry when operating machine. Long hair should be protected.
15. Always maintain proper balance and footing when working around the machine.
16. Maintain equipment with care. Keep cutting tools clean and sharp. Lubricate and change accessories when necessary. Cables and cords should be inspected regularly. Keep controls clean and dry.
17. Before using, check for damaged parts. An authorized service center should perform all repairs. Only identical or authorized replacement parts should be used.
18. Remove any adjusting keys and wrenches before turning machine on.
19. Do not operate the machine unattended.
20. Follow all safety instructions and processing instructions in the MSDS for the material being processed.
21. Use proper precautions with dust collection systems to prevent sparks and fire hazards.
22. Make sure to have proper fire extinguishing equipment on hand at all times.

PREVENT FIRE HAZARDS by using the proper feeds, speeds, and tooling while operating your Techno machine. For example, setting feeds and speeds too low and/or using dull tool bits creates friction at the material. The friction generates heat which can result in a fire that can be drawn through the vacuum table or dust collector without warning. Fire hazard from friction heating caused by dull tools is possible when cutting certain materials, especially composite material such as wood composites, MDF and Particleboard. © 08/2014

CONTACT US TECHNOCNC.COM 29 Trade Zone Drive Ronkonkoma, NY 11779 631-648-7481

CNC
SAFETY AND MACHINE GUIDELINES

It is your responsibility to thoroughly read, understand, and follow all the safety and machine guidelines. Failure to follow these guidelines can result in serious injury or death.
Safety –

· DO NOT remove any of the warning or safety labels mounted on the machine.
· DO NOT disable, bypass, or remove any safety devices equipped with or around your Techno CNC Machine.
· The operation of any Techno CNC machine while under the influence of alcohol or any narcotics is strictly prohibited.
o You should not operate the machine while, or after taking any prescription, or over the counter medication.
· Remove any jewelry while operating the machine.
· Ensure that excess hair is tied back and protected while operating the machine.
· DO NOT wear loose clothing while operating the machine.
· The machine's work bed is not intended as a workbench. Do not use the machine as a workbench or hammer the tabletop.
· Stay clear of the machine while it is in operation.
· DO NOT remove any material while the machine is running.
· DO NOT attempt to stop the machine with your hands or any foreign object.
· Any repairs or service done to the machine should only be conducted by qualified persons.
· Opening the electrical cabinets while the machine is powered can result in high voltage shocks or electrocution.
· Protective eyewear, headgear, clothing, and ear protection should be worn by all personnel working in the vicinity of the CNC machine.
· Keep chemicals and flammable materials away from any of the machine's electronics.
· Always have a suitable and inspected fire extinguisher in the machining area.
· Stay clear of any moving parts when the machine is in operation.
· Keep the work area clean and free of work materials or material byproducts.
· Ensure that the work area is clear of loose parts, work pieces, tools and other items before operating the machine.

CNC
SAFETY AND MACHINE GUIDELINES

Ensure all cutting tools are in good working order before using them on the machine.
· Disconnect power to the machine when it is not in use or prior to servicing.
· Ensure that the proper power voltage is supplied to the machine. Consult with a certified Techno CNC Systems technician for specification.
· Ensure that the machine is grounded to a 10 ft ground rod using the ground lug provided on the back of the machine frame.
· DO NOT attempt to exceed the limits of the machine.
· DO NOT attempt to use the machine for anything other than it intended use.
· Feeds and speeds should be properly calculated. Failure to do so may result in fire, melting, or air born cutting tools.
· Ensure all tooling is properly secured in the machine spindle.
· DO NOT operate the machine while it is unattended.
· ALWAYS wear a full body safety harness when climbing to the top of the machine gantry for maintenance or any other procedures. ALWAYS secure the harness to the steel cable stretched across the length of the gantry bridge.
· Routine maintenance is required. Keep machine mechanics properly lubricated and replace consumable items as needed. Machine cables and cords should be inspected regularly.
· Ensure that all controls are clean and dry.
· Stay alert at all time while operating or working in the vicinity of the machine.

MACHINE GUIDELINES –

· Ensure that all operators have read and understand all the safety instructions, machine connections, and operating instruction provided in this manual.
· Read, understand, and comply with the warning labels and safety instructions.
· Ensure anyone near the machine knows how to perform an emergency stop.
· Only authorized personnel should enter the working area while the machine is in operation.
· Verify that the machine is in good working condition before every use.
· Ensure that all cutting tools, knives, and/or engraving tools are sharp and in good working condition.
· Always ensure that any substrate being processed is securely held to the machine table.

CNC
SAFETY AND MACHINE GUIDELINES

· Stay clear of all moving parts while the machine is in operation.
· All users should complete a training course, read the machine manual, and/or review all tutorials prior to operating the machine.
· All users should be knowledgeable about basic design, operation, cleaning, and maintenance of the machine.
· All users should be knowledgeable regarding materials the CNC Router can cut.
· Operators should have knowledge regarding the proper cutting tools for cutting specific materials.
· Programmers and operators should understand how to properly calculate feeds and speeds.
· Operate the CNC machine in a safe manner according to the safety guidelines.

Personal Protective Gear –

All Operators and personnel working around or near the machine must wear protective equipment based on the machine configuration and the materials being processed.

· Eye Protection
o Eye/face protection must be always worn during machine operation.
· Hearing Protection
o Wear hearing protection when the machine is in operation. Hearing protection should also be used when a vacuum pump and dust collector are running.

· Mask/Respirator
o Some materials, during processing, will create airborne toxins. Wear the appropriate mask/respirator while processing these materials. Refer to the SDS or MSDS for specific information.

· Gloves
o After processing is complete, materials may have sharp edges or be hot to touch. Cutting tools are also sharp and may be hot once processing is complete. Wear suitable gloves to protect your hands.

· Clothing
o Wear appropriate clothing to prevent injury from sharp edges or hot materials and cutting tools. Wearing loose clothing should not be worn to prevent clothing from getting caught in linear mechanics

29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

Techno CNC Systems provides a variety of technical support services with the purchase of any CNC router or plasma table. The following is a brief outline of available support options. On-site service is available for additional fees. To receive a quote for an on-site visit, please inquire at technocnc.com or call us at 631-648-7481.

Phone Support
Speaking with a professional and certified Techno CNC technician is quick and easy. Simply call us at 631-648-7481. Our support team is available Monday – Friday 8:30am – 5:00pm Eastern Standard Time. Techno CNC Systems will make every effort to handle phone calls in a timely manner, but due to the nature of machine diagnosis and the varying capabilities of different machine operators, we cannot guarantee hold times for phone-in support. Technical support includes troubleshooting, machine configuration and assembly, tooling recommendations, parts inquiries, and quality related issues. OPERATIONAL TRAINING IS NOT INCLUDED IN OVER THE PHONE SUPPORT.

E-mail
Techno CNC Systems will return e-mails sent to the support@technocnc.com e-mail address within 24 hours Monday – Friday.

Techno University
Techno CNC Systems provides a number of software and machine training opportunities at our Long Island, New York facility. Please call 631-648-7481 for details.

Cinematic Tutorials
Get started learning all the basics for your Techno CNC machine using our video tutorials listed for each machine. For access to the video tutorials, please visit us at https//:technocnc.com/technical-support/

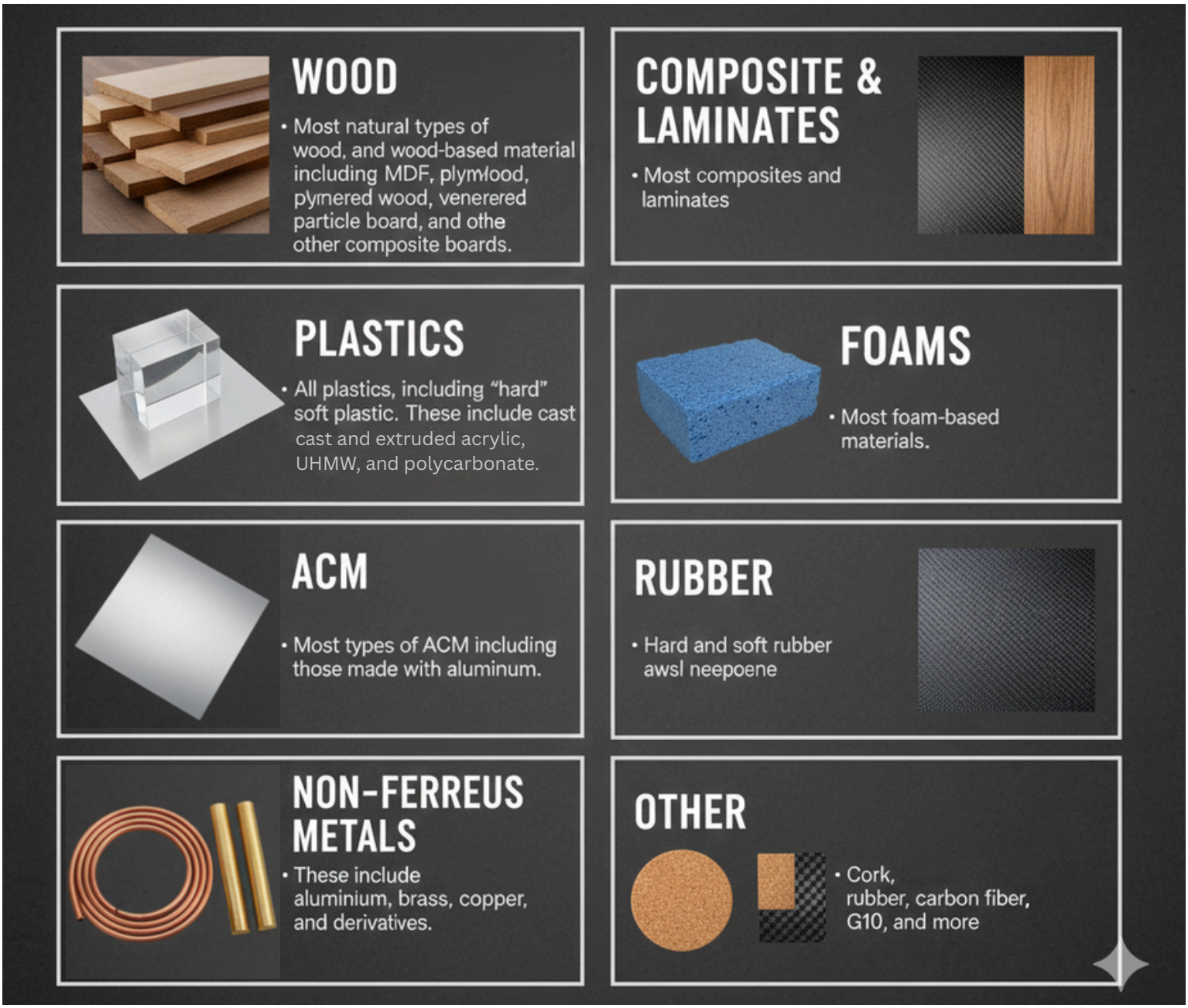
Techno CNC Shop
A variety of consumables, accessories, and cutting tools can be quickly and easily purchased through the Techno CNC Shop. Visit https://technocncshop.com to learn more and start filling your cart.

Techno CNC Systems

29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

THE NEW GENERATION
LC-Plus

MATERIALS
SUITABLE FOR CUTTING



Techno CNC Systems

29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

PROCESSING MATERIAL

It is important to understand the potential hazards associated with processing various materials. Read, understand, and comply with the following information concerning these varied conditions.

COMBUSTIBLES



When processing materials such as wood, MDF, cardboard and other flammable products, the risk of fire exists.
Ensure all operators have access to a suitable fire exinguisher.

DUST EXPLOSION



Processing materials such as MDF or other wood products creates dust. The risk of a **dust explosion exists**.
When collecting chips from non-ferrous metals, hot chips collected can cause a explosion. Consult with a dust collection specialist for a suitable dust collection system for each application.

TOXIC FUMES AND DUST



Processing materials such as MDF. plastics, and some composites can expose the operator and others in the vicinity to toxic fumes, gases, or dust. Ensure a proper respirator is worn when using these materials and working around the machining area.

SHARP EDGES AND CHIPS



Metal materials, natural wood, and composites may have **sharp** edges and/or chips that can cut or pierce yout skin.
Ensure all personnel in the vicinity of the work area wear cut-resistant gloves and protective clothing.

HOT MATERIALS AND WASTE



Processing plastic materials and metals generates heat in both the material being cut and the chips produced from cutting.
This increases the risk of skin burns. Ensure that heat resistant gloves. dothing and face protection are worn when processing these materials.

HEAVY/AWKWARD SHEETS OF MATERIAL



While loading/unioading heavy, large, or awkward sheets of material, use a helper. It is also advised that the use of appropriate lifting and rigging are used to prevent injury or death.

THE NEW GENERATION

LC- *Plus*

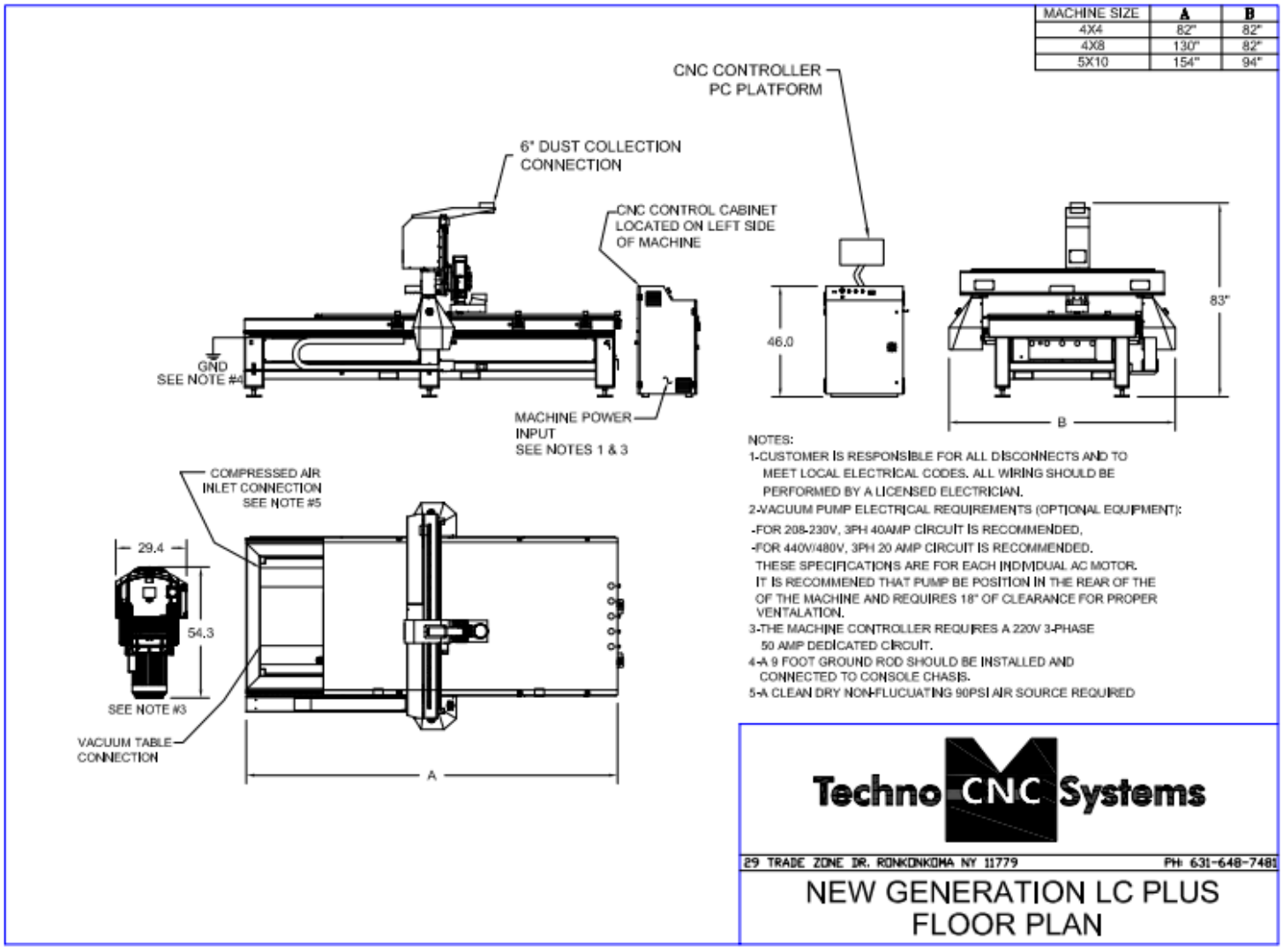
MACHINE SITE PREPERATION

29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

MACHINE SITE PREPARATION

When installing the Techno CNC Systems LC Plus Series machine in your shop, there are numerous factors that influence the potential productivity, ease of use, and safety of the operator. The main factors being discussed include, but are not limited to, the physical layout and placement of the machine in the shop, availability of power, an EMI ground, compressed air, and ventilation.

- When preparing to install the LC Plus Series CNC Router; provide sufficient space all the way around the machine. It is recommended that a 3 ft (914mm) workspace be obstacle free around the entirety of the machine.
- Forklift/load material to be processed opposite the cable carriers only. It is recommended that the gantry be parked towards the back of the machine while loading and unloading material.
- A dedicated earth ground must be provided and should be installed in a manner so that there is no trip hazard. It is recommended that the ground be secured to a copper 10 ft earth ground rod.



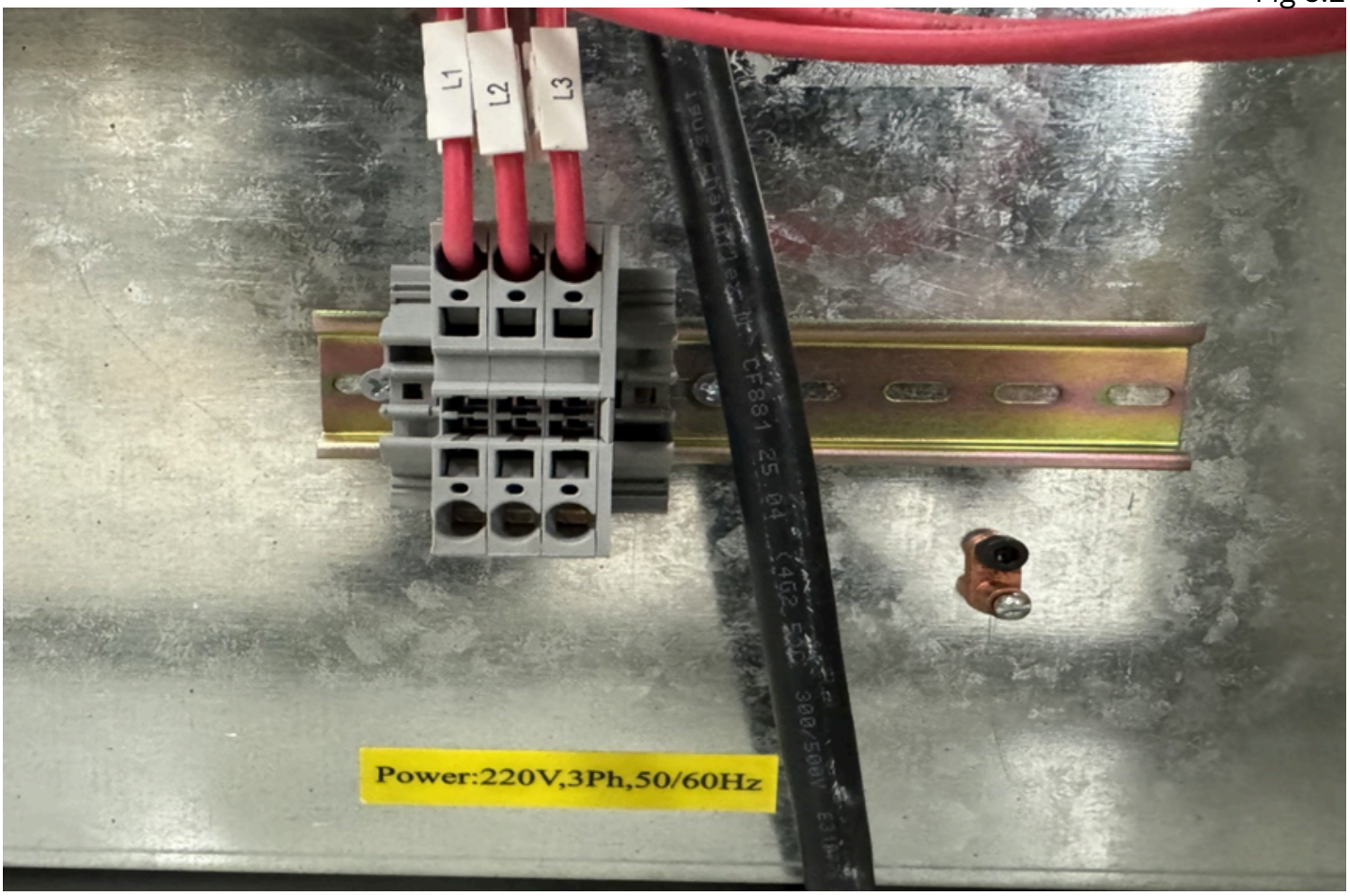
Techno CNC Systems

29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

MACHINE SITE PREPARATION

- 4-6 Leveling feet are included with the LC Plus Series CNC Router. Remove these from one of the boxes included with the machine and install onto all 4-6 base legs of the machine.
- The power requirements for a standard LC Plus Series machine is a 220V 3phase power source connected to its own 50A circuit. Single phase power and 380V – 440V power arrangements can be accommodated. Please contact your sales associate, or Techno CNC Systems' support team for more information.
- The main power terminals are in the front of the control cabinet, towards the bottom right corner of the rear interior.
- Connect the live legs of power to L1, L2, and L3 (Labels may vary). Connect the Ground to the copper ground lug provided, or the green/yellow terminal bridge mounted to the DIN rail.

Fig 5.2



- The LC Plus Series router utilizes a high frequency, air cooled, 12hp automatic tool changing spindle. This advanced and powerful spindle, along with other accessories, requires a constant flow of dry, compressed air. The air pressure requirements to run these components is 90 PSI. The other standard accessories that are equipped on the LC Plus Series include:
  - Pop-up alignment pins.
  - Dust hood.

Techno CNC Systems

29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

MACHINE SITE PREPARATION

- In the back of the machine are a series of regulators and flow control components. Connect an air hose supplying at least 90 PSI to the first 8mm air fitting provided. The fitting should be labeled “Air In”. An adapter similar to the one below can be made up using parts available at most hardware stores, or you can call in to purchase one directly from Techno CNC to better fit a female compressor hose.



Fig 5.3

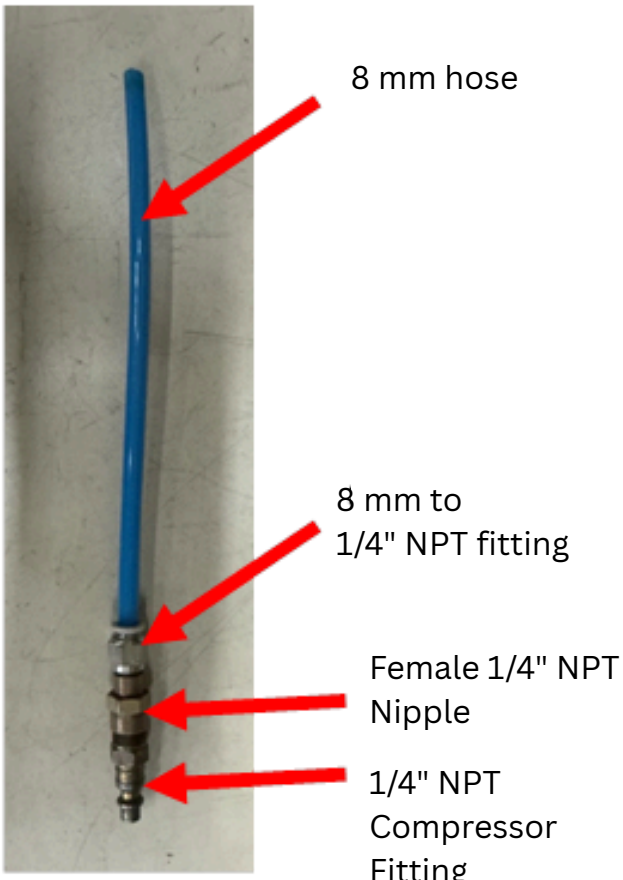
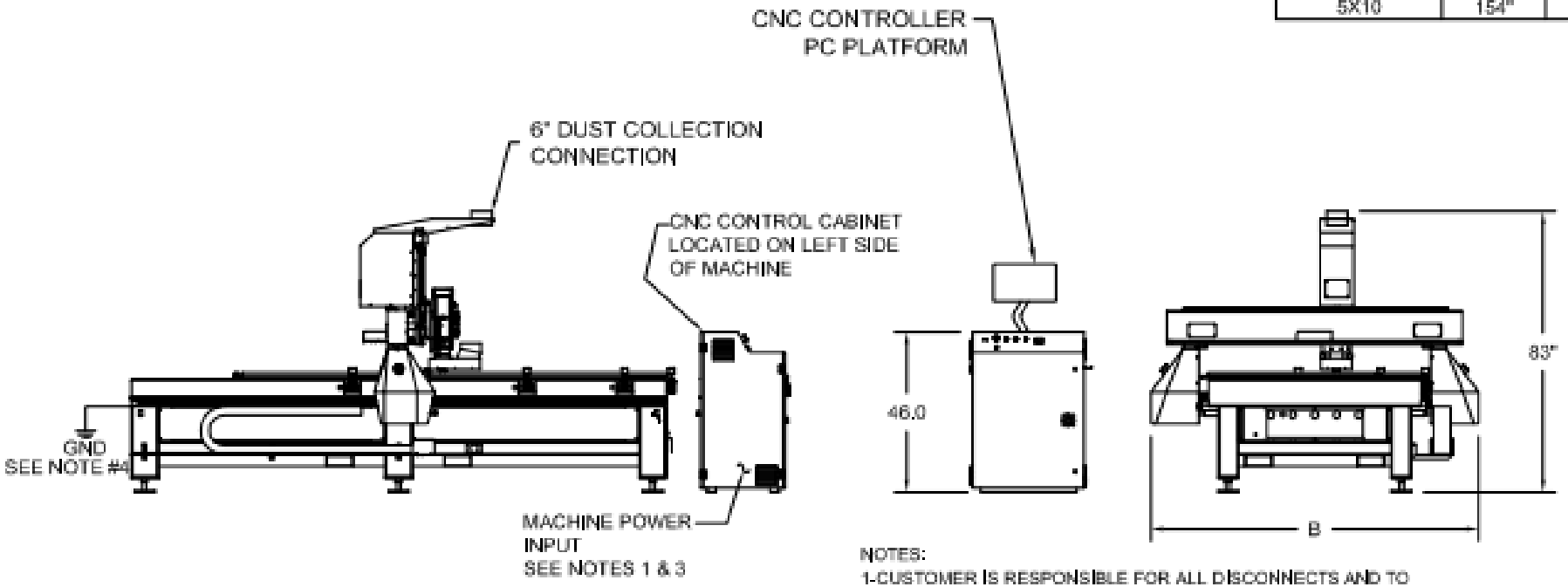
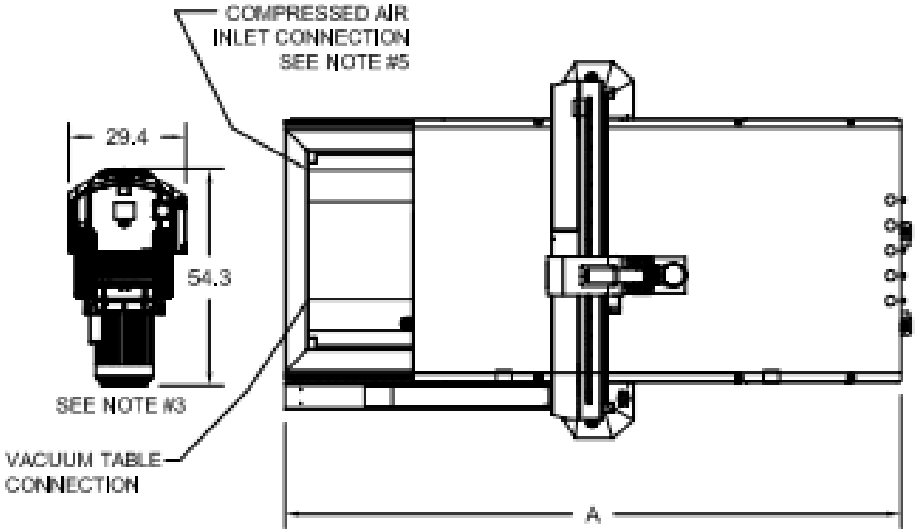


Fig 5.4

- Before beginning to power on the machine and operate its controls, it is important to free up the gantry from any obstructions. Make sure to unload all boxes laid on top of the machine bed, and carefully go through each one to remove any tools, tool boxes, and other accessories that may have been included or purchased.

MACHINE SIZE	A	B
4X4	82"	82"
4X8	130"	92"
5X10	154"	94"





NOTES:

1-CUSTOMER IS RESPONSIBLE FOR ALL DISCONNECTS AND TO MEET LOCAL ELECTRICAL CODES. ALL WIRING SHOULD BE PERFORMED BY A LICENSED ELECTRICIAN.

2-VACUUM PUMP ELECTRICAL REQUIREMENTS (OPTIONAL EQUIPMENT):

-FOR 208-230V, 3PH 40AMP CIRCUIT IS RECOMMENDED,

-FOR 440V/480V, 3PH 20 AMP CIRCUIT IS RECOMMENDED.

THESE SPECIFICATIONS ARE FOR EACH INDIVIDUAL AC MOTOR. IT IS RECOMMENED THAT PUMP BE POSITION IN THE REAR OF THE OF THE MACHINE AND REQUIRES 18" OF CLEARANCE FOR PROPER VENTALATION.

3-THE MACHINE CONTROLLER REQUIRES A 220V 3-PHASE 50 AMP DEDICATED CIRCUIT.

4-A 9 FOOT GROUND ROD SHOULD BE INSTALLED AND CONNECTED TO CONSOLE CHASIS.

5-A CLEAN DRY NON-FLUCUATING 90PSI AIR SOURCE REQUIRED

Techno CNC Systems

29 TRADE ZONE DR. RONKONKOMA NY 11779 PH: 631-648-7481

NEW GENERATION LC PLUS FLOOR PLAN

THE NEW GENERATION

LC-Plus

CONTROL CABINET BUTTONS & FUNCTIONS

Techno CNC Systems

29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

CONTROL CABINET BUTTONS/FUNCTIONS

It important that all operators and personnel that work with the CNC machine learn and understand all the machine controls and functions. This section will summarize the controls mounted on the control cabinet and their functions.



Fig 6.1

A.) Emergency Stop – The E-stop button will cut power to the controls and drives of the system. To disengage the emergency stop, rotate the switch clockwise. The machine will not power on with the e-stop button engaged.

B.) Power On/Off – The power on/off button will power on and shut down the system when pressed in and out.

C.) USB Slots – The USB slots are where the operator will connect their usb drive. The usb drive should contain the .nc g-code files that will be run on the machine. Files put on this usb can be loaded and copied or loaded onto the machine's controller.

D.) VFD Display – The VFD display shows a digital read out of the real time spindle speed. The display will show 0.0 until the spindle is initiated manually, or through running a programmed file. The VFD is preprogrammed prior to delivery. It is not advised to adjust any parameters before speaking with a certified Techno CNC Systems' technician.

E.) Keyboard – The keyboard is used to operate the PC based controller, edit files, parameters, and other PC accessories that would require the user to type in information.

F.) PC Mouse – The mouse is used to select the different applications and controls available on the LC Plus series controller.

Techno CNC Systems

29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

Power Disconnect:

The power disconnect applies power to the rest of the system. Though power may be live at the cables secured to the main power terminal inside of the cabinet, the power disconnect prevents power from getting to the rest of the electronics until it is rotated to the on position. It is highly recommended that the power disconnect be turned off when the machine is not in use. This device is used to protect the sensitive electronics inside of the control cabinet.



Fig 6.2

DISCONNECT OFF



Fig 6.3

DISCONNECT ON

Large PC Monitor

As shown in Fig. 6.4, the LC Plus Series machine comes with a PC-based control system. With that PC controller, there is a touch screen monitor provided that allows users to operate the machine both using the keyboard and mouse as well as touching the monitor screen to navigate the interface and control the machine.



Fig 6.4

After unpacking the monitor from its packaging, secure the unit to the monitor stand using the hardware provided (Fig. 6.5). Once the monitor is safely secured to the stand, connect the power, display, and touch screen cables protruding from the monitor stand (Fig. 6.6).

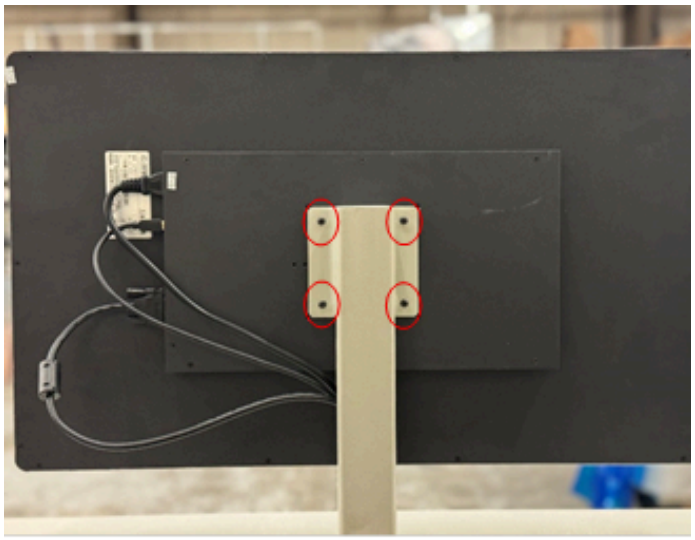


Fig 6.5

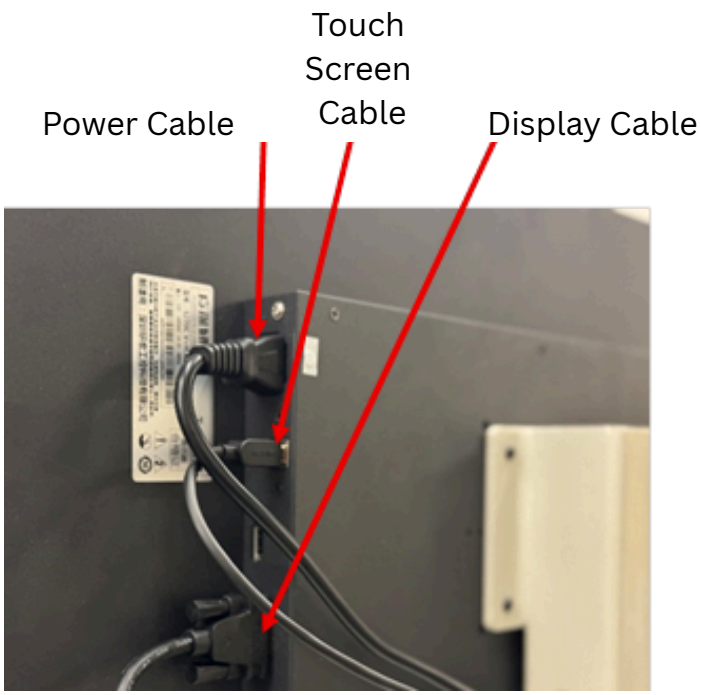


Fig 6.6

The LC plus also comes with a wireless handheld manual pulse generator, or MPG. This unit runs on AA batteries. The MPG is used to remotely jog the machine axes. It is useful for re-learning tool stand locations, touch pad locations, setting accurate work coordinate zeros, and even stepping through programmed g-code files.



Fig 6.7

The handwheel, or MPG has a magnetic back so that it can be secured to any metal part on the machine or control cabinet. On the right side of the control cabinet is a hook that is used for holding the MPG when not in use (Fig. 6.8).



Fig 6.8

Techno CNC Systems

29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

POWER AND AIR REQUIREMENTS

The LC Plus Series is a high voltage machine, equipped with various accessories that require compressed air. This section will give an overview of the voltage and air requirements as well as the connection points for each.

1.Power Requirements:

- Customer is responsible for all disconnects and to meet all local area electrical codes. All wiring should be performed by a licensed electrician.
- The machine is rated to be connected to a dedicated 50A circuit. Failure to power the machine off a dedicated circuit can cause internal damage to the various electronics installed on the machine.
- The machine requires a dedicated 220V, 3-phase, 50amp circuit All electrical connections should be installed by a LICENSED ELECTRICIAN.

·The main power connection is located in the front of the control cabinet.

- Crimped connectors should be used when supplying high voltage power. It is not recommended to connect bear wire to any of the terminals. Failure to do so may cause unexpected shorts, causing excessive damage to the machine's electronics.
- Connect the 3 live legs to the terminals labeled L1, L2, and L3. The ground connection should be secured to the ground lug provided, or the ground terminal bridge next to the 3-phase bridge.

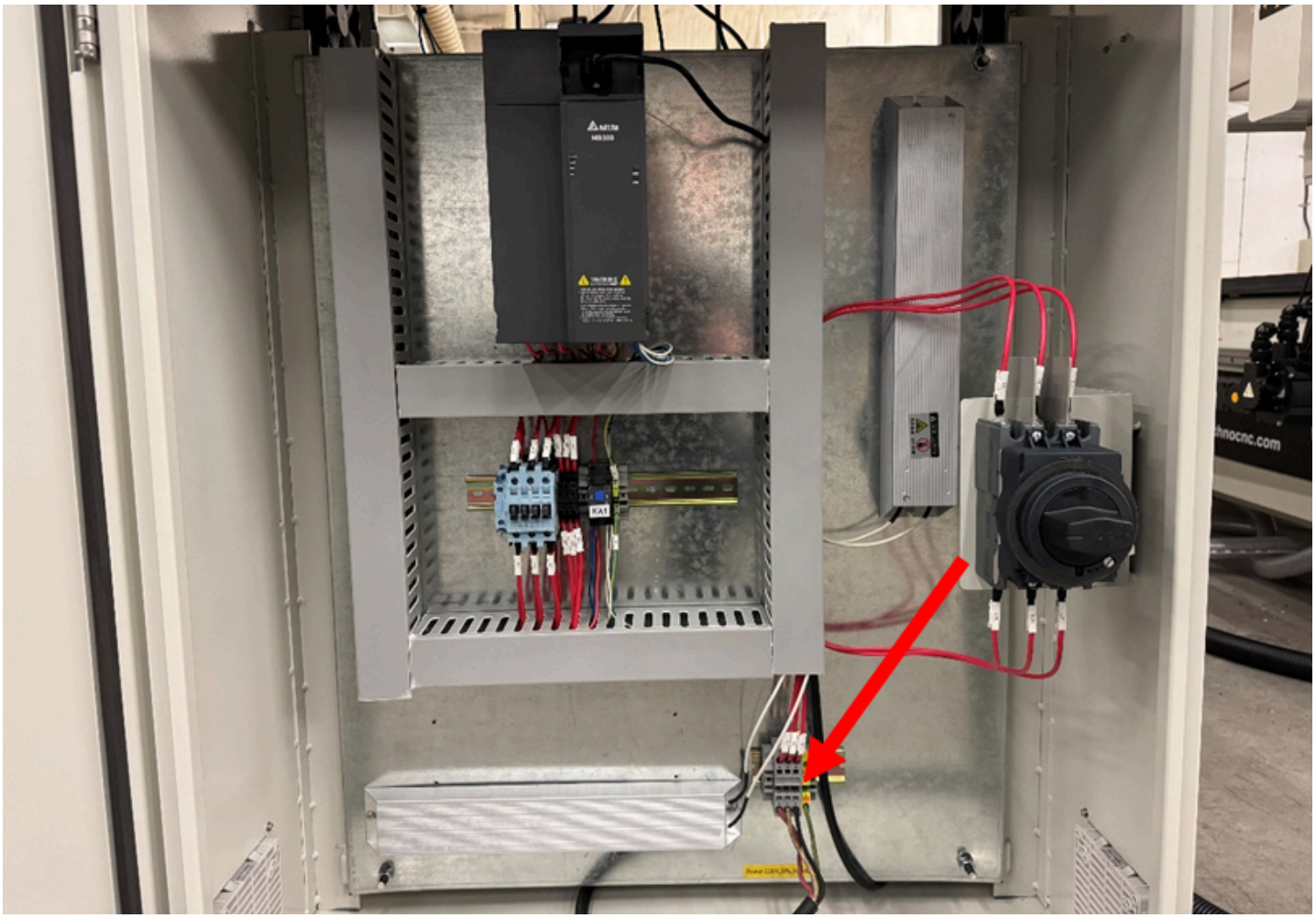


Fig 7.1

Techno CNC Systems

29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

POWER AND AIR REQUIREMENTS

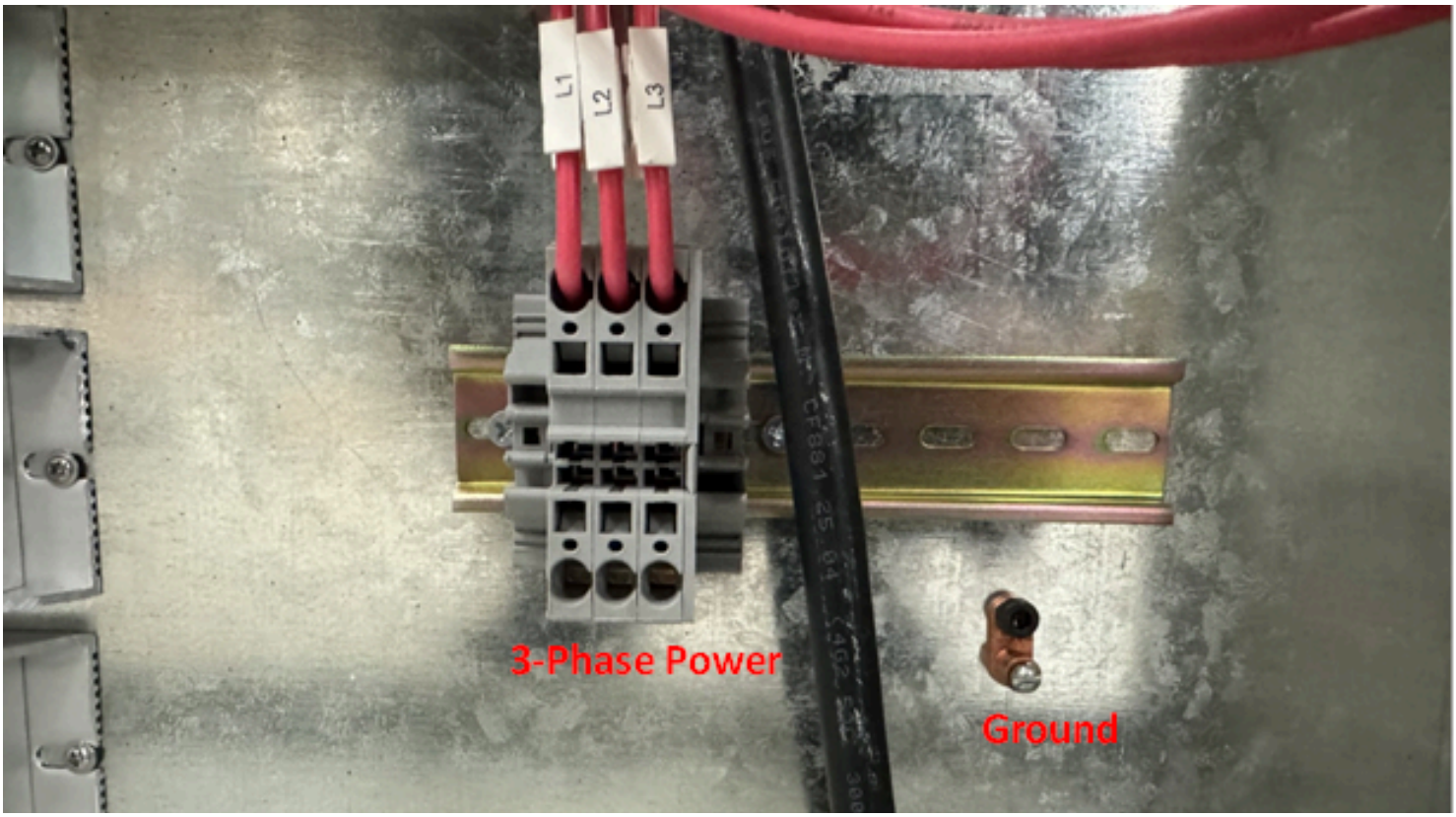


Fig 7.2

1.Air Requirements:

- The LC Plus requires a constant flow of clean, dry, compressed air.

If an air dryer is not in line with the air source for the machine, it is recommended one be installed. Failure to do so will result in premature spindle failure.

- It is recommended to use a constant 90 PSI source. The machine is equipped with its own regulators, so bringing a larger amount of air directly to the regulator is ok. The machine's regulators will then only allow the spec'd air pressure throughout the system.
- The fitting that feeds the main regulator can be found at the back of the machine. The fitting provided is spec'd to fit an 8mm air hose. An adapter or special fitting can be made up using products found at all hardware stores if needed. Please re-read the machine site prep section of this manual for further details.

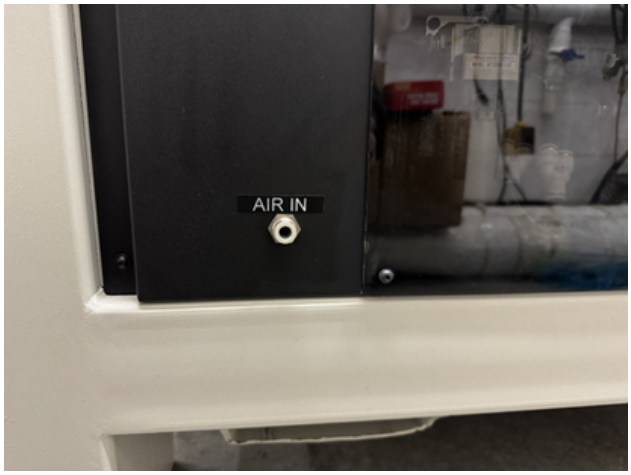


Fig 7.3



Fig 7.4

TOOLING AND COLLETING

29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

TOOLING AND COLLETING

This section will cover the proper tools and collets to use with the LC Plus Series CNC Router as well as an overview on how to properly load tools into the automatic tool changing spindle's tool holders. In addition, the process and proper positioning of tool holders in the automatic tool changing rack will be covered. Do not attempt to run cut files, or automatic tool changes without first reviewing and understanding this process. Failure to do so will result in damage to the machine's mechanics, hardware, tools, and spindle.

1. Parts Overview

When using the LC Plus Series machine, there are several parts to assembling a tool to be run (Fig. 8.1):

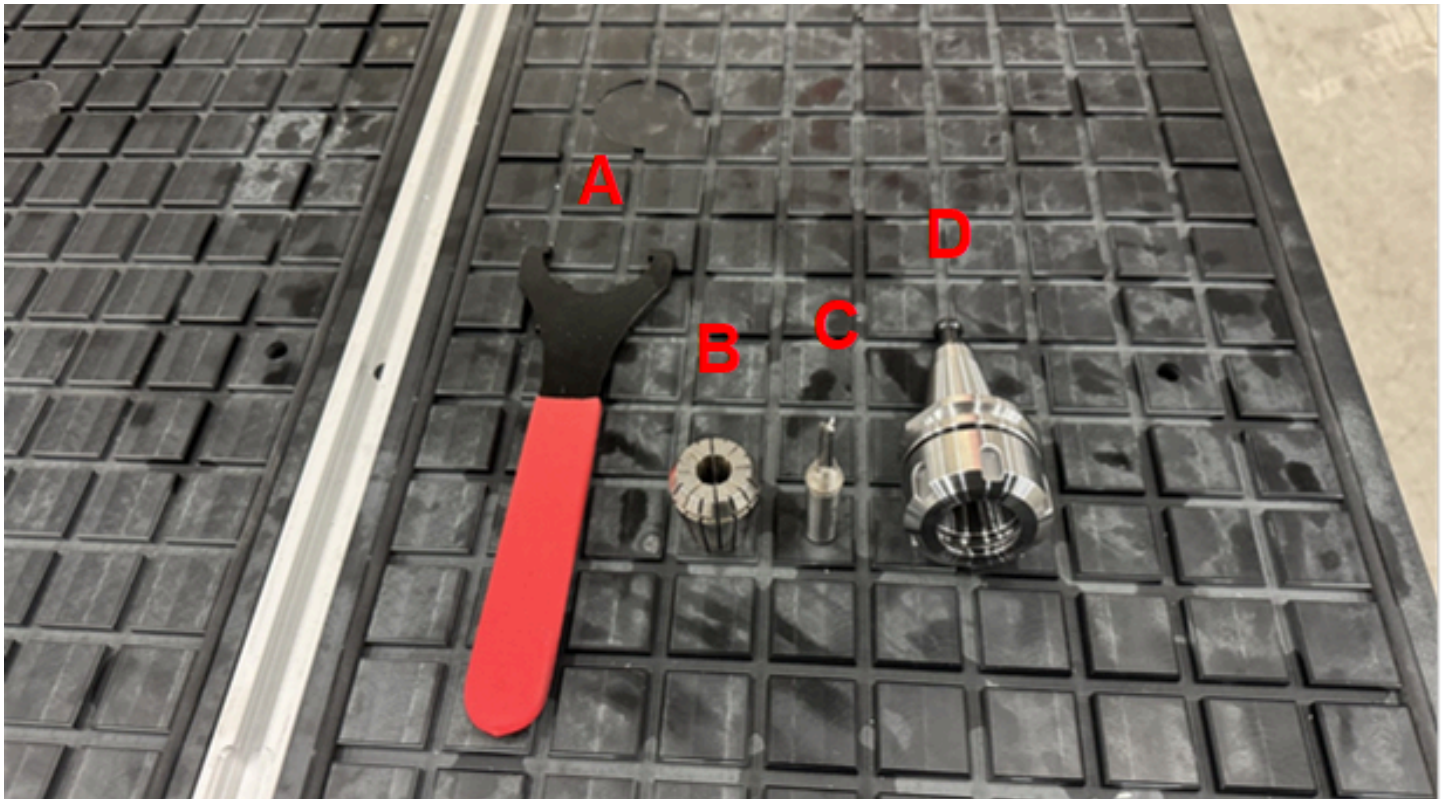


Fig 8.1

Techno CNC Systems

29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

TOOLING AND COLLETING

A. ER32 Collet Wrench: This is a wrench specifically designed for use on tool holders that utilize ER32 collets. This is the standard for the LC Plus Series machine. Techno provides the wrench with any LC Plus Series machine purchased. For information regarding replacements, please see the “Parts and Accessories” section of the manual.

B. ER32 Collet: Collets are the spring steel devices that clamp around the cutting tool inside of the tool holder as you tighten down on the collet nut. These tools are considered consumables and should be replaced annually when working in a production environment. Failure to do so may result in tools coming loose during cutting, broken tools, damage to the vacuum table, or serious injury. These collets come in all standard sizes for most router bits. For replacements, please see the “Parts and Accessories” section of the manual.

C. Router Bit: The router bit is what makes all the magic happen during the cutting process. There are endless options for cutting tools, all specifically aimed around cutting a particular type of material. For cutting tool recommendations based on application, please contact one of our support engineers at 631-648-7481 or visit our online Techno Shop.

D. ISO30 Tool Holder: This is the device that is loaded into the LC Plus Series’ automatic tool changing spindle. Collets and router bits are tightly secured into these tool holders, allowing for the machine to drop off tools in the tool changing rack and pick up others for multi-step cutting files. The ISO 30 tool holder has a few components associated with it. At the top of the conical shape is our pull stud. These are threaded into the top of the tool holder body and should be secured tight. Always ensure the pull stud is tight before using the machine. At the bottom of the conical shape is going to be the threaded collet nut. This nut comes on and off the tool holder when changing out collets. For additional, or replacement tool holders, please see the “Parts and Accessories” section of the manual.

2. Loading Collets and Tools

To begin, first make sure you have an empty ISO 30 tool holder, along with a router bit, and an ER32 collet that has the same dimensioned opening as the shank diameter of the router bit.

a. Unthread the collet nut from the tool holder and insert the large face of the collet into the tool holder, pushing down to ensure it is secured (Fig. 8.2). You should be able to flip the assembly upside down while the collet remains secure. If the collet falls out, re-secure and check.

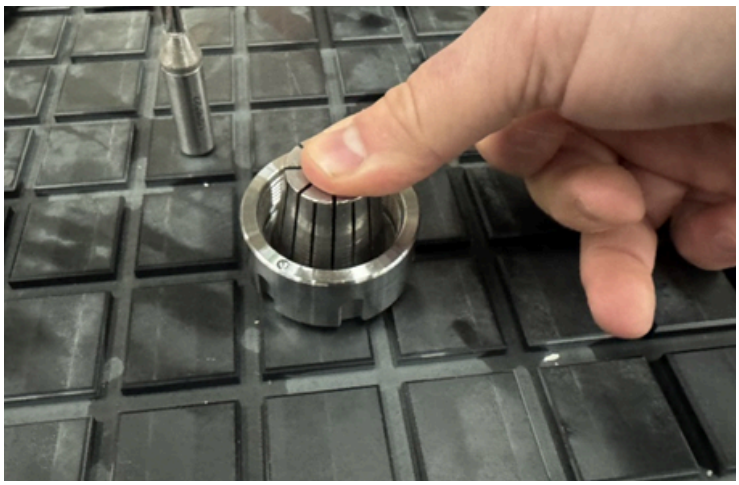


Fig 8.2

Techno CNC Systems

29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

TOOLING AND COLLETING

b. Begin threading the assembled collet and collet nut onto the ISO30 tool holder body. Once a few threads have been establish, carefully install the shank of the router bit so that the collet is positioned about 0.25” – 0.375” away from the cutting edge on the router bit. Once in position, finish hand tightening the assembly (Fig. 8.3).



Fig 8.3

c. Bring the assembly and the ER32 collet wrench to the tool vice located on the back side of the machine frame. Insert the tool holder, cutter side up, on the vice (Fig. 8.4). Make sure to line up the flat edges of the tool holder with the flat edges of the vice (Fig 8.6). Once secured, engage the ER32 collet wrench with the collet nut but sliding the wrench over top of the assembly and coming down on the grooves provided with the collet nut (Fig. 8.5). Tighten the collet nut with the wrench to secure the tool inside the tool holder.

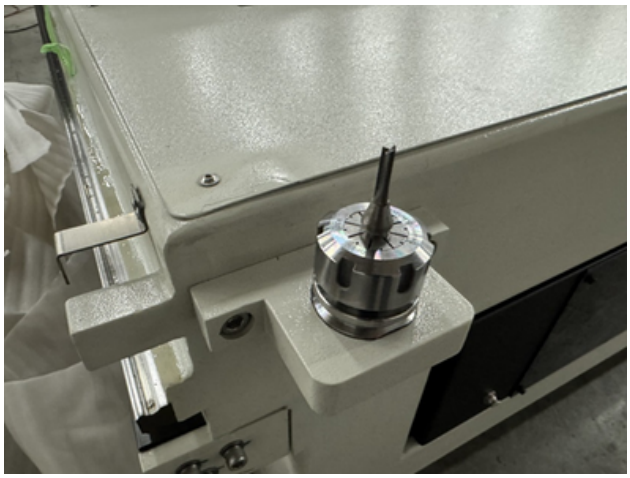


Fig 8.4

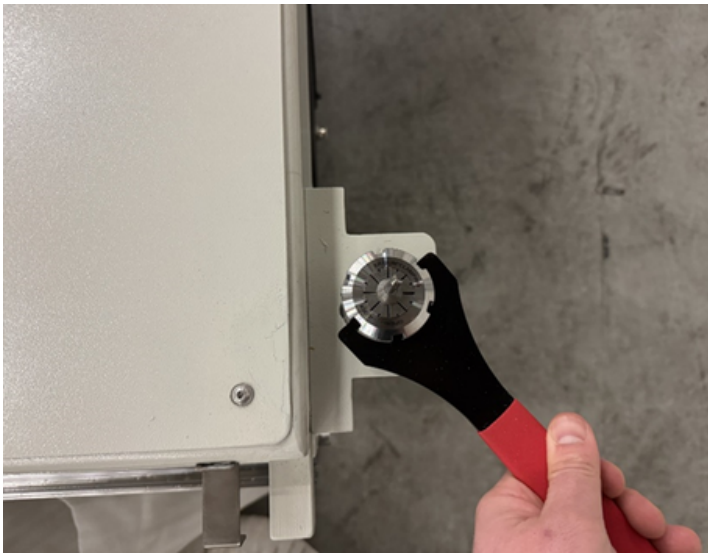


Fig 8.5



Fig 8.6

29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

TOOLING AND COLLETING

d. Once the tool is tight and secure inside its tool holder, load the tool into the spindle or the tool change rack.

- To load a tool into the spindle, press the RED button located on the spindle harness, opening the spindle chuck. A stream of air will be released from the spindle cavity. Insert the tool while simultaneously keeping the redbutton pressed. Once the tool is inserted all the way into the spindle cavity, release the red button and the spindle chuck will grab hold of the pull stud and secure it tightly into the spindle. Once the process is completed, give the tool a small tug to ensure there is not any backlash or play in the tool. If there is, the machine will not run properly and will result in damage or serious injury.

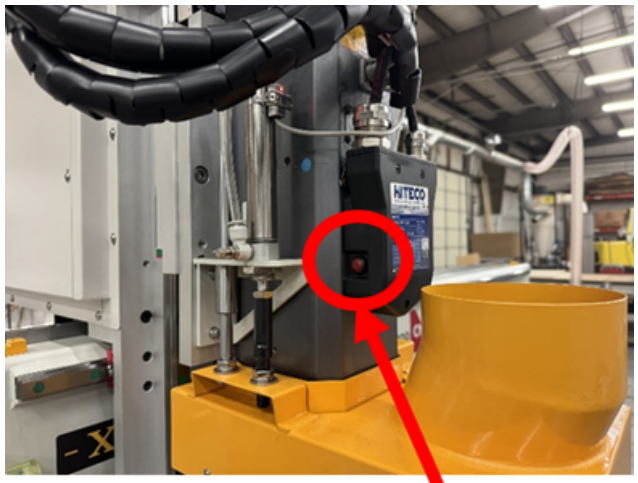


Fig 8.7

Press and hold the red spindle chuck control button. While button is pressed in, the chuck is open. When it is released, the chuck will close.



Fig 8.8

While pressing the chuck control button, insert the ISO 30 tool holder into the spindle. Release the chuck control button once the tool is fully insert into the chuck.



Fig 8.9

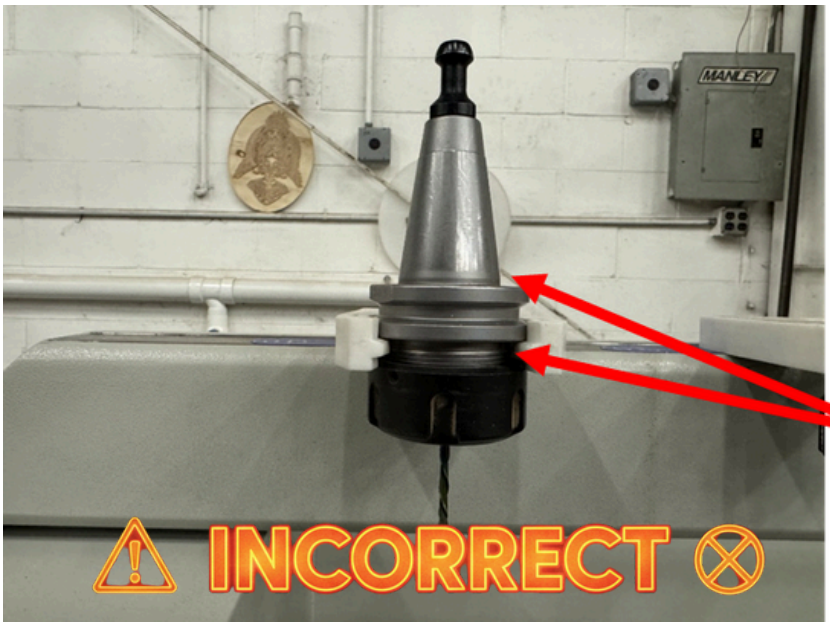
When the tool is loaded, there should not be any gap between the chuck opening and the top of the tool. There should also be no play in the tool, only rotational motion.

Techno CNC Systems

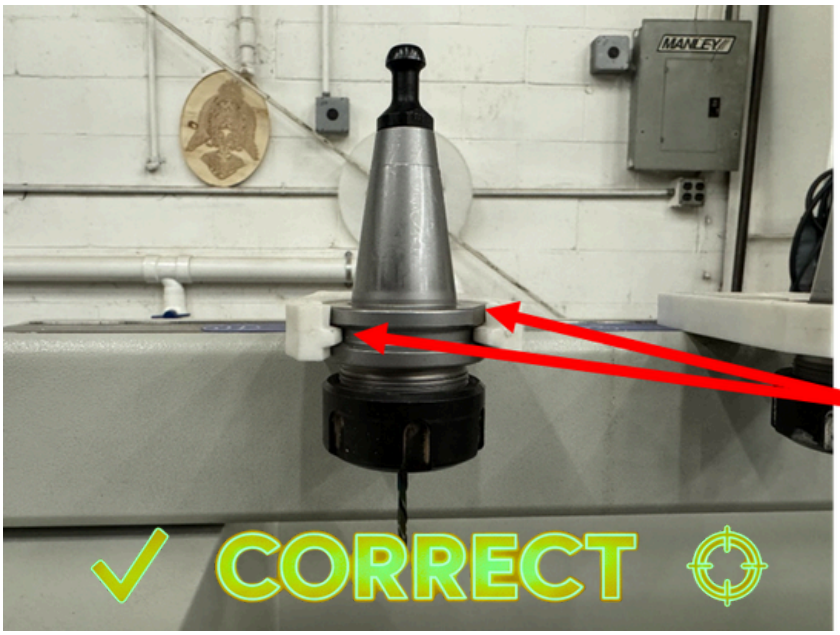
29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

TOOLING AND COLLETING

- When loading a tool into the tool clip, line up the female notched section of the tool holder with the mating notched section of the white tool clip. Rotate and pull the tool back and into the tool clip. When the tool holder is loaded into the clip, the top of the notched section of the tool holder should be flush with the top of the clip. You do not want the threaded part of the tool holder to be sitting on the clip. This is incorrect. Failure to load a tool correctly into a tool clip will result in damage to the machine, tools, tool clips, and/or spindle.



This is the incorrect way to mount a tool into the tool rack. The clip is encapsulating the threaded portion of the tool holder, and the top of the tool holder is sitting above the top of the tool clip. If the machine were to perform a tool change, damage would be imminent.



This is the correct way to load a tool into the tool clips. The top of the tool holder is flush with the top of the tool clip, and the notched section of the tool holder mates perfectly with the notched section of the clip.

LC PLUS
CONTROL INTERFACE CHEAT SHEET

Techno CNC Systems

29 Trade Zone Drive
Ronkonkoma, NY 11779
Email: technosales@technocnc.com
Tel: 631-648-7481

SECTION 9 – CONTROL INTERFACE

This section defines all the button controls on the LC Plus Series control interface. Refer to this section as you continue reading through this manual to reference any buttons, or commands being mentioned.

Upon opening the software. You will be prompted to home the machine axes immediately.

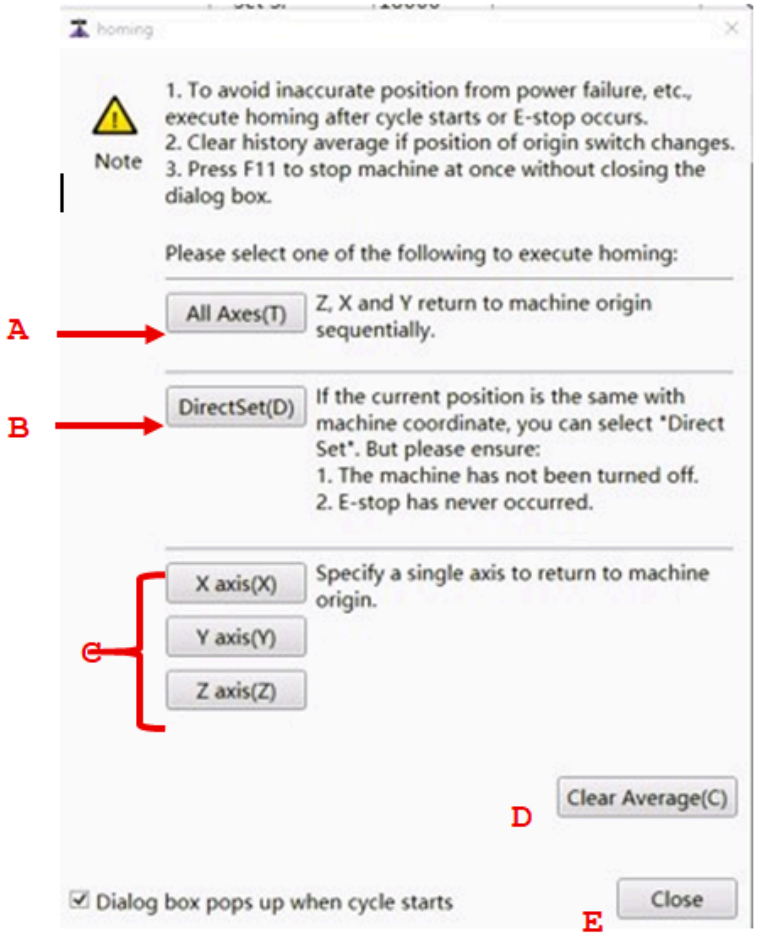


Fig. 9.1

The homing box is a unique feature of the LC Plus machine (Fig. 9.1). This dialogue box appears automatically once the machine has been powered on, and the interface is opened on the PC. Homing the machine is a must during start up so that the machine controller can accurately determine its position on the table before conducting tool changes or running jobs. Using this feature, the operator is able to home all the axes simultaneously or individually for troubleshooting purposes. The labels defined below will go over how each button functions on the homing screen.

A. The button labeled “All Axes(T)” will home each of the axes simultaneously. First, the Z-axis is brought up to its highest position followed by both X and Y, which travels to the front left corner of the machine. This is the negative limit for both axes. After selecting this button, there is a slight delay before the machine proceeds to home each axis. The letter “T” shown in the parentheses of the button represents the keyboard shortcut to perform th function. This keyboard shortcut will only be available while on the homing screen.

B. The DirectSet(D) button will automatically home the machine in the position that it is when the button is selected. It is not advised to use this feature unless the machine coordinates displayed on the interface match the actual location of the machine. Selecting this feature without clearly understanding this concept will result in inaccurate work origin positions, incorrect tool stand locations, and incorrect touch pad locations which will result in damage. The letter “D” in the parenthesis represents the quick key shortcut. Select the letter “D” on the keyboard to run this function when the home screen is open.

C. These three buttons allow the operator to home out each axis individually. Once one axis is selected to be homed, it must complete the homing sequence before selecting another. Use the “X”, “Y”, or “Z” buttons on the keyboard as quick keystrokes for homing individually.

D. The Clear Average(C) button will clear out the average machine coordinate positions on the machine. The machine records its home position as it is cycled, and if there is a discrepancy in this position, the controller will prompt an error message. This feature is useful if the gantry bridge needs to be squared, or the location of the homing switch and its corresponding contact bracket have been moved/adjusted. Use the letter “C” on the keyboard to run this operation from the keyboard.

E. Once the homing sequence has been completed, the operator can close out of the homing screen, navigating to the rest of the interface.

I.Interface Layout

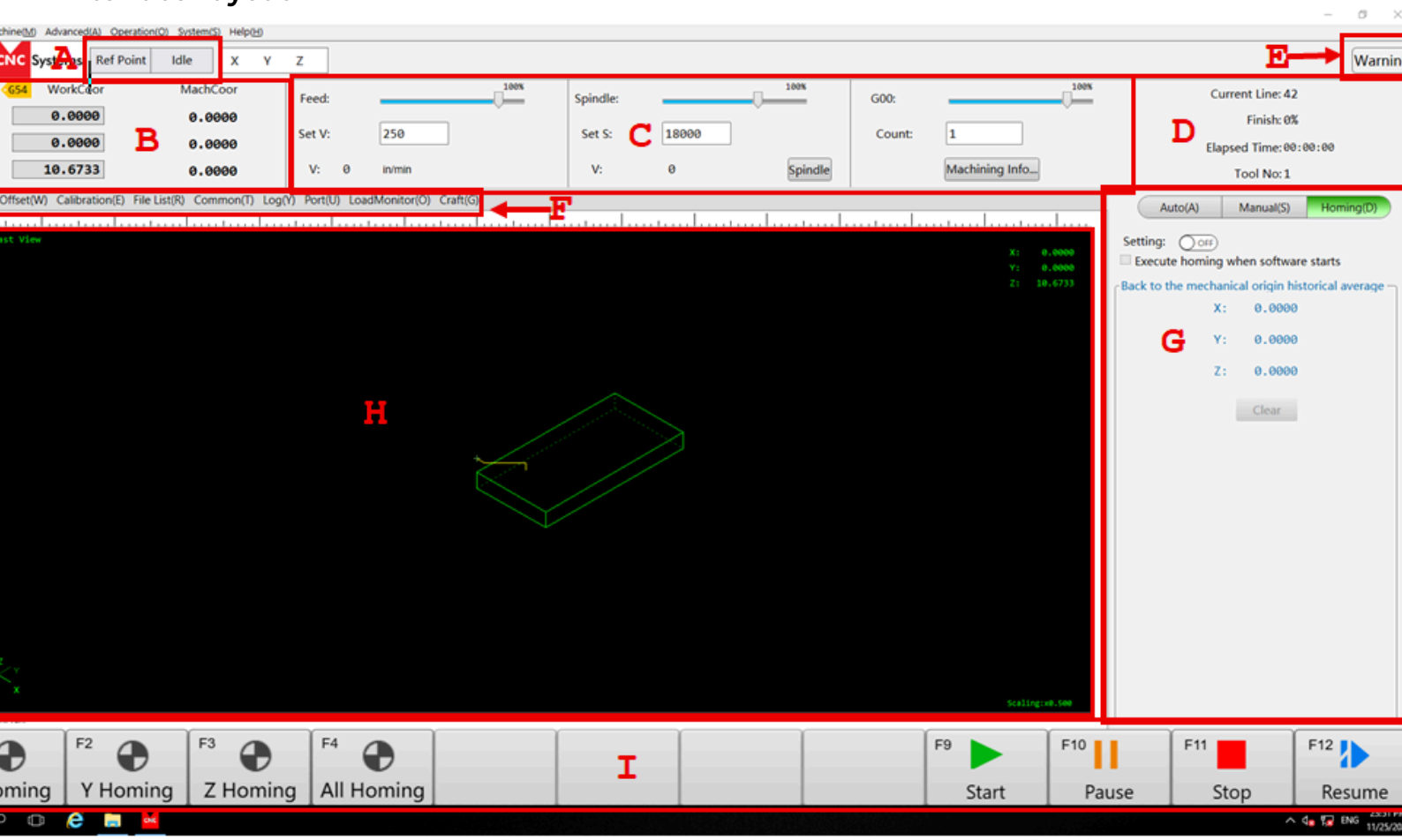


Fig. 9.2

Fig. 9.2 shows the interface as it will be displayed once the homing sequence is complete. The screen in Fig 9.2 has been labeled in sections from A – I to help the user navigate:

A. Selected control mode: This section depicts the motion control mode the machine is currently in. The indicator has two sections, in this case the left side reads ‘ref point’ and the right-side reads ‘idle’. If you change to a different control mode (section G), the new mode will be displayed on the left side. The right side will read either ‘idle’ or ‘running’ if the axes are moving or idle.

B. Live coordinates: Will depict both the absolute and machine coordinates as well as the relative coordinates. The current work offset being used by the controller is also on display. Selecting the relative coordinate boxes also allows the operator to set a new work zero position in the location the machine is currently sitting on. Work zero positions are discussed in more detail later on in section 13 (setting a WCS work origin).

C. Override Control: This section of the interface screen is called the override section. The 3 scroll bars provided allow an operator to override and change the feed rate, spindle speed, and rapid feed rate of the machine both during file operation or manual operation. The scroll bars depict a percentage of the programmed feeds and speeds. Adjusting these scroll bars will change the feeds and speeds by the product of the percentage displayed on the screen.

D. Progress section: This section tracks the machining progression of a specific job. Once a job is loaded up and the machining process is initiated, the progress can be tracked until the program is finished 100%. This section provides real-time feedback on the line #, machine elapsed time and tool #.

E. Warning: Represents the warning tab for the interface. The tab will always display “Warning” but is not relevant unless it starts to flash red or yellow. To get a full display of any warnings or a history of warnings, select this tab to open the warning dialogue box.

F. Command tabs: These are our command tabs; each tab contains a unique interface where the user can customize options. The default is the toolpath tracker (Q). Command tabs will be discussed in more detail later in this manual.

G. Control Mode Interface: The three tabs represented here are the mode tabs. Selecting Homing, Manual, or Auto, puts the machine in a mode that prepares the operator to home the machine, manually jog the machine, or auto run files on the machine respectively. Changing the control mode also changes the bottom toolbar if switched from homing to manual/auto. The control mode interface is discussed in more detail later in this manual.

H. Command tab display: The display represented in this section of the interface is the display for our command tabs. Once a command tab has been selected, the options available, or commands associated with that tab will be on display in this section of the interface. Command tabs will be discussed in more detail later in this manual.

I. Bottom toolbar: Operations include but are not limited to: loading and unloading programs, running programs through auto feed or running using step control of the handwheel. Changing the control mode also changes the bottom toolbar if switched from homing to manual/auto. The control mode interface is discussed in more detail later in this manual.

II. Control mode interface

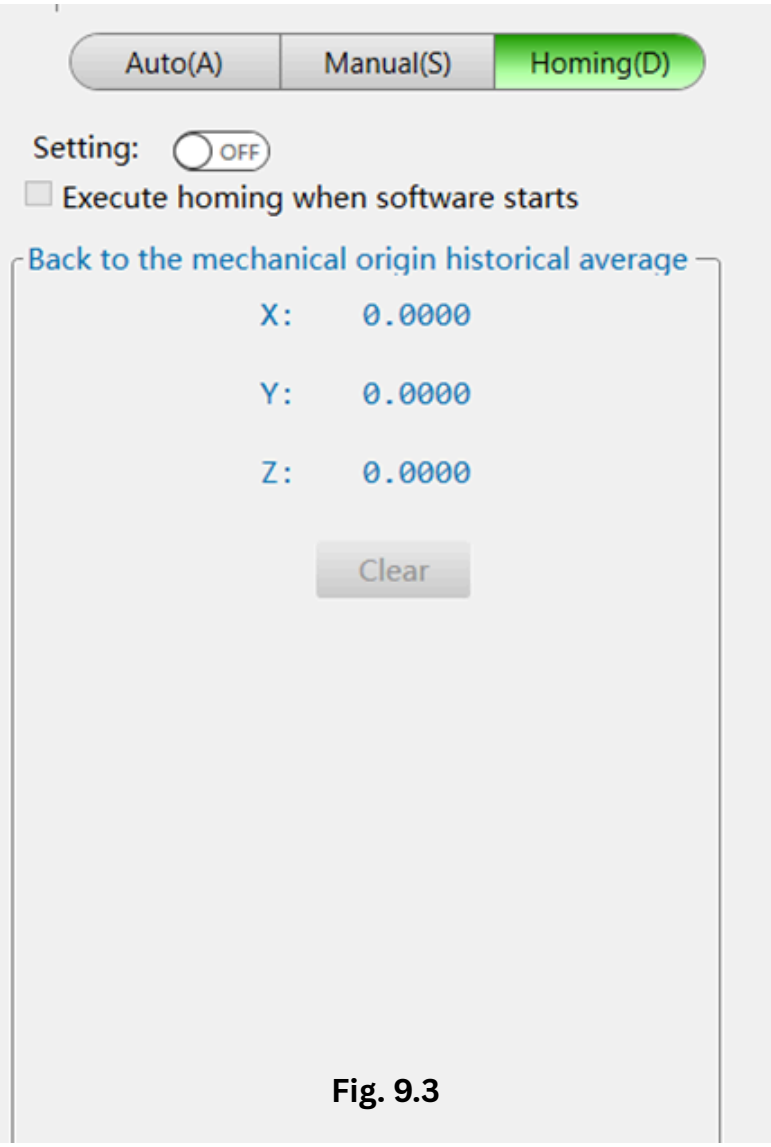


Fig. 9.3

There are three control modes: Homing, Manual and Auto. Changing the mode gives different options for control below. Homing mode (Fig. 9.3) allows the operator to re-home the machine once the initial homing window has been closed. The historical values shown on the display of homing mode represents the machine position of the homing procedure for each axis. These coordinates represent a historical average of this position. If the average changes over time, contact technical support to diagnose what mechanical components could be allowing for deviation.

(Fig. 9.4) shows the bottom toolbar when homing mode is selected. The buttons are used to control the homing mode. Use any of the four buttons to home the corresponding axis, or allalle axes. The F1 – F4 text in the upper left corner of each button represent the keyboard shortcut to run any of these homing procedures.

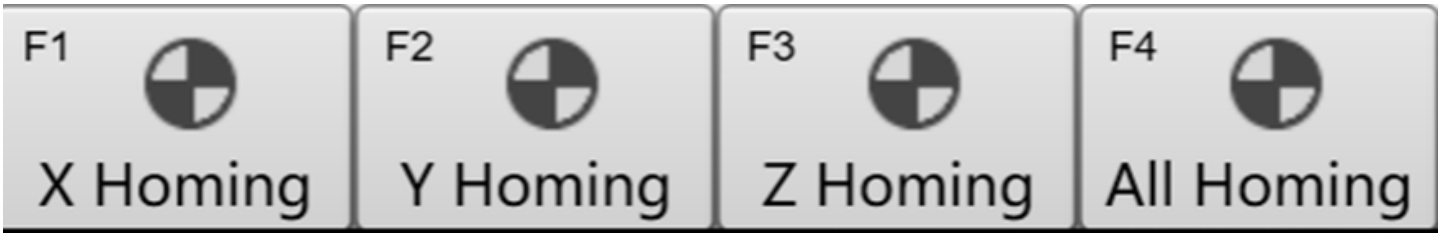


Fig. 9.4

Manual Mode

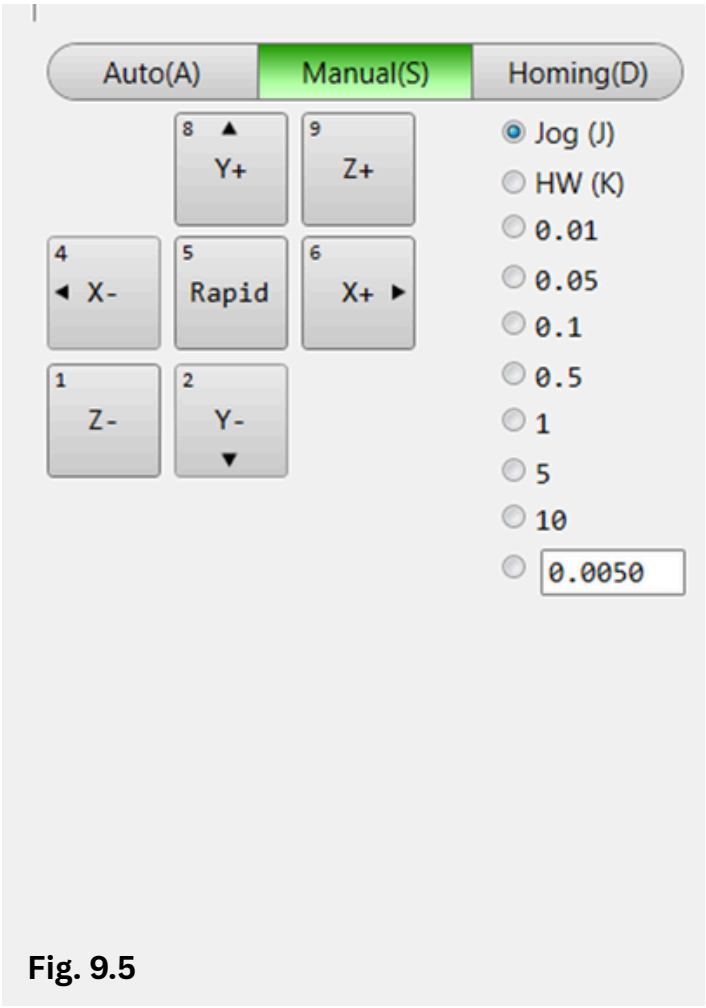


Fig. 9.5

Manual Mode (figure 9.5) Allows the operator to continuously or incrementally jog the machine in one or more axes. On the right-hand side of the screen, are bubbled options for jogging.

“Jog(J)” represents a continuous, smooth jogging motion, where the operator can continuously jog the machine in any of the axes continuously as long as one of the axis control buttons are pressed down. Once released, the machine will stop moving.

“HW(K)” represents our handwheel or MPG as defined in Section 6 of this manual. Select this bubbled option to then be able to use the handwheel to jog the machine.

The bubbled options utilizing numbers as options below the HW option. Select any of these options to incrementally move the machine in any of the axes by the value given in the option. For custom increments, select the last option and type in the desired step value. When selected, the machine will only move this incremental distance for every compression of the jog command buttons.

Manual control mode is further discussed in ‘section 11’ of this manual.

Auto/Manual Mode Additional Commands:



Fig. 9.6

The Spindle(O) button (Fig 9.6) allows the user to manually turn the spindle on and off.



Fig. 9.7

The Dustcover1(H) button (Fig. 9.7) allows the user to manually lift and lower the pneumatic dust good on the machine.

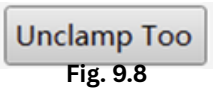


Fig. 9.8

The Unclamp Tool button (Fig. 9.8) allows the user to open and close the spindle chuck through the interface. It is not advised to use this function unless absolutely necessary, or there is an assistant to help hold the tool from dropping out of the spindle.

More Additional Commands



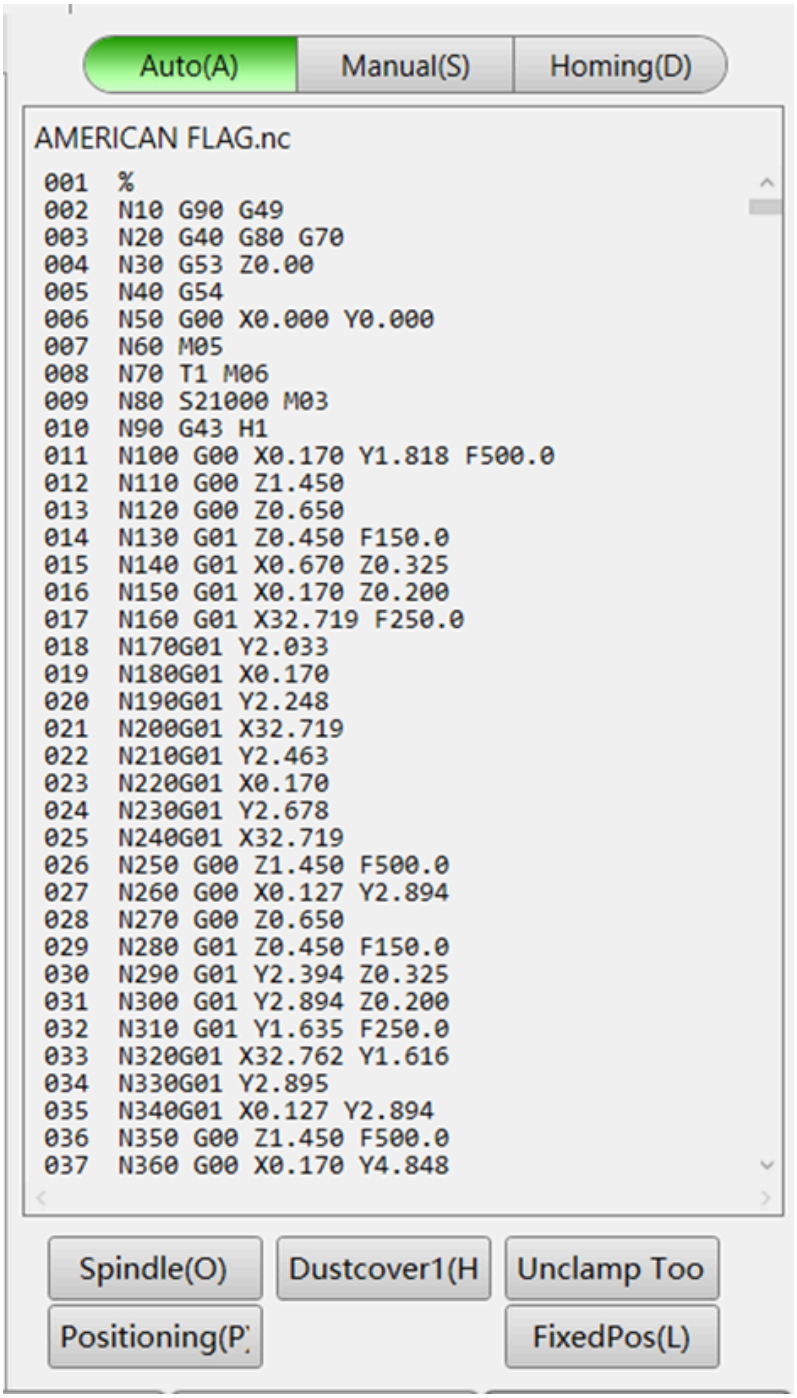
Fig. 9.9

The Positioning(P) button (Fig. 9.9) can be used to move the machine's alignment pins up and down. These pins are used to align cut stock to the table, ensuring that the x/y zero location is the same from sheet to sheet. Make sure to always move the pins down after aligning a sheet to prevent damage to the pins once the cut file begins.



Fig. 9.10

The FixedPos(L) button (Fig. 9.10) is used to send the machine to a set park position at the back of the machine. This is helpful for moving the machine out of the way without the need to manually jog it once a cut file has been completed.



Auto(A) mode(Fig. 9.11) is used for displaying a loaded gcode file. The files on display can be scrolled through. Additionally, lines in the file can be selected and "skipped to" in the event the user has the need to start a job from a particular line.

Auto/Manual Mode Bottom Toolbar



Fig. 9.12

Figure 9.12 shows the bottom toolbar when manual/auto control mode is selected. Below, each operation is defined:



Fig. 9.13

Figure 9.12 shows the bottom toolbar when manual/auto control mode is selected. Below, each operation is defined:



Fig. 9.14

The “All Clear” button (Fig. 9.14) will clear out the current work offset coordinates for all active axes. This means that wherever the machine is currently positioned on all axes will be the new work zero positions.



Fig. 9.15

MDI abbreviates “manual data input”. This button (Fig. 9.15) allows a user to manually input and execute g-code without needing to program and run a file. After selecting this button, a dialogue box will appear where the single line commands can be input and executed.

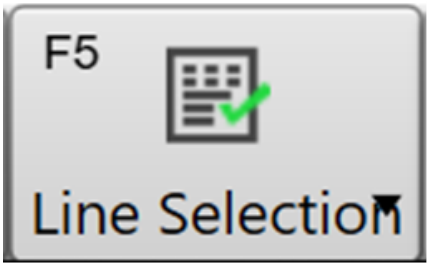


Fig. 9.16

Line Selection (Fig. 9.16) is a feature that will allow the user to skip ahead or start from a specific line in the loaded g-code program file. After using this button, from the Auto mode screen, the operator can select the specific line where the file should start from.



Fig. 9.17

HW Guide (Fig. 9.17) represents “Handwheel guide”. Using this feature, a loaded program can be “stepped” through by rotating the handwheel as opposed to the file being run start to finish completely. This feature can also be used for Para macro sequences such as tool changes.

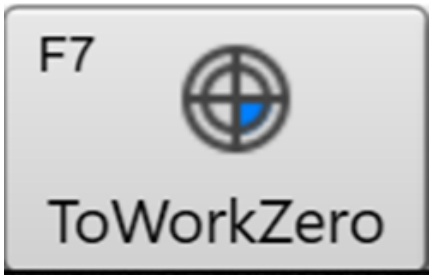


Fig. 9.18

The “ToWorkZero” button (Fig. 9.18) will bring the machine to its current work XY zero location, followed by the z-axis descending to a safe height (roughly 2.5 inches) above the set Z zero position.

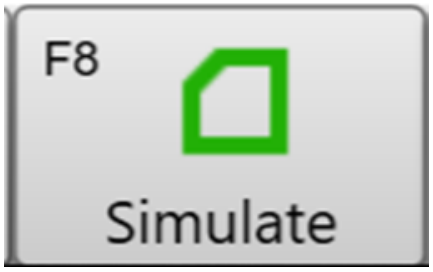


Fig. 9.19

Once a g-code file has been loaded onto the control interface, use the “Simulate” button (Fig. 9.19) to have the software quickly read through the g-code file, prompting any warnings or errors if there is any issue with the way the program is written. Once simulated, the cut path and rapid paths will be displayed on the Track command tab.



Fig. 9.20

The set of buttons depicted in Fig. 9.20 are the commands to start, pause, stop, or resume a programmed g-code file once it has been loaded onto the interface. The Resume feature is used after the pause command has been utilized. The stop feature will reset the file, requiring the user to utilize the Line Selection command (Fig. 9.16) to resume a file from a specific location.

III. Command Tabs

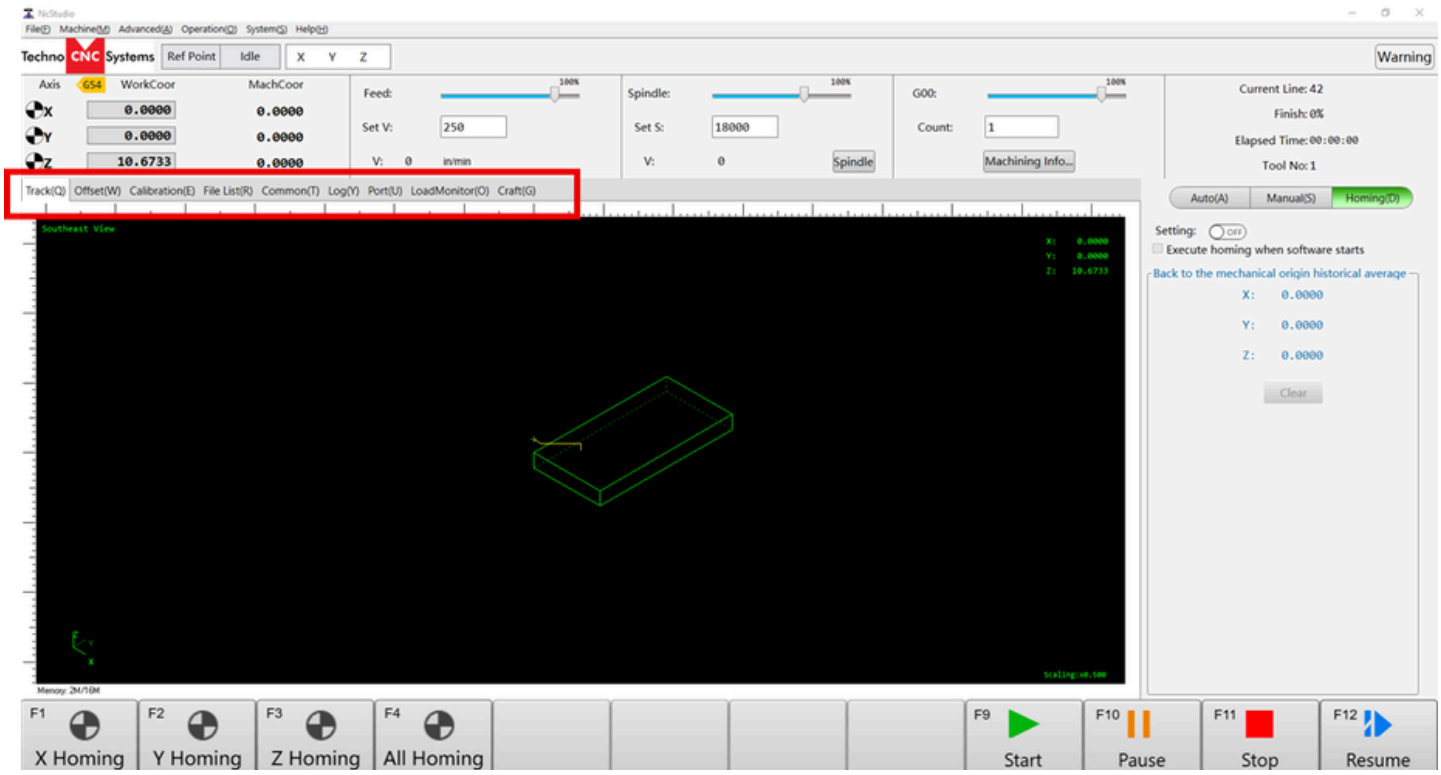


Fig. 9.21

The command tabs (highlighted in Fig. 9.21) allow the user to conduct various functions that are featured on the LC Plus machine. These features include automatic tool changes, file previews, tool measurements, automatic zeroing, and load monitoring.

The individual command tabs are discussed below:

Track (Q)

The first command tab, as shown in Fig. 9.21 is the Track tab. The track tab shows a 3D isometric preview toolpath of the machine movements. This will allow the user to see the tool movement in real time when a program is running. To change the view of the previewer, or to clear the previewer of any tracked movements or files, right click on the screen and select one of the features to do so.

Offset (W)

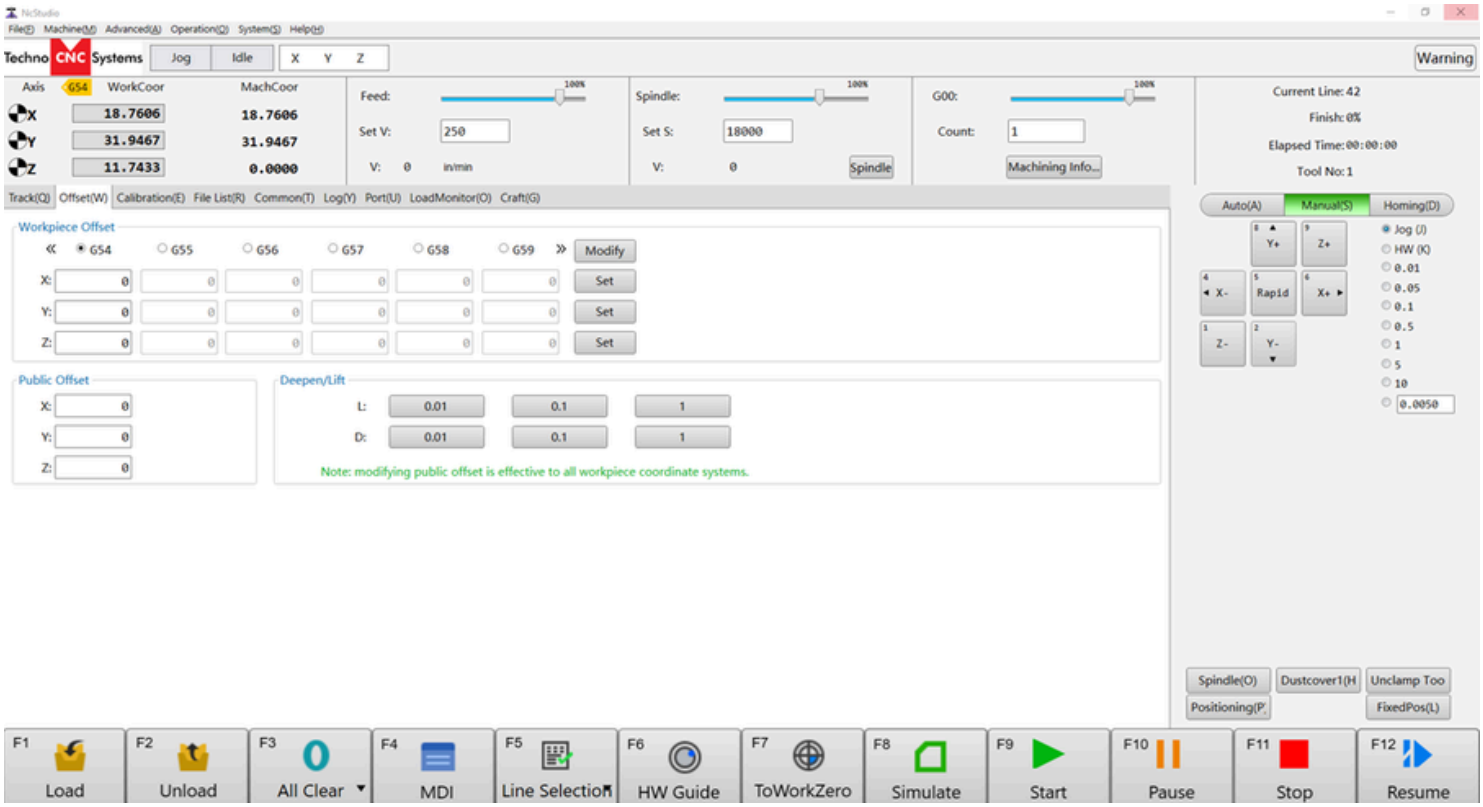


Fig. 9.22

Fig. 9.22 shows the offset command tab. On this page, the operator is able to modify both work piece offsets and public offsets numerically and incrementally.

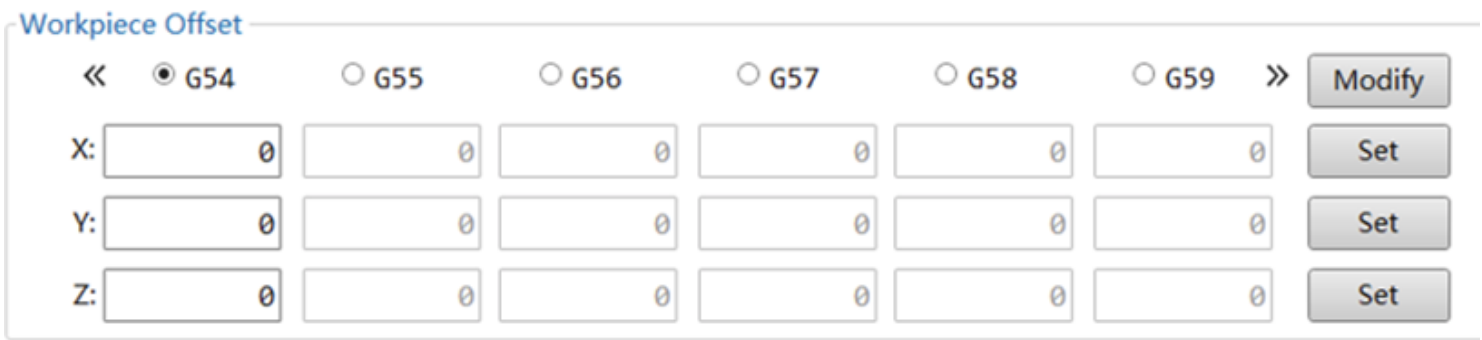


Fig. 9.23

The workpiece offset represents the X, Y, and Z zero positions associated with the six individual offsets allowed for this controller. To modify, or activate an offset, select the bubbled option at the top for offsets G54 – G59, then select “Modify”.
After selection, the bubbled in offset will be editable.
Click on the dialogue box for the corresponding axis to be edited, then input the new machine coordinate value for the work offset zero position.
When finished editing, select “Set” to save the edits.
Be advised, this should only be done if needed to modify the current work offset by a small amount. It is not advised to save a fresh work offset by plugging numbers in without fully understanding where on the table these coordinates are located.



Fig. 9.24



Fig. 9.25

A Public offset (Fig. 9.24) represents an offset from the machine coordinates zero position. Changing a value in this section of the screen will directly affect all work offset positions, tool change positions, or touch probe locations. It is not advised to use this function unless advised by a Techno CNC technician.

The Deepen/Lift dialogue (Fig. 9.25) represents a live depth adjustment feature on this controller. The row labeled “L” will adjust the local, or work offset position of a piece, while the “D” represents a change to the public offset of the z-axis. It is not advised to change the public offset unless instructed by a Techno CNC technician. Any live adjustments to a programmed file should simply be done using the “L” row of values.

Calibration (E)

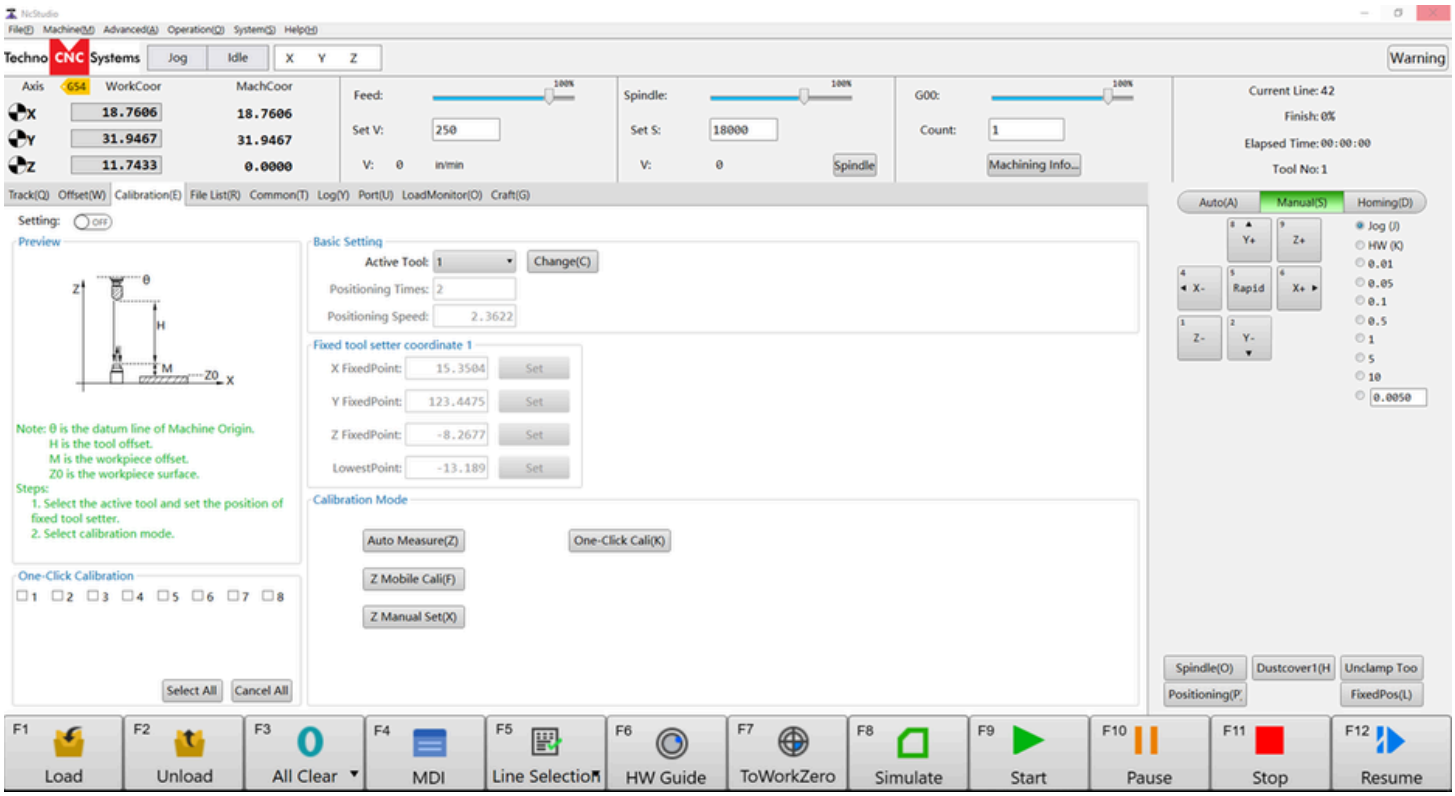


Fig. 9.26

The calibration command tab (Fig. 9.26) is used for all tool change functions, measuring the lengths of tools, and setting z-axis work offsets, both automatically and manually. Navigate to this command tab to perform any of these functions.

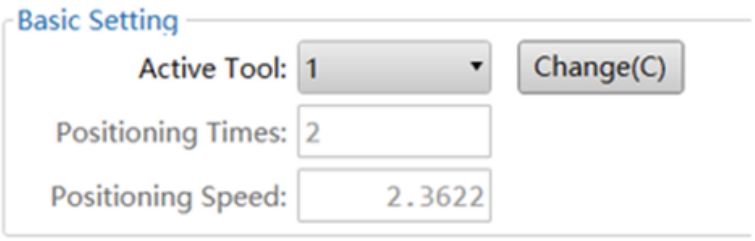


Fig. 9.27



Fig. 9.28

The basic setting portion of the calibration screen (Fig. 9.27) is where the tool changes will be executed. Use the drop down menu next to “Active Tool” to select a tool number to change to. Once the tool number is selected, press the “Change(C)” button to execute the ATC (automatic tool change).

Calibration Mode (Fig. 9.28). Is where tools can be measured or zeroed. Auto measure will measure a tool using the fixed touch probe at the back of the machine. Z Mobile Cali and Z Manual Set are for setting a Z zero position. One-Click Cali is used to execute multiple auto measures based on the selection from the One-Click Cali section of the screen (Fig. 9.29).

Fig. 9.29

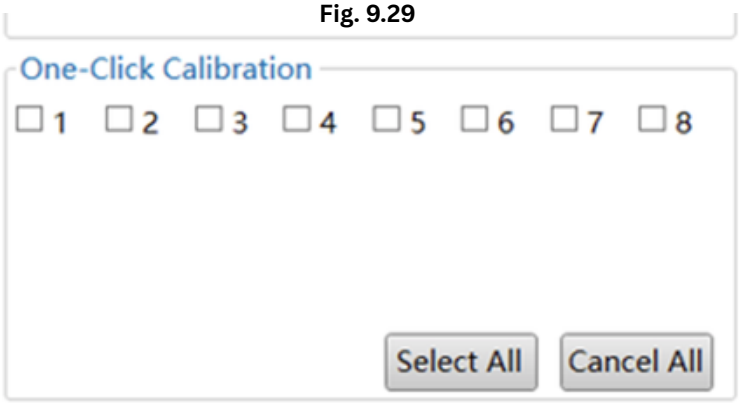
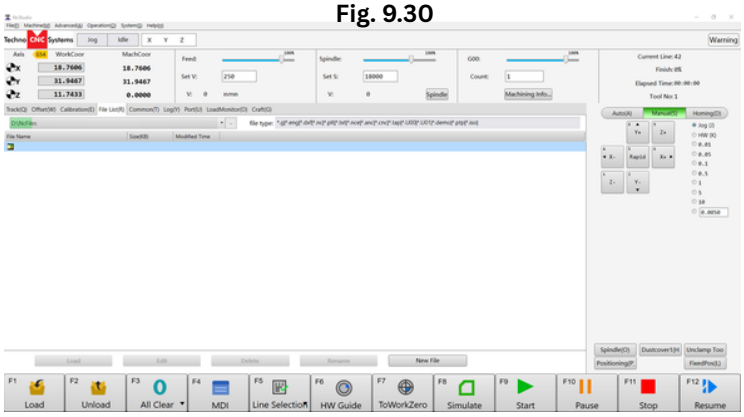


Fig. 9.30



One-Click Calibration (Fig. 9.29) allows the operator to select multiple tools to be measured automatically. Check the boxes next to the tools needing to be measured. Once selected, click the One-Click Cali button from the calibration mode section of the screen (Fig. 9.28). The machine will then automatically measure and change each tool until it has completed the full tool list selection.

The File List tab (Fig. 9.30) is used for loading a file or multiple files to a cut list. Files that have been loaded from a separate directory in the past are also going to be included in this list. To re-run any of these files, simply select them, then press the load button (Fig. 9.13).

The remaining tabs in the command tabs list are used mostly by technicians for setting parameters, determining polarity of inputs, and other tools to help trouble shoot machines as they begin to age. For this reason, these tabs will only be included in a separate appendix at the end of the manual. It is not advised to try and edit any of these parameters in these tabs. Failure to heed this warning will result in incorrect machine operations, potential damage, and guaranteed down time.

LC PLUS
START UP AND HOMING

Start Up and Homing

The LC Plus Series machine is a super easy to use, low maintenance machine. This short section reviews the proper start up procedure for the machine and proper homing to ensure accuracy and repeatability.

Powering On the Controller Cabinet:

1. Before reading through this section, make sure that all safety, power, and air requirements have been met. For more information, please re-visit the safety, and power and air requirements sections of this manual.

2. Turn the power disconnect located on the front door of the controller cabinet (Fig. 10.1) to the ON position (Fig. 10.2).



Fig. 10.1



Fig. 10.2

3. With the power disconnect in the on position, power is being distributed through the controller cabinet. Ensure all E-stop buttons are disengaged by rotating the buttons clockwise. Press the green power on button on the controller cabinet (Fig. 10.3). The button will illuminate a green light, and the machine will begin to boot.



Press the power on button to turn the machine on

Fig. 10.3

When the system first powers on, it will take several moments for the PC to load. Once it has completed loading, the monitor will display the desktop and the control interface icon.



Double-click the Techno CNC Interface icon to open the control interface.

Fig. 10.4

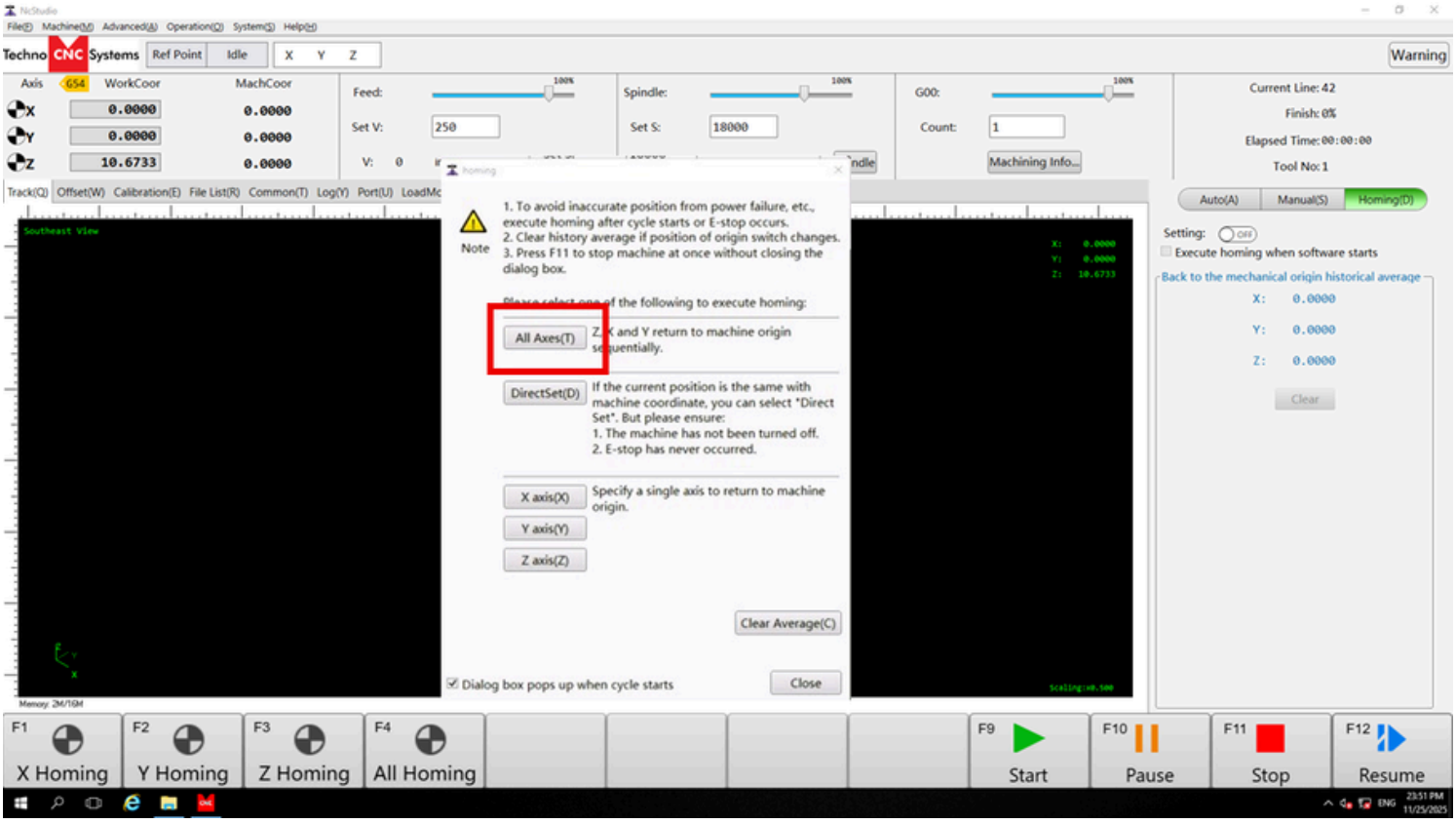


Fig. 10.5

1. The interface will take several moments to load and open up. Once it does open, a dialogue prompt will automatically appear, telling the operator the machine needs to be homed prior to operation (Fig. 10.5). Read through the instructions provided on this dialogue box to become familiar with the purpose of each command provided. To home all axes on the machine, select the “All Axes(T)” button towards the top. After selecting the “All Axes(T)” button, the machine will first raise the z-axis until it has found its limit switch, then the X and Y-axes simultaneously. If the machine is equipped with a 4th axis rotary, the rotary axis will be homed, after the x-axis, but before the y-axis.

LC PLUS
MOTION CONTROL & JOG MENU

Motion Control and Jog Menu

The LC Plus Series controller gives the operator continuous motion control over its multiple axes and allows operators to edit the speeds and distances the machine can travel. This section covers these motion control functions and how to navigate the various manual motion control functions.

The LC Plus Series machine is equipped standard with 3 independent axes, X, Y, and Z. As depicted in Fig. 11.1, the machine's X axis runs along the short length of the table, parallel with the gantry bridge. The Y-axis runs the longer length of the table, and the Z-axis drives the spindle motor up and down, perpendicular to the gantry bridge.

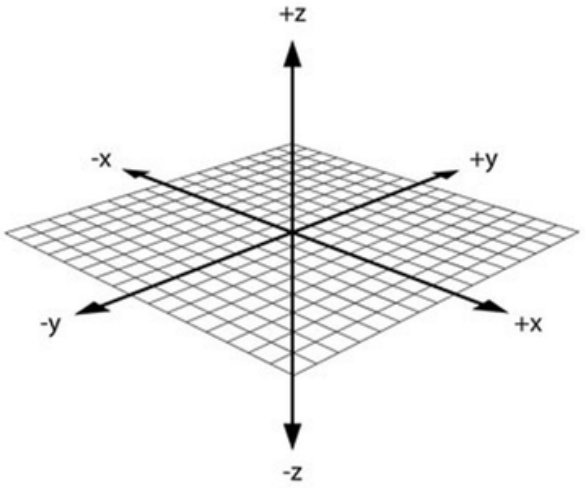


Fig. 11.1

On the right side of the interface, select the Manual tab, to put the machine in a manual jog mode (Fig. 11.2).
Once in manual mode the user can press and hold any of the axis polarity commands to move the machine continuously in any direction.
In the top left corner of each button, there is a number depicted. This number represents the shortcut key that can be used on the keyboard to move the machine in all axes. When jogging the machine manually, the default jog speed will tend to be a bit slow. To increase the speed to a max jog speed, press and hold the "Rapid" or #5 button on the keyboard to move the machine at top speed.

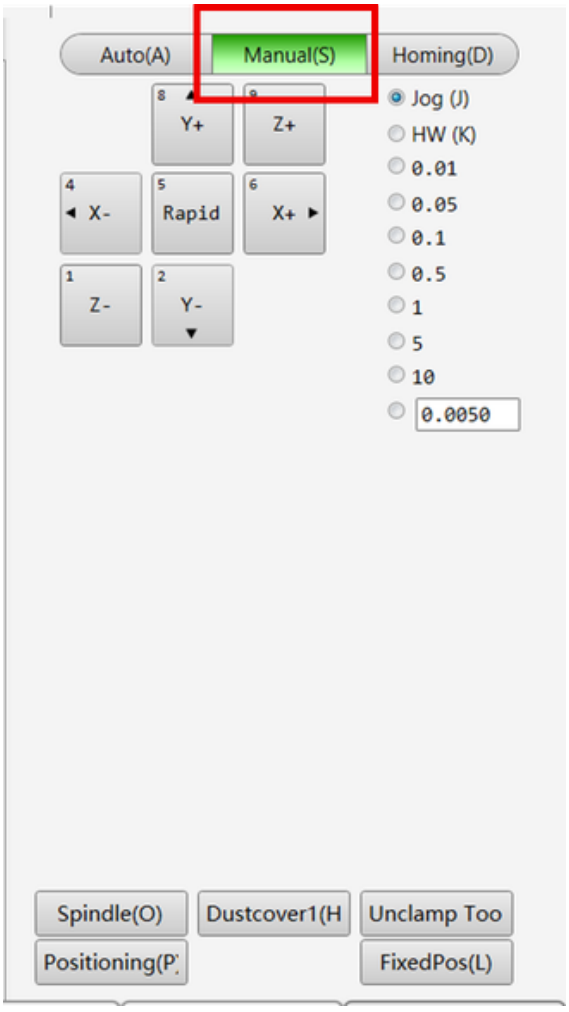


Fig. 11.2

Motion Control and Jog Menus Continued

When the machine is set in “Jog” mode (Fig. 11.3), the operator can continuously move any of the three axes continuously and simultaneously, until the buttons are released.



Fig. 11.3

Put the machine in HW(K) mode (Fig. 11.4) to utilize the wireless multi pulse generator, or handwheel to move the machine around in any of the axes.

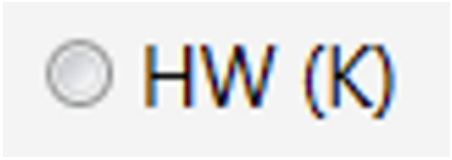


Fig. 11.4

The handwheel (Fig. 11.5) is useful for setting the machine in a precise location, such as setting an X, Y, or Z WCS position without the need to incrementally step the machine into place.
The handwheel has 3 dials to control the machine axes: the largest dial at the top, printed with numbers is used to physically move the machine. Rotating the dial clockwise will move the selected axis in the positive direction, while a counterclockwise rotation will move the axes in the negative direction.
The second dial, displaying X, Y, Z, 4, and 5 is the selection dial. Rotate this dial to the desired axis to be controlled. When the dial is in the Off position, none of the axes will move.
The last dial controls the speed in which each axis moves when rotating the motion control dial. 100x is the fastest speed, 10x is medium, and 1x is barely visible to the naked eye.



Fig. 11.5

The following selections depicted in Fig. 11.6 are various incremental distances that can be selected and function on any axis.
By selecting any of these numbers, the machine will move that exact distance for the commanded axis. For custom increments, select the last option in the selection, then select the editing box to type in the custom value to step, or incrementally move.

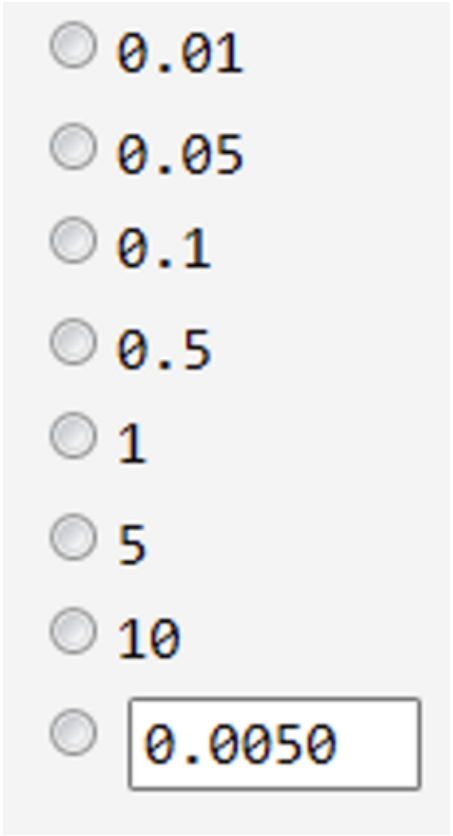
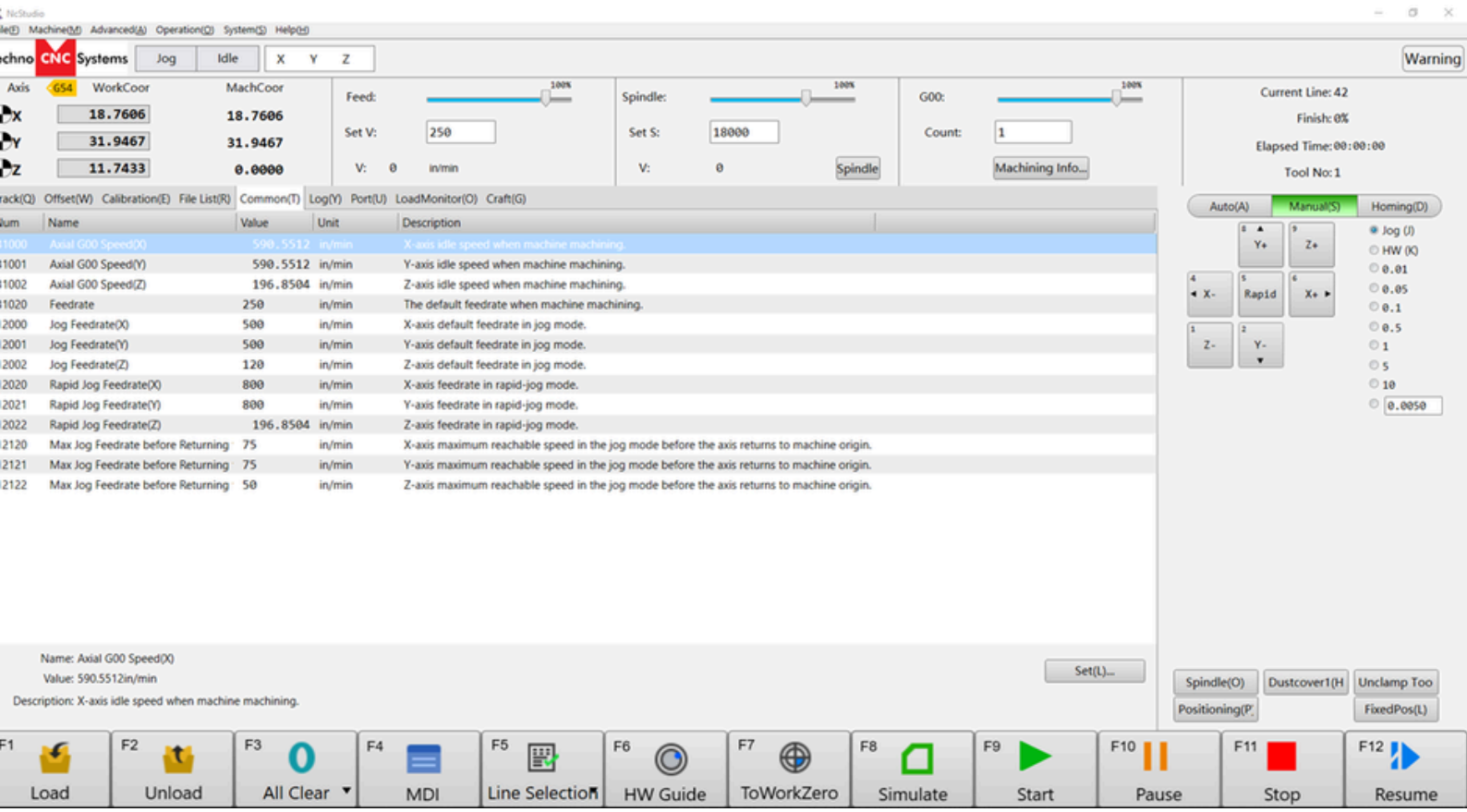


Fig. 11.6

**** Be careful when adjusting the step distance. It is always advised to change the step distance back to 0.005 after any initial change, or immediately switch back to Jog, or HW mode to avoid mistakes, or crashes. ****



The command tab labeled “Common” (Fig. 11.7) depicts multiple speed parameters pertaining to the machines manual and rapid motion controls. Double click on any of these options to change the parameter to a desired speed.

The speeds preset by Techno CNC are set with safety and comfortability in mind for a variety of operators. It is not advised to make changes to these parameters unless there is a clear understanding of how these parameters function and how they will affect the safety and operation of the machine.

For this section, the jog feed rates, and the rapid jog feed rates can be safely adjusted if desired. The jog feed rate parameters will affect the speed in which the machine moves when selecting any of the axis buttons by themselves. The rapid jog feed rate will affect the speed of the machine when the axes are being moved, with the rapid button compressed.

LC PLUS
AUTOMATIC TOOL CHANGES AND MEASUREMENTS

Section 12 – Automatic Tool Changes and Measurements

The LC Plus Series machine is equipped with a 12hp automatic tool changing spindle. This allows users to set up multi tool files and run them as a single file on their machine. The machine executes each tool path and will automatically change its own tools based on tool numbers set in the g-code program. Each tool used on the LC Plus Series tool changing rack/carousel also needs to be measured. This is so that the machine understands how far each of its tools are sticking out of the spindle and is able to locate its Z zero position without the need to recalibrate in between. Review this section to learn how to use the automatic tool changer on the LC Plus Series and how the machine automatically measures the tools set up in its tool changing rack/carousel.

1. Automatic Tool Change

Performing and automatic tool change with the LC Plus Series machine is an easy task, but there are some things to take notice of before executing the function to avoid crashes and damage to the machine:

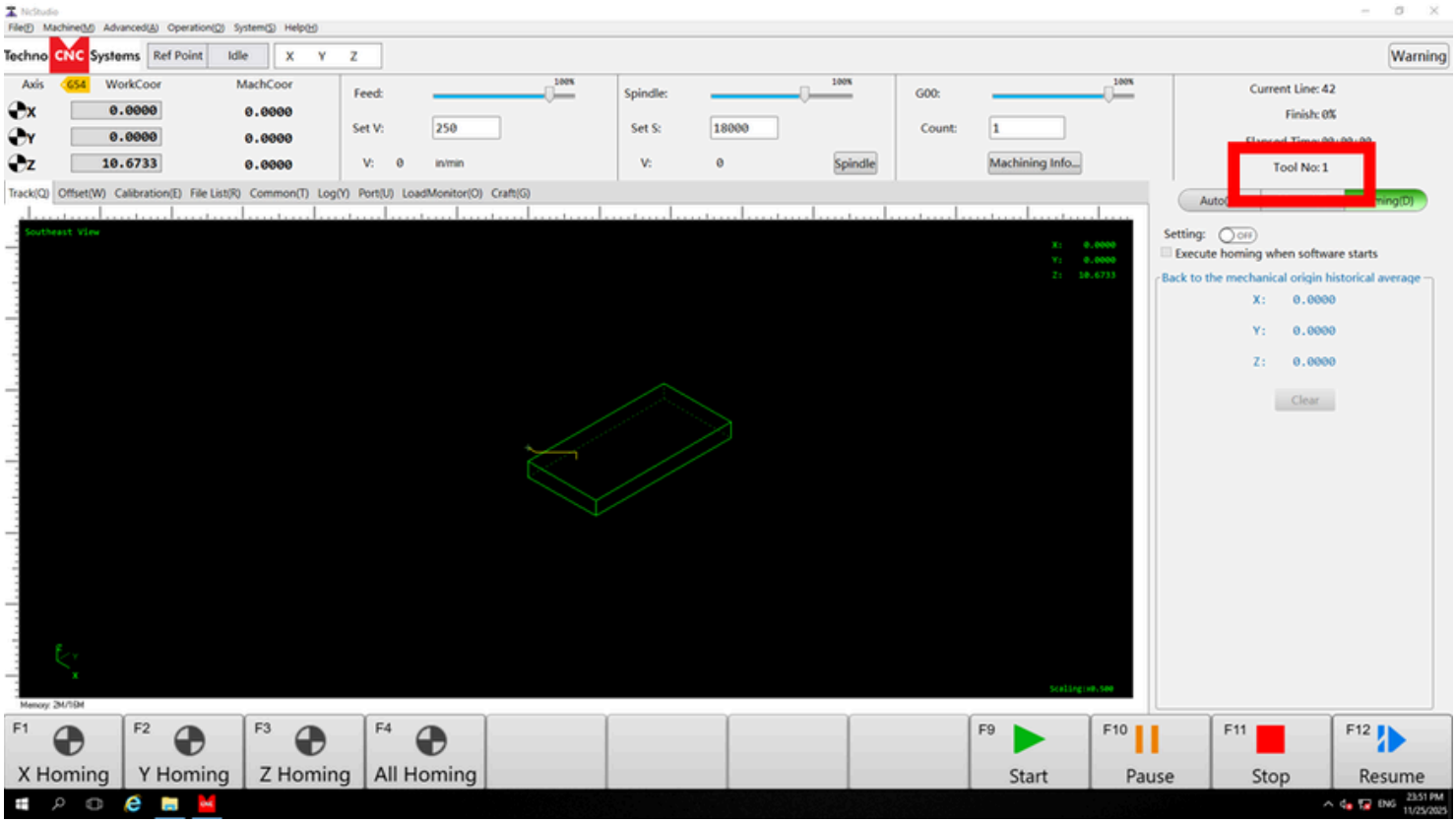


Fig. 12.1

Before conducting a tool change, make sure a tool is first loaded into the spindle, and that the tool loaded is identified with the proper tool number. To check the current tool number, look in the top right corner of the interface (Fig. 12.1) Make sure that the “Tool No.” is the correct tool number. This is the number of the tool slot that the machine will put its current tool away in, before proceeding to pick up the next tool.

Automatic Tool Changes and Measurements Continued

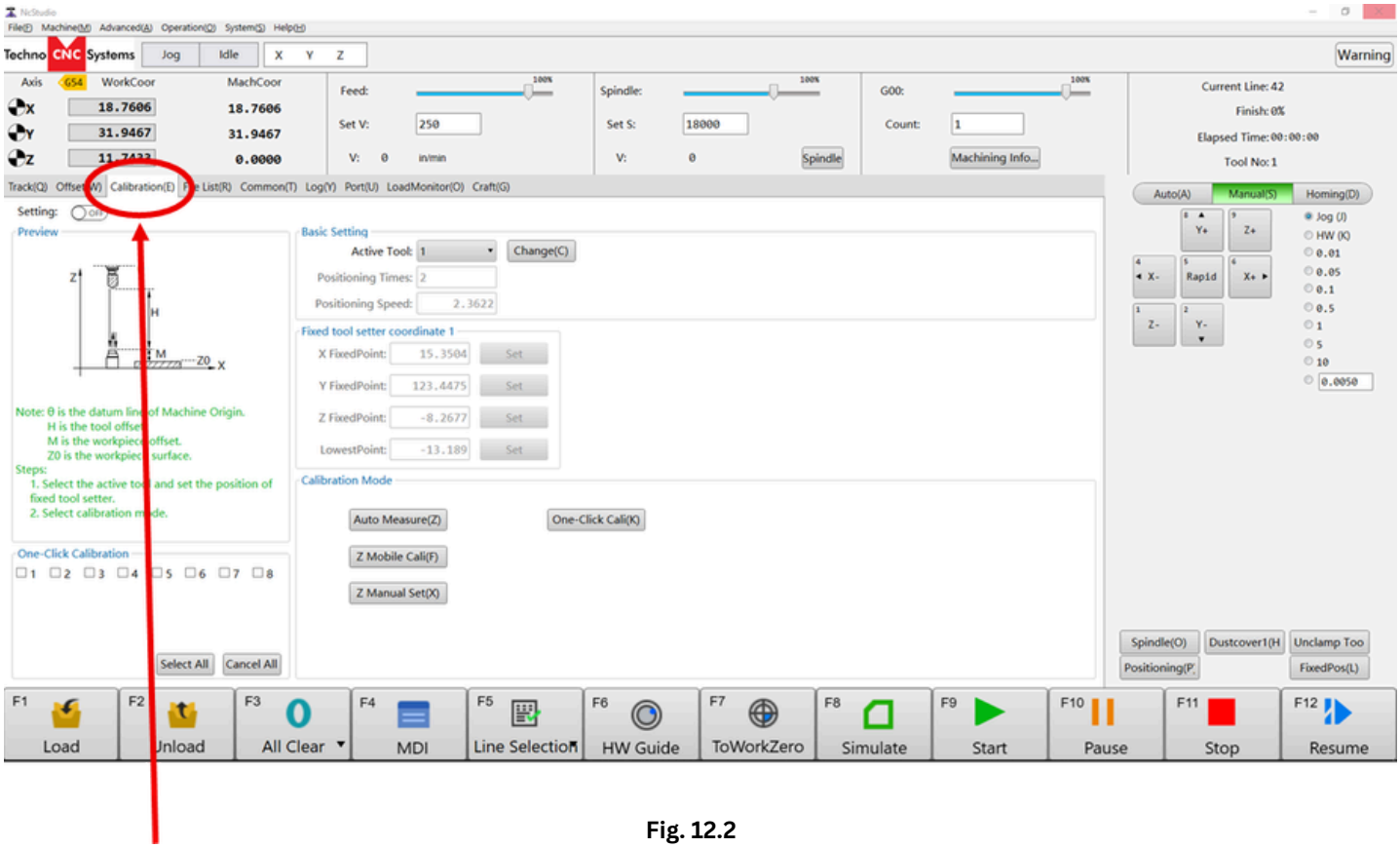


Fig. 12.2

To conduct the tool change, after ensuring the correct tool number is loaded in the spindle, select the calibration command tab (Fig. 12.2). Once in the calibration command tab, at the top of the screen, select the drop-down menu next to “Active Tool”, under the “Basic Setting” portion of the screen. In the drop-down menu, select the tool number the machine should change to. Once the desired tool number has been selected, click the “Change(C)” button to execute the tool change. The machine will then proceed to lift up in the z-axis, raise its dust hood, rapid to the back of the machine in the y-axis, followed by a rapid move in the x-axis to the currently active tool location, proceed to put the tool away in the clip, then lift up and pick up the next tool.

For example: In fig 12.2 we have a case where the spindle is currently holding tool #1 and the control interface is displaying “Tool No. 1”.
Now let’s say you want to change to tool number 3. In order to do so, first ensure that there is a tool in the number 3 slot and there is no tool in the number 1 slot. Next, change the active tool setting to ‘3’ and click change. The machine will return tool number 1 to the correct slot and pick up tool 3.
The tool change has been done successfully.

All ATC positions are pre-set by Techno CNC prior to delivery, so there is no need to perform any additional set up or set up tool stand positions. Techno CNC provides a state-of-the-art machine, that is usable right out of the box.
For more tech supplements on how to re-learn or set tool stand locations, please contact Techno CNC Systems’ support department, or check out the LC Plus Series section of the support website.

Automatic Tool Changes and Measurements Continued

2. Tool Measurements

Once you have selected a new tool for the first time, whether through an automatic tool change procedure or manually, you must now configure the new tool offset measurement.
Importance of Tool Offset Measurement:
When a zero position is established on the z-axis, the controller saves the z coordinate location for that surface, and references that coordinate while it is making depth cuts into the material. If a file uses two or more tools during the cutting operation, the tools that are used will not be protruding out of their tool holders the same distance, thus utilizing different z-axis coordinates to reach the zeroed surface. It is for this reason that tools MUST be measured. By measuring tools, the machine's controller can calculate the correct z-axis coordinate to reach the material zero position as it transitions from one tool to the next.

To enter the tool measurement menu, navigate to the calibration command tab, the same command tab used for changing tools (Fig 12.2).
To measure a single tool, with the proper tool loaded into the spindle, simply select the "Auto Measure(Z)" button located under the calibration mode section of the screen (Fig. 12.3).

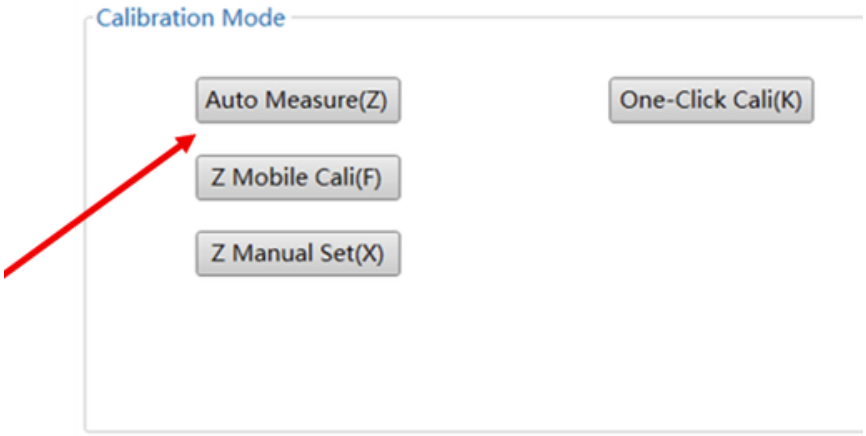


Fig. 12.3

It is important to measure all tools in the tool rack prior to setting a Z zero, or z work offset on the machine.
Continue to change tools and measure each one until all have successfully been measured.
There is a second method to automate this procedure, where multiple tools need to be measured: This feature is the "One-Click Cali" feature. From the One-Click Calibration menu on the screen (Fig. 12.4), select or check the boxes for the tools that need to be measured on the machine. You can also "Select All" if all the tools require measurements.
Once the tools have been selected, select the "One-Click Cali(K)" button under the calibration mode portion of the screen (Fig. 12.5). The machine will then proceed to measure each tool and conduct each tool change, until all the tools in the selection have been successfully measured.

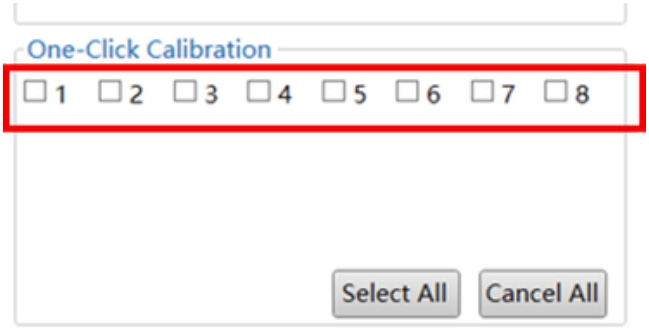


Fig. 12.4

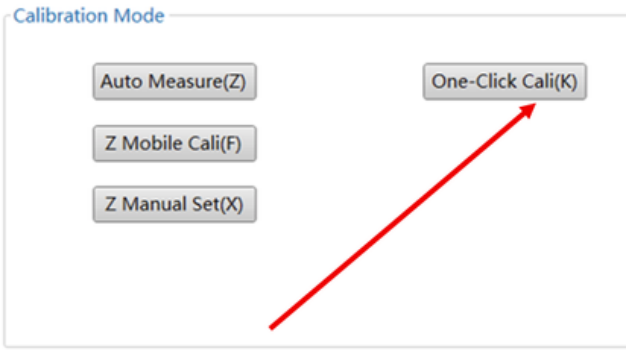


Fig. 12.5

Automatic Tool Changes and Measurements Continued

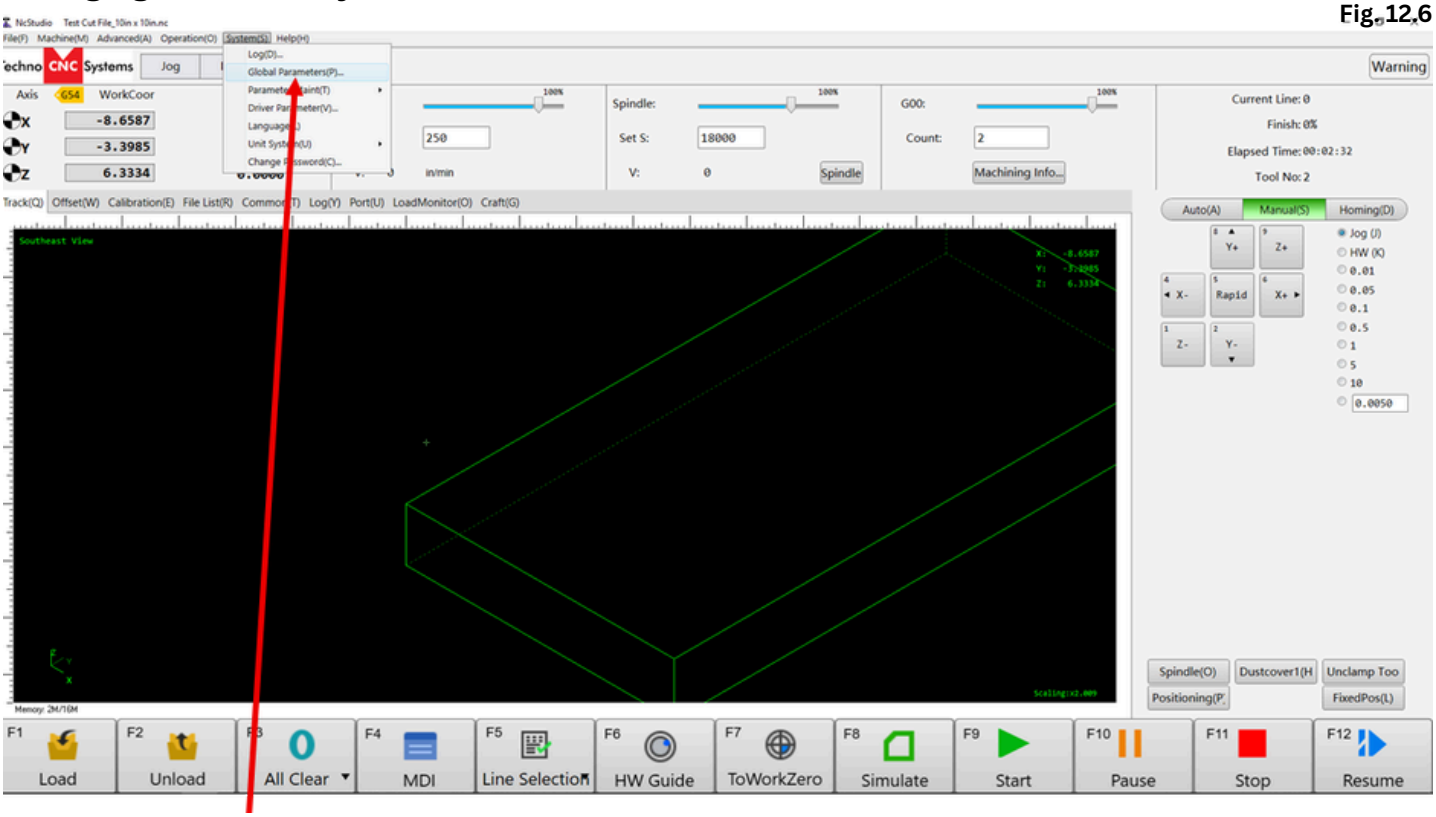
Using the information provided in this section, you should now be capable of setting up the tool change rack with necessary tooling for the application. Make sure to change to each tool, and measure them all prior to loading any jobs, or setting origin positions. Once all the tools have been measured proceed forward onto fixturing and establishing WCS locations to begin running cut files.

3. Manually Changing Tool Identity

Occasionally, the tool loaded into the spindle will not be identified correctly, where the tool number shown on the control screen does not match the empty tool slot at the back of the machine. When met with this issue, there are two options:
1. Change the tool Identity on the controller
2. Re-arrange tools on the tool change rack

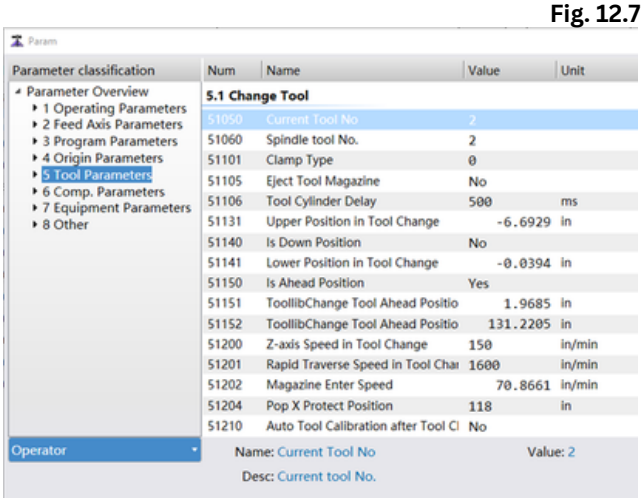
Changing Tool Identity

Fig. 12.6



At the top of the interface select “System” then Global Parameters (Fig. 12.6).

Fig. 12.7



A new menu will appear showing a variety of editable parameters (Fig. 12.7). It is not advised to edit parameters unless instructed by a Techno CNC technician. On the left side of the screen, select the “Tool Parameters” category. One of the first parameters in this category will be “Current Tool No.”. Double click on this variable, and type in the new tool number, then press enter, or OK. When the Global Parameters window is then closed, the new tool identity should be displayed above the mode tabs on the interface.

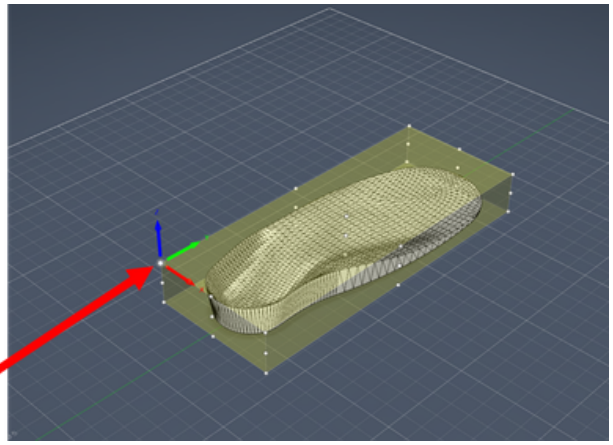
LC PLUS
SETTING A WCS ORIGIN

Setting a WCS Origin

All machines require a set origin, or zero position to work from. This section will cover the quick and easy methods for establishing these origin positions as well as setting multiple WCS positions.

1. Setting an XY WCS position:

Begin by locating the position on your substrate where your programmed WCS is located from your CAM software. Reference Fig 13.1 for an example of a WCS 0 position in CAM. In this example, the front left corner of the substrate will be used to set our XY0 position.



The position of the XY0 position

Fig. 13.1

If a full or half sized sheet of material is being set up for a job, make sure to utilize the locating pins equipped with the LC Plus Series machine (Positioning(P)), to line up and square the sheet to the table. This is also very useful tool for ensuring that the sheets being cut on the machine are in the exact same position from job to job, minimizing the amount of set up required from sheet to sheet.



Fig. 13.2

Raise Pins



Fig. 13.3

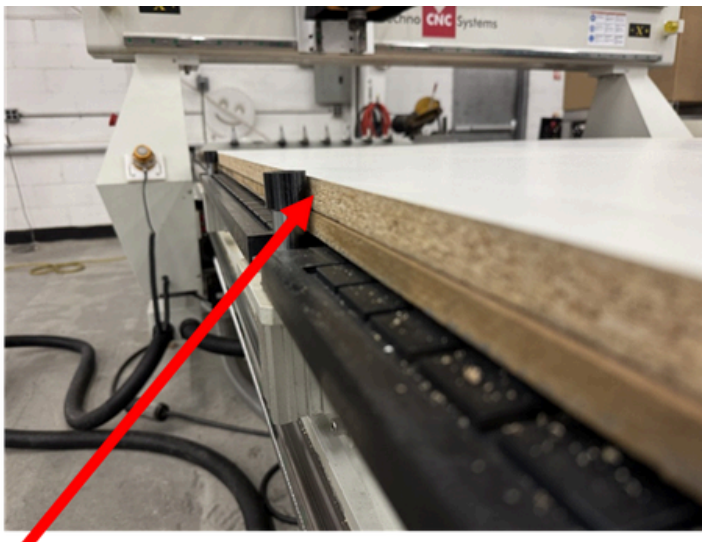


Fig. 13.4

Push substrate up against alignment pins. Then select Positioning(P) to bring pins back down.

Finding X and Y zero

Raise the dust hood on the spindle assembly by pressing the Dustcover1(H) button located at the bottom of the interface when in Manual, or Auto mode.

Using the Jog and handwheel functions on the interface, move the X and Y axes to the corner position of the material to be set as the zero position. Bring the z-axis down in the negative direction so it is easier to see where the tool is in reference to the edges of the board. With a tool in the spindle, try to position each axis, so that half the cutter is hovering over the material, and the other half is not. This will ensure that the center of the tool is set on the corner of the material depicted in Fig. 13.5

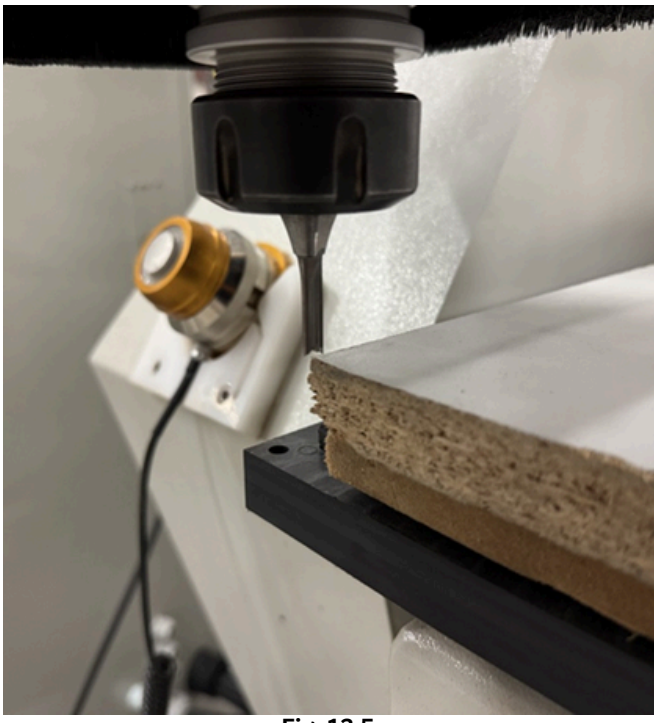


Fig. 13.5

Select the coordinates dialogue box, under the WorkCoor label on the interface (Fig. 13.6). When the box to the corresponding axes is selected, an indicator will appear ensuring you want to set the zero position, or work offset for this axis. Select Yes (Fig. 13.7).

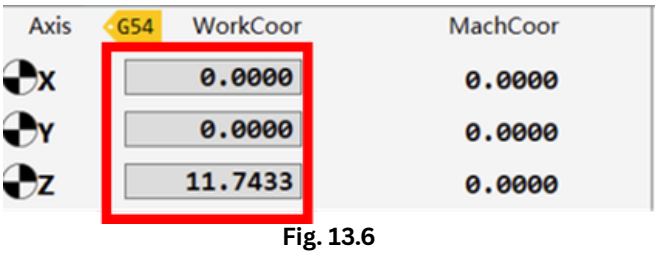


Fig. 13.6

Whichever position the machine is in when the work offsets are set will be where its new zero position is set. Be sure to have the tool in the correct location before proceeding with selecting these options.

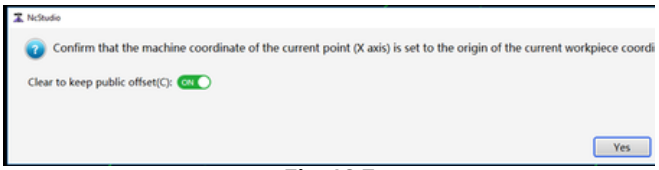


Fig. 13.7

2. Automatically Setting the Z WCS:

Before setting an origin position for the Z-axis, it is important to ensure that the material, or the zeroed surface is at its lowest position, or the same position it will be in during the machining process.

Power on the vacuum pump connected to the machine's vacuum table and open the appropriate zones to the table, drawing your waste board and material down to the vacuum table surface.

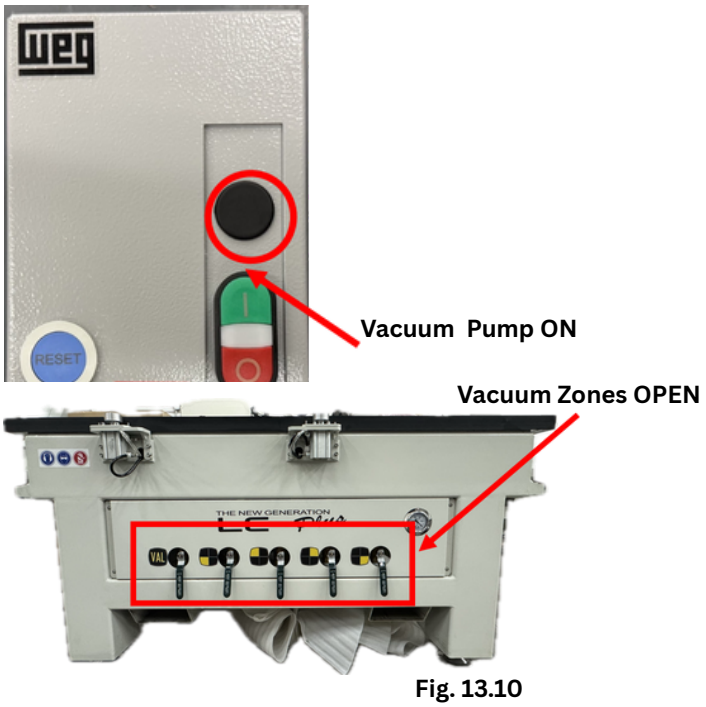


Fig. 13.10

Setting a WCS Origin Continued

With the material being held down, place the Z0 touchpad onto the surface to be made the origin and bring the spindle head over to the same location so that the tool is hovering about 2 inches above the touch pad as shown in Fig. 13.12.



With the tool hovering above the touch pad, navigate to the calibration command tab, and select the Z Mobile Cali(F) button (Fig. 13.13).
The dust hood will raise, if it is not already, and the machine will slowly descend in the z negative direction until it touches off of the mobile touch pad. Once it does, it will lift up a short distance, completing the auto Z0 procedure.

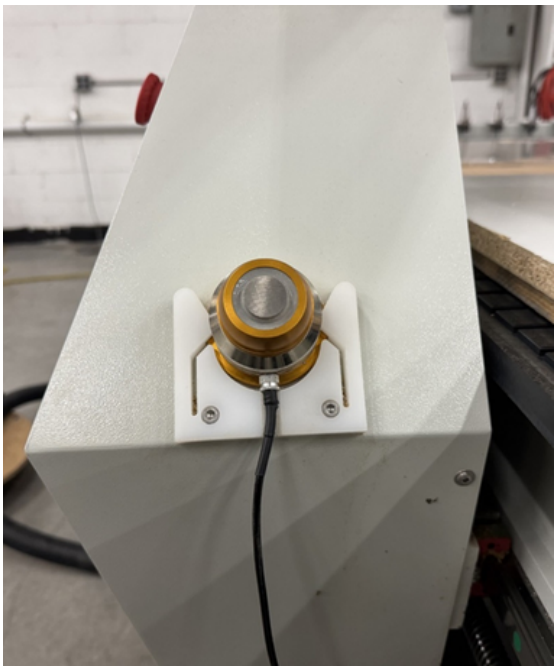


Fig. 13.11

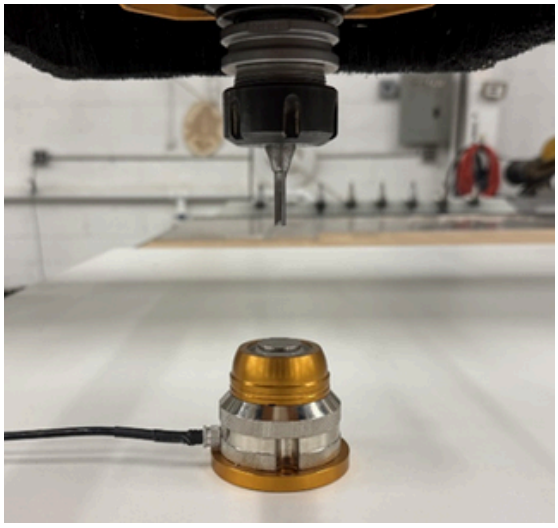


Fig. 13.12

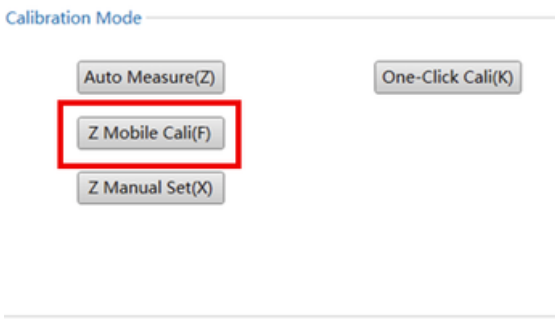


Fig. 13.13

3. Manually Setting a Z0 Work Offset:

If a scenario presents itself, where the Z0 work offset needs to be done manually, this is also an easy procedure with the LC Plus controller.
Begin by following the steps described previously of turning on your vacuum pump and opening up your zones to the vacuum table.
Using the handwheel, bring the tool down to the surface where Z0 will be set (Fig 13.14).
Select the Z-axis coordinate box at the top of the interface (Fig. 13.15) and then select Yes on the confirmation dialogue that will appear (Fig. 13.16). The coordinate will change to 0.000, establishing the surface the tool is on to be the new Z work offset zero.

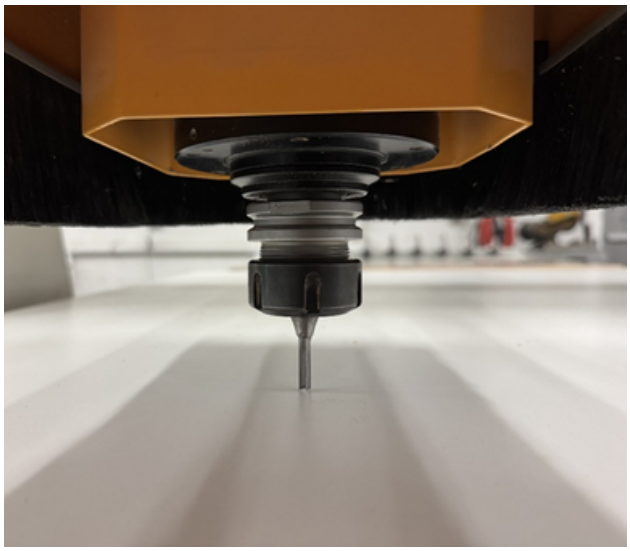


Fig. 13.14

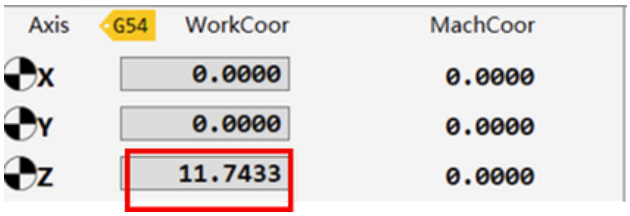


Fig. 13.15

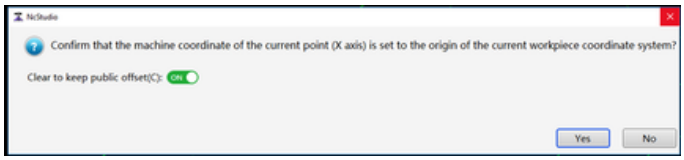


Fig. 13.16

4. Setting Up Multiple WCS Positions:

The LC Plus Series handheld controller can set multiple origin positions and call on them manually through the controller, or within the g-code file being executed. The number of WCS positions available to save is limited to 5 positions. This capability gives users the flexibility to set up multiple fixtures on their table without losing other WCS locations that are used daily. This part of the manual will review activating and setting multiple work coordinate settings as well as their corresponding G code commands.

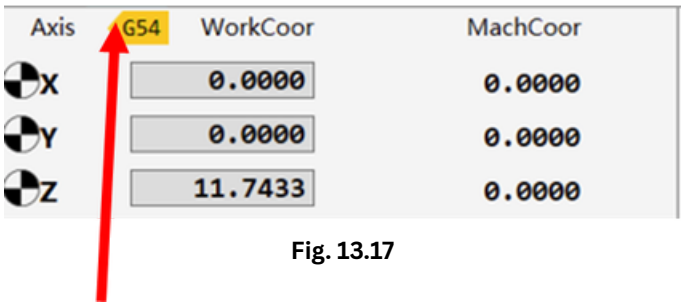


Fig. 13.17

To view what the machine's current active WCS is, have a look at the top of the screen, near the work and machine coordinate displays (Fig. 13.17).

To access or edit other available work offsets, navigate to the Offsets command tab.
From this command tab, all of the available work offsets and their corresponding machine coordinate positions can be viewed and modified (Fig. 13.18 on next page).

Setting a WCS Origin Continued

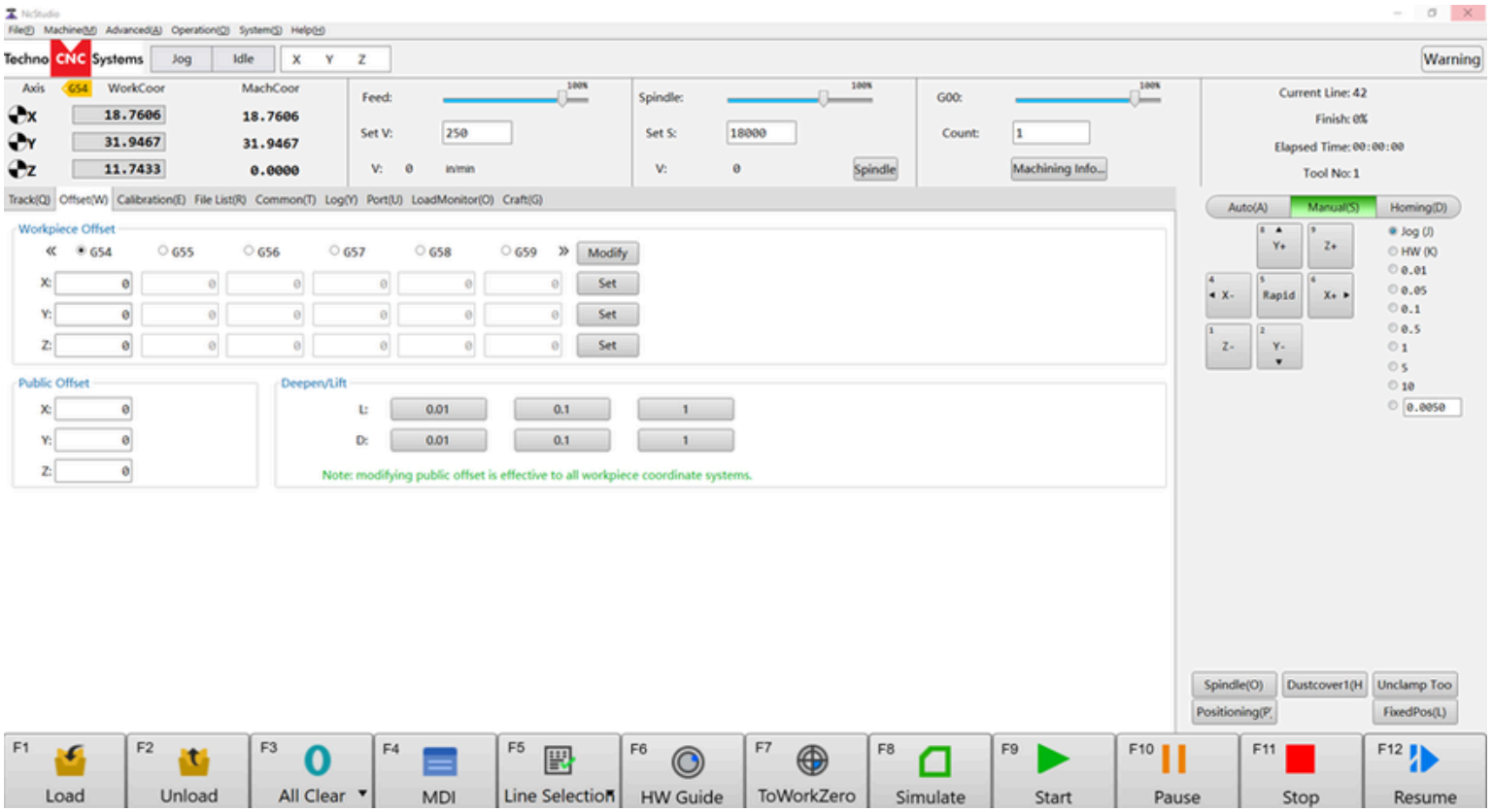


Fig. 13.18

To change, or activate a work offset (Fig. 13.20), simply select the bubble of the desired offset. Once the offset has been selected, press the "Modify" button to both activate and allow for edits to the offset (Fig. 13.19).



Fig. 13.19

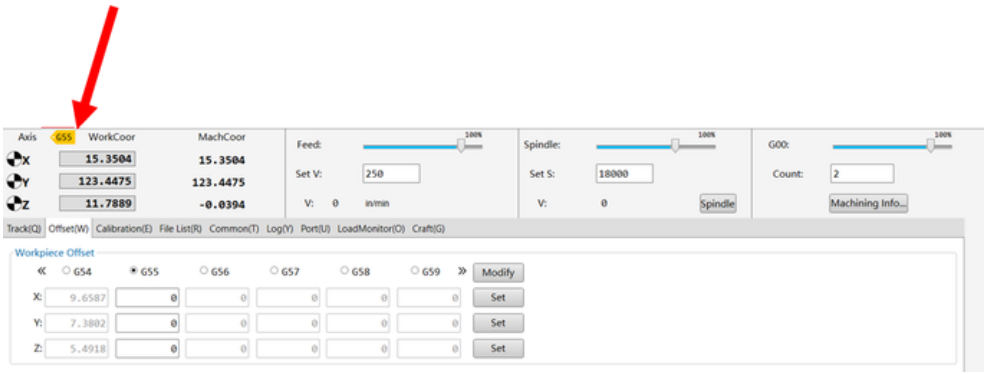


Fig. 13.20

Once an offset is active, it can be modified both numerically and manually using the same steps described previously of setting work offset zero positions. When modifying the offset numerically, select the dialogue box associated with the axis to be modified. In the menu that appears (Fig. 13.21), type in the desired offset value, and select OK. The new number will then appear in the dialogue box for the axis associated with the work offset.

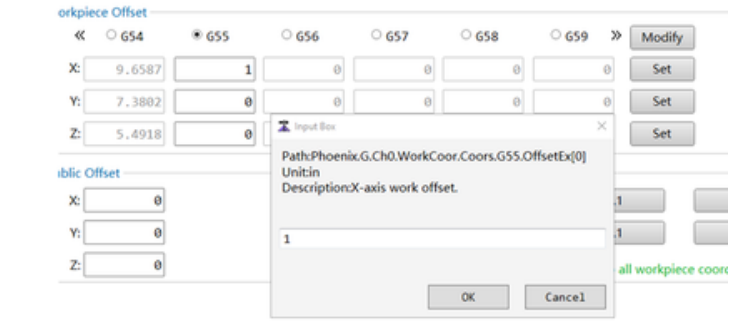


Fig. 13.21

LC PLUS
LOADING A G-CODE FILE

Section 14 – Loading a G-Code File

With all the machine set up taken care of, it is safe to load a programmed g-code file and run it on the LC Plus Series machine. If you do not have programming software or would like to inquire about the packages that Techno CNC offers, please call the main office for more information and formal pricing. Continue reading for more instructions about loading and executing programmed files.



Fig. 14.1

Begin by inserting the USB drive into the machine's control cabinet. Make sure that the programmed g-code file is saved onto this USB.

Like any PC, the files can then be loaded from the USB drive directory.

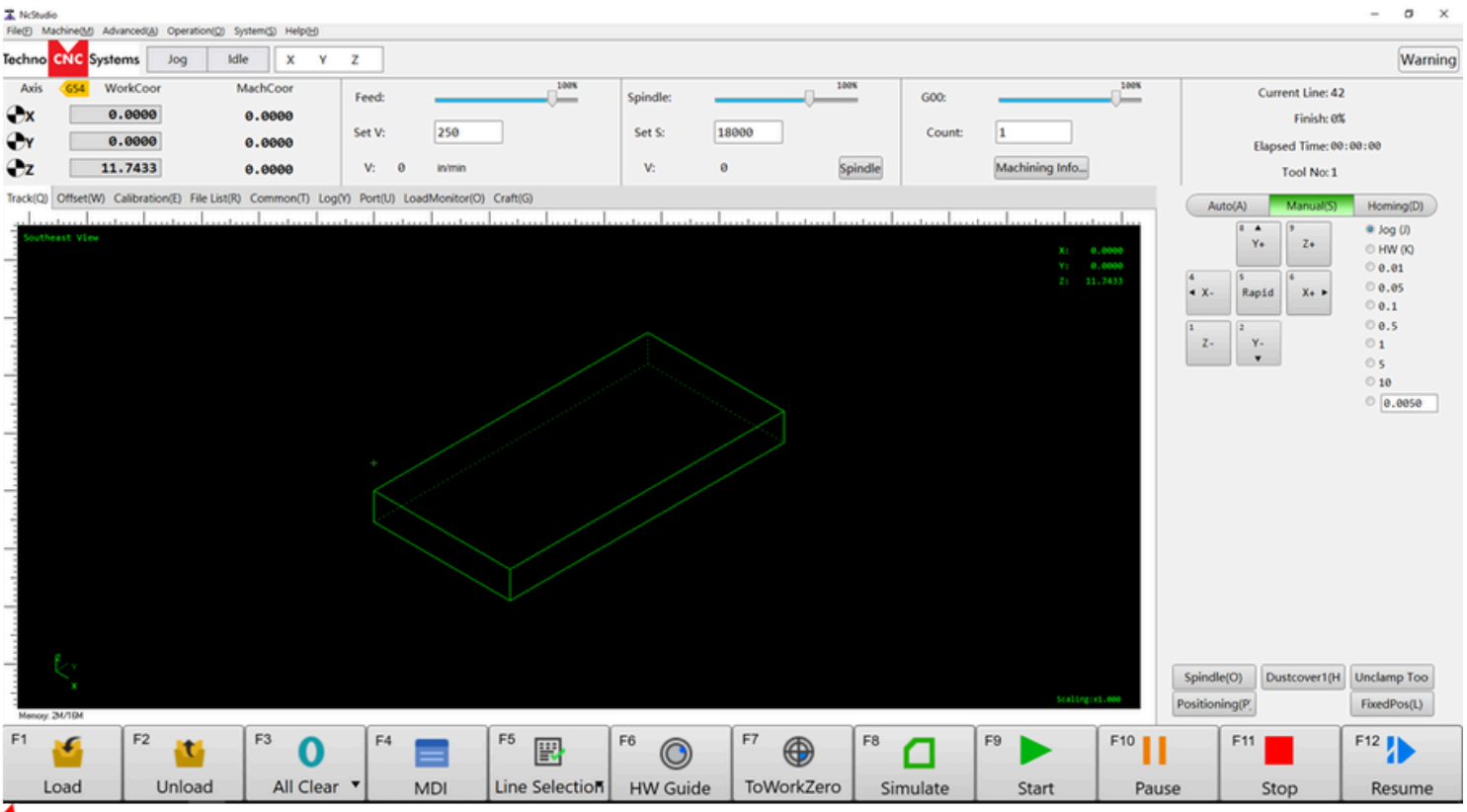


Fig. 14.2

With the USB plugged in, and the machine in manual mode, select the "Load button" at the bottom of the screen (Fig. 14.2).

In the file explorer window that appears, double click, or select and open the file from the USB directory (Fig. 14.3 on the next page).

Section 14 – Loading a G-Code File Continued

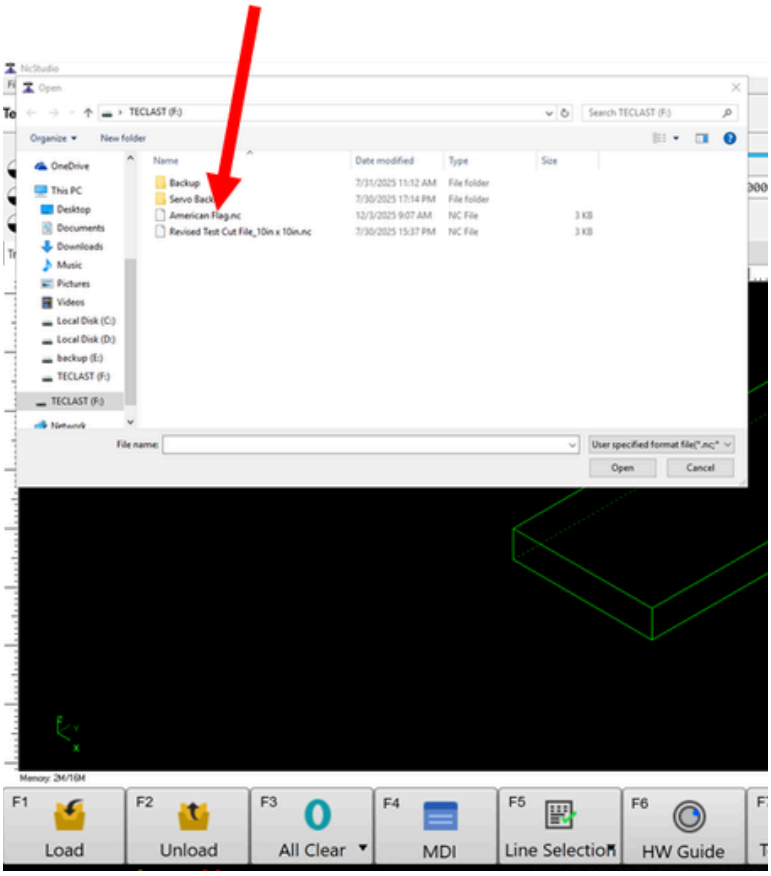


Fig. 14.3

Once the file has been loaded, it is advised to simulate the file (Fig. 12.4).

The simulation process takes a few seconds, and will prompt an error if there is something wrong within the file, preventing any crashes prior to running. The Track or preview screen will then show the cut and rapid paths of the machine for the file to be run (Fig. 14.5).

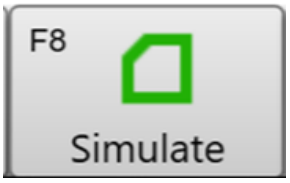


Fig. 14.4

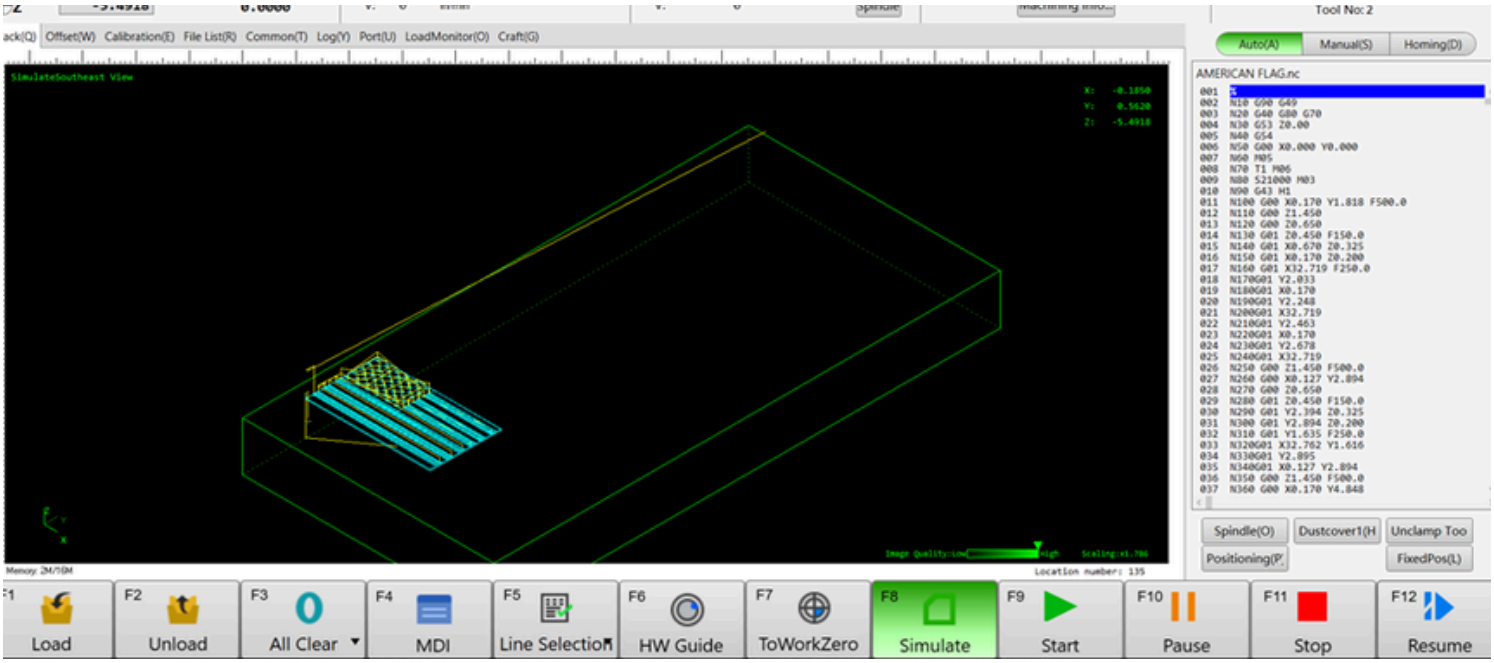


Fig. 14.5

Once the simulation is complete and the operator is happy with the preview, de-select the Simulate button, then press the "Start" button (Fig. 14.6) to begin the cut file.



Fig. 14.6

LC PLUS
ADVANCED CONTROLS

Section 15 – Advanced Controls

Thus far, you have been given the tools and instructions for setting up and running jobs on your LC Plus Series machines. This section will cover some “extras” that the machine’s controls are equipped with. Review this part of the manual for a more in-depth understanding of the CNC controller and the various functions that it offers.

File Skipping:

The controller on the LC Plus Series machine can look ahead and skip portions of a file when desired. It is also capable of picking back up where it left off, or resuming, after it has been stopped mid cycle.

Whether you are skipping ahead in a file, or picking back up where you left off, the procedure will be the same.

To begin, make sure that the file is loaded onto the controller. If the file is being resumed, do not power down or load a different file onto the machine.

On the handheld, select the  menu button → Operations -> Select Line No.

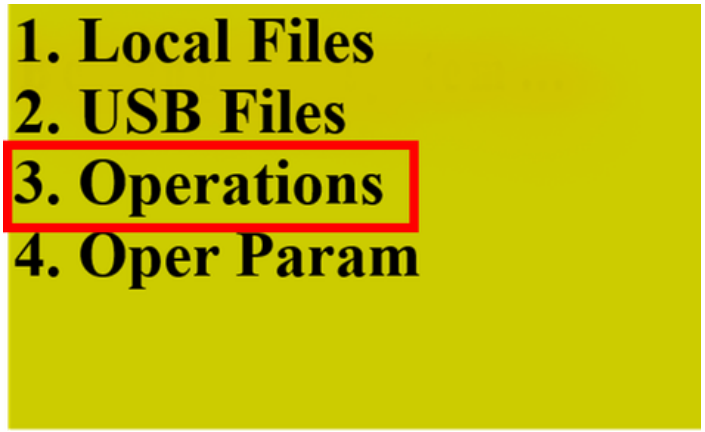


Fig. 15.1

When time comes to resume the file, begin by loading the file onto the controller after the machine has been homed. Proceed by navigating to the Select Line No. option and highlight the start line variable. Input the recorded line number, press OK, then scroll down and select Execute now.

The machine will proceed to start the spindle, move to its cut position at the commanded line number, and descend in the z-axis to resume the cut.



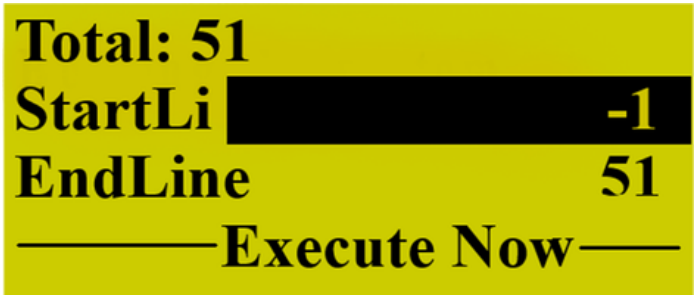
Fig. 15.2

In this menu, depicted in Fig 15.3, you will notice a start line and an end line.

If the file was stopped prior, the start line will be filled in with the last line number that was executed before the machine was stopped. If not, the start line should read 0 or -1.

The end line should match the same number as the “Total” variable.

When a file is being resumed, the start line will automatically be filled in with the last line number of the file that was executed. No change needs to be made to the start line. Use the down arrow to highlight the “Execute Now” option at the bottom. Select execute now using the OK button on the handheld.



The machine will automatically navigate to the main screen of the controller, start the spindle, and move the machine to the first XY cut position before descending in the Z and resuming its cut.

In some cases, the machine needs to be shut down prior to completing a file and resumed at a later time. To resume a file, or skip ahead in a file, prior to shutting down the machine, take note of the “Start Line” after stopping a cut file.

Saving Files Locally

Files that are used often on the LC Plus Series machine can be saved locally so they do not have to always be loaded from a USB drive. By saving files locally, an operator allows themselves to easily load and run a file at any time, with or without a USB.

All locally saved files start on a USB and are plugged into the machine. After selecting a file on the USB, instead of using the #1 button to load the file, use the #3 button to “Copy” the file.

Once the file is copied, the file can then be accessed from the “Local Files” option in the controller’s menu.

The same is true for copying over files locally to a USB. In the Local Files option, select the desired file, and use the #3 button to copy the file over to a USB that is connected to the machine.

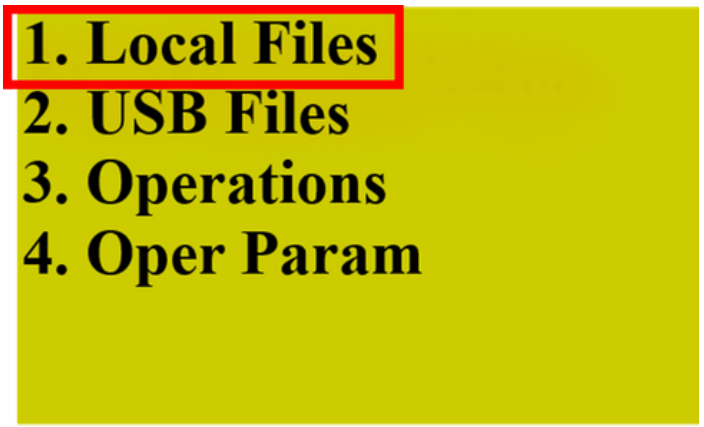


Fig. 15.4

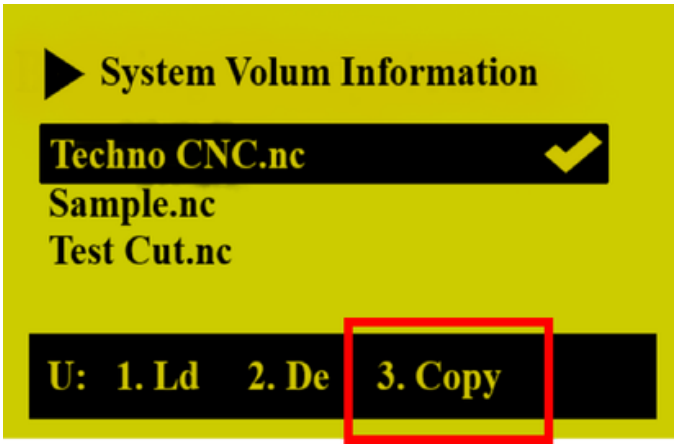


Fig. 15.5

LC PLUS
ERROR MESSAGES AND TROUBLESHOOTING

BASIC TROUBLESHOOTING

Air Alarm	-Check that air is connected to back of machine. -Check that regulator is set to 90 PSI Identify electronic regulator and with the machine powered off, jump the connection on the regulator. Turn the machine back on and see if the error is cleared. If it does, replace the electronic regulator.
NC File Running Error	- There is something wrong with the format or the code included with a g-code file. - Check programmed feed rates to make sure files are using inches/min units. Make sure feed rate does not exceed 800 inches/min
Soft Limit Error	An indication the machine is at the end of travel in either the X, Y, or Z axis.
The Spindle does not turn on when executing a programmed file	- Check that the spindle can turn on/off manually from the handheld controller. If not, check if there are any error codes appearing on VFD display. Call technical support with error code. - Check programmed file for M3 commands. If there are none, then there is no command to turn the spindle on. Check the programmed spindle speed is within the machine's range. Spindle speed range is 6,000 RPM – 24,000 RPM

One or more of the axes are not moving. The coordinates on the handheld change, but the machine does not move.	-Start by powering down the machine for 30 seconds, then restarting. Try to home the machine. If there is still no movement, press the red stop button to stop the homing cycle. -Open the back door of the control cabinet and identify the servo drives inside the cabinet. If there are any with error codes being displayed, take a photo of the error and share it with support@technocnc.com -If there are no errors, check that there are lights illuminated on each drive, indicating they are powered on. -If one or more of the drives do not have power, measure AC power at the terminal bridge, towards the bottom of the cabinet. High voltage lines should be labeled L13, L23, L33 (Same labels as are on the incoming power wires to the drive). If there is no voltage measured on the bridge, check continuity for the 3 power fuses on the front side of the cabinet. Wire labels should match labels on the terminal bridge. Replace fuses as needed.

When powering on the machine; Limit Release Error appears and navigates to the main screen of the controller without prompting a Back to REF Point message.	- The machine was powered on while a limit switch had been sitting on its limit, immediately enabling the limit switch. - Identify the axis prompting the limit release error. - Shut down the machine and physically push the axis off of its limit. This may require more than one person to accomplish. This process will not work for the z-axis. - Switch the polarity of the limit through the controller, allowing the operator to manually jog the axis off its limit. To access the I/O menu, go to menu > Diagnosis > Port List. Once the machine is off its limit, it will again prompt a limit release error. Switch the input back to its original polarity state and exit to the main screen. All errors should be clear, and motion control should be re-established. **- Inputs for axis limits:** GX06: Y-axis Neg. Limit GX05: Y-axis Pos. Limit GX02: X-axis Pos. Limit GX03: X-axis Neg. Limit GX08: Z-axis Pos. Limit

Clamp Signal Not Detected; during an auto tool change sequence	- The spindle does not see that there is a tool in the spindle. - Check that the pull stud at the top of the tool holder is tight and now loose. A loose pull stud will change the position of the tool holder in the spindle and not prompt the necessary input to complete a tool change. - Check that there is no build up of rust forming on the tool holder. If there is, use scotch Brite to polish tool. - Replace the 24V relay in the back of the cabinet labeled KA8. Replace the 24V relay in the back of the cabinet labeled KA7.

TECHNO CNC SYSTEMS LIMITED WARRANTY & COVERAGE

Limited Warranty on Techno Brand Products

Subject to the terms and conditions set forth in this warranty document, Techno CNC Systems LLC ("Techno") warrants its Techno brand products ("Product" or "Products") to the original purchaser for a period of one (1) year against defects in material and workmanship under normal use and conditions ("Product Limited Warranty"). The Product Limited Warranty commences on the date of Product shipment from Techno facilities and expires one (1) year from the ship date ("Product Warranty Period").

Spare or replacement parts ("Part" or "Parts") for Techno Products are warranted to the original purchaser for a period of ninety (90) days against defects in material and workmanship under normal use and conditions ("Parts Limited Warranty"). A Parts Limited Warranty commences on the date of a Part shipment from Techno facilities and expires ninety (90) day from the ship date ("Parts Warranty Period").

A Product Limited Warranty may be validly transferred to one additional party by the original purchaser provided that a reregistration fee is paid to Techno within seven (7) days of transfer of the Product and prior to the expiration of the Warranty Period. Reregistration of any Product warranty does not extend the Warranty Period. A Parts Warranty is not transferable.

Product Limited Warranty and Parts Limited Warranty are hereinafter referred to collectively as "Limited Warranty." Product Warranty Period and Parts Warranty Period are hereinafter referred to collectively as "Warranty Period."

What Is Covered Under the Limited Warranty

During the Warranty Period, Products and Parts that Techno deems validly subject to a warranty claim will be repaired or replaced, in Techno's sole discretion, without charge. Repaired items may include new or refurbished replacement parts. Replaced items may be new or may be manufactured from serviceable used parts. Items that have been repaired and/or replaced will be warranted only for the unexpired portion of the applicable Warranty Period to the original purchaser.

As a condition to this Limited Warranty, customers shall have read the operator's manual and registered the Product or Part with Techno within 30 days of purchase.

What Is Not Covered Under the Limited Warranty

Events that are not covered under this Limited Warranty include:

* Product/Part damage resulting from third-party parts, accessories or systems connected to or used in conjunction with the Product/Part that have adversely affected its operation, performance or durability.
* Product/Part damage caused by normal wear, accidents, improper maintenance, improper use or abuse, alterations, or failure to follow operation and maintenance instructions contained in the operator's manual.
* Products/Parts purchased from any supplier, distributor or dealer not authorized by Techno.
* Labor costs including, but not limited to, such costs as the removal and reinstallation of a component or assembly.
* Insurance and packing costs for a defective items returned to Techno by the customer.
* Product/Part damage caused by electrical surges, improper venting, flooding, fire, freezing, corrosive atmospheric elements, abnormal external temperature, or any event of force majeure such as riot or act of war.
* Noise or vibration unless it is the result of defective material or workmanship of the Product/Part.
* Claims of defective Products or Parts not made in conformance with Techno's return policy as set forth below.
* Transport costs for defective items that require more than one (1) shipping to remedy a claimed defect.

* Claims for personal injuries, incidental or consequential damages, or economic loss (profit or revenue), however caused. i.e. any other incidental, consequential, indirect, special and/or punitive damages, whether based on contract, warranty, tort (including, but not limited to, strict liability or negligence), patent infringement, or otherwise, even if advised of the possibility of such damages. Some states do not allow the exclusion or limitation of certain damages, so the above exclusion or limitation may not apply to a particular customer depending on location.
* Claims for Product components or Parts that are warranted separately by their respective manufacturer. Available warranties covering those components are furnished with each Product and Part. Techno CNC Systems does not assume any warranty obligation or liability for components covered exclusively by the stated warranty of a component's respective manufacturer(s).

Techno's Limited Warranty shall be void in the event of an occurrence of any of the following:

* Failure by the Original Purchaser to register the Product within thirty (30) days of its purchase.
* Where applicable, failure to validly reregister the Product within seven (7) days of transfer of the Product and prior to the expiration of the Warranty Period.
* Improper installation of the Product, including but not limited to, installation in violation of applicable rules, laws or building codes, and installation for non- recommended uses.
* Accident, abuse or misuse of the Product.
* Failure to follow or comply with the user's operational manual.
* Modification, alteration, addition of non-approved components, or misapplication of the Product or Part in any manner.
* Repairs and service conducted by personnel unauthorized by Techno.
* Modifications to, and tampering with, the Product or Part.
* Use of non-standard parts or accessories without prior written approval from Techno.
* Use of Product or part for purposes for which the item was not designed or intended.

Warranty Limitations

Techno's maximum liability hereunder is limited to the original purchase price of the Product or Part.

Techno assumes no responsibility for the selection of any Product or Part for a specific application absent Techno's written approval of such application, and makes no general representations whatsoever in respect to any such selection.

THIS WARRANTY IS IN LIEU OF ALL OTHER WARRANTIES EXPRESSED OR IMPLIED. ALL OTHER WARRANTIES, INCLUDING, BUT NOT LIMITED TO, ANY WARRANTY OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE, WHETHER EXPRESSED, IMPLIED, OR ARISING BY OPERATION OF LAW, TRADE USAGE, OR COURSE OF DEALING, ARE HEREBY DISCLAIMED. THERE ARE NO WARRANTIES THAT EXTEND BEYOND THE DESCRIPTION ON THE FACE OF THISE WARRANTY DOCUMENT.

TECHNO SHALL NOT BE LIABLE FOR INDIRECT, INCIDENTAL, SPECIAL, CONSEQUENTIAL, PUNITIVE OR OTHER SIMILAR DAMAGES THAT MAY ARISE, INCLUDING LOST PROFITS, DAMAGE TO PROPERTY OR INJURIES TO A PERSON, LOSS OF USE, INCONVENIENCE, OR LIABILITY ARISING FROM THE INSTALLATION, SERVICE OR USE OF THE PRODUCT OR PART.

UPON THE EXPIRATION OF THE LIMITED WARRANTY PERIOD, TECHNO'S LIABILITY UNDER THIS WARRANTY SHALL TERMINATE.

Some states do not allow the contractual exclusion or limitation of incidental or consequential damages or personal injury, so the limitations set forth herein may not apply to all customers in all locations.

How To Obtain Warranty Repair/Replacement

All defective items covered under the Limited Warranty must be properly returned to Techno for inspection. Techno reserves the right to not accept returns unless the returned item is accompanied by proof of original purchase, a return authorization number ("RAN") from Techno, and shipped in accordance with packaging and

shipping instructions given to the customer by Techno. Claims and requests for a RAN must be made within seven (7) days of discovery of a defect. Proper packaging and insurance for transportation is solely the customer's responsibility. All returned items must be sent to the Techno facility located in Ronkonkoma, New York (or such other place as Techno specifically designates to the customer) with a statement of the problem and transportation prepaid. If, upon examination, Techno determines that a warranted defect exists, the returned item will be repaired or replaced in Techno's sole discretion at no charge, and shipped prepaid back to the customer. Return shipment will be by common carrier of Techno's choosing. If rapid delivery is requested by customer, then such transport expense shall be borne by the customer.

Warranty inspections and repairs are performed at Techno's New York facility, where all necessary diagnostic and repair equipment is available. This equipment is difficult to transport and field service is accordingly severely limited and will only be supplied at Techno's sole discretion. If field service is required, all service call expenses, including transportation, travel time, subsistence costs, and the prevailing cost per hour (eight hour minimum) are the responsibility of the customer.

In the event that support diagnostics of a covered Product or Part requires an item to be shipped more than one (1) time for any given claimed warranty defect, then the customer shall bear all transport costs.

If an out-of-warranty situation exists, the customer will be notified of the repair or replacement cost. At such time, the customer must issue a purchase order to cover the cost of the repair/replacement or authorize the item to be shipped back to the customer at the customer's expense. In all cases, a restocking charge of twenty (20%) percent will be charged to the customer on all items returned to stock.

Warranty claims will not be reviewed or remedied unless the warranty registration is received by Techno within thirty (30) days of the purchase date. All warranty issues must be handled through Techno.

Techno customer service can be reached by calling 6316487481

Additional Terms & Conditions

TECHNO RESERVES THE RIGHT TO CHANGE DESIGNS, SPECIFICATIONS, PRICES AND ANY APPLICABLE DOCUMENTATION WITHOUT NOTICE TO THE CUSTOMER.

Techno is not liable for delay or failure to perform any obligation hereunder by reason of circumstances beyond Techno's reasonable control. These circumstances include, but are not limited to, accidents, acts of God, strikes or labor disputes, laws, rules, or regulations of any government or government agency, fires, floods, delays or failures in delivery of carriers or suppliers, shortages of materials, and any other event beyond Techno's control.

No legal action arising out of any claimed breach of this Limited Warranty may be brought by the more than one (1) year following date of purchase of a Product or Part. This Agreement shall be governed in all respects by the laws of the State of New York, United States of America. Any legal action brought by a customer against Techno must be brought in the state courts of the State of New York, Second Judicial Department. Some states do not allow the contractual limitation of time periods for bringing suit so the limitations set forth herein may not apply to all customers in all locations.

The terms and conditions contained herein shall constitute the entire agreement concerning the Limited Warranty described herein. No oral or other representations are in effect. No dealer, distributor, or individual is authorized to amend, modify, or extend this Limited Warranty in any manner and only the warranty expressed in this warranty document is extended herein by Techno. Statements made outside this warranty document, such as in dealer advertising or presentations, whether oral or written, do not constitute warranties by Techno and should not be relied upon.

Section headings contained in this warranty document are for informational purposes only and may not be used to limit the terms and conditions set forth in this warranty document.

Form H1219

SECTION 1: TECHNICAL SUPPORT AND ON-SITE SERVICE

Techno CNC Systems provides a variety of technical support services with the purchase of any CNC plasma cutter or router table. The following is a brief outline of available support options. On-site service is available for additional fees. To receive an immediate quote for an on-site visit, please inquire at technocnc.com or call us at 631-648-7481.

PHONE SUPPORT

Speaking with a professional and certified Techno CNC technician is quick and easy. Simply call us at 631-648-7481. Our support team is available Monday – Friday 8:30am – 5:00pm Eastern Standard Time. Techno CNC Systems will make every effort to handle phone calls in a timely manner, but due to the nature of machine diagnosis and the varying capabilities of different machine operators, we cannot guarantee hold times for phone-in support. Technical support includes troubleshooting, machine configuration and assembly, tooling recommendations, parts inquiries, and quality related issues.

OPERATIONAL TRAINING IS NOT INCLUDED IN OVER THE PHONE SUPPORT.

E-mail Techno CNC Systems will return e-mails sent to the support@technocnc.com e-mail address within 24 hours Monday – Friday.

TECHNO UNIVERSITY

Techno CNC Systems provides a number of software and machine training opportunities at our Long Island, New York facility. Please call 631-648-7481 for details

CINEMATIC TUTORIALS

Get started learning all the basics for your Techno CNC machine using our video tutorials listed for each machine. For access to the video tutorials, please visit us at https//:technocnc.com/technical-support/

TECHNO CNC SHOP

A variety of consumables, accessories, and cutting tools can be quickly and easily purchased through the Techno CNC Shop. Visit https://technocncshop.com to learn more and start filling your cart.

Techno CNC Systems

631-648-7481

technosales@technocnc.com

www.technocnc.com

29 Trade Zone Drive
Ronkonkoma, NY 11779

@technocnc

youtube.com/TechnoCNCSystemsLLC

Techno CNC Systems

@technocnc