Eagle eye
CNC ROUTERS

CNC user manual

Table of Contents

Terms and Conditions of Limited Warranty

This warranty is in lieu of all other warranties expressed or implied. All other warranties, including but not limited to any warranty of merchant ability or fitness for a particular, whether expressed, implied, or arising by operation of law, trade usage, or course of dealing, are here disclaimed. There are no warranties that extend beyond the description herein.

Eagle Eye CNC Inc guarantees this CNC system has been quality checked, and tested in the factory and passed with satisfactory results. Eagle Eye Inc warrants that the CNC system will perform in substantial compliance with instructions and documentation supplied. Eagle Eye CNC Inc warrants that the CNC system will be free from defects in material and workmanship for a period of **1 Year** from the date of product delivery. Eagle Eye CNC Inc has the sole obligation, under this warranty, to repair, or at its option, replace defective products, at no additional charge to customer. Eagle Eye does not cover defects caused by (1) normal use of parts subject to continuous or rapid wear (2) non approved alteration by customer of any part or device of the CNC system (mechanical, electrical, electronic, computer software, computer hardware). Eagle Eye CNC Inc makes no guaranty that its' products are fit for any use or purpose to which they may be put by the customer, whether or not such use or purpose has been disclosed to Eagle Eye CNC Inc in specifications or drawings previously or subsequently provided, and whether or not Eagle Eye CNC Inc products are specifically designed and/or manufactured for such use or purpose. All resold sold products constituting accessories and secondary components, including but not limited to, routers, spindles, vacuum pumps, stepping motors, stepping drives, servo motors, servo drives, etc., are subject to their own manufacturers' warranties. The computer warranty of Eagle Eye CNC Inc is in accordance with the computers' manufacturer warranty. Failure is not the responsibility Eagle Eye CNC Inc.

Eagle Eye CNC Inc is not responsible for non-compliance of the CNC system caused by failure to follow the precautions and instructions given in this manual or by improper use or handling of the CNC system. The customer has the right to replacement parts shown to be defective, unless said defects are caused by <u>unauthorized tampering</u>, including the fitting of <u>non-original Eagle Eye spare parts</u> and/ or the replacement of parts described or authorized in this manual unless authorized before hand ***<u>in writing</u>*** by Eagle Eye CNC Inc.

Eagle Eye CNC Inc is not liable for damages: (1) to persons (2) to property (3) to the CNC system (4) incurred for lost production (5) incurred for lost materials (6) incurred for lost income (7) incurred for manufacturing down-time (8) incurred from loss of data or information (9) or any losses economic or otherwise, arising from use of Eagle Eye CNC Inc products or connected to this agreement or items sold under this agreement, whether alleged to arise from breach of contract, expressed or implied warranty, or in tort, including without limitation, negligence, failure to warn, or strict liability.

Eagle Eye is not liable for delay or failure to perform any obligations here under by reason of circumstances beyond its reasonable control. These circumstances include, but are not limited to accidents, acts of God, labor strikes or disputes, laws, rules, regulations or actions 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 the reasonable control of Eagle Eye CNC Inc.

Eagle Eye CNC Inc has a policy of constant development and improvement, and reserves the right to make functional and stylistic modifications to its products, to change the design of any functional or accessory part, and to suspend manufacturing and supply without notice and without obligation to other parties. Eagle Eye CNC Inc reserves the right to make any structural or functional changes to its products, and to change the supply of spare parts without prior notice.

NOTICE: CNC Systems are protected by a machine serialization system. This serialization must be updated from time to time in order to keep the CNC System working. When prompted to enter a vendor update code, contact Eagle Eye CNC Inc at 423-228-2349 for an updated code. Eagle Eye CNC Inc reserves the right to withhold this update code from its customer if the customer's account is past due or if customer refuses to pay for goods or services. Eagle Eye CNC Inc reserves the right to withhold update codes or to decline to issue permanent codes if final payment on a customers' invoice for an original machine purchase has not been paid in full. Issuance of updated codes and the amount of time before expiration of those codes is at the sole discretion of Eagle Eye CNC Inc**.**

The terms and conditions herein shall constitute the entire agreement concerning the terms and conditions for the limited warranty described herein. No oral or other representations are in effect. This agreement shall be governed in all respects by the laws of the State of Tennessee. *No legal action may be taken by any party more than* **1 year** *after the date of purchase*.

Policies and Procedures

Return procedure Policy
Before returning any equipment in or out of warranty, the customer must first receive authorization number and packing instructions from Eagle Eye CNC Inc. No claim will be allowed nor credit given for products returned without such authorization. Proper packaging and insurance for shipping is solely the responsibility of the customer. Upon receiving approval from Eagle Eye CNC Inc, the product should be returned along with statement describing problem or defect with the Eagle Eye CNC Inc product with shipping ***prepaid***. If upon examination by Eagle Eye CNC Inc, warranted defects exist, the product will be repaired or replaced at no charge and returned, prepaid, to the customer. Return will be by common carrier (e.g. UPS). Should expedition be required, this will be at the expense of the customer, not of Eagle Eye CNC Inc. Should an out-of-warranty situation exist, the customer shall be notified of any repair cost. At such time, the customer must issue to Eagle Eye CNC Inc a purchase order to cover the repair cost or authorize the return shipping of product as-is to the customer. A restocking fee of 20% will be charged on items returned to stock.

Service Policy
Repairs are ordinarily done at the Eagle Eye facility in Sale Creek, TN were all necessary instrumentation is available. Service equipment is difficult to transport, and thereby limit these services to be performed at the discretion of Eagle Eye CNC Inc. Should Services be required and provided at the sole discretion of Eagle Eye CNC Inc, any and all relevant expenses incurred, including transportation, travel time, subsistence costs, and prevailing cost per hour (8 hour minimum) are the responsibility of the customer.*see Technical Support policy for details.*call for current pricing.

Shipping Policy (to Customer)
Eagle Eye makes all product shipments FOB (Freight Collect) from the Eagle Eye facility in Sale Creek, TN. All shipments via freight lines (land, sea, or air) will be by customer appointed carrier.
Exception: **Eagle Eye *will not* ship or receive any items by *Central Transport Freight Lines***.
Shipments to customer will be received during regularly scheduled business hours unless other arrangements are necessary. Shipments to customer will be received by customer appointed freight carrier at:
13555 Back Valley Rd.
Suite H
Sale Creek, TN 37373
Upon delivery to agreed customer location, it is solely the responsibility of the customer to **inspect the shipment for any damages** incurred during transport. **Damages to shipment must be noted on the bill of lading at the time of delivery**. Eagle Eye CNC Inc is not responsible for any damages to shipment during transport to the customer. The full value of the shipment must be recorded on the bill of lading in order to receive full compensation for damages incurred during transport to the customer.

Installation and Set up (by Eagle Eye)
Eagle Eye is not a shipping / freight company. Delivery may be provided by Eagle Eye, at its option, to customers requiring installation and set up of the CNC System. The CNC System will be tested and working properly prior to shipment. The customer should be familiar with the operation and use the CNC System and any applicable software prior to delivery for installation and set up. CNC System and Software training are not included in the installation and set up. Upon delivery of the product to customer location Eagle Eye will assist in (1) removal of product from carriers' vehicle. (2) moving of product to its' desired working location. (3) final system assembly as required. Customer shall be solely responsible for electricity supplied to the CNC System from service disconnect breaker. Customer is responsible for meeting any applicable electrical codes.

Policies and Procedures continued

Technical Support Policy

Eagle Eye provides technical support service and assistance for its customers by (1) telephone (2) E-mail (4) technician on site. Technical support is no substitute for CNC System, or software training that is purchased separately by the customer. Technical support is provided at no charge to customer for diagnosing and assisting in the remedy (1) of mechanical problems. (2) of electrical problems (3) of preventative maintenance issues, up to 1 hour per incident for any problem arising with Eagle Eye products which are covered by product the limited warranty. Customer will be charged if technical support or assistance is required if the problem is not covered by product limited warranty. For technical service by telephone the customer will be required to provide information about the Eagle Eye product and the problem. The technician will ask questions concerning the (1) nature of problem. (2) age of product or system. (3) type of product. Customers in need of technical service or assistance on product or system that not covered by the limited warranty will be required to provide the technician with a valid credit card number to cover technical support or assistance service charges. A base charge of $US (call for pricing) will be billed for the first 15 minutes of the service call. A charge of $US (call for Pricing) / minute will be billed after the first 15 minutes of the service call. Pricing will be explained at the time of service rendered. E-mail service will be provided without charge at the discretion of Eagle Eye. If technician on site service is required for customers less than 2 hours driving distance from the Eagle Eye factory, the customer will be billed a charge of $US (call for pricing) / hour. If technician on site service is required for customers more than 2 hours driving distance from the Eagle Eye factory, the customer will be billed a charge of $US (call for pricing) /day on site with a minimum of 4 hours service, and with maximum of 8 hours service for one day. All rooming, travel, and subsistence costs + $US (call for pricing) per diem will be at the customer expense. All parts will be billed in addition to service charges.

Design Files

Eagle Eye offers to its' customers CNC Cutting File Design and Design assistance. The charge for file designing is $US (call for pricing) / hour of design time in increments of 15 minutes with a minimum of $US (call for pricing) charged per file. All files will be transferred by E-mail. Hard copies of design files may be available on CD or DVD upon request with a shipping charge of $US (call for pricing).

CNC System and Software Training

Eagle Eye offers to its' customers, training on CNC Systems and Software. Training classes are provided at the offices of Eagle Eye in Sale Creek, TN.

Start up training class with CNC System Purchase
Machine training at our site at no charge
 2 days for up to 2 Persons at the Eagle Eye Facility……$US (call for pricing)
 (Customer provides transportation and needs of student(s))

Intermediate training class
 2-3 days for up to 4 Persons at the Eagle Eye Facility……$US (call for pricing)
 4 or more persons (up to 6) at the Eagle Eye Facility……..$US (price + $50/ add. person /day)
 (Customer provides transportation and needs of student(s))

Classes will not be mixed by different customers (companies). Each customer (company) will receive training for its' CNC System and Software.

General Information:

- This manual has been prepared by the technical staff of Eagle Eye CNC Inc exclusively for its customers and contains reserved information. Therefore, any partial or total reproduction and/ or discloser to third parties of the content herein is strictly prohibited without the prior written consent of Eagle Eye CNC Inc. This manual is supplied as an integral part of the CNC system and constitutes, at the time of printing, the latest edition of documentation pertaining to the product.

- This manual is to be used by suitably trained personnel only. The information contained in this manual offers no guarantee against risk. The use of the content of this manual is the sole responsibility of the user. Eagle Eye CNC Inc cannot be held responsible or liable for any damage or injury resulting from incorrect use of this document.

- This manual describes the procedures for correct installation of the CNC system. In the case of conflict between these instructions and safety, electrical, or other standards/ codes, please contact Eagle Eye CNC Inc for any corrective and or adaptive measures. Under no circumstances can the instructions contained in this manual substitute technical, electrical, or safety standards/ codes.

- **It the sole responsibility of the CNC system user to thoroughly read and understand this and all documentation supplied with the CNC system in order to avoid any incorrect or dangerous procedures or operations.**

General Safety Symbols:

- Important instructions or precautions are marked with the following symbols.

⚠	**Warning:** Identifies situations that could lead to personal injury.

⚡	**Warning:** Live electrical parts.

i	**Important:** Identifies important operational information.

Safety Warnings and Rules

Fully read and understand this and any manual or other instruction provided with the CNC System. Become familiar with the function, operation, and hazards of the CNC System, its' parts, and accessories.

Personal Safety Rules

- **Only qualified personnel should operate the CNC System.**

- **Stand clear of moving machinery.**

- **Never run CNC equipment unattended.**

- **Stay alert, watch what you are doing and use good sense when operating the CNC System.**

- **Do not use the CNC System while tired or under the influence of drugs, alcohol, or medication**.

- **Keep long hair, clothing, jewelry and gloves away from moving parts.**

- **Always keep body parts away from cutting head.**

- **Always use personal protective equipment (PPE) as required by law including** – Safety glasses, hearing protection, dust or vapor protection, nonskid shoes, and hard hat.

- **Remove collet wrenches before operating the CNC System.**

Work Area Safety Rules

- **Keep work area clean and well lit**

- **Do not operate the CNC System in the presence of flammable liquids, gas, or dust.**

- **Keep bystanders, visitors, and children away while operating the CNC System.**

- **Remove unnecessary articles from work surface before operating the CNC System.**

- **Observe any caution or warning signs supplied with the CNC System.**

System Use and Care

- **Use clamps or other practical way to firmly secure and support the work piece.**

- **Maintain the CNC System with care.**

- **Always use recommended cutting speeds and feed rates.**

- **Keep all cutting tools sharp and clean.**

- **Do not force a tool.**

- **Always use the correct tool for the job.**

- **Check for misalignment or binding of moving parts, and any other condition that may affect the systems operation.**

- **Always keep the CNC System controls clean, and free from dirt, debris, grease, and oil**

- **Always know locations of E-stop or other emergency features on the CNC System.**

Warning: Some dust or fumes, created by power sanding, sawing, grinding, routing, burning, and other construction activities performed on certain materials, are known to be harmful. Personal risk from exposure to these can be significantly reduced by working in well ventilated area, using an adequate dust or fume removal system, working with approved safety and protection equipment.

Safety Warnings and Rules Continued

Fully read and understand this and any manual or other instruction provided with the CNC System. Become familiar with the function, operation, and hazards of the CNC System, its' parts, and accessories.

Electrical Safety Rules

- **Any electrical work should be done by qualified, licensed, electrician.**

- **Always keep the system control box closed during operation.**

- **Always keep electrical guards and covers in place while power is supplied to the CNC System.**

- **Always disconnect power to the CNC System before performing any maintenance procedure.**

- **Always know the location of service disconnect breaker.**

- **Always keep cable and wire carriers clean from dust, dirt, debris, metal chips, oil, grease, water and other materials or fluids.**

- **Check any connectors, plugs, or other electrical device for proper connection before system operation.**

- **Do not operate the CNC System during electrical storms**

Electrical and mechanical service Rules

- **Use only spare parts and accessories recommended by the manufacturer for your make and model.**

- **Repair or maintenance service must be performed only by trained, qualified, service technicians.**

- **Do not modify alter any mechanical or electrical part of the CNC System without prior instruction or authorization.**

Warning: Care must be taken not damage any exposed electrical cabling (or connector), compressed air fitting (or line), vacuum piping, dust collection, fume extraction hoses, or any other connection during loading, unloading, or relocation of CNC equipment. All electrical cables, compressed air lines, vacuum piping, dust collection or fume extraction hoses must be disconnected before moving or relocating the CNC equipment.

General Specification*

Model Number	EG304	EG404	EG408	EG508	EG410	EG510	EG LASER	SIDEWINDER
Cutting Area	32" X 40"	48" X 48"	48" X 96"	60" X 96"	48" X 120"	60" x 120"	EGL?	SW?
Z Axis Clearance	6"	6"	6"	6"	6"	6"	6"	6"
Z Axis Travel	6"	6"	6"	6"	6"	6"	6"	6"
Overall Length	63"	73"	123"	123"	161"	148"	See EG-40 thru. EG 125	See EG-40 thru. EG 125
Overall Width	39"	62"	62"	74"	62"	74"	See EG-40 thru. EG 126	up to 90"
Overall Height	68" Max.	68" Max.	68" Max.	68" Max.	68" Max.	68" Max.	68" Max.	68" Max.
Weight W/O Options	798 lbs.	1037 lbs.	1543 lbs.	1793 lbs.	1793 lbs.	1913 lbs.	See EG-40 thru. EG 126	2450 lbs.

*Specifications may vary with optional equipment added.

General Equipment Description
Handling CNC Equipment

Warning: Forklift or other equipment should be operated by trained, operating personnel only with good understanding of materials handling safety policy, and procedures. Any forklift or other equipment operator should be completely familiar with the equipment, its operation, and functions before unloading or moving any CNC system or equipment.

Important: Use fork lift with fork extensions for unloading and moving the CNC equipment when required. See picture.

Lift Here   ⊢ Use Fork Extensions ⊣
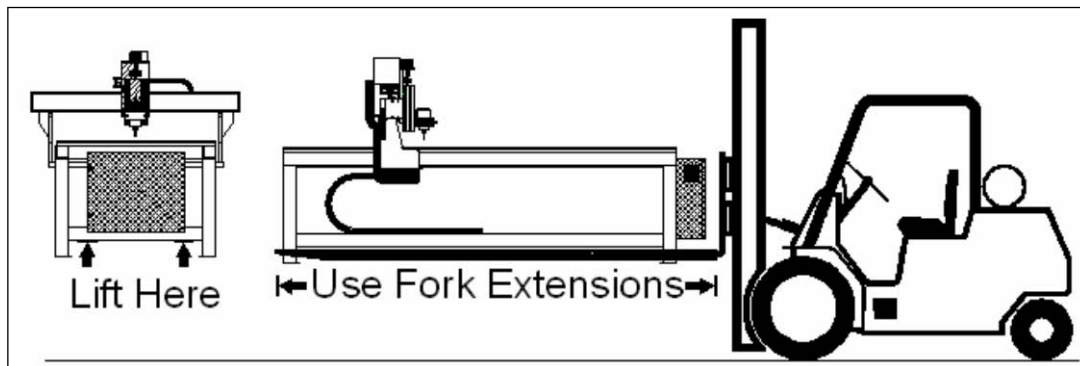
Warning: CNC equipment may not be balanced. Use caution when moving or unloading so as not to damage CNC equipment, fork lift, or operator. See picture**.**

Load May be Unbalanced
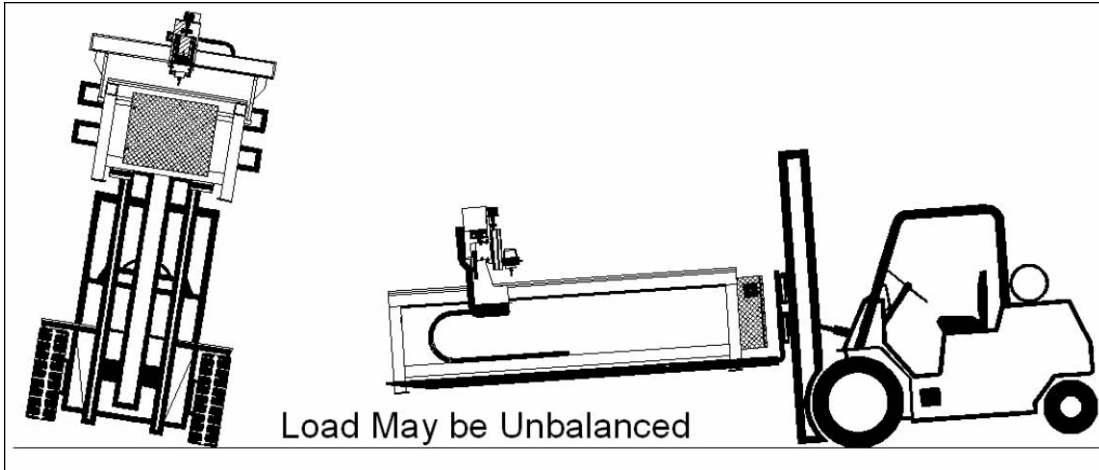
Final Positioning of Equipment*

The final position of the equipment will depend greatly on customers location. Allow for adequate spacing around the equipment for loading / unloading of materials, clamping material, access to vacuum valves. Eagle Eye recommends an access path of at least **3 feet** on three sides of the CNC Equipment and a material **staging area extending 8 or 9 feet** from the loading end of the equipment.
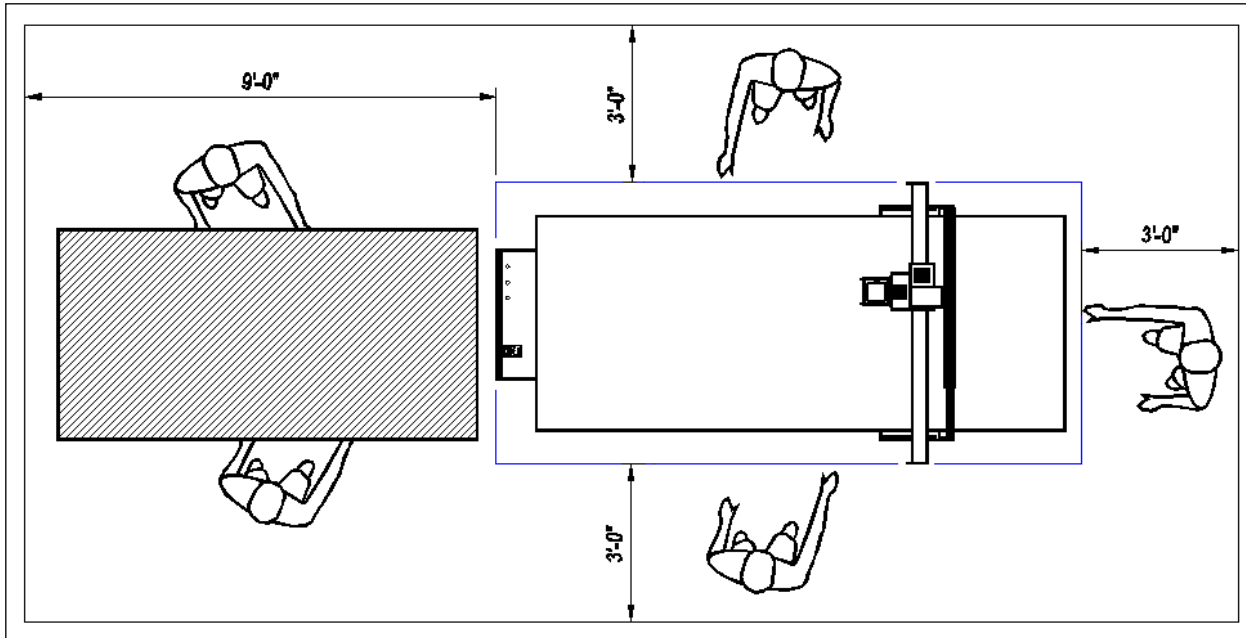
*Final positioning or location of any equipment will be by customer decision.

⚠️ **Warning:** All moving parts must be clear of any obstruction to allow free movement at all times during system operation.

⚠️ **Warning:** Emergency stop should be accessible at all times during system operation. Do not cover, block, or obstruct access to the Emergency Stop or any system controls.

⚠️ **Warning:** Service disconnect should be accessible at all times during system operation. Service disconnect should never locked in during system operation.

Leveling CNC Equipment

The CNC table must be checked for level at several points across the tabletop surface using at least a 4 foot level. Shims of wood, steal, plastic, or rubber may be used to bring table to level with in 1/16 of an inch rise or fall in an 8 foot span. Table should have no teeter in any direction. Should leveling feet be necessary, these may be purchased separately or from Eagle Eye. Leveling feet may also be available from various machinery supply houses, or hardware centers (e.g. McMaster Carr, Grainger, or Reed Tool).

CHECK EQUIPMENT FOR LEVEL AT SEVERAL POINTS
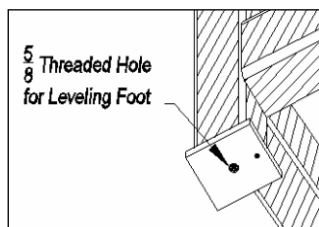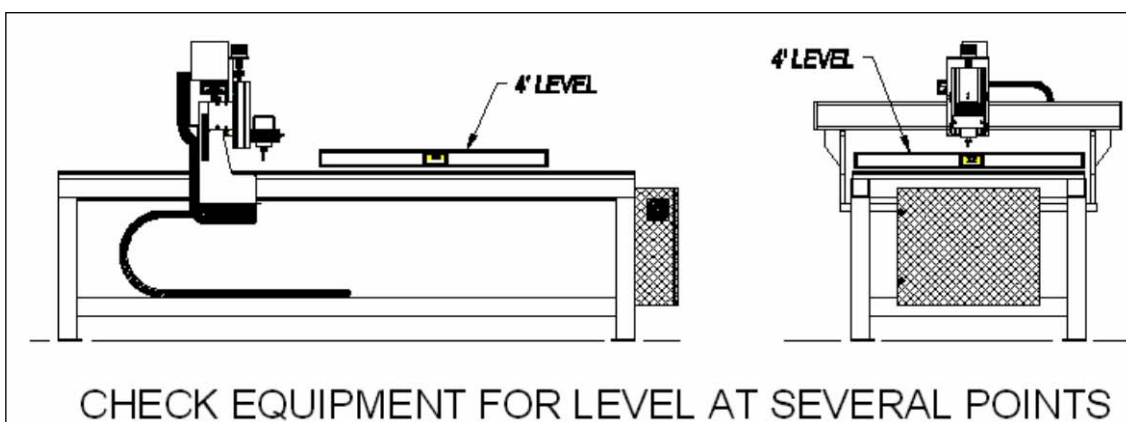
NOTE: Each table leg comes factory drilled and tapped to receive 5/8" UNC thread leveling foot. Leveling feet are not included with the CNC System. (sizes do vary)

Important: Table top surface may not be true. Milling is normally required for truing the tables surface. Customer is responsible for table milling, and truing on any of the equipment working surfaces. Contact Eagle Eye for table milling files if these are not included with the purchase of the CNC System.

Locating the Control PC
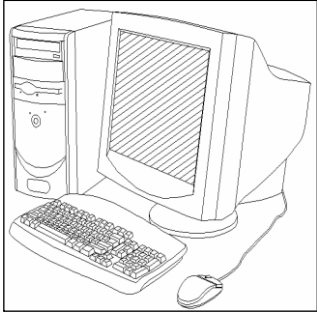
The control PC is part of the CNC system that requires an area for access by the system operator. **Eagle Eye does not provide a stand, or enclosure for the control PC.** The control PC key board and mouse should be easily accessible and functional at all times during system operation. The control PC monitor should also be in plain view by the system operator. The control PC tower should be kept in a ventilated, dust free environment to prevent overheating, and/or malfunction. The control PC CDR drive or Floppy drive should be easily accessible to the operator.
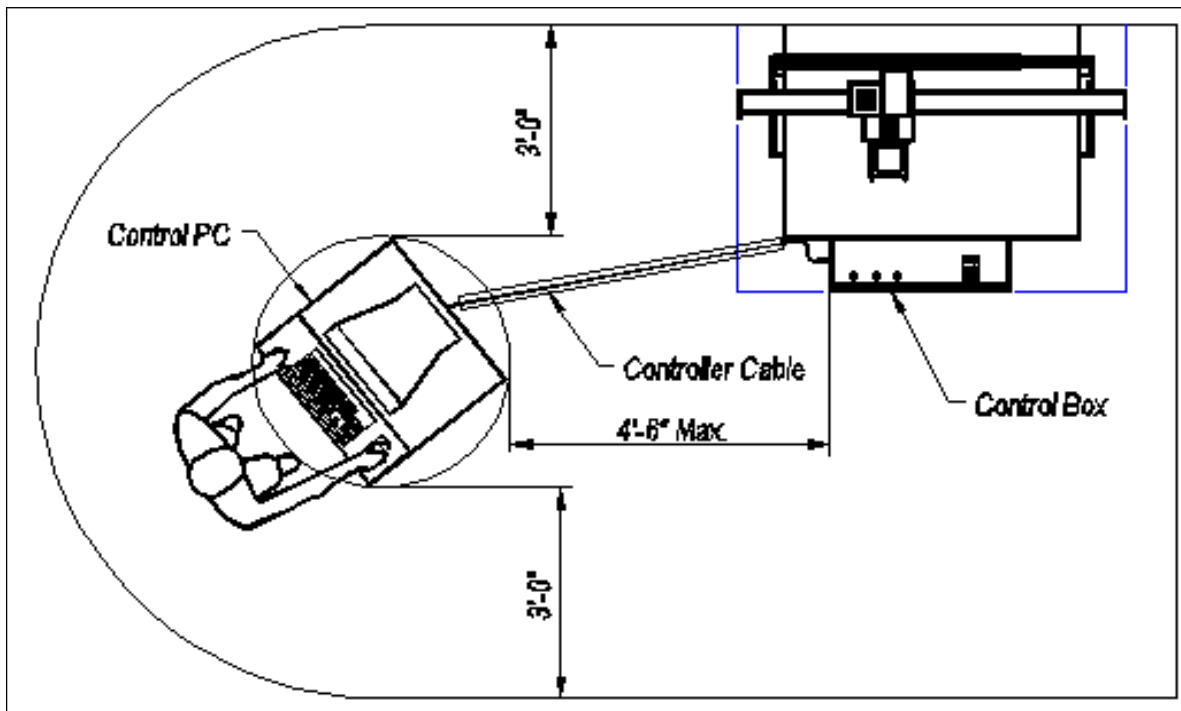
Warning: Protect and secure any cables, hoses or part of the CNC system crossing access the path around the CNC equipment.

Connecting to Electricity
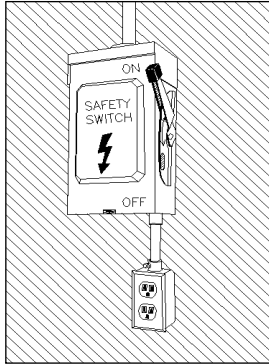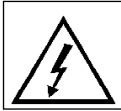
Electrical power to the CNC system is provided by customer. System power should be supplied through an easily accessible fused service disconnect switch or circuit breaker. All system accessories such as vacuum hold down, dust collection, etc. should also be connected with an adequate service disconnect or breaker. Control box maps are provided for approximate power input locations. Input locations may vary by model.
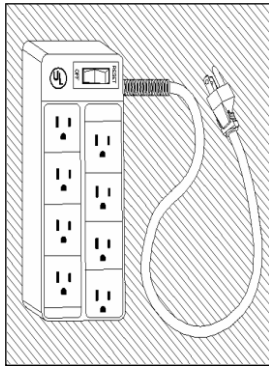
U.L. approved or equivalent **over current, spike, or surge protection should always be used to connect** equipment such as **the Control PC and other accessories** requiring 120v up to 15 amps. Any equipment requiring 120v over 15 amps should be supplied by fused service disconnect switch or breaker.

Important: Any electrical work should be done by qualified, licensed, electrician.

Important: All customer electrical inputs must meet NEC and local electrical codes.

Important: The CNC system should be properly grounded.

Warning: Service disconnect should be accessible at all times during system operation. Service disconnect should never locked in during system operation.

Control Box Map for Stepper Drives

CUSTOMER
INPUT
AC 120V

-X- -Y- -Z-
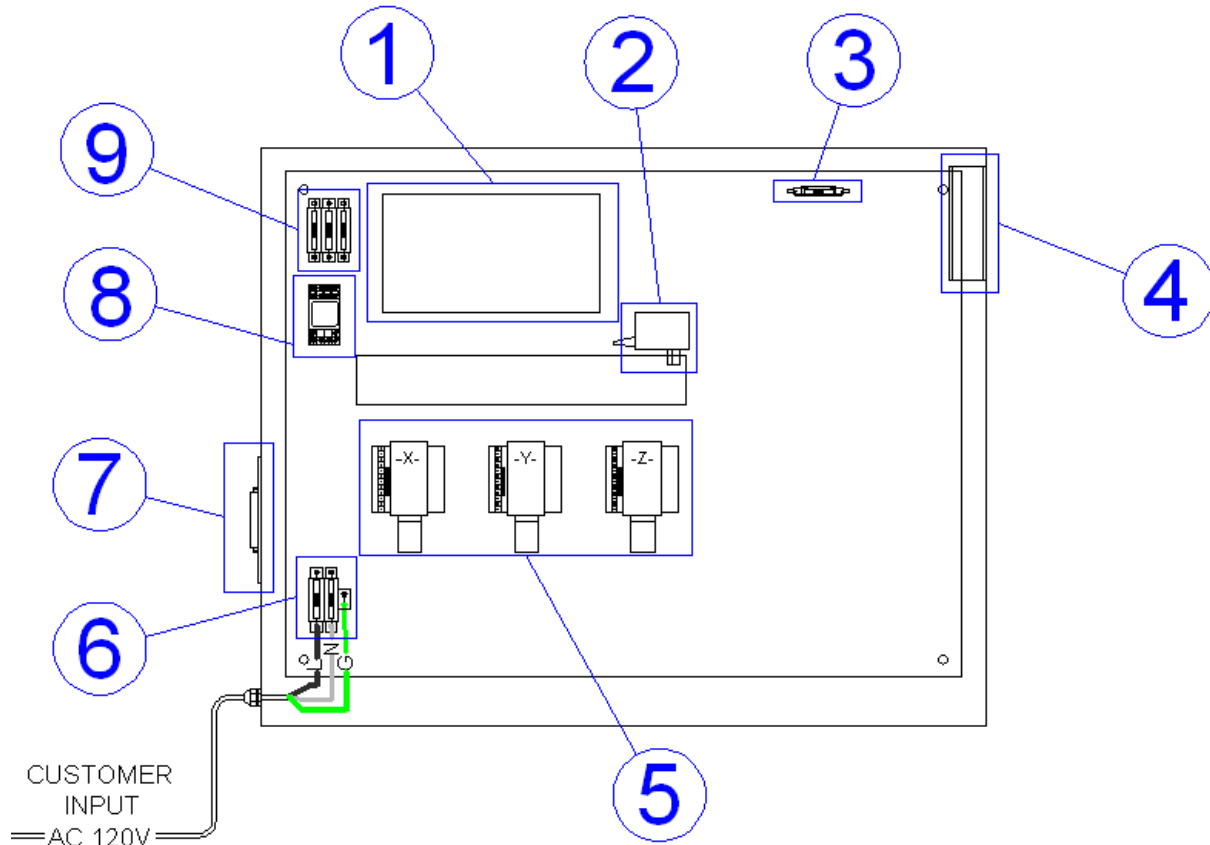
KEY	
1	65v Power Supply
2	12v Power Supply
3	Fuse 120v to Cooling Fan
4	Cooling Fan
5	Micro Stepper Drivers
6	110v / 120v Input Fuse Teminal
7	37 Pin Port
8	Control / Estop Relay
9	Fuses

Note: All customer electrical inputs must
meet NEC and Local electrical codes.

Control Box Map for Servo Drives With 4th Axis

CUSTOMER
INPUT
AC 120V
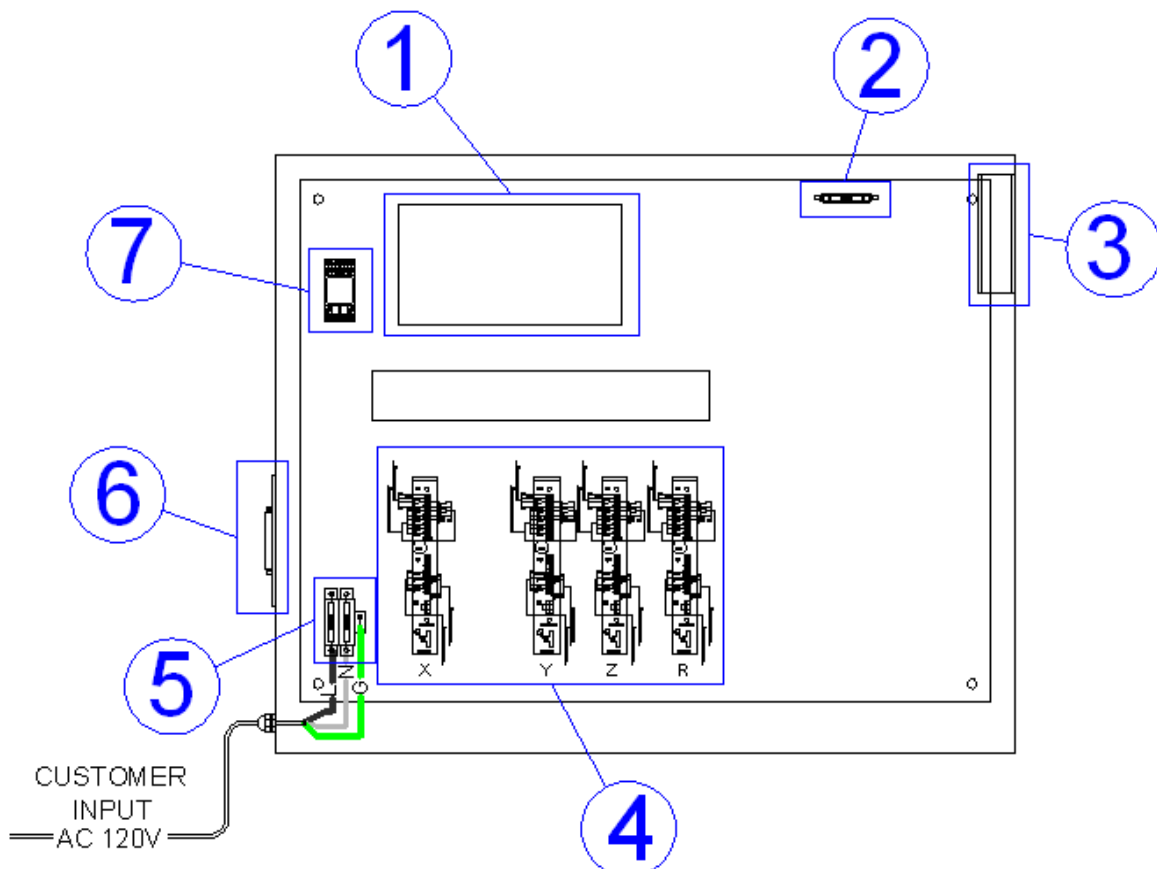
KEY	
1	65v Power Supply
2	Fuses 120v to Cooling Fan
3	Cooling Fan
4	Teknic SST Servo Driver
5	110v / 120v Input Fuse Teminal
6	37 Pin Port
7	Control / Estop Relay

Note: All customer electrical inputs must meet NEC and Local electrical codes.

Control Box Map for Stepper Drives W/ Spindle

CUSTOMER
INPUT
AC 120V

CUSTOMER
INPUT
AC 240/480V

CUSTOMER SUPPLIED
BREAKER / DISCONNE
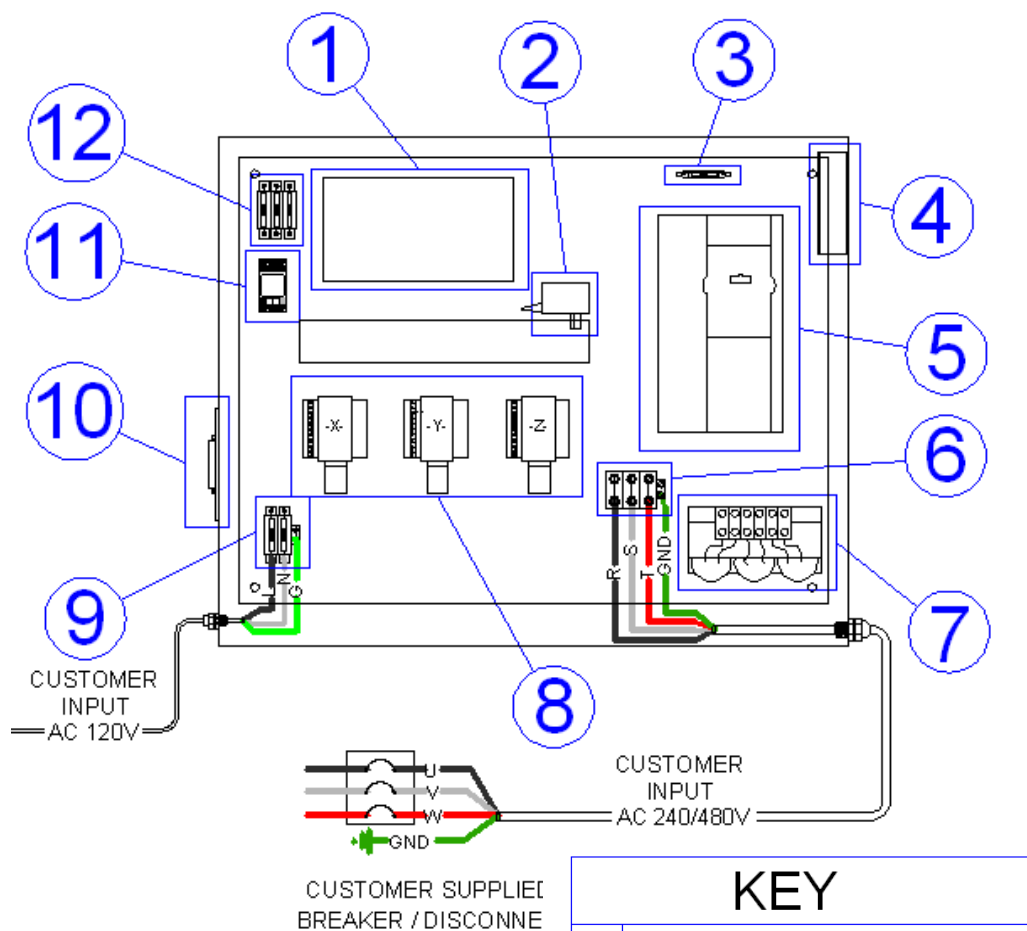
KEY	
1	65v Power Supply
2	12v Power Supply
3	Fuses 120v to Cooling Fan
4	Cooling Fan
5	Delta VFD Inverter
6	240v / 480v Input Line Terminal
7	Line Reactor
8	Micro Stepper Drivers
9	110v / 120v Input Fuse Teminal
10	37 Pin Port
11	Control / Estop Relay
12	Fuses

Note: All customer electrical inputs must meet NEC and Local electrical codes.

Control Box Map for Servo Drives W/ Spindle

① ② ③

⑪

⑩

⑨

⑧

④

⑤

⑥

⑦

CUSTOMER
INPUT
AC 120V

U
V
W
GND

CUSTOMER
INPUT
AC 240/480V
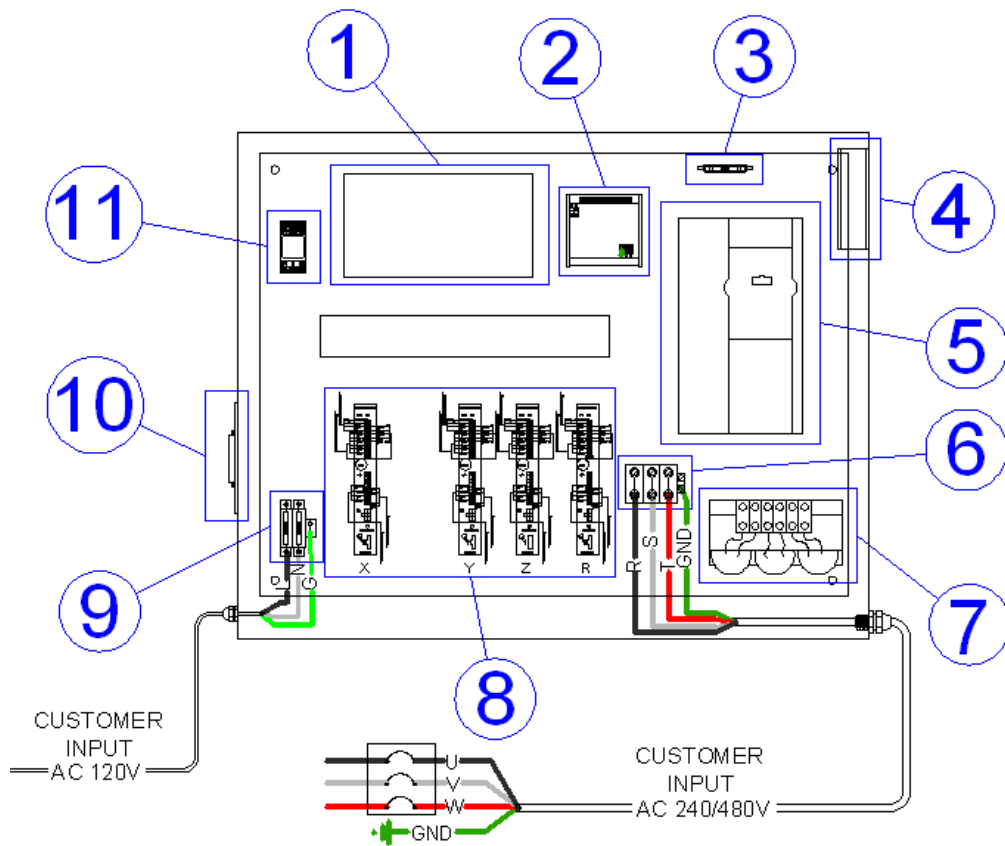
CUSTOMER SUPPI
BREAKER / DISCON

KEY	
1	65v Power Supply
2	24v Power Supply
3	Fuse 120v to Cooling Fan
4	Cooling Fan
5	Delta VFD Inverter
6	240v / 480v Input Line Terminal
7	Line Reactor
8	Servo Drivers
9	110v / 120v Input Fuse Teminal
10	37 Pin Port
11	Control / Estop Relay

i

Note: All customer electrical inputs must
meet NEC and Local electrical codes.

Electrical Input Requirements

	Input Volts	Input Frequency Hz.	AC Motor Drive	Breaker Size
Controls				
Standard Control Pc	110 - 120 v AC 1 Phase	50-60 Hz.	N/A	10 amp
Standard Control Box	110 - 120 v AC 1 Phase	50-60 Hz.	N/A	20 amp
ATC Control Box	230 - 460 v AC 3 Phase	50-60 Hz.	5 - 10 hp VFD	60 amp
Laser Controller	110 - 120 v AC 1 Phase	50-60 Hz.	N/A	20 amp
Cutting Head				
Spindle 3 hp	230 - 460 v AC 3 Phase	50-60 Hz.	3 hp VFD	20 amp
Router 3.25 hp	110 - 120 v AC 1 Phase	50-60 Hz.	N/A	20 amp
Spindle 5 hp	230 - 460 v AC 3 Phase	50-60 Hz.	5 hp VFD	30 amp
Spindle 6 hp	230 - 460 v AC 3 Phase	50-60 Hz.	6 hp VFD	45 amp
Spindle 7.5 hp	230 - 460 v AC 3 Phase	50-60 Hz.	7.5 hp VFD	45 amp
Spindle 10 hp	230 - 460 v AC 3 Phase	50-60 Hz.	10 hp VFD	60 amp
Laser	230 - 460 v AC 3 Phase	50-60 Hz.	N/A	60 amp

See accessories manufacturers specifications for electrical requirements.

ⓘ **Important:** Any electrical work should be done by qualified, licensed, electrician.

ⓘ **Important:** All customer electrical inputs must meet NEC and local electrical codes.

Connecting Control PC To Control Box

| Step 1 | Step 2 | Step 3 | Step 4 |
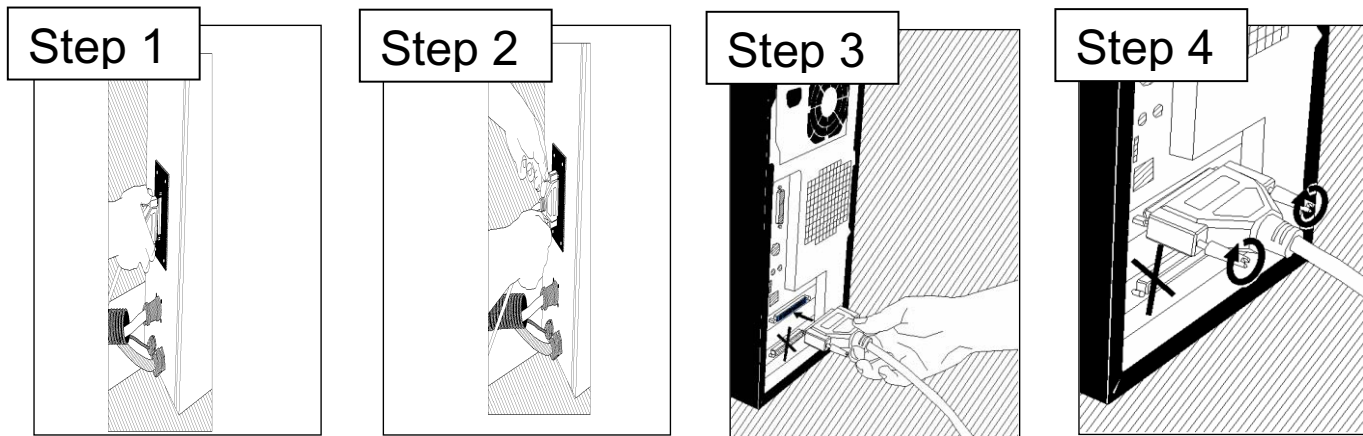
[i] **Step 1**: Connect 37 pin Control Cable to Control Receptacle located on the side of the frame mounted Control Box. (Note: Control Receptacle location may vary with system model)

[i] **Step 2**: Tighten Control Cable grounding / securing screws at Control Receptacle "hand tight". (Note: Never over tighten screws. Over tightening may damage screw threads.)

[i] **Step 3**: Connect the other end of the 37 pin Control Cable to the **correct port**. The position of the port vary depending on make and model of the Control PC. **Standard** systems require **1 Control Cable**, attaching to the **BLUE port** located at the rear of the Control PC.

[i] **Step 4**: Tighten Control Cable grounding / securing screws at Control PC Port "hand tight". (Note: Never over tighten screws. Over tightening may damage screw threads.)

Step 5: Connect Control PC monitor, mouse, keyboard, and power cords according to the PC manufacturers instructions.

For Control PC disconnect from Control Box, reverse steps 1 – 4.

Connecting to Compressed Air
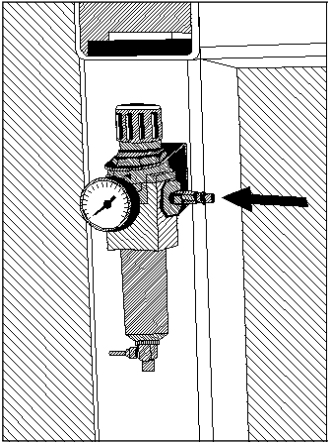
Compressed air connections are supplied by customer.
Standard CNC Systems do not require compressed air for operation. Compressed air is required for Laser, Automatic Tool Change Systems (ATC), and Quick Tool Change Systems (QTC) operation. Compressed air input for ATC, QTC, and Laser is located on the front left table frame leg near the Equipment Control Box. The air supply connection is by ¼" male ARO type quick disconnect hose coupler.
Compressed air for use on the CNC System should be:
(1) Dry
(2) Filtered
(3) Regulated @ 100 PSI w/ at least 25 CFM.
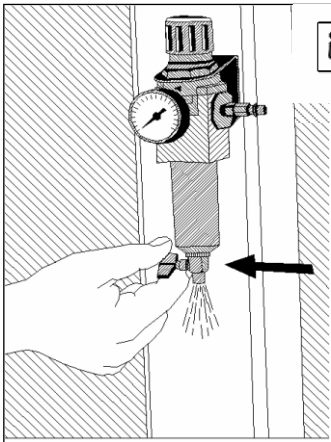
Important: Drain all moisture traps Daily to prevent damage to spindle or other parts.
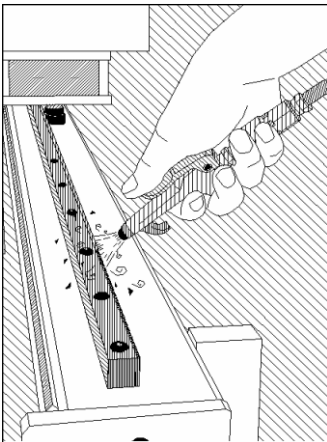
A separate compressed air line is necessary for general equipment maintenance. An air nozzle should be used to blow off the equipment after each shift. Exposed mechanical parts such as the X-Axis geared rack and pinion gear require special attention to keep equipment operating properly. The table cutting surface should be cleared after each cutting job.

Installing Bit And Collet

Step 1	Step 2	Step 3
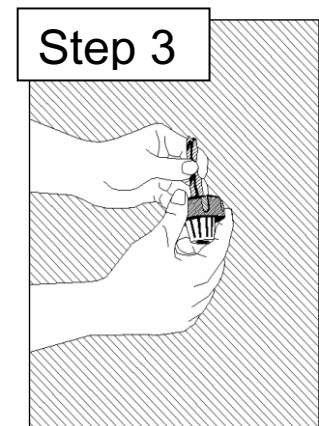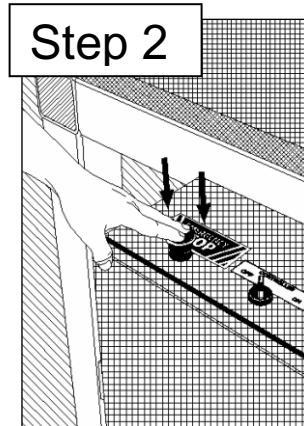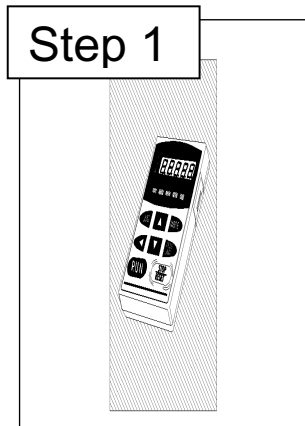
Step: 1   Stop Spindle Rotation

Press the "Stop / Reset" button on the spindle drive control pad. (Applies only to models with High Frequency Spindle) Spindle must be completely stopped before changing or installing bit and collet.

Step: 2   Engage Emergency Stop

Press the "Emergency Stop" button on the control box . (Applies to all models) The Emergency Stop will disable all control drives and cutting head devices such as router or laser. Cutting head must be completely stopped before changing or installing bit and collet.

Step: 3   Insert Bit into Collet

Many bits come with a protective coating which must be removed before inserting a bit into a collet. Most bits come with either 1/4" or 1/2" shank size. The collet size must match the bit shank size. **Use only manufacturers specified collets**.

Warning : Take care not to drop or damage bits. Do not use bits that are bent, broken, cracked, chipped, or damaged any way. Damaged bits are hazardous.

Warning : Bits have sharp edges and should be handled with care to prevent injury.

Installing Bit and Collet Continued

Step 4	Step 5	Step 6	Step 7
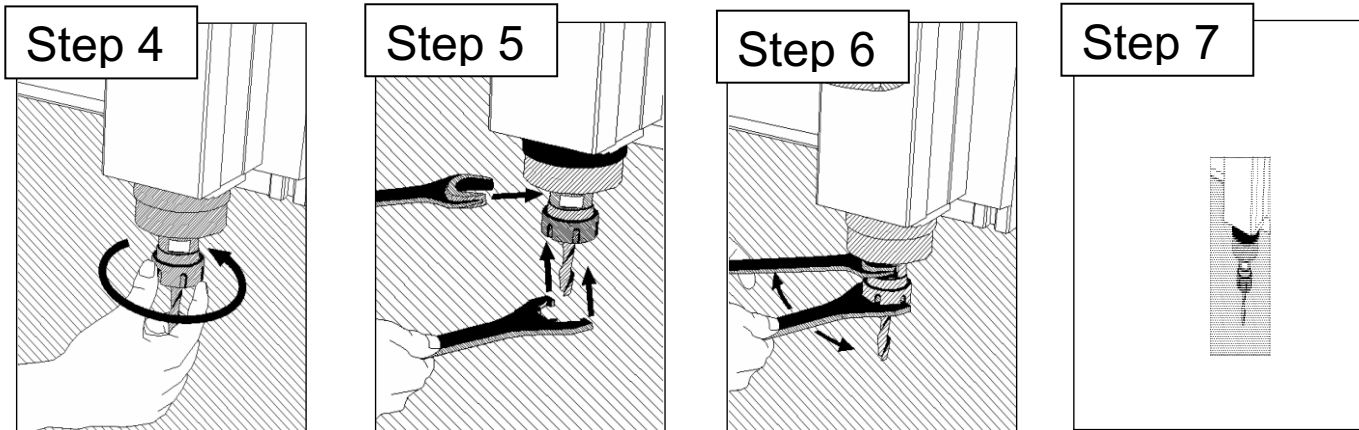
Step: 4   Insert Collet and Bit

The collet is inserted upward into the threaded receiver cone with the cutting end of the bit pointing downward. Hand tighten the collet nut counter clockwise looking from the top of the spindle / router downward.

Step: 5   Position the Collet Wrenches

Use only collet wrenches specified by the spindle / router manufacturer. Wrenches should be included with the CNC System. Additional wrenches may be purchased separately.

Step: 6   Tighten the Collet Nut Securely

Step: 7   Remove Wrenches From Cutting Head

The wrenches should not be left on the cutting surface during operation. Store wrenches in a visible and easily accessible location, on or close to the CNC System. The front right open end of the table frame is a great location for storing wrenches.

Warning : Take care not to drop or damage bits. Do not use bits that are bent, broken, cracked, chipped, or damaged any way. Damaged bits are hazardous.
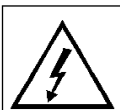
Warning : Bits have sharp edges and should be handled with care to prevent injury.

Vacuum Hold Down System

Connection to the vacuum hold down system is by the customer, including any electrical connection and any vacuum piping / plumbing beyond the control manifold.
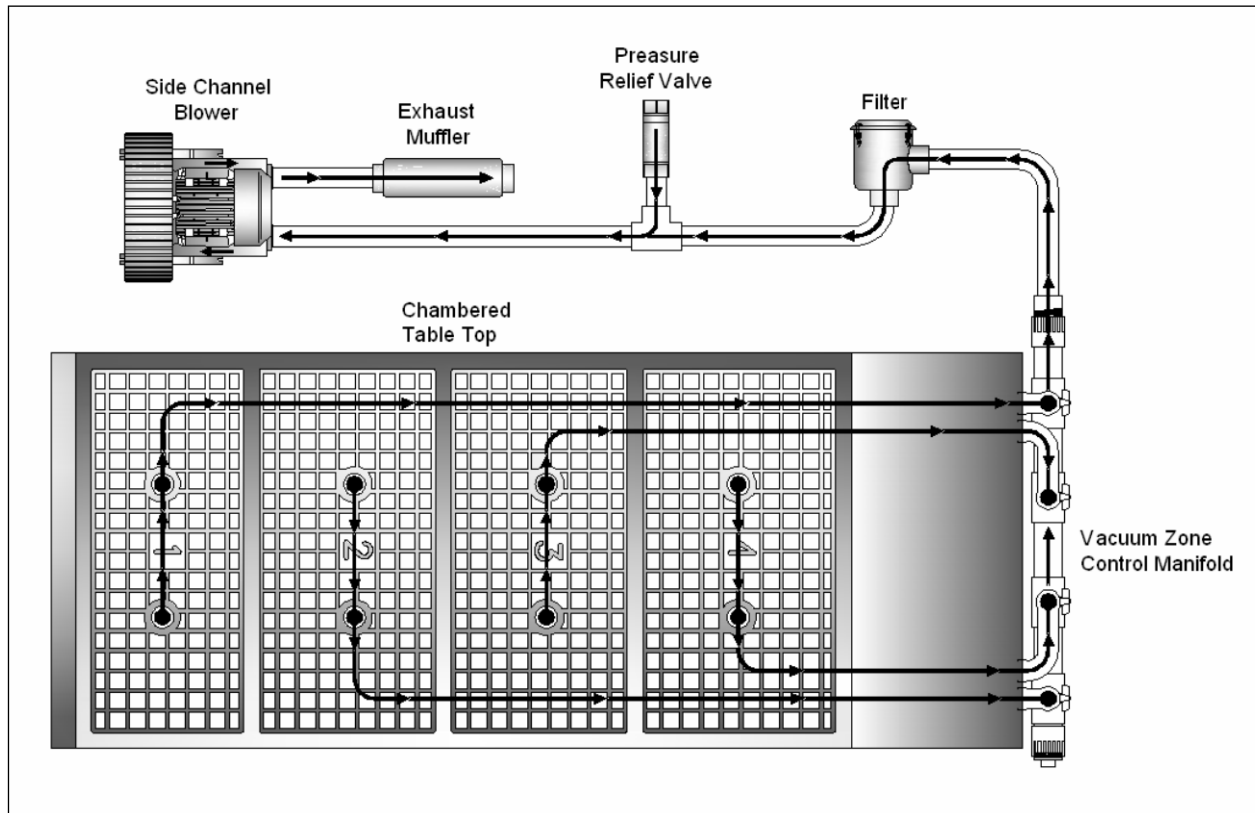
Schematic of vacuum hold down system.
Actual configuration may vary.

Important: Any electrical work should be done by qualified, licensed, electrician.

Important: All customer electrical inputs must meet NEC and local electrical codes.

Warning: Service disconnect should be accessible at all times during system operation. Service disconnect should never locked in during system operation.

Vacuum Hold Down System Continued

Vacuum Hold Down Table Top
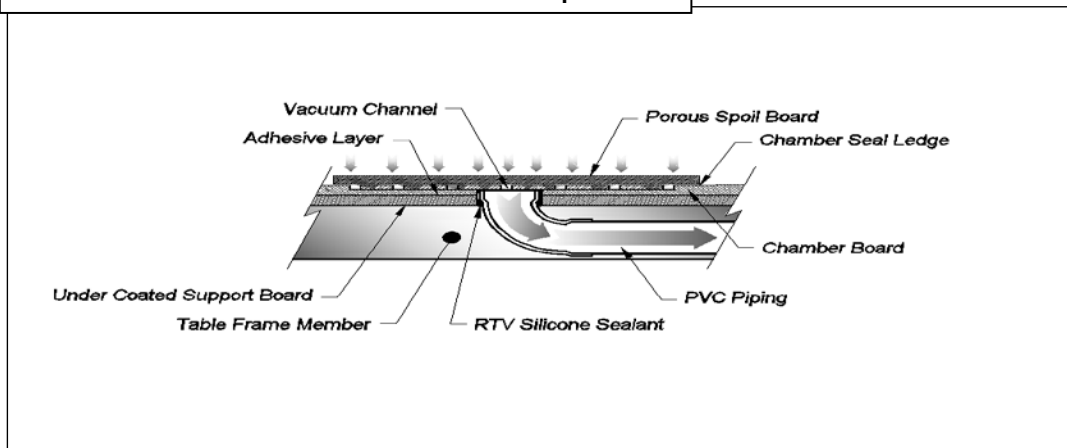
The vacuum hold down system is made up of several components:

Vacuum Source – Vacuum is created by using a motor driven fan to create suction. The vacuum source may be a side channel blower, regenerative blower, rotary vane fan or other source capable of creating enough suction force to hold down material during cutting. Blowers are available from 5hp up to 20hp depending on customers' requirements. Most vacuum systems works well with blowers 10hp to 15hp. Blowers require a dust filter, pressure relief valve, and a muffler to reduce noise. Customer is responsible for any plumbing or wiring. See manufacturers' specifications for wiring and plumbing requirements. Vacuum configuration may vary with customers' requirements.

Porous Spoil Board - The top layer of the vacuum system. The spoil board protects the vacuum chamber board from being routed, or damaged during cutting operations. Suction from the vacuum pump pulls through the spoil board to hold material in place. **MDF** (Medium Density Fiberboard) or **LDF** (Low Density Fiberboard) are good spoil board materials. When job material is routed through, the bit will route into the spoil board. The spoil board may need to be milled from time to time, and replaced as necessary. Use a planing bit to mill the surface the Spoil board. **Do not route through the spoil board!**

Vacuum Chamber Board - The middle layer, and has channels through which air flows into the piping of the vacuum hold-down system. The vacuum chambers are grids, and are divided into separate, which can be selected with the ball valves at the back of the table. The Chamber board should be sealed to increase suction through the spoil board.

Vacuum Piping and Valves – PVC piping connects the Chamber Board through the control valves, to the vacuum manifold, to the filter, to the pressure relief valve, then to vacuum source. Vacuum zones are selected with the ball valves at the back of the table. Any plumbing beyond the control manifold is the supplied by the customer. Plumbing parts are available at many hardware or home centers.

<u>System Maintenance</u>

[i] **Important:** System maintenance should be performed regularly to keep any CNC System in good operating condition.

⚠ **Warning:** Always engage Emergency Stop Button before performing any maintenance procedure.

<u>Between Cutting Jobs</u>

Clear cutting area of debris, chips, and unused or scrap material.

Check condition of the bit or cutting tool, replace if needed.
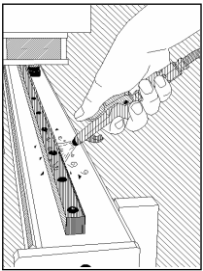
<u>Daily or End of Shift</u>

Same as above.

Use air nozzle to blow off the equipment especially exposed mechanical parts on X-Axis and Z-Axis.

Clean up work area and access area around the CNC System.

Drain moisture traps on compressed air lines.

Inspect all compressed air regulators for correct air pressure

<u>Weekly</u>

Same as above.

Use air nozzle to clean out chips, dust, and debris from the wire and cable carriers.

Lightly oil any exposed steel surfaces such as bearings, guide rails, geared racks, pinion gears, and ball screws. Use only a light machine oil applied with a soft, clean rag. Aluminum and painted surfaces do not require oiling.
Back up CNC cutting files.

[i] **DO NOT USE SOLVENTS OR CORROSIVE CLEANERS / LUBRICANTS ON ANY PART OF THE CNC EQUIPMENT.**

System Maintenance Continued

ⓘ   System maintenance should be performed regularly to keep any CNC System in good operating condition.

⚠   Always engage Emergency Stop Button before performing any maintenance procedure.

⚡   System must be completely powered down from the service disconnect before inspecting or cleaning the control box!!

Monthly

Same as above.

Inspect all equipment for excessive wear, or damage.

Inspect dust filters, clean or replace as necessary.

Open and inspect control box for chips, dust, and debris. If chips, dust, or debris are present clean out with vacuum cleaner. Use care not damage wires, or any electrical components in the control box .

Annually

Same as above.

Contact Eagle Eye for servicing and maintenance of the CNC System as required.

CNC System Operating Procedure

1. Turn on main power to CNC System at the service disconnect switch / breaker.

2. Power up the Control PC, and wait for the computer to boot.

3. Open the WinCnc program.

4. Disengage the Emergency Stop.

5. Turn the Controller Power switch to the ON position (clockwise).

6. Wait approximately 10 seconds.

7. Turn the Servo Enable switch to the Enable position (clockwise).

8. Check limit switches, making sure indicator lights illuminate when touched

9. Click the Home macro button on the WinCnc display console.

10. Secure job material to the table.

11. Insert cutting tool properly.

12. Set the X and Y Local Home to the front left corner of the job material.

13. Set the surface height for the job material.

14. Select the correct job file from the correct drive or port on the Control PC.

15. Start the cutting head.

16. Run the file.

CNC System Shut Down Procedure

1. Make sure all files have stopped running, abort the file if it is running.

2. Turn off the Servo Enable switch to **Disable** position (counter-clockwise).

3. Turn the Controller Power switch to the **Off** position (counter-clockwise).

4. Engage the Emergency Stop button before.

5. Exit the Wincnc Program.

6. Shut down the Windows program.

7. Turn off main power to after the Control PC has completely shut down CNC   System at the at the service disconnect switch / breaker.

Tool Breaks While Cutting

1. **Pause** (**Space Bar)** then **Abort** (**ESC)** the WinCNC software.

2. **Note of the current line number** located in status bar at the bottom of the WinCNC Display console.

3. **Turn Off the cutting head** at the Spindle Control Pad or the On/Off Switch atop the router.

4. **Engage the Emergency Stop Button**.

5. **After all machine components have completely stopped:**

6. **Remove, and Replace the broken tool**.

7. **Reset the Z- Axis zero** by using the Touchtop macro button and the Touchpad.

8. **Turn On the cutting head** at the Spindle Control Pad or the On/Off Switch atop the router.

9. Use the **Restart** button in the WinCNC Display console tool bar **to restart the job**. **Enter the correct Job File Name and Line Number.**

10. **Click OK**. The machine will continue the job where it left off.

CNC Control Software

The CNC System control software is WinCnc Controller written by Microsystems of Buckhannon, Inc. This section is a condensed guide for using WinCnc Controller.
Included are:
Screen Display Console
User Interface Commands
Basic Operation
Keyboard Shortcut Commands
G-Code Commands

 This Guide provides some basic information on how the software interfaces with your CNC machine for a better understanding of how your Machine works. If more information is necessary contact Eagle Eye CNC Inc.

Background Information

WinCnc uses G-code to communicate instructions. Each code is distinct, and affects the system in a different way. Frequently used codes are discussed in this document. A complete list of G-Codes and their functions is provided in the back of this manual.

Control software and the CNC System require an operator. WinCnc serves primarily as a control signal source and only receives information from the systems' mechanical limit switches. No other information is relayed from the CNC Machine to the Control PC. The operator should synchronize the CNC System at every start up. This is to ensure the CNC Machine and the Control Software are working together properly. This is done by '**homing**' the machine. Home the machine whenever you have shut off your system, it loses power, or anytime the two system parts are out of sync.

Screen Display Console

This is a sample of the WinCnc software display.

Menu Bar Pg. 14-17

Viewer Window Pg. 13

Tool Bars Pg. 4 & 5

Command Line Pg. 6

Message Display Window Pg. 6

Axis Window Pg. 3

Transit Speeds Pg. 4

Jog Increments Pg. 7

Console Transit/Jog Buttons Pg. 7 & 8

Custom Macro Buttons Pg. 10 & 11

Status Bar Pg. 3
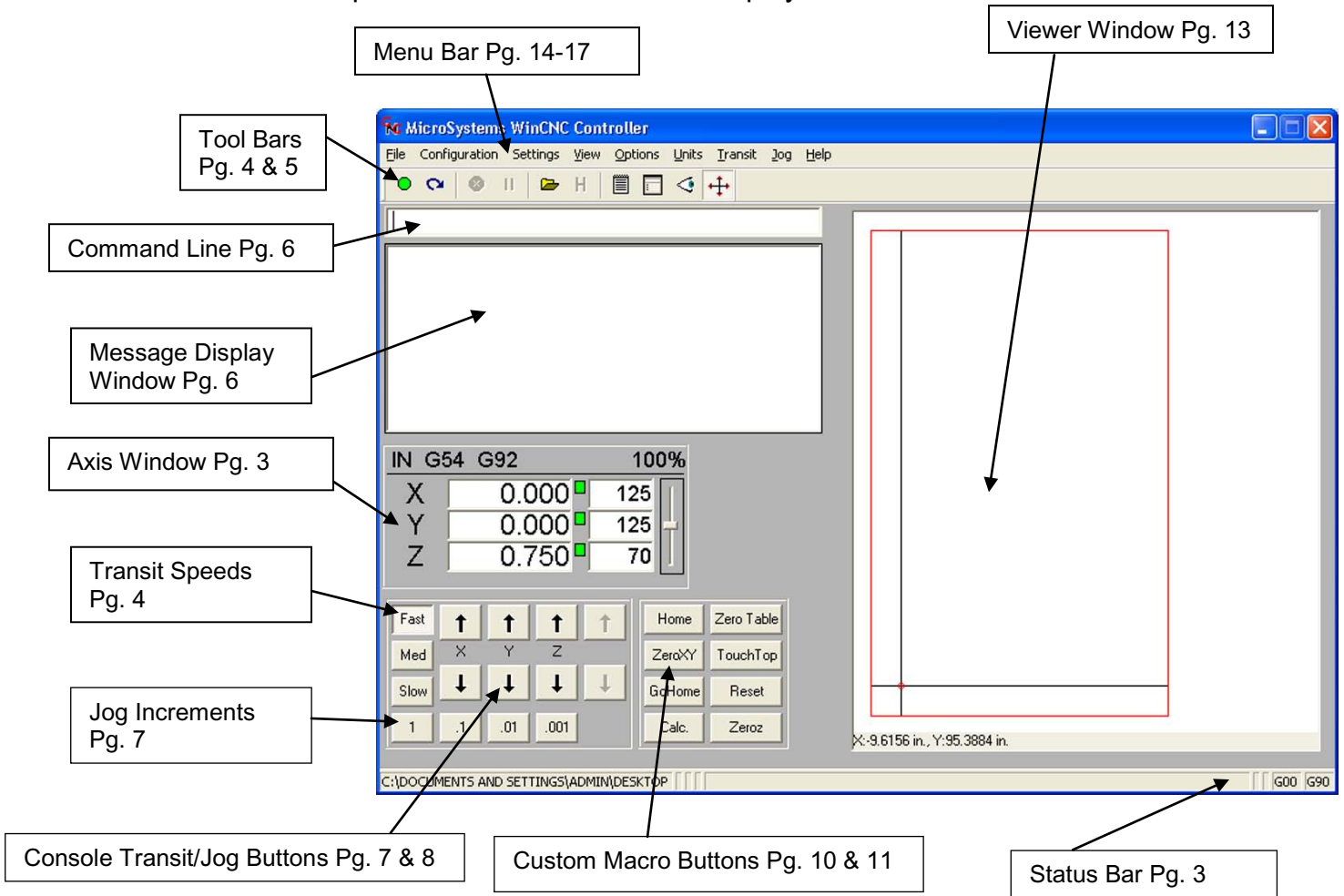
Most machine functions are preformed from the display console buttons or from keyboard commands. For reasons of simplicity and functionality, we will cover the onscreen features. Once that is fully understood, covering some of the deeper features of WinCNC controller found in the "Menu Bar" would give you a clear understand of the potential of the Machine. But for a quick reference, page numbers are provide for each of the program's different components.

Screen Display Functions

The Axis Information Window

The Axis Window gives information about the status of your Machine.

Hardware or Software Limit Indicator
Red Arrow when limit is reached.
Left Side Arrow   =   Low Limit
Right Side Arrow = High Limit

Relative Mode Indicator [G92]
Green Light   =   Local Zero Set

Feedrate Override Slider Bar
Adjusts Feedrate manually.
Slide Up   =   Increase in./min.
Slide Down   =   Reduce   in./min.

```
IN  G54  G92                    100%
◄ X        0.000 ▪    125
  Y ►      0.000 ▪    125
  Z        0.750 ▪     70
```

Axis Feedrate Display
 # = inches / minute

Axis Position Display

Zero Table Indicator [Z –Axis]
Blue Light   =   Zero Table Set
Tool Measure on ATC only

Alternate Workspace Indicator [G55]
Blue Light   =   Alternate Workspace Active
X–Axis and Y–Axis only.

The Status Bar

The status bar is the label along the bottom of the WinCnc window that looks similar to the one shown below. This bar provides the user with the status of several features in WinCnc. Each section in the example below is labeled for clarity.

Current File Directory

G00/G01 Mode Indicator

Current Job Line Number

```
C:\WINCNC\TEST                            Tool 2  Tool 4  G00  G90  Line:0        NUM
```

Current Tool Indicator

G90/G91 Mode Indicator
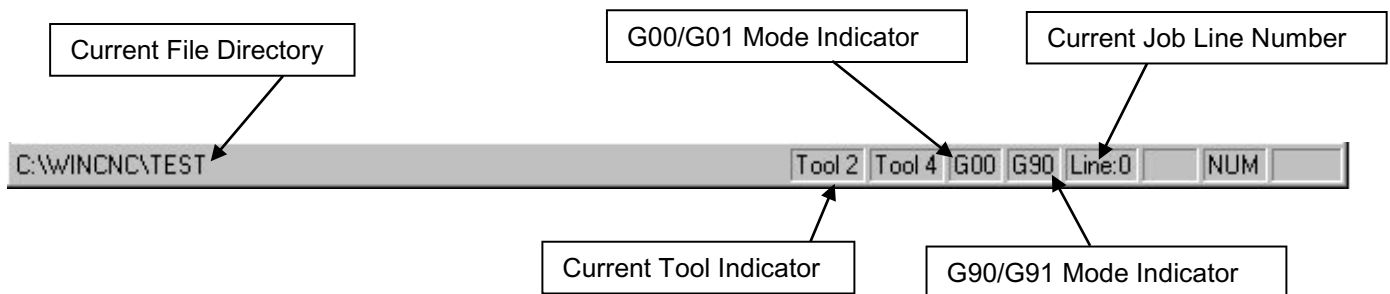
Operations Tool Bar

The tool bar is a collection of shortcut buttons that perform specific actions. The Toolbars in WinCnc are floating and can be positioned or docked to the user's preference. The image below shows both the standard and manual digitizing toolbars both docked in the normal toolbar area.
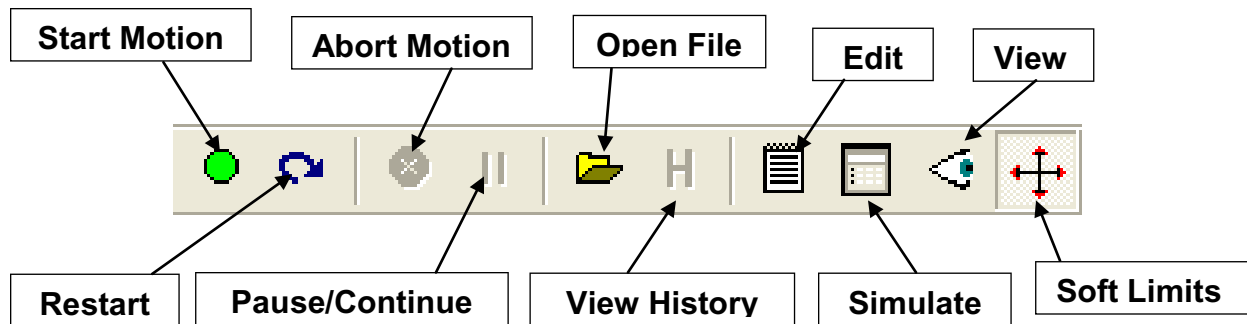
Start Motion: This button will begin the command or job listed in the command line.

Restart: Allows the operator to pick a job file and the line to start that file on. This is useful for skipping over lines in a job file, or if you have aborted a job and want to start back at the point you aborted from. If job is aborted, program remembers the last line, and will return safely there.

Abort Motion: Aborts a running command or job.

Pause/Continue Motion: Pauses/Continues a running command or job. Keyboard Shortcut: Space Bar

Open File: Opens a browse box used to open a job file.

View History: Opens the command history box, which allows operator to execute a command used previously.

Edit: Opens the default editor specified in the WinCnc.ini file. WinCnc uses notepad by default. The editor can be used to open job files, listed in the command line, in the editor.

Simulate: Simulates a file to check for errors and run-time.

View: Opens a file for viewing in the viewer window.

Soft Limits: Enables/Disables Soft limit features.

32

Manual Digitize Tool Bar

If you have a digitize probe, the seven buttons on the manual digitization toolbar are for use only with the Manual Digitize feature, and will only be visible after showing the manual digitize toolbar under the View->toolbars section of the menu bar.
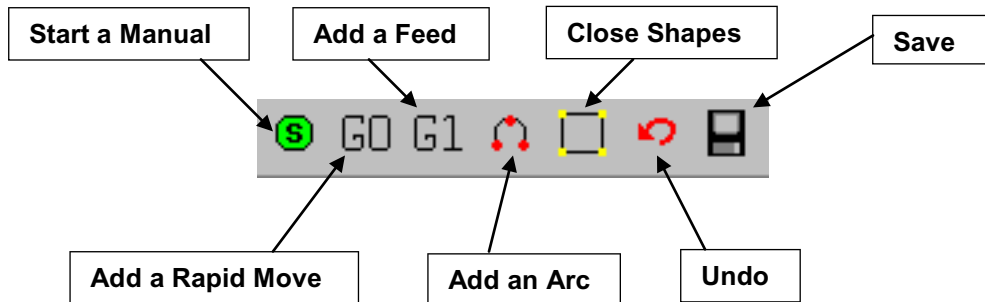
Start a Manual Digitized File: Starts a manual digitized file and enables the manual digitize mode.

Add a Rapid Move: Adds a rapid move to the manual digitized file.

Add a Feed Move: Adds a feed move to the manual digitized file.

Add an Arc Point: Used to add arcs into a manual digitized file.

Close Shape: Used to close the last move in a shape without moving the probe.

Undo: Used to undo previous moves.

Save: Allows you save the manually digitized file in G-Code or DXF file format.

33

The Command Line

```
WTEST.TAP
```

The Command Line is the user Input line for job file names or commands to be executed.

The example above is showing a job named "WTEST.TAP" is ready to run. All jobs end with the extension ".TAP". Once a job title appears in the Command Line, press the **Enter** key or the **Start Motion** button in the tool bar (pg. 8, 9) to run the job from the current zero positions.

The only commands appearing in the Command Line, other than the current job, are commands used for moving the cutting head to a specific point on the table by inputting straight G-Code. Example: Move to position X+ 3", Y+ 3", Z+ 1", type at the command line **X3Y3Z1** and press **Enter**. The machine will move to this position. This is done when more accuracy is needed in locating the cutting head over the cutting surface, than the arrow keys on the keyboard, or log buttons, allow (for more information on keyboard commands see pg 14, 15). Straight G-Code commands may also be entered here (a list of G-Code commands is in Appendix I), but only one code at a time. Creating a ".TAP" file or a ".MAC" (pg. 5) is best for running multiple code entries.

The Message Display Window

```
G3 X32.6715 Y9.5671 I2.7245 J-3.6553
G3 X32.1196 Y8.6453 I5.0934 J-3.6759
G3 X31.5805 Y7.0247 I5.2471 J-2.6456
G3 X31.5054 Y6.147 I5.1103 J-0.8792
G3 X31.5331 Y5.6214 I4.1797 J-0.0429
G0 Z2.1
G0 X0 Y0
Finished Running C:\WINCNC\JOBS\WINCNC.TAP
Runtime: 1 Minute(s) and 3 Second(s)
```
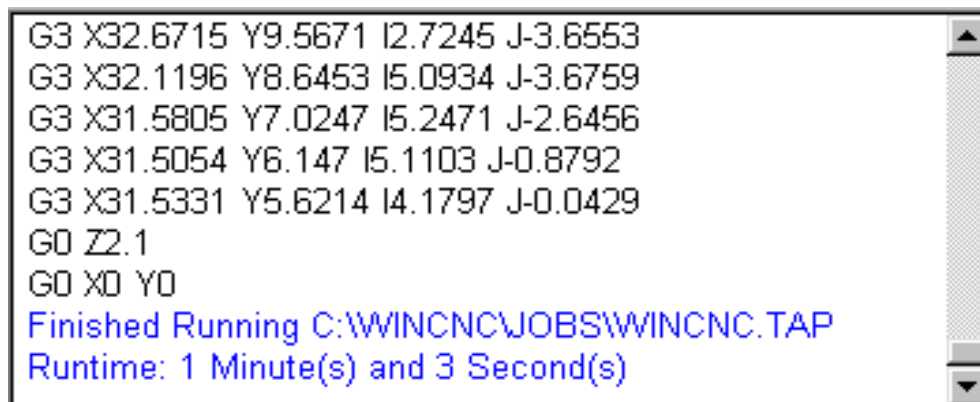
The message display window is the main output window displaying the commands that have been executed, messages to the user, or errors that have occurred during an operation.

Jog Increment Buttons
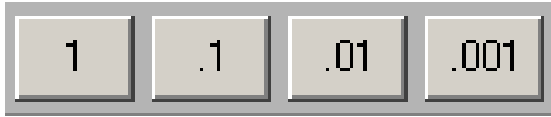
These buttons allow the operator to specify the increment of a jog move.

  = 1 Inch

  = 1/10 Inch

  = 1/100 Inch
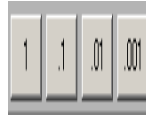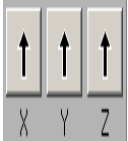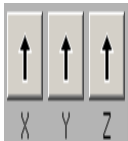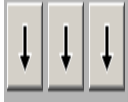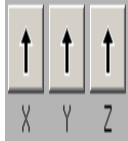
  = 1/1000 Inch

Transit/Jog Buttons
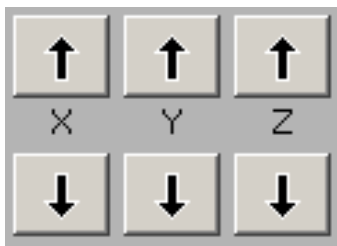
These buttons allow the operator to initiate a manual transit or incremental jog move from the Control PC display console. **Press & hold a button for continuous manual transit jog move** on the selected Axis. **Pressing the button Momentarily** will move the Axis in manual incremental jog mode only **one increment at a time**. Pressing a button momentarily will move the Axis according to the jog increment selected.

These actions can also be performed by using the arrow keys on your keyboard. The **Left and Right arrow keys are used to move the X-Axis** in X-(left) or X+ (right) direction. The **Up and Down arrow keys are used to move the Y-Axis** in Y+ (forward) or Y- (backward) direction. The **"Page Up" and "Page Down" keys are used to move the Z-Axis** in Z+ (up) or Z- (down) direction. The **"Home" and "End" keys are used to move the W-Axis** (Alternate Vertical Axis) in W+ (up) or W-(down). The **"Home" and "End" keys may also be used to rotate a the R-Axis** (Rotary Indexing Axis) R+ (clockwise) or R- (counter clockwise) direction when the CNC System is equipped with a Rotary Axis. **THE "HOME" KEY COMMAND DOES NOT "HOME" ANY SYSTEM AXIS**.

Axis	Direction	Keyboard	Console Button
X- Axis	+		
X- Axis	−		
Y- Axis	+		
Y- Axis	−		
Z- Axis	+		
Z- Axis	−		

36

Alternate Axis	Direction	Keyboard	Console Button
R- Axis	○		
R- Axis	○		
W- Axis	+		
W- Axis	−		
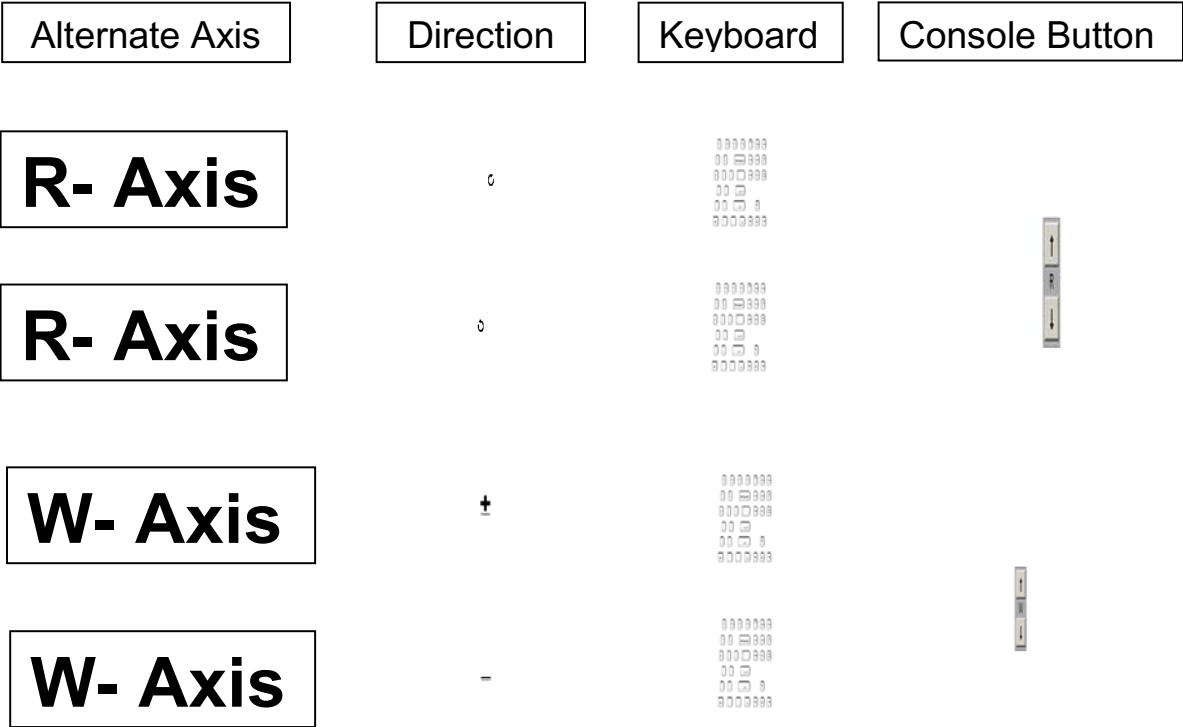
Transit Speeds Buttons

These buttons allow the user to specify the transit speed of the machine.

Transit Speeds	Machine w/ Stepper Motors	Machine w/ Servo Motors
Fast	800 (in/min)	1200 (in/min)
Medium	100 (in/min)	300 (in/min)
Slow	20 (in/min)	20 (in/min)

Custom Macro Buttons

The Custom Macro Buttons are custom control buttons which are shortcuts to activate commonly used commands. Each button uses a Macro File or a .MAC File installed in the control software. Following are some Custom Macro Buttons that come installed with the Machine CNC System and the g-code used to create them. Specific Macro buttons may vary with system model and options.
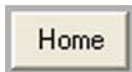
Home Synchronizes each Axis with WinCNC, and provides a reference point for software limits. Software invisible limits programmed into WinCNC to restrict the movement of any axis from over traveling its' workspace. When WinCNC Controller is started, the display is set to the last known position. The G28 command must be used to home the machine. G28 command (1) moves the Z-Axis up to its high limit. (2) moves the X-Axis to its far left limit. (3) moves the Y-Axis to its forward limit. The Home Macro button uses the G28 command to home the system.

Go Home Raises any vertical axis (Z and W Axis), to machine zero then returns the XY Axis to the Local Coordinate Zero.

ZeroZ Performs the same function as the ZeroXY macro button, except placing a local zero on the Z axis only.

ZeroXY Creates a local zero at the current XY position. ZeroXY uses the G92 command on the XY to create the Local Zeros. Although the Machine Zero is still stored in memory, the Local Coordinate System becomes active after using. A Machine Coordinate can still be accessed by using the G-Code command G53 and the desired position. Example: G53XYZ will take the machine to machine zero no matter what local coordinate are set. Using the G53 command doesn't erase the Local Zeros, but uses the Machine Coordinate System instead of the Local Coordinate System. (See page 2 - Local and Machine Coordinate Systems)

Reset Will remove any local coordinates, tool measures, or Zero Table boundaries that might cause WinCNC to give a boundary or software limit error after the CNC routers table or sacrifice material surface has been refinished, or an old tool as been replaced in a Machine equipped with ATC (Automatic Tool Changer).

Zero Table Used with the Touchpad to create a software boundary. This boundary is normally table top or top of the spoil /sacrifice material. Zero Table sets a software boundary beneath the cutting surface to prevent plunging a cutting tool too far below the work material.

TouchTop Creates a Z local zero quickly and accurately on a cutting material's surface. Placing a local Z zero on the material surface is necessary. It indicates where the cutting surface is located in the Z plane. The procedure for using the TouchTop command is similar to the Zero Table except the job material surface is measured instead of the table top surface.

Calc. Quickly opens the Windows Calculator for when quick calculations when necessary.

Menu Bar

The Menu Bar contains many of the main features of the WinCnc Software. It is broken down into several generalized sections, as shown above. The following graphics display their options and uses under each section. With a Standard Machine many of these opinions are not available, and are not necessary to operate the machine. Some of these opinions are also available as buttons on the WinCNC program window.

File Menu

Opens a file

Opens the editor (if a, editor opens filename is in command line)

Simulates a file running to check for errors and run time.

View a file in the viewer window.

Creates a home file at the current machine position.

For importing DXF and HPGL files.*

Exits WinCnc

*Note:The DXF and HPGL import feature of WinCNC should only be used the for simple design. WinCNC does not compensate for tool diameter when converting these files. For design, where intellegent tool pathing is important, use a compatible CAD/CAM program.

Configuration Menu

Starts the WinCnc.ini File Editor

Calibration Settings for D/A

40

Settings Menu

File Configuration | Settings | View Options Units Transit Jog Help

Positions	← ───	View Positions
Home Positions	← ───	View Home Positions
Tool Positions	← ───	View Tool Positions
Local Coordinates	← ───	

View Menu

File Configuration Settings | View | Options Units Transit Jog Help

Resolution	← ───	View current Resolution settings for each axis.
Acceleration	← ───	View current Acceleration settings for each axis
Limits	← ───	View current Limit settings for each axis.
Inputs	← ───	View current input status
Toolbars ▶	✔ Main	
	Manual Digitize	← ─── Display Toolbars
Refresh View F5	← ───	Refreshes the screen.
Clear Screen Ctrl+C	← ───	Clears the message display window

Options Menu

File Configuration Settings View | Options | Units Transit Jog Help

✔ Soft Limits Ctrl+L	← ───	Enable/Disable the Soft Limits Option.
Keyboard Ctrl+K	← ───	Enable/Disable the Keyboard Transit/Jog Control.
Buttons	← ───	
Auto Repeat	← ───	Used to Customize User Buttons.
Auto Run	← ───	Auto populate the command line with last Command.
Auto Preview	← ───	Enables/Disables the Auto run feature.

41

Units Menu

File Configuration Settings View Options |Units| Transit Jog Help

Set units of measurement to Inches. ———————→ ✔ Inches

Set units of measurement to Centimeters. ————→ Centimeters

Set units of measurement to Millimeter ————————→ Millimeters
Millimeters.

Transit Menu

File Configuration Settings View Options Units |Transit| Jog Help

Set Transit speed to Slow ———————→ Slow   F2

Set Transit speed to Medium ——————→ Medium F3

Set Transit speed to Fast ———————→ ✔ Fast   F4

Jog Menu

File Configuration Settings View Options Units Transit |Jog| Help

Set Jog Increment to 0.001 ——————→ 0.001   F6

Set Jog Increment to 0.01 ———————→ 0.01   F7

Set Jog Increment to 0.1 ————————→ 0.1   F8

Set Jog Increment to 1 ————————→ 1   F9

Set Jog Increment to Custom ——————→ Custom F10

Help Menu

File Configuration Settings View Options Units Transit Jog |Help|

Activates the in Program help menu. ———→ Help Topics   F1

Opens the Updating Utility. ——————→ Update Program

Opens the About WinCnc window. ———→ About

About WinCnc

The About Box in the Controller software contains important information about the specific software package. The About Box displays the security key serial number, the software version number, the user level, the maximum number of axis, the table size limitation, and which additional features the user has enabled. If you do not have a feature enable that you would like or need, you can contact Eagle Eye, Inc for an upgrade.

About MicroSystems WinCNC Controller ☒

Key Information:
Company: MicroSystems WinCNC Controller

Serial Number: 9999

Software Version: 0.1.04

User Level: 4

Key Version: 5.00

Max. Axis: 4

Max. Table: No Limits

Days Remaining: Unlimited

Maximum Velocity: Unlimited

Feature Enables:
☑ Six Axis Card ☑ Com. DLL
☑ Pump Control ☑ Tangential Knife
☑ Tool Changer ☑ Rotating Saw
☑ Serial Keypad ☑ D/A Output
☑ Digitizer Probe ☑ Vector Laser
☑ Velocity Override ☑ View Window

Copyright (C) 1999-2004   [OK]

Viewer Window

The Viewer Window provides visual information about the status of the WinCnc software and equipment. The Viewer Window also allows job previewing. will also display a G-code file line by line as it is run by WinCnc. G0 moves and G1 moves are distinguished by the color of the line in the Viewer Window. G0 moves are shown as a black dashed line. G1 moves are displayed as a solid blue line. You can also re-center, and zoom in or out within the viewer.
Commands used for The Viewer Window:

 To Preview a File:

With the file name in the command line select the **"View"** button under the **"File"** menu, press Ctrl+V, or press the **"View"** button on the toolbar. The file will be displayed line by line automatically when you run the file.

Viewer Window Adjusting controls:

Zoom in Incrementally – click left mouse button.

Zoom out Incrementally – click right mouse button.

Selected Zoom – hold down the left mouse button and drag box around area to zoom in on.

Pan or re-center object – hold control and click either mouse button.

Reset Image – hold shift and click either mouse button.

Viewer Window Display

Job Preview

Local Zero
(Starting Point)

Machine Zero

WinCNC

Tool Location
(Red Dot)
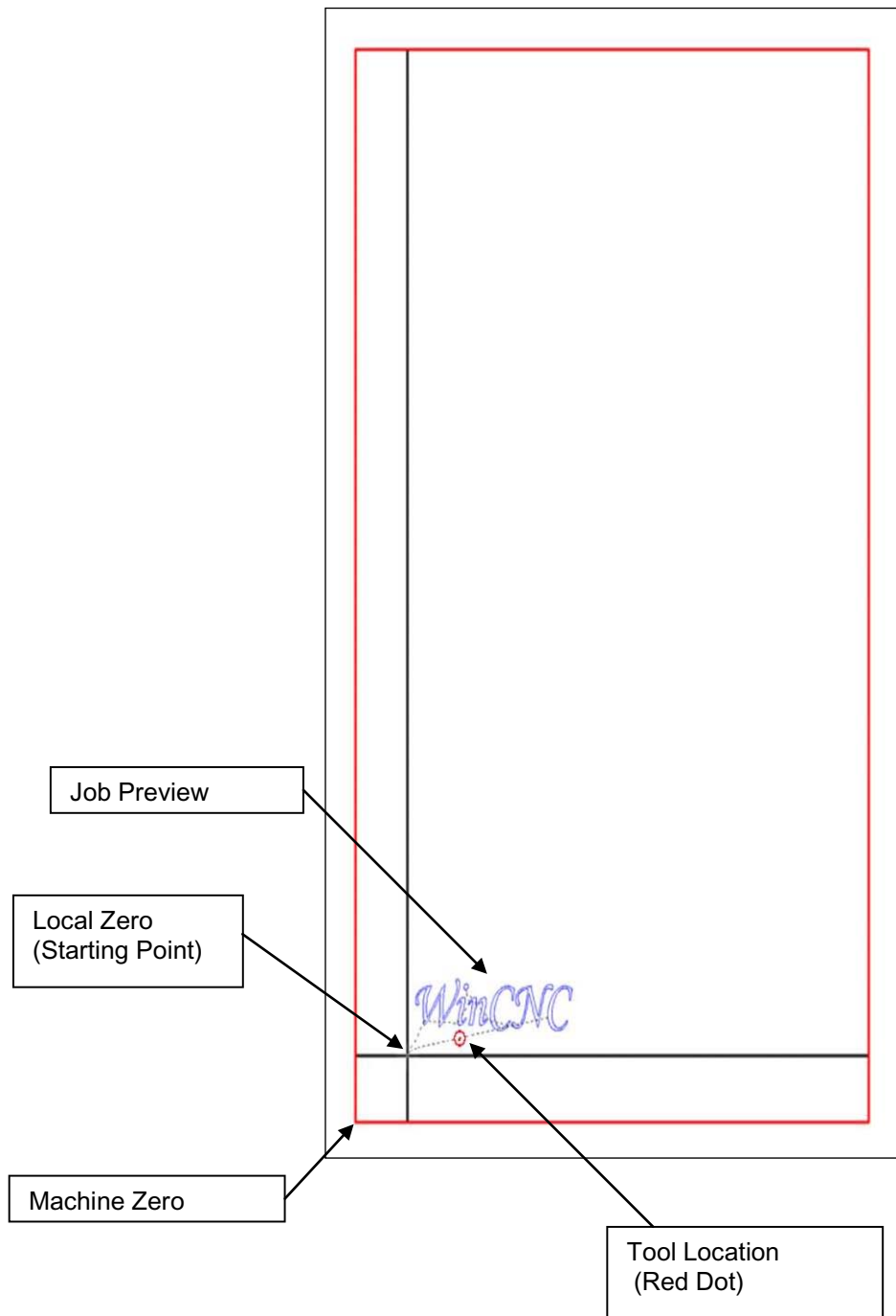
Homing the System

Machine Zero: An absolute position located close the front left corner of the table. The system finds its "Home" position or starting point for all Axis by moving to its X-Low, Y-Low, and Z-High limit switches. This point is stored in memory and all other points are created from referencing that position.

Home	Use the **Home** button or type **G28** then **Enter** at the "Command Line", for moving to the Machine Zero for all Axis. Use any time the system started to synchronize WinCNC with each Axis.

Local Zeros: Axis starting positions which may be set any were within the cutting area of the CNC System. This is normally set to the front, left corner of the job material.

ZeroXY	Use the **ZeroXY** button or use the **G92**, **G92.1** or **G92.2** commands to set Local Zeros. (See the G-Code Command list for explanations)

GoHome	Use the **Go Home** to return the cutting head to its' Local Home position for the X and Y-Axis. The Z or W-Axis will have to be reset at this point with the Touchpad.

Local Zero / Home

Job Material

Machine Zero / Home
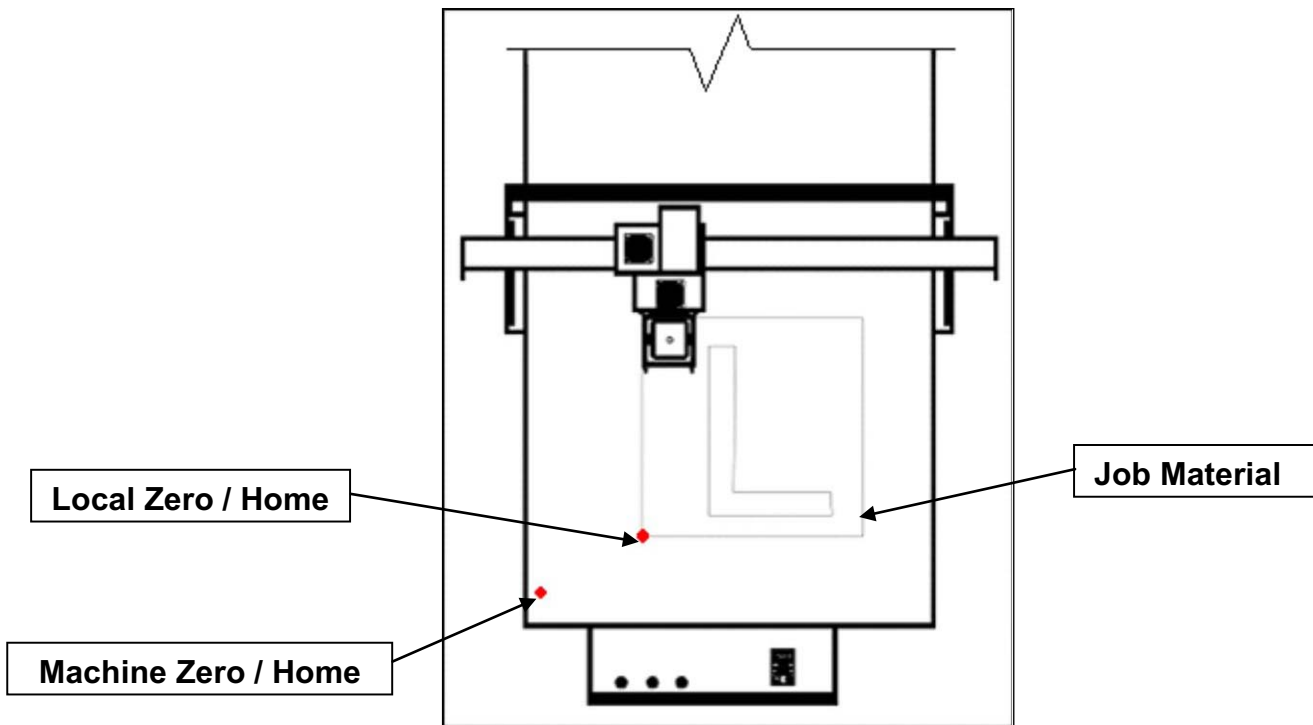
Using the Touchpad

The Touchpad is used to find the top surface height of the job material, and set that point as local zero for the Z-Axis. This position is different on any cutting tool and job material. Setting the material height between jobs saves time, and saves material.

Completely stop the cutting head rotation before using the Touchpad.

Before running the Touchtop or Zero Table files, place the Touchpad on top of the desired material surface. The Locate the cutting head tool over the Touchpad. The Z -Axis will lower slowly until it contacts the Touchpad, then raises up .75 inches off the material's surface when it is complete.
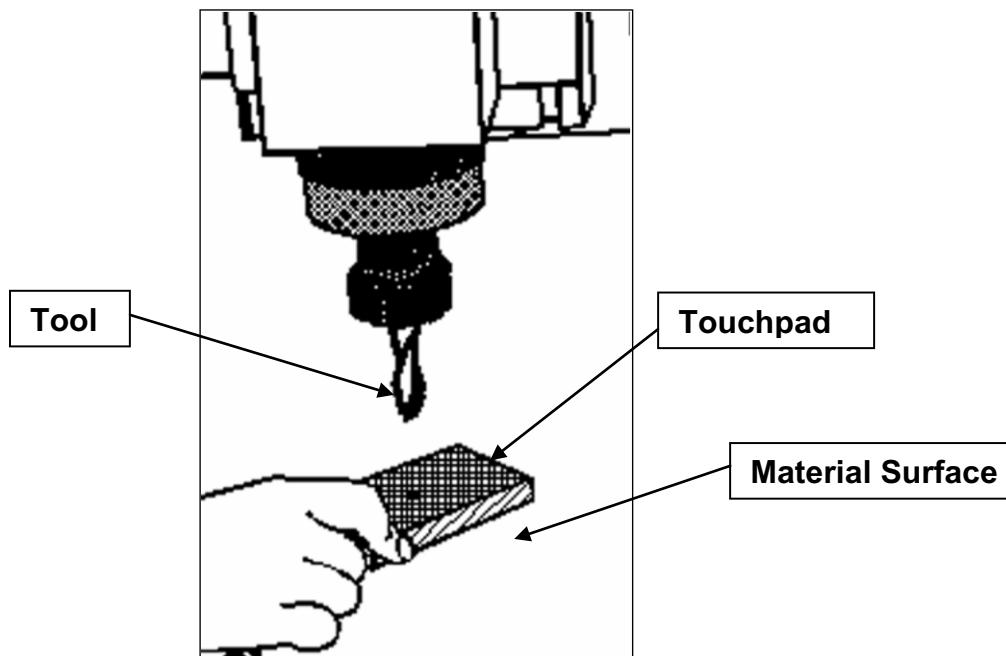
Setting the material surface height manually

1. Using the **Page Down** key, carefully bring the tool tip down slowly to the surface of the job material. Use the small incremental jog commands to fine-tune the position. Be careful do not gouge the job material.

2. At the command line, type **G92Z**, press the **Enter** key. Local Zero is set for the Z-axis.

3. Use the **Page Up** key to Raise the Z-Axis cutting tool off of the job material.

G90 and G91 Modes

There are two different ways that WinCNC controller finds position once a start point is established.

G90 Mode

G90 mode is called absolute mode. In G90 mode values specify positions. If the X position is 20 and G0 X8 is specified the machine would move –12 inches in X to X8.

In G90 mode the situation is different. The initial of the machine position will not affect where the part runs since the first move will go to the absolute position specified on the table. Instead of setting the machine position properly before running a G90 program. This is the mode we recommend for the Machine CNC router. Most G90 programs are written relative to a starting position of X0Y0 with Z0W0 being the top of the job material. All that is then necessary to run the program is to insure that the current **Local Zero** is set to match the program before running.

Alternately **Local Zeros** may be set by shifting **Machine Zero** the desired amount using G92.1 Using G92.1 it is not necessary to move to the job material first.

G91 Mode

G91 mode is called relative or incremental mode. In G91 mode values specify distances. For instance if the X position is currently 20 and G0 X8 is specified the machine would move +8 inches in X to X28.

A G-Code program written in G91 mode may be run from any position. Since the moves are relative, the starting position does not matter, the program will run properly from any starting position. However to get the proper results the machine must be set to the proper position in relation to the job material before the piece is run.

The axis coordinate status does not have any effect on how the program runs. It does however affect the coordinate display while the program runs.

Example

G90	Commands in G90 mode
G92.1**X**20**Y**20**Z**1	Move Local Zero 20" in XY and 1" above the table

The coordinate display shows the current position relative to the work piece.

When writing a G Code program the user must always remember to set a starting point using the ZeroXY and set the Z or W zero at the top of the job material using the Touchtop of ZeroZ macro buttons.

When running the program the machine is positioned to this starting point and G92, the g-code used then the ZeroXY macro files, is used to set position to the start position for the program. Alternately G92.1 may be used to shift **Machine Zero.**

Keyboard Shortcut Commands

File Menu Shortcuts	
Open	Enter(only when command line is blank), (Ctrl+0)
Edit	(Ctrl+E)
Simulate	(Ctrl+S)
View	(Ctrl+V)
Create Home File	(Ctrl+H).
View Menu Shortcuts	
Refresh View	(F5)
Clear Messages	(Ctrl+C)
Options Menu Shortcuts	
Keyboard	(Ctrl+K).
Soft Limit	(Ctrl+L)
Transit Menu Shortcuts	
Slow	(F2).
Medium	(F3)
Fast	(F4)
Jog Menu Shortcuts	
.001	(F6)
.01	(F7)
.1	(F8)
1	(F9)
Custom	(F10)
Help Menu Shortcuts	
Help	(F1)
Tool Bar Shortcuts	
ESC	Aborts the current file or command.
SPACE	Pauses or continues a paused file or command. Enter will also continue a paused file or command.
TAB	Opens the command history box.
Ctrl+R	Opens the restart file box.
Feed Rate Override Shortcuts	
Insert	Increases override rate
Delete	Decreases override rate
Ctrl+either	Resets feed rate to 100%. No override settings

Notes

Notes

Appendix I

Command Reference:

Here is a list all commands supported by WinCNC Controller.
Parameters in [Brackets] are optional.

Values:

XYZWIJ	Axis Specification [X#] [Y#] [Z#] etc.	Axis values are specified with a decimal point. A value with no decimal is read as an integer value. No value is equivalent to specifying 0.

Example: XYZ is equivalent to **X0Y0Z0.**

F	Feed Velocity F#	Velocity is stored separately for Linear XY, Linear ZW. Arc's Rapid and Feed Velocity is stored separately for Linear Moves. Independent velocities are stored based on the XYZW specified in the line containing the F# command. Velocity is specified in inches per minute.
P	Stored Positions P#	Move to the selected stored position.
H	Stored Home Positions H#	Move to the selected stored Home position.
S	Spindle Speed S#	Sets the spindle speed to the given value.

Example: F60 sets XY Feed Velocity to 60. G1 X F60 sets XY Feed Velocity to 60. G2 X2 F60 sets arc feed Velocity to 60. Feed Velocity changes affect all axis in a Vgroup.

[ ]	Comment	Used to add comments to programs. A closing bracket is optional.

Example: [This is a comment]

{ }	Braces	Used to substitute axis or parameter values in a line of G-Code. Internally defined variables as well as constant numbers can be used within the braces. Following is a list of internally defined variables and what they represent.

G Codes:

G0	**Rapid Move** G0 [X#] [Y#] [Z#] [W#]	Moves to the position specified at Rapid velocity. G0 is modal. After a G0 is executed lines with no g-code command are executed as a G0.
G0.1	**Rapid Move with Vertical Lift** G0.1 [X#] [Y#]	First lifts all vertical heads then moves the position specified at Rapid velocity then drops the vertical heads back to their previous positions.

Example: X1Y1 is equivalent to G0 X1Y1 if mode is G0

G1	**Feed Move** G1 [X#] [Y#] [Z#] [W#]	Moves to the position specified at Feed velocity. G1 is modal. After a G1 is executed lines with no g-code command are executed as a G1.

Example: X1Y1 is equivalent to G1 X1Y1 if mode is G1

G2	**Clockwise Arc** G2 [X#] [Y#] [I#] [J#] [Z#] [W#]	Moves to the position specified at Feed velocity. I is the X distance to the center point. J is the Y distance to the center point. If no XY move is specified a full circle is cut. If no I or J is specified previous I J values are kept. Helical Interpolation is supported in Level 2+ Only.
G3	**Counter Clockwise Arc** G3 [X#] [Y#] [I#] [J#] [Z#] [W#]	Moves to the position specified at Feed velocity. I is the X distance to the center point. J is the Y distance to the center point. If no XY move is specified a full circle is cut. If no I or J is specified previous I J values are kept. Helical Interpolation is supported in Level 2+ Only.
G4	**Dwell** G4 X[#]	Stops movement for the time specified by the X value in seconds. There is no limit to delay time. If no time is specified then the machine will be stopped until the operator pushes ENTER. Place a comment after the dwell to prompt the operator.

Example: G4 [Ready To Start Section 2] Never use Dwell to stop the machine while changing parts!Instead program a single part and use the TAB key at the Program prompt.This will repeat the last part cut.

G9	**Smoothing** G9 S[#]	Enables vector smoothing. S is smoothing factor, 1 being no smoothing. Highest smoothing factor is machine dependent, stepper machines typically use S5, and servo machines typically use S1. Too high a factor can result in missed steps or motor stalls. Smoothing can also be set in the WINCNC.INI file using G09 = S#
G20	**Set Unit to Inches**	Converts unit of measure to inches.
G21	**Set Unit to Centimeters**	Converts unit of measure to centimeters.
G22	**Set Unit to Millimeters**	Converts unit of measure to millimeters.
G23	**Return to last used unit**	Returns to the last used unit of measure.

G28	**Return to machine zero** G28 [X] [Y] [Z] [W]	Moves specified Axis's to Lo Limit for XY – Hi Limit for ZW. Moves specified Axis's to WINCNC.INI specs from limits. Sets all Axis positions to 0. All Axis' are moved if none are specified.
G31	**Stop file parsing**	Used with the M28 command to measure machine position after touching a limit switch.

Example: L91 G1 Z-10 F20 M28 G31 M37 Z1

G43	**Tool Length Offset On** G43 [Z] [W] [U] [V]	Sets tool length offsets to the values set by the last M37 command. G43 alone turns all offsets on.
G49	**Tool Length Offset Off** G49 [Z] [W] [U] [V]	Cancels tool length offsets. G49 alone turns all offsets off.
G50	**Scaling Mode Off**	Cancels the scaling mode.
G51	**Scaling Mode On** G51 [X#] [Y#] [Z#] [I#] [J#] [K#] [C#]	Enables the scaling factor. Scale each axis type based on the values specified by X# Y# Z#. The object will be centered based on the I# (X center) J# (Y) K#(Z). You can center all axis using the C# line.
G53	**Rapid Move** G53 [X#] [Y#] [Z#] [W#]	Moves to the position specified at rapid velocity, ignoring tool measures and local coordinates. G53 alone will lift all vertical or type 3 axis.
G54	**G54 Workspace** Rotational: G54 [X#] [Y#]	Select the Z head. If G54 is commanded from G55, G56, or G57 mode, the Z head will be moved to the current XY position.
G55	**G55 Workspace** Rotational: G55 [X#] [Y#]	Select the W head. If G55 is commanded from G54, G56, or G57 mode, the W head will be moved to the current XY position.
G56	**G56 Workspace** Rotational: G56 [X#] [Y#]	Select the U head. If G56 is commanded from G54, G55, or G57 mode, the U head will be moved to the current XY position.
G57	**G57 Workspace** Rotational: G57 [X#] [Y#]	Select the V head. If G57 is commanded from G54, G55, or G56 mode, the V head will be moved to the current XY position.
G73	**Chip Break Cycle** G73 [X#] [Y#] [Z#] [R#] [Q#] F#	Moves to XY specified at Rapid velocity. On the first peck, moves to R retract height at Rapid velocity, moves to Q peck depth at F Feed velocity, lifts 0.05" at Rapid velocity to break the chip, moves to next peck depth at Feed velocity. The cycle repeats until Z depth is reached. Then moves back to retract height R at Rapid velocity. Without this cycle, some materials can produce a long "stringer" which can become a safety issue to the operator.
G80	**End Drill Cycle**	Used to disable the G81 Drill Cycle mode. Note: Switching to G0 or G1 will also break out of the drill cycle mode.
G81	**Drill Cycle** **G81 [X#] [Y#] [Z#] [R#] [F#]**	Moves to XY specified at Rapid velocity. Moves to R (clearing height) at Rapid velocity. Moves to Z specified at Feed velocity. Moves back to R at Rapid velocity. F allows you to set the feed rate.

G82	**Dwell Cycle** G82 [X#] [Y#] [Z#] [R#] [P#] F#	Moves to XY specified at Rapid velocity. Moves to R (clearing height) specified at Rapid velocity. Moves to Z specified at Feed velocity. If P is specified, bit pauses that amount of time at the bottom of the hole, then moves back to R at Rapid velocity. P is measured in milliseconds (thousandths of one second). F allows you to set the feed rate.
G83	**Peck Drill Cycle** G83 [X#] [Y#] [Z#] [R#] [Q#] F#	Moves to XY specified at Rapid velocity. On the first peck, moves to R retract height at Rapid velocity, moves to peck depth Q at Feed velocity, then moves back to retract height at Rapid velocity. On each subsequent peck, moves at Rapid velocity to within 0.05" of previous peck depth, then moves at Feed velocity to next peck depth. Cycle repeats until Z depth is reached. Then moves back at Rapid velocity to retract height R. The F allows user to set the feed rate.
G90	**Absolute Mode**	Can be specified with other G Codes on any line. XYZW values from the current line forward are read as absolute coordinates. IJ values are always relative to the current XY position, not absolute positions, regardless of G90/G91 mode.
G91	**Relative Mode**	Can be specified with other G Codes on any line. XYZW values from the current line forward are read as relative movements from the current position.
G92	**Set Local Coordinates** G92 [X#] [Y#] [Z#] [W#]	Used to specify a new coordinate system for running absolute mode programs. Use G92 alone to restore the Machine Coordinates. G92 X0 Y0 Z0 W0 sets the current position to zero. G92 then restores the Machine Coordinates values.
G92.1	**Shift MZ Coordinates** G92.1 [X#] [Y#] [Z#] [W#]	Similar to G92 except that the coordinate system produced shifts Machine Zero by the amount specified. This is useful since a given local coordinate system can be set without positioning the head to a certain position first. G92 X10 Y10 sets the absolute position X10 Y10 to X0 Y0 in local coordinates. G92.1 may also be used to restore a single axis to absolute coordinates. G92.1 X0 leaves YZW local coordinates but sets X back to absolute coordinates.
G92.2	**Shift LZ Coordinates** G92.2 [X#] [Y#] [Z#] [W#]	Similar to G92.1 except that the coordinate system shift is added to the current local coordinates instead of replacing them. This is useful for 'jogging' a local position. If the Z head has been set for running a G90 mode file but then needs to set to cut .010" deeper G92.2Z-.01 will accomplish this with a single command. Can be useful as a macro.

L Codes:

L0, L2	Ignored	Ignored commands are not executed but not flagged as errors.
L1	**Spindle Speed Control using the spin.mac macro**	Calls the spin.mac macro for spindle speed control. Spin.mac must be a file in the WinCnc directory.
L3	**Set Home Position** L3 [X] [Y] [Z] [W]	Stores current position of each axis specified. Values specified are ignored.
L4	**Return To Home** L4 [X] [Y] [Z] [W]	Moves each axis specified to the last **L3** position stored. Values specified are ignored.
L10	**Cut Array** L10 [R#] [C#] [X#] [Y#]	Sets up array cutting. Repeats all code following until the end of the file or another L10 is reached. Code following L10 is run until the end of file or another L10. Program pointer is moved back to initial L10. The head is moved to the next column or row specified by XY at rapid velocity. Cycle repeats until all array points have been cut. L10 specified without RCXY values can be used to end an array cut. Lines that follow will not be repeated.
L11	**Set Axis Mapping** L11 [X] [Y] [Z] [W]	Allows re-mapping of axis inputs. Specify the axis's that should receive the input from XYZW in that order. Specify L11 alone to return to normal parsing.

    Example: L11 XYWZ swaps the W and Z axis, L11 XYZW moves Z and W together.

L12	**Set Axis Mapping** L12 [X] [Y] [Z] [W]	Allows re-mapping of axis inputs. Specify the axis's that should receive the input from ZW in that order. This command works like L11 except that the XY inputs are skipped. Specify L12 alone to return to normal parsing.

    Example: L12 WZ swaps the W and Z axis, L12 ZZ moves Z and W together.

L13	**Set Axis Mapping** L13 [X] [Y] [Z] [W]	Allows re-mapping of axis on a pulse basis. This command works like L11. Specify L13 alone to return to normal mapping.
L20	**Enable Soft Limits** L20 [X1, X2]	Enable Soft Limit and Boundary Checking: L20 X1 enables Boundaries only. L20 X2 enables Soft Limits only. L20 alone enables both.
L21	**Disable Soft Limits**	Disable Soft Limit and Boundary Checking Soft Limits and Boundaries are used to define a cutting area which is checked during parsing of a file or command. This effectively keeps the machine from moving out of a defined area. L21 disables Soft Limit and Boundary checking.
L22	**Set Lo Boundaries** L22 [X#] [Y#] [Z#] [W#]	Set Lo Boundaries to values specified. If no values are given then all Lo Boundaries are set to Wincnc.INI defaults.
L23	**Set Hi Boundaries** L23 [X#] [Y#] [Z#] [W#]	Set Hi Boundaries to values specified. If no values are given then all Hi Boundaries are set to Wincnc.INI defaults.

L28	**Set Position** L28 [X#] [Y#] [Z#] [W#]	Sets Machine Coordinates. Machine Coordinates specify the fixed machine zero point for your machine. L28 is normally used only to write a custom homing program. L28 does not move the machine, but sets the current position to the values specified. Use G92 to set a local coordinate system for running absolute mode programs from any table position.
L30	**Knife Mode On**	Enables support for the Knife Mode.
L31	**Knife Mode Off**	Disables support for the Knife Mode.
L32	**Saw Mode On**	Enables support for the Saw Mode.
L33	**Saw Mode Off**	Disables support for the Saw Mode.
L34	**AutoAuxOut Enable** L34 [D#]	Enables an auxiliary output that will turn on during G1/G2/G3 moves, and will turn off for all other moves. L34D# sets the dwell time in seconds used when the channel is set to on and L35D# sets the dwell time in seconds when the channel is set to off. When the 'D' parameter is used with the L34 command the "AutoAuxOut" mode does not change. L34 parameters are set in the WINCNC.INI file by the "Autoaux =" line.
L35	**AutoAuxOut Disable** L35[D#]	Disables the auxiliary output that is turned on by the L34 command. L35D# sets the dwell time in seconds when the channel is set to off. When the 'D' parameter is used with the L35 command the "AutoAuxOut" mode does not change.
L40	**Feed Rate Override** L40 [P#]	Allows user to set the feed rate override by command. P must be within the bounds specified in the ini file. If not, the command is ignored. The L40 command without a P# resets the feed rate to the default.
L50	**Generate a Set Frequency** L50 L50 [F#]	Generates a set frequency using the ISA 7120 card where F# is the number of desired pulses per second. L50 alone will stop the output signal.
L51	**Set D/A** L51 [V#] [P#] [S#]	Sets the output voltage for DA. The V parameter is used to set the DA by voltage, the P to set the DA by a Percentage, and the S is used to set the DA by spindle speed. ONLY 1 PARAMETER CAN BE USED.
L53	**Set Vertical Head Height**	Raises all vertical heads as high as possible while keeping the bits at equal height.
L55	**Pump Velocity Mode On**	Enables velocity matched pump control.
L56	**Pump Purge Mode On**	Sets pump to purge mode.
L57	**Pump Off**	Stops the pump.
L58	**Pump Off with Auto Reverse**	Stops the pump, using auto reverse.
L59	**Set Up Pump Max** L59 [M#]	Sets the pump max to the specified M parameter.
L61	**Set Laser Controller Power** L61 [W#]	Sets the power of the laser controller using the RS232 connection. The W parameter can be any value from 0 to 100. Requires lasercont = setting.

L70	**Re-Enter Last Command**	When L70 is used in a file the last used command will automatically be re-entered on the command line after file completion.
L82	**Measures the Surface** L82 L82 [Z#]	L82Z# measures the surface with the probe and sets the measured point to the Z value given. L82 alone will set the measured point to Z0.
L83	**Measures Z Depth** L83 X[#] Y[#]	L83 command will do a one-shot Z probe at the specified location, and record the XYZ position values in a comma delimited text file named scan.csv. Where X# and Y# specify the (X,Y) coordinates to move to before doing the depth probe.
L90	**One Time Absolute**	L90 is used in a command to specify G90 coordinates on one line without changing program mode to G90.
L91	**One Time Relative**	L91 is used in a command to specify G91 coordinates on one line without changing program to G91.
L92	**Cancel G92 Restore** L92	The current G92 Local Coordinate system is saved every time a program is started or a sub-program is called using M98. When the program is finished the original G92 system is restored. This is not desirable if the program is being used to preset a system configuration for other programs. L92 causes the normal G92 Coordinate restore to be skipped after the program containing the L92 is finished.
110	**Tool Change** L110 [T#]	Sets current tool number for machines with an automatic tool changer. The tool number is available as {TP1} in subsequent commands.
L111	**Test Tool Number** L111 [T#]	Run next line if tool number is equal to T#
L111.1	**Test Tool Number with Dual Tool Changers** L111.1 [T#]	Run next line if both tool numbers are equal to T#
L112	**Test Tool Number** L112 [T#]	Skip next line if tool number is equal to T#
L113	**Spindle Speed Conditions** L113[L#][H#][V#]	Skip the next line if the V# is not greater than the L# and less than the H#.
L120	**Tool Change** L120 [T#]	Sets current tool number for machines with an automatic tool changer. The tool number is available as {TP2} in subsequent commands.
L201	**Set 201 Signal Command** L201 [Command]	Sets the command used with the 201 signal type.
L202	**Set 202 Signal Command** L202 [Command]	Sets the command used with the 202 signal type.

210	**Selects Alternate Low Limits** L210 [X] [Y] [Z] [W]	Selects axis to use alternate low limits
L211	**Selects Alternate High Limits** L211 [X] [Y] [Z] [W]	Selects axis to use alternate high limits
L212	**Selects Primary Limits** L212	Selects the Primary Limits for all axis.
L213	**Dual Axis, Select Main** L213 [X] [Y] [Z] [W]	Selects the main motor on a dual motor axis.
L214	**Dual Axis, Select Secondary** L214 [X] [Y] [Z] [W]	Selects the secondary motor on a dual motor axis.
L215	**Dual Axis, Select Both Axis** L215 [X] [Y] [Z] [W]	Selects both motors on a dual motor axis.
L249	**Laser Power Setting** L249 [P#]	Using the L249 command allows you specify the laser power. Example: L249P50 (sets laser power to 50)
L250	**Vector Laser Off**	Disables vector laser cutting.
L251	**Vector Laser On**	Enables vector laser cutting.
L251.1	**Laser Dwell** L251.1 [D#]	Turns laser on for specified period of time.
L300	**Lock Axis** L300 [XYZW]	Locks specified axis motion. Any motion commanded to a locked axis will be ignored.
L301	**Unlock Axis** L301 [XYZW]	Unlocks specified axis motion.
L400	**Create Home File** L400	Creates a home file at the machines current position.
L401	**Recall Positions** L401	Moves all axis to the positions where they were at previous to the last file ran.

L500	**Store Absolute Positions** L500 [X, Y, Z..] [P H T] L# C# N"Pos Name"	[X, Y, Z..] – axes to set positions for if none specified then all axis positions are stored. [P H T] – Type of Position (P)osition, (H)ome Position, (T)ool Position. Can only use one. L# - 0 no vertical head life, 1 include head lift. C# - Clearance for vertical heads if L1. N"Name" – Name for specified position. Must be less than 16 characters long.
L501	**Store Local Positions** L501 [X, Y, Z..] [P H T] L# C# N"Pos Name"	[X, Y, Z..] – axes to set positions for if none specified then all axis positions are stored. [P H T] – Type of Position (P)osition, (H)ome Position, (T)ool Position. Can only use one. L# - 0 no vertical head life, 1 include head lift. Default is L1. C# - Clearance for vertical heads if L1. N"Name" – Name for specified position. Must be less than 16 characters long.
L800	**Disable Probe** L800	Disables the probe while transiting.
L801	**Enable Probe** L801	Enables the probe while transiting.
L802	**Scan Given Rectangle** Uses an up, down, side to side probe typeL802 [X# Y# I#]	Scans a given rectangle XY with an increment of I in an up, down, and side to side style scan.
L803	**Scan Given Rectangle** Uses an up – down probe type L803 [X# Y# I#]	Scan a given rectangle XY with an increment of I in an up down style scan.

M Codes:

M11	**Auxiliary Output On** M11 [C#]	Controls auxiliary outputs. Channel specified by # is turned on. Port and bit addresses are setup in WINCNC.INI. **Note: For use with a Drill Bank** A d# can be added to the M11 command for use with a drillbank. The d# is the binary number of channels up to the n# specified in the drillbank line to turn on. **Example:**M11D7 would turn on the first 3 channels of a ten channel drill bank.
M12	**Auxiliary Output Off** M12 [C#]	Controls auxiliary outputs. Channel specified by # is turned off. Port and bit addresses are setup in WINCNC.INI.
M13	**Auxiliary Output On** M13 [C#]	Same as M11, except the output is turned off without pausing motion.
M14	**Auxiliary Output Off** M14 [C#]	Same as M12, except the output is turned off without pausing motion.
M17	**Pause Until Input On** M17 [C#]	Wait for channel specified by # to come on. Port and bit addresses are setup in WINCNC.INI.
M18	**Pause Until Input Off** M18 [C#]	Wait for channel specified by # to go off. Port and bit addresses are setup in WINCNC.INI.
M17.1	**Run Next If Input On** M17.1 [C#]	Run next command only if channel specified by # is on. Port and bit addresses are setup in WINCNC.INI.
M17.2	**Run Next if Aux Out is On** M17.2 [C#]	Run the next command only if channel specified by the # is on. Port and bit addresses are setup in WINCNC.INI.
M18.1	**Run Next If Input Off** M18.1 [C#]	Run next command only if channel specified by # is off. Port and bit addresses are setup in WINCNC.INI.
M28	**Disable Limit Abort**	M28 is used to write your own home program instead of using G28 to home the machine. CNC Controller normally aborts a program when a limit switch is encountered. Adding M28 to a G0 or G1 command disables the program abort and instead stops the limited axis while allowing other axis's to continue. Limit Cushions are disabled while using M28 so any move done with M28 should be done slowly to avoid machine damage. M28 must be specified on each line separately.
M29	**Disable Limit Abort**	M29 is used to write your own home program instead of using G28 to home the machine. Adding M29 to a G0 or G1 command disables the program abort and instead stops the current G Code line but continues on the following line. M29 must be specified on each line separately.
M37	**Measure Tool** **M37 [Z#] [W#] [U#] [V#]**	M37 is used to set Tool Length Offsets. The end of the tool is first set to a known position. M37 is then specified to set the tool length offset.

M37.1	**Measure Tool for Calibrating G37** M37.1 [Z#] [W#] [U#] [V#]	M37.1 is used to store a tool measure in the TM1 variable. This command is used primarily with calibrating G37 for automatic tool measuring.

Example: Jog the Z head to the spoilboard. Enter **M37Z0** to the Z work coordinate to 0.
Once Tool Length Offsets are set, Boundaries may be used to insure that the table is not routed by accident. Normally zero values are set to the top of the spoilboard. Use G92 to set material thickness if programs reference zero from the top of material. This eliminates re-measuring tool for different thickness materials. **G43** and **G49** are used to enable and disable offsets.

M98	**Subprogram Call** M98 <filename>	Run a subprogram. A subprogram may be any G Code file. Control is passed back to the calling program when finished. To run a G90 mode subprogram from any position on the table use G92X#Y#Z#W# to set the local coordinates at the beginning of the file.
M98.1	**Subprogram Call** M98.1 <filename>	Same as M98, except line numbers in the subprogram are not counted. This is most useful for macros that call subprograms.
M99	**End Subprogram**	Ends a subprogram.

Machine Codes:

M3	**Turns Spindle On**	This command will turn on the spindle, if the Machine is equipped with the automatic spindle feature. Use the 'S' command, above, to set speed of spindle in RPM.
M5	**Turns Spindle Off**	This command will turn off the spindle, if the Machine is equipped with the automatic spindle feature.
T[tool number]	**Change Tool** Example: T1- Changes tool One T2- Changes tool Two, etc.	Enter T and the tool number to change tools on a Machine Equipped with ATC.
TC	**Loads Tool**	Loads current tool if the active tool is unloaded. Only on Eagle Eyes equipped with ATC.
TU	**Unloads Tool**	Unloads current tool into the tool holders. Only on Machine equipped with ATC.

Troubleshooting

1. The machine is cutting circles or arcs that are oblong or not true circles.

    A1. Check for looseness in the axes by pushing on the different axes while the control box is on, but make sure that the cutting head is disengaged. Once the looseness is found, adjust the axis bearing so they are tight against the drive rails, but not so tight the axis cannot move freely. Call Eagle Eye tech support if you need assistance.

2. When I start the software I try to move or home the machine and I get an "Abort Enable I" massage.

    A1. Make sure that the machine is plugged in with the controller engaged and the Emergency Stop disengaged.

    A2. Make sure the 37-pin connector cable is properly connected to the machine into the computer's blue 37-pin connector port.

    A3. Call Eagle Eye tech support, professional assistance is required.

3. I hear noise from the motors but that axis is not moving. Or moves to a certain point and stops. But the other axis will move fine.

    A1. Check gears and tracks to make sure nothing is obstructing the movement of the machine. Or the bearings need adjustment.

    A2. Other unknown cause, possible defective component in control system, contacting Eagle Eye is recommended.

4. In WinCNC I get the message "Abort limit switch ……" appears in middle of cut file with the hard limits have clearly not been touched.

    A1. Check Emergency Stop button and make sure it is not presses.

    A2. You machine is getting electrical noise that causes the limit switches to trigger. Call Eagle Eye tech support for assistance.

5. One axis's motor will move but it shutters, or moves a short distances and spins back a little then repeats this process again, or the axis moves to a curtain point then stops.

A1. Check and make sure that there is nothing blocking the movement of that axis. Check the gears and ball screw (if problem is on the Z axis) for any debris. If you see debris, remove it. If you cannot see any debris, turn off your control box and wait a few seconds, try moving the axis by pushing on that axis in the direction of the problem, in the case of the Z axis turn the ball screw with your finger. If the control box is off and the axis in question will not move, or comes to a curtain point and stops something is definitely blocking one of the gears of rails. Closer investigation is recommended to make sure nothing to in the gears or rails.

CONSULTING
SALES
SUPPORT

Eagle eye
CNC ROUTERS

13555 Back Valley Road
Sale Creek, TN 37373
423-228-2349 ph
423-228-2346 fax
www.eagleeyecnc.com
www.youtube.com/eagleeyecnc
sales@eagleeyecnc.com