LAGUNA

Swift Series:
Swift Vacuum (Multi-Tool) MT

For immediate service on any Laguna Tools products:

Call: +1 (800) 234 1976
or
Email: customerservice@lagunatools.com

LAGUNA **SuperMax**

LAGUNA AMERICAN HEADQUARTERS

Texas: 744 Refuge Way Suite 200, Grand Prairie, Texas 75050, U.S.A. Phone: +1-800-332-4094

Huntington Beach: 7291 Heil Ave Huntington Beach, CA 92647, U.S.A. Phone: +1-949-474-1200

South Carolina: 825 Bistline Dr. Ste 101, West Columbia, SC 29172, U.S.A. Phone: +1-800-234-1976

Minnesota: 5250 West 74th St, Edina, MN 55439, U.S.A Phone: +1-949-474-1200

lagunatools.com

supermaxtools.com

lagunacleanair.com

lagunalathe.com

LAGUNA EUROPE

Walker Rd, Bardon Hill, Coalville LE67 1TU, United Kingdom. Phone: +44-1530-516921

lagunatools.uk

DAKE CORPORATION

724 Robbins Road, Grand Haven, MI 49417, United States +1-800-937-3253

dakecorp.com

LAGUNA

LAGUNA

LAGUNA

Contents

Troubleshooting   100

Maintenance   102

Command Reference   103

Warranty Information   116

General Information

SAVE THESE INSTRUCTIONS
Refer to them often and use them to instruct others.

Please read and understand all warnings and operation instructions before using any tool or equipment. Always follow basic safety precautions to reduce the risk of personal injury. Improper operation, maintenance or modification of tools or equipment could result in serious injury and property damage. There are certain applications for which tools and equipment are designed. This product should *NOT* be modified and/or used for any application other than for which it was designed.

NOTICE! It is important for you to read and understand this manual. The information it contains is provided for your safety while assembling and operating this machine.

Safety Signs and Call-Outs:

⚠ **DANGER** An imminently hazardous situation which, if not avoided, will result in death or serious injury.

⚠ **CAUTION** A potentially hazardous situation which, if not avoided, could result in death or serious injury.

⚠ **WARNING!** A potentially hazardous situation which, if not avoided, may result in minor or moderate injury.

NOTICE A helpful tip from our technical staff. Sometimes displayed as *NOTICE!* instead.

	Disconnect from power before proceeding.		Wear ear protection.
	Be aware of possible laceration danger.		Wear Eye Protection.
	Be aware of possible crushing danger.		Wear a full face shield.
	Electrical Hazard.		Wear lung protection.
			Requires X People

Safety Rules

PLEASE READ AND UNDERSTAND ALL SAFETY WARNINGS AND OPERATING INSTRUCTIONS BEFORE USING THIS MACHINE

FAILURE to follow all instructions listed below, may result in electric shock, fire, and/or serious personal injury or property damage.Woodworking can be dangerous if safe and proper operating procedures are not followed. As with all machinery, there are certain hazards involved with the operation of the product. Using the machine with respect and caution will considerably lessen the possibility of personal injury. However, if normal safety precautions are overlooked or ignored, improper operation or assembly of this machine could result in personal injury to the operator. Safety equipment such as guards, push sticks, hold downs,feather boards, goggles, dust masks and hearing protection can reduce your potential for injury. But even the best guard won't make up for poor judgment, carelessness or inattention.

ALWAYS use common sense and exercise caution in the workshop. If a procedure feels dangerous, don't try it.Consult a professional to find an an alternative procedure which is safer and more efficient.

NOTICE! Your personal safety is YOUR responsibility.

⚠ **WARNING!** This machine was designed for certain applications only. We strongly recommend that this machine not be modified and/or used for any application other than that for which it was designed. If you have any questions relative to a particular application, DO NOT use the machine until you have first contacted the manufacturer to determine if it can or should be performed on the product.

⚠ **WARNING!** If you have any questions relative to its application DO NOT use the product until you have contacted the manufacturer and we have advised you. When using an electrical machine, basic precautions should always be followed, including the following:

Read and understand all warnings and operation instructions before using any tool or equipment. Always follow basic safety precautions to reduce the risk of personal injury. Improper operation, maintenance or modification of tools or equipment could result in serious injury and property damage. There are certain applications for which tools and equipment are designed. This product should NOT be modified and/or used for any application other than for which it was designed. It is important for you to read and understand this manual. The information it contains relates to protecting your safety and preventing problems.

Machine & Workshop Safety Instructions

READ AND UNDERSTAND ALL WARNINGS AND OPERATING INSTRUCTIONS BEFORE USING THIS EQUIPMENT. FAILURE TO
FOLLOW ALL INSTRUCTIONS LISTED BELOW, MAY RESULT IN ELECTRIC SHOCK, FIRE, AND/OR SERIOUS PERSONAL INJURY
OR PROPERTY DAMAGE.

MACHINERY CAN BE DANGEROUS IF SAFE AND PROPER OPERATING PROCEDURES ARE NOT FOLLOWED. AS WITH ALL
MACHINERY, THERE ARE CERTAIN HAZARDS INVOLVED WITH THE OPERATION OF THE PRODUCT. USING

THE MACHINE WITH
RESPECT AND CAUTION WILL CONSIDERABLY LESSEN THE POSSIBILITY OF PERSONAL INJURY. HOWEVER, IF NORMAL
SAFETY PRECAUTIONS ARE OVERLOOKED OR IGNORED, PERSONAL INJURY TO THE OPERATOR MAY RESULT. SAFETY
EQUIPMENT SUCH AS GUARDS, PUSH STICKS, HOLD-DOWNS, FEATHER BOARDS, GOGGLES, DUST MASKS AND HEARING
PROTECTION CAN REDUCE YOUR POTENTIAL FOR INJURY. BUT EVEN THE BEST GUARD WON'T MAKE UP FOR POOR
JUDGMENT, CARELESSNESS OR INATTENTION. ALWAYS USE COMMON SENSE AND EXERCISE CAUTION IN THE WORKSHOP.
IF A PROCEDURE FEELS DANGEROUS, DON'T TRY IT. FIGURE OUT AN ALTERNATIVE PROCEDURE THAT FEELS SAFER.
REMEMBER: YOUR PERSONAL SAFETY IS YOUR RESPONSIBILITY.

OWNER'S MANUAL. READ AND UNDERSTAND THIS OWNER'S MANUAL BEFORE USING MACHINE.

TRAINED OPERATORS ONLY. UNTRAINED OPERATORS HAVE A HIGHER RISK OF BEING HURT OR KILLED. ONLY ALLOW

TRAINED/SUPERVISED PEOPLE TO USE THIS MACHINE. WHEN MACHINE IS NOT BEING USED, DIS-CONNECT POWER,
REMOVE SWITCH KEYS, OR LOCK-OUT MACHINE TO PREVENT UNAUTHORIZED USE—ESPECIALLY AROUND CHILDREN. MAKE
YOUR WORKSHOP KID PROOF!

DANGEROUS ENVIRONMENTS. DO NOT USE MACHINERY IN AREAS THAT ARE WET, CLUTTERED, OR HAVE POOR LIGHTING.
OPERATING MACHINERY IN THESE AREAS GREATLY INCREASES THE RISK OF ACCIDENTS AND INJURY.

MENTAL ALERTNESS REQUIRED. FULL MENTAL ALERTNESS IS REQUIRED FOR SAFE OPERATION OF MACHINERY. NEVER
OPERATE UNDER THE INFLUENCE OF DRUGS OR ALCOHOL, WHEN TIRED, OR WHEN DISTRACTED.

ELECTRICAL EQUIPMENT INJURY RISKS. YOU CAN BE SHOCKED, BURNED, OR KILLED BY TOUCHING LIVE ELECTRICAL
COMPONENTS OR IMPROPERLY GROUNDED MACHINERY. TO REDUCE THIS RISK, ONLY ALLOW QUALIFIED SERVICE
PERSONNEL TO DO ELECTRICAL INSTALLATION OR REPAIR WORK, AND ALWAYS DISCONNECT POWER BEFORE ACCESSING
OR EXPOSING ELECTRICAL EQUIPMENT.

DISCONNECT POWER FIRST. ALWAYS DISCONNECT MACHINE FROM POWER SUPPLY BEFORE MAKING ADJUSTMENTS,
CHANGING TOOLING, OR SERVICING MACHINE. THIS PREVENTS AN INJURY RISK FROM UNINTENDED START-UP OR
CONTACT WITH LIVE ELECTRICAL COMPONENTS.

EYE PROTECTION. ALWAYS WEAR ANSI-APPROVED SAFETY GLASSES OR A FACE SHIELD WHEN OPERATING OR OBSERVING

MACHINERY TO REDUCE THE RISK OF EYE INJURY OR BLINDNESS FROM FLYING PARTICLES. EVERYDAY EYEGLASSES ARE
NOT APPROVED SAFETY GLASSES.

WEARING PROPER APPAREL. DO NOT WEAR CLOTHING, APPAREL OR JEWELRY THAT CAN BECOME ENTANGLED IN MOVING
PARTS. ALWAYS TIE BACK OR COVER LONG HAIR. WEAR NON-SLIP FOOTWEAR TO REDUCE RISK OF SLIPPING AND LOSING
CONTROL OR ACCIDENTALLY CONTACTING CUTTING TOOL OR MOVING PARTS.

HAZARDOUS DUST. DUST CREATED BY MACHINERY OPERATIONS MAY CAUSE CANCER, BIRTH DEFECTS, OR LONG-TERM
RESPIRATORY DAMAGE. BE AWARE OF DUST HAZARDS ASSOCIATED WITH EACH WORKPIECE MATE-RIAL. ALWAYS WEAR A
NIOSH-APPROVED RESPIRATOR TO REDUCE YOUR RISK.

HEARING PROTECTION. ALWAYS WEAR HEARING PROTECTION WHEN OPERATING OR OBSERVING LOUD MACHINERY.
EXTENDED EXPOSURE TO THIS NOISE WITHOUT HEARING PROTECTION CAN CAUSE PERMANENT HEARING LOSS.

REMOVE ADJUSTING TOOLS. TOOLS LEFT ON MACHINERY CAN BECOME DANGEROUS PROJECTILES UPON STARTUP. NEVER
LEAVE CHUCK KEYS, WRENCHES, OR ANY OTHER TOOLS ON MACHINE. ALWAYS VERIFY REMOVAL BEFORE STARTING!

USE CORRECT TOOL FOR THE JOB. ONLY USE THIS TOOL FOR ITS INTENDED PURPOSE—DO NOT FORCE IT OR AN
ATTACHMENT TO DO A JOB FOR WHICH IT WAS NOT DESIGNED. NEVER MAKE UNAPPROVED MODIFICATIONS, MODIFYING
TOOL OR USING IT DIFFERENTLY THAN INTENDED MAY RESULT IN MALFUNCTION OR MECHANICAL FAILURE THAT CAN
LEAD TO PERSONAL INJURY OR DEATH!

AWKWARD POSITIONS. KEEP PROPER FOOTING AND BALANCE AT ALL TIMES WHEN OPERATING MACHINE. DO NOT
OVERREACH! AVOID AWKWARD HAND POSITIONS THAT MAKE WORKPIECE CONTROL DIFFICULT OR INCREASE THE RISK OF
ACCIDENTAL INJURY.

CHILDREN & BYSTANDERS. KEEP CHILDREN AND BYSTANDERS AT A SAFE DISTANCE FROM THE WORK AREA.STOP USING
MACHINE IF THEY BECOME A DISTRACTION.

GUARDS & COVERS. GUARDS AND COVERS REDUCE ACCIDENTAL CONTACT WITH MOVING PARTS OR FLYING DEBRIS. MAKE
SURE THEY ARE PROPERLY INSTALLED, UNDAMAGED, AND WORKING CORRECTLY BEFORE OPERATING MACHINE.
FORCING MACHINERY. DO NOT FORCE MACHINE. IT WILL DO THE JOB SAFER AND BETTER AT THE RATE FOR WHICH IT WAS

DESIGNED.

NEVER STAND ON THE MACHINE. SERIOUS INJURY MAY OCCUR IF MACHINE IS TIPPED OR IF THE CUTTING TOOL IS
UNINTENTIONALLY CONTACTED.

STABLE MACHINE. UNEXPECTED MOVEMENT DURING OPERATION GREATLY INCREASES RISK OF INJURY OR LOSS OF
CONTROL. BEFORE STARTING, VERIFY MACHINE IS STABLE AND MOBILE BASE (IF USED) IS LOCKED.

USE RECOMMENDED ACCESSORIES. CONSULT THIS OWNER'S MANUAL OR THE MANUFACTURER FOR RECOMMENDED
ACCESSORIES. USING IMPROPER ACCESSORIES WILL INCREASE THE RISK OF SERIOUS INJURY.

UNATTENDED OPERATION. TO REDUCE THE RISK OF ACCIDENTAL INJURY, TURN MACHINE OFF AND ENSURE ALL MOVING
PARTS COMPLETELY STOP BEFORE WALKING AWAY. NEVER LEAVE MACHINE RUNNING WHILE UNATTENDED.

MAINTAIN WITH CARE. FOLLOW ALL MAINTENANCE INSTRUCTIONS AND LUBRICATION SCHEDULES TO KEEP MACHINE IN
GOOD WORKING CONDITION. A MACHINE THAT IS IMPROPERLY MAINTAINED COULD MALFUNCTION, LEADING TO SERIOUS
PERSONAL INJURY OR DEATH.

DAMAGED PARTS. REGULARLY INSPECT MACHINE FOR DAMAGED, LOOSE, OR WRONGLY ADJUSTED PARTS—OR ANY
CONDITION THAT COULD AFFECT SAFE OPERATION. IMMEDIATELY REPAIR/REPLACE BEFORE OPERATING MACHINE. FOR
YOUR OWN SAFETY, DO NOT OPERATE MACHINE WITH DAMAGED PARTS!

MAINTAIN POWER CORDS. WHEN DISCONNECTING CORD-CONNECTED MACHINES FROM POWER, GRAB AND PULL THE
PLUG—NOT THE CORD. PULLING THE CORD MAY DAMAGE THE WIRES INSIDE. DO NOT HANDLE CORD/PLUG WITH WET
HANDS. AVOID CORD DAMAGE BY KEEPING IT AWAY FROM HEATED SURFACES, HIGH TRAFFIC AREAS, HARSH CHEMICALS,
AND WET/DAMP LOCATIONS.

Electrical Safety

Grounding Methods

Grounding Methods Provided by CSA Group. (A) Receptacle with nominal rating less than 150 volts. (B) 150 volt receptacle without grounding pin fitted with adapter. (C) 150-250 volt receptacle.
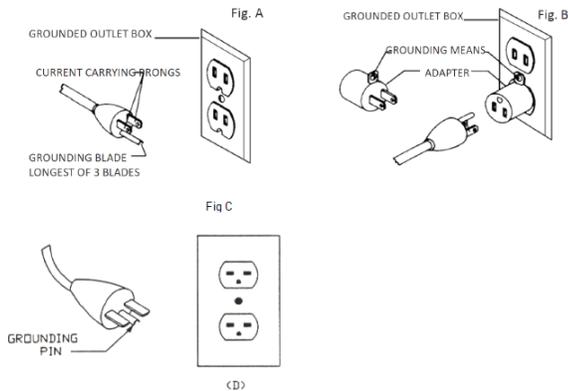
1. All grounded, cord-connected machines:
In the event of a malfunction or breakdown, grounding provides a path of least resistance for electric current to reduce the risk of electric shock. This tool is equipped with an electric cord having an equipment-grounding conductor and a grounding plug. The plug must be plugged into a matching outlet that is properly installed and grounded in accordance with all local codes and ordinances. Do not modify the plug provided - if it will not fit the outlet, have the proper outlet installed by a qualified electrician.
Improper connection of the equipment-grounding conductor can result in a risk of electric shock. The conductor with insulation having an outer surface that is green with or without yellow stripes is the equipment-grounding conductor. If repair or replacement of the electric cord or plug is necessary, do not connect the equipment-grounding conductor to a live terminal.
Check with a qualified electrician or service personnel if the grounding instructions are not completely understood, or if in doubt as to whether the tool is properly grounded.
Use only 3-wire extension cords that have 3-prong grounding plugs and 3 pole receptacles that accept the tool's plug.
Repair or replace damaged or worn cord immediately.

2. Grounded, cord-connected machines intended for use on a supply
circuit having a nominal rating less than 150 volts:
This tool is intended for use on a circuit that has an outlet that looks like the one illustrated in Fig. A. The tool has a grounding plug that looks like the plug illustrated in Fig A. A temporary adapter, which looks like the adapter illustrated in Fig B may be used to connect this plug to a 2 pole receptacle as shown in Fig B if a properly grounded outlet is not available. The temporary adapter should be used only until a properly grounded outlet can be installed by a qualified electrician. This adapter is not permitted in Canada. The green-colored rigid ear, lug, and the like, extending from the adapter must be connected to a permanent ground such as a properly grounded outlet box.

3. Grounded, cord-connected tools intended for use on a supply circuit

having a nominal rating between 150 – 250 volts, inclusive:

This tool is intended for use on a circuit that has an outlet that looks like the one illustrated in Fig C. The tool has a grounding plug that looks like the plug illustrated in Fig C. Make sure the tool is connected to an outlet having the same configuration as the plug. No adapter is available or should be used with this tool. If the tool must be reconnected for use on a different type of electric circuit, the reconnection should be made by qualified service personnel; and after reconnection, the tool should comply with all local codes and ordinances.

Receiving Your Machine

Your machine will likely be delivered by a third party service. Before you unpack your machine, you will first need to inspect the packing, invoice and shipping documents supplied by the driver.

Ensure that there is no visible damage to the packing or the machine. You must do this prior to the driver leaving. All damage must be noted on the delivery documents and signed by you and the delivery driver. You must then contact the seller, Laguna Tools, within 24 hours. It is advisable to photograph any shipping damage to support an insurance claim.

Note: You may find sawdust in your machine upon arrival. This is because the machine has been tested prior to shipment from the factory and / or Laguna Tools. Laguna Tools endeavors to test machines prior to shipping to customers as movement can take place during transportation. It must be noted that additional machine movement can take place between Laguna Tools and the end user and some adjustments may have to be undertaken by the customer. These adjustments are covered in the various sections of this manual.

⚠ WARNING! Most large machinery will be delivering on a tractor trailer 48'-53' long. Please notify Sales Representative with any Delivery Restrictions.

Customer is required to have a forklift (6000lb. or larger is recommended) with 72" forks or fork extensions and operator.

Note any visible damage, torn packaging, scuffs or any abnormal marks on the delivery receipt or Bill of Lading (BOL).

Delivery Protocol Sheet

[Click to select date]   BILL OF LADING – SHORT FORM – NOT NEGOTIABLE   Page 1 of 1

SHIP FROM	Bill of Lading Number:
Laguna Tools 744 Refuge Way Suite #200 Grand Prairie, TX 75050 SID No.:	BAR CODE SPACE

SHIP TO	Carrier Name:
[Name] [Street Address] [City, ST ZIP Code] CID No.:	Trailer number: Serial number(s):

THIRD PARTY FREIGHT CHARGES BILL TO	SCAC:
[Name] [Street Address] [City, ST ZIP Code]	Pro Number: BAR CODE SPACE

Special Instructions:	Freight Charge Terms (Freight charges are prepaid unless marked otherwise): Prepaid ☐ Collect ☐ 3rd Party ☐ ☐ Master bill of lading with attached underlying bills of lading.

CUSTOMER ORDER INFORMATION

Customer Order No.	# of Packages	Weight	Pallet/Slip (circle one)		Additional Shipper Information
			Y	N	
			Y	N	
			Y	N	
			Y	N	
Grand Total					

CARRIER INFORMATION

Handling Unit		Package						LTL Only	
Qty	Type	Qty	Type	Weight	HM (X)	Commodity Description Commodities requiring special or additional care or attention in handling or stowing must be so marked and packaged as to ensure safe transportation with ordinary care. See Section 2(e) of NMFC item 360		NMFC No.	Class

Where the rate is dependent on value, shippers are required to state specifically in writing the agreed or declared value of the property as follows: "The agreed or declared value of the property is specifically stated by the shipper to be not exceeding _____ per _____.	COD Amount: $ Fee terms: Collect ☐ Prepaid ☐ Customer check acceptable ☐

Note: Liability limitation for loss or damage in this shipment may be applicable. See 49 USC § 14706(c)(1)(A) and (B).

Received, subject to individually determined rates or contracts that have been agreed upon in writing between the carrier and shipper, if applicable, otherwise to the rates, classifications, and rules that have been established by the carrier and are available to the shipper, on request, and to all applicable state and federal regulations.	The carrier shall not make delivery of this shipment without payment of charges and all other lawful fees. Shipper Signature _____

Shipper Signature/Date	Trailer Loaded: ☐ By shipper ☐ By driver	Freight Counted: ☐ By shipper ☐ By driver/pallets said to contain ☐ By driver/pieces	Carrier Signature/Pickup Date
This is to certify that the above-named materials are properly classified, packaged, marked, and labeled, and are in proper condition for transportation according to the applicable regulations of the DOT.			Carrier acknowledges receipt of packages and required placards. Carrier certifies emergency response information was made available and/or carrier has the DOT emergency response guidebook or equivalent documentation in the vehicle. Property described above is received in good order, except as noted.

Machine Placement

Your machine will likely be delivered by a third party delivery service. Before unpacking your new machine, first inspect the packaging, the invoice, and the shipping documents supplied by the driver. When unpacking your machine, separate all enclosed items from the packing materials and inspect them for damages. Ensure that there is no visible damage to either the packaging or the machine **BEFORE** the driver leaves. Save all packaging materials until you are satisfied with the machine and/or have resolved any issues concerning any missing or damaged items.

NOTICE! All shipping damage must be noted upon delivery and signed by the owner and the delivery driver. If you find any damaged items in your package, you must contact Laguna Tools to file a complaint. In order to return damaged goods under the limited warranty to Laguna Tools, Inc., you MUST have the original packaging. All claims of loss or damaged goods must be reported to Laguna Tools within 24 HOURS of delivery. Please contact the Laguna Tools, Inc. Customer Service Department to make claims for any damaged items/parts.

NOTICE! It is probable that you will find sawdust within your machine. This is because the machine has been tested prior to shipment from the factory and or Laguna Tools. We test machines prior to shipping to customers, but movement can take place during transportation. Some adjustments may have to be undertaken by the customer. These adjustments are covered in the various sections of this manual.

PLACEMENT: Before you remove your machine from the packaging, select the area where you will use your machine. There are no hard and fast rules for its location, but below are a few guidelines:

1. There should be sufficient area at the front of the machine to allow you to work on it comfortably.
2. There should be sufficient area at the back of the machine to allow access for adjustments and maintenance to be conducted.
3. Adequate lighting. The better the lighting the more accurately and safely you will be able to work.
4. Solid floor. You should select a solid flat floor, preferably one made of concrete or something similar.
5. Locate it close to a power source and dust collection.
6. Allow an area for the storage of blanks, finished products and tools.

LOCATING YOUR MACHINE: The physical environment where you locate your machine is important for safe assembly and operation of your machine. Before removing your dust collector from the packaging consider the weight load, electrical installation requirements, lighting, dust collection, and space allocation available for the band saw and accompanying materials.

SPACE ALLOCATION/DUST COLLECTION: Consider the largest size and length of wood or other materials which will be processed through your machine. Leave ample around the machine for the operator to handle both the equipment and the materials begin cut. Leave enough space around the machine to open or remove doors/covers as require by the maintenance described in the owner's manual. Allow enough space for proper dust collection from your machine. For optimal operation, ensure that your machine is located in a dry environment free from excessive moisture, extreme weather conditions, hazardous chemicals, or airborne abrasives.

ELECTRICAL REQUIREMENTS: Place your machine near an existing power source with the appropriate voltage required to operate your machine. Ensure that all power cords are protected from traffic, moisture, chemicals, or other hazards. For you safety, have a qualified electrician assess your electrical needs and grounding if you have any doubts about your own ability to do so. We **DO NOT** recommend that you use an extension cord to supply power to your machine.

LIGHTING: Ensure that the lighting your band saw is placed under is adequate enough that regular operation and maintenance can be performed safely. Any glares, shadows, or strobe lighting which may distract or prevent the operator from safely operating the machinery should be removed from the working area.

MACHINE WEIGHT LOAD: Ensure that the surface where your machine is located can bear the full weight of your machine as well as any additional equipment which may be placed on the band saw. Also consider the weight of the operator and any materials which may be stored around the machine when selecting a location for your machine.

Unpacking Your Machine

To unpack your machine, you will need tin snips, a knife, and a wrench.

1.) Using the tin snips, cut the banding that is securing the machine to the Pallet [if fitted].

⚠ WARNING! **EXTREME CAUTION MUST BE USED BECAUSE THE BANDING WILL SPRING AND COULD CAUSE INJURY.**

2.) Remove the box from the CNC machine if fitted and any other packaging material. The parts ordered with the machine will be packed on or inside the machine. Note. The machine is heavy, and if you have any doubt about the described procedure, seek professional assistance. Do not attempt any procedure that you feel is unsafe, or that you do not have the physical capability of achieving.

3.) Use a forklift with sufficient lifting capacity and forks that are long enough to extend the complete width of the machine. *NOTICE:* One should obtain a 7000 lbs. forklift with 6' fork extensions

4.) Remove the securing bolts that attach the machine to the pallet [if fitted].

5.) Approaching the machine from the side, lift the machine on the frame taking care that there are no cables or pipes around the forks.

6.) Move the machine to the required position and lower gently to the floor.

7.) Level the machine so that all the supporting feet are taking the weight of the machine and no rocking is taking

place.

Machine Specifications

	Swift 24" x 36"	Swift 48" x 48"	Swift 48" x 96"
Motor	2hp very quiet spindle	2 hp very quiet spindle	2 hp very quiet spindle
Spindle	1 or 3 industrial induction spindle, liquid-cooled	1 or 3 industrial induction spindle, liquid-cooled	1 or 3 industrial induction spindle, liquid-cooled
Spindle RPM	5,000 - 24,000	5,000 - 24,000	5,000 - 24,000
Controller	HHC Standard Hand Held	HHC Standard Hand Held	HHC Standard Hand Held
Dust Chute Diameter	2 7/8" 4 inches with adapter	2 7/8" 4 inches with adapter	2 7/8" 4 inches with adapter
Volts	220V single phase/ 20 amp	220V single phase/ 20 amp	220V single phase/ 20 amp
Ball Screw Adjustable on all Axis	Yes	Yes	Yes
Controller	Laguna HHC Controller	Laguna HHC Controller	Laguna HHC Controller
Gantry Clearance	5 inches	6 inches	5 inches
Machine Work Table	T-slot	T-slot	T-slot
Z Axis	Precision Ball Screw	Precision Ball Screw	Precision Ball Screw
X and Y Axes	Precision Ball Screw	Precision Ball Screw	Precision Ball Screw
Machine Foot Print	45" w x 61" l x 53" h	73" w x 80" l x 55" h	67" w x120" l x 58" h
Work Envelope	24" x 36"	48" x 48"	48" x 96"

Machine Overview

Swift Series: Swift MT (Multi-Tool)

Sign Making - Vinyl Cutting - Production
The Laguna Swift MT has everything you need to handle your routing, cutting, and creasing operations.
With the Laguna Vision System, you can quickly find registration marks to identify parts and adjust the cutting path,
giving you perfect cutouts in printed shapes.

Features:
- 3HP Liquid-Cooled Industrial Spindle
- LAGUNA Vision System
- Tangential Knife System
- One-Piece Welded All-Steel Frame
- 4 to 6, Zone Phenolic Vacuum Table
- Helical Rack and Pinion on X and Y-Axis
- Ground Ball Screw on Z-Axis
- 7" Gantry Clearance

Special Features:

Machine Footprint (W x L x H):
- 57.99" x 67.92" x 75.12"
- 57.99" x 135.83" x 75.12"
- 69" x 160" x 78"

Type of Table
- Phenolic Universal Grid-Type Vacuum Table With T-Slot Feature
Four (4) – Six (6) Independently Controlled Vacuum Zones

Spindle Details
- 3HP Liquid-Cooled Spindle **RPM:** 6,000 rpm – 24,000 rpm
Dust Port Diameter: 4"

Gantry Details
- Rapid Travel Speed: 900"/min.
- Max Cut Speed: 600"/min.
- Gantry Height: 12"
- Gantry Clearance: 7"
- 20mm Heavy Duty Linear Bearing on All Axis
- Helical Rack-And-Pinion Drive System X & Y-Axis

Cooling Systems
- Liquid-Cooling System

Power Requirements
- 200 Volts Electrical Power

- 40 Amps
- 3 Phase or 1 Phase with Vacuum w/Variable Frequency Drive-VFD
-

<u>Worktable Size (W x H):</u>

- 4' x 4'
- 4' x 8'
- 5' x 10'

Introduction to CNC Machines

Introduction to the CNC

The CNC is designed to give you years of safe service. Read this owner's manual in its entirety before assembly or use. The advantage of the CNC machine is that it can, in most cases, fully machine the complete job without it being removed from the table so that you have finished parts of high accuracy that are totally repeatable. If the relevant software is purchased and installed, it can also produce intricate carvings. Nesting is also a valuable feature of CNC machining that saves on waste and costs. It is possible to reduce the number of different machines in the shop as the CNC will perform a multitude of functions and meet the needs for cabinet makers and serious woodworkers.

⚠️ **CAUTION** Like all machines, there is danger associated with the machine. Injury is frequently caused by lack of knowledge or familiarity. Use this machine with respect. If normal safety precautions are overlooked or ignored, serious personal injury may occur. As the CNC is under the control of the on board machine controller, it is important that you are clear of the cutter when operating the machine.

Parts of the CNC Machine:
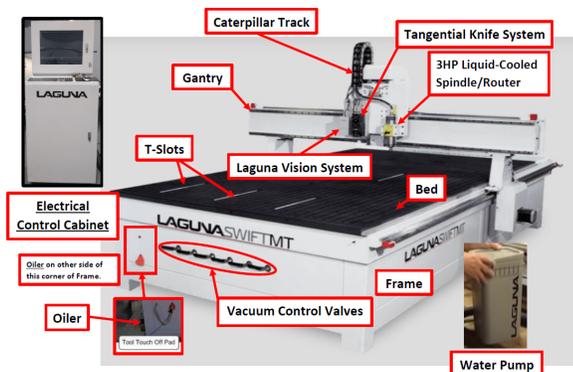
Part Name	Description
Bed	The bed of the machine consists of a heavy steel frame and a "T-Slot" Table. The "T" slots are used to clamp the job or fixtures to the bed.
Gantry	The gantry straddles the bed and carries the router spindle motion system. It is moved along the length of the bed by a precision pinion gear system that is controlled by the machine controller.
Spindle/Router	The router spindle is moved along the gantry by a precision ball screw system that is controlled by the machine controller.
Frame	The frame is a heavy welded construction that supports all the other parts of the machine.
Electrical Control Cabinet	The electrical control cabinet is located on the side of the machine in a dust free enclosure.
Caterpillar Track	The caterpillar track runs along the side of the machine in a trough and carries all the electrical cables and the spindle cooling tubes.
Water Pump	The water pump provides coolant for the router spindle motor. Running the router spindle without the cooling pump running can lead to spindle bearing failure.

Assembly & Setup

<u>Installing the Water Pump:</u>

The following is the assembly & set-up for the Swift MT's water pump.

1) Locate the two holes on the top of the pail.

2) Place the two separate clear hoses into the two holes on the top of the pail. ***NOTICE!*** The fit will be tight, so keep dirt out of the pail.

3) Place the end of the clear hose (should now be threaded through the pail's holes) into the blue fitting on the water pump.
Once attached, the fitting with lock in automatically.

NOTICE! To release, the hose from the fitting, press down on the top of the blue ring on the fitting and the hose will be released.

Note: One will need to provide a coolant tank with a minimum capacity of 5 gallons. **If the shop temperature is high, the tank size**
will have to be larger. If your shop is likely to be subject to freezing temperatures, Antifreeze must be added to the cooling water.

4) Place the water pump into the pail.

5) Place the water pump's black electrical cord (110V) into the pail's slot. Ensure that the black cord is leading to an electrical outlet.

6) Fill the about 3/4 full of water. ***NOTE: Using distilled water is recommended.***

7) Place the water pump into the pail and seal the lid on the pail.

8) Place the pail underneath the work bench,work station or any place so that the pail is out of the way.

9) Place the electrical plug into any 110V outlet to start operation.

PC Components

There are circuit boards that have been installed into the PC's tower. If the PC is to be replaced the cards must be re-installed into the new PC tower.
The WinCNC software must then be re-installed.
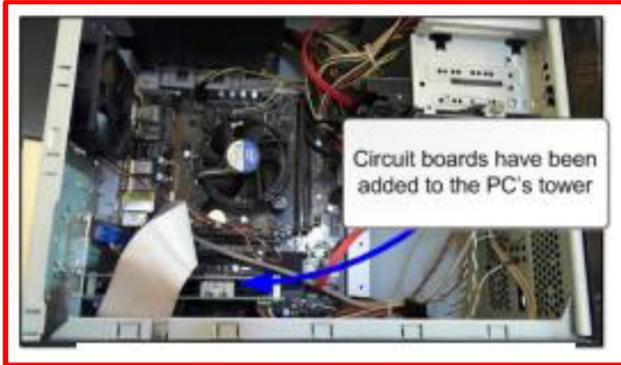
PC Tower showing Internal PCI

Setting Up the Electrical Cabinet

1.) Position the electrical cabinet so that it is perpendicular to the machine.
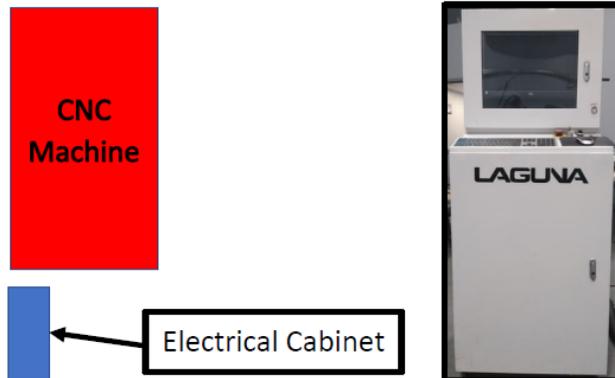
2.) Connect PCI cables to PC. *These cables are labeled. Note: The machine will malfunction if the PCI cables are connected incorrectly.*
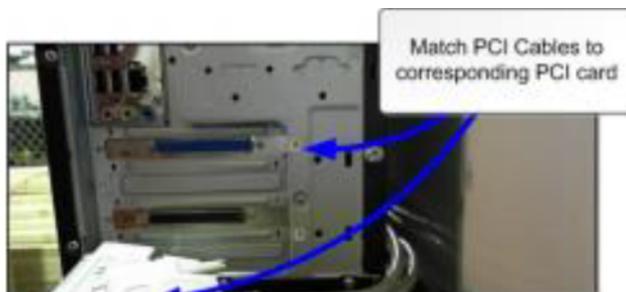
3.) Route power to the machine for the PC.

The machine requires **220V Electrical Power** connected to the primary power switch located on the front of the cabinet.
WARNING – Contact an electrician if there is any uncertainty about how to properly connect your machine to power.

1.) Connect 220V to the primary switch.

2.) Route power supply through the side openings on the electrical cabinet.

Power Supply Connections

Fitting the Dust Hose

1.) Fit the dust hose to the dust shroud and secure with the clamp. Ensure that it is tight as it is very inconvenient to have it fall off during production.
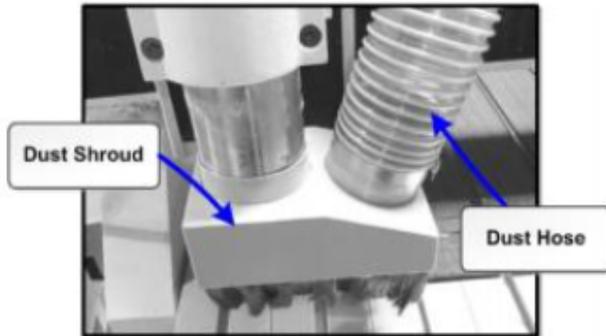
2.) Fit the shroud to the router head. Adjust the shroud so that the brushes are even with the tip of the router bits that you will be using. If it is too low the brushes may drag on the job being cut and could be cut by the router bit. If it is too high and is not in contact with the job, suction may not be optimal. Once adjusted, clamp with the clamping Allen screw.

3.) Feed the dust hose through the **Dust Hose Support**.
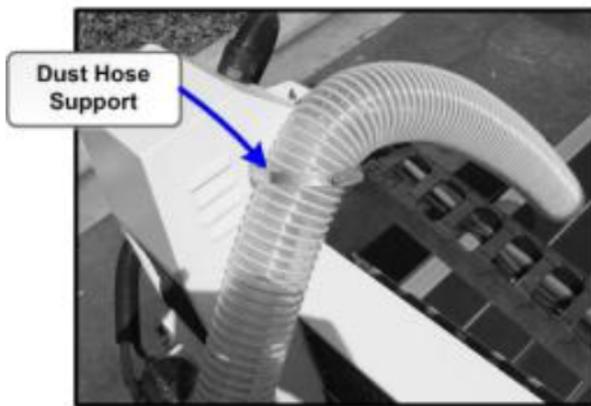
4.) Fit the dust hose adapter to the end of the dust hose and secure with the clamp. The adapter allows a 4" hose to be connected to the machine.

5.) The head of the machine will move across the complete table and the dust hose will follow the head. If there is insufficient slack, the hose may break or damage the dust shroud. It is suggested that the hose be suspended from the ceiling of your shop with sufficient slack so that it will not restrict movement. It will also be out of the way and not cause a tripping hazard.

Control Panel

Functions of the control panel:

⚠ WARNING! If the spindle is run without cooling then it could be damaged and fail.

Cabinet Lock

TRIPPED

I-ON

O-OFF

RESET
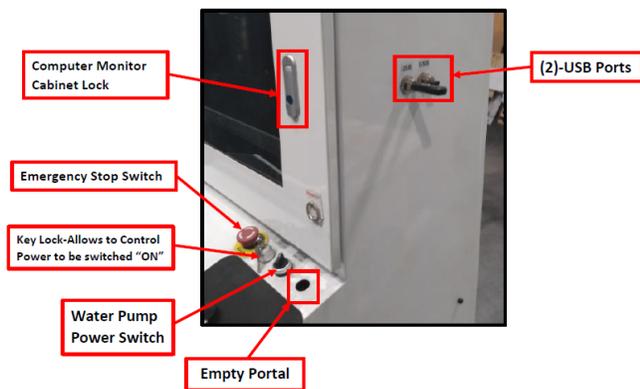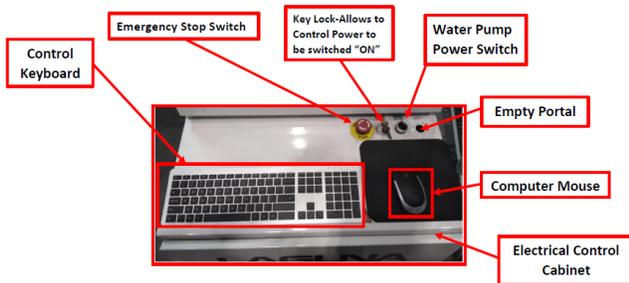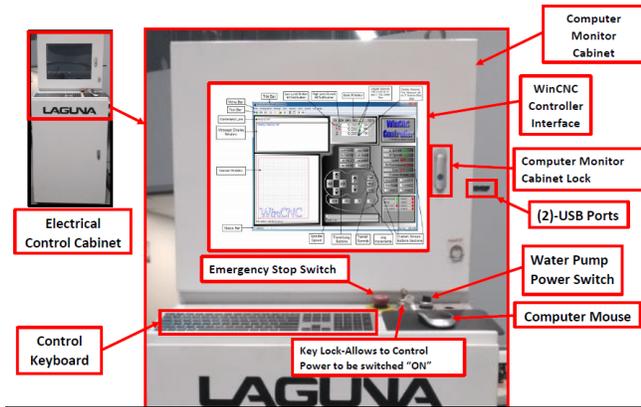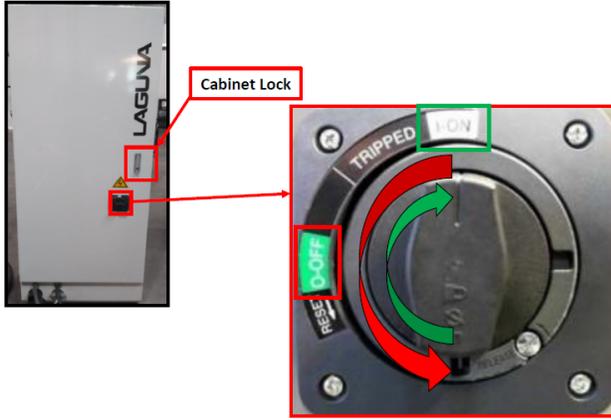
Primary Power Switch in the "ON" Position.

Controller Interface Description

Features of the controller interface.

<u>**Screen Display Overview:**</u>

1. Menu Bar
2. Tool Bar
3. Command Line
4. Message Display window
5. Viewer Window
6. Status Bar
7. Spindle Speed Control
8. Transit/Jog buttons
9. Transit Speeds
10. Jog Increments
11. Emergency Switch State Feedback
12. Tool touch-off pad state feedback
13. Touch probe switch state feedback
14. Command Buttons : See Button Functions
15. Spindle ON/OFF Control
16. Blue Square: Tool measure set for Z
17. Green Square: Local zero is set for Y
18. Red Square: indicates absolute coordinates are being displayed
19. Axis Window
20. Low Limit Switch Hit Notification

Menu Bar

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 below. The menu bar selections are described in the following pages. If a menu option has a shortcut key, the shortcut key combination is listed in parenthesis () after the name of the menu option.

1) File Menu:

Menu Button	Function
Open (CTRL+O)	Opens a file.
Edit (CTRL+E)	Opens the editor. If a file name is in the command line, the editor opens that file.
Simulate (CTRL+S)	Simulates a file running to check for errors and estimate run time.
View (CTRL+V)	Views a file in the viewer window.
Restart (CTRL+R)	Opens the file restart dialog box to restart a file.
Create Home File (CTRL+H)	Creates a home file at the current machine position.
Import 1.7.1. Import => DXF	Allows the user to import DXF (Drawing Exchange Format) file.
Import => HPGL	Allows the user to import HPGL (Hewlett-Packard Graphics Language) file

Note: The DXF and HPGL import feature is intended for cutting simple designs. WinCNC does not compensate for tool diameter when converting these files. For more complicated design, where intelligent tool pathing is important, please use a compatible CAD/CAM program.

MENU BAR:

Menu Button	Function
Digitize 1.8.1. Digitize => Manual	Enable/Disable the Manual Digitizing Toolbar.
Digitize => Automatic	Automatic digitizing (L802, L803).
Digitize => Frame	Skeletal Digitizing (L810). This scanning method scans a center line or spine, and horizontal lines or ribs of the object.
Digitize => Automatic	Automatic digitizing (L802, L803)
Digitize => Frame	Skeletal Digitizing (L810). This scanning method scans a center line or spine, and horizontal lines or ribs of the object.
Digitize => Outline	Creates an outline trace around an object.
Digitize => Laser => Run Scan	Opens the laser digitization dialog box to set up laser scanning parameters.
Digitize => Laser => Reprocess Data	Opens a data file from a previous laser scan used to re- generate a 3D object with different filtering options to improve the quality of the 3D object.
Exit	Closes WinCNC.

2) Configuration Menu:

Menu Button	Function
D/A Calibration	Calibration Settings for D/A (Digital to Analog) spindle speed control.

3) Settings Menu:

Menu Button	Function
Positions	Opens the positions dialog box with options to view and edit stored positions. (Ex: G53 P# to call position in G-Code)
Home Positions	Opens the positions dialog box with options to view and edit stored home positions. (Ex: G0 H# to call home pos.)
Tool Positions	Opens the tool positions dialog box with options to view and edit stored tool positions for use with automatic tool changers. (This requires an additional software feature to be enabled)
Tool Library	Edits and views the defined tool library.
Local Coordinates	View XYZ coordinates.

4) View Menu:

Menu Button	Function
Resolution	Views current resolution settings for each axis.
Acceleration	Views current acceleration settings for each axis.
Limits	Views current limit settings for each axis.
Inputs	Views real-time input states.
Outputs	Views real-time output states.
Toolbars	Enable/Disable the main toolbar.
Toolbars =>Size	Change toolbar from small to large.
Refresh View (F5)	Refreshes the screen.
Clear Screen (CTRL+C)	Clears the message display window

5) Options Menu:

Menu Button	Function
Buttons	Used to customize buttons.
Auto Repeat	Auto populate the command line with the last command executed.
Auto Run	Enable/Disable the auto run feature.
Auto Preview	Enable/Disable the auto preview feature.
Single Step (CTRL+P)	Enable/Disable single line execution of G-Code files.

6) Units Menu:

Menu Button	Function
Inches	Sets the unit of measure to inches.
Centimeters	Sets the unit of measure to centimeters.
Millimeters	Sets the unit of measure to millimeters.

7) Jog Menu:

Menu Button	Function
0.001 (F6)	Set's jog increment to 0.001 units of measure.
0.01 (F7)	Sets the jog increment to 0.01 units of measure.
0.1 (F8)	Sets the jog increment to 0.1 units of measure.
1 (F9)	Sets the jog increment to 1 unit of measure.
Custom (F10)	Sets a custom jog increment.

8) Help Menu:

Menu Control	Function
Help Topics (F1)	Activates the in-program help menu.
Update Program	Opens the update utility.
About	Displays the WinCNC "About" box which contains important information about your specific software package. The About box displays your security key serial number, the software version number, your user level, the maximum number of axes, the table size limitation, and which additional features you have enabled. If you do not have a feature enabled that you would like or need, you can contact your software vendor for an upgrade.

Tool Bar

The **Tool Bar** is a collection of shortcut buttons that perform specific actions. The tool bars in WinCNC are floating and can be positioned or docked according to the user's preference. The image below shows the standard and manual digitizing tool bars docked in the normal toolbar area. The explanation of each button on the toolbar is as follows. The explanations move sequentially from left to right along the toolbar.

Tool Bar Overview:
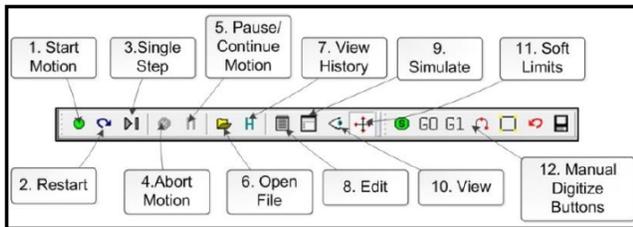
Tool Bar Button	Function
Start Motion	This button will begin the command or job listed in the command line.
Restart	Allows you to pick a job file and the line to start that file on. This is useful if you want to skip over lines in a job file, or if you have aborted a job and want to start back at the point where you aborted from. You can also choose to run the file in single step mode.
Single Step	When selected, this allows you to execute one line of a G-Code file each time you hit ENTER.
Abort Motion	Aborts a command or job that is running.
Pause/Continue Motion	Pauses/Continues a command or job that is running.
Open File	Opens a browse box that is used to open a job file.
View History	Opens the command history box, which allows a user to execute a command used previously. When an error is found in the WINCNC.INI file, the line containing the syntax error is displayed here in red when WinCNC starts.
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.

Manual Digitize Buttons

Manual Digitize Buttons: 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 > Tool bars section of the menu bar.
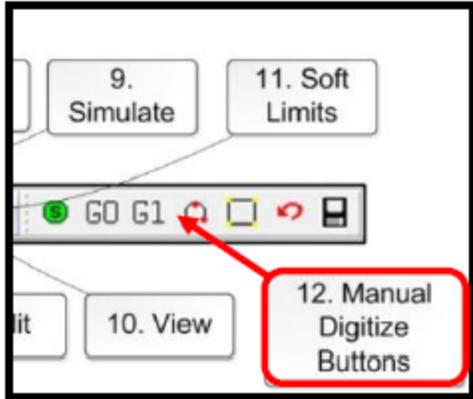
Manual Digitize Button	Function
Start a Manual Digitized File	Starts a manual digitized file and enables the manual digitize mode.
Add a Rapid Move	Adds a rapid move to the manual digitized file.
Add a Feed Move	Adds a feed move to the manual digitized file.
Add an Arc Point	Used to add arcs into a manual digitized file.
Close Shape	Used to close the last move in a shape without moving the machine.
Undo	Used to undo previous moves.
Save	Allows you to save the manual digitized file in either G-Code or DXF file format.

Axis Window

The axis window is the primary display for current information about each axis on your machine. The current positions and velocities for each axis are displayed in the text boxes. Displayed above the position and velocity boxes is the current unit of measure, any currently active modes, and the current feed rate override percentage.

Red Triangles to the left and right of each axis label indicate low or high limit switch hits. A **Red Triangle** to the left of an axis label indicates a low limit switch hit, and to the right indicates a high limit switch hit.

A **Green Box** displayed to the right of the position box indicates a temporary workspace **(G92)** setting, and a **Blue Box** indicates a tool length measure or workspace setting. To the far-right side of the axis window is the feed rate override control slide bar.
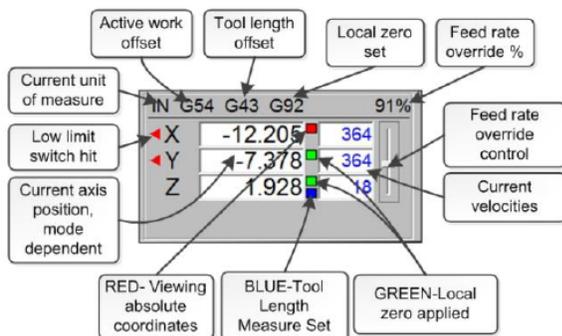
Axis Window Button	Function
Set XY	Executes a G92XY command. Sets a local zero for the X and Y axes.
Set Z	Executes a G92Z command. Sets a local zero for the Z axes.
Clear	Executes a G92 command. Clears all local zeros set by the G92 command.
Seek Home	Executes a G28 command. Homes the machine and resets **Machine "0"-Zero Coordinates.**
Touch Top	Measures the length of the tool mounted in the spindle. See Chapter **"Basic Operations"** section **"Measuring Tool Length"** for procedure.
Calculator	Opens Windows calculator application.
Meas. Z	Executes a M37 command. This setting can be used to set a maximum depth that the Z head can travel. See Chapter **"Basic Operations"** section **"Setting Tool Length Offset"** for procedure.

Transit Speed Buttons

Transit/Jog Buttons.
Manual Transit Buttons. Hold the directional button
down to continuously move the machine.
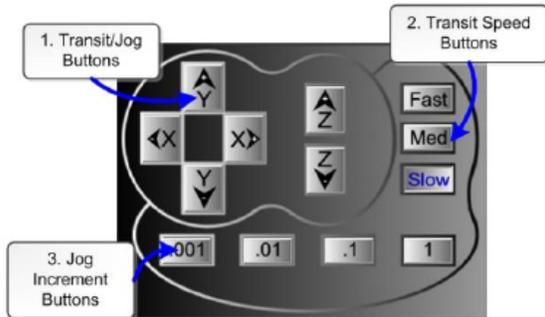
Transit Speed Buttons.
Allows the user to adjust between
3-Three predefined Transit Speeds (Fast, Med., Slow).
The speeds can vary for each Axis.

Transit Speed Button	Fast	Medium	Slow
X	400 in / min	200 in / min	50 in / min
Y	400 in / min	200 in / min	50 in / min
Z	100 in / min	50 in / min	25 in / min

The table below shows the Default Settings:
Jog Increment Buttons.
These Buttons allow you to specify the increment of a jog move.
The default **unit of measure is "inches"**.

Input Indicators
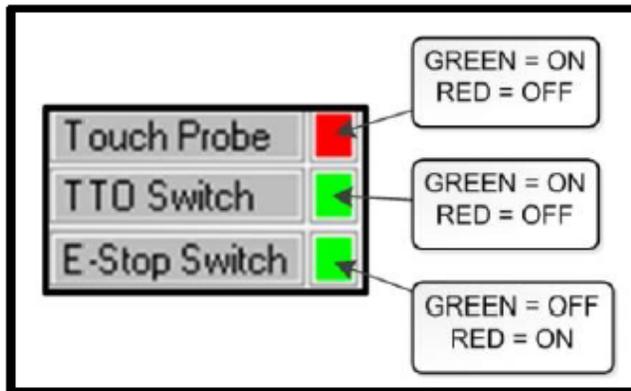
Touch Probe – Indicates if touch probe has been triggered.
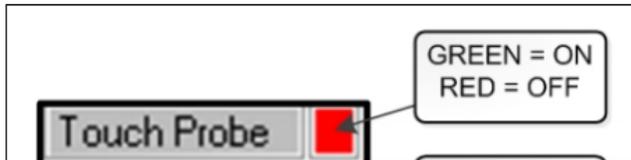
TTO Switch – Indicates whether the tool touch pad has been grounded.

E-Stop Switch – Indicates if the **"Emergency Stop Switch"** has been triggered.
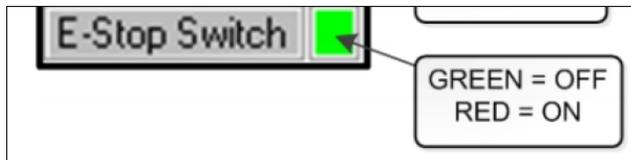
Basic Machine Operations

The following is the procedure to start up the machine:

1) Power **"ON"** the PC.
2) Start **WinCNC** by either clicking on the desktop shortcut or locating the executable file winCNC.exe on the (C:) drive in the WinCNC folder.

NOTE: <u>It is important that the WinCNC application be running on the PC before the machine is powered. This is to ensure that the user will always have control over the machine. If the machine is powered without the WinCNC application running, the machine may be susceptible to electronic noise causing abnormal behavior.</u>

3.) Turn the main power switch to the "ON" position. The switch is located on the back of the electrical cabinet.
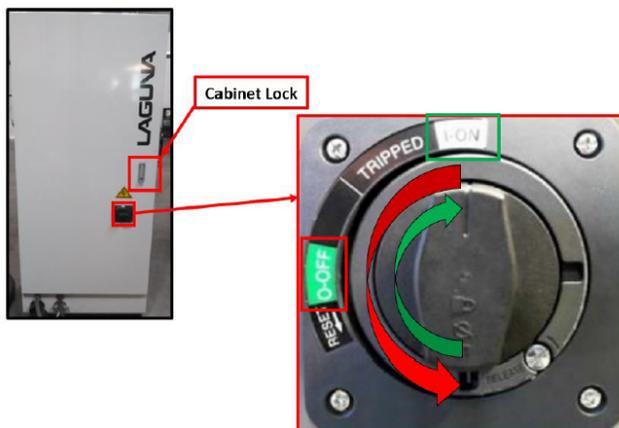
4.) Turn the machine key the right towards the **"ON"** Position.

5.) Turn the Black Water Pump Switch to the right towards the **"ON"** Position.

Shutting Down The Machine

The following is the procedure to properly shut down the machine:

1.) Turn the power key to **"OFF"** position.

2.) Turn the Main power switch to the **"OFF"** position.
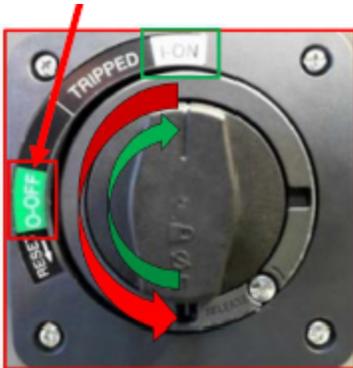
3.) **Close the desktop WinCNC** application.

"WinCNC Desktop"

4.) **Shut Down or Turn "OFF"** the PC.

Emergency Stop Button

Emergency Stop Button

There is an **"Emergency Stop Switch"** located on the electrical cabinet. Press during any machine operation to halt the machine.

Reset after an **"Emergency Stop Button"** – Rotate the **"Emergency Stop Switch"** Clockwise a 1/4 turn.

Keyboard Emergency Stop Option

"**ESC" Key – Pressing the "ESC" key** on the keyboard will abort the current file or command being executed.
NOTICE! The program or command will have to be reentered after aborting.
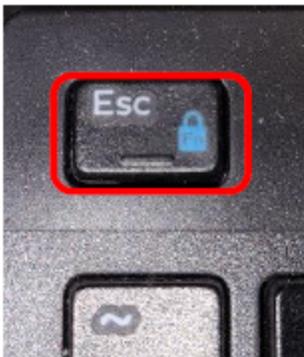
"**SPACE BAR" – Pressing the "SPACE BAR"** will pause a file or commands execution.
Press the "ENTER" key to resume executing the program or command.

Assembly & Setup

Homing The Machine

⚠ WARNING! It is important that the machine be homed upon start up to ensure the absolute coordinates are reset. This is because a machine can lose its position when it is in the OFF state.
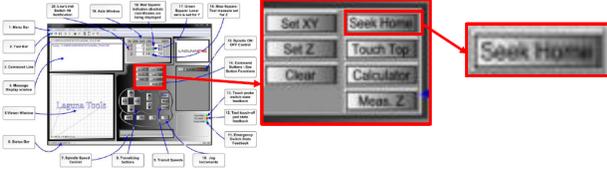
1) After the machine is powered **"ON"**, **"Click the "SEEK HOME" Button"** located in the display screen. The machine will automatically seek home and zero out the machine coordinates displayed in the Axis Window.
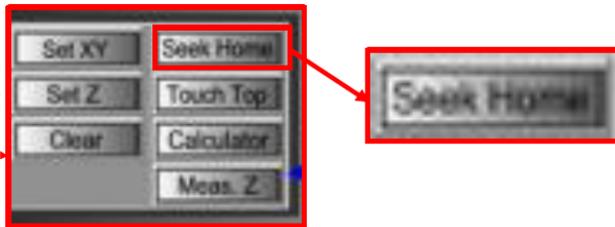
Finding a Router Bit

1.) Select a **"Router Bit"** and its relevant **"Collet".**

Note: The Collets and Spindle Collet Hole must be cleaned regularly. Ensure that the slots in the Collets are free of sawdust as sawdust builds up and will stop the Collet from compressing. If the Collet or Spindle Hole are not clean, the router bit may not run true, and this will affect the performance of your machine.

2.) Fit the collet into the spindle nut. Press the collet into the spindle nut until it snaps into place, as shown in the figure shown below.

Spindle Nut

Collet

=

Collet plus Spindle Nut
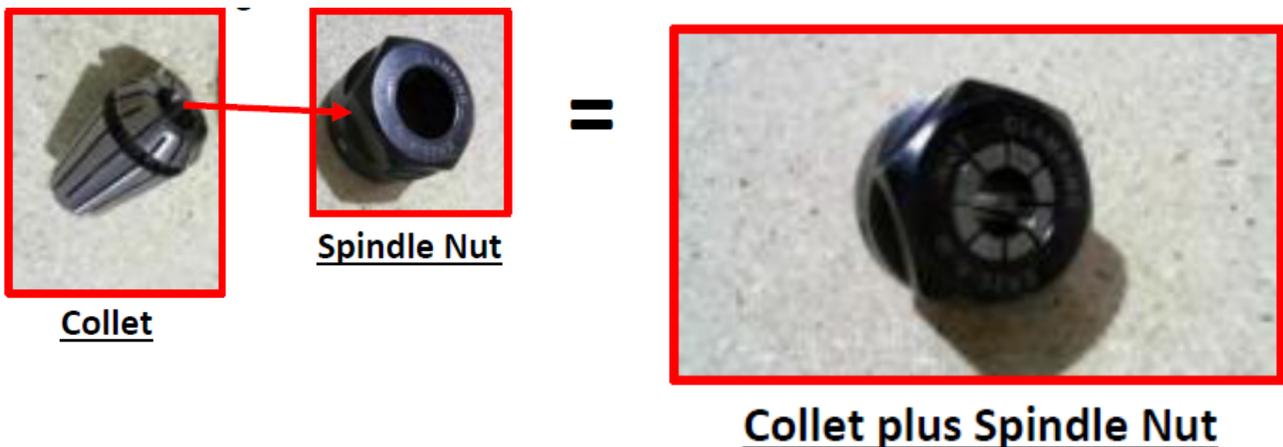
Note: The router bit must not be fitted into the collet until the collet has been fitted into the spindle nut. With the router bit fitted into the collet, the collet cannot compress and snap into the spindle nut. The face of the collet and the face of the spindle nut will be close to flush.

Note: To remove the collet, hold the spindle nut and press the collet on the side. The collet will compress and pop out. Do not try to remove the collet while a cutter is fitted as the collet will not compress and pop out.

4.) Fit the Spindle Nut and Collet Assembly onto the Spindle thread by hand.

Collet + Spindle Nut

Electronic Spindle
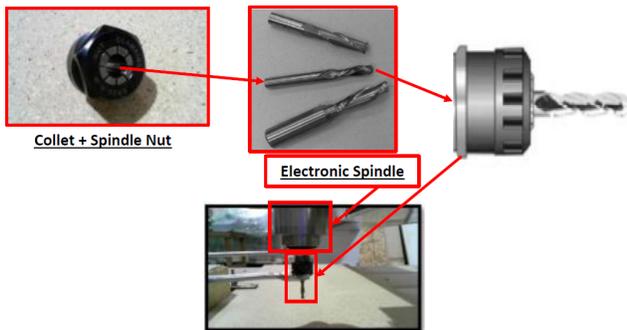
5.) Press the bit into the collet but note, **the flute of the router bit must not be inside the collet and must be a minimum of 1/16 " outside the collet.** Hold the Router Spindle with the supplied wrench and tighten the Collet with a second wrench. **Do not over Tighten.**

The flute of the router bit must not be inside the collet and must be a **minimum of 1/16 "** outside the collet.

Note: Use this process for all other router bits that you need to fit but you will have to change the collet if the shank of the router bit is a different size.

Spindle Operation

The spindle can either be controlled using commands or by the spindle control buttons in the controller interface.
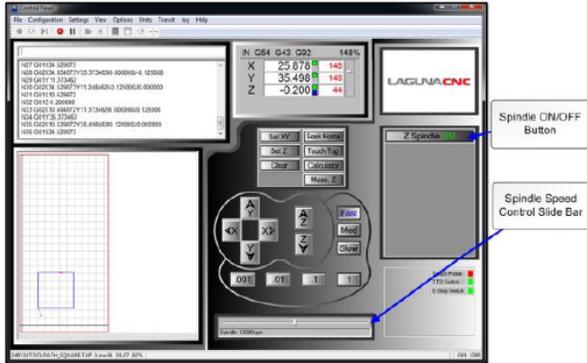
Controller Interface Spindle Controls

NOTICE!: There will be a 10 second delay when the spindle is initially turned **"ON"**. This is to allow the spindle to reach the correct RPM before executing any other commands. This is intended to protect the machines components and optimize the cutting process. The user will be unable to perform any actions during this time. The only way to abort this command is to **"Press the "Esc" key on the keyboard"**.

Spindle Control using the Command Line.

M3 – Turns the Spindle **"ON"**. Entering M3 in the command line will power the spindle, but it may not begin rotating unless an RPM is set.

Adjust Spindle RPM manually by entering the **character 'S'** followed by a number between **0-24000.**

Example:	S12000	Sets Spindle Speed at 12000 RPM
	M3	Turn "On" Spindle
	M5	Turn "Off" Spindle

Spindle control using the Display screen.
Pressing the "Z" Spindle Button will turn the Spindle **"ON"** or **"OFF"**.

Controller Interface Spindle Controls
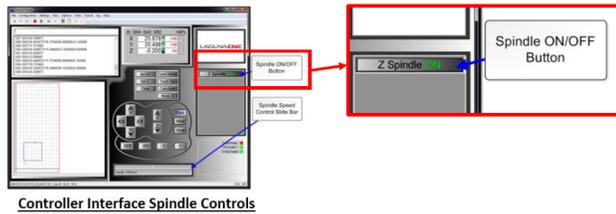
The Spindle Speed can be adjusted while spindle is **"OFF"** or **"ON"**.
The Spindle Speed can be adjusted while **"Executing Code"**.

The bar can be adjusted by "Clicking and Dragging the marker or using the keyboards "+" and "-"key on the keypad.
The RPM will increase by 240 RPM per button "Click if using the Keyboard Shortcut".

Controller Interface Spindle Controls
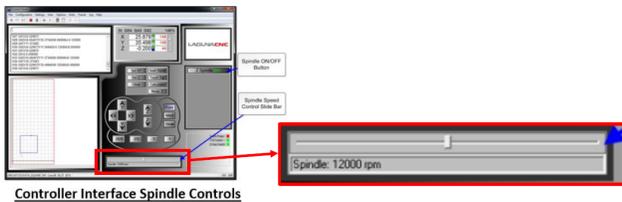
The bar can be adjusted by using the keyboards "+" and "-"key on the keypad of one's Keyboard.

Measurement Tool Length

Measuring tool length is a necessary step prior to cutting. It establishes the correct Z zero plane in the workspace coordinate system.

Method One:
Manual Method:

1) Mount the tool in the spindle.
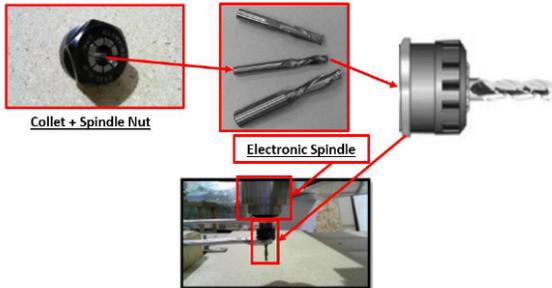
2.) Bring the tool over the material that will serve as the "Z0" plane for the application.

3.) Using the slow transit speed setting, lower the spindle down close to the table.

4.) Slide a Sheet of Paper under the tool.

Lower router bit on top of the paper until there is tension
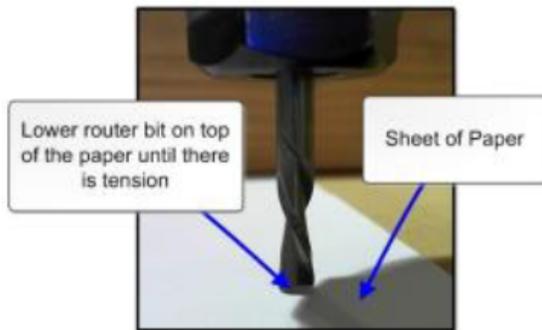
Sheet of Paper

5.) Lower the tool down using the 0.001 or 0.01 incremental transit setting until there is tension on the paper but enough clearance to slide the sheet of paper out.
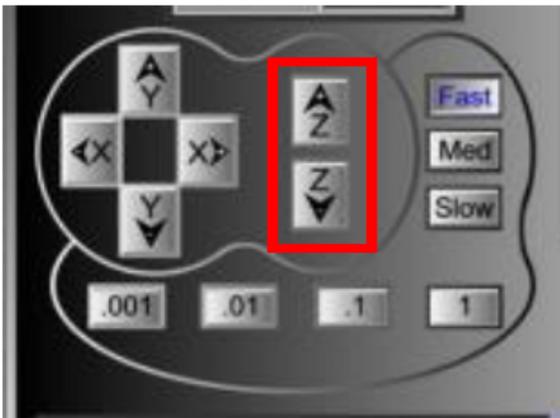
6.) **Press the "Set Z" Button** in the Control Interface.
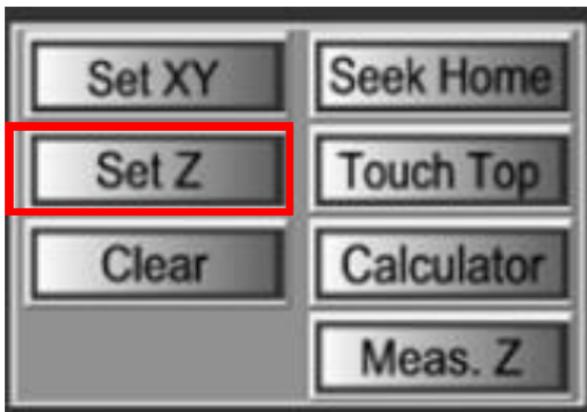
6.1) A **Green Box** will appear next to the **"Z" Coordinate** in the access window.

Method Two:
Automated Method

⚠ **WARNING!** Make sure the touch pad is functioning properly.

1) To verify proper functionality, position the router bit away from the table and then touch the tool touch-off pad to the router bit. At the same time look for a change from red to green on the TTO Switch indicator, as seen in the figures below.
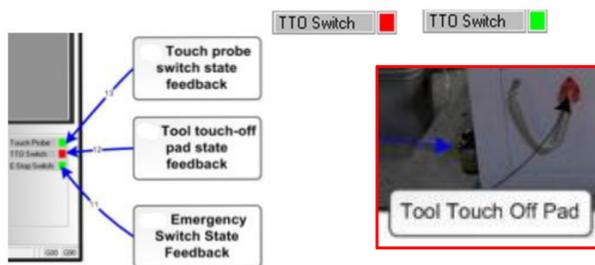
2.) Fit the router bit into the router head (See Basic Operations, Fitting the Router Bit).
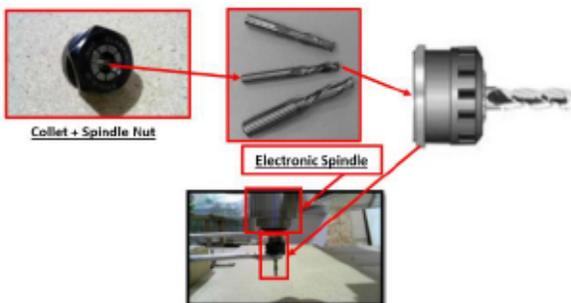
3.) Lower probe close to the top of the material and position the touch pad under the router bit so that the bit will contact the switch. Shown in figure below.
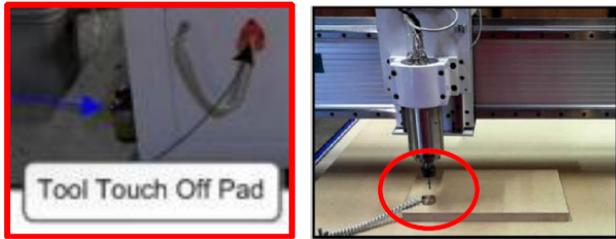
4.) In the control screen press the "Touch Top Button".
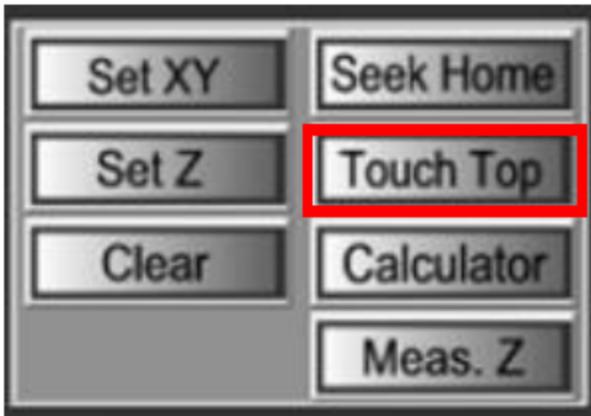
4.1.) The spindle will automatically lower down until it contacts the **"Tool Touch-Off Pad"**. After making contact the spindle will retract to the Z-Home Position.
In the photo below: **Executing a Tool Touch Off using the Touch Pad.**
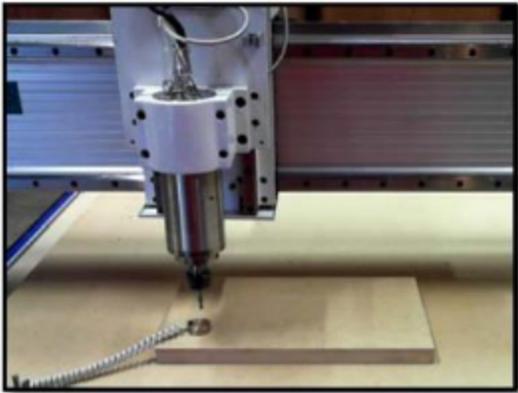
4.2.) Commands being executed will be visible in the message display window. A **Green Box** should be visible in the Axis window to the right of the **"Z" Coordinate**. This **Green Box** indicates that the local 0-Zero has been set for the Z-Axis.
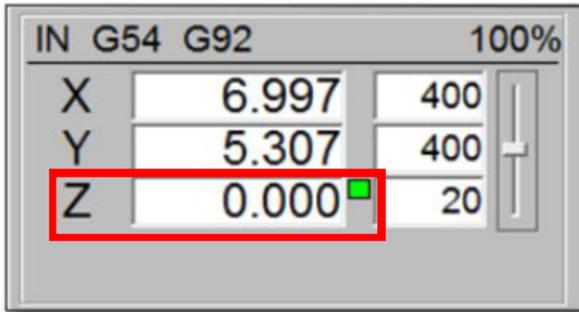
A Green Box in the axis
window indicates the local
zero has been set.

Setting Tool Length Offset

The **"Meas. Z Button"** executes a M37 command. The M37 command sets a Tool Length Offset. The Tool Length Offset establishes a Z-Plane that the Spindle will not cut past.
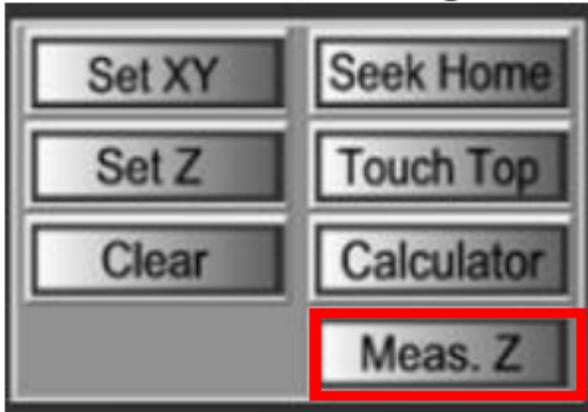
1.) Verify that the **"Tool Touch Off Pad"** is functioning properly,

2.) Place the **"Tool Touch Off Pad"** on top of the table or the material that will serve as the **Boundary Plane**.
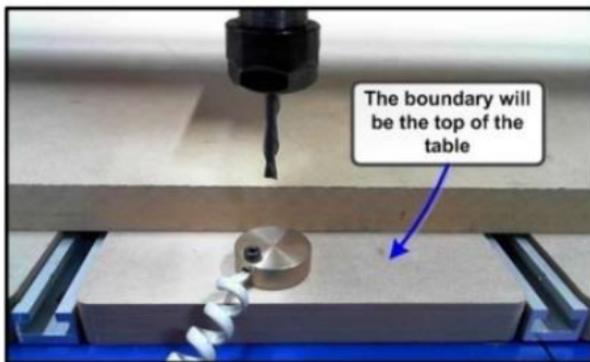
3.) Press the **"Meas. "Z" Button"** located in the WinCNC Controller Interface.

4.) There now should be a **"Blue Box"** located in the axis window next to the Z-Axis as shown to the Left. This indicates that a tool measure is in effect.
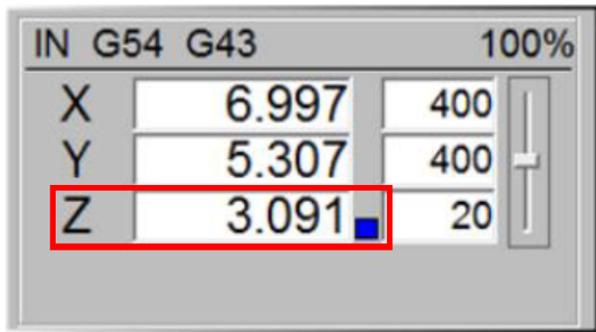
IN G54 G43		100%
X	6.997	400
Y	5.307	400
Z	3.091 ■	20

Vacuum (SV 200 400 Valve) Adjustment

NOTE: Torx bit TX30 X2 10mm wrenches

1) Remove the plastic valve cover.

2) Remove the Two Torx screws are found in between the plastic valve cover and bottom cover of the impeller.

3) The valve stem will be exposed. Crack the two nuts apart and adjust the "tension nut" to achieve desired vacuum level.

NOTICE   This should be done while the pump is running. A gauge should be used at the inlet port of the pump, to accurately adjust the valve.

Loosen the valve tension for less vacuum. Tighten the valve tension for more vacuum.

⚠ WARNING! Do not exceeded recommended vacuum levels. Call the Service department for exact values.

4) Once the desired vacuum level is achieved, Loctite the nuts and tighten them against each other.

5) Reassemble the valve cover.

Aspire to Cutting Tutorial

Description: This section is a tutorial on how to take a design made in computer-aided design (CAD) software, generate post processor code, upload to the controller, and begin cutting. It is assumed the user is familiar with their preferred CAD software, so the design process in CAD software is omitted. For this tutorial, a simple design will be created using Aspire 4.510 to demonstrate concepts.

1.) Create your design in CAD Software.

2.) Save Tool Path.

    2.1.) Before a tool path can be saved, a post processor must be selected. The correct post processors must be added manually to the CAD software being used. This will be demonstrated next for Aspire 4.5.

Note: For 3D applications, the winCNC_CUT3D.pp file must be used to generate a tool path.

    2.1.1.) Go to the winCNC website at http://wincnc.net/ .

    2.1.2.) Navigate through the website and download the corresponding post processor file.

Navigation through the winCNC.net website to locate correct post processor file.
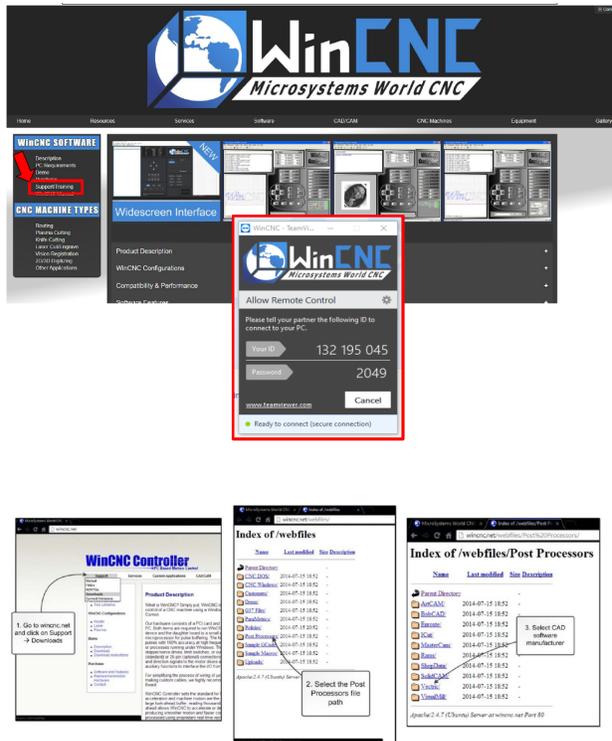
2.1.4) Move the post processor file to the folder where all the other post processor files are located for the CAD software.

Downloading a post processor file

2.1.5) Save tool path.
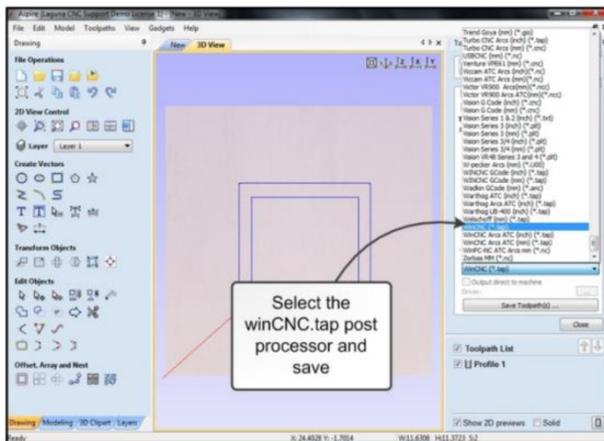
**Saving a tool path in Aspire using the
winCNC.tap post processor.**

3.) Load program into the WinCNC Controller.

    3.1.) Go to File → Open and find the post processor file. The display will be like the figure to the right.

4.) Mount Cutting Material to table.

5.) Mount Tool into Spindle.

6.) Home the Machine. **"Click the "SEEK HOME" Button"** located in the display screen. The machine will automatically seek home and zero out the machine coordinates displayed in the Axis Window.
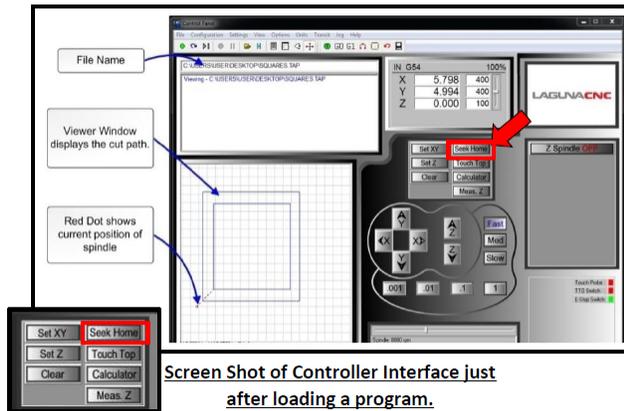
Screen Shot of Controller Interface just
after loading a program.

7.) **Optional –** Use the **"Meas. Z Button"** on the screen display to set a maximum depth that the tool will not exceed. This will prevent the tool from going below a set depth and prevent any damage to the machine and tools.

7.1.) Place tool touch off pad on top of spoil board or on top of the table. This location will vary depending on the application. For this tutorial, a spoil board is not being used, so the tool touch pad is placed on the tabletop.
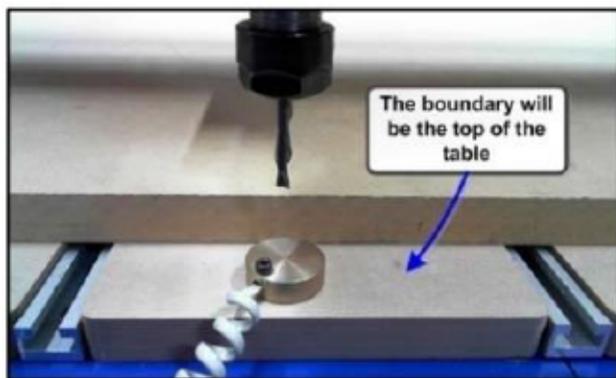
7.2.) Verify Touch Pad Operation (See Setting Tool Length Section).
Press the Meas. Z Button (**Blue Box**).
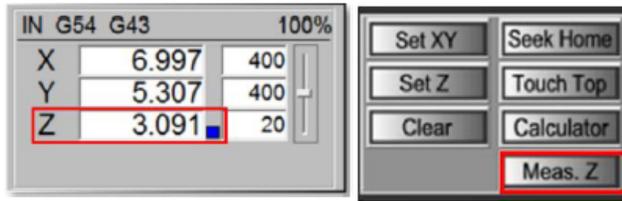
8.) Measure Tool length (See "Measuring Tool length"). Begin with Step #2.) Slide 60.

9.) Set X, Y work Coordinates.

    11.1.) Start by relocating your spindle to the XY datum position that was declared in the CAD software.

    11.2.) Press the "Set XY" Button. Green Boxes should appear in the axis window as shown.
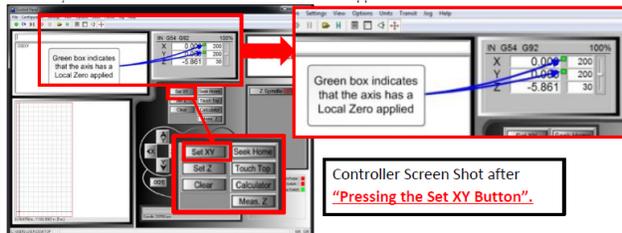
10.) Make sure spindle cooling is on, whether it is liquid cooled or fan cooled.

11.) **"Press the Start Button"** to begin executing.
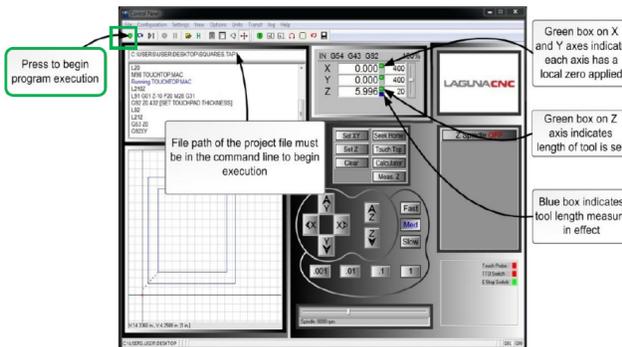
    11.1. **"Emergency Stop" options** include the **"Emergency Stop Switch"**, the **"ESC" Button"** on the keyboard or the space bar.

3D Scanning Guide

The WinCNC controller supports 3D digitizing scanning. This feature uses a touch probe to sample points in 3D space. The 3D digitizing scan outputs a file in STL format. The following section is a walkthrough on how to perform a 3D scan.

1). Mount the (TP-1000 Assembly) Touch Probe into the Spindle.
    1.1.) Using a suitable Collect (3/8") and Clamping Nut (ER20-A-W).
    1.2.) Fasten the Touch Probe's Connector into the connector on the Gantry.
    1.3.) Bundle the Touch Probe's Wire and secure it into the Clip located on the Spindle Mounting Plate. Leave extra slack between the Clip and Connector to allow for travel along the Z-Axis.
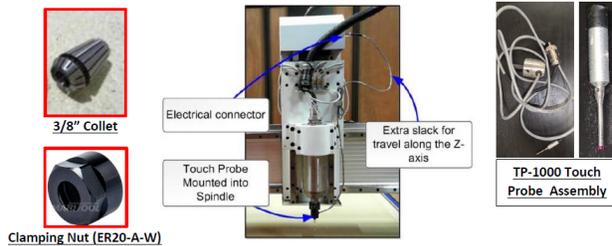
2.) Verify that the touch probe is working properly before proceeding to prevent damage to the probe.
    2.1.) Manually activate the touch probe and verify that the touch probe indicator in your WinCNC controller changes state from **Red** to **Green**, as seen to the right
    2.2.) If touch probe indicator does not change state, do not proceed until fault is corrected.
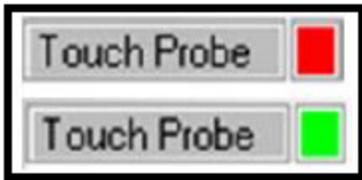
3.) Fasten the part to be scanned to the Spoil Board.
    3.1.) This can be done by using **double-sided tape or vacuum** if your machine is equipped with a vacuum table.
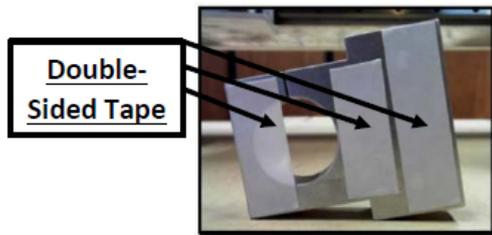
4.) Enter the "**L82Z0 Command"** in the Command window.
    4.1.) Position the Touch Probe such that it can travel in the Negative Z-Direction and Touch Off on what will be the "Z" 0-Zero Plane. Normally this is the material the object is mounted to.
    4.2.) The "**L82Z0 Command"** measures the surface with the probe and sets the measured point to be the "Z" 0-Zero Plane.

Controller Screen Shot after execution of a "L82Z0 Command".

Note: This step is not required to execute a 3D-Scan, but it will correctly establish the "Z" 0-Zero Plane for rendering in CAD software. Otherwise, the "Z" 0-Zero Plane will need to be adjusted in CAD software.

5.) Manually position the probe to the corner that will serve as the X0,Y0 work coordinates, in other words the negative-most corner. The probe will travel in the positive X and Y direction.

Positive Y Direction

Positive X Direction
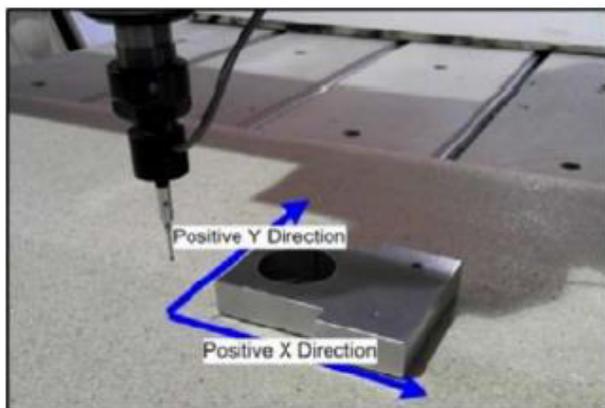
Starting location for a 3D Scan.

6.) Set the local zero for the X and Y-Axis by **Pressing the "Set XY" Button**.

Note: Step 6 is not necessary to execute a 3D-Scan. If the probe needs to be repositioned at the same starting location, setting the local zero for X and Y will allow for accurate repositioning.

7.) Open the 3D-Scan Dialog Box.
   7.1.) Go to File→Digitize→Automatic
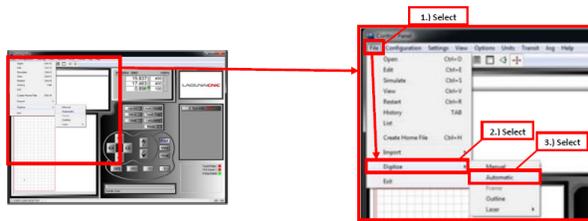
8.) There are multiple settings that must be chosen before proceeding.
   8.1.) **Probe Type:** The touch supplied will be the Up/Down/Side option.
   8.2.) **Axis:** Choose the axis for width, length and retract. The default settings are conventional.
   8.3.) **Area:** The area dimensions that are to be scanned.
     8.3.1.) **Retract:** This will need to be set if an up/down probe is being used. The retract height must be set higher than the highest point on the scan area.
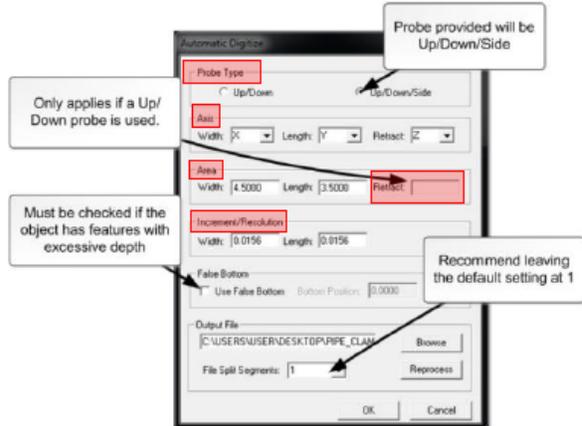
   8.4.) **Increment/Resolution:** The distance between sample points. The smaller the increment (Ex. 1/64") the more detail the scan will output but the scan time will increase.
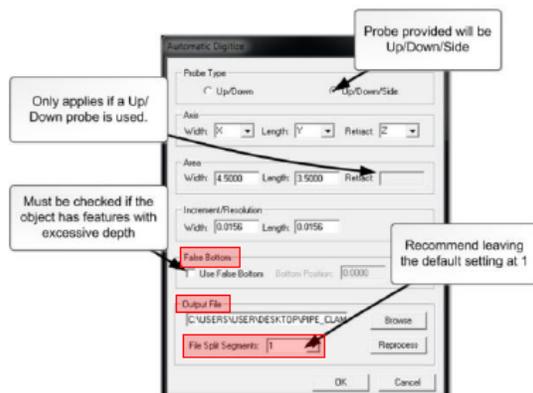
   8.5.) **False Bottom:** This needs to be set if the part to be scanned has holes deeper than the length of the touch probe. Not setting a false bottom on a part with deep features may result in damage to the touch probe and the machine.
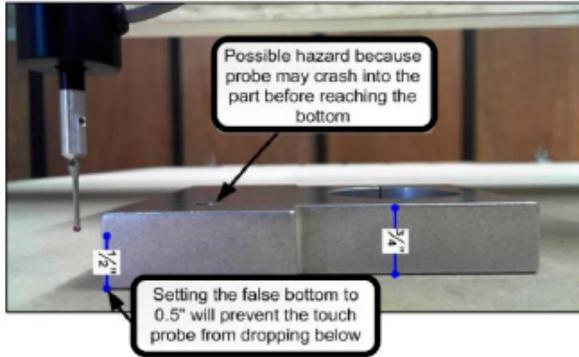
False Bottom Description.

8.6.) **Output File:** A file destination must be declared. This is the location the output file for the scan will be saved.
Note: If a scan is aborted in the middle of the scanning process, a partial file will still be generated.
8.7.) **File Split Segments: Do not Change.** Default needs to be "**1**".

9.) After all settings have been set, **the scanning process will begin once the "OK" Button in the Automatic Digitize dialog box is pressed.**
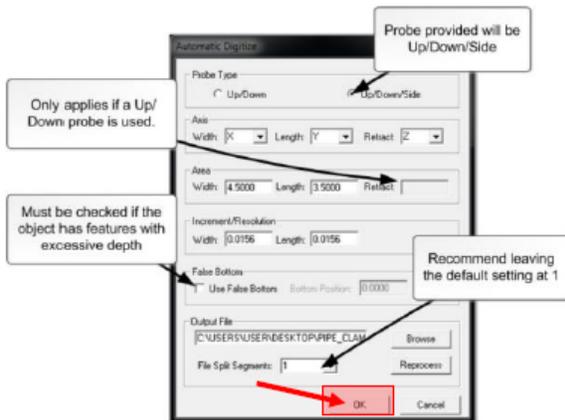
Note: Pressing the "Esc" Button Key on your keyboard will abort any machine operation. Pressing the Spacebar will pause program execution and pressing "Enter" will resume the machine operation.

10.) Upon completion of the 3D-Scan, a notification will be present in the command console in the WinCNC controller interface, as in the image below.
11.) The probe will move in the **Positive "Z" Direction** upon completion.

Edge Outling/Scanning Guide

1.) Mount touch probe into spindle. See Step 1 of the 3D-scanning tutorial.

2.) Verify that the touch probe is working properly. See Step 2 of the 3D-scanning tutorial.

3.) Fasten part to be scanned onto the table. See Step 3 of the 3D-scanning tutorial.

4.) Select a starting location.

4.1.) The touch probe needs to be between ¼" to 1" away. These values can vary depending on the "First Forward Move" setting located in the outline scan window.

4.2.) Lower the probe on the z-axis so that the probe stylus will contact the part when scanning.

4.3.) Unique to the edge scan is that there is no restriction on the starting location.
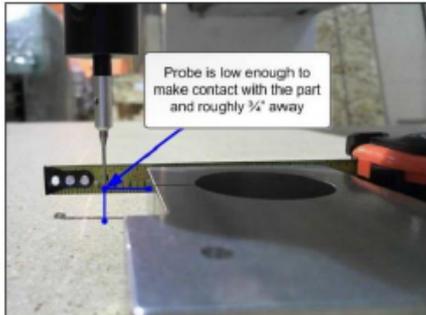
Starting location for an Edge Scan.

Note: Depending on the starting location, the X and Y scan directions will be affected. These settings can be adjusted in the outline scan dialog box.

5.) Open the Outline Scan Dialog Box.
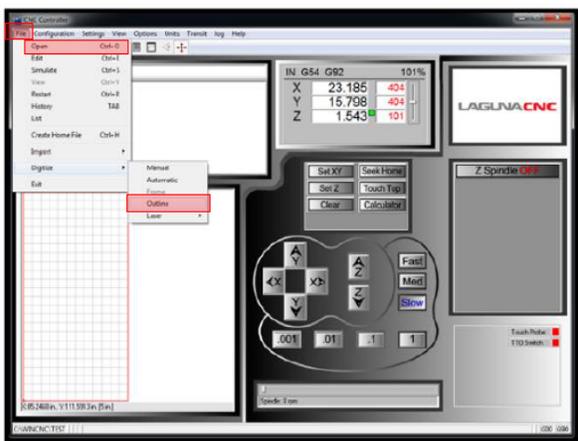
Navigation to the Edge Scan Window.
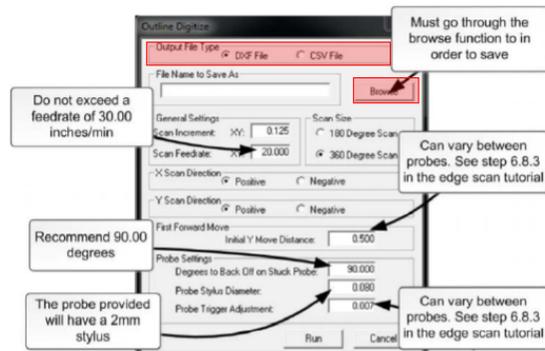
Edge Scan Window Overview.

6.) Adjust the necessary settings in the dialog box-

6.1.) Select the **Output File Type. DXF or CSV file.** DXF File format is conventional across CAD software.

6.2.) Must **"Click Browse"** to select a destination for the output file. **File will not save otherwise.**

6.3.) **General Settings:**

6.3.1.) **Scan Increment**: The distance between each sample point. The smaller the scan increment the more accurate the scan will be.

6.3.2.) Scan Feed-Rate:
6.3.2.1.) It is recommended that the feed-rate does not exceed 30 in/min.
6.3.2.2.) A feed-rate greater than 30 in/min will negatively affect the accuracy of the scan.
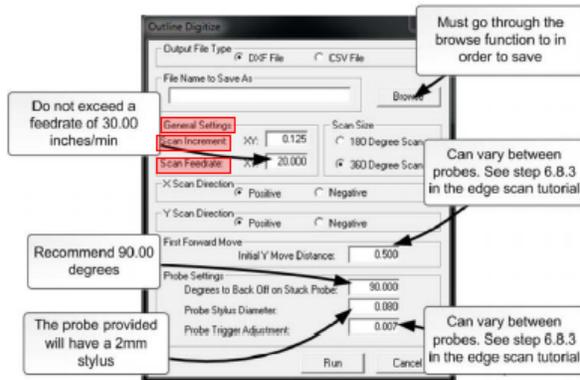
6.4.) Scan Size:
6.4.1.) **360 degrees:** Scans the entire perimeter of a part.
6.4.2.) **180 degrees:** Only scans half of the part.

6.7.) First Forward Move:

6.7.1.) **Initial Y move Distance:** How far the probe initially travels in the Y-Direction.
6.7.2.) If the probe is not triggered within the distance specified in the initial Y move distance the program will abort. This is a safety mechanism to prevent damage to the probe.
6.7.3.) It is recommended that a short distance be used relative to the starting position of the probe.
6.7.4.) If the probe is 1 inch away from the material, a corresponding setting would be slightly over 1 inch.

6.8.) Probe Settings:
6.8.1.) **Degrees to back off on Stuck Probe:**
6.8.1.1) Recommended setting is 90 degrees.
6.8.2.) **Probe Stylus Diameter:** The diameter of the tip of the touch probe.
6.8.2.1.) If using touch probe provided, stylus diameter is 2mm. This size will vary depending on the manufacturer of the touch probe.
6.8.2.2.) Note if using inches as the primary unit of measure the diameter must be converted into inches. 2mm is approximately 0.0787".
6.8.3.) **Probe Trigger Adjustment:** The distance the probe's internal switch must travel before the switch is triggered.
6.8.3.1.) How to calculate Probe trigger adjustment.
6.8.3.2.) Position the probe as close as possible without contacting the material.
6.8.3.3.) Using the 0.001 incremental setting, jog the probe until it is just touching the material.
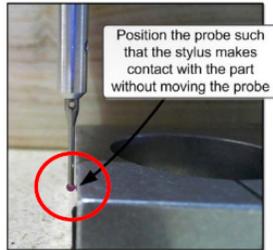
6.8.3.3.) Step 2-Two in measuring probe trigger adjustment.

6.8.3.4.) Execute G92X0 command to set a local X0 coordinate. This step is not required but it will help to track the number of moves until the probe's switch is triggered. This will be used to monitor distance traveled in the x- direction until the probe is triggered.

6.8.3.5.) Move in the positive (+) X-Direction 0.001" inches at a time until the touch probe indicator is triggered, at which point the probe trigger adjustment will be the X-Coordinate displayed in the Axis window. In this example, in the figure below the figure is 0.004".

7.) **Press Run:**
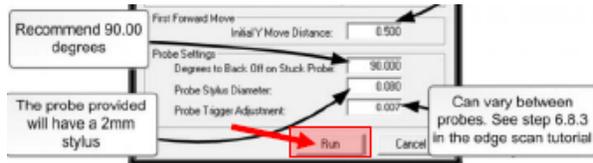
Programming & Operation of the Electric TCM-3 Tangential Cutting (Knife) Module

The TCM-3 is a processing unit for CNC-machines that is designed to cut various materials such as foils and flocking materials.

Below is a work flow diagram that captures the steps required to execute a program. The following sections outline each step in more detail.
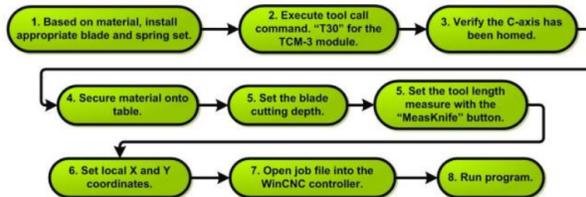
Figure: Tangential Knife (TCM-3) Workflow Diagram.

TCM Blade and Spring Selection

SPRING OVERVIEW:
The type of the spring set determines the coarse adjustment of the pressing force. The fine adjustment will be done by the adjustment wheel that changes the pre-load of the springs. If the spring set does not have enough pressing force the blade may be forced up away from the material.

Table- Available Spring Sets

Suggested Spring Sets:

Spring Set ID	Pressing Force
S5	5N/510g
S10	10N/1.020kg
S18	18N/1.836kg
S28	28N/2.856kg

N= Newton (N)= The Newton (Symbol: N) is the International System of Units (SI) derived unit of force. A newton is defined as the force which gives a mass of 1 kilogram an acceleration of 1 metre per second, per second, 1 kg·m/s 2.

1 Newton =0.102 kg·m/s 2
1kg=1000g

Figure: Example of available Springs.

Suggested Spring Sets	
Spring Set ID	Pressing Force
S18	18N/1.836kg
S28	28N/2.856kg

Changing Springs and Fine Adjustment:

1.) Use the Adjustment Wheel to release Spring Tension.

1a.)Remove the 4-Four Torx (T10) Screws and Front Cover.
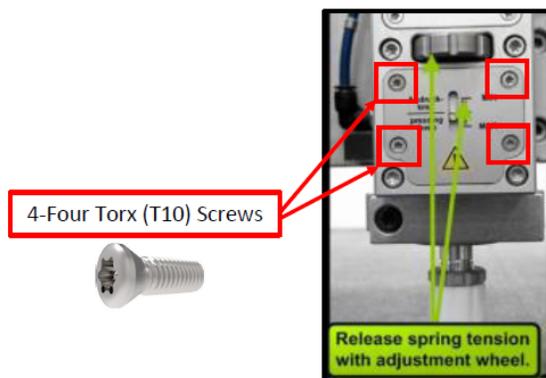
2.) Using a pick tool, remove and replace springs.

Pick Tool Set

Replace springs.
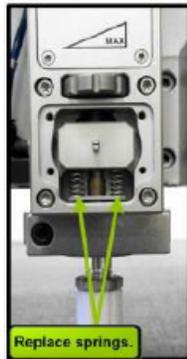
3.) Reinstall Front Cover.

4.) Use the fine adjustment wheel set desired pressing force.

Blades

When selecting a blade, it is important to consider material thickness and tensile strength. Different carbide metal blades with a shank diameter of 6mm are available for various cutting applications. The table below presents a selection of the most used blades.

Table of Carbide Blades:

CARBIDE BLADE SELELCTION SMARTSHOP MT				
Blade ID	**Cutting Edge**	**Length of Cutting Edge**	**Total Length**	**Typical Applications**
E12	2	12 mm	25 mm	Universal blade for various materials such as cardboard, gasket material, foam rubber, cork, useable on both sides.
E18	1	13.5 mm	25 mm	Universal blade for various materials such as cardboard, gasket material, foam rubber, cork, one-sided blade for fine lines.
E25	1	25 mm	39 mm	Universal blade for various materials such as cardboard, gasket material, foam rubber, cork, one-sided blade for fine lines.
E28	1	30 mm	45 mm	Universal blade for various materials such as cardboard, gasket material, foam rubber, cork, one-sided blade for fine lines.
E30	1	2.5 mm	25 mm	Special blade for TCM module; Wedge blade for normal foils and writings.
E50	1	3.5 mm	25 mm	Special blade for TCM module; Wedge blade for flock textile foils, felt, cardboard.
E70	1	8 mm	25 mm	Special blade for TCM module; Wedge blade for textile foils, felt, cardboard, rubber.
E85	1	50 mm	65 mm	Special blade for EOT module; e.g. for soft polyurethane foam panels.
E87	1	70 mm	83 mm	Special blade for EOT module; e.g. for soft polyurethane foam panels.
E92	1	120 mm	133 mm	Special blade for EOT module; e.g. for soft polyurethane foam panels.

1.) Unscrew & Remove the Gliding Element.

2.) Insert blade with the Weldon Surface Facing the Set Screw.
3.) Torque down the 2.5mm Set Screw against the Weldon Surface.
4.) Reinstall the Gliding Element.
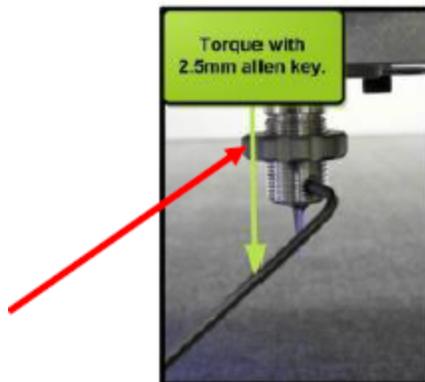
Sample Project (Blade Selection):

Either the E12 or E50 blades will be sufficient to cut 0.020" styrene. These blades were chosen because of the thickness of the blade's spine as well as a more robust design. Blades like the E70 and E18 will most be damaged due to a thinner spine near the tip of the blade.

Blade Examples:

CARBIDE BLADE SELELCTION SMARTSHOP MT				
Blade ID	**Cutting Edge**	**Length of Cutting Edge**	**Total Length**	**Typical Applications**
E12	2	12 mm	25 mm	Universal blade for various materials such as cardboard, gasket material, foam rubber, cork, useable on both sides.
E50	1	3.5 mm	25 mm	Special blade for TCM module; Wedge blade for flock textile foils, felt, cardboard.

Homing the C-Axis

The C-Axis refers to any axis that rotates around the Z-axis, such as the TCM-3 and EOT-2 Modules.
It is necessary to home the C-axis prior to using the modules. This will ensure blade direction is accurate.

If the C-Axis is not homed, the blade may be in the incorrect position during a cut, most likely leading to a broken blade. The C-Axis automatically homes when a tool number is called. Depending on the last state the machine was in, one of the two modules will home during the initial homing sequence.

Figure: Position of Blade after successfully homing the C-Axis.

Adjusting the Cutting Depth

Cutting depth is the length of blade protruding past the nylon gliding element. The gliding element also aids in holding down the material as the blade is dragged through. The TCM-3 module can also be used without the gliding element. In this case, only the tool length offset needs to be set. Skip to the following section.
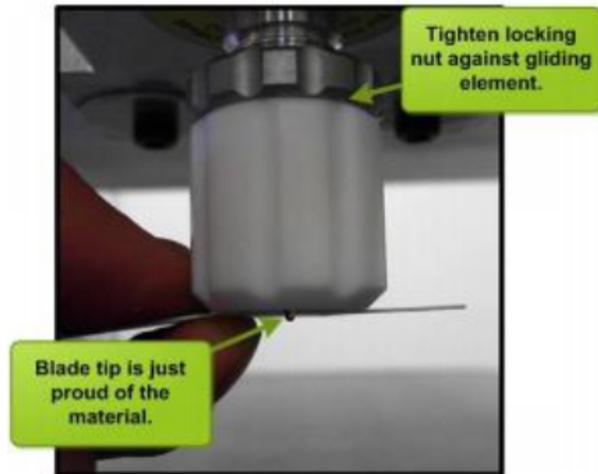
Figure: Setting the Cutting Depth.

1.) Take a small sample of the material and hold it up behind the blade.
2.) Loosen the locking nut and adjust the gliding element until the knife tip is just protruding past the material.
   2a.) Note: This tutorial assumes that the material is to be cut all the way through.
3.) Tighten the locking nut and gliding element against each other without affecting the cutting depth.
4.) The cutting depth can be adjusted precisely if the gliding element is mounted. The cutting depth increases 1.0mm per revolution. In practice it´s possible to adjust the depth in increments of 0.05 - 0.1mm.

Setting Tool Length Offset

<u>**TCM-3 SETTING KNIFE LENGTH (TOOL LENGTH OFFSET):**</u>
The purpose of setting a tool length offset is to store a point in the Z-axis in which the knife will lower to prior to cutting. The **"Buttons "MeasKnife" and "MeasSaw"** are used to store this value. When a tool length offset is active there will be a blue next to the corresponding axis. G43 mode will also be active.

<u>**SETTING TOOL LENGTH OFFSET:**</u>
1.) Select the TCM-3 tool by executing a tool call command. For the TCM-3 module the tool command is T30. The order of events that follow a tool call are outlined below.
    1a.) The TCM-3 module will extend, and all other modules will retract
    1b.) Blue boxes will appear next to the X, Y, and Z axes.
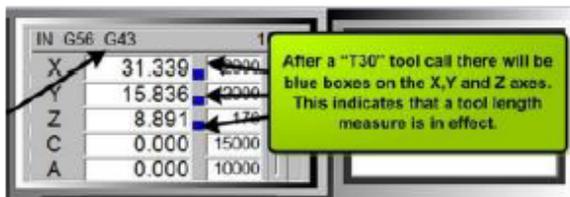
    1c.) The head will shift along the X-axis the corresponding offset.
2.) Using the manual jog controls in the WinCNC interface slowly lower the knife down into the material.
3.) Once the knife has been lowered to the appropriate cutting height. **"Click "MeasKnife"**.

Figure: Controller State after a T30 command.
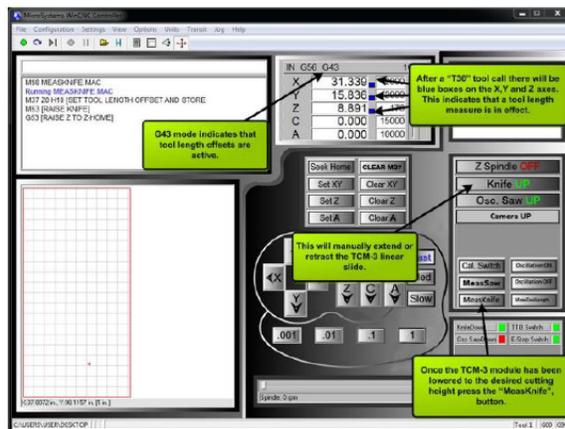
Figure: Drag knife extended after T30 command. process.

Figure: Demonstrating tool length measure
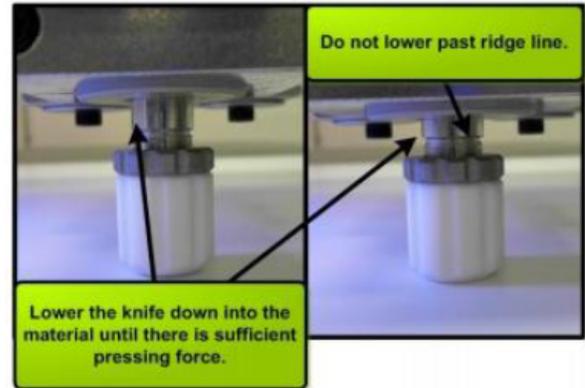
4.) After pressing the **"MeasKnife" Button**, the linear slide will retract, and the tool head will return to the "Z"-home position.

TCM-3 Setting Work Coordinates

1.) Manually Jog the Tool Head, until the Knife Point is at the desired XY Origin Point.

Positioning the cutting module to set local coordinates.

2.) **"Press the "Set XY" Button".** Green Boxes will appear next to the X and Y-Axis.
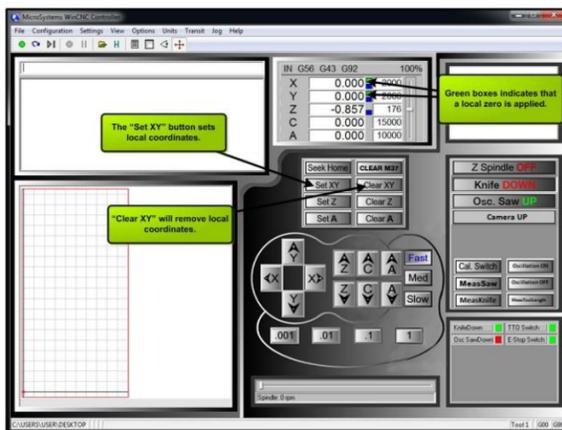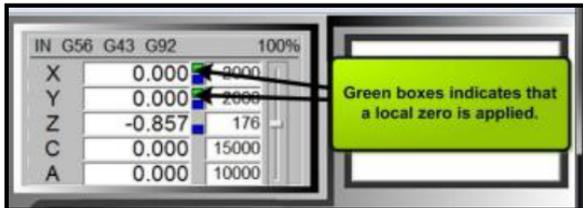
Figure-Controller overview of setting and removing local coordinates.

Once the previously discussed steps are completed G-Code can then be executed. Import a program file using the **"Drop down menu File --> Open"**. Select the desired file.
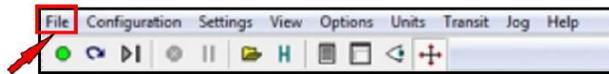
The program can be previewed in the viewer window by **"Clicking the View Button on the Tool Bar"**. Execute the program by clicking on the green start button or pressing "Enter" on the keyboard. The file path must be in the command line for the program to execute.
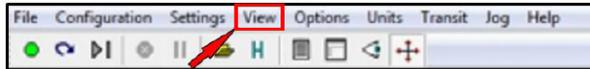
Preliminary Checks

The cutting blades are tungsten carbide, which means they stay sharper longer. But carbide is also brittle and can break if stress is applied in the wrong place. That is why doing a test cut in the air is an important optional preliminary step to avoid breaking a blade.

Things to look for are:
• Positions in the cut path where the blade makes sharp turns without lifting.
• Cutting edge of the blade not facing the correct direction.
• Cutting speed.

To Execute a Cut in the Air:
• Partially lower the knife so that there is ample clearance for the knife to safely execute the program without crashing into any obstructions.
• Press the "MeasKnife" or "MeasSaw".
• Load program into **Wincnc. File --> Open**, select program. **"Click on the "View" icon in the Tool Bar"**. This will display the cut path in the viewer window.
• Execute the program.

Post Processor

POST PROCESSOR:

The Post Processor will require commands unique to the WinCNC Control System. Post processors can be found at the wincnc.net website. See the **"CNC swift series with WinCNC Manual"** for more details on locating and implementing new post processors. If using the program **Aspire made by Vectric**, the post processor is **WinCNCKnife.pp**.

DRAWING TIPS:

When using the tangential cutting modules, it is important to be aware of the angles and arcs used in the graphics. In the graphic below there are both acute angles as well as curves.
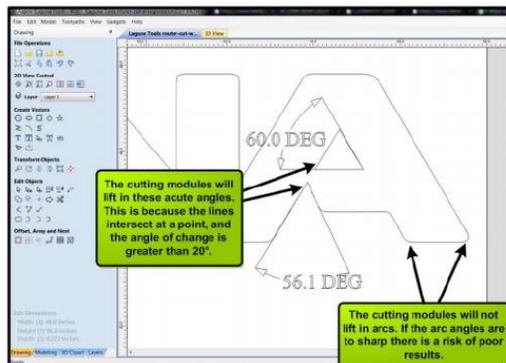
Figure: Example graphic highlighting cutting module behavior.

Advanced Settings

ADJUSTING LIFT ANGLE:
The default lift angle is set at 20°. This means that the tangential module will lift away from the material before rotating in a location where the angle of change is greater than 20°.
• Increasing this value will decrease the number of lifts but increase the risk of damaging a blade.
• Decreasing this value will increase the number of lifts but reduce the risk of blade or material damage.
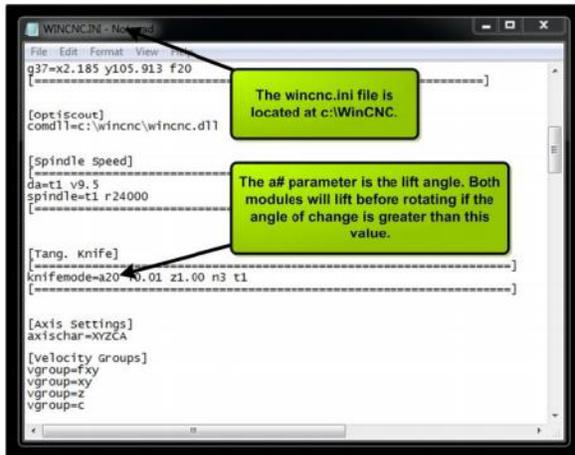
Figure: Location of the lift angle parameter.

<u>**Laguna Spindle Options:**</u>

- **3 HP Liquid Cooled- This Spindle is to be used!**

Electro-Spindle Assembly: The Electro-Spindle is moved along the gantry by a precision Helical Rack and-Pinion Ball Screw
 system that is controlled by the machine controller.

Programming & Operation of Laguna Vision System- Modular Optical Recognition System (Optiscout-Vision System) for multi-functional flatbed machines for industrial usage in exact-outline digital finishing processes.

OPTISCOUT OVERVIEW AND WORKFLOW-The purpose of Optiscout is to compensate for deviations that arise

during the preparation process. Below is a workflow diagram that captures the steps required to utilize the Optiscout system. The following sections outline each step in more detail.
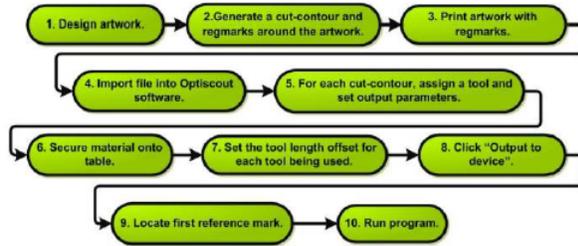
Figure-Vision System Workflow Diagram

The suggested Design Software for use with Optiscout are the following:

1) Adobe Illustrator.

2) CorelDRAW.

3) Optiscout Design.

For this walkthrough, Adobe Illustrator will be used to demonstrate the necessary steps to using your vision system. Keep in mind that the design software will vary, but the concepts remain the same.

ARTWORK LAYERS:
The first step is to create the Artwork Layer. This layer will contain all the graphics that will be printed but not to be cut or routed. As shown in the figure below.

Figure: Artwork designed in Adobe Illustrator.

GENERATING CUT-CONTOUR AND REGISTRATION MARKS:

CUT PATH LAYER:
As the name suggest the cut path is the layer that the CNC machine will trace over with either a tangential knife or router bit.
1) There can be as many cut path layers as the project demands. Each cut path layer can then be assigned a specific tool.

2) For a quick transition into Optiscout software the layers need specific names.

REGMARK LAYER:
Registration marks, video marks, regmark's, jog marks are all synonymous terms. These terms refer to reference markers that the camera will locate and use to rotate and scale the output objects. The criteria for the registration's marks are the following:
1) The layer needs to be called "regmarks" case sensitive.

2) At least 3 marks should be present.

3) The size of the marks needs to be 0.25 or 0.197 inches.

4) Circles should be black when on white substrate. A contrasting color on colored substrate.

5) Marks need to lie within attainable range of the camera.

6) The placement of remarks should result in a large triangle which covers the object to be compensated. See figure below for an example.

7) Use more regmarks for more accuracy.
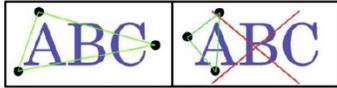
Figure: (Left) Good Coverage, (Right) Poor Coverage of the Graphic's Area.
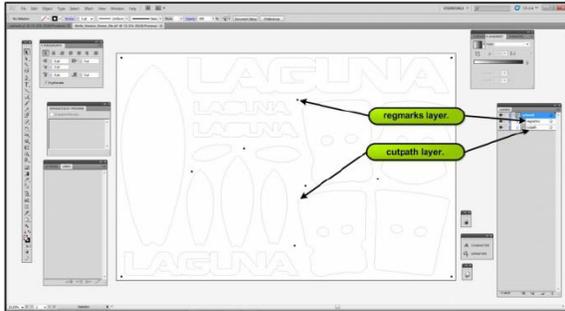
Figure: Example Cut path and Regmark layers, Artwork Layer Turned Off.

PRINT ARTWORK:

Export the Artwork and Regmark layer to your RIP Software for printing.

Note: Make sure that the dimensions of the file will not be changed in the RIP software.

IMPORT FILE INTO OPTISCOUT:

1) Start Optiscout.

2) Click on the "File import" button located in the macros tab.

3) Select the Illustrator file.
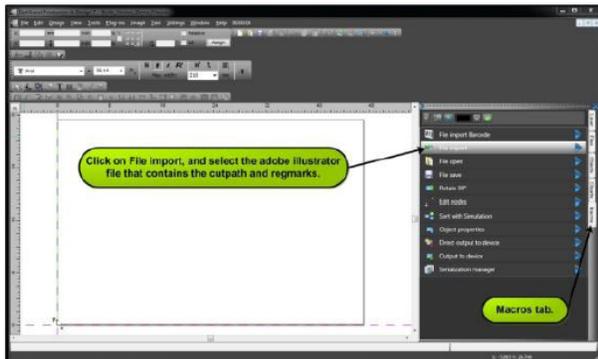
Figure: Screen Shot when Optiscout is first opened.

SELECTING A TOOL AND MATERIAL

Figure: For each Cut path, a Tool and Material must be selected.

EDITING OUTPUT PARAMETERS:

To edit a tools output parameters such as feed rate and cutting depth, "Click" on the Settings Button shown in figure 8 below:
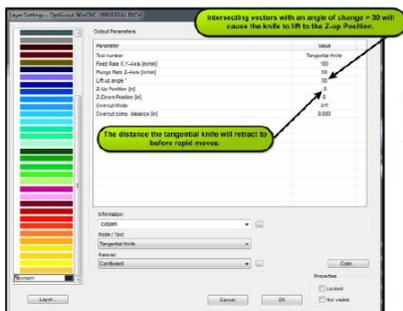
Figure: Output Parameters Dialog Box for the Tangential Knife.

Figure: Highlighting the Output Settings Button.

SECURE MATERIAL ONTO TABLE:

This step is dependent on the user's application and techniques. For this demonstration, the tangential cutting module is being used so cutting mat will be placed down onto the table and a sheet of styrene will placed on top.

SETTING THE TOOL LENGTH OFFSET:

ROUTER:

A tool length offset must be stored prior to executing output code for the router. When the router is called, a T1 command is executed. This tool call will activate the last stored tool length offset.

1) Secure a router bit into the spindle.

2) Lower the bit to where Z0 will be. Normally this will be the top of the material, but there are exceptions.

3) Enter the command M37Z0H1 or press the "MeasureToolLength" Button. This will store the tool length offset into memory. See Figure Below.

Figure: Overview of Controller Interface and storing Tool Length Offsets.
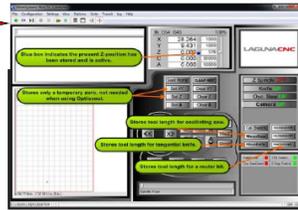
Tangential Module:

As with the router a tool length offset must be set prior to executing code for a tangential module. This subsection applies to the oscillating knife and crease wheel as well.

1) Verify the linear slide has been extended.

2) Lower the Z-Axis until the desired cutting height is reached.

3) Press the respective tool measure button. See the figure below.

Figure: Overview of Controller Interface and storing Tool Length Offsets.
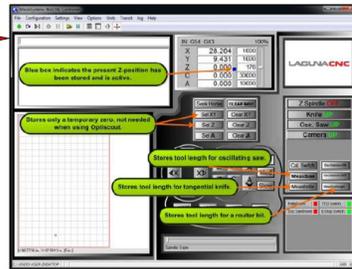
OUTPUTTING CODE:

Press the output to device button located in the macros tab.

Adjust any relative parameters such as the park position.
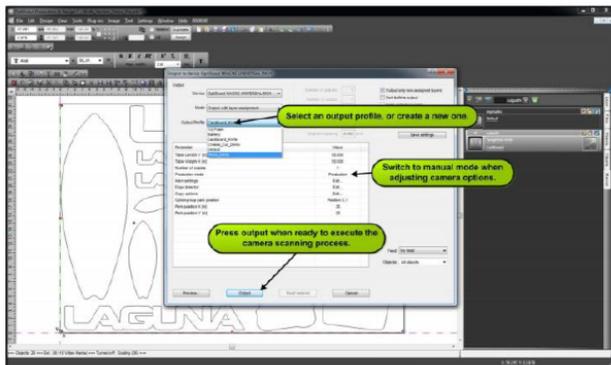
Figure: Screen Shot highlighting the Output Parameter Dialog Box.

Once the output button is pressed the camera will lower to the stored camera height and rapid to a starting location. It will then idle and wait for the user to manually jog the camera over the first marker. See next section.

LOCATE FIRST REFERENCE MARK:

Using the keyboard hot keys, manually jog the camera over the first reference mark. Once visible in the Optiscout Software, **"Press the Target Button".** This will start the scanning process. Each dot will be visited by the camera unless redundancy is detected. Once complete, Optiscout will output code to the controller, which will begin executing.
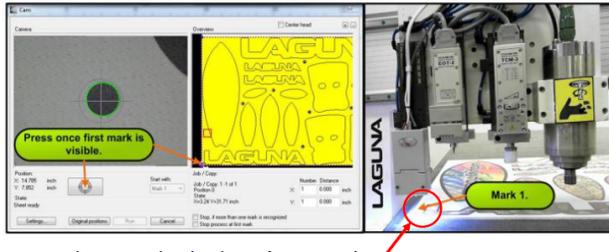
Figure: Locating the First Reference Mark.

Troubleshooting

Issue	Potential Solution
MOTOR TRIES TO START BUT WILL NOT TURN	1.) With the power disconnected from the machine. Try to turn the spindle by hand. If the spindle will not turn, check the reason for the jamming. 2.) Motor faulty. Replace the motor. 3.) If the Spindle runs without coolant. Replace the motor.
MACHINE WILL NOT STOP	This is a very rare occurrence as the machine is designed to be fail-safe. If it should occur and you cannot find and fix the source of the problem, seek professional assistance. The machine must be disconnected from the power and never run until the fault has been rectified. 1.) Internal breaker faulty. Replace the breaker.
MOTOR TRIES TO START BUT WILL NOT TURN	1.) With the power disconnected from the machine. Try to turn the spindle by hand. If the spindle will not turn, check the reason for the jamming. 2.) Motor faulty. Replace the motor. 3.) If the Spindle runs without coolant. Replace the motor.
MOTOR OVERHEATS	The motor is designed to run hot, but should it overheat it has an internal thermal overload protector that will shut it down until the motor has cooled and then it will reset automatically. If the motor overheats, wait until it has cooled and restart. If the motor shuts down consistently, check for the reason. Typical reasons are dull cutting tools, no water in the coolant tank, blockage in the coolant pipe and excessive ambient temperature.
SQUEAKING NOISE	1.) Check the Bearings.
SPINDLE SLOWS DOWN DURING A CUT	1.) Dull cutting tools. Replace the tool or have it Re-sharpened. 2.) Feeding the wood too fast. Slow Down the Feed-Rate.
MACHINE VIBRATES	1.) Machine not level on the floor. Re-level the machine ensuring that it

Issue	Potential Solution
	has no movement.

Maintenance

MAINTENANCE RECOMMENDATIONS:

Daily Checks:
• Clean the machine and lubricate unpainted surfaces with a Teflon lubricant. Wipe off any excess and buff with a dry polishing cloth. This will reduce the likelihood of rust forming.
• Check cutter teeth for chips and dullness.
• Generally, inspect the machine for damage and loose or warn parts.
• Collets and spindle collet hole must be cleaned regularly. Ensure that the slots in the collets are free of sawdust as sawdust builds up and will stop the collet compressing. If the collet or spindle hole is not clean, the router bit may not run true, and this will affect the performance of your machine.

Weekly Checks:
• Clean the Cutters.
• Check cutter teeth for chips and dullness.
• Generally, inspect the machine for damage and loose or warn parts.
• Check the dust extraction for blockages and any large bits that could cause blockages.

Note: If ones Business is a Quality Management System (QMS) Certified Facility, suggest incorporating the Maintenance Recommendations and Trouble Shooting Methods into the Facilities QMS Preventive Maintenance System or Machine Maintenance System.

Maintenance Tips
• Keep the guide rails clean. After cutting, debris will build up on the rails and collect near the bearings.
• Periodically lubricate guide rails using the oiler. One pump is sufficient for one lubrication.

• Periodically do maintenance on the PC tower by removing dust buildup. This will prevent overheating issues and possible malfunctioning.
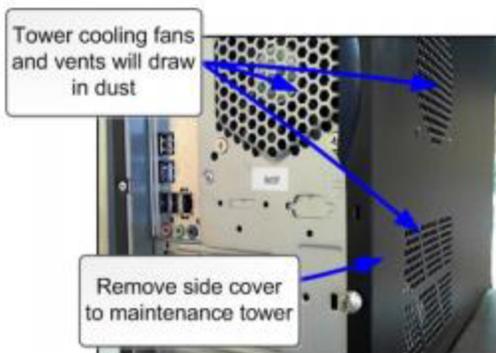
Command Reference

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. Example: X1Y1 is equivalent to G0 X1Y1 if mode is 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.
G1	Feed Move - G1 L# 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 the mode is G1. L# is used to allow setting laser power in vector cutting (1-100.)
G2	Clockwise Arc - G2 L# X# Y# I# J# Z# W# K# R#
	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. Any additional axis spec that is not part of the arc itself will move that axis simultaneously throughout the arc. L# is used to allow setting laser power in vector cutting (1-100). R# is the radius, if R# is used IJK cannot be used.
G3	Counter Clockwise Arc - G3 L# X# Y# I# J# Z# W# K# R#
	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. Any additional axis spec that is not part of the arc itself will move that axis simultaneously throughout the arc. L# is used to allow setting laser power in vector cutting (1-100). R# is the radius, if R# is used IJK can not be used.
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] **WARNING: 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# A#
	Used to set the smoothing factor. When the XY direction of motion changes, this setting can reduce the "slowdown" to improve the smoothness of motion. Increasing the S# decreases the slowdown of the machine

	when turning corners. 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#. S# - smoothing factor to be used. (not a specific unit of measure) A# - used to change the angle used in the velocity-matching algorithm. The A# value represents an angle. If a small angle, such as say 20, is specified, the G9=# will not apply to any angle smaller than 20.
G20	Converts unit of measure to inches.
G21	Converts unit of measure to centimeters.
G22	Converts unit of measure to millimeters.
G23	Returns to the last used unit of measure.
G28	Return to machine zero - G28 XYZW
	Moves specified axes to Lo Limit for XY - Hi Limit for ZW (these are default, they can be reversed). Moves specified axes to WINCNC.INI specs from limits. Sets all axis positions to 0. All axes are moved if none are specified.
G31	Stop file parsing
	Must be used with the M28 command to measure machine position after touching a limit switch. Example: L91 G1 Z-10 F20 M28 G31 M37 Z1.
G40	Cutter Compensation Off
	Turns off cutter compensation.
G41	Cutter Compensation Left - G41 O#
	Compensates for differences in nominal and actual tool diameters. Starts to the left on the first move. O# is the compensation amount on machines without a tool changer.
G42	Cutter Compensation Right - G42 O#
	Compensates for differences in nominal and actual tool diameters. Starts to the right on the first move. O# is the compensation amount on machines without a tool changer.
G43	Tool Length Offset On - G43 ZWUV
	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 ZWUV
	Cancels tool length offsets for axis specified. G49 alone turns all offsets off.
G50	Scaling/Rotation Mode Off
	Cancels the scaling or rotation mode.
G51	Scaling Mode On - G51 X# Y# Z# I# J# K# C# W#
	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 axes using the C# line.
G52	Rotation Mode On - G52 I# J# A#
	Enables G-Code rotation. G-Code commands or files will be rotated by the specified number of degrees around the specified center point. I# is the X center point of rotation. J# is the Y center point of rotation. A# is the degrees to rotate.

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 axes.
G54	G54 Workspace - Rotational: G54 X# Y#
	Select the Z head. If G54 is selected 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
	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 the 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. U 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.
	**If running a .tap, .mac, .nc, etc., file, and there is a G92 in the file, and the user wishes to maintain that G92 after the file is done and closed, then a L92 must be added at the end of the file.
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, G92.2 Z-.01 will accomplish this with a single command. Can be useful as a macro.
G92.3	Stores and Restores G92 values
	Allows the user to switch back and forth between local and absolute coordinates. Use the G92.3 command without any parameters or axis characters. If G92 positions are set on any axis then they will be stored and cleared. The next G92.3 call will restore the previous G92 values even if they have been cleared by using G92.
G93	Inverse Time Feed Mode
	In inverse time feed rate mode, an F word means the move should be completed in [one divided by F number] minutes. For example, if the F number is 2.0, the move should be completed in half a minute. When the inverse time feed rate mode is active, an F word must appear on every line which has a G1 motion, and an F word on a line that does not have G1 is ignored. Being in inverse time feed rate mode does not affect G0 motions. It is an error if: inverse time feed rate mode is active and a line with G1 does not have an F word.
G94	Cancels the Inverse Time Feed Mode
	Turns off the Inverse Time Feed Mode.

Keyboard Hotkeys

File Menu Shortcuts	
Open	Enter (with blank command line) CTRL+ O
Edit	CTRL + S
Simulate	CTRL + V
View	
Create Home File	CTRL + H
View Menu Shortcuts	
Refresh View	F5

Edit	CTRL + C
Options Menu Shortcuts	
Keyboard	CTRL + K
Soft Limit	CTRL + L
Transit Menu Shortcuts	
Slow	F2
Medium	F3
Fast	F4
Jog Menu Shortcuts	
0.001	F6
0.01	F7
0.1	F8
1	F9
Custom	F10
Help Menu Shortcut	
Help	F1
Tool Bar Shortcuts	
ESC	Aborts the current file or command
SPACE	Pauses a file or command
ENTER	Starts/restarts a file or command
TAB	Opens the command history box
CTRL + R	Opens the restart file box
Feed Rate Override Shortcuts	
INSERT	Increase override rate
DELETE	Decreases override rate
CTRL+ either	Resets feed rate to 100%. No override settings
Spindle Speed	
Spindle Speed	+ - (plus and minus)

L-Codes

L1	Save Positions - L1 N# V#
	Store values is the saved positions file. Can store a value in one of the 30 existing 'H' registers. N# is the desired 'H' register number, can be 0 to 29. V# is the desired value. Can be the result of a math operation or a constant.
L1.1	Save 'H' Register
	Save the 'H' register value to a file.
L1.2	Load 'H' Register
	Load the 'H' register value from a file.
L3	Set Home Position - L3 XYZW
	Stores current position of each axis specified. Values specified are ignored.
L4	Return to Home
	Moves each axis specified to the last L3 position stored. Values specified are ignored.
L5	Set Contouring Velocity - L5 XYZ
	Sets the contouring velocity in units per minute.
L6	Set Acceleration - L6 XYZ
	Sets the acceleration in units per minute per second. L6 alone restores the original values.
L7	Set Acceleration Percentage L7 P#
	Sets the acceleration to a percentage of the current acceleration. P# is the percentage (1-100) to set. L7 alone restores the original.
L8	Turns Backlash Compensation On, Restore INI values for all axes.
	Examples: L8XY - Turn Backlash Compensation On, Restore INI values for specified axes. L8X.001 - Turns Backlash Compensation On, Set new value for specified axis.
L9	Turns Backlash Compensation Off.

L10	Cut Array - L10 R# C# X# Y# E#
	Sets up array cutting. Repeats all code following until the end of the file or another L10 is reached. 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. If an 'E' is found on the L10 command line then WinDNC will skip the final move back to the starting position at the end of the array. If no 'E' is found then WinDNC will move back to the starting position at the end of the array. **A G92XY is required after the L10 Line for the cut array to work. X# is the units to move the x axis before starting the next cut. Y# is the units to move the y axis before starting the next cut. R# is the number of rows. C# is the number of columns. Example: L10 r5 c5 x4 y4 e1 or L10 r1 c4 x5 y0 G92xy G92xy m98 Texas.tap G90 G90 G01 x3 G01 y0 G01 x0 **End with L10 if using more

L11	Set Axis Mapping - L11 XYZW
	Allows re-mapping of axis inputs. Specify the axes 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 axes, L11 XYZZ moves Z and W

L12	Set Axis Mapping - L12 ZW
	Allows re-mapping of axis inputs. Specify the axes that should receive the input from ZW in that order. This command works like L11 except that the XY inputs are not used. Specify L12 alone to return to normal parsing. Example: L12 WZ swaps the W and Z axes, L12 ZZ moves Z and W together.

L13	Set Axis Mapping - L13 XYZW
	Allows re-mapping of axis on a pulse basis. This means when the controlling axis takes a step, the slave will follow in that direction. Specify L13 alone to return to normal mapping. Examples: L13 ZW (Z is controlling axis, W follows Z.) L13 WZ (W is controlling axis, Z follows W.) L13 XY (X is controlling axis, Y follows X.)

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 Low Boundaries - L22 X# Y# Z# W#
	Set Low Boundaries to values specified. If no values are given then all Low Boundaries are set to WINDNC.INI defaults.

L22.1	Restore Low Boundaries - L22.1 XYZW
	Restores default Low Boundaries to the specified axes.

L23	Set High Boundaries - L23 X# Y# Z# W#
	Set High Boundaries to values specified. If no values are given then all High Boundaries are set to WINDNC.INI defaults.

L23.1	Restore High Boundaries - L23.1 XYZW
	Restores default High boundaries to the specified axes.
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.
L29	Clear Home Flags - L29 XYZ
	Clears all require home flags. Axes can be used with the command to clear require home flags only on the specified axes.
L29.1	Ignore Home Flags
	Ignores require home flags for all axes during execution of the currently running file.
L30	Knife Mode On
	Enables support for the Knife Mode.
L30.1	Knife Parameters L30.1 A# Z# P#
	Set Lift Angle, Lift Position, and Plunge depth for knife. When amount of change in the angle of the knife exceeds A# the knife will lift to make the turn. A# is also used for angle correction in Target Mode: see **Knifemode**=. Z# is the position to lift to. P# is the position that the knife will plunge to.
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. L34 D# sets the dwell time in seconds used when the channel is set to on and L35 D# 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. L35 D# 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.
L36	Torch Height Control On.

L36.1	Auto Voltage Set Mode On.
L36.2	Reset Auto Voltage Set Status.
L37	Torch Height Control Off.
L37.1	Auto Voltage Set Mode Off.
L38	Torch On.
L39	Torch Off.
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.
L41	Shell Command - L41 C:\Path\Program.exe
	Allows the user to run an executable program from within WinCNC. **NOTE:** There is no error reporting for this command. If the path or filename are incorrect the command will fail without aborting.
L48	Laser Height Control
	Sets the target voltage to the current input voltage.
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	Laser Height Control
	Sets the target voltage to the current input voltage.
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.
L60	Turn Virtual Input Off - L60 C#
	Turn specified virtual input off, # can be 1-64.
L61	Turn Virtual Input On - L61 C#
	Turn specified virtual input on, # can be 1-64.

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 Z#
	L82 Z# 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.

L82.1	Sets the Surface for Probe Digitization
	Sets a G92 for vertical axis based on the L82.3 calibration.

L82.2	Moves the Surface for Probe Digitization
	Adjusts G92 positions set by the last L82.1 based on a new measurement.

L82.3	Calibrate Digitizing Probe
	Calibrates, or stores, the distance from the probe contact point to the table surface.

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
	The current G92 Local Coordinate system is saved every time a program is started or a subprogram 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.

L110	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, or {TN1} and {TN2} for dual tool changer machines.
	The stored tool measure is reloaded for the new tool.
	(See M37 command for storing tool measures)

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 FileName T# C# S#
	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.

L204	Activate Signals
	L204 C# activates signals and enables the specified channel.
	L204 activates all signals and enables.

L205	Deactivate signals
	L205 C# deactivates signals and enables the specified channel.
	L205 deactivates all signals and enables.

L210	Select Alternate Low Limits - L210 XYZW
	Selects axis to use alternate low limits.

L210.1	Select Alternate Low Limit Pin - L210.1 XYC#
	Selects alternate low limit pin from the specified C# channel. If no channel is specified, the original settings are restored.

L211	Select Alternate High Limits - L211 XYZW
	Selects axis to use alternate high limits.

L211.1	Select Alternate High Limit Pin - L211.1 XYC#
	Selects alternate high limit pin from the specified C# channel. If no channel is specified, the original settings are restored.

L212	Select Primary Limits
	Selects the primary limits for all axes.

L213	Dual Axis, Select Main - L213 XYZW
	Selects the main motor on a dual motor axis.

L214	Dual Axis, Select Secondary - L214 XYZW
	Selects the secondary motor on a dual motor axis.

L215	Dual Axis. Select Both Axes - L215 XYZW
	Selects both motors on a dual motor axis.

L249	Laser Power Setting - L249 S# P# D# O# M# I#
	Using the L249 command allows you to specify the laser power. Example: L249 P50 (sets laser power to 50). S# - 0 or 1 sets smart engrave off/on. P# - sets the laser pulse width in percentage (1-100). D# - sets the laser pulse duty cycle (1-100). O# - sets the bitmap engraving offset. M# - sets the bitmap engraving margin. I# - reverses color polarity.

L250	Vector Laser Off
	Disables vector laser cutting.

L251	Vector Laser On - L251 P# D# L#
	Enables vector laser cutting. P# - laser power (1-100). D# - sets the distance for the distance based vector laser firing. L# - sets the distance for the off signal when perforating.

L251.1	Laser Dwell - L251.1 D#
	D# - time in seconds for dwell.

L252	Bitmap Engraving
L252.1	Stores rotation for future use.
L253	Bitmap Engraving rotated 90 degrees clockwise.
L253.1	L253.1 Stores rotation for future use.
L254	Bitmap Engraving rotated 180 degrees clockwise.
L254.1	Stores rotation for future use.
L255	Bitmap Engraving rotated 270 degrees clockwise.
L255.1	Stores rotation for future use.

L256	Offset for bitmap engraving row alignment L256 R#
	Offset on positive motions to compensate for laser firing delays to line up bitmap rows during dual motion engraving. R# sets the radius used for rotary engraving in the current unit of measure.

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
	Creates a home file at the machine's current position.

L401	Recall Positions
	Moves all axes to the positions where they were at previous to the last file run.

L500	Store Absolute Positions - L500 [X, Y, Z...] [P H T] L# C# N"Name"
	[X, Y, Z...] - axes to set positions for. If none is specified then all axes positions are stored. [P H T] - type of position (P)osition, (H)ome Position, (T)ool Position. Can only use one value. L# - 0 no vertical head lift, 1 include head lift. C# - Clearance for vertical heads if L1. N"Name" - Name for specified position. < 16 characters long.

L500.2	- L500.2 T# N# A# C# O# H# M#
	T# specifies the tool number. N# specifies the nominal diameter for the tool. A# specifies the actual diameter for the tool. C# specifies the diameter compensation for the tool. O# specifies the height offset for the tool. H# specifies the tool height. M# specifies the maximum current allowed for the tool. This command sets up a tool library instead of using the dialog boxes.

L501	Store Local Positions - L501 [X, Y, Z...] [P H T] L# C# N"Name"
	[X, Y, Z...] - axes to set positions for, if none specified then all axes positions are stored. [P H T] - type of position (P)osition, (H)ome Position, (T)ool Position. Can only use one value. L# - 0 no vertical head lift, 1 include head lift. Default L1. C# - Clearance for vertical heads if L1. N"Name" - Name for specified position. < 16 characters long.

L600	Disable Mapping Adjustments
	Disable mapping adjustment for the specified axis. If no axis is specified, mapping adjustments for all axes are disabled.

L601	Enable Mapping Adjustments
	Enable mapping adjustment for the specified axis. If no axis is specified, mapping adjustments for all axes are enabled.

L602	Start Mapping Setup
	Starts mapping setup for specified axis. First axis specifies the axis to map and the second axis specifies the axis to adjust. The axis map will automatically move to the start map position. No axis spec stops the mapping setup.

L603	Store Current Adjustment and Move - L603 (NPSE)
	Stores the current adjustment and moves the map axis to the specified map increment. N is next, P is previous, S is start, and E is end. The default move is to the next map increment position.
L604	Move Map Axis By Increment - L604 (NPSE)
	Moves the map axis to the specified map increment. N is next, P is previous, S is start, and E is end. The default move is to the next map increment position.
L605	Store Current Adjustment
	Stores the current adjustment.
L800	Disable Probe
	Disables the probe.
L801	Enable Probe
	Enables the probe.
L802	Scan Given Rectangle - L802 X# Y# I#
	Scans a given rectangle XY with an increment of I in and up/down, and side-to-side style scan. (Use automatic scan).
L803	Scan Given Rectangle - L803 X# Y# I#
	Scans a given rectangle XY with an increment of I in an up/down style scan.
L810	Scan Slice (Drop-off Detection) - L810 FILENAME.EXT X# Y# I# K#
	Digitizes a slice (line), either in X direction, or Y direction and saves the points to a text file. If a drop-off is encountered, or distance stated in X or Y is reached, the slice is saved and the scan is ended. **Only one axis can be scanned at a time: X or Y**. Uses up/down, and side-to-side probe type. FILENAME.EXT - replace with your filename. **Do not surround with quotation marks.** X# - is the actual X axis distance that you want the probe to travel looking for an edge. If it finds one, the file is saved and the scan ends. Y# - works like X#, except on the Y axis. I# - is the maximum increment for the horizontal axis. K# - is the vertical increment (how high the probe raises).
L820	Disable axisadj
	Disables automatic axis position adjustment.
L821	Enable axisadj
	Enables automatic axis position adjustment.
L901	Limit Switch Debounce - L901 D#
	Changes the limit switch debounce. D# can be 1-30.
L910	Disable G-Code Line Display
	Disabling G-Code display when running large G-Code files can reduce CPU and memory usage resulting in better overall computer system performance.

L911	Enable G-Code Line Display
	Enabling G-Code display when running large G-Code files can increase CPU and memory usage resulting in worse overall computer system performance.

M-Codes

M11	Auxiliary Output On - M11 C#
	Controls auxiliary outputs. Channel specified by # is turned on. Port and bit addresses are set up in WINCNC.INI file. **Note: For use with a Drill Bank** A d# can be added to the M11 command for use with a drill bank. The d# is the binary number of channels up to the n# specified in the drill bank 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 set up in WINCNC.INI.

M13	Auxiliary Output On - M13 C#
	Same as M11, except the output is turned on 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 set up in WINCNC.INI.

M17.1	Run Next if Input On - M17.1 D# C#
	Run next command only if channel specified by # is on. Port and bit addresses are set up in WINCNC.INI. D# can be 0 or 1. D0 sets the simulate logic to false. D1 sets the simulate logic to true.

M17.2	Run Next if AuxOut is On - M17.2 D# C#
	Run the next command only if channel specified by the # is on. Port and bit addresses are set up in WINCNC.INI file. D# can be 0 or 1. D0 sets the simulate logic to false. D1 sets the simulate logic to true.

M17.3	Run Next if Virtual Input On - M17.3 C#
	C# is a "virtual input" channel, will run the next line if the specified "virtual input" channel is on and skip the next line if the channel is off. # can be 1-64.

M17.4	Run Next if C# is 1, Otherwise Skip. M17.4 C#
	C# can be 0 or 1. 1 will run the next line, 0 will skip the next line.

M18	Pause Until Input Off - M18 C#
	Wait for channel specified by # to go off. Port and bit addresses are set up in WINCNC.INI.

M18.1	Run Next if Input Off - M18.1 D# C#
	Run next command only if channel specified by # is off. Port and bit addresses are set up in WINCNC.INI. D# can be 0 or 1. D0 sets the simulate logic to false. D1 sets the simulate logic to true.

M18.2	Run Next if AuxOut is off - M18.2 C#
	C# is an "auxiliary output" channel, will run the next line if the specified output channel is off and skip the next line if the channel is on. # can be 1-64.

M18.3	Run Next if Virtual Input is Off - M18.3 C#
	C# is a "virtual input" channel, will run the next line if the specified "virtual input" channel is off and skip the next line if the channel is on. # can be 1-64.

M18.4	Run Next Line if C# is 0, Otherwise Skip - M18.4 C#
	C# can be 0 or 1. 0 will run the next line, 1 will skip the next line.

M28	Disable Limit Abort
	M28 is used to write your own home program instead of using G28 to home the machine. WinCNC 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 axes 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 O# Z# W# H# T#
	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. O# manually specifies the tool height offset. H# is the tool number of the currently loaded tool. Use this with mode 0 only, M0 on the ATC line in the WinCNC.ini file. T# is the tool number of the currently loaded tool. Use this with mode 1 only, M1 on the ATC line in the WinCNC.ini file. Example: Jog the Z head to the spoil board. 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 spoil board. 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.

M37.1	Measure Tool for Calibrating G37 - M37.1 Z# W#
	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.

M37.2	Measure Tool for Calibrating G37 - M37.2 Z# W#
	M37.2 is used to store a tool measure in the TM2 variable. This command is used primarily with calibrating G37 for automatic tool measuring.

M37.3	Shift Workspace Offsets M37.3 S# H# T# M# X# Y# Z# ...

M37.3	Shift Workspace Offsets M37.3 S# H# T# M# X# Y# Z# ...
	M37.3 is used to shift workspace offsets for tool measures, tool measure switch heights, or individual axis.
	S# is the shift amount to be used for tool measures and tool measure switch positions. (usually a negative #).
	H# is the H register number to be shifted by the shift amount. H0 will shift all H registers.
	T# is the tool number height to be shifted by the shift amount. T0 will shift all tool numbers.
	M# is the measure switch height to be shifted by the shift amount. M0 will shift both switches.
	X# Y# Z# ... shifts each axis workspace offsets by the specified value.
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.
M99.1	Abort
	Initiates a commanded program abort.

Messages and Error Codes

Error codes will be displayed in the message display window of the screen display. The list below is in alphabetical order.

2-Bit Bitmap Engrave Not Enabled	Tried to open a 2-Bit bitmap without the feature being enabled in the software.
8-Bit Bitmap Engrave Not Enabled	Tried to open an 8-bit bitmap without the feature being enabled in the software.
Aborted By User	Escape button pushed while running.
Acceleration Out of Bounds	Acceleration rate too high for resolution. Maximum (acc*res) in steps is 65535.
Arc Radius	Distance from start point to center is not equal to distance from endpoint to center. Arc Radius errors can occur from running a G2 or G3 intended for G91 mode in G90 mode and vice versa.
Arc Too Small	Specified arc move is too small.
Arcs Disabled When X or Y scaled	When scaling is enabled, arcs are not supported.
Rotating Saw Not Enabled	The rotating saw feature is not enabled.
Auto-AuxOut Not Enabled	User has tried to use an Auto-AuxOut without the "autoaux=" line being set in the WINCNC.INI file.

BMP File Invalid Header	Tried to open an incompatible bmp file.
Boundary Exceeded	Move specified would result in the head being moved outside the workspace boundaries.
CNC.MAC Not Found	The CNC.MAC file is not found.
Command Not Implemented	This command has not yet been fully implemented.
Continuing.....	Motion is continuing.
Could Not Open File	Requested file could be opened. May be in use by another program.
D/A Not Enabled	D/A channel is not set up in the WINCNC.INI file.
Demo Count Exceeded	Demo mode only. 25 commands have been completed.
Depth Must Be Lower Than Retract	Tried to retract a tool to a height that was below current depth.
Device Not enabled	The requested device is not enabled.
Digitizer Probe Not Enabled	The digitizer probe feature has not been enabled.
Error - Syntax - Command or File Not Found	Non-supported code or invalid file name entered.
Errors Reading CNCSCRN.INI	The CNCSCRN.INI file contains errors or is not found.
Errors Reading WINCNC.INI	The WINCNC.INI file contains errors or is not found.
Escape Button Pushed While Running	User has aborted motion with the ESC key.
File Not Found	The specified file name cannot be found.
File Not Found	A specified file cannot be found.
Finished Running "Filename"	The file "Filename" is complete or has been aborted.
Forced Limit Exceeded	Move specified would result in the head being moved outside the forced limits.
Helical Interpolation Not Supported	No Z or W moves during an arc.
Illegal Value	The value specified is invalid.
Invalid L10 Command	L10 command has invalid parameters.
Invalid P1 Command	P1 command has invalid parameters.
Key Error #	A software key read error has occurred.
Key Not Found	The software key cannot be found.
L10 in Subprogram	L10 command found in subprogram.
Limit Switch	Limit switch encountered while running.
Line Too Long	Input line more than 256 characters.
Macro Expansion Error	Occurs when macro commands exceed 256 characters.

Memory Error	Insufficient or bad memory.
Multiple Commands	Input line contains more than one command.
Not a BMP File	Tried to open a non-bmp file for bitmap engraving.
Parameter Not Specified	A required parameter is not specified.
Parameter Out of Range	Specified parameter is out of range.
Pausing....	Motion is pausing.
Pump Control Not Enabled	The pump control feature has not been enabled.
Runtime: #d Minute(s) and #f Second(s)	Run time for the last file in minutes and seconds.
Soft Limit Exceeded	Move specified would result in the head being moved outside the soft limits.
Soft Limits Not Enabled	Softlimits are set as disabled.
Subprogram Nesting Too Deep	Too many nested subprogram calls.
Tangential Knife Not Enabled	The tangential knife feature is not enabled.
Tool Changer Not Enabled	Tool changer is not set up in the WINCNC.INI file.
Unknown	An unknown error has occurred.
Unknown	An error has occurred that is undefined in WinCNC.
Unknown Position Specified	The position specified is invalid.
Unsupported G Code	G Code in input line which is not supported.
Unsupported L Code	L Code in input line that is not supported or ignored.
Unsupported M Code	M Code in input line which is not supported.
Upgrade needed for Key Version 5, or 6 Axis card	And upgrade of the daugtherboard is needed.
User Level 5 Required	A Level 5 user license is required.
Vector Laser Not Enabled	Vector laser feature not enabled in the software.
Wrong Key Found	The software key found is incorrect.

Warranty Information

Dealer Machine Warranty

New woodworking machines sold by Laguna Tools carry a two-year warranty effective from the date of dealer invoice to customer/end-user. Machines sold through dealers must be registered with Laguna Tools within 30 days of purchase to be covered by this warranty. Laguna Tools guarantees all new machine sold to be free of manufacturers' defective workmanship, parts, and materials. We will repair or replace, without charge, any parts determined by Laguna Tools, Inc. to be a manufacturer's defect. We require that the defective item/part be returned to Laguna Tools with the complaint. The end-user must request an RMA (return material authorization) number from Customer Service and include the (RMA) number with all returned parts/components requesting warranty coverage.* Any machines returned to Laguna Tools must be returned with packaging in the same way it was received. If a part or blade is being returned it must have adequate packaging to ensure no damage is received during shipping. In the event the item/part is determined to be damaged due to lack of maintenance, cleaning or misuse/abuse, the customer will be responsible for the cost to replace the item/part, plus all related shipping charges. This limited warranty does not apply to natural disasters, acts of terrorism, normal wear and tear, product failure due to lack of maintenance or cleaning, damage caused by accident, neglect, lack of or inadequate dust collection, misuse/abuse or damage caused where repair or alterations have been made or attempted by others.

***NOTE: Issuing an RMA number is for referencing materials and issues, it does NOT indicate warranty acceptance/conformity.**

**Any machines returned to Laguna Tools must be returned with packaging in the same way it was received. If a part or blade is being returned it must have adequate packaging to ensure no damage is received during shipping. In the event the item/part is determined to be damaged due to lack of maintenance, cleaning or misuse/abuse, the customer will be responsible for the cost to replace the item/part, plus all related shipping charges.

We require that the defective item/part be returned to Laguna Tools with the complaint. The end-user must request an **RMA (Return Material Authorization) Number** from Customer Service and include the (RMA) number with all returned parts/components requesting warranty coverage.

Sample RMA

CNC Limited Warranty

New CNC machines sold by Laguna Tools carry a one-year warranty effective from the date of shipping. Laguna Tools guarantees all new machine sold to be free of manufacturers' defective workmanship, parts, and materials. We will repair or replace without charge, any parts determined by Laguna Tools, Inc. to be a manufacturer's defect. We require that the defective item/part be determined to be damaged due to lack of maintenance, cleaning or misuse/abuse, the customer will be responsible for the cost to replace the item/part, plus all related shipping charges. This limited warranty does not apply to natural disasters, acts of terrorism, normal wear and tear, product failure due to lack of maintenance or cleaning, damage caused by accident, neglect, lack of or inadequate dust collection, misuse/abuse or damage caused where repair or alterations have been made or attempted by others. Laguna Tools, Inc. is not responsible for additional tools or modifications sold or performed (other than from/by Laguna Tools, Inc.) on any Laguna Tools, Inc. woodworking machine. Warranty maybe voided upon the addition of such described tools and/or modifications, determined on a case-by-case basis. Software purchased through Laguna Tools, Inc., is not covered under this warranty and all technical support must be managed through the software provider. Normal user alignment, adjustment, tuning, and machine settings are not covered by this warranty. It is the responsibility of the user to understand basic woodworking machinery settings and procedures and to properly maintain the equipment in accordance with the standards provided by the manufacturer.
Parts under warranty are shipped at Laguna Tools, Inc.'s cost either by common carrier, FEDEX ground service or a similar method. Technical support to install replacement parts is primarily provided by phone, fax, e-mail, or Laguna Tools Customer Support Website. The labor required to install replacement parts is the responsibility of the user. Laguna Tools is not responsible for damage or loss caused by a freight company or other circumstances not in our control. All claims for loss or damaged goods must be notified to Laguna Tools within twenty-four hours of delivery.

****Please contact our Customer Service Department for more information. Only NEW machines sold to the original owner are covered by this warranty. For warranty repair information, call 1-800-332-4094. Copyright 2013 Laguna Tools, Inc. ****Warning – no portion of these materials may be reproduced without written approval from Laguna Tools.**

Modifications Policy

No Modifications Allowed or Sold.

Laguna Tools, Inc. is not responsible for additional tools or modifications sold or performed (other than from/by Laguna Tools, Inc.) on any Laguna Tools, Inc. woodworking machine. Warranty maybe voided upon the addition of such described tools and/or modifications, determined on a case-by-case basis. Normal user alignment, adjustment, tuning, and machine settings are not covered by this warranty. It is the responsibility of the user to understand basic woodworking machinery settings and procedures and to properly maintain the equipment in accordance with the standards provided by the manufacturer. Parts, under warranty, are shipped at Laguna Tools, Inc.'s cost either by common carrier, FEDEX ground service or a similar method. Technical support to install replacement parts is primarily provided by phone, fax, e-mail, or Laguna Tools Customer Support Website. The labor required to install replacement parts is the responsibility of the user. Laguna Tools is not responsible for damage or loss caused by a freight company or other circumstances not in our control. All claims for loss or damaged goods must be notified to Laguna Tools within twenty-four hours of delivery. Please contact our Customer Service Department for more information. Only new machines sold to the original owner are covered by this warranty.

For warranty repair information, call 1-800-332-4094.

Standard Damage Notification Sheet

The machines are thoroughly tested before leaving any or our Laguna Tools facilities, but that does not mean the machines would not experience any damage in transit.

Before signing the *Bill of Lading* (See Example Below) when the trucking company drops off the machine, **visually inspect the entire crate and check for any damage.**

LAGUNA *SuperMax*

LAGUNA AMERICAN HEADQUARTERS

Texas: 744 Refuge Way Suite 200, Grand Prairie, Texas 75050, U.S.A. Phone: +1-800-332-4094
Huntington Beach: 7291 Heil Ave Huntington Beach, CA 92647, U.S.A. Phone: +1-949-474-1200
South Carolina: 825 Bistline Dr. Ste 101, West Columbia, SC 29172, U.S.A. Phone: +1-800-234-1976
Minnesota: 5250 West 74th St, Edina, MN 55439, U.S.A Phone: +1-949-474-1200

LAGUNA EUROPE

Walker Rd, Bardon Hill, Coalville LE67 1TU, United Kingdom. Phone: +44-1530-516921