# spectraLIGHT Turning Center

# User's Guide

## For Windows

 Manchester, New Hampshire, U. S. A. March, 1998

The information contained in this guide was accurate at the time of its printing. Light Machines Corporation reserves the right to change specifications and operational performance of its products without notice. Any changes or corrections to the information contained in this publication will be incorporated in future issues.

This publication (34-7611-0003, March, 1998) corresponds to the spectraLIGHT CNC Turning Center package, including the WSLT Control software.

Printed in U.S.A.

# spectraLIGHT Turning Center

## User's Guide

### For Windows

34-7611-0003

# User’s Guide

Section A: Installation

Section B: spectraLIGHT System Hardware

Section C: Getting to Know the Control Program

Section D: Tutorial: Turning a Sample Part

# Reference Guide

## WARNING

The operation of rotating machinery should only be attempted by experienced, knowledgeable individuals!

Read the entire contents of this guide before running the spectraLIGHT Machining Center.

To avoid possible injury *always* observe the safety precautions described in this User's Guide.

# Contents

# User's Guide

## Section A: Installation

## Section B: spectraLIGHT System Hardware

## Section C: Getting to Know the Control Program



## Section D: Tutorial: Machining a Sample Part

# Reference Guide

## Section E: Control Program Reference

## Section F: Basic CNC Programming

## Section G: More CNC Programming



## Section H: Optional Machining Capabilities

## Section I: General Machining Information



## Section J: Safe Turning Center Operation

## Section K: G and M Codes Listed by Group



## Section L: Robotic Integration



## Index

# Installation

# Getting Ready for Installation

Before connecting and running your new Turning Center, you should:

1. Check your shipment to make sure you received everything you need.
2. Register your Turning Center so you are covered by your warranty.
3. Prepare a work space for the Turning Center and controller.
4. Unpack and set up the Turning Center.
5. Install the interface card in the computer.

Once these procedures are complete, connect the spectraLIGHT Turning Center and Controller Box to your personal computer, and install the Control Program.

## Check Your Shipment

The first thing you must do after receiving your Turning Center is inspect the packaging for any visible signs of damage. If there is damage to the outside packaging, contact the shipping company as well as Light Machines Corp. After checking the packaging, locate the packing slip. This slip lists all of the items you should have received with your Turning Center. Check all of the items on the list. If any item is missing, contact Light Machines' Customer Service Department (800/221/2763).

The Turning Center components include:

- spectraLIGHT Turning Center
- Controller Box
- Interface Card
- Control Program Software
- Documentation
- Accessory Kit

## Register Your Turning Center

You'll find a registration card in the small box with the documentation and software disks. Clearly print all the requested information and return this card to Light Machines Corporation.

## Prepare the Work Place

Make sure you have all the items on hand necessary to perform the installation. To install the spectraLIGHT Turning Center, you must have:

- A sturdy table on which you'll place the Turning Center and your computer. Placing the table against a wall provides more stability. Make sure the wall has a 120VAC, 15 Amp polarized outlet, or a 220VAC, 8 Amp outlet for international users.
- A personal computer running Windows 95 or Windows NT version 3.51 (or greater). See page A-19 for a complete list of the necessary computer equipment.
- Your *PC Owner's Manual* or equivalent documentation.

**Note:**

We recommend the use of a voltage surge protector and line filter for your computer system.

## Unpack the Turning Center

For each standard Turning Center you order, you should receive two large cartons. One carton contains the Turning Center. The other carton contains the Controller Box, the Interface Card, the Control Program software, the documentation, and the Accessory Kit.

1. Unpack the Controller Box container and use the packing slips to confirm that you have received all the items listed.
2. Open the Turning Center container and remove the foam inserts.
3. Lift the Turning Center out of the box and onto the table.
4. Inspect the Turning Center chassis for signs of visual damage such as a broken shield, a dent in the chassis, or damaged cables. If any damage is noted, or if you find any discrepancies between the packing slips and the items received, call Light Machines' Customer Service Department (800/221/2763).
5. Remove the protective paper from the safety shield.

**CAUTION**

The spectraLIGHT Turning Center weighs approximately 80 pounds. Be very careful when lifting it.

**IMPORTANT!**

Be sure to keep all of the original cartons in which the spectraLIGHT Turning Center was shipped. Should any components need to be returned to the factory, repack them exactly as they were received.

*Light Machines will not be responsible for any damage caused during shipping when components are not returned in the original cartons.*

# Hardware Installation

The following paragraphs review the procedures for installing the hardware components of the spectraLIGHT Turning Center. You should already have your personal computer set up in accordance with the directions in the computer owner's manual.

The first thing you have to do is install the spectraLIGHT Interface Card in your personal computer.

**WARNING**

**Do not connect power to the Turning Center or the computer until instructed to do so in the following procedures.**

## Installing the Interface Card in the PC

**Note:**

The Interface Card is factory set to operate in the address range reserved for Bisync cards on the PC I/O address map "03A0 hex ". If another expansion card is already installed to operate at this address, contact Light Machines' Technical Support Department for installation assistance.

The Interface Card can be installed in any full-size slot designated for expansion card use. Refer to your computer owner's manual to determine particular expansion card restrictions.

**WARNING**

**Do not plug the power cord from the PC chassis into an AC outlet until all installation procedures have been completed and the chassis cover has been closed.**

### Opening the PC Chassis

To install the Interface Card, you must remove the cover of the personal computer. Refer to the installation instructions supplied with your computer for details on removing the cover. Generally, the cover is secured by four screws through the rear panel; however, some personal computers may have push latches, or screws in different locations.

**WARNING**

**Disconnect power from your personal computer before opening its chassis cover!**

**Turn off the power switch and remove the power cord to assure that no electrical potential is present when the cover is removed.**

Set the cover aside and locate an open slot in which to install the Interface Card. Remove the blank slot cover (if any). Removing the slot cover requires removing a screw at the top rail of the rear panel. You may choose to discard the cover, but save the screw for installing the Interface Card.

## Unpacking the Interface Card

The card is shipped inside an antistatic envelope. Be careful not to create any static discharge when removing the card from the envelope; touch a grounded surface such as the PC enclosure first. Slide the card out of the envelope and inspect it for signs of damage, such as bent or broken components or a warped circuit card. If damage is noted, contact Light Machines Corporation immediately.

## Inserting the Interface Card

The following procedures describe how to insert and secure the Interface Card in the computer.

1. Grasp the Interface Card at the front and back.
2. Position the card above the bus connector at the chosen slot. The interface connector on the end of the card should face the rear panel of the computer chassis.
3. Slide the card into the bus connector. The interface connectors on the card should protrude from the rear panel of the computer. Carefully wiggle the Interface Card back and forth to assure its tightness in the bus connector. Components on the Interface Card should not touch adjacent cards or other components.
4. Secure the Interface Card to the top rail of the rear panel with the screw you saved when removing the blank slot cover.
5. Pay particular attention to the location of the slot in which you inserted the Interface Card. Do not get the Interface Card mixed up with the parallel port or serial port which use the same types of connectors.

**Note:**

It may be beneficial to label the connectors on the back panel of the PC for easy identification.

## Checking Your Installation

After installing the Interface Card, replace the computer chassis cover. Connect the computer power cord and turn the computer on. The computer should perform an internal check, then start the Windows operating system.

If the PC fails to start-up, turn off the power, open the chassis and check your installation to be sure that the Interface Card is located in an appropriate slot and is properly seated.

When the C:> prompt appears, turn off the power and install the other hardware components.

# Connecting the Turning Center

The following paragraphs review the procedures for connecting your computer with the Turning Center and Controller Box. The Interconnection Diagram on the following page has been provided as a visual aid for the recommended connections.

### WARNING

**Do not connect power to the Turning Center, the Controller Box or the computer until instructed to do so in the following procedures.**

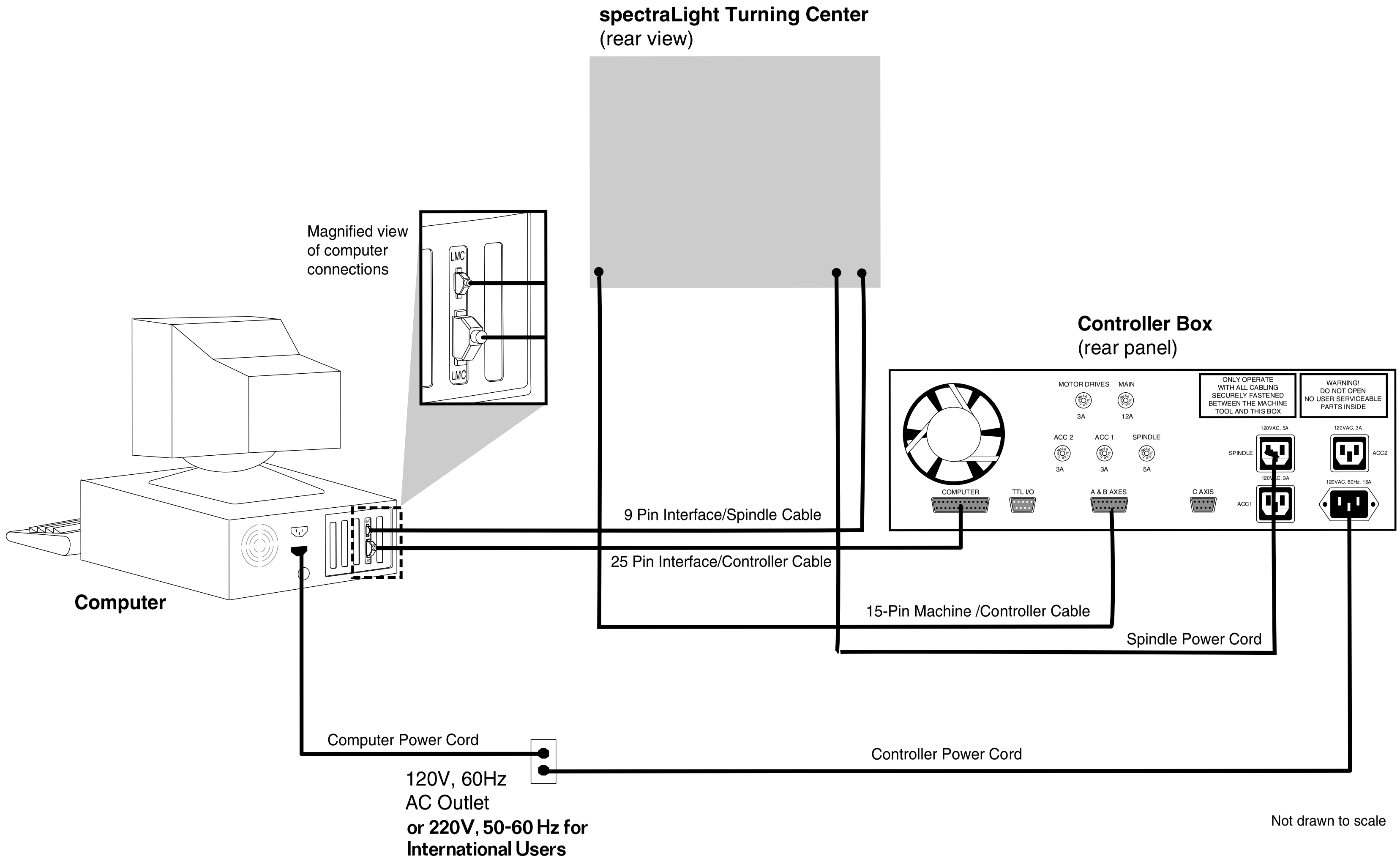
**Never connect or disconnect the cables with the power on! This will cause damage to the Controller Box drive components.**

**Operate the Turning Center with all cables firmly secured.**

## Connecting the Turning Center to the Controller Box

The interface cables and power cords are long enough to allow the Controller Box to be located up to five feet away from the Turning Center. Make sure the Turning Center is placed on a stable, flat surface and leveled properly.

The Controller Box can be placed beside the Turning Center, or mounted on a shelf beneath the Turning Center. Make sure the power switch on the front of the Controller Box is readily accessible. Keep in mind that you may need to check the fuses on the rear panel of the Controller Box. Locate the Controller Box in an area where it will not be exposed to metal chips or cutting fluid.




## spectraLIGHT Turning Center System Interconnection Diagram

(rear view of system)

## Connecting the Turning Center to the Controller Box, Cont.

1. Route the Spindle  Power Cable from the Turning Center to the Controller Box as shown in the Interconnection diagram.
2. Insert the AC power plug from the Turning Center's Spindle Power Cable into the three-prong receptacle marked SPINDLE on the rear panel of the Controller Box.
3. Insert the 15-Pin plug from the Turning Center into the 15-Pin receptacle marked A & B AXES on the rear panel of the Controller Box.
4. Make sure the 15-Pin Connector is secured with screw locks.

### CAUTION

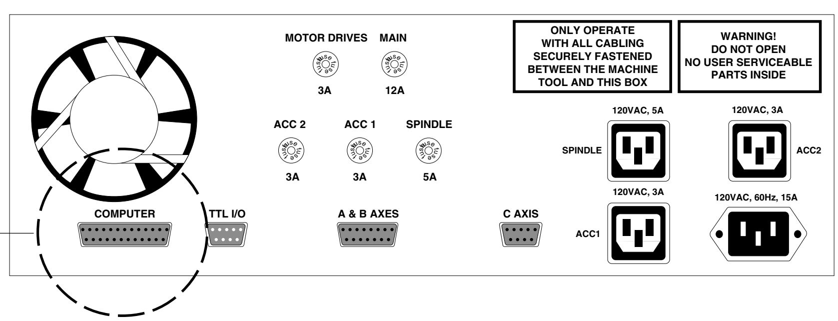
Keep  the AC power cords separate from the interface cables. The power cords can create noise problems with the signal lines in the interface cables.

## Connecting the Turning Center to the Computer

The Interface Card/spindle cable is attached to the rear panel of the Turning Center and is terminated with a 9-Pin plug. Insert this plug into the 9-Pin receptacle on the computer Interface Card you installed earlier.

## Connecting the Computer to the Controller Box

1. Route the 25-Pin Interface/Controller Cable between the Computer and the Controller Box.
2. Connect the end of the cable marked COMPUTER to the 25-Pin connector protruding from the LMC Interface Card in the rear of the computer. As mentioned before, make sure you are plugging the cable into the Interface Card connector, not the parallel port.
3. Connect the other end of the cable to the 25-Pin connector marked COMPUTER on the rear panel of the Controller Box.
4. Make sure all connectors are secured.




The location of the "Computer" connector on the Controller Box.

# Connecting Power to the Hardware Components

The  Controller Box must be plugged into a grounded 120VAC, 60Hz, 15A polarized wall outlet (220VAC, 50-60 Hz, 8A for International Users), as shown in the Interconnection Diagram. This outlet must be capable of supplying up to 12 Amps (6 Amps) of power to the Controller Box.

Locate the loose power cord that came with the Turning Center; this is the power cord for the Controller Box. Insert the receptacle end of this cord into the 120VAC (220VAC) three-prong connector on the back of the Controller Box. Insert the plug end of this cord into a grounded, three-hole, 120VAC, 60Hz (220VAC, 50-60Hz) wall outlet.

# International Users

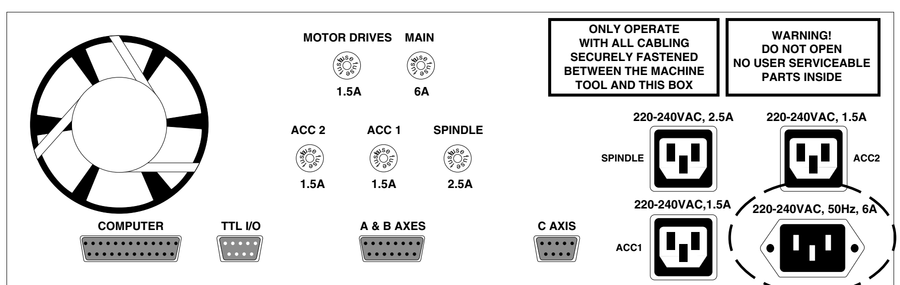
If you are an international user and you need to change the plug on the end of the electrical cord to accommodate 220VAC, 50 Hz service, the wiring is color coded as follows:

Line: white

Neutral: black

Ground: green with a yellow stripe

Rear view of the International Controller Box

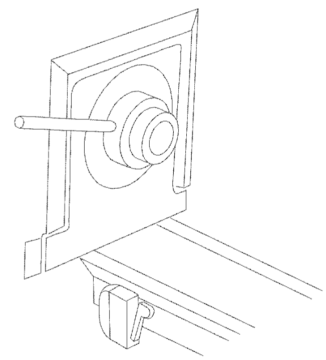



This plug termination is a standard computer type receptacle. International users need to source a computer power cable for their country.

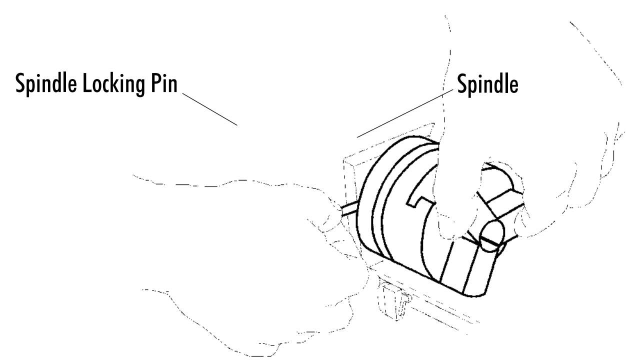
# Chuck Installation

A three jaw chuck is a standard accessory for the spectraLIGHT turning Center. Installation instructions for the chuck follow.

1. Remove any device attached to the Turning Center spindle.
2. Insert the Spindle Locking Pin (from the Accessory Kit) through the hole in the spindle to lock it in place.



3. Holding the Locking Pin with one hand, thread the chuck onto the spindle nose and hand tighten securely. Remove the Locking Pin.

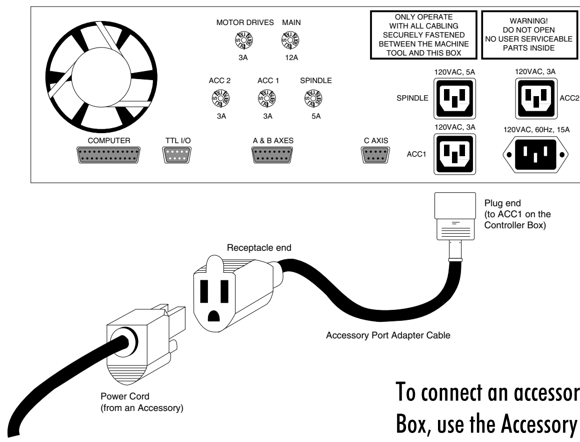


# Optional Connections

If you have purchased the optional Air Chuck, plug the power cord from the solenoid valve on the vise into the receptacle end of the accessory port adapter cable. Plug the other end of the accessory port adapter cable into the receptacle labelled ACC 1 on the rear panel of the Controller Box.

**Note:**

Remove the protective cap from the plug end of the Accessory Port Adapter Cable before inserting it into the accessory connector on the Controller Box.




To connect an accessory to the Controller Box, use the Accessory Port Adapter Cable

You can also connect a second 120VAC accessory to your lathe. Plug the accessory's power cord into the receptacle end of another accessory port adapter cable. Insert the plug end of the cable into the receptacle labelled ACC 2 on the rear of the Controller Box; the current draw for such accessories, however, is limited to 3 amps.

A 9-Pin male connector (labeled TTL I/O) is provided on the rear panel of the Controller Box for interfacing to an I/O device such as a robot. See the Reference Guide, Section L for details on interfacing with robots.

# Automatic Tool Turret Installation

The following paragraphs describe the installation procedures for the Automatic Tool Turret option.

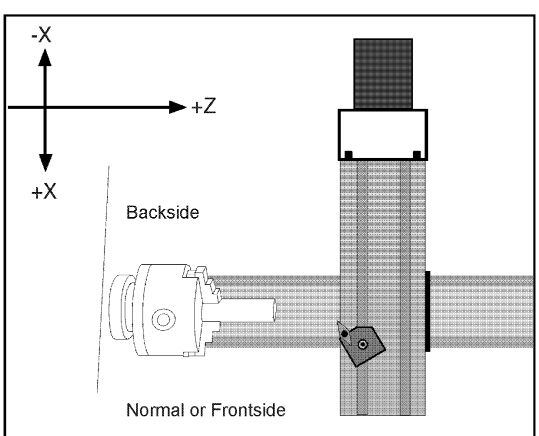
## Control Program Software

After you install the Control Program software on your computer, run the Setup.exe program that was installed in the WSLT program group. In the Configure WSLT Window that appears, select the Control tab. You must check Tool Turret Installed on the Turret Settings box before you can use the turret on the spectraLIGHT Turning Center. You will need to adjust your NC coding for tool turret installation.

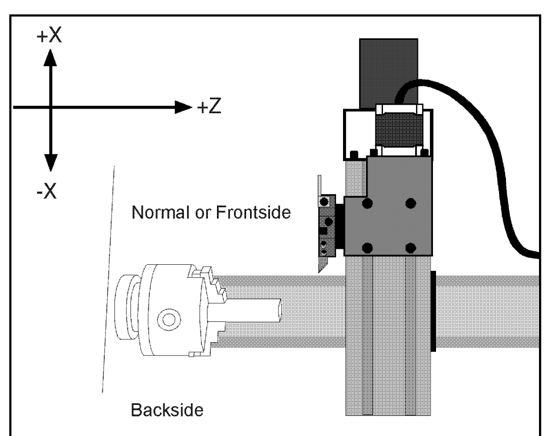
## Programming Considerations

It is important to understand that since the turret is mounted on the backside of the cross slide (on the backside of the workpiece), the polarity of the X axis is reversed. Therefore, you will see references to this reversed X axis situation throughout this guide.

The reversal in X axis polarity does not mean you must program in the -X quadrant to get your programs to work. Write your NC programs as you normally would, in the +X quadrant, and the Control Program software will take care of the conversion by your indication of the installed tool turret, and normal or frontside versus backside operation. The only type of programming you should not consider with the turret is cutting from backside, which is now the front side.




Tool Post




Tool Turret

# Installing the Turret

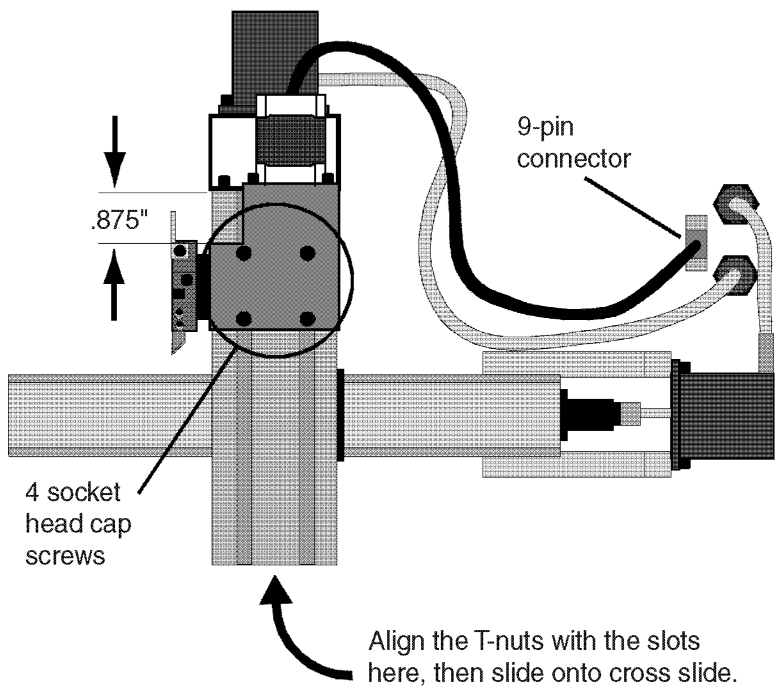
## Mount the Turret

To mount the turret body onto the top edge of the cross slide, align the T-nuts on the turret body with the cross slide slots. Slide the turret along the slots until the turret is approximately .875" from the upper edge of the cross slide. Tighten the four socket head cap screws using a 5/32" hex key. The stepper motor on the turret body should extend out over the X axis stepper motor.

1. Mount the turret body onto the cross slide about .875" from the upper edge.

   Be careful not to overtighten the hold down bolts.

2. Connect the tool turret cable (the black cable in the illustration) to the 9-pin connector on the right side of the rear panel as shown here.



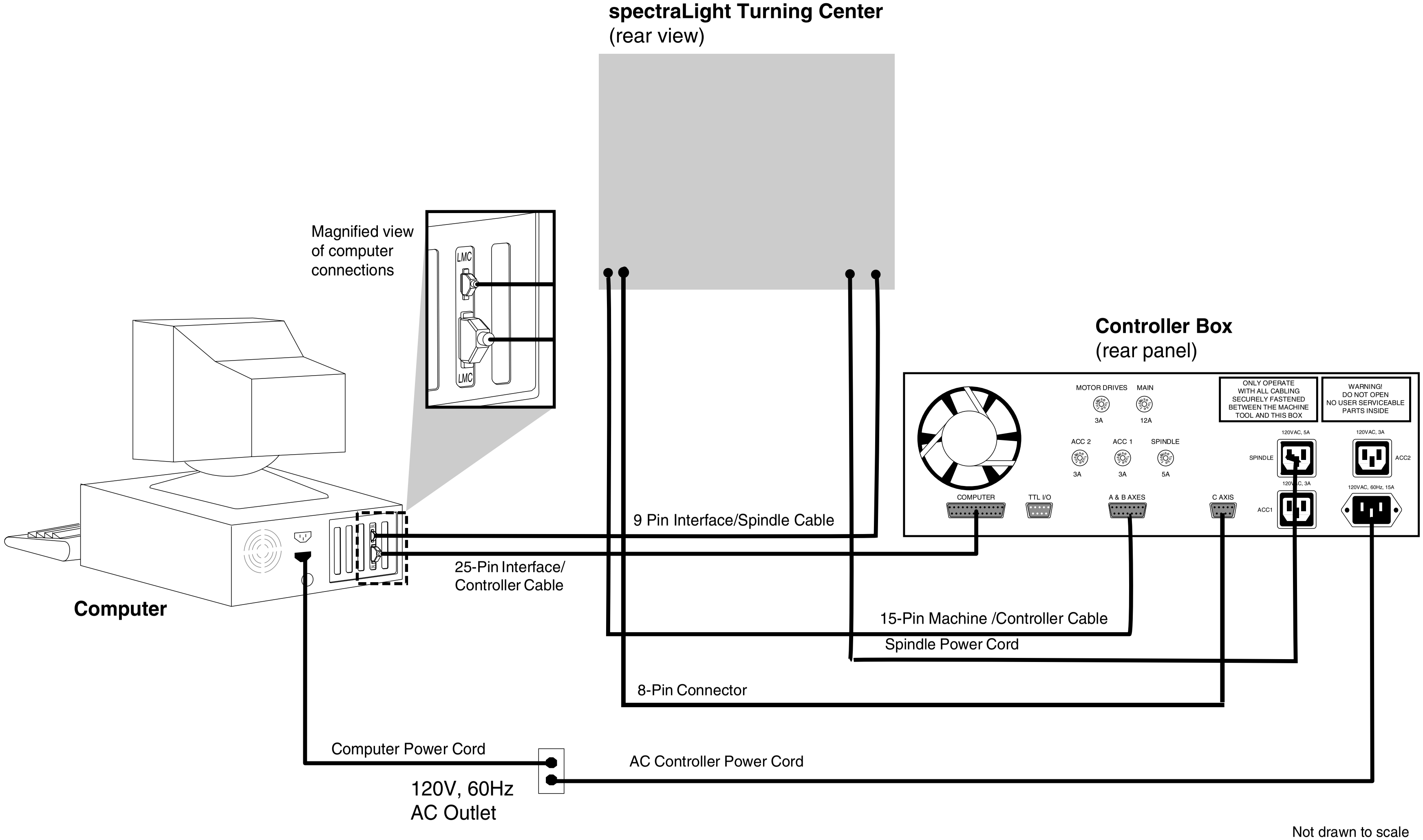
## Make the Connections

Attach the 9-pin connector from the turret stepper motor to the 9-pin connector on the Turning Center as shown.

You'll find an 8-Pin round connector on the rear panel of the Turning Center. This is the turret interface connector.

Use the cable with an 8-Pin circular connector at one end and a 9-pin connector at the other end to connect the Turning Center to the Controller Box. Insert the 8-Pin circular connector of the cable into the 8-Pin circular receptacle on the rear panel of the Turning Center. Push and turn the connector collar to lock the connector in place.

Insert the 9-pin connector of the cable into the mating 9-pin connector labeled C AXIS on the rear panel of the Light Machines Controller Box (see Interconnection Diagram on following page). Tighten the screws on the connector.



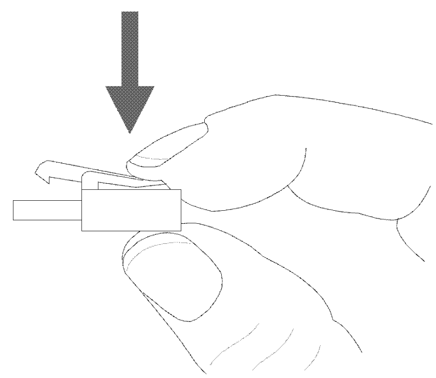

# spectraLIGHT Turning Center Interconnection with Optional Tool Turret

## Alter the Limit Switch Polarity

To complete the turret installation, you must alter a connection to change the sense of the limit switch. This is to accommodate the reversal in X axis polarity mentioned earlier.

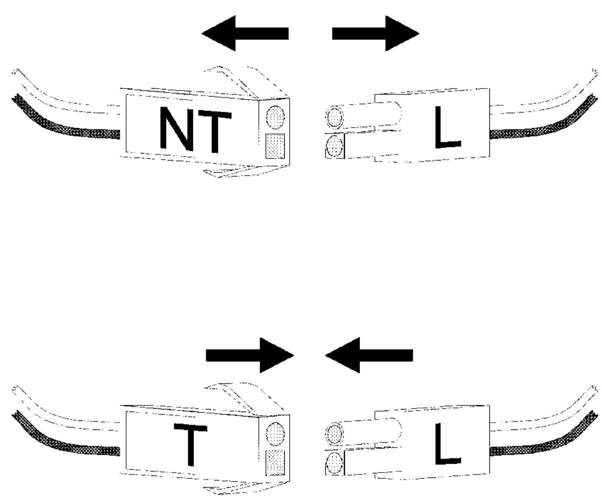
Turn the Turning Center onto its side (spindle side down) so you can safely reach the connectors underneath. If you look under the Turning Center, you can see that there are several small white connectors. Locate the two connectors marked NT (for non-turret) and L (for limit). They should be connected to each other.

To disconnect the NT and L connectors, press down and hold the lever on the L connector and pull them apart.



To disconnect the L and NT connectors, depress and hold down the lever on the L connector, then pull the two connectors apart.

Locate the connector marked T (for turret). Connect the L and T connectors. You will be able to feel the two connectors “snap” together. Carefully lower the Turning Center back onto it’s feet.

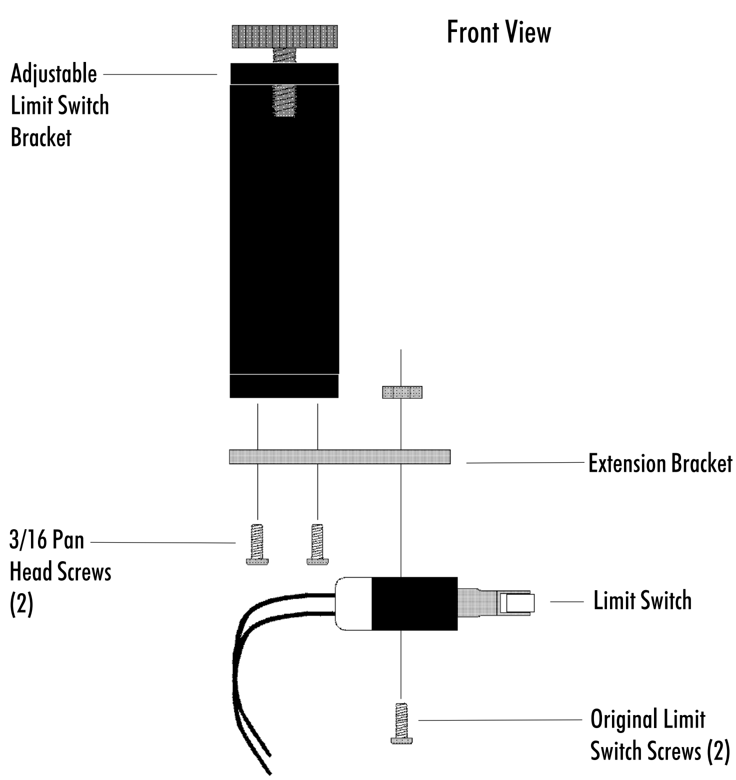



Disconnect the L and NT connectors and connect the L and T connectors. The connectors are keyed (one square pin and one round pin) so there is no chance of mixing up the pins.

## Add the Z Limit Switch Extension Bracket

With the tool turret mounted to the cross slide, the tool turret face may hit the chuck with the limit fully extended. The solution to this problem is to attach an extension that moves the limit switch out far enough so the tool turret face does not contact the chuck. To install the extension:

1. Locate the Z limit switch on the lathe bed. (For the location of the limit switch, see the illustration on page B-3). The limit switch is fastened to the bed by an adjustable bracket. Loosen the thumb screw to remove the adjustable bracket and, therefore, the limit switch from the lathe bed.
2. Remove the screws that hold the limit switch onto the adjustable bracket. Retain the screws, you will use them again later in this procedure.
3. Remove the extension bracket (and accompanying 3/16 pan head machine screws) from its package. The extension bracket can be found in the Accessory Kit that was shipped with the Turning Center.
4. Using the 3/16 pan head machine screws that came with the extension bracket, attach the extension bracket to the adjustable bracket as shown above.
5. Attach the limit switch to the opposite end of the extension bracket, using the two screws that you retained earlier.
6. Return the adjustable bracket to its original position on the bed and tighten the thumb screw.



Important! You may use standard off-the-shelf tooling, but it may have to be modified so it does not strike the lathe bed.

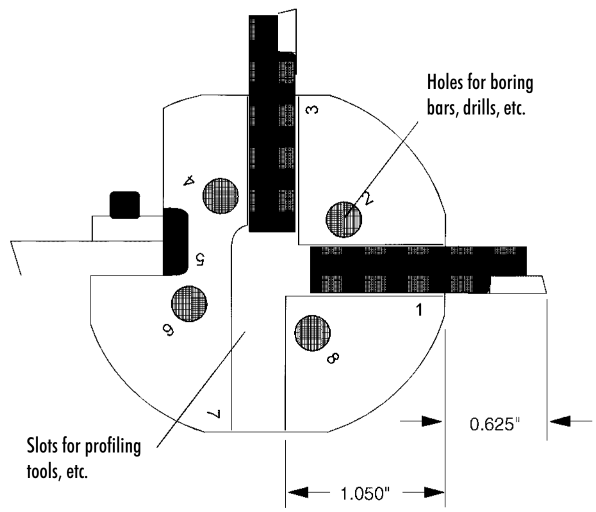
The tool tip must not extend more than 0.625" from the tooling plate.

The tool length must fit within the 1.050″ slot.

Light Machines offers a Tooling Package that contains a variety of standard tooling which has been modified especially for use with the tool turret.

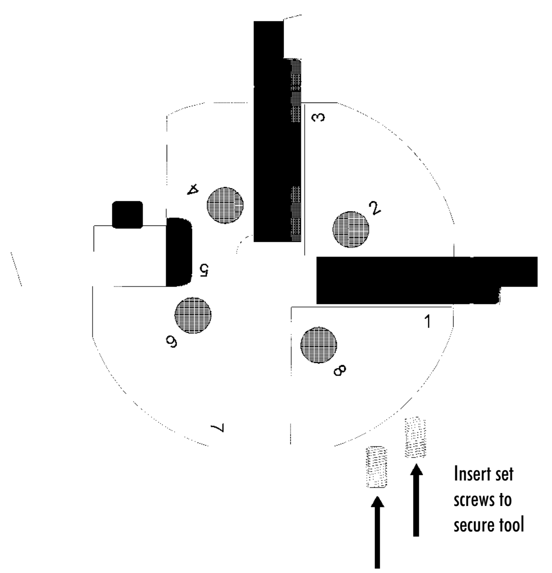
## Mounting the Tools

The tool turret will support up to eight different tools at one time. All tools must be mounted upside down in the tooling plate since, when using the tool turret, the workpiece rotates in a clockwise direction which brings the material into the tool from the bottom, up.




The tooling plate has three slots (1, 3 and 7) for holding right-hand, left-hand, profiling and external threading tools, and four holes (2, 4, 6 and 8) for holding boring, drilling and internal threading tools. Tool position 5 is for the cut off tool. To mount a tool, lay it in a slot, or insert it in a hole, and tighten the two set screws.

Important! Before cutting, make sure that each tool has sufficient clearance so it does not ram into the chuck or workpiece when another tool is selected.



## Computer System Requirements

The spectraLIGHT Control Program runs on a 586/120MHz personal computer. The computer must have:

- Windows 95 or Windows NT 3.51 or higher. (Windows NT users: check the Readme file for new information).
- 16MB RAM minimum for Windows 95. 20MB RAM minimum for Windows NT.
- A 3.5" floppy drive.
- A hard drive with at least 5MB of available space.
- An available expansion slot.
- VGA graphics controller and monitor.
- A Windows-compatible mouse.

CAUTION

The spectraLIGHT master disks are shipped write-protected (the write-protect window is open) to prevent accidental destruction of the software. *Never* remove the write protection! Create and use a working copy of the disks. Always store disks in a safe place away from heat, sunlight and static.

CAUTION

Make sure you read all the safety instructions in this guide before you attempt to run the Control Program with the Turning Center for the first time.

# Software Installation

The spectraLIGHT Control Program must be installed on the hard drive on your computer. The spectraLIGHT Control Program is shipped on two 3.5", 1.4MB disks. The Control Program must be installed on a hard drive running either Windows 95, or Windows NT version 3.51 (or higher). You must have at least 5MB of free space on your hard drive to perform this installation. (See above for system requirements.)

## Installing the Control Program

The following instructions assume that your hard drive is drive C, and your floppy drive is drive A.

1. Turn on the computer. Wait for it to go through its internal checks and for it to complete the start up process.
2. When your Windows desktop appears, insert the spectraLIGHT disk in the computer floppy drive.
3. Using the Windows Explorer, (Start Menu>Programs>Windows Explorer) open the floppy drive. Note: If you are installing on Windows NT, use either the File Manager to access the floppy drive, or select "Run" from the Program Manager.
4. Double click on Setup.exe to start the installation.
5. The Welcome screen appears. You are warned to exit all other running programs. If no other programs are running, click Next.
6. The next screen requests that you enter the destination directory for the Control Program. If you would like to place the Control Program in a directory other than the default directory, click on Browse and select an alternate destination. Otherwise, click Next.
7. A window appears, displaying installation progress, and prompting you on how to proceed. After installation is complete, you are prompted to view the Readme file. It is beneficial to view the Readme file at this time because it contains important information about the software and the lathe that may not be included in this guide.
8. Run the Control Program by double clicking the program icon.
9. If running Windows NT, you need to reboot the computer.

## Uninstalling the Control Program

**Note:**
Using the Uninstall program will not delete any NC files that you may have created or used with the software.

In the event you need to remove the Control Program from your hard drive, there is an uninstall program included on the software disks. The uninstall program was copied onto your hard drive when you installed the Control Program.

To uninstall the Control Program, just double click the Remove Program icon (it should be in the same folder as the Control Program). A message appears asking if you are sure you wish to remove the program and all its files. Click on Yes to uninstall, or Cancel to exit the Uninstall program.

## The SETUP Program

The Control Program automatically sets most variables for you.

If you will be using the Automatic Tool Turret, you must confirm this in the Configure WSLT Window. Choose the Control tab, and on Turret Settings, check Tool Turret Installed.

If you need to access the Setup Program for other configurations, see the Reference Guide: Section E in this manual.

## Software Start-Up Problems

If you are experiencing start-up problems after installing the Control Program software, check the items listed below.

- Are the computer and monitor adequately supplied with power from an AC outlet?
- Is the monitor correctly connected to the computer?
- Are all the interface connections secure?
- Are the monitor and the computer turned on?
- If you are using a mouse, was the mouse driver software loaded be fore the Control Program software?
- Do you have enough memory on the computer to run the Control Program? More RAM can be gained by closing other applications.

If the WSLT Window is still not displayed after following the steps in the Control Program Installation process, refer to your *PC Owner's Manual* or call Technical Support for assistance.

# Technical Support

Should you require technical assistance, contact your local Light Machines dealer. If you are unable to resolve your problem through your local dealer, free technical support is available by phone, fax or email from 8:15 A.M. to 5:00 P.M. EST.

## Before Calling

Make sure you have the following information gathered before contacting our Technical Support group.

- The product serial number.
- The name of the owner of the product.
- The specifications of your computer (e.g. hard drive size, clock speed, etc.).
- Notes on any Control Program error messages.

When you call, make sure you have access to both your Turning Center and your computer. This will allow our technical support representatives to *walk through* the problem with you. Our technical support numbers are:

| | |
|---|---|
| U. S. | **(800) 221-2763** |
| Canada | (800) 637-4829 |
| Fax | (603) 625-2137 |
| email | support@lmcorp.com |
| WWW Site | http://www.lmcorp.com/lmcorp |

## Warranty

Light Machines' products (excluding software) carry a one-year limited warranty from date of purchase. Defective products may be returned for repair or replacement according to the conditions outlined in the Terms and Conditions of Sale agreement. If you need to return a product, call Light Machines and a Technical Support representative will issue you a Return Materials Authorization number (RMA). You must write the RMA and your return address on the outside of the product carton or crate. Failure to do so can result in a delay in the return of your product.

IMPORTANT!

Be sure to keep the pallet and all of the original cartons in which the Turning Center was shipped. Should any components need to be returned to the factory, pack them for shipping exactly as they were received.

Light Machines will not be responsible for any damage incurred during shipping when components are not returned in the original packing materials.

User's Guide: Section B

# spectraLIGHT System Hardware

An Introduction to the spectraLIGHT Turning Center

Option Installation

Maintaining the Turning Center

Maintaining the Controller Box

Maintaining the PC in a Shop Environment

# An Introduction to the spectraLIGHT Turning Center

The spectraLIGHT Turning Center is a two-axis tabletop lathe which you can run directly from your personal computer. The spectraLIGHT Control Program, which you load onto your computer, accepts standard EIA RS-274D G&M codes that CNC machine tools recognize. You'll find a comprehensive list of options and accessories for the spectraLIGHT Turning Center on the last page of this guide.

## Features

Some of the spectraLIGHT Turning Center's most notable hardware and software features include:

- X axis travel of 4.6 inches (116mm)
- Z axis travel of 6 inches (152mm)
- Feed rates up to 12 ipm (rapid traverse up to 30 ipm)
- Feed rate and spindle speed override functions
- Tailstock
- EIA RS-274D standard G&M code programming
- Multiple tool programming
- Computer-controlled spindle speeds from 200 to 2,500 RPM
- A built-in full-screen NC program editor
- User's Guide and Introduction to CNC Workbook
- An on-line help utility
- Safety shield and limit switches
- Emergency stops from the Turning Center and computer keyboard

## The Turning Center Components

There are, of course, more components on the Turning Center than those shown here, but, to begin, you need only be concerned with the depicted major components.

## The Main Machine Components

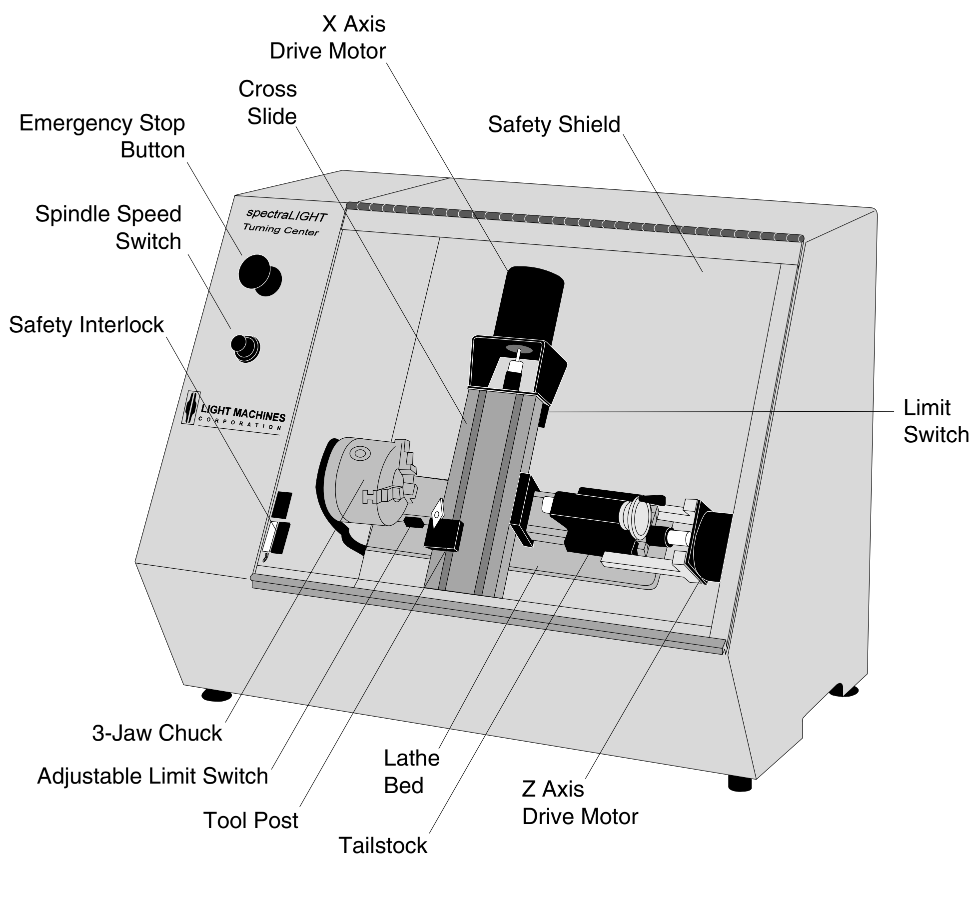
The Safety Shield encloses the turning area to help protect the operator from flying chips. A magnetic Shield Interlock Switch prevents the machine from operating with the shield open.

The X and Z motion of the machine is performed by Stepper Drive Motors on each axis. There is an adjustable Limit Switch mounted under the chuck to prevent the machine from crashing.

The Spindle Head supports a 1/2 hp DC permanent magnet Spindle Motor.

You can't miss the most important control on the machine; the Emergency Stop button. When pressed, this bright red palm button immediately halts machine operation. To resume operation, the button must be pulled back out. It's important that this button be pressed before performing any manual operations, like changing the stock or the tooling.

Use the Spindle Speed switch to establish the spindle speed. The *min* and *max* positions on the switch are equivalent to approximately 200 (min) to 2,500 (max) RPM. Use the *Computer* position when you want to set spindle speeds from the Control Program.



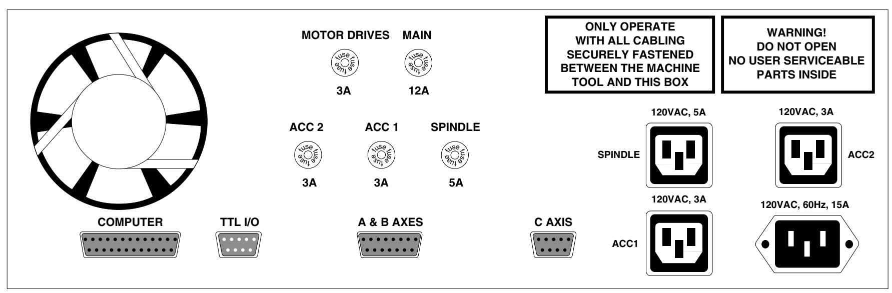
## The Controller Box

The Controller Box houses the power- and interface-related controls for the machine. The power switch and keylock switch are on the front panel of the box. The Key Lock switch keeps unauthorized persons from turning on the machine. Once the Key Lock is in the unlocked position, the Power switch can be turned on. The Power switch lights up when power is on.

The rear panel houses the power and interface connectors, and the fuses. The machine has five fuses; for Main Power, the Spindle, the Stepper Motors, and the two accessory outlets (labeled ACC 1 and ACC 2). The values for these fuses are clearly printed next to the fuse receptacles. *Do not use fuses with amperages other than those shown on the machine.*

The interface connectors are used to interface the Controller Box with the Turning Center and your computer. Cables are provided in the Accessory Kit for this purpose. Refer to Section A for correct installation procedures.

Controller Box Rear Panel



## The Accessory Kit

The Accessory Kit that comes with the Turning Center contains all the tools and hardware necessary for installing and maintaining the Turning Center. It also contains a collet and tools to get you started; other tool holding devices and tools are available as options.

The accessories that come standard with the Turning Center are:

- One indexable Tool Holder
- One carbide profiling tool
- One spindle locking pin
- Five Allen keys (5/32", 3/32", 1/8", 5/64", and 9/64")
- One 25-Pin Cable
- One AC Power Cord

## Available Options

Light Machines Corporation offers a variety of Turning Center accessories, CAM software, curriculum and documentation. For more information about these products, call your local Light Machines dealer, or call Light Machines directly at (800)221-2763 or (603)625-8600.

A convenient listing of products is located on the last page of this User's Guide.

# Maintaining the Turning Center

Performing preventative maintenance, like routine cleaning, lubrication, and correct adjustment, on your spectraLIGHT Turning Center ensures a longer, trouble-free life for the machine.

## Adjusting the Gibs and Spindle

**Note:**

Routine cleaning and lubrication, and correct adjustment of the gibs will ensure a long wear life.

Gibs are fitted at the rear of the saddle and on the right-hand side of the cross slide. A gib is a wedge that is adjusted by loosening the gib locking screws and pushing the gib inward until any play is eliminated. Correct adjustment of the gibs will ensure smooth and steady operation of the slide.

If any end play develops in the main spindle, it can be eliminated by readjusting the preload unit.

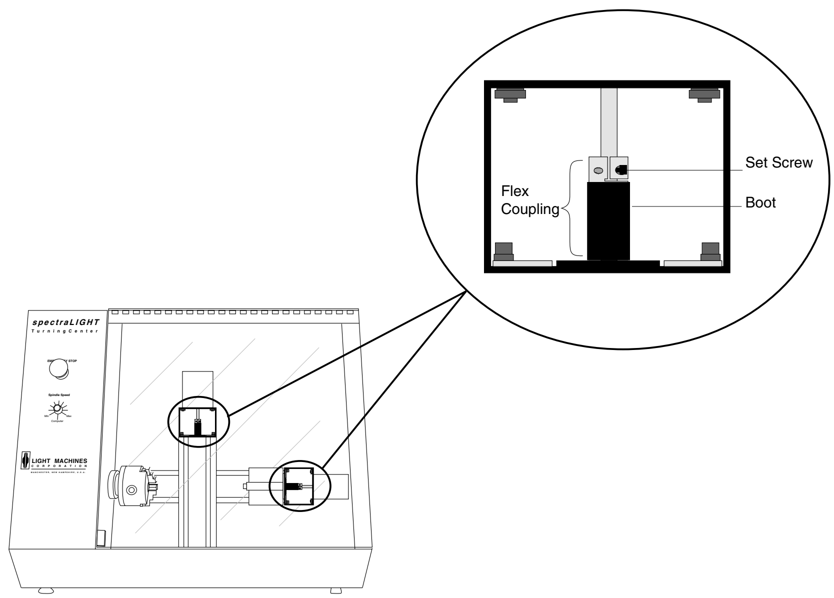
1. Remove the upper safety shield from the Turning Center. Remove the drive belt.
2. Remove the main pulley from the spindle.
3. Loosen the setscrew and tighten the preload nut while turning the spindle until the play is removed.
4. Tighten the preload nut just enough to remove excess play or the bearings may be damaged (0.0002-inch play is recommended).

# Adjusting the Lead Screw Thrust Bearings

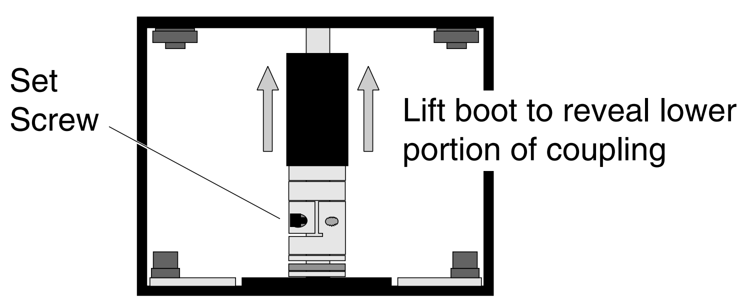
You should periodically inspect the lead screw bearings because they can loosen over time. These brass bearings are mounted on the lead screw next to the flexible couplings on each axis. Tighten the bearings whenever you measure the backlash and it is more than 0.008-inch, or whenever you see a gap between one of the bearings and the cross slide.

## To adjust the lead screw bearings on all axes:

1. There is a black boot covering the lower portion of the flexible coupling. Loosen the set screw that is visible above the black boot.




2. Slide the black boot up to expose the lower half of the flexible coupling.
3. Loosen the set screw on the lower half of the flexible coupling.



## To complete the X axis adjustment:

4. Press the lathe table toward the front of the Turning Center by exerting pressure on the stepper motor.
5. While still pressing the stepper motor, press the flexible coupling in the same direction.
6. While holding the stepper motor and flexible coupling in place, tighten the lower set screw, then release the flexible coupling and stepper motor.
7. Slide the boot down and tighten the top set screw.

## To complete the Z axis adjustment:

4. Pull the cross slide toward the Z axis stepper motor.
5. While still pulling the cross slide, press the flexible coupling in the opposite direction, toward the cross slide.
6. While holding the cross slide and flexible coupling in place, tighten the lower set screw, then release the flexible coupling and cross slide.
7. Slide the boot down and tighten the top set screw.

## Replacing the Cross Slide Saddle Nut

If the backlash is still more than 0.008 inch after you tighten the bearings, you may need to replace the cross slide saddle nut.

To replace the cross slide saddle nut:

1. Remove the X axis stepper motor from its bracket by removing the four screws holding it to the bracket.
2. Turn the flexible coupling to the right to turn the cross slide lead screw. As you turn the lead screw, you can see that it pulls the cross slide in the +X direction (toward the saddle). Turn the lead screw until the front-most edge of the cross slide is flush with the front-most edge of the saddle.

   You should now be able to see the saddle nut. The saddle nut is inserted in a recess in the front of the saddle.
3. The saddle nut is held in place by an set screw. The set screw is embedded inside the saddle directly beneath the saddle nut. The only way to access the set screw is from beneath the front of the saddle. You should be able to probe the underside of the saddle with an Allen wrench, locate the set screw and loosen it to release the pressure on the saddle nut.
4. Once you have loosened the set screw, press the cross slide down toward the front of the Turning Center (in the -X direction). The cross slide lead screw should move with the cross slide and appear out of the recess in which the saddle nut resided.
5. The saddle nut should be threaded onto the end of the lead screw. Unscrew the old saddle nut and screw on a new one (part number L4089).
6. Push the cross slide back into its original position. The lead screw will follow, pulling the new saddle nut into the recess in the front of the saddle.
7. Reach back under the saddle with the Allen wrench and tighten the set screw until you can feel a slight resistance when you turn the lead screw.
8. Turn the flexible coupling to the left to move the cross slide down in the -X direction (enough so you can replace the X axis stepper motor).

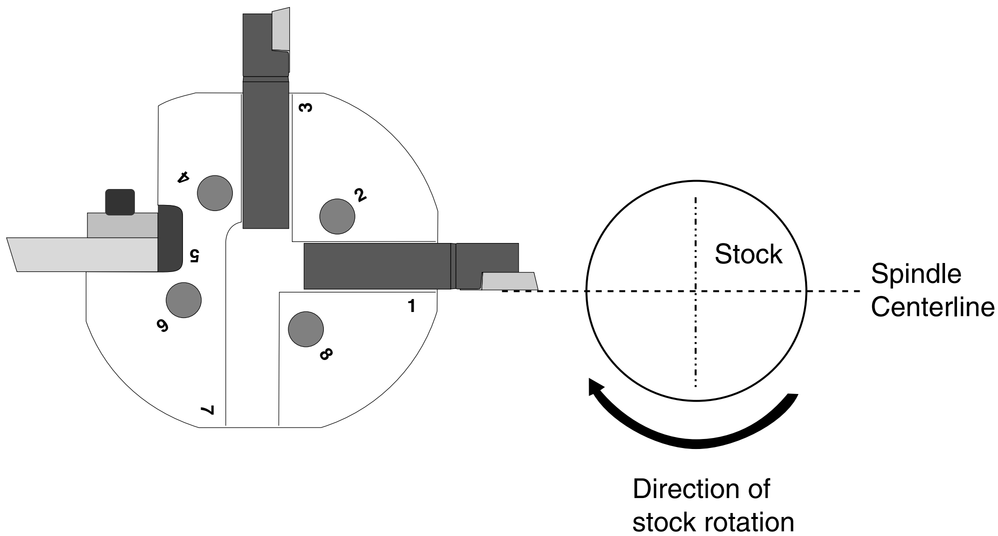
## Resetting Backlash Compensation

Backlash is the amount of play in the lead screws. Backlash compensation removes excess play whenever the machine changes direction. The amount of play in these screws is minimized by adjustments made during assembly and test of the Turning Center at the factory. However, you should still check the backlash compensation after the first four or five hours of use, and every 10 to 20 hours of use thereafter, or whenever you find parts outside acceptable tolerances.

1. Mount a dial indicator on the lathe bed or chip tray with the tip of the indicator touching a tool post or the cross slide. Carefully align the indicator parallel to the axis you are calibrating. Calibrate the X axis first, then the Z axis.
2. Turn on the computer and the Controller Box, and start the Control Program.
3. Select Backlash from the Setup Menu, and set the distance for all axes to zero.
4. Set the speed for all axes to 3ipm (75mm/min.).
5. Use the Jog Keypad to jog 0.020-inch in one direction along the X axis. Take a reading from the dial indicator.
6. Jog 0.020 in the other direction on the X axis and take a second reading from the dial indicator.
7. Calculate the difference between the two readings and subtract this value from 0.020 to get the actual backlash.
8. Select Backlash from the Setup Menu and change the X axis distance to the value found in Step 7.
9. Align the dial indicator on the lathe bed parallel to the Z axis and follow Steps 5 through 8 to determine the actual backlash value for the Z axis.
10. Select Backlash from the Setup Menu and change the Z axis distance to the value found in Step 7.
11. Reactivate the safety interlock on the shield if you have deactivated it.

# Adjusting Turret Tool Heights

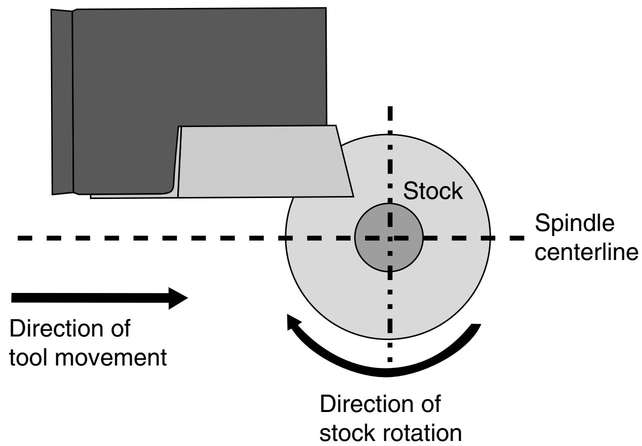
The Automatic Tool Turret (ACC-5601) is an option that allows your Turning Center to make automatic tool changes while running NC programs. It is critical for part tolerances that the tool tip and spindle centerline lie on the same plane.



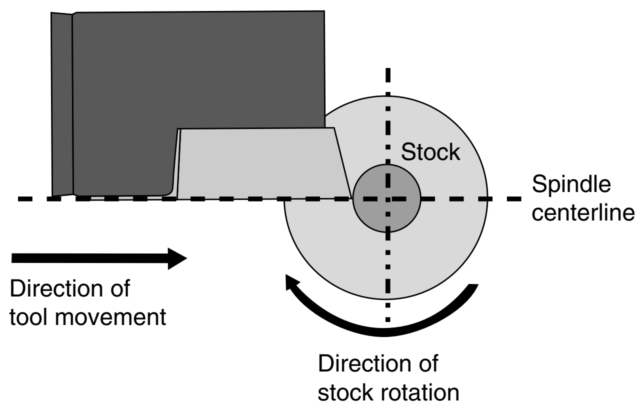

Because different tools may be set up differently, you may need to make some gross adjustments of the tool turret before you can make fine adjustments and start machining parts. There are several steps involved in tool turret adjustment:

1. Mount all the tools you intend to use.
2. Attach a chuck to the spindle and mount a dead center in the chuck. The dead center is used as a reference for the spindle centerline.
3. Rotate each tool as close as possible to the tip of the dead center. Some tools may fall below the spindle centerline, some may fall above the centerline. Some tools may be even with the centerline. Record the distance from the centerline for each tool.
4. If any tools fall below the spindle centerline, place a shim beneath the turret to bring the *lowest* tool even with the spindle centerline. (The shim thickness should be equal to the distance between the lowest tool and the spindle centerline.) This means that the remaining tools will be above the centerline, but that will be remedied by making fine adjustments.

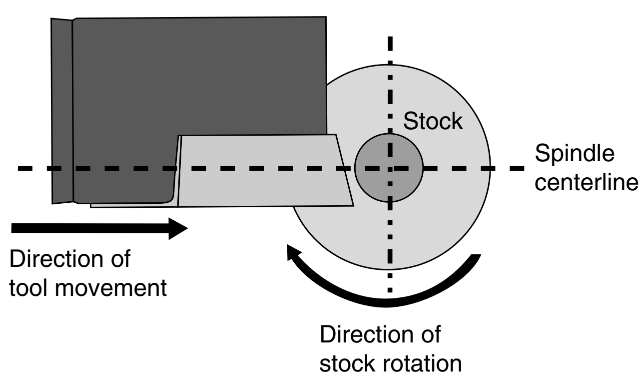
A cutting tool higher than the spindle centerline leaves a small nub on the stock when performing facing cuts.




A cutting tool properly positioned along the spindle centerline cuts the stock cleanly.




A cutting tool lower than the spindle centerline rubs against the workpiece as it approaches the center of the stock.



5. Make the fine adjustments to each tool.

a. Remove the dead center and place a piece of stock in the chuck.

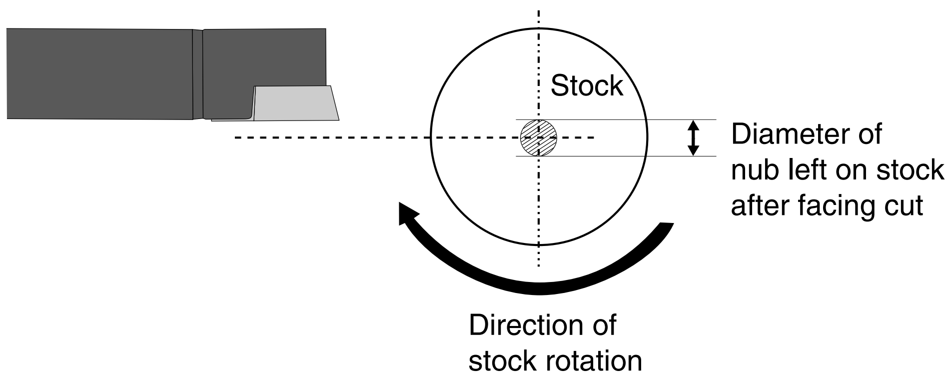
b. Rotate a tool into cutting position and make a facing cut on the stock.

c. Stop the machine just before the tool reaches the spindle centerline. Visually check the position of the tool in relation to the center of the stock.

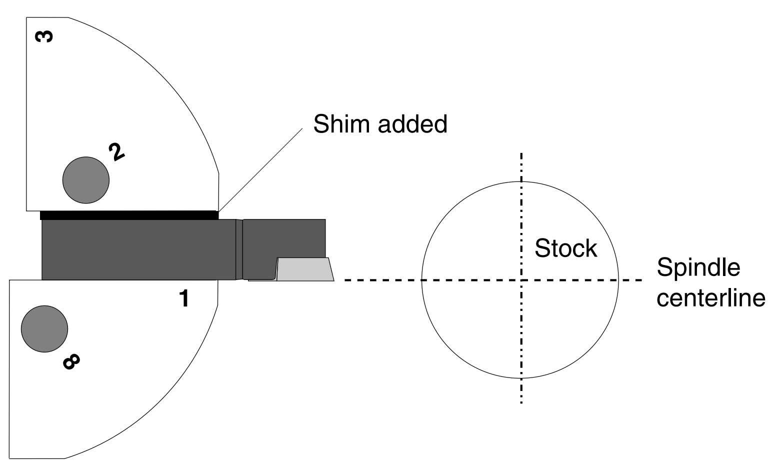
   If the tool is even with the centerline, you do not need to make any further adjustment.

   If the tool is below the centerline, adjust the shim below the turret and begin the fine adjustment process again.

   If the tool is above the centerline, allow the tool to continue past the center of the stock. Stop the machine. There should be a small nub left at the center of the face of the stock. Use calipers to measure the diameter of the nub of material. Divide the measured value by two. The resulting value is the amount that the tool must be shimmed to bring it down to the centerline of the spindle.




Place shims above the tool holder as shown below. The thickness of the shims should be equal to the radius of the leftover nub.

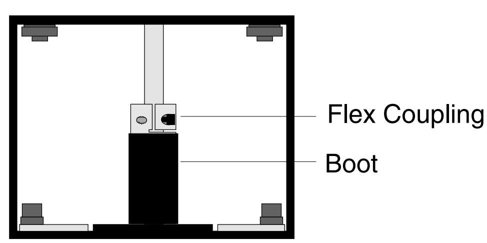


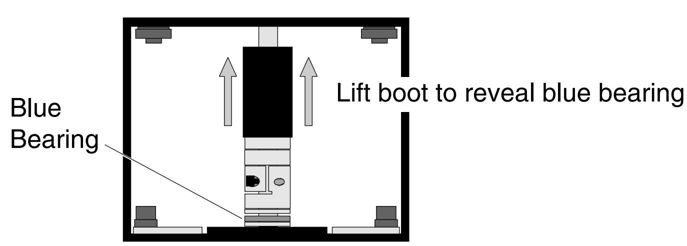
## Lubricating Turning Center Components

If the Turning Center sits idle for two weeks or more, you should clean and oil it to prevent dust buildup. In addition, you should always clean the cross slide, saddle, and bed at the end of each work session. Periodically maintain the following parts:

- **Bed**, **saddle**, **cross slide**. Use a light oil, such as sewing machine oil, on all points where there is a sliding contact. You should do this immediately after each cleanup.
- **Lead screw**, **cross slide screw**. Place a light oil along all threads regularly. At the same time, check that the threads are free of any metal chips. Also put a few drops of light oil between the lead screw bushings and the stationary bushings to minimize wear and increase the capabilities of the drive motors.
- **Tailstock spindle**. On a weekly basis, wind out the spindle as far as it will go and oil it with a light oil.
- **Headstock bearings**. These bearings are lubricated at the factory and sealed to last the lifetime of the machine. They should not need further lubrication if the seals remain unbroken.
- **Thrust bearings**. For every 80 hours of machine use, you should lubricate the bearings on both axes with an all-purpose grease.

  The bearings are located under the flexible couplings inside the stepper motor brackets. Slide the black boot covering the flexible coupling to expose the blue bearing. After lubricating, make sure to slide the black boot back to its original position.



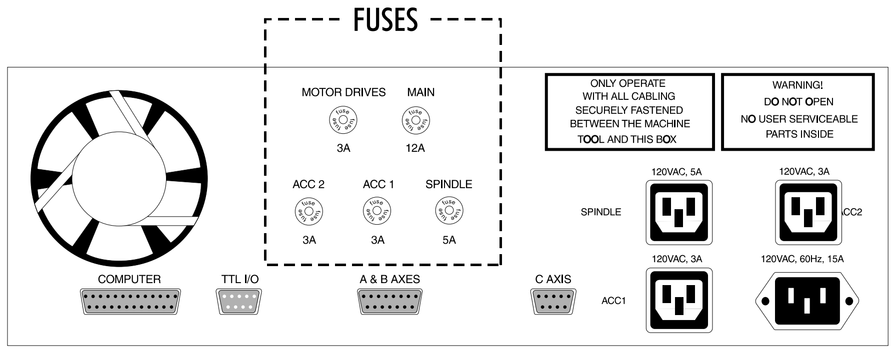



# Maintaining the Controller Box

The spectraLIGHT Controller Box requires no special maintenance, except that it should be kept in a dust-free environment.

If you are having problems with the AC Outlet function, or if the stepper motors are not running, you may need to change a fuse in the Controller Box. A blown fuse can occur if you overload one of the AC outlets. To change a fuse:

Controller Box Fuse Locations




1. Turn the power off, and disconnect the power cords to the Controller Box and the Turning Center.

## WARNING

**Changing Fuses With The AC Power Connected Can Cause Electric Shock!**

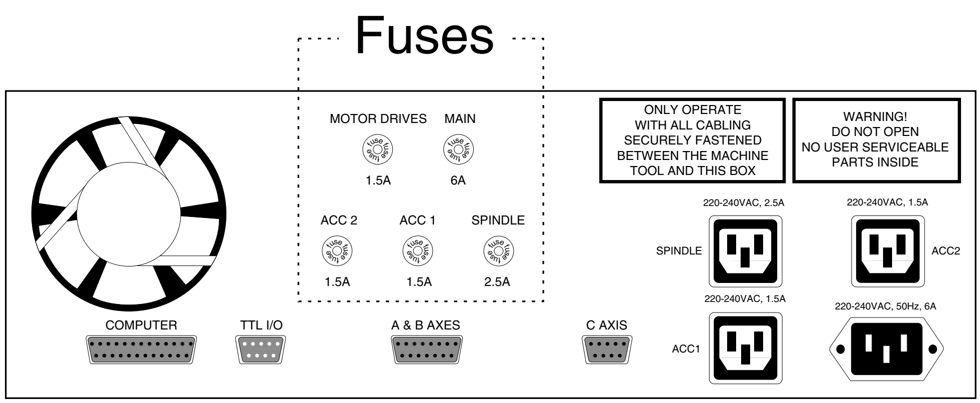
**Always disconnect the main AC power before servicing the Controller Box or Turning Center.**

2. The fuse holders are externally mounted on the rear panel of the Controller Box and are labelled to correspond with each AC outlet. Locate the correct fuse holder.
3. Remove the fuse by turning the fuse cap counterclockwise while pressing slightly inward. The fuse is removed with the cap.
4. Visually inspect the fuse. If the fuse element appears broken, it is blown. You can also use an ohm meter to check the continuity of each fuse.

5. Replace the blown fuse with a standard Slo-Blo fuse of the appropriate rating. Three-amp (1.5A for International Users) fuses are used for the motor drives, ACC 1, and ACC 2. A five-amp (2.5A) fuse is used for the spindle, and a twelve-amp (6A) fuse is used for main AC power.

6. Replace the fuse holder cap by pushing inward and turning clockwise.

7. Reconnect power.




Fuse locations on the International Controller Box

The International Controller Box uses three 1.5A fuses, one 6A fuse and one 2.5A fuse.

# Maintaining the PC in a Shop Environment

Here are a few general guidelines for maintaining your personal computer and software in a shop environment. See your owner's manual for maintenance procedures specific to your computer.

## Caring for the Computer

Follow these general rules for computer care.

- Keep the computer and peripherals (mouse, keyboard, external drive, printer) out of direct sunlight and away from sources of heat and in a relatively clean environment (i.e., not right next to the foundry room).
- Keep liquids (soda, coffee, cutting fluid, grease) away from the computer and peripherals.
- Keep oil, grease, metal chips and excess dust away from the computer, keyboard and floppy disks. You should consider erecting a clear plastic shield between the computer and the lathe to keep chips off the computer.
- Use grounded, three-prong outlets for the computer and peripherals. Take precautions against current overload. A line-surge suppression unit can be purchased at your local computer store to help alleviate this problem.
- Don't block the vent holes in the computer or drives; they are required for air circulation.

## Caring for Floppy Disks

Floppy disks are simple to use but require a few precautions to maintain their integrity.

- Don't touch the magnetic disk part of the disk (the shiny, record-like part inside the disk jacket). Dust or grease from your hand can ruin any part of the disk that you touch and can possibly destroy the entire disk.
- Keep disks in a disk box or special disk container instead of spreading them out on your work space.
- Handle disks gently, don't bend or crease them.
- Don't write on disk labels with a ball point pen. If you must write on the label, always write very lightly with a felt tip pen.
- Keep disks in a clean, cool environment away from excess amounts of dust, heat, or sun.
- Beware of getting machining fluids on the disks. If you spill a liquid or cutting fluid on a disk, it is almost percent certain that the disk and all the data on it can never be used again.
- Keep disks away from all magnetic sources including telephones, high-voltage power sources and lathe motors.
- Make backup copies of all NC program disks each time you update them.
- Print copies of all NC programs in case of disk failure or lost disks.

No matter how cautious you are, disks will go bad; they develop bad blocks (unreadable surfaces). If this happens while you are editing a program, the program will be lost. The solution to this disaster is simple—make backup copies.

# Getting to Know the Control Program

# Starting the Control Program

To start the spectraLIGHT Control Program:

From **Windows 95** select Start from the Task Bar, then select "Progams" and the WSLT folder. In that folder, select WSLT Windows CNC Control. You can also create a Windows 95 shortcut by opening the WSLT directory and dragging WSLT.EXE onto your desktop.

From **Windows NT** double-click the WSLT icon in the WSLT Group.

You should have the controller box connected and powered up before starting the control software, unless you are going to be working in the Simulate Mode. The safety shield should be closed, and the Emergency Stop button pulled out.

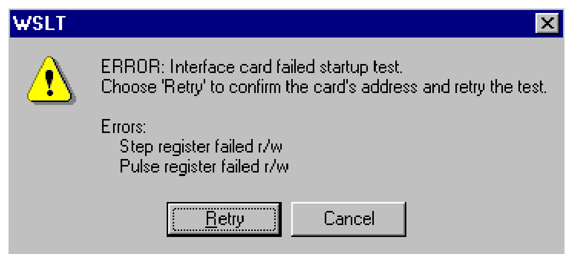
## Starting the Control Program in Simulate Mode

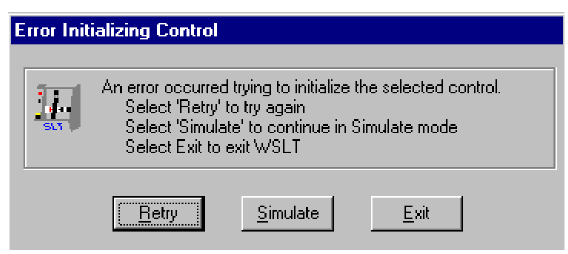
If this dialog box appears when you start the Control Program, the interface card that came with your Turning Center has not been properly installed (see Reference Guide: Section E).

If you do not have the interface card installed, you can still edit and verify NC part programs without the Turning Center by running the Control Program in Simulate Mode.

To start the Control Program in Simulate Mode, select Cancel.

When the next dialog box appears, select Simulate.



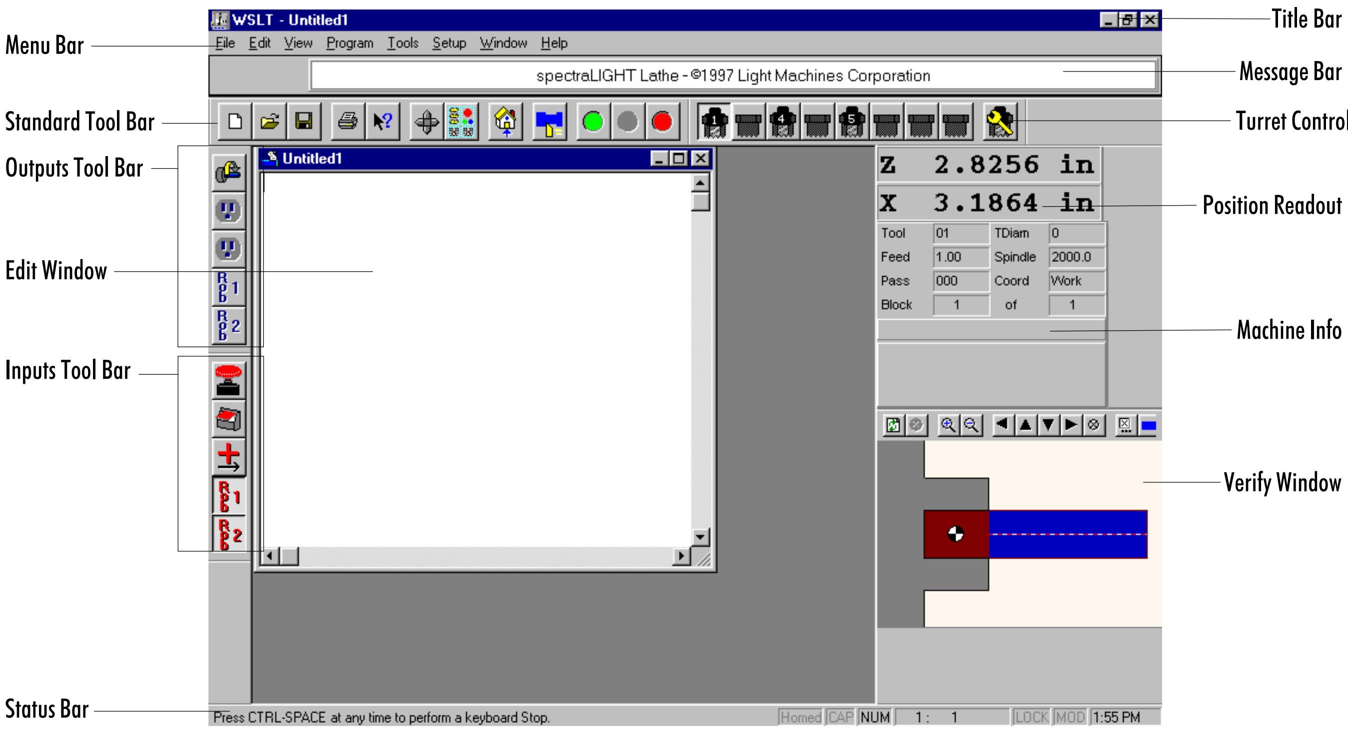
# If You Need Help...

You can access online help by using the commands under the Help Menu, or by pressing F1.

For information on many of the functions and screens in the WSLT software, you can also refer to the Reference Guide: Section E.

# Exploring the Control Program Screen

You should become familiar with the main parts of the Control Program screen before you begin using the Control Program to run NC part programs. The following are the default components that make up the screen.

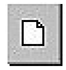
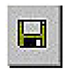
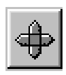
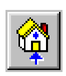
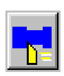
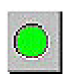
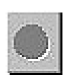
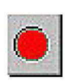



## Menu Bar

The Menu Bar contains all of the menu commands for the Control Program. For an explanation of each menu and its relative commands, refer to the Reference Guide: Section E.

## Standard Tool Bar

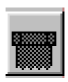
The Standard Tool Bar provides easy access to the most often used commands available in the Control Program.

| | Tool | Function |
|---|---|---|
|  | New | Begin a new NC part program file. |
|  | Open | Open an existing NC part program file. |
|  | Save | Save current NC part program file to disk or drive. |
|  | Print | Send program to printer. |
|  | Context Help | Obtain help on selected object. |
|  | Jog Control | Access Jog Control Panel. |
|  | Operator Panel | Access Operator Control Panel. |
|  | Home | Open Set/Check Home dialog box. |
|  | Verify | Verify the current NC part program. |
|  | Run (green) | Run the current NC part program. |
|  | Pause (yellow) | Pause the currently running NC part program. |
|  | Stop (red) | Halt the currently running NC part program. |

# Turret Control Toolbar

In order to make use of this toolbar, the use of the optional Tool Turret must be specified in the Setup Program. See page A-12 for directions on setting the Control Program Software for the Tool Turret.

The Turret Control Toolbar configures the optional Tool Turret by specifying which tool is in which station to make tool changes more accessable. To change the tool, activate one of the eight Tool Station buttons. The tools must be defined in the Tool Library dialog box which an be accessed through the Tool menu.

| Tool | | Function |
|---|---|---|
|  | Tool Station (1) | Applies the tool in the Station (1) to the NC program. |
|  | Tool Station | Undefined Tool Station. |
|  | Configure Turret | Configures which tool will be placed in each station. |

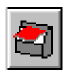
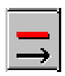
# Outputs Tool Bar

The Outputs Tool Bar is an active tool bar. It provides switches to supply power to the spindle, and to the Accessory outlets on the Controller Box. Switches for Robotic outputs 1 and 2 are also provided. Power is on when the buttons are depressed.

| Tool | | Function |
|---|---|---|
|  | Spindle Output | Provides power to the spindle. |
|  | Acc1 Output | Provides power to the Acc1 outlet on the Controller Box |
|  | Acc2 Output | Provides power to the Acc2 outlet on the Controller Box |
|  | Robot 1 Output | Provides power to the TTL I/O connector on the Controller Box for Robotic Output 1 (Reference Guide: Section L). |
|  | Robot 2 Output | Provides power to the TTL I/O connector on theController Box for Robotic Output 2 (Reference Guide: Section L). |

# Inputs Tool Bar

The Inputs Tool Bar is an inactive tool bar. It provides information only on the state of the Emergency Stop, the Safety Shield, and the negative limit switch. Indicators for Robotic inputs 1 and 2 are also provided. An input is active (on) when the button is depressed.

| Tool | | Function |
|---|---|---|
|  | E-Stop | Indicates when the Emergency Stop is pressed. |
|  | Safety Shield | Indicates when the Safety Shield is open. |
|  | Negative Limit | Indicates when the Negative Z Limit switch has been hit. |
|  | Robot 1 Input | Indicates when Robotic Input 1 (Reference Guide: Section L) is on. |
|  | Robot 2 Input | Indicates when Robotic Input 2 (Reference Guide: Section L) is on. |

## Edit Window

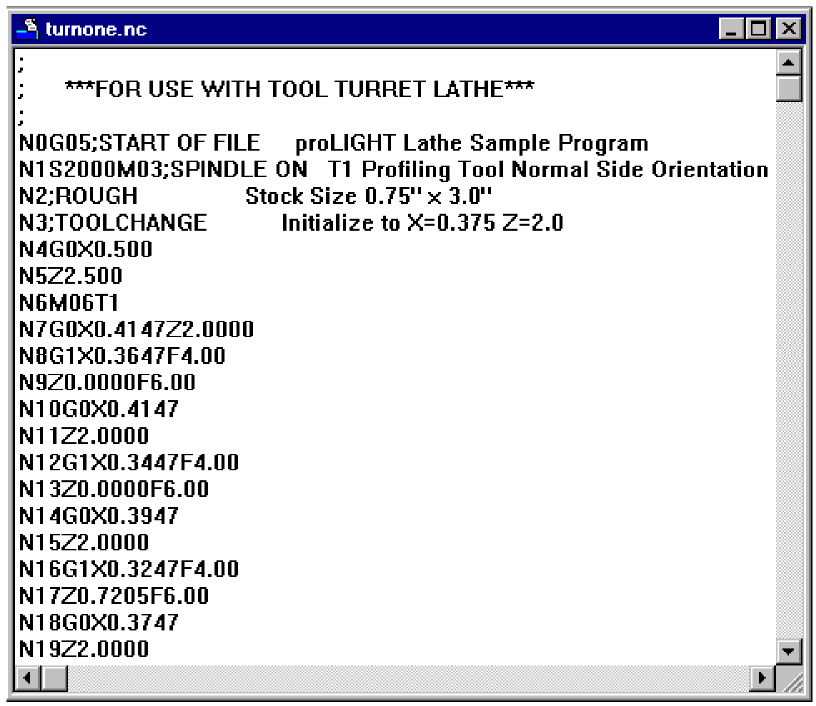
**IMPORTANT!**

<u>Always</u> verify NC programs after editing to ensure that your changes will not cause a tool crash!

Whenever you open an NC part program file it appears in its own edit window. These windows have the same characteristics as other Windows 95 windows (scroll bars, minimize/maximize buttons, etc.). You can have multiple edit windows open at a time, the maximum number of which depends on available memory.

By default, each new window is locked; you can not edit a locked window. To unlock the window, use the Lock command under the Edit Menu.

Here is how the edit window for the NC part programTURNONE.NC appears.

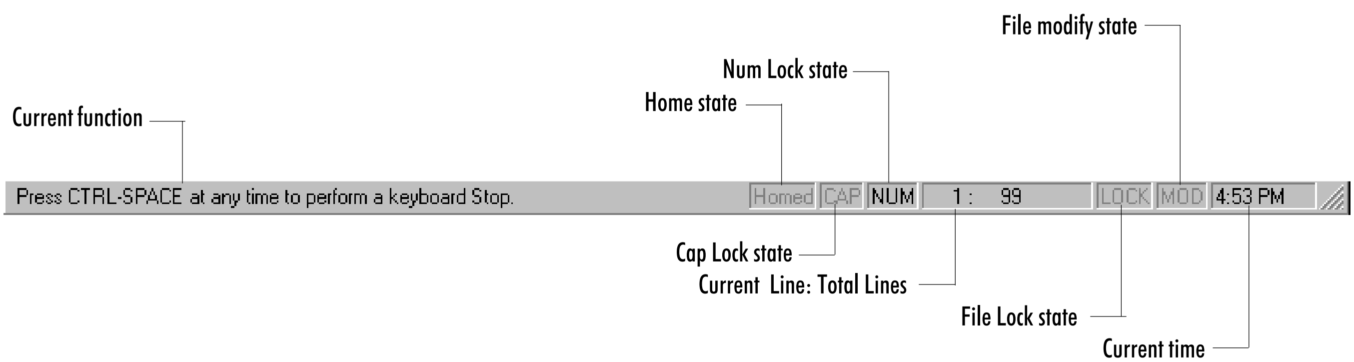


## Status Bar

The left side of the Status Bar provides information about the currently selected function. The right side of the status bar provides information on:

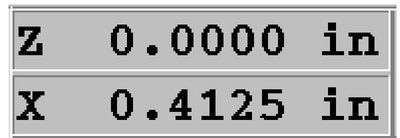
- Whether or not the Turning Center is homed
- Whether or not the Caps Lock key is activated
- Whether or not the Num Lock key is activated
- The current line and total number of lines in the program
- Whether or not the current NC part program is locked
- Whether or not the current NC part program has been modified
- The current time according to your computer

When the indicator is dimmed, the function is in the off condition.



## Position Readout

The Position Readout provides information on the current X and Z coordinates of the tool position. The units of measure in the Position Readout are determined by the Units command under the Setup Menu.




Tips:

◆You can double-click on the Position Readout window to bring up the Go To Position dialog box.

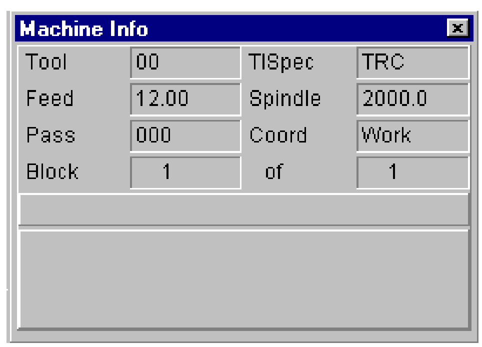
◆You can move the Position Readout window to another location on the screen. Hold Ctrl down, then click and drag the window. Once you release Ctrl and the mouse key, the window stays in the new position. If you move the window back to the docking area, it will automatically dock.

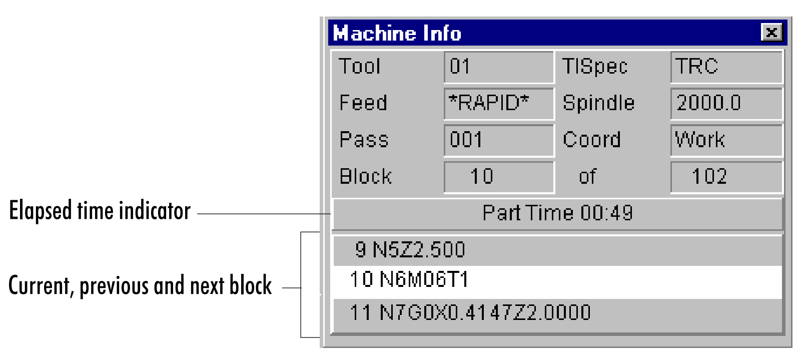
◆ If you want the window to remain floating, click the right mouse button on the window, and uncheck "dockable." The window can only be resized when it is not dockable.

◆For more information on moving, resizing and docking windows, refer to the Reference Guide: Section E.

## Machine Info Panel

The Machine Info Panel provides information on the current tool, tool reference point, feed rate, spindle speed, number of passes made, coordinate system in use, as well as the current block and total number of blocks in the program.




When a part program is running, the Info Panel also provides dynamic display of the elapsed machining time and highlights the block of code that is currently being executed.

Tips:

◆You can move the Machine Info Panel to another part of the screen. Hold Ctrl down, then click and drag the panel. Once you release Ctrl and the mouse key, the panel will stay in the new position. If you move the panel back to the docking area, it will automatically dock.

◆ If you want the panel to remain floating, click the right mouse button on the panel, and uncheck "dockable." The panel can only be resized when it is not dockable.

◆For more information on moving, resizing and docking panels, refer to the Reference Guide: Section E.

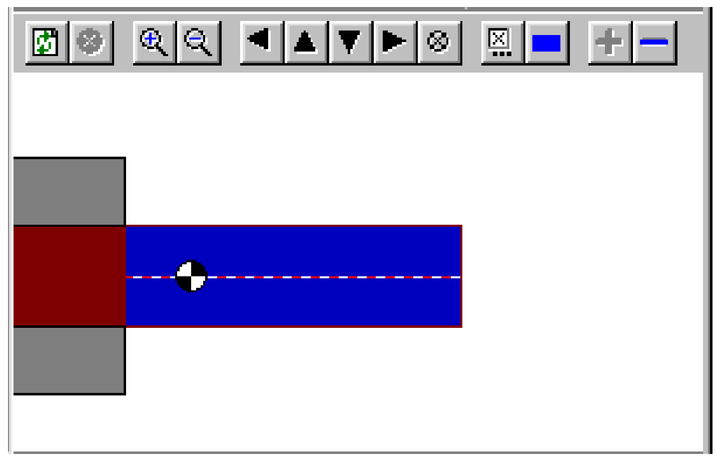
Tips:

- You can double-click on the window to bring up the Verify Setup dialog box.
- You can move the Verify Window to another part of the screen. Hold Ctrl down, then click and drag the window. Once you release Ctrl and the mouse key, the window will stay in the new position. If you move the window back to the docking area, it will automatically dock.
- If you want the window to remain floating, click the right mouse button on the window, and uncheck "dockable." The window can only be resized when it is not dockable.
- For more information on moving, resizing and docking windows, refer to the Reference Guide: Section E.

# Verify Window

The Verify Window displays a simulation of your part program when you select the Verify command from the Program Menu, or when you click the Verify Program button on the Standard Tool Bar.

Many elements in the Verify Window can be altered according to your preferences in the Display section of the Verify Setup dialog box. The view of the workpiece can be centered, zoomed in or out, color coded and instantly updated after the window is resized.



User Guide: Section D

# Tutorial: Machining a Sample Part

Safely Running the Turning Center

Running a Sample NC Program

# Safely Running the Turning Center

Like any other power tool, the spectraLIGHT Turning Center is a potentially dangerous machine if operated in a careless manner. The importance of safely operating the spectraLIGHT Turning Center, including the need for protection against personal injury and the prevention of damage to the equipment, can not be stressed enough. You will find more information on safe machining in the Reference Guide: Section J.

## Safety Rules

Proper setup of the Turning Center is essential for safe machining. These procedures must be followed each time a new tool is mounted. General setup requirements for the Turning Center include checking components for cleanness and lubrication, mounting the cutting tool, setting the limit switch, mounting the workpiece, and setting the spindle rotation speed.

The following safety rules should be practiced by all operators of the spectraLIGHT Turning Center for each use.

### Checking Lathe Components

Always examine the bed saddle, cross slide and lead screw to be sure they are free of shavings and particles from previous operations. Remove such debris from the lathe to avoid possible binding of components which may result in possible damage to the lathe, the workpiece or the operator.

Always make sure the machine is properly lubricated.

### Remove Adjusting Keys and Wrenches

Make a habit of checking that keys and adjusting wrenches are removed from the Turning Center before operating the machine.

### Do Not Force a Tool

Select the feed rate and depth of cut best suited to the design, construction and purpose of the cutting tool. It is always better to take too light a cut than too heavy a cut.

### Use the Right Tool

Select the type of cutting tool best suited to the turning operation. Don't force a tool or attachment to do a job it wasn't designed to do.

## Mounting the Cutting Tool

Each cutting tool used in the machining operation must be sharp and tightly inserted in the tool post. The cutting edge of the tool must be on the centerline or just below the centerline (0.004  inch or 0.1mm maximum) of the axis of rotation of the lathe.

## Secure the Workpiece

The workpiece can be held in a three or four jaw chuck, or with a collet. Be certain that you have firmly secured the workpiece on the cross slide and the cutting tool in the spindle before turning on the spindle motor.

## Turn the Spindle By Hand Before Starting

Manually turning the spindle allows you to safely determine that the tool will not hit the Turning Center bed, cross slide, or stock on start up.

## Set the Spindle Rotation Speed

The spectraLIGHT Turning Center is equipped with an electronically-controlled spindle motor which produces a comprehensive range of spindle rotation speeds. Speed can be set with the Control Software or by using an S code in the NC program. With the Control Software select Spindle from the Setup menu, and adjust the spindle speed. The spindle can also be turned on or off from this dialog box.

## Tighten All Holding, Locking and Driving Devices

Tighten the work holders and tool holders. Do not overtighten these devices. Overtightening may damage threads or warp parts, thereby reducing accuracy and effectiveness.

## Making Emergency Stops

Before you run the spectraLIGHT Turning Center for the first time, you should know how to stop the machine should an emergency situation arise. There are a number of ways an emergency stop can be initiated on the Turning Center: by pressing the Emergency Stop button, by simultaneously pressing the Control and Space Bar keys on the computer keyboard, by activating one of the limit switches, or by activating the safety shield interlock switch.

**Note:**

You should use the Emergency Stop button to disconnect power to the Turning Center when changing tools, or when mounting or removing a workpiece.

Place a Pause command and a Tool Change command (M06) in your part program. Once the pause is executed, push in the Emergency Stop button and open the Turning Center shield.

When you are finished with the changes, close the shield. Pull out the Emergency Stop button, then press Enter on the computer keyboard to resume running the program.

### Stopping with the Emergency Stop Button

There is an Emergency Stop button located on the front panel of the Turning Center; it has an oversized red cap. Before power can be applied to the Turning Center, the Emergency Stop button must be pulled fully out from the front panel. The full-out position allows power to be supplied to the Turning Center. The Emergency Stop button disables the spindle even if the computer is turned off.

In the event that a tool crashes into the workpiece, you can immediately kill power to the Turning Center by pushing in the Emergency Stop button. The Emergency Stop button should be your first target in an emergency situation. Pushing in the Emergency Stop button terminates the part program. Wait until the Turning Center has completely stopped moving before opening the safety shield.

When the tool crash has been cleared and the Emergency Stop button is reset (pulled back out), edit the part program to remove the cause of the tool crash before running the program again. Reset the tool position using the Set Position command from the Setup Menu.

### Stopping with the Computer Keyboard

The execution of the part program can be interrupted by pressing keys on the computer keyboard. Unlike using the Emergency Stop button, this method of stopping the Turning Center does not cause the software to lose track of the tool position.

To stop the part program with the keyboard, press the Control key and Space Bar simultaneously. The cutting stops immediately and the cutting tool remains in position. To restart the program from a keyboard-generated stop, select the Run/Continue command from the Program Menu. In the Start At Line box, enter the number of the last line executed, then click on the Run Program button. (When you stopped the program, the last line executed is displayed on the Machine Info Panel, and the cursor is placed on the same line in the file.)

## Stopping with a Limit Switch

The spectraLIGHT Turning Center is equipped with a limit switch on the Z to sense the end of travel in the negative Z direction (i.e., when the tool travels left towards the work piece). If the Z axis travel exceeds the end of travel, the limit switch is activated and shuts down machine operation.

Once the limit switch is activated, the tool must be jogged away from it using the Jog Control Panel (see the Reference Guide, Section E). To move the spindle away from the limit switch, you must jog it in the opposite direction, in the positive Z direction.

If the spindle comes close enough to the end of travel to activate the limit switch, the following procedure must be followed to restore normal operation.

1. Select **Jog Control** from the View Menu or from the Standard Tool Bar.
2. Click on the appropriate jog key on the jog keypad, to move the spindle away from the triggered limit switch.
3. Check your initial machine set up to make sure it was performed correctly.

# Running a Sample NC Program

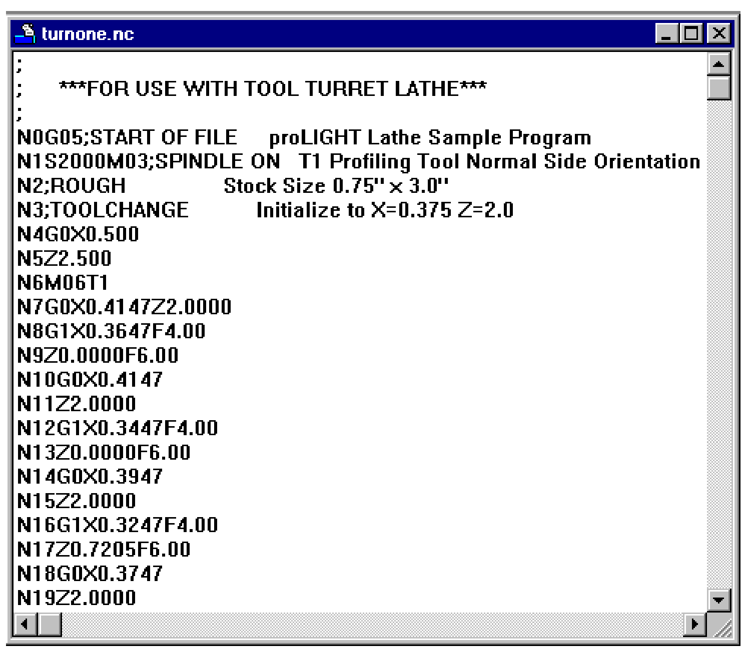
When you installed the spectraLIGHT Control Program an NC part program file, named Turnone.nc, was copied into the WSLT/SAMPLES directory along with the other files. The Turnone.nc program is meant to turn a 3" (length) x 0.75" (diameter) cylindrical piece of machinable brass, aluminum, Delrin or wax. You will be using this file to create your first workpiece on the Turning Center.

**WARNING**

**Do not attempt to operate the spectraLIGHT Turning Center without reviewing all of the safety precautions set forth in the Reference Guide: Section J.**

## Open Turnone.nc

1. Select the **Open** command from the File Menu, or click on the Open button on the Standard Tool Bar. The Open dialog box appears.
2. Double-click on the **Turnone.nc** file name, or click on the file name then click on the **Open** button. The edit window for Turnone.nc appears.


turnone.nc

```
;
;     ***FOR USE WITH TOOL TURRET LATHE***
;
N0G05;START OF FILE     proLIGHT Lathe Sample Program
N1S2000M03;SPINDLE ON   T1 Profiling Tool Normal Side Orientation
N2;ROUGH              Stock Size 0.75" x 3.0"
N3;TOOLCHANGE          Initialize to X=0.375 Z=2.0
N4G0X0.500
N5Z2.500
N6M06T1
N7G0X0.4147Z2.0000
N8G1X0.3647F4.00
N9Z0.0000F6.00
N10G0X0.4147
N11Z2.0000
N12G1X0.3447F4.00
N13Z0.0000F6.00
N14G0X0.3947
N15Z2.0000
N16G1X0.3247F4.00
N17Z0.7205F6.00
N18G0X0.3747
N19Z2.0000
```

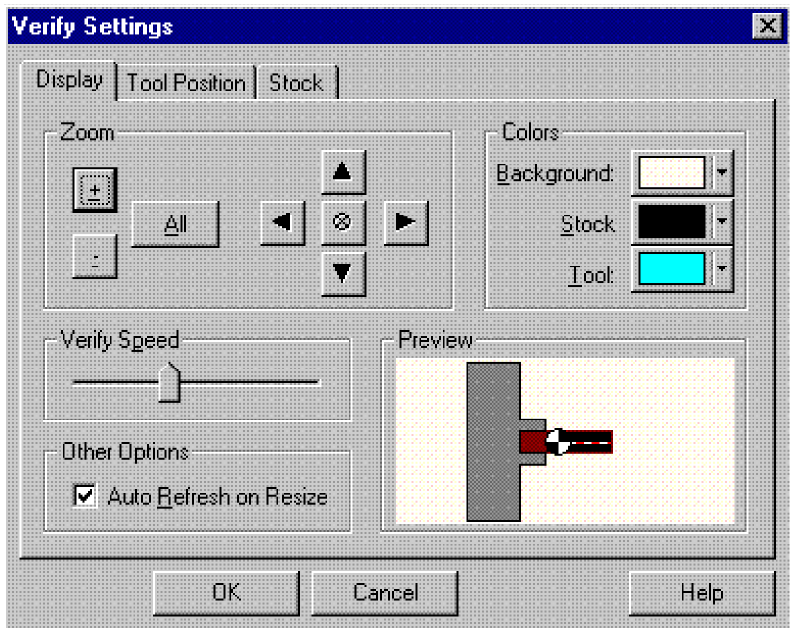
# Adjust the Verify Settings

After opening the NC program, you need to adjust the Verify Settings for the part you are about to turn. To view the **Verify Settings** dialog box, select **Verify Window** from the View Menu; double click on the Verify window when it appears. Alternatively, you may select **Verify Settings** from the Setup menu.

The Verify Settings dialog box allows you to alter the viewpoint of the tool and workpiece in the Display Window.

The Display panel allows you to:

- Zoom in, zoom out or fill the window by selecting All
- Center the workpiece in the Verify Window
- Decide which colors will indicate the background, stock, and tool in the Preview Window
- See which settings you have chosen for the Verify Program in the Preview Window
- Choose a speed at which the verify process will be executed in the Verify Window
- Automatically update the display in the Verify Window after you have resized the window by checking this option

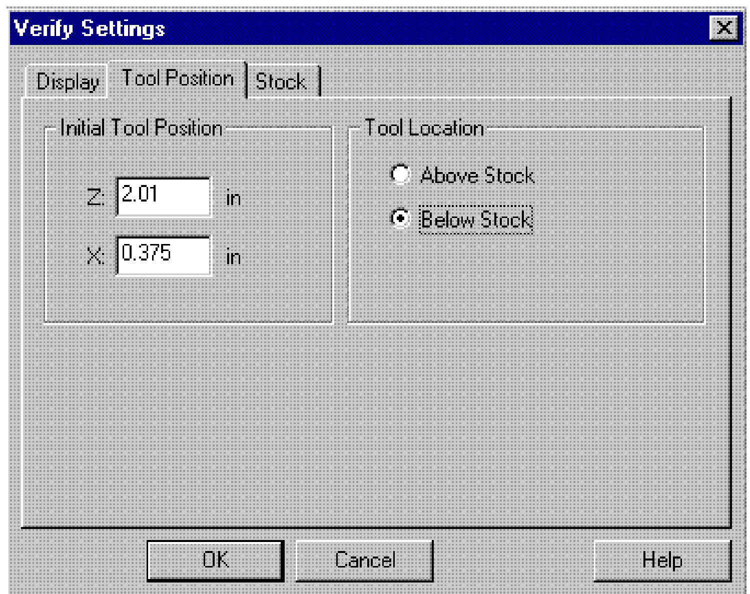


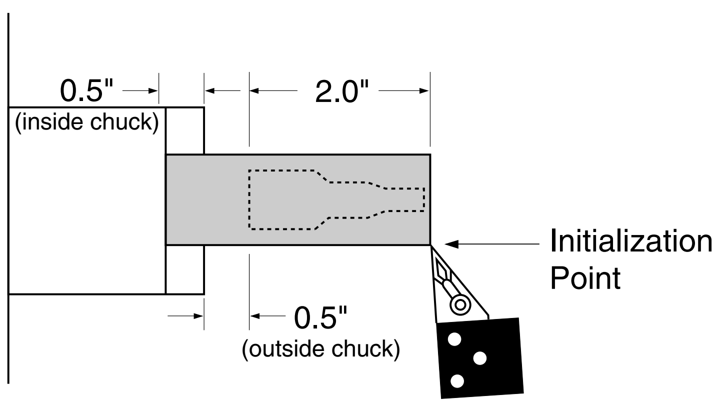

# Adjust the Display

1. Select the **Display** tab.
2. Use the **Zoom** controls to alter the size of the workpiece, and to position the workpiece in the Verify window, as shown in the Preview Window.
3. Use the **Verify Speed** control to alter the speed at which the verify program will be executed in the Verify Window. If you would like to view the verify program in slow motion, move the slider to the left. If you would like to view the verify program at a faster speed, move the slider to the right.
4. Use the **Colors** controls to choose the color indicators for the background, stock and tool in the Verify Window.
5. Check **Auto Refresh on Resize** so that you may adjust the Verify Window size if necessary without disturbing the Verify program.

## Adjust the Tool Position

1. Select the **Tool Position** tab.
2. Enter the **Initial Tool Position** for the Turnone.nc part program. The positions are Z=2.01" and X=.375". These points are based on the origin and the size of the stock and are always positive values.
3. Select **Below Stock** for the **Tool Location** for this sample NC program. If you have a tool turret installed, you will need to select **Above Stock**.




Initial Tool Position

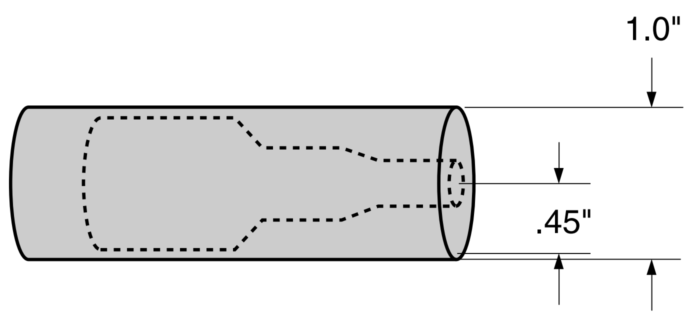
## Determining the Stock Size

1. Check your NC part program for the maximum X value. This value is used to determine the required diameter of the workpiece. For instance, if the maximum X value in your program is 0.45 inches, you will need a workpiece with a 1-inch diameter.

   When writing a part program for a turned part, the maximum X value in the program is equal to the radius of the largest portion of the finished part. Multiply the radius of the largest portion of your finished workpiece by 2 to approximate the required diameter of the workpiece.

2. Check your NC part program for the maximum Z value. This value is used to determine the required length of the workpiece. If the maximum Z value in your program is 1.9 inches, you would assume that you need a workpiece 2 inches long. In addition, you have to account for the stock held inside the chuck by adding 0.5 inch to the workpiece length. Also, you should add another 0.5 inch so that the tool does not hit the chuck.

   Therefore, to turn a part with a finished length of 1.9 inches, you need a workpiece thatis 2" + 0.5" + 0.5", or 3 inches long. When entering this stock size in the stock window, only the usable dimensions are defined.

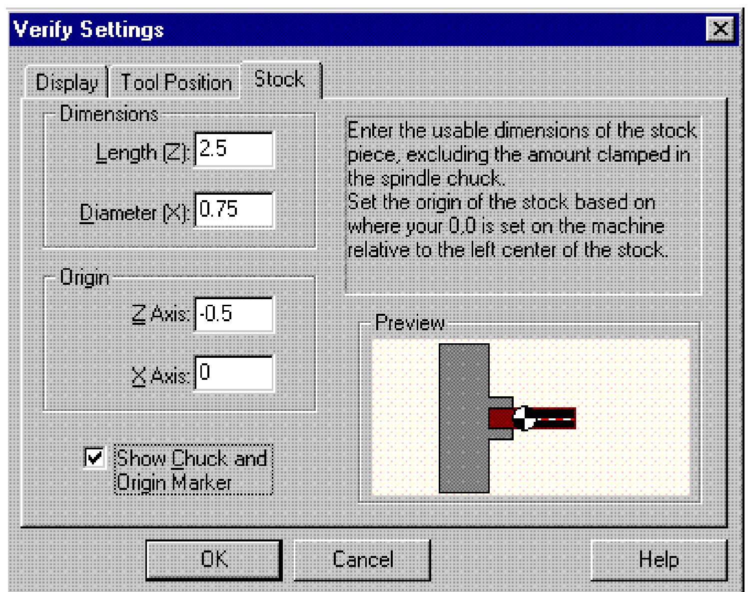


## Adjust the Stock

1. Select the **Stock** tab.
2. Enter the stock **Dimensions** for the Turnone.nc part program. The stock dimensions are Z=2.0" and X=0.75". These dimensions should define the usable dimensions of the stock, excluding the amount clamped in the spindle chuck.
3. Set the **Origin of Stock** to Z=0.5" and X=0". The origin of the stock is based on where your 0,0 point is set on the machine, relative to the left center of the stock.
4. Check the box labeled **Show Chuck and Origin Marker** to view these items in the **Preview** and **Verify Windows**.
5. Select **OK**. The dialog box closes, and your changes are applied to the workpiece in the Verify Window.

The Stock panel allows you to:

- Enter the dimensions of the workpiece
- Set the point of origin for the workpiece
- Choose to view the Chuck and Origin Marker in the Preview and Verify Windows

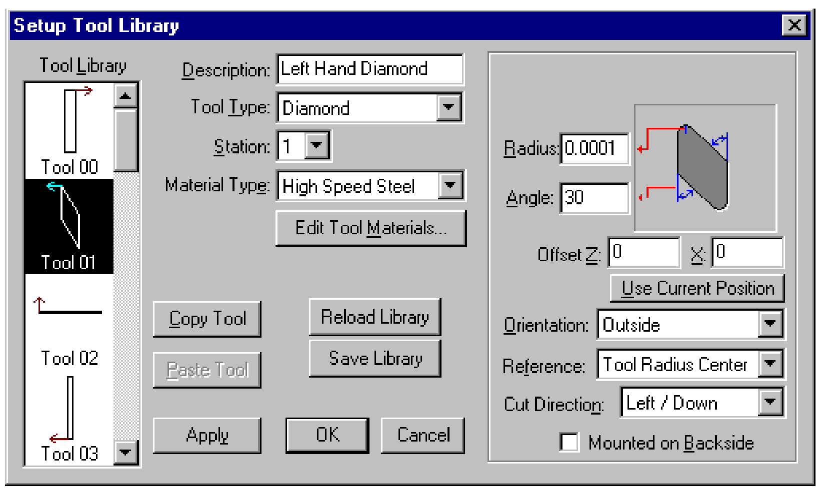


# Define the Tool

To turn this part, you will use a diamond profiling tool. You will use the parameters for this particular tool for the tool path verification as well. To define the tool parameters, first select the tool from the tool library, then select the tool for verification.

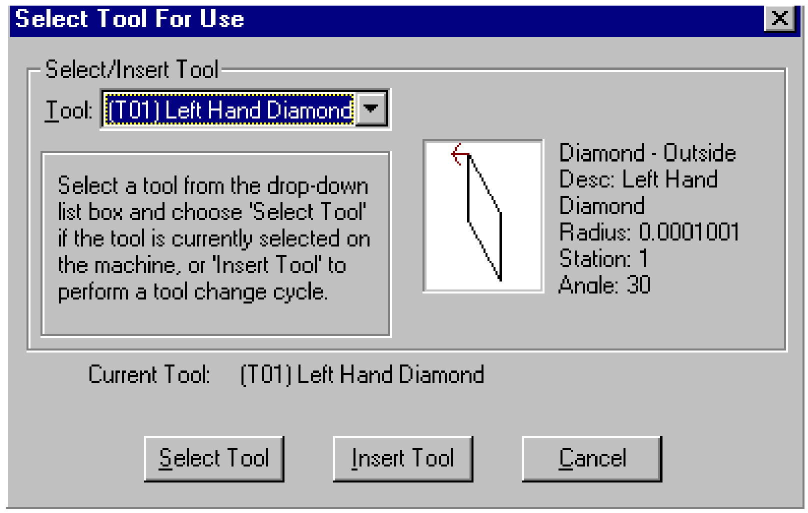
## Select the Tool from the Library

1. Select Setup Library from the Tools Menu. The Setup Tools dialog box appears. There are already a number of tools defined. From these, select Tool 01, a left hand diamond profiling tool with outside orientation.
2. Click on the **Apply** button.
3. Click on **OK** to exit the Tool Library.



## Select the Tool for Verification

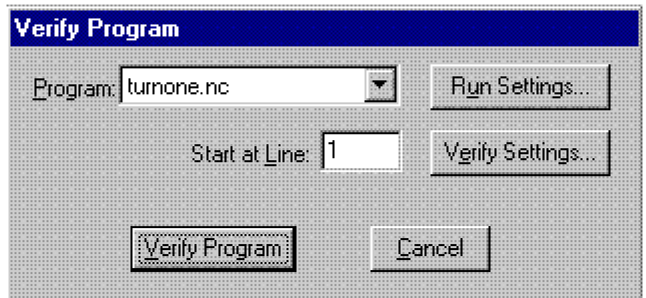
1. Select the **Select Tool** command from the Tools Menu. The Select Tool for Use dialog box appears.
2. Select the tool you previously defined by using the Tool pull-down list.
3. Click on the **Select Tool** button.



## Verify Turnone.nc

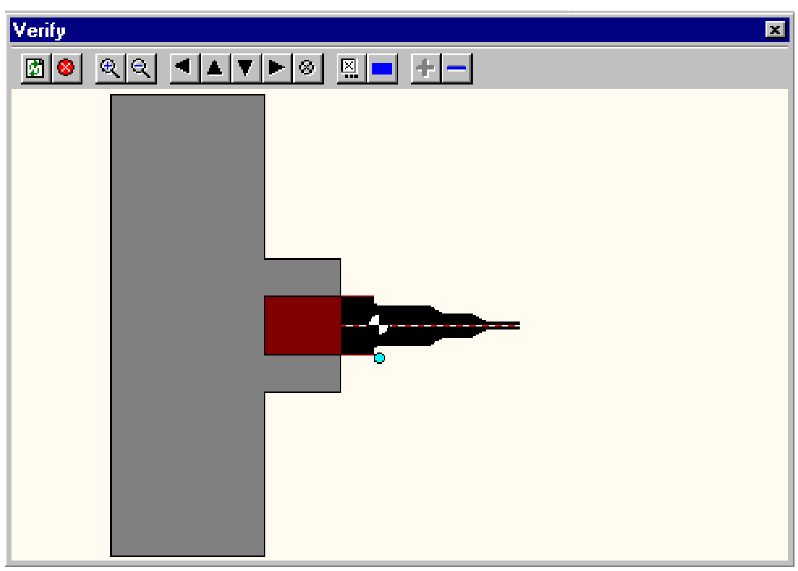
Tool path verification allows you to check for programming errors before actually running the part program on the Turning Center.

1. Select Verify from the Program Menu or from the Standard Tool Bar. The Verify Program dialog box appears. The default starting line for the program is Line 1. When verifying a program for the first time, you should begin on Line 1.




2. Click on the Verify Program button, then watch the Verify Window. You will see the Turnone.nc program executed on the cylindrical workpiece. Turnone.nc has a programmed pause. The operator must press **Cycle Start** from the Operator Panel or **Go** from the status line to see the entire verification process.

3. After verification is completed, the Normal Program Stop dialog box is displayed. Select OK.



## Dry Run the NC Program

Before you run your part program for the first time, you should perform a dry run (run the program with no stock mounted). This will ensure that all the movements of the Turning Center make sense and that the tool is in no danger of striking any fixtures of the Turning Center. Although you should dry run the program with no stock mounted, set the point of origin using the workpiece and then remove it.

**Note:**

To turn the spindle speed down, select the Operator Panel, and adjust the spindle speed to 0%. As the program runs you may want to increase the speed.

Begin with the Emergency Stop button pressed in, and the spindle speed turned all the way down. The tool should be mounted in the tool post.

1. Mount the workpiece in the chuck. Your workpiece should be a cylindrical piece, 3" by 0.75" diameter.
2. Close the Safety Shield and pull out the Emergency Stop button.
3. Select Jog Control from the View Menu (or the Standard Toolbar). The Jog Keypad appears.
4. Use the Jog Keypad to jog the tool to the bottom of the front right corner of the workpiece (the end of the stock furthest from the chuck and at the edge of the stock). If a Tool Turret is installed, jog the tool to the top of the front right corner of the workpiece, instead of the bottom.

**To jog the tool:**

Click on the appropriate axis buttons on the Jog Keypad.

The Enable button allows the arrow buttons on the computer keyboard to also be used to jog the tool. Once a button on the Jog Control Panel has been activated, the Enable button is automatically activated.

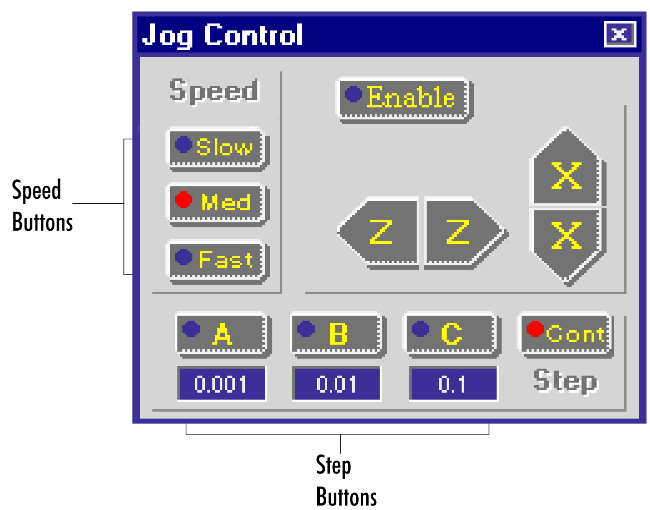
The tool moves at the speed and distance selected using the Speed and Step buttons.

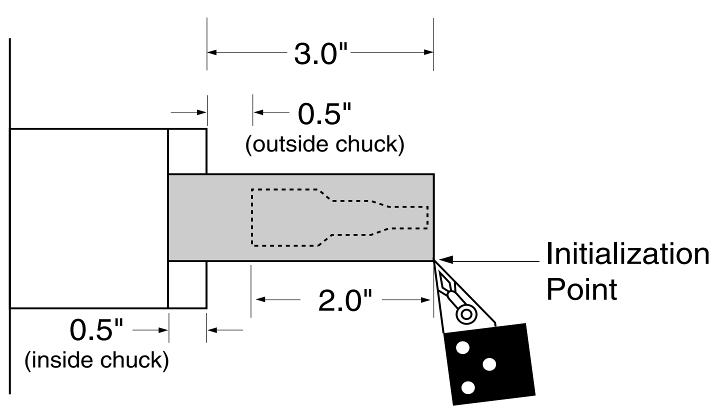
The Speeds and Steps (distances) on the Jog Keypad are defined by the Jog Settings command under the Setup Menu.

To move the tool in a continuous motion, select Cont. The tool will continue to move as long as the axis button is depressed.

**Note:**

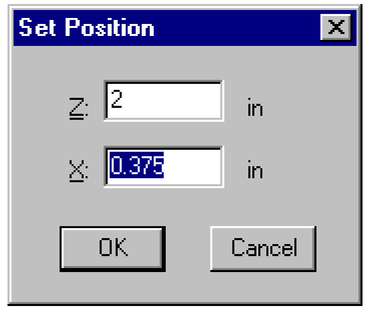
Double-clicking on the Jog Keypad opens the Jog Settings dialog box.



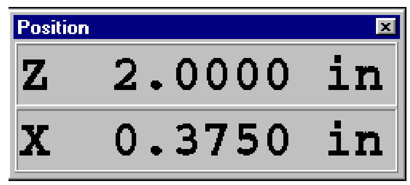



5. Select **Set Position** from the Setup Menu. The Set Position dialog box appears.

6. Enter  Z= 2.0 and X= 0.375.




7. Click on **OK**. The values in the Position Readout all change to zero.

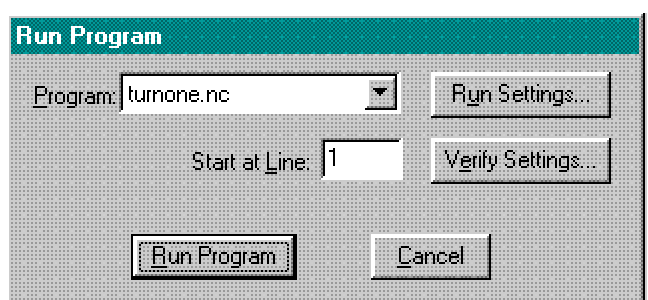



8. Jog the tool up and away from the workpiece. Press the Emergency Stop button, open the Safety Shield and remove the workpiece.

9. Return the Safety Shield to the closed position and pull out the Emergency Stop button.

10. Put on a pair of safety glasses and complete the Safety Checklist (refer to the Reference Guide: Section J).

11. Select **Run/Continue** from the Programs Menu. The Run Program dialog box appears.




12. Click on the **Run Program** button.

13. As the part program runs, observe the tool motion in relation to the chuck (and eventually the workpiece). Look for signs of a possible tool crash and be prepared to press the Emergency Stop switch on the Turning Center. Edit the program as required. When you are satisfied that the tool motions are correct, you can mount the workpiece.

## Mount the Workpiece

1. Using the Jog Keypad, jog the tool post up and out of the way.
2. Before mounting the workpiece, push the Emergency Stop button in.
3. Mount the 3"x.75" workpiece in the chuck. Take care to position the workpiece perpendicular to the tool post.
4. Pull the Emergency Stop button out.
5. Jog the tool to position the center of the tool tip at the top of the front, right corner of the workpiece. Jog until the tip of the end mill touches the surface of the workpiece.
6. Select the **Set Position** command from the Setup Menu and enter Z= 2.0 and X= 0.375. Select the Operator Panel and check that the spindle speed is set to 100%.

The workpiece is now correctly mounted.

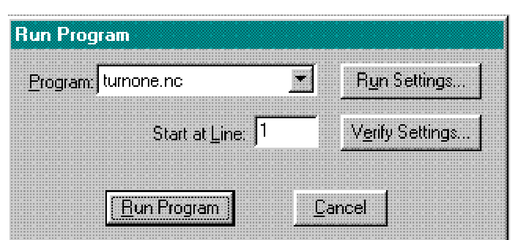
## Run the Program

Before executing the Turnone.nc program, check that all safety precautions have been taken. The Turning Center safety shield should be closed, and you should be wearing safety glasses.

If anything goes wrong, immediately press the Emergency Stop button on the Turning Center to stop the operation. A safety checklist has been provided in the Reference Guide: Section J of this guide. Post a copy of this checklist near the Turning Center and review it before you run any NC program.

To run the program:

1. **After reviewing the Safety Checklist**, select the **Run/Continue** command from the Program Menu. The Run Program dialog box appears.



**To change any of the Run Settings:**

Click on an item's checkbox. The Run Settings include:

**Single Step** Allows you to run the NC part program one line at a time, pausing after each line is executed.

**Optional Skip** Recognizes the optional skip code (/ ).

**Optional Stop** Pauses the NC program at any M01 code.

**Enable Subprograms** Must be on if the NC program uses subprograms.

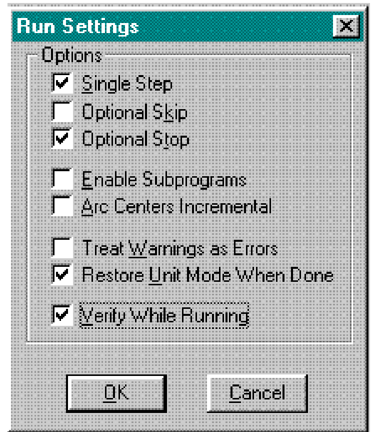
**Arc Centers Incremental** Recognizes the % code indicating that the center of an arc is an incremental value relative to the start of the arc.

**Treat Warnings as Errors** Halts the NC program at a warning as though it were an error.

**Restore Unit Mode When Done** Restores the original unit mode (inches or metric) regardless of the units used in the current NC program.

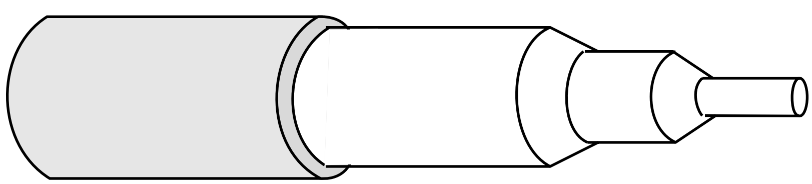
**Verify While Running** Allows tool path verification to occur while the NC program is running on the Turning Center.

2. Make sure that the **Start Line** box is set to line 1 of the program.
3. Click on the **Run Settings** button. The Run Settings dialog box appears.




4. Make desired changes in the Run Settings dialog box, then select **OK**.
5. Click on the **Run Program** button to begin running your program.
6. After the part is turned, press the Emergency Stop button before opening the safety shield and removing the finished part.

Your finished part should look similar to the part pictured here.



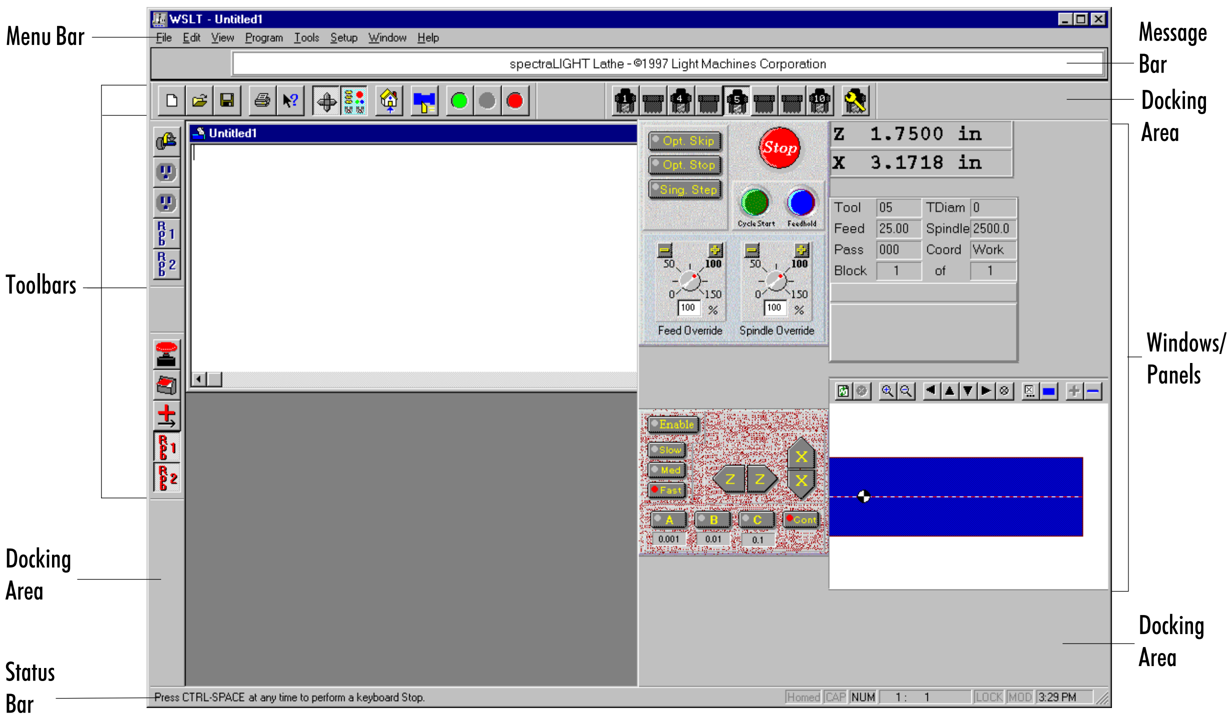
-

Reference Guide: Section E

# Control Program Reference

# About the Control Program Interface

The Control Program interface (the screen) is composed of several components that allow you to create NC part programs and interact with the Turning Center.




# Using the Message Bar

The Message Bar is located directly beneath the Menu Bar. When an NC program is running or being verified, the Message Bar displays the name of the NC file currently being run, or the most recent operator message. When a program is running, the Message Bar also displays control buttons (Go or Stop). When there is no program running, the Message Bar displays the Control Program copyright notice.

# Using Windows and Panels

Windows can be used to display information or accept input from the machine operator. Some windows can be docked or they can be floating windows (see *Docking and Floating Windows and Toolbars* later in this section). Windows are activated or hidden using the commands under the View Menu.

Panels are used to control machine operation. They can also be docked or they can be floating windows.

The following windows and panels are available:

- Program Edit Windows
- Position Window
- Machine Info Window
- Verify Window
- Jog Control Panel
- Operator Panel

# Using Program Edit Windows

When you open an existing NC part program file, or create a new one, the program appears in a program edit window. Program Edit Windows have all of the features common to other windows, including a title bar which displays the program file name and controls for minimizing, maximizing, and closing the window.

Program Edit Windows appear in the Edit Area (the large central area) of the screen. The Edit Area can be closed and moved. First select **Cascade** from the Window menu, then the window is movable and resizable. The Edit Area can contain multiple Edit Windows.

If a Program Edit Window is locked, the background of the edit window is gray. In the locked position, no edits can be made. If the window is unlocked, it is white. The lock command can be found in the Edit menu of the menu toolbar.

When other windows, panels, and toolbars are closed, the space that their docking areas occupy is given to the Edit Area.

When other windows, panels, and toolbars are open, the space that their docking areas occupy is taken from the Edit Area.

The Windows Menu has several commands for managing program Edit Windows, and allows you to select a particular window from a list of all current windows, identified by file names.

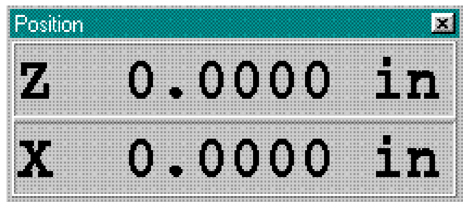
## Using the Position Window

This is a dynamic display. When a program is running or being verified, the current position of the tool is indicated here.

If you double-click on this window, the Goto Position dialog box appears, allowing you to move the tool to specific coordinates.

**Note:**

When running the Control Program in Simulate mode, the Goto command will not move the machine to the specified coordinates. It will simulate movement, showing the tool moving to those coordinates in the Position Window.



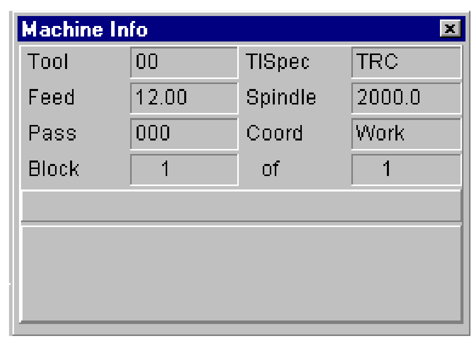

## Using the Machine Info Window

The information displayed in the Machine Info Window varies with the particular operation being performed:

**Tips:**

- You can move the Machine Info Window to another part of the screen by holding the Ctrl key down, then clicking and dragging the Machine Info Window.
- When the Machine Info Window is floating (not docked) you can resize it just like any other window. To prevent the window from ever docking, right-click on the window and un-check the Dockable command.

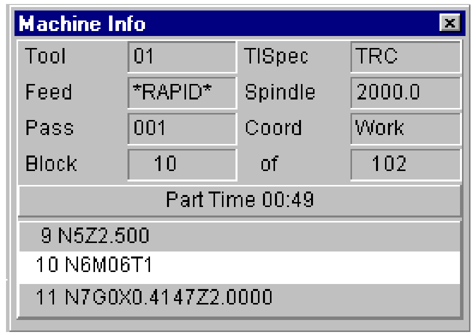
**When the machine is idle**, the Machine Info Window displays information about the:




- Current tool number
- Tool reference point
- Feed rate
- Spindle speed
- Number of passes* made for the current program
- Coordinate system being used
- Current block number*
- Number of blocks* in the current program

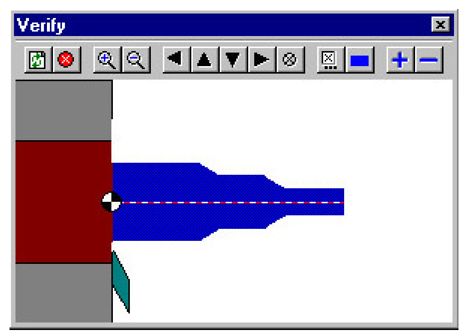
(*The passes, current block, and total number of blocks refer to the last program verified or run.)

**When a program is running,** the Machine Info Window becomes a dynamic display. In addition to updating the previously mentioned information, each line of code (as it is executed) is displayed along with the previous and next lines of code. Also, a clock provides the elapsed run time for the program.




**When a program is being verified**, the Machine Info Window displays the current line of code, plus the previous and next lines, along with all information given for an actual run of the program.

## Using the Verify Window




The Verify Window can be opened and closed by selecting the Verify Window command from the View Menu.

The Verify Window displays a simulation of your part program (tool path verification) when you select the Verify command from the Program Menu, or when you click the Verify Program button on the Standard Toolbar.

Choices made in the Verify Setup dialog box will be displayed in this Verify Window. These choices include a centered view, a view zoomed in or out, colors, the speed of the Verify Program and updating the Window after resizing it.

Tips:

- You can double-click on the Verify Window to bring up the Verify Setup dialog box.
- You can move the Verify Window to another part of the screen by holding the Ctrl key down, then clicking and dragging the window.
- Pressing the right mouse key while in the Verify window displays a pop-up menu with Verify-related commands.
- When the Verify Window is floating (not docked) you can resize it just like any other window. To prevent the window from ever docking, right-click on the window and uncheck the Dockable command.

## Using the Jog Control Panel

The Jog Control Panel is accessed by selecting the Jog Control command under the View Menu, or by clicking on the Jog Control button on the Standard Toolbar. The Jog Control Panel allows you to manually move (jog) the tool on the Turning Center.

Each axis on the machine is represented by buttons. The Z axis is represented by two horizontal buttons, one for negative motion and one for positive motion. The X axis is represented by two vertical buttons, one for negative motion and one for positive motion. Pressing any of the axis buttons moves the tool in the indicated direction as long as the system is not in Simulate Mode.

Jogging occurs in specific increments of speed and distance. The speed and distance values are selected on this panel as well. You can alter the speed and distance parameters for jogging by selecting the Jog Settings command under the Setup Menu, or by double clicking on the Jog Control Panel.

To jog a tool:

1. Define the Speeds and Steps (distances) for jogging by selecting the **Jog Settings** command under the Setup Menu.
2. Click on the **Axis** button on the Jog Keypad to move the tool in the desired direction. The tool moves at the speed and distance indicated by the Speed and Step buttons.

### Tips:

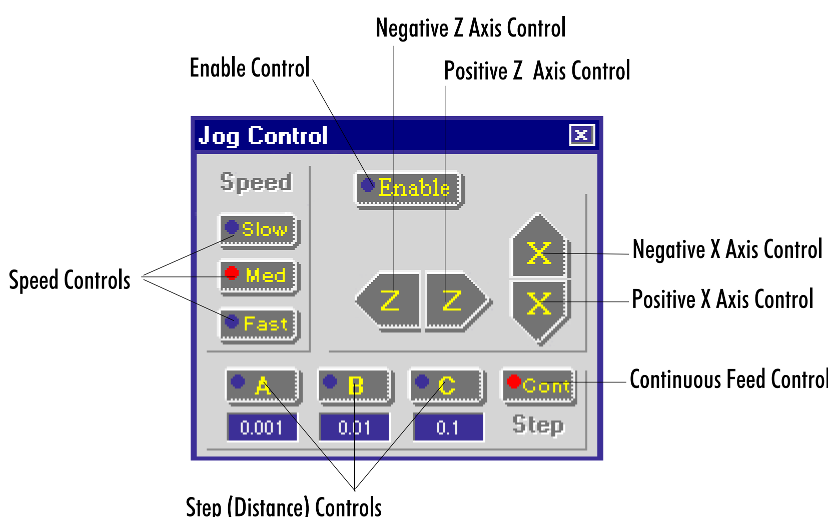
- Double-clicking on the Jog Control Panel opens the Jog Settings dialog box.
- If you wish, you can move the Jog Control Panel to another part of the screen by holding the Ctrl key down, then clicking and dragging the Jog Control Panel.

### Note:

The X Axes are reversed when the Tool Turret is present. This is because the Tool Turret positions the tool(s) above the workpiece, and must work in a positive quadrant. Therefore, the values of X on are reversed to make proper accommodations in the NC program.

### Caution

Wtih the Tool Turret installed, the Jog Control Panel allows jogging in both the positive (backward, counterclockwise) and negative (frontward, clockwise) directions. Jogging a complete rotation in the negative direction is allowed. However, only short positive distances are allowed per button press. the positive motion is reserved for manually servicing the Tool Turret. All other rotations should be in the negative direction to avoid damge to the Tool Turret.




Jog control Panel Operations without Tool Turret Installed

### Tip:

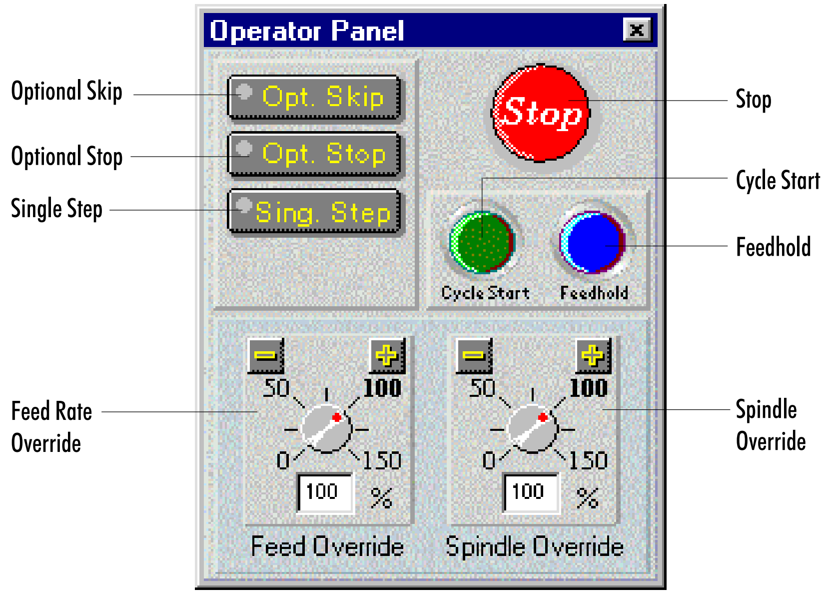
To move the tool in a continuous motion, select Cont. The tool will continue to move as long as the axis button is depressed.

# Using the Operator Panel

The Operator Panel provides controls that are used while running an NC program on the Turning Center.

**Tip:**

You can move the Operator Panel to another part of the screen by holding the Ctrl key down, then clicking and dragging the Operator Panel.




The Operator Panel controls include:

- **Optional Skip**: Allows you to execute or ignore any optional skips ( / codes) you have embedded in the NC program.
- **Optional Stop**: Allows you to execute or ignore any optional stops (M01 codes) you have embedded in the NC program.
- **Single Step**: Causes the NC program to pause after each block is executed. This allows you to check each step of the cutting operation.
- **Stop**: Immediately halts the currently running NC program. This button works the same as the Ctrl + Space Bar combination.
- **Cycle Start**: Begins running the current NC program from the beginning or from a paused condition.
- **Feedhold**: Pauses the currently running NC program. To continue running the program from a Feedhold, press the Feedhold button again or press the Cycle Start button.
- **Feed Rate Override**: Overrides the programmed feed rate.
- **Spindle Speed Override**: Overrides the programmed spindle speed.

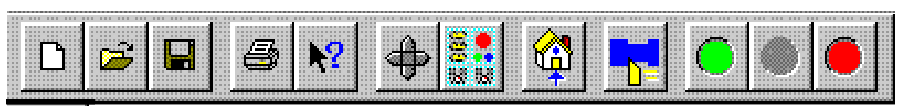
# Using Tool Bars

Toolbars carry buttons that correspond to frequently used menu commands. You can click on these buttons to quickly select the associated menu command. Toolbars also operate Outputs and display the state of Inputs. Toolbars can be placed anywhere on your screen, and can be hidden if you do not use them often or want the additional space for program windows. Toolbars are revealed and hidden using the Toolbars command under the View Menu.

The toolbars include:

- Standard Toolbar
- Inputs Toolbar
- Outputs Toolbar
- Turret Control Toolbar

# Using the Standard Toolbar



The Standard Toolbar provides quick-access buttons for the following menu commands:

**Context-sensitive**

The results of your action are dependent on the item you click, or on the operation you are currently performing.

| Command: | Used to: |
|---|---|
| **New** | Create a new program edit window |
| **Verify** | Verify the current NC part program |
| **Run** | Run the current NC part program |
| **Pause** | Pause the currently running NC part program |
| **Stop** | Immediately halt the currently running NC program |
| **Open** | Open an existing NC program file |
| **Save** | Save an NC program file |
| **Print** | Print an NC program |
| **Context Help** | This button can help you instantly find information on the items you see on the screen. For instance, click on the Context Help button, then click on a menu item, toolbar button, window or other screen element. The Help topic for that particular item appears. |

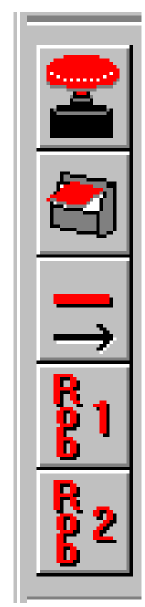
| | |
|---|---|
| **Jog Control Panel** | Show or hide the Jog Control Panel |
| **Operator Panel** | Show or hide the Operator Panel |
| **Home** | Sets/Checks the machine's Home, or reference point |
| **Verify** | Verify the current NC part program |
| **Run** | Run the current NC part program |
| **Pause** | Pause the currently running NC part program |
| **Stop** | Immediately halt the currently running NC program |

## Using the Inputs Toolbar

The Inputs Toolbar isn't really a toolbar, in that you can not use it to interact with the various Control Program inputs. It is a monitoring device that keeps track of the state of the various machine inputs.

The state of each input is indicated by the position of its button. If a button is depressed, the input is "on" or "high." If a button is not depressed, the input is "off" or "low." You can also check the condition of an input by clicking on it or by holding the mouse over the input button. The state of the input is displayed on the Status Bar  at the bottom of the screen.

The inputs on the Inputs Toolbar include:



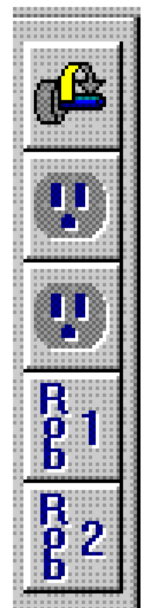

- The **Emergency Stop** condition. This input is in the "on" condition (depressed) if the Emergency Stop button on the Turning Center is pushed in.
- The **Safety Shield** condition. This input is in the "on" condition (depressed) if the Safety Shield on the Turning Center is open.
- The **Negative Limit** condition. This input is in the "on" condition (depressed) if the negative Z axis limit has been hit.
- The **Robot Input 1** condition. This input is in the "high" condition (depressed) if robotic input #1 (on the TTL I/O connector on the Controller Box) is currently in a high state.
- The **Robot Input 2** condition. This input is in the "high" condition (depressed) if robotic input #2 (on the TTL I/O connector on the Controller Box) is currently in a high state.

# Using the Outputs Toolbar

The Outputs Toolbar provides quick-access buttons for controlling the system outputs.

The state of each output is indicated by the position of its button. If a button is depressed, the output is “on” or “high.” If a button is not depressed, the output is “off” or “low.” You can also check the condition of an output by holding the mouse over the output button. Clicking on the button will change the state of the output. The name of the output is displayed in a tool tip, and the state of the output is displayed on the Status Bar at the bottom of the screen.

The output controls on the Outputs Toolbar include:

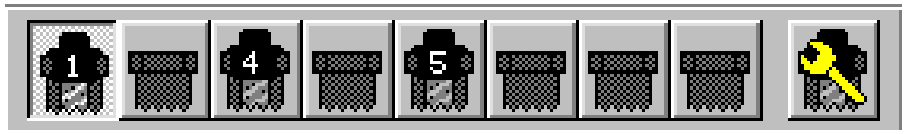



- The **Spindle** control. This button turns the spindle on and off.
- The **ACC1** control. This button turns the Accessory 1 output on and off.
- The **ACC2** control. This button turns the Accessory 2 output on and off.
- The **Robot Output 1** control. This button toggles Robotic Output 1 between high and low conditions. When this button is depressed, it places Robotic Output 1 (on the TTL I/O connector on the Controller Box) in the “high” condition. When this button is not depressed, Robotic Output 1 is in the “low” condition.
- The **Robot Output 2** control. This button toggles Robotic Output 2 between high and low conditions. When this button is depressed, it places Robotic Output 2 (on the TTL I/O connector on the Controller Box) in the “high” condition. When this button is not depressed, Robotic Output 2 is in the “low” condition.

**Note:**

Refer to Section L of this guide for more information on robotic interfacing.

# Using the Turret Control Toolbar




Note:

This Turret Control Toolbar is only used with the optional Tool Turret

The Turret Control Toolbar provides quick-access buttons for tools which are defined under the Tool menu in the Tool Library. The button that is depressed is the tool that is selected for use in an NC program.

The Tool Turret controls on the Turret Control   Toolbar include:

- **Tool Station 1** through **Tool Station 8.**In the toolbar above, Tool Stations #1, #4 and #5 are defined, while Tool #1 is selected. Five Tool Stations are undefined and are still available for input information and use. The tools must be defined in the Tool Library dialog box which can be accessed throught he Tool menu.

# Using the Status Bar

The Status Bar displays miscellaneous information about the Turning Center


and the computer. The left side of the Status Bar is reserved for operator messages such as the one displayed here.

If any of the following are grayed-out, the feature is considered “off.”

- The machine **Homed** state: Black if the machine is currently homed.
- The **Caps Lock** key state: Black if the Caps Lock feature is on. Some NC programmers prefer to type their programs in capital letters. When the Caps Lock feature is on anything you type will be displayed in capital letters. Press the Shift key to type lower case letters.
- The **Num Lock** key state: Black if the Num Lock feature is on. Some NC programmers prefer to use the numeric keypad on the keyboard to enter figures. The Num Lock feature must be on to do this.
- **Current Line : Total Lines**: Displays the line the cursor is currently on, and the total number of lines in the program.
- The file **Locking** state: Black if the NC program file is locked.
- The file **Modified** state: Black if the NC program has been modified since being opened.

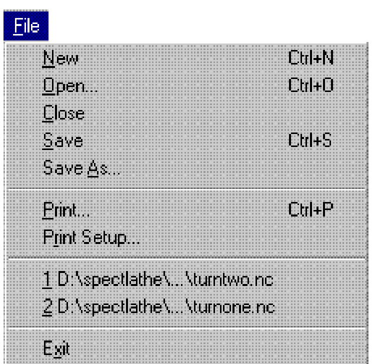
- The **Configure Turret** button at the end of the toolbar configures the Turret by specifying which tool is in which station.

# Using the Menu Bar

The Menu Bar is located at the very top of the screen. It lists the categories of commands into which the Control Program operations are grouped.

The available menus are:

- File Menu
- Edit Menu
- View Menu
- Program Menu
- Tools Menu
- Setup Menu
- Window Menu
- Help Menu



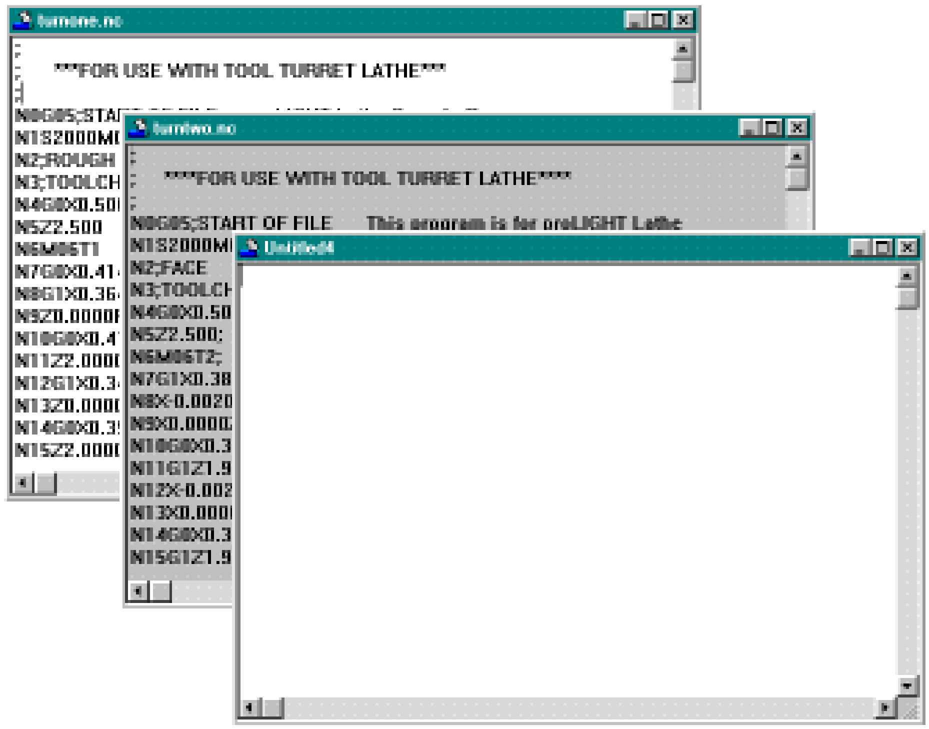

## File Menu

The File Menu provides typical file management commands and the command to exit the Control Program.

| Command: | Used to: |
|---|---|
| **New** | Create a new program window. |
| **Open** | Open an existing file. |
| **Close** | Close an open program window. |
| **Save** | Save a program. |
| **Save As** | Save a program under a different filename or location. |
| **Print** | Print an open NC program. |
| **Print Setup** | Set up your printer for printing. |

| | |
|---|---|
| **Recently opened files** | Open one of the eight most recently used files. |
| **Exit** | Exit the Control Program. |

Here is a group of three Edit Windows open at the same time; two previously existing windows and one new window. As windows are opened they are cascaded (offset from each other) so that each one is visible. This layout can be chosen in the Window menu, with the Cascade command. If a file is "locked," the background of the Edit Window is gray, as is the window for Turntwo.nc shown here.

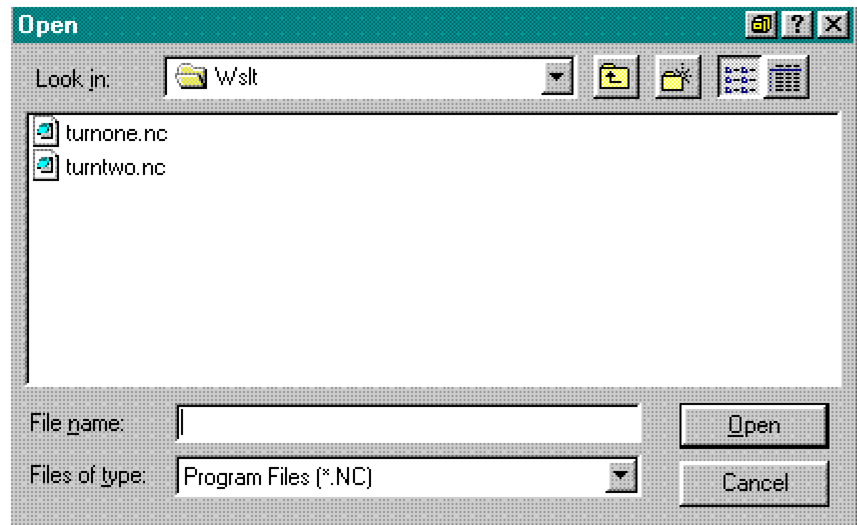



## New Command

Use the New command under the File Menu to create a new program edit window. You can create a new program edit window at any time. The number of program edit windows that you have open at one time is limited by the amount of memory on your computer.

To create a new program window, select **New** from the File Menu, or press **Ctrl+N**. A new program window is created. The filename on the Title Bar is "Untitled," indicating that this is a new program. The program will remain untitled until you save it. You should save a new program before it is run or verified.

This is a typical Open file dialog box.




## Open Command

Use the Open command under the File Menu to open an existing NC program. The number of program edit windows that you have open at one time is limited by the amount of memory on your computer.

### To open an existing NC program:

1. Select **Open** from the File Menu, or press **Ctrl+O**. The Open dialog box appears.
2. In the dialog box, locate and highlight the desired NC file.

3. Click the **Open** button or press **Enter**. The selected NC program file is opened. The Title Bar on the Edit Window displays the name of the file.

### To select a file that is already open:

- If the open file has changed since it was opened, you are prompted to reload the original version of the file or to cancel the opening procedure.
- If the open file has not changed since it was opened, it becomes the active Program Edit Window.

## Close Command

You can close a program window at any time. Unless you have already done so, you will be prompted to save any changes made to the program file.

To close a program window:

1. Make sure the program window you want to close is selected.
2. Select one of several ways to close the open window:
   - Select the **Close** command from the File Menu.
   - Single-click the icon on the far left of the Title Bar and select **Close** from the drop-down menu. (If the Edit window is maximized, the icon will be at the far left of the Menu Bar.)
   - Double-click the icon on the far left of the Title Bar. (If the Edit window is maximized, the icon will be at the far left of the Menu Bar.)
   - Click on the **Close** button on the far right of the Title Bar. (If the Edit window is maximized, the icon will be at the far right of the Menu Bar.)
   - Press **Ctrl**+**F4**.

3. If there are unsaved changes to the current program, the File Save dialog box appears, prompting you to save the changes. Click one of the buttons in the dialog box:
   - Click **Yes** to save the changes.
   - Click **No** to discard the changes.
   - Click **Cancel** to exit the dialog box without saving the changes or closing the program window.

**Tip:**

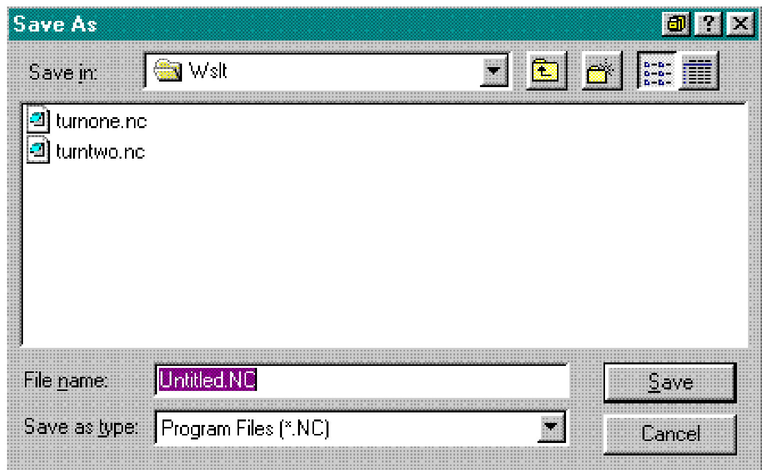
If you enable the AutoSave feature (see Setup Menu/Preferences), your work will be saved automatically at regular intervals. Use of the AutoSave feature is recommended; if AutoSave is not enabled, you should save your files frequently as you work.

## Save Command

Use the Save command under the File Menu to save the current program as an NC file.

**Note:**

When you name a file, consider whether this file will be used on older systems running DOS or Windows 3.1 before you take advantage of Windows 95 long file names.




If the current NC program was previously saved, selecting Save saves the changes to the same file without bringing up a dialog box. If the current program is new (and still has the name "Untitled"), selecting Save brings up the Save As dialog box, with which you name, choose a location for, and save the new program.

To save a program, select **Save** from the File Menu, or press **Ctrl+S**.

The current program is saved to a file. If this is a new program, the Save File As dialog box appears. Choose a name and location for the new file.

## Save As ... Command

Use the Save As command under the File Menu to save the current program to an NC file using a new name or location.

### The Save As Dialog Box

The Save As dialog box in the WSLT Control Program functions in the same manner as in other Windows 95 applications.

**Note:**

When you name a file, consider whether this file will be used on older systems running DOS or Windows 3.1 before you take advantage of Windows 95 long file names.

To use the Save As dialog box:

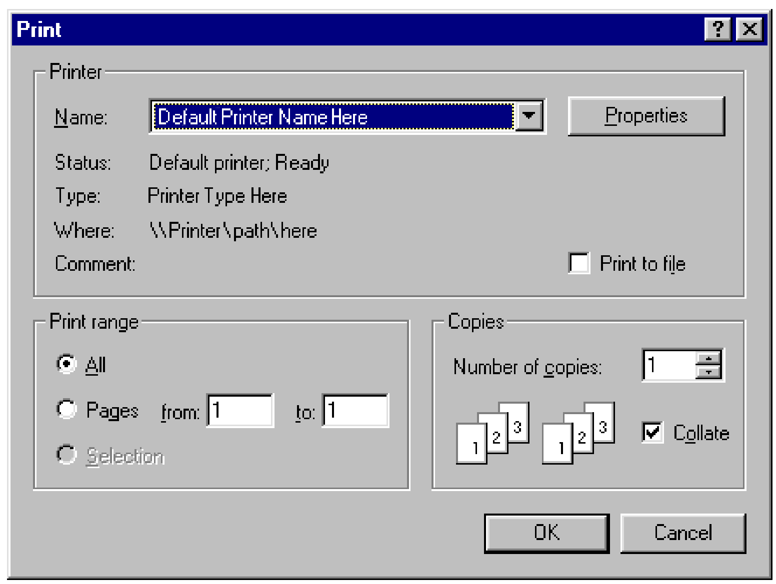
1. Select a destination for the file using the **Save in:**, **Up one level** and **Create new folder** buttons.
2. Enter a filename in the **File Name:** field.
3. Select a file type in the **Save as type:** field.
4. Click **Save** or press **Enter** to save the file.
5. Click **Cancel** or press **Esc** to cancel and exit the dialog box.

### To save a program using a new name or location:

1. Select **Save As** from the File Menu. The Save As dialog box appears (see above). It displays the name and location of the current program file.
2. Choose a new drive and directory for the file, if desired.
3. Type in a new file name, if desired.
4. Click **Save** to save the file, or **Cancel** to exit the dialog box.

**Note:**

The default printer name will reflect the designated Windows default printer.




## Print Command

Use the Print command under the File Menu to print the current NC program.

To print the program:

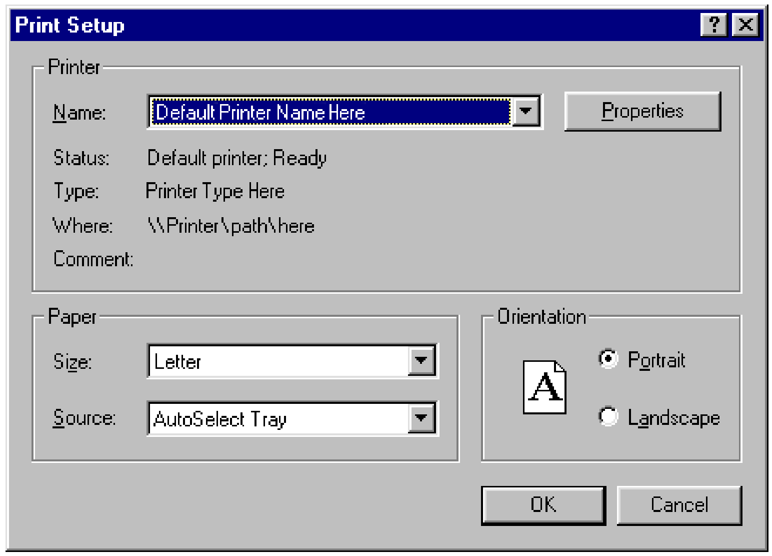
1. Use one of the following methods to open the Print dialog box:
   - Click the **Print** button on the toolbar.
   - Select the **Print** command from the File Menu.
   - Press **Ctrl+P**.

   The Print dialog box appears.

2. Choose the desired print options in the dialog box. Clicking **Properties** opens the Print Setup dialog box.
3. This dialog box will give options that are specific to your printer, such as paper, graphics, and device options. customize your print job then select **OK**.
4. Select **OK** to print, or select **Cancel** to exit the Print dialog box without printing the NC program.

Note:

The default printer name will reflect the designated Windows default printer.



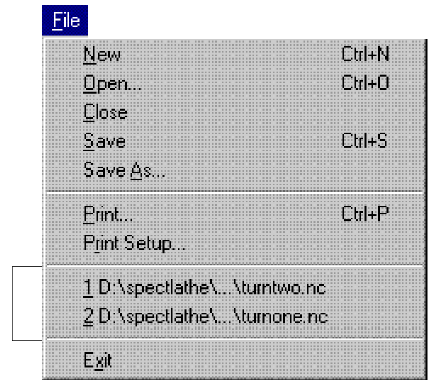

You can print to any printer that is supported by Windows. See your printer manual or Windows documentation for more information on installing and using printers with Windows.

## Print Setup Command

Use the Print Setup command under the File Menu to select a printer. The Print Setup dialog box allows you to establish parameters for printing your NC part programs.

To choose print settings:

1. Select **Print Setup** from the File Menu. The Print Setup dialog box appears.
2. Select the desired print settings, including:
   - ◆ The destination printer.
   - ◆ The size of the paper.
   - ◆ The paper tray.
   - ◆ The orientation of the paper.
3. Select **OK** to print, or select **Cancel** to exit the Print dialog box without setting the printing parameters.




## Opening a Recent Program

Use the numeric (1, 2, 3, … 8) commands under the File Menu to open any of the most recently opened files (up to the last eight files).

The names and paths of the most recent files appear in the reverse order in which they were opened.

For instance, in this example, 1 turntwo.nc is the most recently opened file.

The file 2 turnone.nc is the second most recently opened file.

To open one of the listed NC program files, simply select the filename from the list. The recent program you selected is opened. Once the file Edit Window is open, the Title Bar displays the name of the program file.

## Exit Command

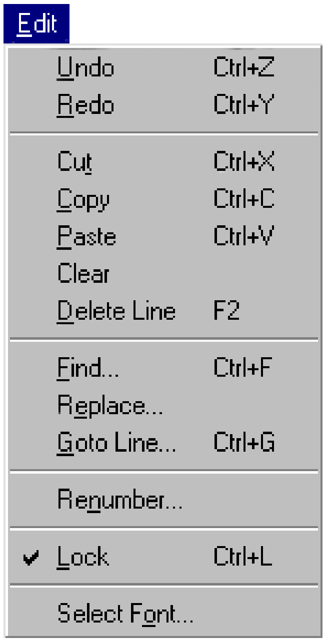
Use the Exit command under the File Menu to exit the Control Program. You should always exit the Control Program before you exit Windows.

Use one of the following methods to exit the Control Program:

- ◆ Select the **Exit** command from the File Menu.
- ◆ Single-click the icon on the far left of the Control Program Title Bar. Select **Close** from the drop-down menu.
- ◆ Double-click the icon on the far left of the Control Program Title Bar.
- ◆ Click on the **Close** button on the far right of the Title Bar.
- ◆ Press **Alt+F4**.

If there are unsaved changes to any program window, a dialog box appears for each unsaved program window, prompting you to save the changes.

- Click **Yes** to save the changes and exit.
- Click **No** to ignore the changes and exit.
- Click **Cancel** or press **Esc** to cancel the Exit command and return to the Control Program.




**Note:**

If your NC program is locked, the only editing options available to you are Find, Goto Line, and Select Font. See page E-27 for information on unlocking a program.

## Edit Menu

The Edit Menu provides typical text editing commands. Before you can edit the text in an NC program, you must select the text.

| Command: | Used to: |
| --- | --- |
| **Undo** | Undo the most recent editing command. |
| **Redo** | Redo the most recent Undo command. |
| **Cut** | Cut selected text to the Windows clipboard. |
| **Copy** | Copy selected text to the Windows clipboard. |
| **Paste** | Paste text from the Windows clipboard into the current NC program. |
| **Clear** | Delete selected text. |
| **Delete Line** | Delete the line the cursor is currently on. |
| **Find** | Locate a sequence of characters in an NC program. |
| **Replace** | Replace one sequence of characters with another, one or more times. |
| **Goto Line** | Jump to a particular line in the NC program. |
| **Renumber** | Modify or insert N codes in an NC program . |
| **Lock** | Lock or unlock the Program Edit Window to prevent or allow modification to the NC program. |
| **Select Font** | Change the font currently being used. |

## Undo Command

The Undo command reverses the most recent editing action taken. It is useful for recovering from accidental deletion or inclusion of a block of text.

To undo the last change, select **Undo** from the Edit Menu, or press **Ctrl+Z**.

- If your last editing action deleted selected text, the text is restored.
- If your last editing action deleted a character, the character is restored.
- If your last editing action pasted text, the text is removed.
- If your last editing action typed a character, the character is removed.
- If Undo is grayed-out in the Edit Menu, no changes can be undone.

### Selecting Text

Use the mouse or a Shift+Arrow key combination to select a portion of your NC program for cutting, pasting, or copying.

To select text using the editing keys:

1. Using the arrow keys, position the cursor at the beginning of the text to be selected.
2. Press the shift key and hold it down while using the arrow keys to move the cursor to the end of the text to be selected.
3. Release the shift key.

To select text using the mouse:

1. Place the cursor at the beginning of the text to be selected.
2. Click and hold the left mouse button.
3. Move the cursor to the other end of the text to be selected.
4. Release the mouse button.

## Redo Command

The Redo command reverses the action of the most recent undo command. For example, if you delete a portion of text in an NC program, then decide that wasn't such a good idea, the Undo command will return the text to its original position. If you then decide that the deletion was appropriate after all, select the Redo command to once again remove the text.

## Cut Command

Use the Cut command under the Edit Menu to remove text from the NC program (the text is copied to the Windows clipboard). The text can then be pasted anywhere in the current program, into another program, or into another application such as Notepad. The text remains on the Windows clipboard until it is replaced by another Cut or Copy operation.

To cut text to the Clipboard:

1. Select the text you wish to cut.
2. Select **Cut** from the Edit Menu, or press **Ctrl+X**.

## Copy Command

Use the Copy command under the Edit Menu to duplicate selected text in an NC program.

To copy text to the Clipboard:

1. Select the text you wish to copy.
2. Select **Copy** from the Edit Menu, or press **Ctrl+C**. The selected text is copied to the Clipboard.

## Paste Command

You can use the Paste command to insert text from the Windows clipboard into your NC program.

To paste text from the Clipboard:

1. Place the cursor at the point in the NC program where you wish to insert text that has been previously cut or copied to the Windows clipboard.

2. Select **Paste** from the Edit Menu, or press **Ctrl+V**. The contents of the clipboard are inserted into the program. If this menu command is grayed-out, there is no text on the clipboard to paste.

## Clear Command

Use the Clear command under the Edit Menu to delete selected text from your NC program. The text is not copied to the Windows clipboard. You can also use the Delete key on your keyboard to achieve the same effect.

To delete text using the Clear command:

1. Select the text you wish to delete.
2. Select the **Clear** command from the Edit Menu. The selected text is deleted.

## Delete Line Command

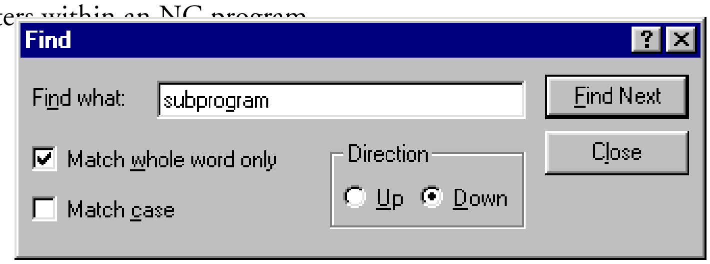
Use the Delete Line command under the Edit Menu to delete an entire line of NC code from a program without selecting it first.

To delete a program line using the Delete Line command:

1. Place the cursor anywhere on the line of code you wish to delete.
2. Select the **Delete Line** command from the Edit Menu, or press **Ctrl+Y**. The line of code is deleted.

## Find Command

Use the Find command under the Edit Menu to locate a particular sequence of characters within an NC program.




To use the Find command:

1. Select **Find** from the Edit Menu, or press **Ctrl+F**. The Find dialog box appears.
2. Enter the character sequence for which you are looking in the **Find what:** box.
3. Check the **Match whole word only** box to restrict the search to whole words, skipping partial words.
4. Check the **Match Case** box to restrict the search to finding only those text strings that match the case (upper or lower) of the text that you entered.

5. Select **Up** or **Down** from the Direction box to search through the text before or after the cursor position, respectively.

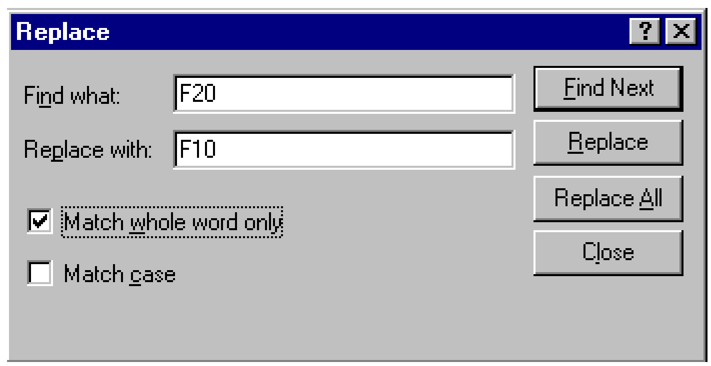
6. Click **Find Next** or press **Alt+F** to begin the search. Click **Close** or press **ESC** to exit the Find dialog box without performing the search.

## Replace Command

Use the Replace command under the Edit Menu to replace an existing character string with a new character string.

If the Match Case box is checked, this feature is "case-sensitive." If you enter a character string in lower case letters, but your NC program is written in all upper case letters, Replace will not find any occurrence of the string. Make sure you enter the existing character string exactly as it is shown in the NC program.

To use the Replace command:



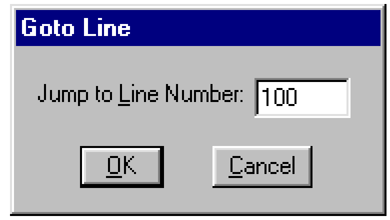

1. Select **Replace** from the **Edit** Menu. The Replace dialog box appears.
2. Enter the existing character string in the **Find what:** box.
3. Enter the new character string in the **Replace with:** box.
4. Check the **Match whole word only** box to restrict the search to whole words, skipping partial words.
5. Check the **Match Case** box to restrict the search to finding only those text strings that match the case (upper or lower) of the text that you entered.
6. To locate the first and successive instances of the character string, click **Find Next**.
7. Select **Replace** to replace the currently located character string which has been highlighted for you with the word or phrase which you have indicated. **Replace** changes one word at a time, starting at the location of the cursor.

8. Select **Replace All** to replace all similar character strings throughout the NC program. Replace All will automatically locate each instance of the word or phrase and change all of them at once and will present a dialog box telling you how many changes were made.

9. Click **Close** to exit the dialog box .

**Note:**

The line number is counted starting at one and increments in steps of one, regardless of the NC code block sequence number. That is, the Goto Line does not reference the "N" code in the NC file.




## Goto Line Command

Use the Goto Line command under the Edit Menu to move the cursor to a specific line in the NC program. This command is also available using the Program Edit Window Pop-up Menu.

To use the Goto Line command:

1. Select **Goto Line** from the Edit Menu, or press **Ctrl+G**. The Goto Line dialog box appears.

2. Enter a line number in the **Jump to Line Number** box. The cursor moves to the specified line in the NC program. If the line number entered is larger than the number of lines in the program, the cursor is moved to the end of the program.

**IMPORTANT!**

None of the Renumber actions can be undone!

The Undo command will not undo the effects of the Renumber command. Removed comments must be reentered manually. You should save a copy of the program to another file using the Save As so you can easily recover if the effects of using Renumber are not what you expected.

## Renumber Command

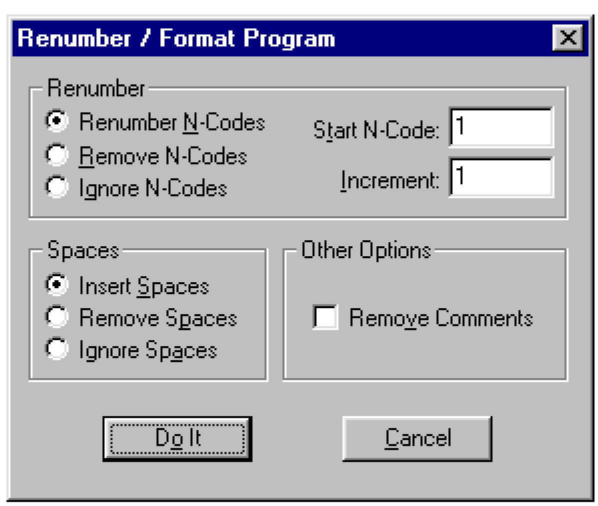
Use the Renumber command under the Edit Menu to alter the N codes in your NC program.

The Renumber command can be used to:

- Insert N codes in a program that has none.
- Remove N codes from a program.
- Renumber the N codes in a program.

**Note:**

The number of a line in a program and the number of the corresponding N-code are only the same if the first N code in the program is N1 and each N code thereafter is incremented by 1.



- Insert, remove or ignore spaces between NC commands.
- Remove comments from the program.

### Insert N Codes

To insert or renumber the N codes in your program:

1. Select **Renumber** from the Edit Menu. The Renumber/Format Program dialog box appears.
2. Select **Renumber N Codes** or press **Alt+N**.
3. Click on the **Start N Code** box (or press **Alt+T**), then enter the number of the first N code. The default starting number is N1.
4. Click on the **Increment** box (or press **Alt+I**), then enter the increment you wish to use. For instance, if you wish to have each N code numbered in increments of 5, enter 5 in the Start N Code box and enter 5 in the Increment box. The N code sequence will then be: N5, N10, N15, N20…and so on.

   This option is useful if you are renumbering a portion of the program to be inserted into another program, and the N codes must start with a number greater than 1. Using increments greater than 1 allows you to insert additional numbered lines without having to renumber the whole program.
5. Select **Do It**, or press **Alt+O**, to execute the Renumbering options you selected. Select **Cancel**, press **Alt+C**, or press **Esc**, to exit the Renumber/ Format Program dialog box without altering the NC program.

### Renumbering and Subprograms

Although the Renumber command automatically changes P codes used with M99 codes, it does not renumber P codes used with M98 codes, nor does it renumber O codes. Although O codes are not altered, the lines which they occupy are counted. So, the very next N code is numbered as though the O code has been renumbered, too.

For instance, N41X
N42X…
O25G…
N44…

In this example, although the O code has not been renumbered, the line it resides on has been counted. The N code on the following line reflects the next number in the sequence.

### Insert or Remove Spaces

To insert or remove spaces between the NC words in your program:

1. Select **Renumber** from the Edit Menu. The Renumber/Format Program dialog box appears.

2. Choose a Spaces option.
   - **Insert Spaces** inserts a space between each NC word (to the left of the comment code).
   - **Remove Spaces** removes any spaces between NC words (to the left of the comment code).
   - **Ignore Spaces** ignores any spaces in the NC program.
3. Select **Do It**, or press **Alt+O**, to execute the Spaces options you selected. Select **Cancel**, press **Alt+C**, or press **Esc**, to exit the Renumber/Format Program dialog box without altering the NC program.

Note:

The Undo command will not undo the effects of the Remove Comments command. Removed comments must be entered again manually. You should save a copy of the program to another file using the Save As command so that you can easily recover if the effects of using Renumber are not what you expected.

### Remove Comments

To remove comments from your program:

1. Select **Renumber** from the Edit Menu. The Renumber/Format Program dialog box appears.
2. Select **Remove Comments**.
3. Select **Do It**, or press **Alt+O**, to execute the Remove Comments command. Select **Cancel**, press **Alt+C**, or press **Esc**, to exit the Renumber/Format Program dialog box without altering the NC program.

### Remove N Codes

To remove the N codes from your program:

1. Select **Renumber** from the Edit Menu. The Renumber/Format Program dialog box appears.
2. Select **Remove N Codes** or press **Alt+R**.
3. Select **Do It**, or press **Alt+O**, to execute the Remove N Codes command. Select **Cancel**, press **Alt+C**, or press **Esc**, to exit the Renumber/Format Program dialog box without altering the NC program.

Note:

When multiple program Edit Windows are open, each is individually locked or unlocked.

## Lock Command

Use the Lock command under the Edit Menu to prevent or allow changes to your NC programs. When an NC program is unlocked, it can be modified by the commands on the Edit Menu. When an NC program is locked, the program cannot be changed by any commands. By default, when you open a file it is automatically locked to prevent accidental changes. You can change this default setting in the Preferences dialog box (Seup Menu/Editor Tab).

There are three ways to determine if a selected program is locked or unlocked:

1. If a check mark appears next to the Lock command, the current NC program is locked.
2. A locked NC program can be identified by the background color of its Edit Window. If the background is gray, the file is locked; if it is white, the file is unlocked.

**Note:**

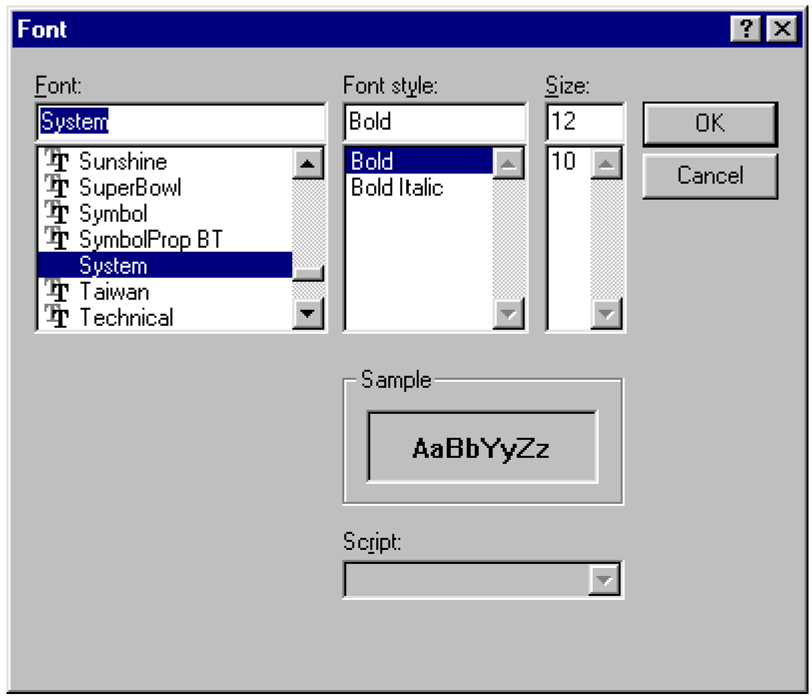
The Undo command does not undo a font settings change.

3. The Lock Indicator on the Status Bar is gray if the program is unlocked, black if the program is locked.

To toggle an NC program between the locked and unlocked states, select the **Lock** command from the Edit Menu, press **Ctrl+L**, or double-click the **Lock Indicator** on the Status Bar.

## Select Font Command

Use the Select Font command under the Edit Menu to change the font settings for open NC programs. The font settings control the font and font size used in the program Edit Window. The fonts listed are the true type fonts already installed on your system. Font Settings are intended for viewing and printing purposes only. They do not affect the NC program in any way and are not




stored within the program file. All open program windows use the same font settings.

To use the Select Font command:

1. Select the **Select Font** command from the **Edit** Menu. The Font dialog box appears.
2. Select a font from the **Font** list.
3. Select a **Font Style**.

4. Select a font size from the **Size** list.
5. Click **OK** to change the font, or click **Cancel** or press **Esc** to exit the Font dialog box without changing the fonts.


The checkmarks on this View Menu indicate that the Machine Info Window and the Verify Window are both open. The other windows, or panels, are not open.

## View Menu

The View Menu commands control the display of windows and toolbars.

| Command: | Used to: |
| --- | --- |
| **Position** | Open or close the Machine Position Window. |
| **Machine Info** | Open or close the Machine Info Window. |
| **Jog Control** | Open or close the Jog Control Panel. |
| **Operator Panel** | Open or close the Operator Panel. |
| **Verify Window** | Open or close the Verify Window. |
| **Toolbars** | Open or close one of the toolbars. |

### Position Command

Use the Position command on the View Menu to open or close the Position Window.

To open or close the Position window, select **Position** from the View Menu. A check mark appears next to the Position command when the Position Window is open.

### Machine Info Command

Use the Machine Info command on the View Menu to open or close the Machine Info window.

To open or close the Machine Info window, select **Machine Info** from the View Menu. A check mark appears next to the Machine Info command when the Machine Info Window is open.

### Jog Control Command

Use the Jog Control command on the View Menu to open or close the Jog Control Panel. You can also use the Jog Control button on the Standard Toolbar.

To open or close the Jog Control window, select **Jog Control** from the View Menu. A check mark appears next to the Jog Control command when the Jog Control Panel is open.

## Operator Panel Command

Use the Operator Panel command on the View Menu to open or close the Operator Panel.

To open or close the Operator Panel, select **Operator Panel** from the View Menu. A check mark appears next to the Operator Panel command when the Operator Panel is open.

## Verify Window Command

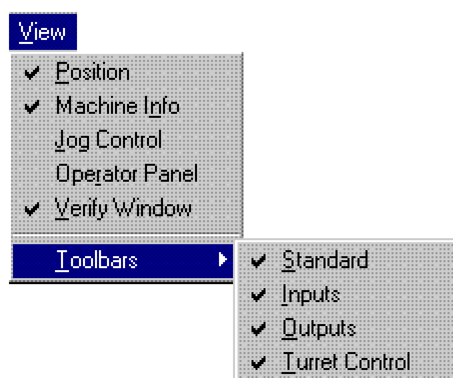
Use the Verify Window command on the View Menu to open or close the Verify Window.

To open or close the Verify Window, select **Verify Window** from the View Menu. A check mark appears next to the Verify Window command when the Verify Window is open.

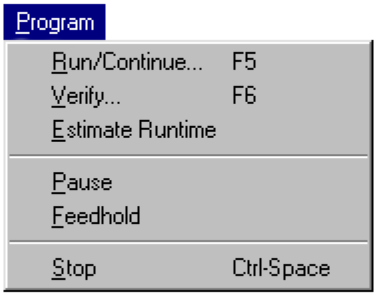
## Toolbars Command

Use the Toolbars command on the View Menu to show or hide the Toolbars.

To show or hide a toolbar:



1. Select **Toolbars** from the View Menu. The list of available toolbars is displayed. Toolbars that are visible have a check mark beside them.
2. Select the toolbar that you wish to show or hide.



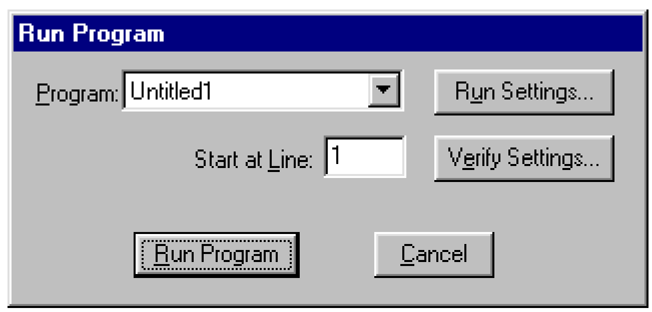

## Program Menu

The Program Menu commands allow you to Run, Verify, or Stop an NC program.

| Command: | Used to: |
|---|---|
| **Run/Continue** | Start running or resume running the current NC program. |
| **Verify** | Verify the current NC program. |
| **Estimate Runtime** | Estimate the runtime of the current NC program. |
| **Pause** | Pause the currently running NC program. |
| **Feedhold** | Stop movement of all axes. |
| **Stop** | Immediately halt the currently running NC program. |

### Caution

*Always* wear safety glasses and close the safety shield before running an NC program on the Turning Center. *Always* observe set up and safety precautions.




### Run/Continue Command

The Run/Continue command under the Program Menu runs the current NC program on the Turning Center. When you select Run/Continue from the Program Menu, the Run Program dialog box appears.

The Run Program dialog box allows you to select an NC program to run, to set the program block from which to begin running the program, and to access the Run Settings and Verify Settings dialog boxes.

1. Select an NC Program.

   If you have more than one NC program open, use the **Program** drop-down list, or press **Alt+P**, to select the program you wish to run.

2. Select a Starting Line.

When you are running an NC program for the first time, it is wise to start the program from the first line. When you start at a line other than line one, the control program parses through the program to the specified start point. As it parses, it performs operations such as turning the spindle on, but it will not execute a Dwell or Pause command, and it will not move the tool.

To specify a starting block, click on the **Start at Line** box, or press **Alt+L**, and enter the line number.

3. Set the Run Settings

   To bring up the Run Settings dialog box click on the **Run Settings** button, or press **Alt+U**.

4. Set the Verification Settings

   To bring up the Verify Settings dialog box click on the **Verify Settings** button, or press **Alt+E**.

5. Run the Program

   To start running the program, click on the **Run Program** button, or press **Alt+R**. To cancel running the program, click on the **Cancel** button, or press **Esc**.

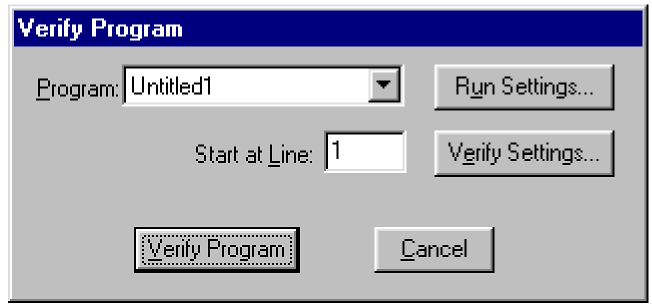
While a program is running, the Machine Info Window and the Message Bar keep you informed by providing information on:

- The name of the NC program.
- Which tool is being used.
- The tool reference.
- The feed rate.
- The spindle speed.
- The number of passes made.

- The coordinate system in use.
- Which block is currently being executed.
- How many blocks are in the program.

**Note:**

If you are verifying a part program for the first time, you should begin the verification at line one.



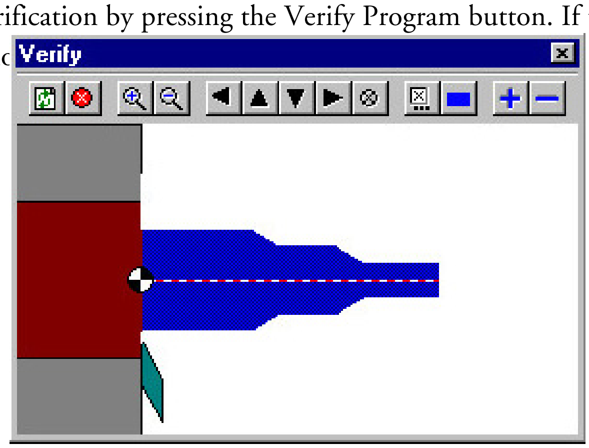

- Operator messages such as which block paused the program or the error that caused the program to stop.

## Verify Command

The Verify command allows you to view tool path verifications of your NC part programs. When you select Verify from the Program Menu, the Verify Program dialog box appears.

This dialog box allows you to select a program to verify from a pull-down list of currently open NC part programs. Prior to verifying the part program, you may wish to alter the Run Settings, alter the Verification Settings, or select a starting line in the program. The default starting line is line one.

Begin the verification by pressing the Verify Program button. If the Verify Window is nc



Note:

The Calculated Distance is the total distance that the tool and the workpiece move in relation to each other. Every linear and rapid motion (no arcs) has a length (length=Sqrt)(dx*dx+dz*dz). These lengths are added together, yielding the total distance. The feed rate of each segment is divided into the length to calculate the "ideal" time for each segment (assuming zero-communications overhead and infinite acceleration and deceleration). These times are added together to yield the total estimated time.

Tool path verification is displayed in the Verify Window. The workpiece and tool are displayed according to the choices you made in the Verify Settings dialog box.

Here is an example of how the Verify Window may appear for the part program Turnone.nc.

## Estimate Runtime Command

Use the Estimate Runtime command to calculate the approximate amount of time the spectraLIGHT Turning Center requires to turn your part, and the approximate distance the machine travels while turning your part.

The Estimate Runtime command accounts for Dwell times and subprograms when calculating estimated run time, but it can not account for stops that have indefinite length of stop time. These program stops include:

- Pause (G05/M00)
- Chain (M20)
- Skip (/)
- Wait for input high/low (G26/G25)
- Write to file (M22)
- Rerun (M47)

The Estimate Runtime command treats M47 Rewind codes as M2 End of Program codes.

Note:

An NC program can also be stopped by pressing the Emergency Stop Button on the front panel of the Turning Center, or by tripping a limit switch.

This command also verifies the syntax of your NC programs while calculating the estimated run time. If an error is found, the Estimate Run Time command alerts you with a dialog box, and places the cursor near the error.

## Pause Command

Use the Pause command to pause a running NC program. Pause may also be used during tool path verification. The pause is not immediate; it takes effect after the current NC block has been executed.

### To use the Pause command:

Select **Pause** from the Program Menu, or click the Pause button on the Standard Toolbar.

### To resume running a program after a Pause:

Press **F5**, click the **Run** button on the Standard Toolbar, or click the **Go** button on the Message Bar.

## Feedhold Command

The Feedhold command is very similar to the Pause command. It is used to pause a running NC program. The differences between a Pause and a Feedhold are:

- A Feedhold pause the NC program immediately; it does not wait until the current block is executed.
- Feedhold does not work during tool path verification.

### To use the Feedhold command:

Select **Feedhold** from the Program Menu.

### To resume running a program after a Feedhold:

Press **F5**, click the **Run** button on the Standard Toolbar, or click the **Go** button on the Message Bar.

## Stop Command

You can use the Stop command under the Setup Menu to halt a running NC program. The Turning Center immediately halts cutting and the current tool position is stored by the computer.

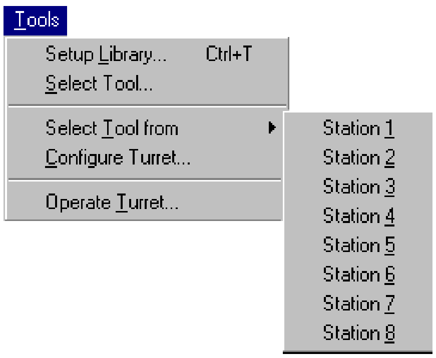
### To use the Stop command:

1. Select **Stop** from the Program Menu, press **Ctrl**+**Space**, or click the **Stop** (red) button on the Standard Toolbar.
2. A message box appears. Clear the box by clicking **OK** or pressing **Enter**. You are automatically returned to the Edit mode.

### To restart the NC program:

1. Manually jog the tool so it is above the workpiece to avoid a tool crash.

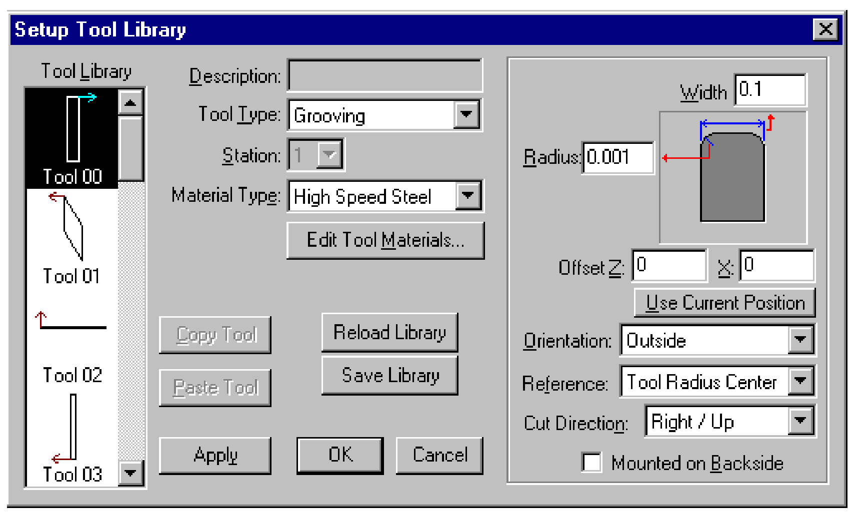
2. Select the **Run/Continue** command to restart the NC program. You will not have to reset the initial tool position (assuming the first block of your program moves the tool to the start position).




# Tools Menu

The Tools Menu commands allow you to select tools, and to set up and use a tool library.

| Command: | Used to: |
| --- | --- |
| **Setup Library** | Define tools used with the Turning Center. |
| **Select Tool** | Select a tool for use on the Turning Center. |




## Setup Library Command

Use the Setup Library command under the Tools Menu to assign parameters to multiple types of tools used on the Turning Center. When you select the Setup Library command, the Setup Tool Library dialog box appears.

### Creating a New Tool

There are two ways to create a new tool in the Setup Tool Library dialog box.

- Use the Copy and Paste buttons to copy an existing tool in the Tool Library box and paste it into the Tool Library box under an unassigned tool number.

Note:

Station is used for tool turret configuration.

Assigned tool numbers are displayed with a tool icon. Unassigned tool numbers have no tool icon.

- Manually create a tool using the features available in the Setup Tool Library dialog box.

## To manually create a new tool:

1. Select an unassigned tool number from the **Tool Library** list.
2. Select a tool type, such as Grooving, from the **Tool Type** drop-down menu.
3. Enter a name for the tool in the **Description** field.
4. Enter the **Material Type** from which the tool is made, such as High Speed Steel.
5. Enter a tool **Width**, **Radius**, **Right Angle**, and/or **Angle** depending on the type of tool that you have selected.
6. Enter a tool **Offset** value. You may also click on the **Use Current Position** button to establish the current Z, X position of the tool as the Offset.
7. Enter an **Orientation**, either inside (for boring, tapping or drilling applications),outside (for cuts made on the outside of a workpiece), or facing.
8. Enter a point of **Reference**, either Tool Radius Center for the middle of the workpiece or Theoretical Sharp Corner for the top or bottom edge of the workpiece.
9. Enter a **Cut Direction**, either Right/Up (cutting away from the chuck) or Left/Down (cutting toward the chuck).
10. If the tool is mounted on the backside, check the box labeled **Mounted on Backside**. See page A-12 for a diagram.
11. Apply the new parameters to the selected tool number by clicking the **Apply** button.
12. Press **Enter** or click on **OK** to accept the new tool information. Click on **Cancel** to exit the Tool Library dialog box without changing the tool library.

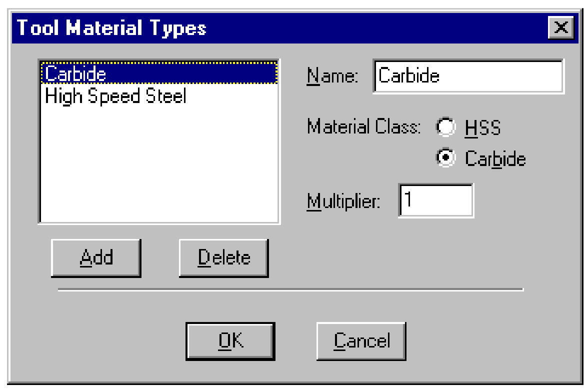
## To alter an existing tool:

1. Select an existing tool from the **Tool Library** list.

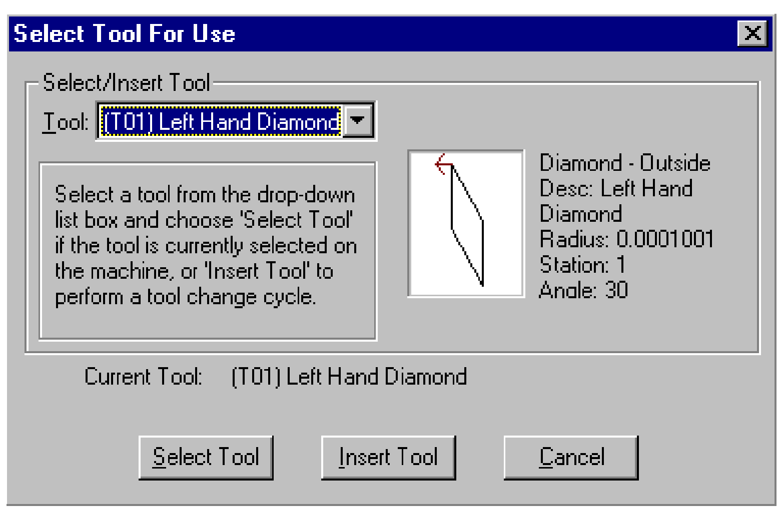
## Creating and Editing Materials

There is a secondary library for tool materials. You can use this library to create new materials or edit existing materials. To do this, click on the **Edit Tool Materials** button, then:

1. Click on the **Add** button.
2. Enter a material **Name**.
3. Select a **Material Class**.
4. Enter a **Multiplier**. This should be set to "1" for now. This is used when integrating with CAM for calculating feed rates and spindle speeds when generating tool paths.
5. Click on the **Delete** button to remove tool materials you no longer need.
6. Press **Enter** or click on **OK** to accept the new material. Click on **Cancel** to exit the Tool Material Type dialog box without changing the material library.




2. Make the desired changes to the tool parameters, then click on the **Apply** button.



3. Press **Enter** or click on **OK** to accept the new tool information. Click on **Cancel** to exit the Tool Library dialog box without changing the tool library.

## Select Tool Command

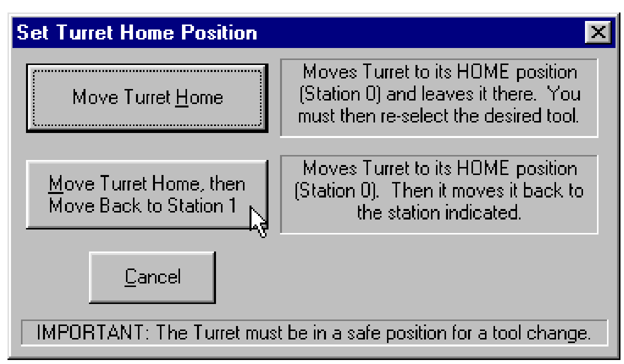
Use the Select Tool command under the Tools Menu to select a tool for use on the Turning Center.

To select a tool:

1. elect the **Select Tool** command from the Tools Menu. The Select Tool ɔr Use dialog box appears.
2. elect a tool from the drop-down **Tool** list. The tool parameters appear ı the window to the right of the list.
3. elect an action to exit the dialog box:
   - Click **Select Tool** button if the tool is already in the mounted spindle.


   - Click **Insert Tool** to perform a tool change cycle. When the Turning Center spindle stops, you are prompted to insert a tool into the tool post. After you insert the tool and press F5, the Turning Center returns to its original position.

     When using a tool turret which has already been homed, the Insert Tool command will turn the turret to the station that holdes the se-



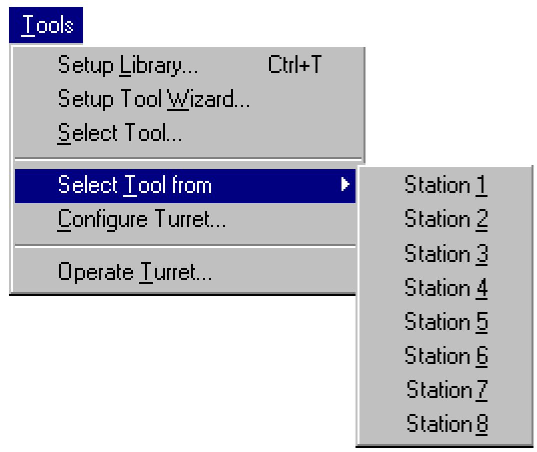

     lected tool. If the turret has not yet been homed, the following dialog box will appear.

     Click on OK to home the tool turret.

## Select Tool From Command

After choosing and mounting tools in the turret, you must identify for  the software program which tools are in the turret and which positions they are in. When you have defined the tools and assigned each one a tool station, the turning program will then know which position to rotate the turret to when cutting. The foolowing tool turret commands will be available to you.

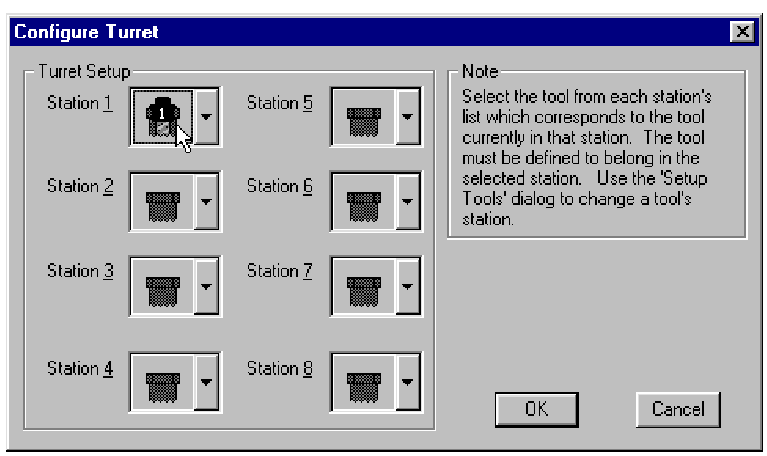
The Select Tool From command provides quick access to each of the tools which are defined for the tool turret. Each of the eight stations are listed for easy selection.




## Configure Turret Command

Once you have defined tools to be used by the tool turret, you must identify where the tools are located in the tool turret. This allows the software to move the turret to the proper place to perform the cutting operations.

All eight tool stations are displayed in the Configure Turret dialog box. when you defined the tool, you also should have selected a station. If you click on the station box, the tool number (which may be  different than the station number) will show below the box, allowing you to selectit.



## Operate Turret Command

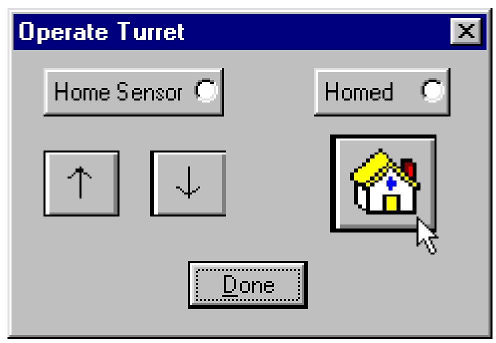
The Operate Turret dialog box allows you to home the Tool Turret, as well as test turret operation.

Click on the **Home** button, indicated by the icon, to move the turret to the home position. When the turret reaches the home position, a black dot appears the circle on the **Homed** button.

The up and down arrows under the Home Sensor button allow you to check the operation of the sensor. Clicking on either arrow will move the turret in the indicated direction, until the turret encounters either a hard stop or trips the home sensor. When the Home Sensor is activated, a dot appears on the button. Clicking on the button has no affect on the sensor.

The **Home Sensor** button and arrows are primarily for diagnostic purposes, and are used to verify the operation of the turret and the sensor.

Click on the **Done** button to exit the Operate Turret dialog box.



The Set Turret Home Position box appears. You may click on Move Turret Home to move the turret to its home position and leave it there. you must then reselect the desired tool. You can also Click on Move Turret Home, then Move Back to Station1, or a different station as indicated. The turret will then home and then move to the proper station.

- Click **Cancel** to exit the Select Tool for Use dialog box without selecting a tool.


## Setup Menu

The Setup Menu commands control the parameters for tool positioning, jogging, running and verifying programs, coordinate systems, tool offsets, etc.

| Command: | Used to: |
|---|---|
| **Set Position** | Establish the X and Z position of the tool. |
| **Zero Position** | Set the current tool position to X0, Z0. |
| **Jog Settings** | Establish speed and distance parameters for jogging the tool. |
| **Run Settings** | Establish options for running an NC part program. |
| **Verify Settings** | Establish options for verifying an NC part program. |
| **Set/Check Home** | Establish or check a fixed known position on the machine. |
| **Goto Position** | Automatically move the tool to a specific set of coordinates on the Turning Center. |
| **Units** | Select Inch or Metric units of measure. |
| **Coordinate Systems** | Define multiple coordinate systems for the Turning Center. |
| **Offsets** | Modify the table of Offset values used for certain NC codes. |
| **Spindle** | Specify a spindle speed if you have not used an S code in your NC program. |

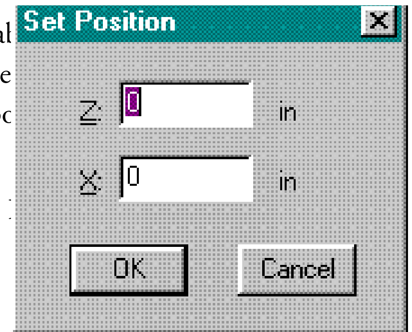
**Note:**

This command sets the position of the tip of the tool to the specified values. It takes the tool's defined x and z offsets into account.

| | |
|---|---|
| **Backlash** | Define the amount of play in the Turning Center turning screws. |
| **Soft Limits** | Establish software limits for each axis that are different than the actual fixed hardware limits on the Turning Center. |
| **Preferences** | Establish defaults for saving files and security features. |

## Set Position Command

Use the Set Position command under t
positions for the tool. This command estal
in relationship to the Machine Coordinate
dinates for the tool also defines the zero pc
absolute motion.




This command is also available under the

To set a new position:

1. Move the tool to the desired position.
2. Select **Set Position** from the Setup Menu. The Set Position dialog box appears.
3. Enter a new **Z** position.
4. Enter a new X position.
5. Press **Enter** or click on **OK**.

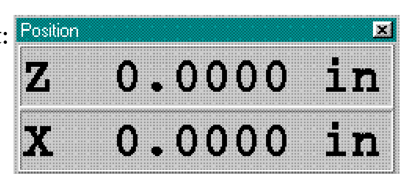
The new position is displayed in the Position Window.

## Zero Position Command

Use the Zero Position command under the Setup Menu to reset the point of origin (0,0) at any position on the Turning Center. Since the tool length and the workpiece position on the cross slide may vary from one tooling setup to another, the zero position must be initialized each time the setup is changed.

This command is also available under the Position Window Pop-up Menu.

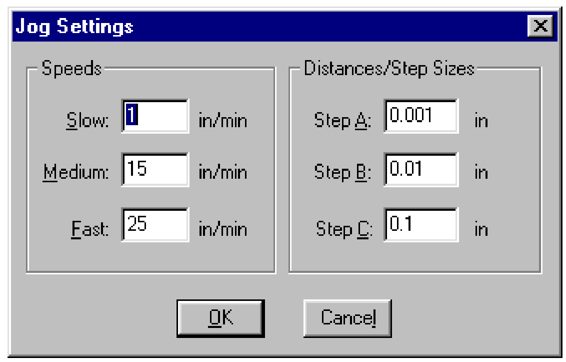
To set the zero point:



1. Move the tool to the point on the workpiece you intend to establish as the zero point.
2. Select **Zero Position** from the Setup Menu. The new position (0,0) is displayed in the Position Window.

## Jog Settings Command

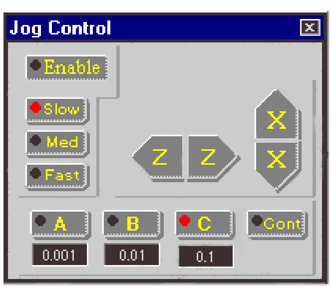
Use the Jog Settings command to open the Jog Settings dialog box. This dialog box is used to enter speed and distance values for the Jog Control Panel. The




Jog Settings command is also available through the Jog Control Panel Pop-up Menu.

To set jog parameters:

1. Select **Jog Settings** from the Setup Menu. The Jog Settings dialog box appears.
2. Enter the desired jog speeds and distances.
3. Select **OK** or press **Enter**. The new values are applied to the Jog Control Panel.




After setting the jog speeds and steps in the Jog Settings dialog box, the tool is easily moved using the Speed and Step buttons on the Jog Control Panel.

### Jog Speed

The jog speed is the rate at which the tool moves along the X and Z axes. Select the speed by entering the desired speed in the Jog Settings dialog box, then press the appropriate button (Slow, Medium or Fast) on the Jog Control Panel.

The default values for speed are:

- 1 ipm for Slow
- 15 ipm for Medium
- 25 ipm for Fast

These feed rates can be set as high as 30 inches per minute.

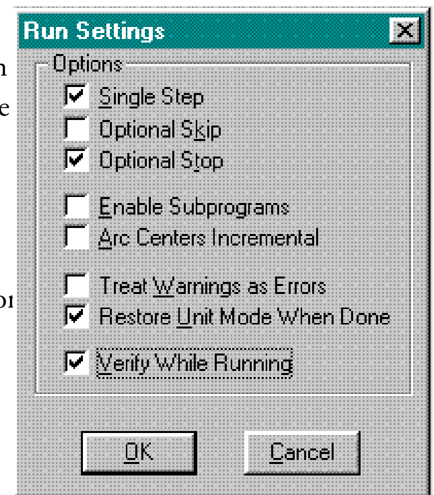
### Jog Distance (Steps)

Distance values determine how far the tool moves each time a key is pressed. Referred to as Steps, the distance is selected by pressing the A, B or C buttons on the Jog Control Panel. Using the Jog Settings dialog box, the distance can be set at a low value (for instance 0.0005 inch) to move the tool for a precise cut, or at a high value (e.g. 5 inches) to position a tool.

These are the default values for steps:

- 0.001 inch for Step A
- 0.01 inch for Step B
- 0.1 inch for Step C

Pressing the Continuous (Cont.) button
as the axis button is depressed. Once the
stops.




## Run Settings Command

Use the Run Settings dialog box to set o
program. The available options are:

- Single Step
- Optional Skip
- Optional Stop
- Enable Subprograms
- Arc Centers Incremental
- Treat Warnings as Errors
- Restore Unit Mode When Done
- Verify While Running

### Single Step

This option inserts a Pause after each block of the NC program. To move on to the next block in the program, you can:

- Click the **Run** button on the Standard Toolbar.
- Click the **GO** button on the Message Bar.
- Press **F5**.
- Press **Enter**.

◆ Select **Run/Continue** from the Program Menu.

## Optional Skip

Use this option to enable or disable the optional skip code. The optional skip code allows you to skip blocks of code as the NC program is run.

Make sure to check off the Optional Skip box in the Run Settings dialog box or activate the Optional Skip button on the Operator Panel. Then place a forward slash ( / ) in front of each line in the NC program you want to skip.

With Optional Skip off, each skip code is ignored and each block of code is executed. With Optional Skip on, each skip code is recognized and each block of code that has been tagged with a skip code is skipped.

To execute particular blocks every nth pass, place a number after the optional skip. For example: **/5G28; Home every fifth pass**

## Optional Stop

Use this option to enable or disable the optional stop code (M01). The optional stop code allows you to place an optional stop in your NC program.

Make sure to check off the Optional Stop box in the Run Settings dialog box or activate the Optional Stop button on the Operator Panel. Place an M01 on the line of code where you would like to pause.

With the Optional Stop option on, the M01 works like a G05. With Optional Stop off, the M01 code is ignored, the other codes on the block are executed as usual.

Note:

The NC program itself can override the default by placing the Incremental Arc Centers (%) or Absolute Arc Centers ($) codes in the first line of the file.

## Enable Subprograms

Use this option to enable or disable the use of subprograms. With this option disabled, M98 commands generate an error. Running or verifying a program with subprograms enabled takes longer to start because the software parses the entire file for subprogram information. This extra delay should only be noticeable with very large programs.

## Arc Centers Incremental

Use this checkbox to specify the default mode for programming arc centers. If this box is checked, the default mode is the Fanuc mode, in which arc centers are always incremental. If this box isn't checked, the default mode is EIA-274, in which arc centers follow the general programming mode: absolute when the mode is absolute, and incremental when the mode is incremental.

## Treat Warnings as Errors

This command is used for special applications, such as laser welding, where you don't want any unexpected pauses in the program execution. For example, when a warning is displayed and the program pauses, waiting for your input before it continues.

When this item is checked, any warning will halt the program, performing a program Stop. When motion is stopped, all outputs are turned off.

### Restore Unit Mode When Done

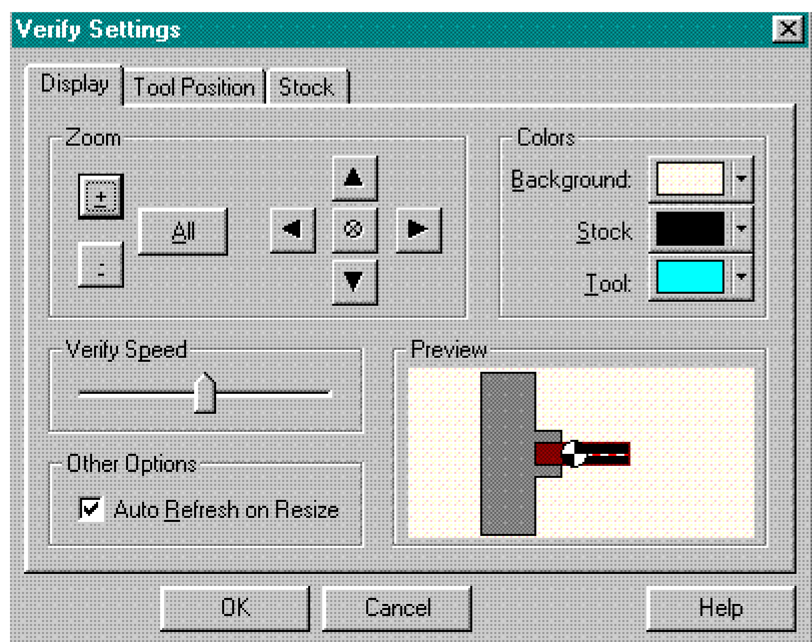
If you normally work in one unit mode (inch or metric) but would like to run a program in another mode without disrupting your default settings, check this box.

Select a specific unit mode by using one of the G20/G21 or G70/G71 commands at the beginning of your NC program. Once the NC program is executed, your default unit mode will be restored.

For instance, if you normally work in Inch Mode, but have a particular program you would like to run in Metric Mode, check the **Restore Unit Mode When Done** box. Place a G21 code at the beginning of your program, then run it. When the program is finished, the default for your system will still be Inch Mode.

### Verify While Running

If this box is checked, the Verify window will display the program verification while the program is running. The verification does not show exactly what is




happening on the Turning Center. There is a delay between each tool motion. You will see each tool motion on the screen, but the *screen will pause* until the machine finishes the motion and the next program block is read.

# Verify Settings Command

Use the Verify Setup dialog box to control the appearance of the tool path verification. You  can also access this dialog box using the Verify Window Pop-up Menu.

The Verify Settings dialog box is tabbed, with the settings organized into three groups: the Display Panel, Tool Position Panel and Stock Panel.

## Display Panel

The Display Panel allows you to control the view. It offers a selection of view preferences.

**Zoom**

Use these buttons to control the size of the stock in the Verify Window. You can click on the buttons or use Alt key combinations (**Alt++**, **Alt+-**, and **Alt+A**). Each mouse click or key combination used zooms by an increment of one.

| Button | Function |
|---|---|
| + | Zoom in on the stock. |
| - | Zoom away from the stock. |
| All | Fit the stock into the window. |
| Arrows | Move in the indicated direction. |
| | Center the stock. |

**Colors**

Use these buttons to select which colors will indicate the background, stock, and tool in the Preview Window. Color options can be viewed by clicking on the down arrow to access a bar of various colors.

**Verify Speed**

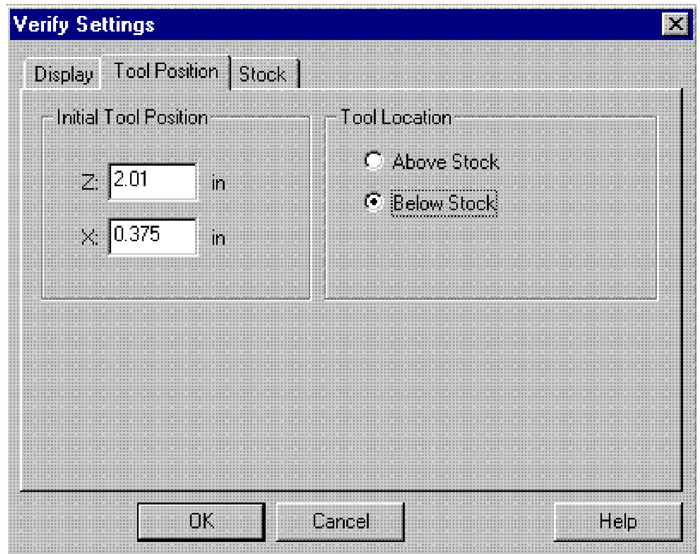
Use the slider to alter the speed at which the Verify Program will be executed in the Verify Window. If you would like to view the verify program in slower motion, move the slider to the left. If you would like to view the Verify Program at a faster rate, move the slider to the right.

## The Preview Window

The Preview Window appears in the Display and Stock Panels. The Preview Window shows you approximately what the Verify Window will look like. The preferences for the Preview and Verify Windows are set in the Display Panel.

### Other Options

If the Auto Refresh on Resize box is checked, the Verify Window automatically refreshes the display of the workpiece if the Verify Window is resized during verification. When Auto-refresh is disabled, you will need to manually cause the verify window to update by selecting the Redraw command from the window's context menu. This setting only affects refreshing the window when no verification is in progress.




## Tool Position Panel

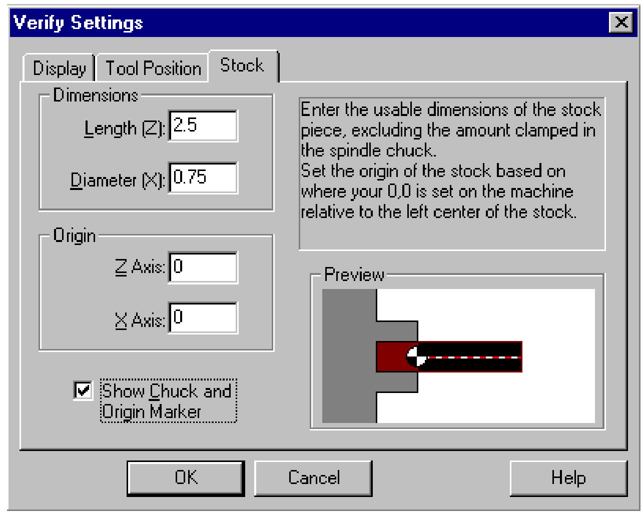
The Tool Position Panel controls the initial tool position and the tool location in reference to the stock.

### Initial Tool Position

You can select a tool start point using Initial Tool Position. Only use Initial Tool Position for verification, not for actually running a program. The Initial Tool Position is used when you verify a program; when verifying an NC program while running it, the actual physical position of the tool is used as the initial tool position.

### Tool Location

The specification of tool location depends on where your tool will be mounted according to the NC program you are using.  You must check ei-




ther **Above Stock** or **Below Stock**. Usually the a single tool is mounted on the tool post below the stock and the optional tool turret is mounted above the stock.

## Stock Panel

The Stock Panel allows you to specify particular information about the stock and tool position for tool path verification.

All stock values are interpreted according to the Units Mode in effect at the time they are set. To quickly see which Units Mode is currently in effect, check the Position window.

### Dimensions

Use this area to set the dimensions of the stock used in the verification process. You will see the stock in the Preview Window change as soon as you enter a dimension.

### Origin

## Buttons

The **OK** button at the bottom of the dialog box applies the changes you have made and closes the dialog box.

The **Cancel** button closes the dialog box without applying any of the changes you have made.

The **Help** button brings up the Help topic.

Use this area to adjust the verification for different workpiece setups. Most NC programs set the 0,0 point at the left center of the stock, immediately in front of the chuck. The origin of the stock must be based on this 0,0 point.

The origin of the stock is always 0,0. The stock initialization point varies depending on where the origin is relative to the stock.

**Show Chuck and Origin Marker**

**Note:**

While the Reference Point Wizard is active, the machine feed rate is limited to 2.5ipm. You can not alter this feed rate.

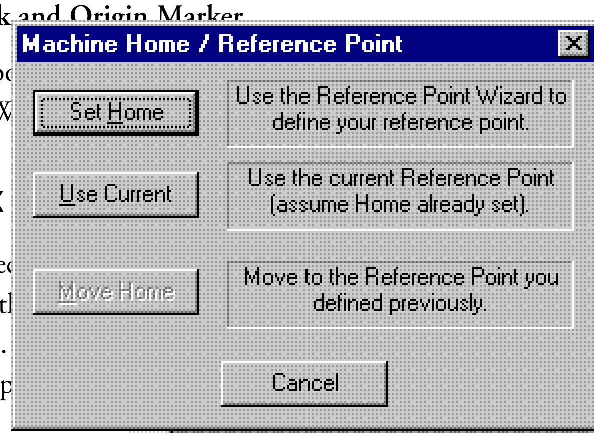
Check this bo... he Preview and Verify W...

## Set/Check ...

**Important:**

If you are using G27 or G28 codes in your NC program, you must use the Reference Point Wizard to home the machine first.

The Set/Chec... a reference position on t... position on the machine. ... he following dialog box ap...



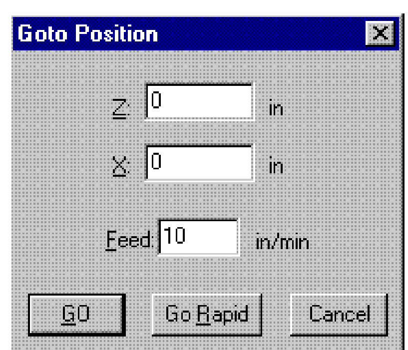

**Note:**

If you would like the tool to travel at the Rapid feed rate, click on the Go Rapid button instead.

The **Set Home** button opens the Reference Point Wizard, which aids you in setting a machine reference point. Just follow the directions.

The **Use Current** button uses the last defined Reference Point. Home must already be set in order for this button to work.

1. Enter the coordinates for the new tool position.
2. Enter the feed rate at which you would like the tool to travel.
3. Click on the **Go** button. The tool moves to the new position at the defined feed rate.



The **Move Home** button moves the machine to the reference point coordinates so you can manually measure the machine position to make sure no position has been lost.


IMPORTANT!

Machine coordinates are established by Homing the system to establish a point of origin at the ends of travel on the Turning Center. Once established, these coordinates remain fixed. Each time you turn on the Turning Center, the current machine position is stored as the machine coordinate system X0, Z0.

Work coordinates are different. When you use the Set Position command to set a point of origin on the workpiece, you are actually entering a position that is offset from the fixed machine position. The same thing occurs when you use a G92 code.

Work Coordinates are not fixed; they can be established anywhere on the system by using either the Set Position command or the G92 code.

Within the Work Coordinates you can set separate coordinate systems using the codes G54 through G59, or the Setup Coordinate Systems dialog box.

If you reset the Work Coordinates either with a G92 code or in the dialog box, the G54-G59 Coordinate System offsets will change accordingly.

If you Home the machine while using Coordinate systems, the offsets will not be affected.

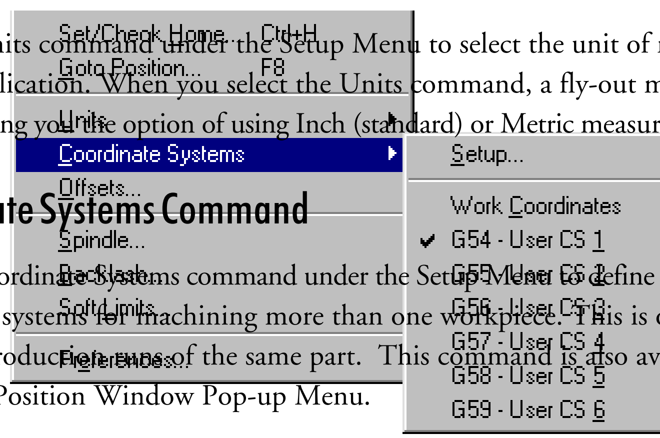
## Goto Position Command

The Goto Position command opens the Goto Position dialog box. Use the Goto Position dialog box to move the tool to a particular coordinate position on the Turning Center.

This command is also available under the Position Window Pop-Up Menu and Jog Control Panel Pop-up Menu. To use the Goto Position command:

## Units Command

Use the Units command under the Setup Menu to select the unit of measure for the application. When you select the Units command, a fly-out menu appears allowing you the option of using Inch (standard) or Metric measurement.

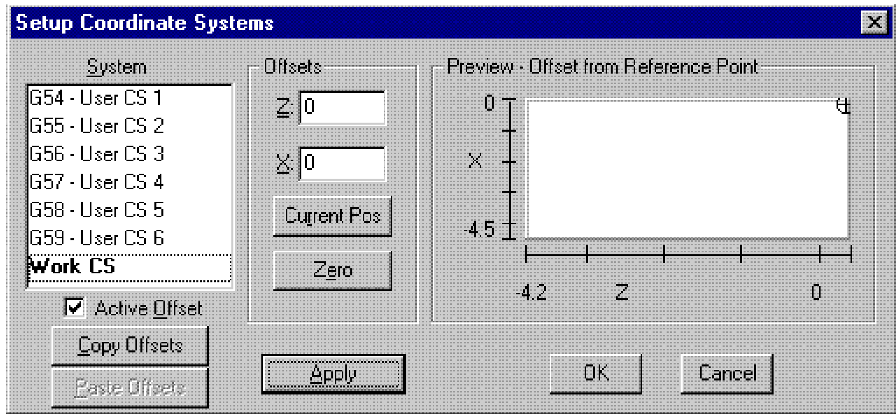



## Coordinate Systems Command

Use the Coordinate Systems command under the Setup Menu to define multiple coordinate systems for machining more than one workpiece. This is often done for production runs of the same part. This command is also available under the Position Window Pop-up Menu.

### To select an existing coordinate system:

1. Select **Coordinate Systems** from the Setup Menu. The coordinates fly-out menu appears.




2. Select an existing coordinate system from the fly-out menu. Select one of the available coordinate systems available, CS1 through CS6, (these are equivalent to using the codes G54 through G59 in your NC program).

   OR

   Select the **Work Coordinates** command to cancel the Coordinate System offsets and return to Work Coordinates.

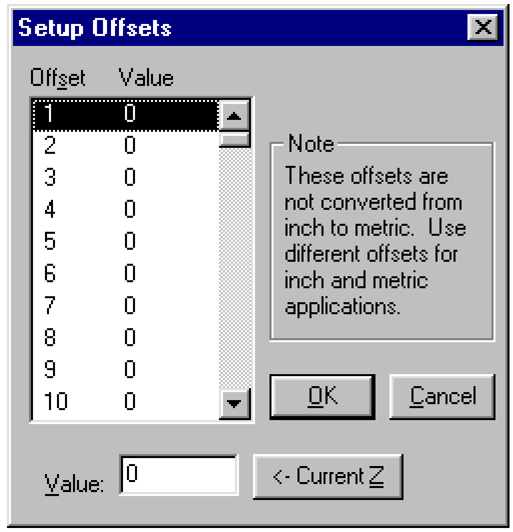
### To define a new coordinate system:

1. Select **Coordinate Systems** from the Setup Menu.
2. Select **Setup** from the fly-out menu. The Setup Coordinate Systems dialog box appears.
3. Select a coordinate system, then enter and apply the offsets.
   a) Select a CS from the **System** list.

      The system titled "Work CS" contains the current values for the work coordinate system. This is like adding a G92 code to your NC program. You can change the default for the work coordinate system by entering new values in the Offsets boxes. These values are an offset from the true origin of a coordinate system and affect all coordinate systems.

   b) Select the offsets for a user coordinate system by:
      - Entering X and Z offset values in the **Offsets** boxes.

- Clicking the **Current Pos** button to establish offset values based on the current tool position.
- Selecting a Marker in the **Preview Area** for the coordinate system and dragging it to the desired position.




- Copying offsets from one CS to another using the **Copy Offsets** and **Paste Offsets** buttons.

c) Make the currently selected CS the active CS by checking the **Active Offset** box.

d) Apply the coordinates or exit the dialog box:

- The **OK** button applies the changes you have made and closes the dialog box.
- The **Cancel** button closes the dialog box without applying any of the changes you have made.
- The **Apply** button applies the changes you have made to the selected CS (you can still cancel changes once they have been applied by selecting the Cancel button).

## Offsets Command

Use the Offsets command under the Setup Menu to compensate for variations in the cutting tools being used. The offset values are used for cutter compensation and tool offset adjustment NC codes.

### To enter a compensation offset:

1. Select **Offsets** from the Setup Menu. The Setup Offsets dialog box appears.

2. Click on an Offset Number (the numbers 1 through 199 are available). This number only acts as a designation (a name) for the offset.

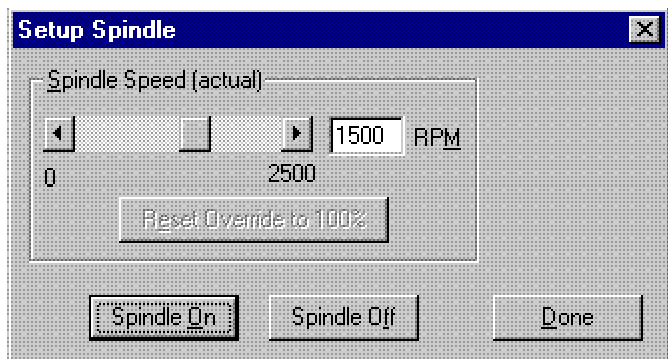
Note:
This feature does not override an S code in the NC program.

3. Enter an Offset Value in the **Value** box, or select the **Current Z** button to use the current Z position (also useful for defining tool heights if you are using G43 or G44 to specify tool heights).

4. Press **Enter** or click on **OK**. The Offset Value has been associated with the Offset Number. The next time you open the Offsets table, you will see the new offset.

### The Offset Table

The Offset Table stores up to 200 values which are used in several operations including tool offset adjustment, cutter compensation, and tool length compensation to ensure uniform application of an offset value. The numbers are stored as unit-less values; their interpretation depends on the Units default currently in effect.




Set the values in the Offset Table using the Offsets command under the Setup Menu.

## Spindle Command

Use the Spindle command under the Setup Menu to specify a spindle speed if you have not used an S code in your NC program.

The spindle speed is primarily determined by the Mode Switch on the Turning Center front panel. If the Mode Switch is set to Manual Mode, spindle speed is controlled by the Spindle Speed Control Knob, also on the front panel. If the Mode is set to computer, spindle speed is determined by an S code in the NC program. If there is no S code in the NC program, spindle speed is determined by the Setup Spindle dialog box.

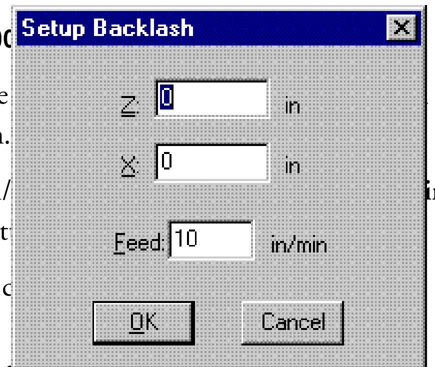
### To specify a spindle speed:

1. Select the **Spindle** command from the Setup Menu. The Spindle Setup dialog box appears.

**Note:**
You should not specify too slow of a backlash feed rate or you will notice delays each time an axis with backlash changes direction.

2. Select a spindle speed by entering a value in the RPM box, or by using the slider and arrow buttons.

### Also in this dialog bc




- Reset the spindle … the **Reset Override to 100%** button.
- Change the On/ … **indle On** and **Spindle Off** but…
- Select **Done** to c…

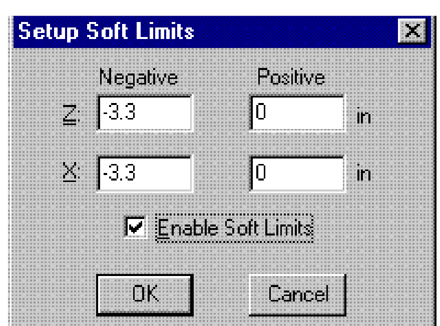
## Backlash Command

Use the Backlash command under the Setup Menu to define the amount of play in the Turning Center turning screws. The system default is set at a backlash value of 0.0 on both axes, with a feed rate of 10 ipm.

To establish new backlash parameters:

1. Select **Backlash** from the Setup Menu. The Setup Backlash dialog box appears.

**IMPORTANT!**
Soft limits are defined in relation to the machine coordinates; therefore, you must home the Turning Center before using soft limits. Soft limits are not enforced if the Turning Center has not been homed.



**Note:**

You must be within the Soft Limit range in order to Enable the Soft Limits. If you are outside of the Soft Limit range and Soft Limits are enabled when you close the dialog box, they will be automatically disabled and you will be instructed to jog the machine to within the Soft Limits. After you do so, you can open the Soft Limits dialog box and enable them.

2. Enter the desired backlash distances and feed rate.
3. Press **Enter** or click on **OK** to accept the new backlash parameters, or click on **Cancel** to exit the Setup Backlash dialog box without setting new backlash parameters.

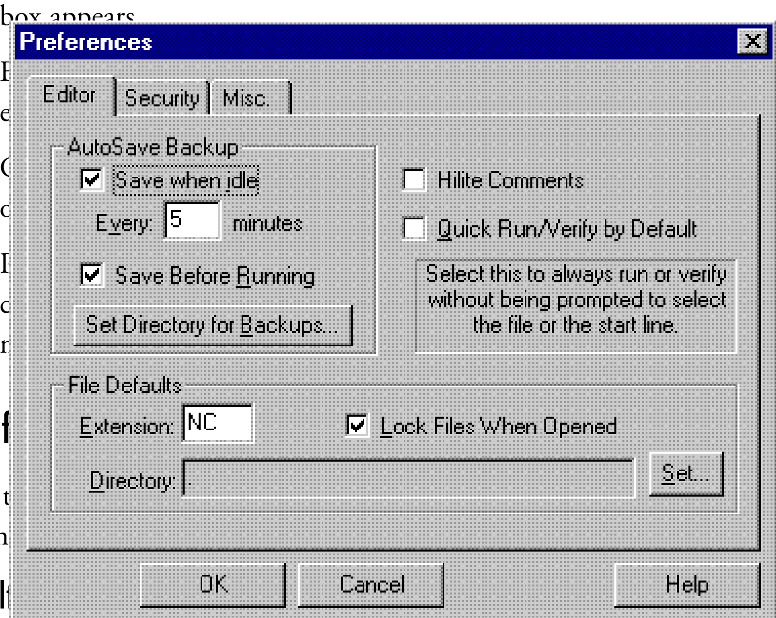
## Soft Limits Command

Use the Soft Limits command under the Setup Menu to establish software limits for each axis. The limits are different than the actual fixed hardware limits on the Turning Center. Soft limits can confine the tool travel to an area smaller than the normal maximum travel.

The Turning Center shuts down if it trips a soft limit, just as it does when it trips one of the limit switches. This is helpful when working with devices such as robots, or when you have installed fixtures within the normal work area that you don't want the tool to hit.

To establish software limits on the Turning Center:

1. Select **Soft Limits** from the Setup Menu. The Setup Soft Limits dialog box appears.
2. E … n to e …
3. C … is c …
4. I … or c … ing r …




## Pref…

Use t … ts for savin…

### To al…

1. Select the **Preferences** command from the Setup Menu. The Preferences dialog box appears.

2. Select one of the **Editor** ,**Security** or **Misc.** preferences tabs.

## Editor Preferences

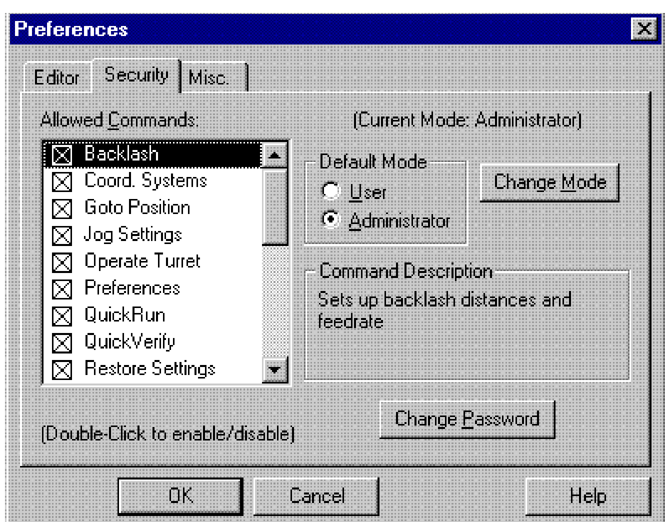
The Editor preferences allow you to automatically save your NC program files at regular intervals, and establish a default directory in which to store your files. When you select the Editor tab in the Preferences dialog box, the Editor panel appears (depicted above).

**To select AutoSave features:**

- Select **Save when idle** and enter a value in the **Every:___ minutes** box to save your NC programs automatically at the specified time increment.
- Select **Save Before Running** to save changes to your NC program prior to running it for the first time with the changes.

**To select File Default features:**

- Enter an **Extension** for your NC part program files. The default is "NC."
- Select **Lock Files When Opened** to have your NC programs locked by default. Deselect this feature to have your NC programs unlocked by default.
- Select the **Set** button to specify a target directory in which to save your NC program files. The default directory appears in the Directory box.



**Other features:**

- Select **Hilite Comments** to distinguish the comments from commands in your NC program.
- Select **Quick Run/Verify by Default** to always run or verify without being prompted to select the file or the start line.

**To exit the Preferences dialog box:**

Press **Enter** or click on **OK** to accept the new preference settings, or click on **Cancel** to exit the Preferences dialog box without setting new Editor or Security preferences.

**To access Help for this panel:**

Select the **Help** button to access the Help files for this panel.

IMPORTANT!

As a security measure, the following Setup features are disabled by default when the software is in User Mode.

- Coordinate Systems (and related features)
- Offsets
- Soft Limits
- Preferences
  The command is enabled, but selecting it will prompt for the administrator password before the Preferences dialog box appears.
- Backlash

The Restore Settings command under the Help Menu is also disabled by default.

## Security Preferences

The Security preferences allow you to control which features others may use. When you select the Security tab in the Preferences dialog box, the Security panel appears.

The Security preferences panel offers two modes, User and Administrator. Administrator Mode allows a supervisor, such as a teacher in a classroom, to turn commands on or off using the Allowed Commands list. User Mode does not have access to this feature.

**To secure the software using Administrator Mode:**

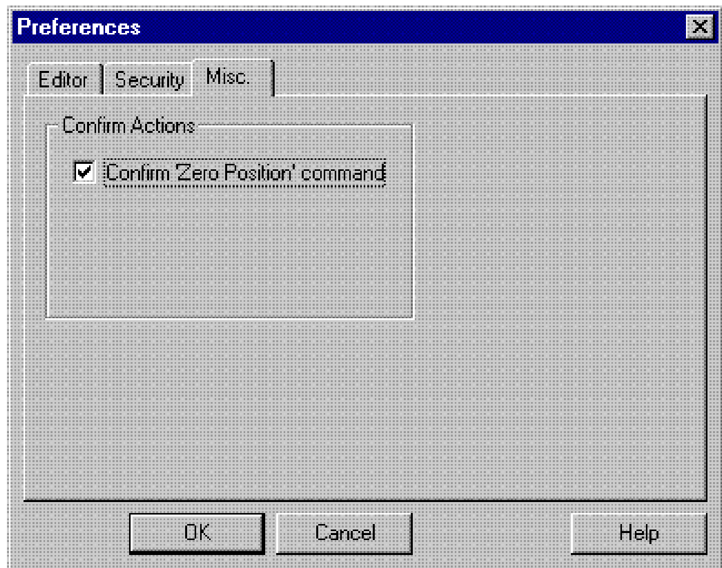
1. Set the **Default Mode** to **User.** (This will not change the current mode.)
2. Use the **Change Password** button to create a password. The default password is blank — no password.
3. Select the **Allowed Commands**. Double-click on the listed commands to enable or disable them. If the commands are enabled, they are marked with an X. A description of each selected command is displayed on the right side of the panel.
4. Use the **Change Mode** button to change to User Mode.
5. Select **OK** or press **Enter** to exit the Preferences dialog box.

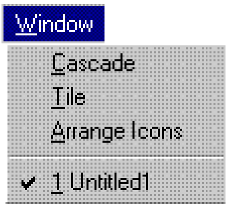
The software is now running in User Mode. The next time you open the Preferences dialog box, the Security preference panel is displayed in User Mode. In this mode it is not possible to turn commands on or off.

**To return to Administrator Mode:**

## Misc. Preferences

The Misc. preferences allows you to choose to Confirm the reference point by checking the Confirm 'Zero Position' Command box. By checking this box, the warning "Are you sure you want to reset the current position to ZERO on all axes?" will appear every time the Zero Position Command is activated in the Setup menu. You must then select Yes to apply the command.






1. Use the **Change Mode** button to toggle the Mode from User to Administrator. A dialog box appears, prompting you to enter your password.
2. Enter your password and press **Enter** or click on **OK**.

**To change your password:**

1. Click on the **Change Password** button.
2. Enter your current password.
3. Enter the new password.
4. Enter your new password again to verify that it is correct.

**To exit the Preferences dialog box:**

Press **Enter** or click on **OK** to accept the new preference settings, or click on **Cancel** to exit the Preferences dialog box without setting new Editor or Security preferences.

**To access Help for this panel:**

Select the **Help** button to access the Help files for this panel.

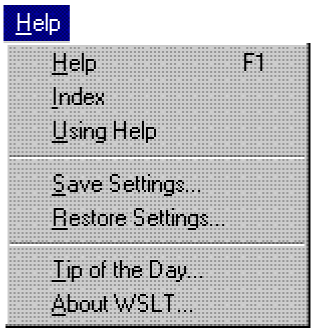
# Window Menu

The Window Menu commands allow you to manipulate the arrangement of the Program Edit Windows.

| Command: | Used to: |
|---|---|
| **Cascade** | Layer the open edit windows. |
| **Tile** | Tile the open edit windows. |
| **Arrange Icons** | Arrange any minimized edit windows along the bottom of the edit area. |
| **Window List** | Display and select between the currently open NC programs by their file names. |

## Cascade Command

Similar to the standard Windows/Cascade command, this command places the open Program Edit Windows in a layered format, cascading down to the right with the currently selected window on top.




## Tile Command

Similar to the standard Windows/Tile command. Places the open Program Edit Windows in a tiled format, filling the Edit Area from top to bottom.

## Arrange Icons Command

When you minimize a Program Edit Window, it becomes a small icon. The Arrange command under the Window Menu arranges these icons along the bottom of the edit area.

## Window List Command

Lists the currently open Program Edit Windows. The currently selected window is designated with a check mark. You may select any window by clicking on the window itself, or by selecting the window name from this list.

# Help Menu

The Help Menu commands allow you to navigate through the Help files, to save or restore parameters set throughout the current session, and provides handy tips and other information about the Control Program.

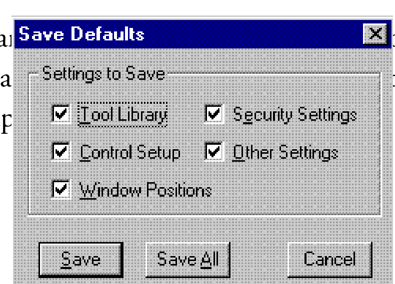
| Command: | Used to: |
|---|---|
| **Help** | Display Help for the current task or command. |
| **Index** | List Help topics. |
| **Using Help** | Display instructions about how to use Help. |
| **Save Settings** | Saves the current machine and application settings. |
| **Restore Settings** | Resets the machine and application settings from defaults. |
| **Tip of the Day** | Display a Tip of the Day. |
| **About WSLT** | Display program information, version number and copyright. |

**Note:**

On all panels the Default button resets the Control Program to use factory set defaults. All values on the panels are based on the unit of measure selected on the Setup program Welcome Panel.

## Help Command

Use the Help comma ou can also press the **F1** key to get informa ted command on a drop-down or pop-up



## Index Command

Use the Help Index command access an index of available Help topics.

## Using Help Command

Use the Using Help command to obtain information on how to use the Help utility.

## Save Settings Command

The Save Settings command brings up a dialog box that allows you to retain current library, security, screen and control settings as defaults.


- Click on a particul on the **Save** button to save the selected it
- Click on the **Save** thout having to select each one.
- Click on the **Canc** lts dialog box without changing the existi

## Restore Settings Command

The Restore Settings command allows you to restore all or some of the current settings to the defaults you set using the Save Settings command (User Defaults) or to the factory set defaults.

- Click on a particular setting(s) to tag it, then click on a Restore From option. Click the **Restore** button to restore the selected items and exit the dialog box.
- Click on the **Restore All** button to restore all settings without having to select each one.

- Click on the **Cancel** button to exit the Restore Defaults dialog box without changing the existing defaults.

### Tip of the Day Command

This command brings up information about the operation of the Turning Center, tips and tricks for using the Control Program and NC programming ideas.

### About WSLT ... Command

This command simply brings up an information box. Included is information on the Control Program version number, the release date, and copyright information.

# Selecting Commands

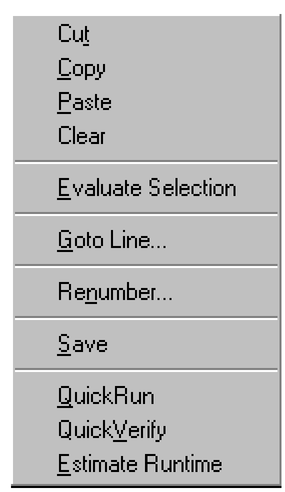
There are a few different ways to select commands in the Control Program. Use the method that is most convenient for you.

## Select a Command Using Pop-Up Menus

Clicking the right mouse button on certain windows or panels brings up a pop-up menu. Each pop-up menu is context-sensitive. Commands which cannot be performed at that time are grayed out.

To select a pop-up menu command:

1. Position the cursor on the window or panel.
2. Click and hold down the *right* mouse button. The context-sensitive menu appears.
3. Highlight a command by moving the mouse pointer over it, then release the right mouse button.




The following windows have pop-up menus:

- Program Edit Window
- Position Window
- Verify Window
- Jog Control Panel

The Machine Info Window and Operator Panel only provide the Dockable and Hide commands on their Pop-up menus.

## Program Edit Window Pop-up Menu

The Program Edit Window Pop-Up Menu contains different combinations of these commands, depending on whether the file is running or being verified, and whether or not text is selected.

### Cut Command

Cut is the same as selecting the Cut command from the Edit Menu.

### Copy Command

Copy is the same as selecting the Copy command from the Edit Menu.

### Paste Command

Paste is the same as selecting the Paste command from the Edit Menu.

### Clear Command

Clear is the same as selecting the Clear command from the Edit Menu.

### Evaluate Selection Command

Evaluate Selection will process an equation which is highlighted and replace it with a result. If the selected text is not a valid numerical equation, there is no result.

### Goto Line Command

Goto Line is the same as selecting the Goto Line command from the Edit Menu.

### Renumber Command

The Renumber command is the same as selecting Renumber from the Edit Menu. A dialog box appears that allows you to alter the N codes in the NC program.

### Save Command

Save brings up the Save As dialog box.

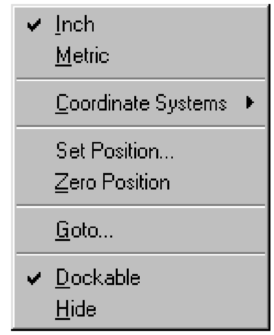
### QuickRun Command

The QuickRun command is a shortcut that runs the currently selected NC part program. When you click the right mouse button on this command, the program behaves as though you had selected the Run/Continue Command from the Program Menu with the following exceptions:

- You do not have the option of selecting a starting line.

- You do not have the option of changing any settings.

### Quick Verify Command

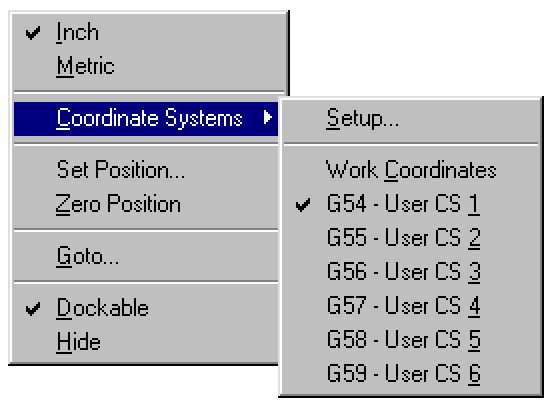



The Quick Verify command is a shortcut that verifies the currently selected NC part program. When you click the right mouse button on this command, the program behaves as though you had selected the Verify Command from the Program Menu with the following exceptions:

- You do not have the option of selecting a starting line.
- You do not have the option of changing any settings.

### Estimate Runtime

This command performs the same function as the Estimate Runtime command under the Program Menu.




## Position Window Pop-up Menu

The Position Window Pop-Up Menu contains these commands:

### Inch Command

Automatically switches the units of measure for the current program to inch units.

### Metric Command

Automatically switches the units of measure for the current program to metric units.

### Coordinate Systems Command

Produces a fly-out menu that allows you to set up and select coordinate systems just as you would using the Coordinate  Systems command from the Setup Menu.

### Set Position Command

Opens the Set Position dialog box.

### Zero Position Command

Sets the current tool position to zero on both axes.

### Goto Command

Opens the Goto Position dialog box.

### Dockable Command

The Dockable command toggles the window between being a dockable window and being undockable. See Docking and Floating Windows for more information.


This command is available on all pop-up menus for the windows and panels available under the View Menu.

### Hide Command

The Hide command closes the window. To open the window again, select it from the View Menu.

This command is available on all pop-up menus for the windows and panels available under the View Menu.

## Verify Window Pop-up Menu

The Verify Window Pop-Up Menu contains these commands:

### Setup Command

This command opens the Verify Settings dialog box.

### Redraw Command

This command repeats the most recent tool path verification simulation.

### Stop Redraw Command

This command interrupts the verification currently in progress.

### Reset Command

This command resets the Verify Window; it clears the tool path lines and resets the tool to the starting position.

### Dockable Command


The Dockable command toggles the window betweer
dow and being undockable. See Docking and Floatin
information.

This command is available on all pop-up menus for the windows and panels available under the View Menu.

### Hide Command

The Hide command closes the window. To open the window again, select it from the View Menu.

This command is available on all pop-up menus for the windows and panels available under the View Menu.

## Jog Control Panel Pop-up Menu

The Jog Control Panel Pop-Up Menu contains these commands:

### Setup Command

This command opens the Jog Settings dialog box.

### Goto Command

This command opens the Goto Position dialog box.

### Set/Check Home Command

This command opens Machine Home/Reference Point dialog box.

### Dockable Command

The Dockable command toggles the window between being a dockable window and being undockable. See Docking and Floating Windows for more information.

This command is available on all pop-up menus for the windows and panels available under the View Menu.

### Hide Command

The Hide command closes the window. To open the window again, select it from the View Menu.

This command is available on all pop-up menus for the windows and panels available under the View Menu.

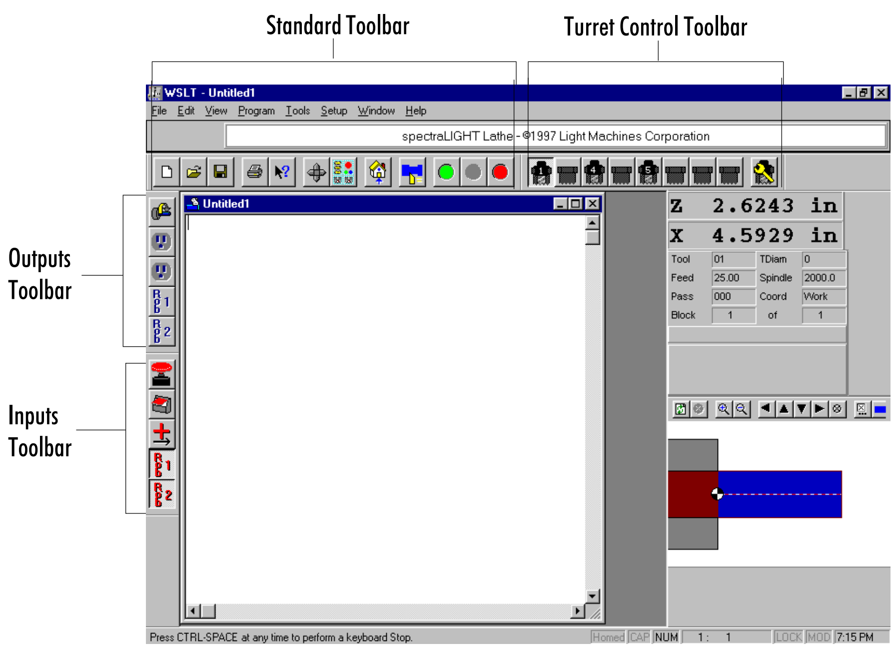
## Select a Command Using Hot Keys

Some menu commands have one or more key designations next to them; those are the *hot keys* for that command. Pressing the hot key(s) selects the corresponding command. For example, press **Ctrl+S** to save your file. Other hot keys are also available for controlling machining functions; for instance, use the **Ctrl+KeyPad+** combination to increase feed rate override.

Here's a list of the available hot keys:

| Key(s): | Menu/Command: |
|---|---|
| **Ctrl+C** | Edit Menu/**Copy** |
| **Ctrl+F** | Edit Menu/**Find** |
| **Ctrl+G** | Edit Menu/**Goto Line** |
| **Ctrl+H** | Setup Menu/**Set/Check Home** |
| **Ctrl+L** | Edit Menu/**Lock** |
| **Ctrl+N** | File Menu/**New** |
| **Ctrl+O** | File Menu/**Open** |
| **Ctrl+P** | File Menu/**Print** |
| **Ctrl+R** | Setup Menu/**Run Settings** |
| **Ctrl+S** | File Menu/**Save** |
| **Ctrl+Space** | Program Menu/**Stop** |
| **Ctrl+T** | Tools Menu/**Setup Library** |
| **Ctrl+V** | Edit Menu/**Paste** |
| **Ctrl+X** | Edit Menu/**Cut** |
| **Ctrl+Y** | Edit Menu/**Delete Line** |
| **Ctrl+Z** | Edit Menu/**Undo** |
| **Ctrl+Shift+Z** | Setup Menu/**Zero Position** |
| **F1** | Help Menu/**Help** |
| **F2** | Edit Menu/**Delete Line** |

| | |
|---|---|
| **F5** | Program Menu/**Run/Continue** |
| **F6** | Program Menu/**Verify** |
| **F8** | Setup Menu/**Goto Position** |
| **Ctrl+KeyPad+** | **Increase Feed Rate Override** |
| **Ctrl+KeyPad-** | **Decrease Feed Rate Override** |
| **Ctrl+Backspace** | Edit Menu/**Undo** |
| **Shift+Delete** | Edit Menu/**Cut** |
| **Shift+F1** | Help Menu/**Context Help** |
| **F4** | Activate **Jog Control** |
| **Ctrl+F5** | **QuickRun** |



| | |
|---|---|
| **Ctrl+F6** | **Quick Verify** |
| **Ctrl+TAB** | **Next Edit Window** |
| **Ctrl+Insert** | Edit Menu/**Copy** |
| **Shift+Insert** | Edit Menu/**Paste** |
| **Pause** | Edit Menu/**Pause** |

### Select a Command Using Toolbars

**Docking Area:**

The gray portion of the screen where toolbars, windows and information areas are placed as stationary objects.

The Tool Bars contain buttons that correspond to frequently used menu commands. Clicking a button on a toolbar is equivalent to selecting the same command from a menu, and is usually quicker. The Control Program provides Standard, Input, Output , and Turret Control toolbars. Use the commands under the View Menu to control whether each tool bar is displayed or hidden.

## Positioning Screen Components

All the toolbars, windows and panels found under the View Menu can be repositioned.

The toolbars include:

- Inputs Toolbar
- Outputs Toolbar
- Standard Toolbar

The windows include:

- Position Window
- Machine Info Window
- Jog Control Panel
- Operator Panel
- Verify Window

The Control Program interface is easily configured and optimized by opening, closing, resizing, and repositioning the screen components.

### Positioning Toolbars

You can reposition any of the toolbars (Standard, Inputs, Outputs, or Turret Control) simply by dragging them off their docking areas. Once away from the docking area, the toolbar becomes a floating window, which can be treated the same as any other window (E.G. move it, resize it, close it). To move the toolbar, click on the toolbar background (the area behind the buttons) and drag.

### Positioning Windows and Panels

The windows and panels, (Position Window, Machine Info, Jog Control Panel, Operator Panel, Verify Window) are initially docked on the docking area on the right side of the screen. If these items are moved away from the docking area, they become floating windows or panels until they are moved back to the docking area. (See *Docking and Floating Windows and Toolbars* on the following page.) The Verify Window is initially not dockable.

#### To move a window or panel (create a floating window):

1. Press and hold the Ctrl key.
2. Click on the window/panel with the left mouse button.
3. Hold and drag the window/panel off the docking area.

5. When the floating window is over its new location, release the mouse button. You can now treat the area the same as you would any other window (e.g., move it, resize it, close it).

### To return a floating window to its docking area:

(The Dockable command must be invoked for a window/panel before it can be docked. See *Docking and Floating Windows and Toolbars*.)

1. Click on the title bar of the floating window.
2. Drag the area back to the docking area.
3. Release the mouse button.

## Positioning Program Edit Windows

Program Edit Windows can be moved, resized or closed just like any other window. The only restriction is that the Edit Windows can not be moved out of the edit area. For instance, you can not move an Edit Window to a docking area.

All the toolbars, windows and panels found under the View Menu can be docked or floating.

The toolbars include:

- Inputs Toolbar
- Outputs Toolbar
- Standard Toolbar
- Turret Control Toolbar

The windows include:

- Position Window
- Machine Info Window
- Jog Control Panel
- Operator Panel
- Verify Window

## Saving the Component Position

After changing a component's position on the screen, you can save it's position by checking the Window Positions box on the Save Settings dialog box (select Save Settings from the Help Menu). The next time you use the Control Program, the component appears in the new position. To restore the default position (the factory preset position), use the Restore Settings command under the Help Menu.

## Docking and Floating Windows and Toolbars

Windows and Toolbars can behave in two ways; they can be placed in a stationary state, docked, or they can be in a free-floating state, and moveable.

### Docking Screen Components

A docked screen component is fixed in place, unlike a floating component which can be placed anywhere on the screen. When a screen component is docked, the window frame and title disappear.

### Floating Screen Components

A floating screen component can be moved to any position on the screen, unlike a docked screen component, which is fixed in place. When a screen component is floating it has a window frame and title.

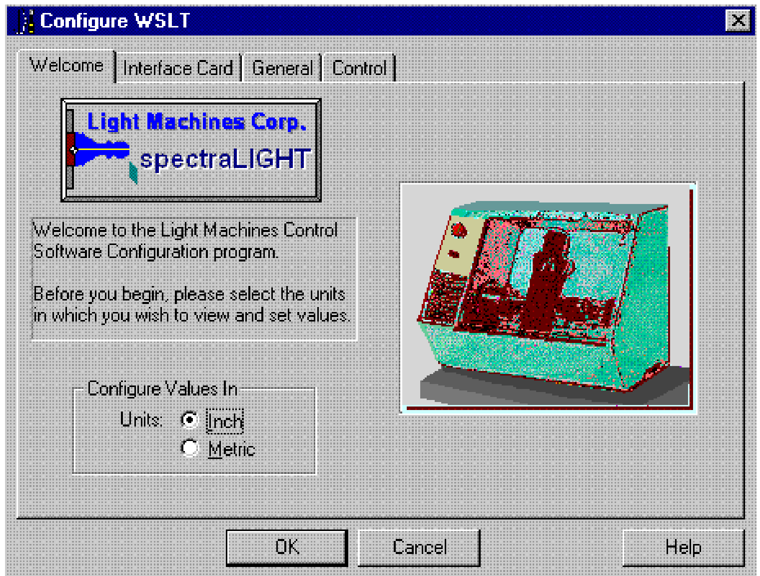
#### To select the Docking/Floating state of a toolbar:

- Click on the background area of the toolbar and drag it away from the docking area to float it.
- Click on the background or the title bar of the toolbar and drag it to the docking area to dock it.

### To select the Docking/Floating state of a window:

1. Position the mouse pointer over the window you would like to dock/float.
2. Click the right mouse button.
3. Select Dockable from the drop-down menu.

When the Dockable command is checked, the window can be dragged to a docking area and docked. Dockable windows can **not** be resized.




When the Dockable command is not checked, the window will float and can not be docked. Some floating windows can be resized.

# Using The Setup Program

The Setup Program allows you to set program and hardware defaults. In order to access the Setup Program, you must first exit the Control program. Locate the spectraLIGHT LATHE folder on your hard drive, (the default installation location is C:\ProgramFiles\Programs\LMC\WSLT) and double click the Setup Icon. The program will start and you will see the Welcome screen. You may choose from the file tabs to view the settings for each category.

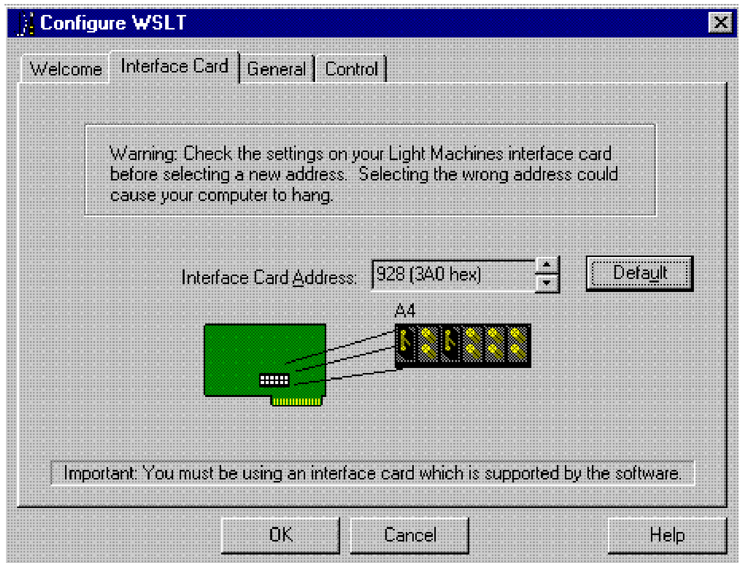
The Setup Program provides settings under the following categories:

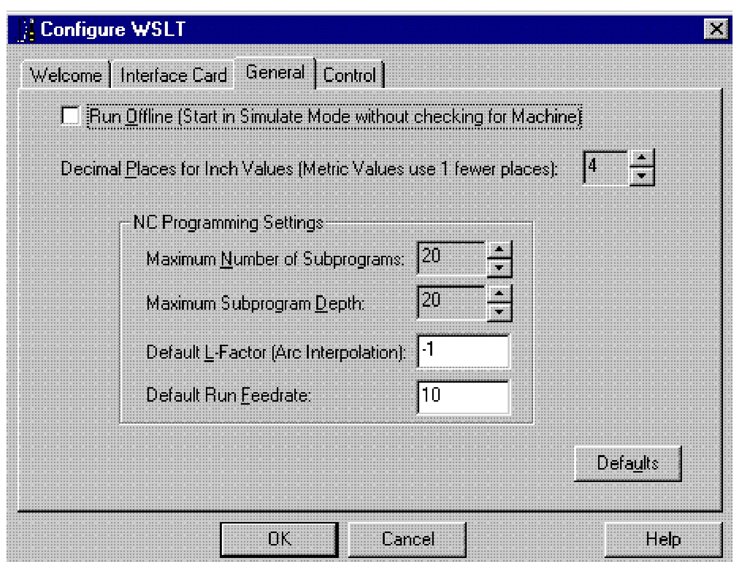
- Welcome
- Interface Card
- General

- Control

**Note:**

On all panels the Default button resets the Control Program to use factory set defaults for the values on the current panel. All values on the panels are based on the unit of measure selected on the Welcome Panel.



## Welcome Panel

This panel provides one option, **Units Default**. This option sets the default unit of measure for the Control Program and the Setup Program. When running the Control Software, you can change the default using the Units command under the Setup Menu.

## Interface Card Panel

This panel allows you to change the **Interface Card Address**. Once you have installed the Interface Card and software, and performed the initial Setup, this address should not change. The only time you might consider changing the address is if there is a hardware conflict.

## General Panel

This panel allows to you alter several software defaults.

Note:

Increasing the maximum number or depth of subprograms requires slightly more memory. Unless you use NC programs with complex subprograms, you should not need to change these values.

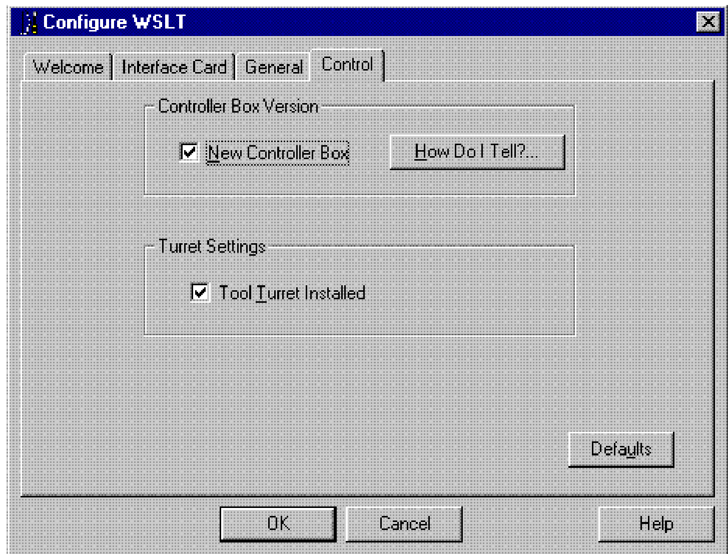
- **Run Offline** starts the Control Program in Simulate Mode without checking for a machine connection.

  Simulation Mode is provided to allow the Control Program to be used when your computer is not connected to a spectraLIGHT Turning Center. This is primarily to support the off-line development of NC programs.

  NC program verification and running is supported in Simulation Mode. Programs will Run in the same amount of time that they would if a machine were attached (excluding stops that have indefinite length of stop time, see Estimate Runtime).

  The principal difference between Simulation Mode and Normal Mode is, since no machine is available to send and receive data, there are no Inputs and Outputs. The Inputs and Outputs buttons on the toolbars are inoperable. If your program contains commands to wait for certain input values that are not the default, the events will never occur. If the values are the default, the wait will occur immediately.

- **Decimal Places for Inch Values** controls the display of values in dialog boxes. When in Metric mode, the software displays 1 less than the specified number of decimal places.
- **NC Programming Settings** controls several programming options:
  - The Maximum Number of Subprograms.
  - The Maximum Depth of Subprograms; the number of nested subprograms.




  - The Default L-Factor; the angle at which a line segment approximates a portion of an arc.
  - The Default Run Feed Rate; the initial feed rate when running or verifying an NC program; used until a feed rate is specified in the NC program.
- The **Default Button** resets all configurations in the General Panel to the default settings of the control software. The picture on the previous page shows the default settings.

# Control Panel

The **New Controller** Box option lets the controller know which version of Controller electronics you are using. The **How Do I Tell** button explains the difference between the controllers.

The **Turret Settings** Box option lets the control software know if an optional Tool turret is installed or not.

Reference Guide: Section F

# Basic CNC Programming

# The Elements of an NC Part Program

Part programs generally incorporate two types of instructions: those which define the tool path (such as X and Z axis coordinates), and those which specify machine operations (such as turning the spindle on or off). Each instruction is coded in a form the computer can understand.

An NC program is composed of *blocks* (lines) of code. The maximum number of blocks per program is limited by the memory (RAM) on your computer. You can, if necessary, chain programs together to form very large part programs.

Each block contains a string of *words*. An NC word is a code made up of an alphabetic character (called an *address character*) and a number (called a *parameter* or *value*). There are many categories of address characters used in NC part programs for the spectraLIGHT Turning Center (see Categories of NC Code).

Each block of NC code specifies the movement of the cutting tool on the Turning Center and a variety of conditions that support it. For example, a block of NC code might read:

```
N1G90G01X.5Z1.5F1
```

If the machine is currently set for inch units, the individual words in this block translate as:

**N1** This is the block sequence number for the program. Block 1 is the first block in the program.

**G90** This indicates absolute coordinates are used to define tool position.

**G01** This specifies linear interpolation.

**X.5** This specifies the X axis destination position as 0.5".

**Z1.5** This specifies the Z axis destination position as 1.5". The cutting tool will move to the absolute coordinate position (0.5,1.5).

**F1** This specifies a feed rate of 1 inch per minute, the speed at which the tool will advance to the specified coordinate points.

# Categories of NC Code

Many categories of NC codes can be used to program the spectraLIGHT Turning Center. Here is a list of the NC codes (designated by the address character) supported by the spectraLIGHT Turning Center.

| Code: | Function: |
|---|---|
| % | Incremental Arc Centers (Fanuc). |
| $ | Absolute Arc Centers. |
| \ | Skip. |
| / | Optional skip. |
| F | Feed rate in inches per minute; with G04, the number of seconds to dwell. |
| G | Preparatory codes. |
| H | Input selection number |
| I | Arc center, X axis dimension (circular interpolation). |
| K | Arc center, Z axis dimension (circular interpolation). |
| L | Loop counter; Program cycle (repeat) counter for blocks and subprograms, angle of arc resolution. |
| M | Miscellaneous codes. |
| N | Block number (user reference only). |
| O | Subprogram starting block number. |
| P | Subprogram reference number (with M98 or M99). |
| Q | Peck depth for pecking canned cycle. |
| R | Arc radius for circular interpolation (with G02 or G03); Starting reference point for peck drilling (with canned cycle codes). |
| S | Spindle speed. |
| T | Tool selection. |
| U | Incremental X motion dimension. |
| W | Incremental Z motion dimension. |
| X | X axis motion coordinate. |
| Z | Z axis motion coordinate. |
| ; | Comment. |

## Incremental Arc Center (% Code)

The incremental arc center code selects the Fanuc mode for programming arc coordinates. This mode is selected for the entire NC program as well as for any chained programs.

In the Fanuc mode, arc centers are always incremental, regardless of whether the system is in G90 (absolute) or G91 (incremental) mode. In contrast, arc center specifications in EIA-274 mode follow the selected programming mode, absolute or incremental.

You can specify the default arc center mode in the Run Settings dialog box in the Setup menu.

**This character must stand alone on the first line of the NC program in which it appears.**

## Absolute Arc Centers ($ Code)

The absolute arc center code selects the EIA-274 mode of programming arc coordinates. This mode is selected for the entire NC program as well as for any chained programs.

In the EIA-274 mode, the mode of programming arc centers follows the selected programming mode, absolute (G90) or incremental (G91). In contrast, arc center specifications in Fanuc mode1 are always incremental, regardless of whether the system is in absolute or incremental mode.

You can specify the default arc center mode in the Run Settings dialog box.

**This character must stand alone on the first line of the NC program in which it appears.**

## Skip (\ Code) and Optional Skip (/ Code)

Note:

The optional skip (/) code works only when the Optional Skip parameter from the Run Settings dialog box is on.

The Skip and Optional Skip codes allow you to skip particular lines of code in your program.

### To use the Skip code (\):

Place the code at the beginning of the line you wish to skip. When you run the NC program, the specified line will be skipped.

## To use Skip code (\) with a parameter:

Use the Skip code with a parameter to instruct the Control Program to execute the line of code every nth pass. Place the code at the beginning of the line you wish to skip. The syntax is: **\n**, where **n** is the number of passes between executions.

For example, if you want to execute a block of code every 5 passes, place **\5** as the first code at the beginning of the block.

## To use the Optional Skip code (/):

1. Place the code at the beginning of the line you wish to skip.
2.. Select the Optional Skip option from the Run Settings dialog box or the Operator Panel.

When you run the NC program, the specified line will be skipped. If you do not select the Optional Skip option in the Run Settings dialog box, the skip code is ignored and the line is executed normally.

## To use the Optional Skip code (/) with a parameter:

Use the Optional Skip code with a parameter to instruct the Control Program to execute the line of code every nth pass. Place the code at the beginning of the line you wish to optionally skip.

The syntax is: **/n**, where **n** is the number of passes between executions.

For example, if you want to execute a block of code every 5 passes, place **/5** as the first code at the beginning of the block.

# Feed Rate (F Code)

The F Code is used to specify the rate of speed at which the tool moves (the feed rate in inches or millimeters per minute, depending on the selected unit of measure). For example, F3 equals 3 ipm when in inch units or 3mmpm when in metric units.

The feed rate should be set to a low value (up to 30 ipm) for cutting operations. Feed rate values are in millimeters per minute (mpm) when using metric units. The Control Program limits the programmed feed rate so it doesn't exceed the maximum allowed by the Turning Center.

The F Code is also used to specify the number of seconds to dwell when used with the G04 code. For example, G04F5 causes the machine to dwell for five seconds.

# Preparatory Codes (G Codes)

G codes take effect before a motion is specified. They contain information such as the type of cut to be made, whether absolute or incremental dimensioning is being used, whether to pause for operator intervention, and so on.

More than one G code from different groups can appear in each NC block. However, you may not place more than one G code from the same group in the same block.

The G codes supported by the spectraLIGHT Turning Center Control Program fall into the following groups:

- The Interpolation Group
- The Units Group
- The Wait Group
- The Canned Cycle Group
- The Programming Mode Group
- The Preset Position Group

## The Interpolation Group

The Interpolation Group allows you to specify the type of motion for interpolation. These G codes are retained until superseded in the NC program by another code from the Interpolation Group.

The supported interpolation G codes are:

**G00** Rapid traverse
**G01** Linear interpolation (default)
**G02** Circular interpolation (clockwise)
**G03** Circular interpolation (counterclockwise)

Refer to Section G for more information on these functions.

## The Units Group

By default, an NC program is interpreted using the units of measure (inch or metric) specified using the Units command on the Setup Menu.

The codes in the Units Group, G70 (inch) and G71 (metric), are used to override the Units command for the entire program.

If the code is placed at the beginning of the program before any tool motions are made, that unit of measure is assumed for the entire program. Otherwise, it affects the rest of the program following the code. You can use these codes to switch between inch and metric modes throughout your program at your convenience.

The Fanuc equivalents, G20 (inch) and G21 (metric), can also be used.

The Units Group Codes are:

**G70** Inch Units
**G71** Metric Units
**G20** Fanuc inch units
**G21** Fanuc metric units

## The Wait Group

Wait Group codes apply only to the block in which they appear. The program does not continue until the wait conditions are satisfied.

The supported Wait Group codes are:

**IMPORTANT!**

Do not use G04 for tool changes in the middle of a program; use G05 for this purpose.

**G04** Dwell (wait): Pause between motions on all axes for the number of seconds specified by the F code, then continue the program. This is used primarily for robot integration. Because the F code is used to specify the number of seconds, you cannot also specify a new feed rate in the same block. Example: G04F10;Wait for 10 seconds

**G05** Pause: Used for operator intervention. Stop motion on all axes until the operator manually resumes program execution using the Run/ Continue command.

**G25** Wait until TTL input #1 (Robot 1 or user input 5) goes high before executing the operations on this block. Used for robot synchronization (see Section L for more information). Use the H code to specify an input other than the default, H5.

**G26** Wait until TTL input #1 (Robot 1 or user input 5) goes low before executing the operations on this block. Used for robot synchronization (see Section L for more information). Use the H code to specify an input other than the default, H5.

## The Canned Cycle Group

Canned cycle codes allow you to perform a number of tool motions by specifying just one code. Canned Cycle codes are typically used for repetitive operations to reduce the amount of data required in an NC program. Canned cycle codes are retained until superseded in the program by another canned cycle code.

The supported Canned Cycle codes are:

**G32** Canned cycle thread cutting
**G72** Canned cycle arc turning, clockwise
**G73** Canned cycle arc turning, counterclockwise
**G77** Canned cycle side turning
**G79** Canned cycle end turning
**G80** Canned cycle cancel
**G81** Canned cycle drilling
**G83** Canned cycle peck drilling

Refer to Section G for more information on these functions.

## The Programming Mode Group

Programming mode G codes select the programming mode, absolute (G90) or incremental (G91). These codes remain in effect until superseded by each other. The default code on program start up is G90.

With absolute programming, all X and Z coordinates are relative to origin of the current coordinate system. With incremental programming, each motion to a new coordinate is relative to the previous coordinate.

The supported Programming Mode codes are:

**G90** Absolute programming mode
**G91** Incremental programming mode

## The Preset Position Group

The preset position G codes move the tool to a predetermined position, or affect how future motions will be interpreted.

The supported Preset Position codes are:

**G28** Set reference point: This code moves the machine to its home position and sets the machine position to 0,0. The G28 code performs an automatic calibration of the axes.
**G29** Return to reference point: Moves the tool to a coordinate specified by XZ. Typically used after a G27 or G28 code.
**G92** Preset position: This code works like the Set Position command under the Setup Menu. The X and Z coordinates following a G92 code define the new current position of the tool.
**G98** Rapid move to initial tool position after canned cycle complete.
**G99** Rapid move to point R (surface of material or other reference point) after canned cycle complete.

## The Coordinate System Group

Use the Coordinate System codes to establish multiple coordinate systems on one or more workpieces to create multiple parts.

For instance, you can run a part program using a typical coordinate system (with the point of origin on the left side of the workpiece next to the chuck, along the centerline of the workpiece), then select another coordinate system which has its origin at a different point on the surface of the workpiece.

Multiple coordinate systems can be useful for different size workpieces, or for special set-up conditions

There are seven coordinate system codes. One of these codes (G53) is used to make a rapid traverse motion to specified machine coordinates. The other six codes allow you to make up to six individual parts on the same workpiece by specifying different work coordinate systems for each part.

The coordinate system codes are G54 through G59, referring to coordinate systems 1 through 6 respectively. These coordinate systems may be set through the Coordinate Systems command on the Setup Menu.

### The Polar Programming Group

The Polar Programming Group codes allow you to perform polar programming operations, based on polar coordinates. Refer to Section H for more information on using polar programming.

The supported Polar Programming codes are:

**G15** Polar programming OFF
**G16** Polar programming ON

### The Compensation Functions Group

Use the compensation NC codes to automatically compensate for scaling and rotation adjustments. Refer to Section H for more information on using scaling and rotation.

The supported Compensation codes are:

**G50** Cancels scaling.
**G51** Invokes scaling.
**G68** Invokes rotation.
**G69** Cancels rotation.

## Input Selection Number (H Code)

The H code is used to specify inputs and outputs in robot integration.

Use the H code in conjunction with:

- The wait codes G25 and G26, to specify the input number. If the H code is **not** used with these G codes, input 5 is assumed.
- The transmit codes M25 and M26 for interfacing with robots or other external devices, to specify the output number. If the H code is not used with these M codes, output 4 (Robot 1) is assumed.

## X Axis Coordinate of Center Point (I Code)

In absolute programming mode (G90), the **I** code specifies the X axis coordinate of the center point of an arc or circle when using circular interpolation. In incremental mode (G91), the **I** code specifies the X axis distance from the start point of motion to the center point of the arc for circular interpolation. If no I code is specified, the system uses the current X axis location as the X axis center of the arc.

In Fanuc mode, all arc centers are incremental.

The I code is also used with the G51 code to specify the scale factor for the X axis when performing scaling functions, including scaling each axis and mirror scaling. Refer to Section H for more information on using scaling.

## Z Axis Coordinate of Center Point (K Code)

In absolute programming mode (G90), the K code specifies the Z axis coordinate of the center point of an arc or a circle when using circular interpolation. In incremental mode (G91), the K code specifies the Z axis distance from the start point of motion to the center point of the arc for circular interpolation. If no K code is specified, the system uses the current Z axis location as the center of the arc.

In Fanuc mode, all arc centers are incremental.

The K code is also used with the G51 code to specify the scale factor for the Z axis when performing scaling functions, including scaling each axis and mirror scaling. Refer to Section H for more information on using scaling.

## Angle of Arc Resolution, Loop Counter (L Code)

The L code specifies the angle of arc resolution in circular interpolation programming. When the system executes a circular motion, it actually splits the arc into a series of line segments to approximate the circle. The L code specifies the angle in degrees which a line segment approximates a portion of the arc. The smaller the angle, the smoother the cut. A negative value for L will generate a *normalized* L factor (degrees x radius {in inches}) so larger radii have smaller degree values. For example, with the default L factor of -1:

- An arc with a radius of 1 inch will have line segments approximating every 1 degree of the arc.
- An arc with a radius of .5 inches will have line segments approximating every 2 degrees of the arc. (Degrees = -L/R, or Degrees = -(-1) / .5, which is 2)

**Note:**

The line segments generated by a normalized L code are always approximately the same length regardless of the arc's radius. (The length of the arc segment being represented by each line segment is exactly the same).

The default setting for the Turning Center is 2 degrees, and typically this will work quite well. You may notice the Turning Center hesitating on arcs if the resolution is too fine. The L code can be a fraction of a degree (such as L.5), but it must be large enough so the Turning Center will move at least the minimum motion (.0005") on each of the straight line motions.

Use the L code with:

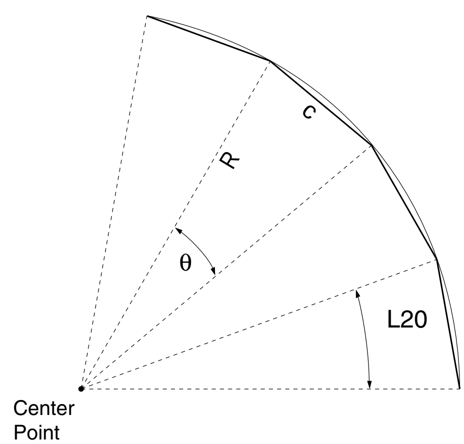
- The M98 code as a loop counter for subprograms.
- The M47 code as a program cycle counter, to repeat a program a finite number of times.
- The G27 code to specify tolerance with homing commands (this is an LMC-specific NC language extension). The difference between the current position and 0 is compared to a tolerance value specified using the Setup Program; use the L code to override this tolerance value.
- An L code that is not used by one of these codes is used as an Arc Resolution factor.

The equation to calculate the minimum arc angle is: $\theta = \frac{360c}{2pR}$

For example, to calculate the minimum arc value for a chord length of .001", and an arc with a radius of .5", you would write the equation as:

$$\theta = \frac{360(.001)}{2\pi.5} = .1146^{\circ}$$

Therefore, L=.1146




An Arc with an L Value of Twenty Degrees

# Miscellaneous Codes (M Codes)

Note:

All M codes used to turn on a device, such as the spindle, execute at the beginning of the tool motion for that block of NC code.

All M codes used to turn off a device execute after the tool motion for that block is completed.

To avoid confusion, it is sometimes easier to place M codes in a block separate from the motion commands.

M codes control a variety of Turning Center functions while a part program is running. Only one M code should be specified per NC block. M codes and motion commands should be placed on separate blocks to avoid confusion over whether an M code is activated during or after a motion command.

M codes can also be used to chain a second program to the end of a part program, or to repeat an NC program.

The supported M codes are:

**M00** Pause: Allows you to place a pause in your code. Acts like a G05 pause.

**M01** Optional Stop: Allows you to place an optional pause in your code. Place an M01 in the block of code where you would like to pause. There are switches to activate or deactivate the Optional Stop code in the Run Settings dialog box and on the Operator Panel.
With Optional Stop on, the M01 works like a G05 pause.
With Optional Stop off, the M01 code is ignored, and the other codes on the block are executed as usual.

**M02** End of Program: Takes effect after all motion has stopped; turns off drive motors, and all outputs, including the spindle and the accessory outlets.

**M03** Spindle Motor On: Activated concurrently with motion specified in the program block; remains in effect until superseded by M05.

**M05** Spindle Motor Off: Activated after the motion specified in the program block; remains in effect until superseded by M03.

**M06** Tool Change: Pauses all operations, turns off spindle, retracts the cross slide for tool change.

**M08** Accessory #1 On: Turns on ACC 1 accessory AC outlet concurrently with the motion specified in the program block; remains in effect until superseded by M09. This is the same as using the M25 H2 codes.

**M09** Accessory #1 Off: Turns off ACC 1 accessory AC outlet after the motion specified in the program block is completed; remains in effect until superseded by M08. This is the same as using the M26 H2 codes.

**M10** Accessory #2 On: Turns on ACC 2 accessory AC outlet concurrently with the motion specified in the program block; remains in effect until superseded by M11. This is the same as using the M25 H3 codes.

**M11** Accessory #2 Off: Turns off ACC 2 accessory AC outlet after the motion specified in the program block is completed; remains in effect until superseded by M10. This is the same as using the M26 H2 codes.

**M20** Chain to Next Program: This code is used to chain several NC files together. It appears at the end of a part program and is followed on the next line by the file name of another program which is executed when all motion stops. Here's an example of a part program chain to another program:

```
N37 Z.2
N38 M20
PROGRAM2.NC; Chain to PROGRAM TWO
```

If the two programs you are chaining are not in the same directory on your computer, you must specify the full path name for the next program file. If the software cannot locate the specified file, you will be prompted to find it.

**M22** Output current position or other information to file.

**M25** Set TTL output #1 (Robot 1 or Output 4) On: Used for robot synchronization (see Section L for more information). Use the H code to specify an output other than the default, H4.

**M26** Set TTL output #1 (Robot 1 or Output 4) Off: Used for robot synchronization (see Section L for more information). Use the H code to specify an output other than the default, H4.

**M30** End of program: Same as M02.

**M35** Set TTL output #2 (Robot 2, or Output 5) On: Used for robot synchronization (see Section L for more information).

**M36** Set TTL output #2 (Robot 2, or Output 5) Off: Used for robot synchronization (see Section L for more information).

**M38** Drive MotorsStandby: Typically used to turn the drive motors to standby (low power) before a pause (G05); activated after the motion specified in the program block. M38 is useful when the machine is left unattended for a long time, such as the interval before reloading a part.

**M47** Rewind: Restarts the currently running program; takes effect after all motion comes to a stop. Typically used with an L code to repeat a program a set number of times.

**M98** Call to subprogram. Use the P code to specify the subprogram starting block number. Use the L code to specify the number of times the subroutine is executed. You can nest subprogram calls to a depth of 20.

**M99** Return from Subprogram; Goto.

**M105** Operator Message (nonstandard Light Machines code).

**M111** Home the X axis.

**M112** Home the Z axis.

## M22: Output Current Position to File

Note:

Information about digitizing is provided with the digitizing package. For additional information, please call Light Machines Technical Support.

The M22 code is used to write information to a file while a program is running. Typically, this code is used when digitizing to write the current X and Z machine coordinates to a file. The proper format for using this code is: M22(*filename*) Data to Write to File.

The first time the Control Program encounters an M22 code, it opens the specified file. You must enclose the name of the file in parentheses for the Control Program to recognize it. If you do not specify any DatatoWritetoFile text, the default data is output. This default is the current position, equivalent to specifying 'X@X Z@Z'. The @X,@Z 'macros' are replaced by the actual machine position when the data is written.

Each M22 code automatically adds a linefeed to the end of its output so the next M22 starts on a new line.

If the file name is followed by ",A" (e.g., test.nc,A), the Control Program does not delete previous information from the file, it appends the information to the end of the existing information. If the file does not exist, it is created.

**Note:**

All text on the same block, after the closing parenthesis, is output to the file, with all valid macros being replaced as it is written.

If you use more than one M22, only the first occurrence must have the file name in the parentheses. The remaining M22's may have empty parentheses, ( ), or may specify a different file.

If you want to generate more than one file at a time, you must include the filename each time you specify M22. If a filename is not specified, the first file opened is used.

**Example:**

```
. . .          ; code to move to position
; Open my1.xz, discard contents, write coordinates
M22(my1.xz)
. . .          ; code to move to next position
; Append to currently open data file
M22( )
. . .          ; code to move to next position
;Open my2.xz and append coordinates
M22(my2.xz,A)
```

Special codes that can be used with M22 to generate run-time reports.

| | |
|---|---|
| @X | Current X position (in current coordinate system) |
| @Z | Current Z position (in current coordinate system) |
| ~ (tilde) | New line (starts a new line in the file) |
| @TD | Time of day (12hour): "11:59:59AM" |
| @TC | Time (elapsed) for cycle: "99:11:59" (0's trimmed from left) |
| @TT | Time total (of program run): "99:11:59" |
| @TA | Time Average (per cycle): "99:11:59" ("??:??" if first part) |
| @TL | Current Tool number: "5" |
| @C | Cycle number (current pass): "3" |
| @D | Date: "12/31/97" |
| @FN | Current file (without path): "PART.NC" ("UNTITLED.NC" if untitled) |
| \t | Tab |
| \\ | Outputs a single backslash ('\') character to the file |

**Example:**

```
; Start of file
… ; Process a single part
; Output part time statistics to file c:\Reports\Stats.txt (c:\Reports directory must exist)
M22 (c:\Reports\Stats.txt,A) Part #@C processed in @TC.
M47 L50 ; We want to process 50 parts.
```

## M99: Return from Subprogram, Goto

The M99 code has two specific uses; it can be used as a command to return from a subprogram or it can be used as a goto command.

### Using M99 with subprograms

When used in a subprogram, this code returns you to the block following the last M98 (Call to Subprogram) command. If the M98 used an L code to specify multiple calls to the subprogram, the M99 will return to the block containing the M98 until all the specified number of subprogram calls have been made; then it will proceed to the block following the M98.

You can use the P code plus a block number to override the block returned to; however, if this feature is used from a nested subprogram call, all return targets are discarded. The rules for a Goto target block apply to this use as well.

### Using M99 as a Goto command

This command can be used in the main NC program as a Goto command to jump to any line *before* the first subprogram (as denoted by the O code).

Use the P code to identify the block number being jumped to. Control is transferred to the first occurrence of this N code; it cannot be used to transfer control between chained programs (see M20).

This command can be used anywhere in the program to change the flow of program execution. It is good programming practice to place this command on a line by itself to improve the program's readability.

## M105: Operator Message

This command is used to display messages in the Control Program. It provides a way to display messages to the operator on the message bar while an NC program is running. You can also pause the program with a custom message if the first character is "!". This is a nonstandard, Light Machines code.

By default, the message is centered, displayed as a Normal Message, and is persistent (not cleared until the program clears it or until the next message is displayed).

### Using the Correct Format for this Code

```
M105(the message);comment
```

For instance, the following line of code displays a simple message:

```
M105(End of Roughing Segment);Normal Message; doesn't pause
```

### Messages Altered Using Alternate Characters

! Displays the message and performs a pause requiring operator intervention to continue.

~ Displays the message as a Warning Message.

\b Beeps when the message is shown.

### Using the M105 Code with an Alternate Character

```
M105(alternate character plus the message) ;comment
```

For example:

```
M105(~WARNING);Warning Message , doesn't pause
```

### Other Examples of Using this Code

```
M105( ) ; clears current message
M105(!Please stop and read this!) ; Normal Message, pauses
M105(~!I MEAN IT!) ; Warning Message, pauses
M105(\b\b\b) ; Clears current message, beeps 3 times, and doesn't
pause
```

## Block Number (N Code)

N codes have two uses:

- To provide destinations for Gotos (M99) elsewhere in the program.
- To clearly show the organization of the code and improve readability.

Using the N code is optional; however, when you do use the N code, it must be the first character in the block.

Other than for the above stated uses, N codes are ignored by the Control Program. Their presence, absence, or sequential value does not affect the execution of the NC program in any way (unless the target of a goto is missing).

You may have N codes on some blocks and not on others. N code sequence numbers do not have to be in order, but regular sequential order does make it easier to follow and reference sections of the program. The Control Program can change the N codes in a program by inserting, removing, or renumbering the codes.

## Subprogram Block Number (O Code)

The O code is used to indicate the start of a subprogram, and must be followed by a number which identifies the subprogram. The O code replaces the N code in the first block of the subprogram.

To call a subprogram, use the M98 code; use the P code to specify which subprogram to execute. To return from the subprogram, use the M99 code.

Only the first block in the subprogram contains the O code. The remaining blocks may contain N codes. The O and N code numbers may be used to help identify and set apart the subprogram to improve readability, for example:

```
M98 P50000 ;call to first subprogram
…;after first subprogram is finished, M99 code returns to this point
…
M98 P60000 ;call to second subprogram
…;after second subprogram is finished, M99 code returns to this point
…
O50000 ;start of subprogram
N50010 ;first line of subprogram
N50020 ;second line of subprogram
N50030 M99 ; last line of subprogram
…
O60000 ;second subprogram
N60010 ;first line of second subprogram
N60020 ;second line of second subprogram
N60030 M99 ; last line of second subprogram
```

## Subprogram Reference Number (P Code)

Use the P code with:

- The G31 code to reference a goto target block.
- The M98 code to reference a subprogram using the subprogram block number.
- The M99 code to specify a return block number as a goto target.

## Peck Depth (Q Code)

The Q code is used with the G83 code in canned cycle peck drilling to specify the incremental depth of each peck.

## Radius of Arc, Drilling Start Location (R Code)

As an alternative to specifying the center point of an arc (I Code or K Code) you can specify the arc radius. Use the same value for the radius in both absolute and incremental programming modes.

G02 or G03 specifies the direction of motion.

Positive values for R (radius) are specified for arcs up to 180°. Negative values are used for arcs greater than 180°. Full circle arcs cannot be performed with an R code. Split the circle into two arcs, or use center point (I and K) values for full 360° circles.

Use the R code in canned cycles to specify a Z axis reference point for peck drilling. The point can be at the material surface or at another reference point. The R code is also used to specify the rotation angle, in degrees, with the G68 code.

## Spindle Speed (S Code)

Use the S code to set the spindle speed from within the NC program. Spindle speed is specified by the address character "S" followed by a parameter that represents the speed in RPM. For example, S750 is the designation for a spindle speed of 750 RPM. For the spindle speed to have an effect the spindle must be turned on by the M03 command. If the spindle is off, the spindle speed is stored and used when the spindle is turned on again within the program. Use the M05 command to turn the spindle off.

CAUTION

Using multiple tools is an advanced operation, and should not be attempted by persons unfamiliar with using the spectraLIGHT Turning Center.

## Tool Selection (T Code)

T codes specify the tool (number) in the Tool Library in multiple tool machining operations. Tools are specified by the address character "T" followed by a parameter that represents the number of the tool. For example, T3 is the designation for tool number three.

Note:

Do not place absolute and incremental commands in the same block. For example: G90X1V1 will not produce the expected motions.

## X Axis Coordinate (X or U Code)

An X code specifies the coordinate of the destination along the X axis. A **U** code is used in absolute programming mode (G90) to specify an incremental X motion. You cannot use the U code to mix incremental and absolute programming in the same block.

## Z Axis Coordinate (Z or W Code)

The **Z** code specifies the coordinate of the destination along the Z axis (spindle axis). A **W** code is used in absolute programming mode (G90) to specify an incremental Z motion. You cannot use the W code to mix incremental and absolute programming in the same block.

## Comment Codes

The Control Program allows you to add comments (notes) to your NC code lines. The Control Program recognizes two comment codes, a semicolon ";" and an open parenthesis "(". These two comment codes are equivalent. The use of either of these codes within an NC block indicates that a comment follows.

Comments must follow all other NC codes in the block. Comments are ignored when the part program is executed. Comments can be placed on a block without any NC codes to document what is occurring within a program. NC programmers use these comments to annotate their programs.

Here is an example of an NC block with a comment:

```
X0Z0;MOVE TO ZERO POINT
```

The comment tells us that the X and Z codes in this block command the cutting tool to move to the zero point (coordinate 0,0).

Comments can be combined with the G05 pause and the M06 Tool Change codes to display messages to the operator during program execution. Here is an example of an NC block with a pause coded comment:

```
M05(SPINDLE OFF
G05(ROUGH DIAMETER SHOULD BE 0.5 in.)
```

When the program pauses, the comment is displayed on the message bar, telling the operator to verify the diameter of the workpiece before continuing. The M105 code provides a more versatile and powerful message facility.

The Control Program can strip comments from a program with a single command; however, the comments can not be subsequently replaced.

# General Programming Suggestions

The following rules should be followed when writing NC part programs.

1. The sequence of words (address characters plus parameters) in an NC block must appear in the following order: **%, $, \, /, N (O), G, X (U),Z (W), I,K, R, Q, L, F, S, T, M, P, ;** A different order may cause unpredictable results.

2. In many cases, a word need not be repeated in the next block (line). The system assumes no change in codes unless a new code appears. Some of the codes this does *not* apply to: N words, I and K, G04, G05, G25, G26, G92, F used for dwell, M02, M20, M25, M26, M30, M47, M98 or M99.

3. You can use more than one G code in a block; however, you can use only one G code from any one group in a single block.

4. N codes (sequence numbers) are not required in a part program; however, they can be useful in identifying a block when editing a long NC part program.

   An O code is required to mark the beginning of a subprogram and does not have to be in sequence with the N codes.

5. The first instruction in a part program should move the tool to the starting position. This makes restarts much easier.

6. The last block of a program should move the tool back to the starting position. The tool will then be in position to start cutting another part.

7. Part programs should reference the zero point with Z0 at the point where the tool just touches the work piece. This convention allows for standardization of programming.

8. Before running an NC part program:
   a. Look for the typical coding error that places two X codes or two Z codes in the same block.
   b. Be sure that all required coordinates have been written into appropriate blocks.
   c. Verify the part program to discover any program errors.
   d. Run the part program without mounting stock in the Turning Center to see if the tool movements are logical.

9. The first portion of a part program should turn on the spindle and establish the feed rate and spindle speed.

10. M codes should be placed on separate blocks to avoid confusion over whether an M code is activated during or after a motion command.

11. Double-check all program blocks against your coding sheet to locate and correct typographical errors.

Reference Guide: Section G

# More CNC Programming

Linear Interpolation Programming

Circular Interpolation Programming

Rapid Traverse Programming

Canned Cycle Programming

Spline Interpolation Programming

Subprogram Programming

# Linear Interpolation Programming

Linear interpolation is the movement of the tool in a line from its current position to a coordinate location specified by an NC code. Here's a typical line of NC code using linear interpolation:

```
N5G90G01X1Z.5F2
```

Broken down into individual words:

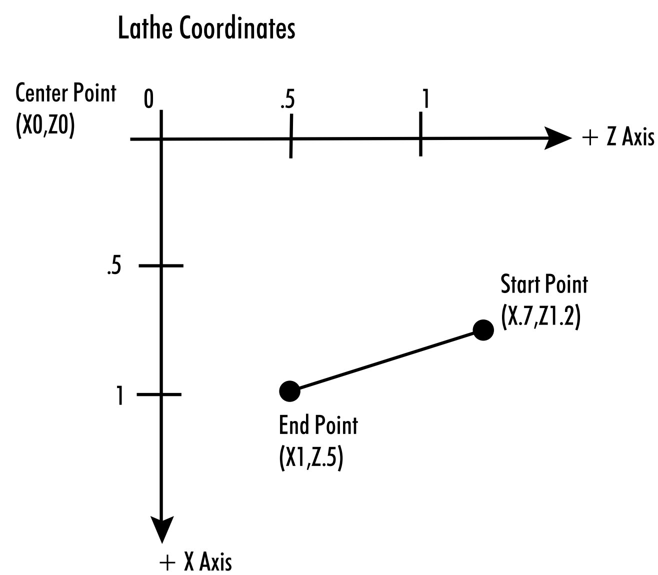
| | |
|---|---|
| **N5** | The line sequence number is 5 |
| **G90** | Coordinates are given using absolute dimensioning |
| **G01** | Linear interpolation is specified |
| **X1** | X axis coordinate of end point = 1 |
| **Z.5** | Z axis coordinate of end point = .5 |
| **F2** | Feed rate is 2 inches per minute |

The G01 code *is* required when switching from circular interpolation or rapid traverse positioning back to linear interpolation. If we assume the current position of the tool is X.7, Z1.2, the tool movement generated by the above line is something like this:

An equivalent movement is achieved with incremental dimensioning (G91):

```
N5G91G01X.3Z-.7F2
```

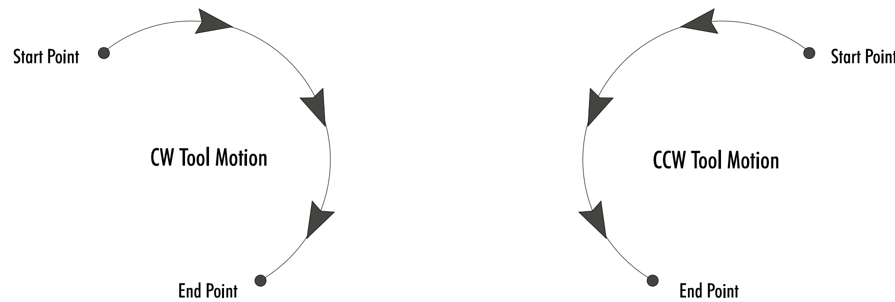
Typical tool movement using linear interpolation.



# Circular Interpolation Programming

Circular interpolation moves the cutting tool along an arc from the starting point specified in one line, to an end point specified in the next line. The curvature of motion is determined by the location of the center point (I or K), which must also be specified in the second NC line.

The direction of rotation from the starting point determines the actual shape of the arc relative to the spindle axis. A G02 code moves the tool in a clockwise (CW) motion from the starting point. A G03 code moves the tool in a counterclockwise (CCW) motion from the starting point.




G02 (CW) and G03 (CCW) cutting paths.

Here are two typical lines of NC code using circular interpolation:

```
N9G90X0Z1;SET START POINT
N10G02X1Z0I0K0F2;CLOCKWISE TO X1,Z0
```

The first line defines the starting point. The second line defines the end point and the center of the arc. Broken down into individual words, the second line reads:

**N10** The line sequence number is 10

**G02** The tool will proceed in a clockwise direction from the starting point to specified (X, Z) coordinates; center point of arc is specified by (I,K) coordinates

**X1** X axis coordinate of end point = 1

**Z0** Z axis coordinate of end point = 0

**I0** I coordinate of center point of arc = 0 (relative to start point)

**K0** K coordinate of center point of arc = 0 (relative to start point)

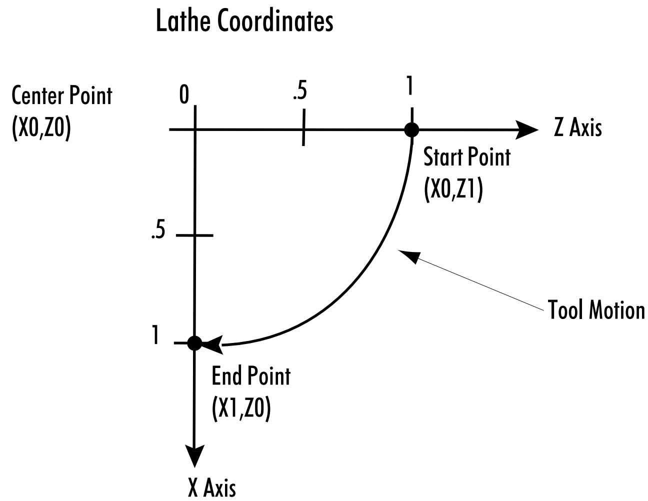
**F2** Feed rate is 2 inches per minute

### Note:

This program was written for Incremental Arc Center Mode. Either the % code was used at the beginning of the program, or Arc Center Incremental was selected in the Run Settings dialog box.

Assuming the start point is X0,Z1, the tool path generated by the preceding lines is something like this:

Typical tool movement using circular interpolation.




An equivalent movement is achieved with incremental dimensioning (G91):

```
N9G91X0Z1;SET START POINT
N10G02X1Z-1I0K-1F2;CLOCKWISE TO X1,Z0
```

In this NC line, the X and Z values are the distance the tool is to move from its current position. In both cases, the I and K values are equal to the X and Z distance from the start point to the center point.

## Rapid Traverse Programming

On the spectraLIGHT Turning Center, the rapid traverse code (G00) can move the tool at the maximum available feed rate (30 ipm) to specified coordinates. Rapid traverse is used to reposition the tool before ending a program, or in preparation for the next cut.

**WARNING**

**The tool should not be engaged in a cutting operation while traversing to a new location!**

Rapid traverse can be used for all tool positioning motions. This will reduce the run time for the part program. The G00 code remains in effect until linear (G01) or circular (G02, G03) interpolation is again specified. Linear or circular interpolation resumes at the feed rate last specified prior to the rapid traverse motion(s) unless you specify a new feed rate.

Rapid traverse is not affected by the feed rate scale factor. If all the positioning motions are done in rapid traverse (G00) and all the cutting motions are done in linear (G01) or circular interpolation (G02, G03), cutting rates can be adjusted up or down using the feed rate scale factor without affecting rapid traverse motions.

Here's a sequence of typical NC lines using rapid traverse:

**G90G01X1F2;** MOVE IN A STRAIGHT LINE TO X = 1 AT 2 IPM
**G00X2;** RAPID TRAVERSE TO X=2
**X3;** RAPID TRAVERSE TO X=3
**G01X4;** MOVE IN A STRAIGHT LINE TO X=4 AT 2 IPM

# Canned Cycle Programming

Canned cycle commands allow you to perform many operations by specifying just a few codes. They are typically used for repetitive operations to reduce the amount of data required in an NC program. Canned cycle codes are retained until superseded in the program by another canned cycle code.

The supported canned cycles codes are:

| | |
|---|---|
| **G32** | Canned cycle thread cutting (see Section H for more information) |
| **G72** | Canned cycle arc turning (clockwise) |
| **G73** | Canned cycle arc turning (counterclockwise) |
| **G77** | Canned cycle side turning (Fanuc G90) |
| **G79** | Canned cycle end turning (Fanuc G92) |
| **G80** | Canned cycle cancel |
| **G81** | Straight drilling |
| **G83** | Peck drilling |

These codes are used in conjunction with canned cycle codes:

| | |
|---|---|
| **G98** | Rapid to initial position after canned cycle complete; this is the system default. |
| **G99** | Rapid to point R after canned cycle complete. |

| | |
|---|---|
| **K Code** | Specifies the number of repeats. The default is 1. When K=0, drilling data is stored. |
| **P Code** | Specifies the length of dwell time in seconds. |
| **Q Code** | Specifies the depth of cut. In peck drilling each peck uses the same Q value. The Q value is always positive. If a negative value is specified, it is converted to a positive value. In side and end turning cycles the Q value is used to specify the depth of each roughing cut. |
| **R Code** | Used for specifying a starting reference point for peck drilling or for specifying tapers for side and end turning cycles. The point can be at the material surface or at another reference point. |

# Using G72 and G73

Note: A canned radius turning cycle can only be performed in one quadrant; it can not be performed across quadrants.

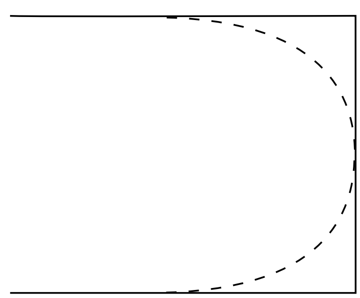
G72 can be used to cut an arc in one quadrant in a clockwise direction, for instance:

```
G0X0.001Z1;START POINT
G72X.5Z.5I0K.5Q.04
G80
```

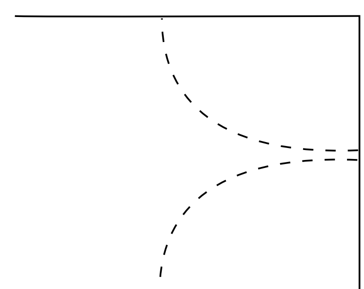
In the second block of code, G72 specifies a clockwise arc, uses I and K codes to specify the center point of the arc, and uses a Q for the depth of the cut. G73 works the same way, except it specifies a counterclockwise arc. These cuts are made using a facing orientation and a profiling tool.

The dotted lines represent the tool paths performed by typical G72 and G73 motions.

```
G0X0.001Z1
G72X.5Z.5I0K.5Q.04
G80
```

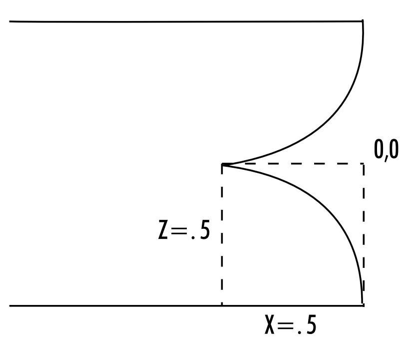


```
G0X0.001Z1
G73X.5Z.5I0K.5Q.004
G80
```



Here is an example of a G73 code used with a negative Q value.

```
G0X.5Z1
G73X0.001Z.5I.5K.5Q-.04
G80
```



## Using G77

The G77 code can be used to perform four functions: for straight side turning, for roughing cuts (using the Q code for depth of cut), for making tapers and for boring.

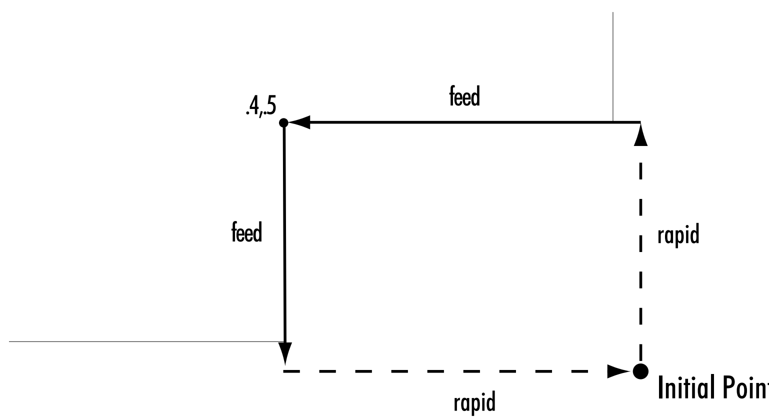
*Straight Side Turning*

Straight side turning is a simple operation with the G77 code. The tool makes a rapid move to the X depth, feeds to the Z coordinate, then backs out and returns to the start point. The following is an example of code using straight side turning.

```
G0X.6Z.8
G77X.4Z.5F10
G80
```

The tool movement for this example would be similar to that shown below.

A canned cycle using G77 for straight side turning.



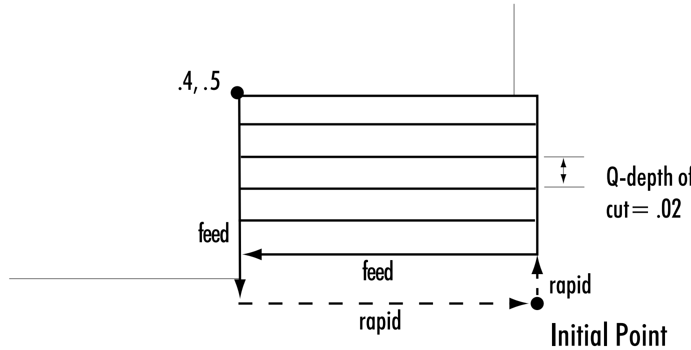

## Roughing Cuts

Performing roughing cuts with the G77 code works the same as straight side turning, except that a Q code is placed in the block to specify the depth of each cut. For instance, this block

```
G0X.6Z.8
G77X.25Z.5Q.02F10
G80
```

tells the tool to make 0.0200-inch roughing cuts until it reaches the specified X and Z coordinates.

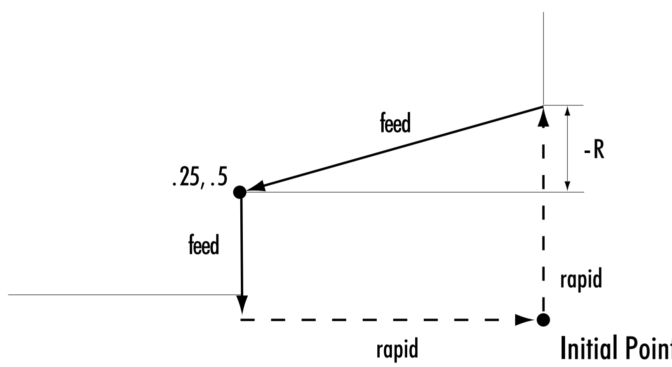
A canned cycle using G77 for roughing cuts.



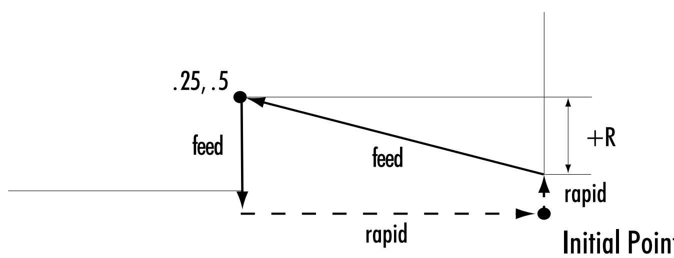

## Adding Tapers

Tapers are created by adding an R code. The following examples show a positive taper (using a negative R value) and a negative taper (using a positive R value).

The block G77X.25Z.5R-.1F10 produces Positive Taper - uses a negative R value.



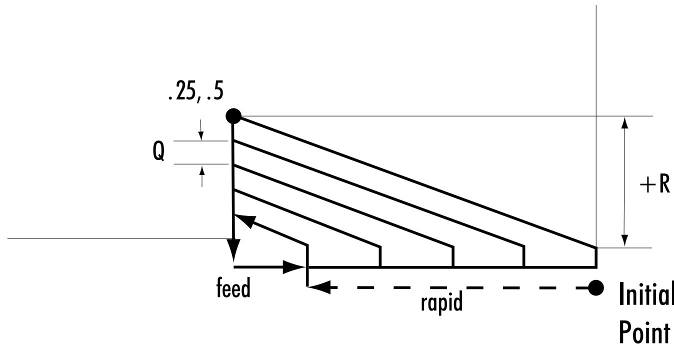

The block G77X.25Z.5R.1F10 produces Positive Taper - uses a positive R value.



This block

G77X.25Z.5Q.02R.1F10

produces a Positive Taper created with roughing cuts (Q code).

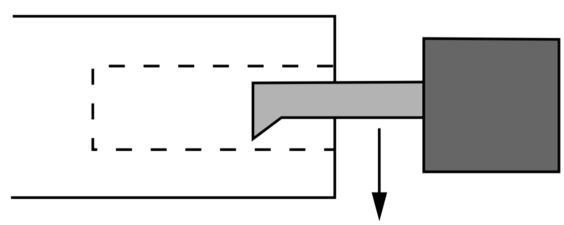


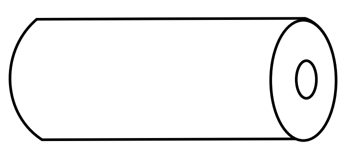

## Boring

Boring (inside turning cycle) inside is done by using a smaller X value at the beginning of the operation than at the end of the operation. The tool moves into the workpiece nears the center, then moves outward with each cut. An R value can be added to turn an inside taper. To perform a boring operation, the code would be something like:

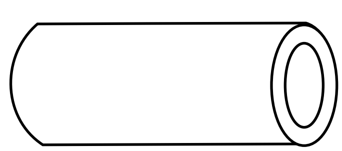
```
G0X0
G77X.375Z.5Q.03F10
G80
```

A canned cycle operation using G77 for boring.





Before Boring



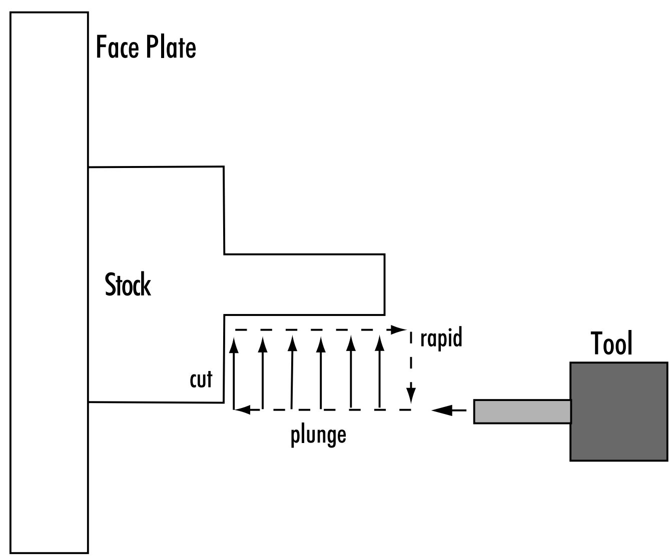
After Boring

# Using G79

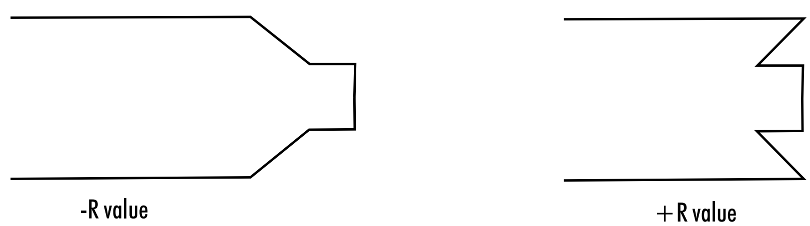
The G79 code can be used to perform an end turning cycle. The cuts are made on the face of the stock. G79 can be used with a Q code and R code just like G77. A typical use for G79 is shown in the following example.

```
G0;START POINT
G79X.2Z.8Q.03F10
G80
```

A canned cycle operation using G79.




Canned cycle cuts made using G79 with positive and negative R codes.




# Using G80

To cancel a canned cycle, use the G80 code. This code cancels the currently running canned cycle and resumes normal operation. All other turning data is canceled as well. You can also cancel canned cycles by using a G00 or G01 code; a G80 is automatically performed before the G00 or G01.

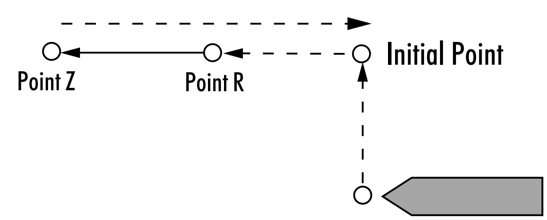
# Using G81 and G83

The G81 code performs straight drilling operations. G83 is used for peck drilling. By specifying an R value of zero, the tool will return to the initial point after drilling to point Z. The G98 code is the default for rapid movement to the initial point, or you could also use G99 to rapid to point R. We placed both rapid return codes in this sample to show how they should be placed in the program.

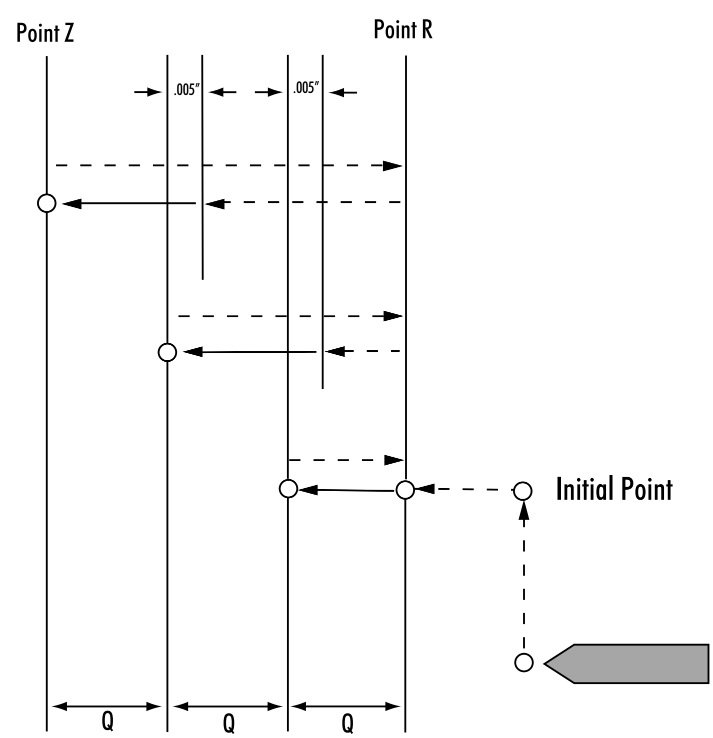
```
G0X0Z1;RAPID TO 0,1
G81Z.9R1F2;CENTER DRILL TO DEPTH OF Z.9 FROM Z1 FEED 2,
RAPID TO INITIAL POINT
G83Z.5R1Q.1F3;PECK DRILL TO Z.5 FROM Z1 EACH PECK .1,
RAPID TO POINT R
G80;CANCEL CANNED CYCLE
M2;END PROGRAM
```

This program will generate tool motions similar to this:

A canned drilling cycle using G81 and G98. The tool is returned to the initial point at the end of the canned cycle.




A canned pecking cycle using G83 and G99. The tool rapids to point R instead of the Initial Point at the end of the canned cycle.



# Spline Interpolation Programming

Note:

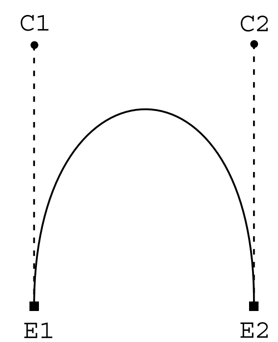
AutoCAD and AutoSketch curves are B-splines and are programmed with G02 and G03 NC codes.

In addition to Linear and Circular interpolation, the Turning Center supports geometric curve definition using Spline interpolation. This feature is included for advanced users who wish to use a CAD software package to generate drawing information (end point and control point information) for use on the spectraLIGHT Turning Center.

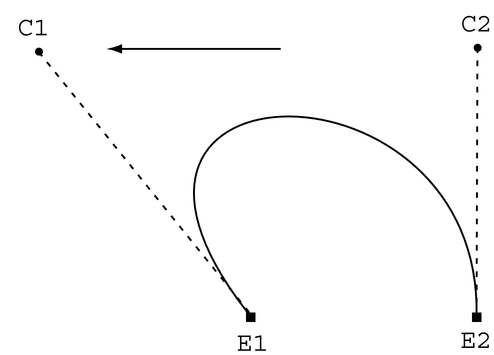
Whereas circular interpolation defines arcs as portions of a circle, spline interpolation defines curves based on non-colinear points. These more flexible curves are called Bezier curves. Unlike arc coordinates defined by a start point and an end point relative to a center point, coordinates on Bezier curves are defined relative to two *end points* and two *control points*.

### What is a Bezier Curve?

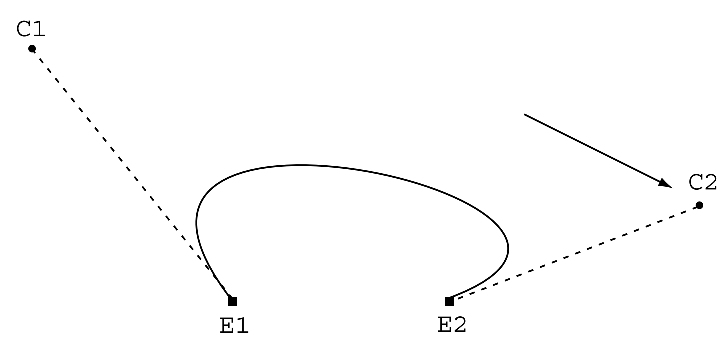
A Bezier curve is a single line drawn from end point 1 (E1) to end point 2 (E2). The curve follows tangent lines (shown dotted) that run from each end point to its control point.




When control point 1 (C1) is moved, the path between C1 and E1 remains tangent, pulling the curve out from its original position.




When control point 2 (C2) is moved, the curve is pulled along the tangent line that runs between C2 and E2.



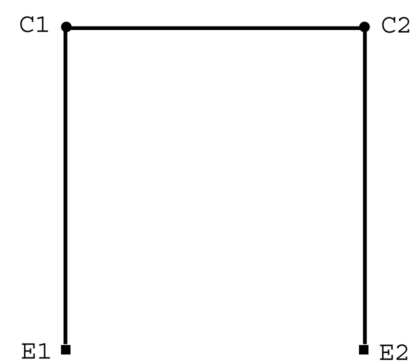
## Bezier Curve Characteristics

The end points of a Bezier curve are *anchored* and do not move, so the shape of a Bezier curve is dependent on the position of the control points. The control points can change the shape of the curve in any number of ways, allowing for greater flexibility in curve definition.

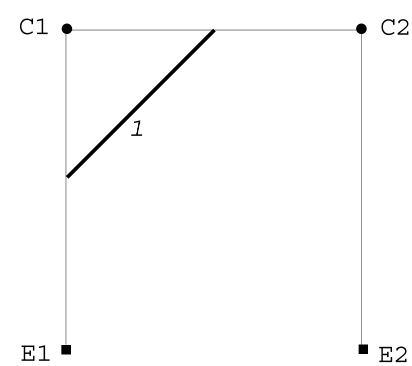
## Geometrically Defining a Bezier Curve

Normally, Bezier curves used in machining operations are generated by a CAD software package; they are not performed manually. All you need to know is where the start, end and control points are. Once you plot the coordinates for these points, the computer can perform the mathematical functions necessary to generate the Bezier curve. The following is a simple method for producing a Bezier curve by construction.

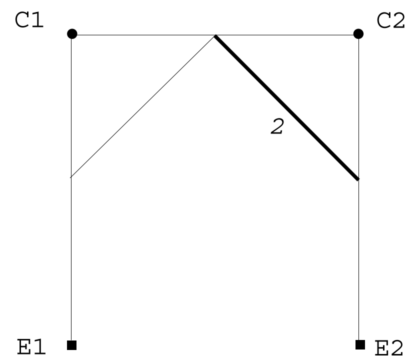
First, the start, end and control points are established and lines are drawn between them. The lines are drawn from E1 to C1, from C1 to C2, and from E2 to C2.



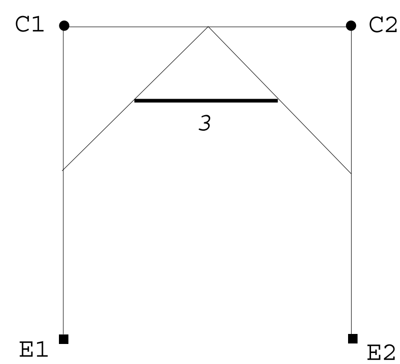

Next, a line (line 1) is drawn from the midpoint of the E1/C1 line to the midpoint of the C1/C2 line.



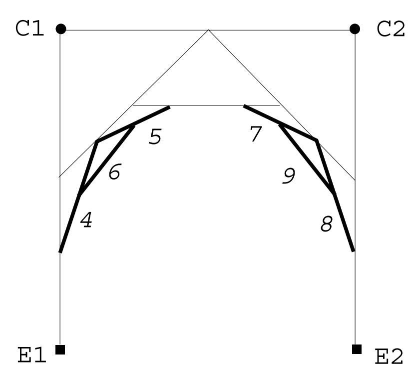
A line (line 2) is drawn from the midpoint of the C1/C2 line to the midpoint of the C2/E2 line.



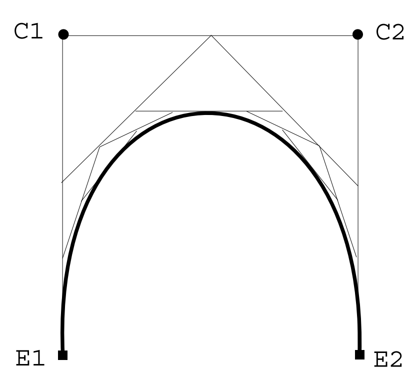

A line (line 3) is drawn from the midpoint of line 1 to the midpoint of line 2.




The above procedure is repeated using lines 1, 2 and 3 to produce lines 4 through 9.




This process is repeated until a Bezier curve is formed.



# Programming with Spline Interpolation

You can assign coordinates to the start, end and control points of a curve and enter this information as part of your NC part program. Here is a block of NC code using spline interpolation:

```
N24G0X.5Z3;START POINT
N25G101X2Z.5I1.5K3A2C1.5
N26G1Z0
```

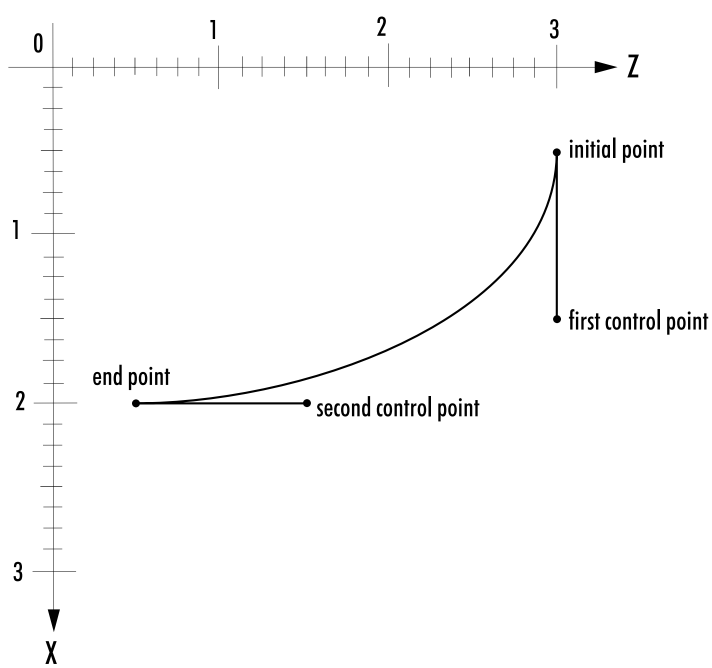
Block N24 moves the tool to the start point.

The second NC block (N25) can be analyzed as:

| | |
|---|---|
| **N25** | Block sequence number = 25 |
| **G101** | The preparatory code for spline interpolation |
| **X2** | X-axis coordinate of the end point |
| **Z.5** | Z-axis coordinate of the end point |
| **I 1.5** | X-axis coordinate of the first control point |
| **K 3** | Z-axis coordinate of the first control point |
| **A2** | X-axis coordinate of the second control point |
| **C1.5** | Z-axis coordinate of the second control point |

If the start point is X.5, Z3, the curve generated will be similar to the one shown below.

A Bezier Curve with Assigned Coordinates.



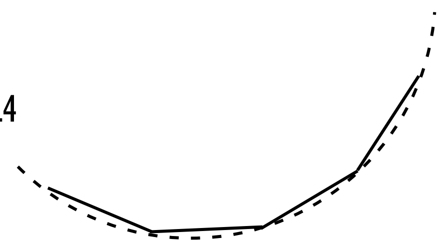
## Using the L Code in Spline Interpolation

Previously, when you used circular interpolation, you defined segments of an arc in degrees using the L code. In spline interpolation the L code can be used to define the number of segments required to complete a curve like the one shown on the previous page. The more segments there are in the curve, the smoother the curve approximation by the cutter.

Values for L that are too large or too small can create problems. Large values for L will bog-down the computer and cause the tool motion to slow down.

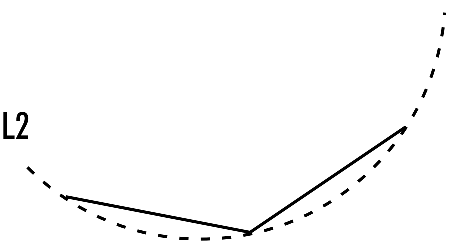
Small L values will cause the curve to be very rough.

L4



Large L values will bog down the the computer and cause extremely slow tool motion.

L2



# Subprogram Programming

Subprograms are used to execute repetitive routines in an NC program. Since a subprogram can be called again and again, you don't have to enter the same data more than once. This is especially useful if the machining operation you wish to repeat is lengthy or complex. The NC codes used for subprogramming on the spectraLIGHT Turning Center are:

**M98** Call to subprogram.

**M99** Return from subprogram.

**P Code** The P code is used to reference the first line of the subprogram (which begins with an O code). The P code immediately follows an M98.

**L Code** The L code is used as a loop counter when used in subprogramming. The computer executes the subprogram as many times as defined by the L code. For instance, if the code is L5, the subprogram is executed five times. (Optional)

**O Code** The O code replaces the N code on the first line of a subprogram.

**Note:**

The L code is also used as a program cycle counter. For instance, if the last line of NC code in your program is M47L10, the program executes ten times. After the tenth time, the computer ignores the rewind and runs the remainder of the program.

A subprogram is *called* by an M98 and a P code. When an M98 calls the subprogram, the main program is interrupted while the subprogram is executed.

The P code references the subprogram's address (the first line of the subprogram). The first line of the subprogram uses an O code instead of an N code for line numbering.

**Note:**

You can also perform an M99P "line number" at the end of the subprogram. This returns to the main program at the specified line. It's like a return with a go to.

When the M99 is executed, the main portion of the NC program continues to execute from the line after the subprogram was called.

Subprogram calls can also be *nested* within other subprogram calls. This means that while a subprogram is being executed, it can call another subprogram. The default number of levels that subprograms can be nested is 20 levels deep. You can change the default by using the Setup Program (click on the Setup icon in the WSLT program group).

# A Sample NC Subprogram

This subprogram makes a series of light chamfered grooves. It's set up for stock with a 0.75" diameter and 2" length so you should use a piece slightly longer 0.75" X 2.5".

Note that the file uses absolute programming in the main program and incremental programming in the subprogram. This makes it necessary to use the G90 after the subprogram is executed to allow motion back to the start point.

This block calls the subprogram (M98) that begins on block O1000 (referenced by P1000) and tells it to repeat five times (L4). The M99 cancels the subprogram.

```
;THIS FILE FOR 2.5 INCH BY 0.75 INCH STOCK MOUNTED IN CHUCK
;USE WITH A PROFILING TOOL NORMAL SIDE ORIENTATION
;SET THE START POINT AT Z2 X0.375
G0G90M03;ABSOLUTE PROGRAMMING
G0X0.380Z2
M98P1000L4;CALLS SUBPROGRAM 1000 AND EXECUTES IT 4 TIMES
G90;ABSOLUTE PROGRAMMING
G0X0.38
G0Z2
M02;END OF PROGRAM
O1000;START OF SUBPROGRAM
G91;INCREMENTAL PROGRAMMING SELECTED
G1X-0.040Z-0.040F3
G1Z-0.125
G1X0.040Z-0.040
G0Z-0.20
M99;END OF SUBPROGRAM
```

Reference Guide: Section H

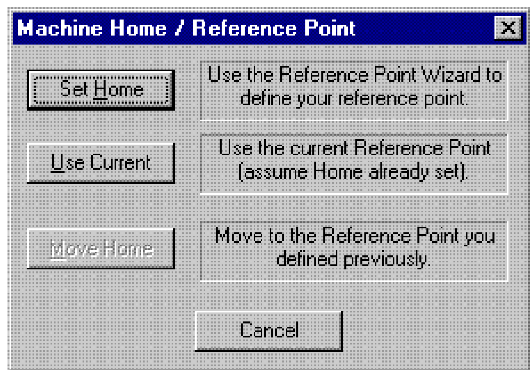
# Optional Machining Capabilities

# Using the Homing Commands

The Set/Check Homing command under the Setup Menu allows you to establish a point of origin at the ends of travel on the Turning Center. The homing commands (G27, G28, G29) allow you to return to and check this established position. The Turning Center uses this point as a reference for all machine coordinate movements. This allows you to use the Soft Limits and Coordinate Systems commands (under the Setup Menu) to move the Turning Center consistently to the same location.

Before you can use any homing commands, or the Soft Limits and Coordinate Systems commands, you must use the Set/Check Home command to establish an initial reference point. See Section E for information on using the Set/Check Home command.

G28 sets a machine reference point, similar to the Set/Check Home command from the Setup Menu.



# Using G28

The G28 code moves the machine to the coordinates previously established by the Reference Point Wizard as the home position, and then sets the machine position to 0,0. If a reference point has not been defined, a warning is generated and the G28 command is skipped. The G28 code performs an automatic calibration of the axes.

## Using G28 in an NC Program

The G28 code homes the machine and sets the Machine Coordinates to zero. Use a G27 if you want to check the home position, but do not want to set that position to zero.

The G54 calls up coordinate system one, which contains the offset values relative to the machine's home position. These would be the values which were entered for coordinate system one in the Offset from Machine Zero dialog box under the Set Coordinates dialog box. The coordinates in the Position Window on the screen change to the coordinates of "coordinate system one."

The next line (G0X0Z0) would call for the machine to perform a rapid traverse motion from the Home position to point 0,0 in "coordinate system one."

## Using G28 Before Setting Soft Limits

Remember, Soft Limits are based on Machine Coordinates. You can not use Soft Limits until you have set the machine to the reference point using the Set/Check Home command under the Setup Menu.

# Using G29

The G29 code moves the tool at a rapid traverse rate to a coordinate specified by XZ. If you have set an intermediate point on one or more axes, the machine first rapids from the current position to the intermediate point then continues to the specified destination. If you command a G29 code in Incremental mode, your specified XZ point is relative to the intermediate point. If you have not specified an intermediate point, your specified XZ point is relative to the current position. Use the G29 code after a G28 code to return the tool to a position closer to the part. The example below shows the use of a G28 code and a G29 code.

```
N1G28X2Z-1; INTERMEDIATE POINT THEN HOME
N2G29X4Z1; GO TO G29 POINT
```

## Using Polar Programming

Using polar coordinates allows you to specify a radius and an angle by specifying a G16 code (polar programming on), then X and Z codes. The X code specifies the radius. The Z code specifies the angle in degrees. A G15 is used to cancel the polar programming mode. This programming method can be used in both absolute and incremental programming.

Polar programming is especially useful when writing programs for machining bolt holes. An L code can be used as a multiplier for the angle value. For instance, this bit of code:

```
G0X0Z0.07
M03
G16
G91X2Z0
M98P1L12
M2
O1Z30
G81Z-.1R0
G80
M99
```

combines the use of polar programming with a canned cycle and a subprogram to drill a hole at 30° increments. The L value was determined by dividing 360 degrees by 30.

You can also cut an arc using code similar to:

```
G16
G91X2Z0
M98P1L360
G15
M2
O1Z1
M99
```

In polar programming, the center point is the origin if you specify G90 and the radius (X code). If you specify G91 and the radius (X code), the center point is the current point. If you specify only the angle (Z code), the center point is the current center.

# Using Scaling and Rotation Codes

Scaling and rotating codes can be used separately or they can be combined. Each of these functions is described in the following paragraphs.

## Scaling

Use the scaling codes to scale one or more axes of a part from a fixed scaling origin. You can scale the entire piece uniformly, or set different scaling factors for each axis. Use the following codes for scaling:

**G50** Cancels scaling
**G51** Invokes scaling

Given the information that in G51 X and Z are the origin of the scale, I and J are the scale factors. I = X and J = Z.

Here is an example of an NC program line using Scaling. In this example, X,Z is set at zero and is the origin of scale, and both axes are scaled by two.

G51X0Z0I2.J2.

## Rotation Codes

Rotation codes allow you to rotate a programmed shape around a rotation origin. Lathe programs are rarely rotated, however, it funtions in the same manner as scaling.

**G68** Invokes rotation
**G69** Cancels rotation

Here is an example of an NC program line using Rotation. In this example, X,Z is the rotation center, and 90 is the rotation value in degrees.

G68X0Z0R90.

# Multiple Tool Programming

Note:

There is a greater risk of tool crashes when there are more tools to crash, and there is more room for errors in larger NC programs.

The spectraLIGHT Turning Center allows you to designate up to 8 different tool offsets for tool changes during turning operations. Using multiple tool programs provides the power-user with a great deal of flexibility and freedom from tool changes. It also allows the user to create more complex parts on the Turning Center. To designate the tool number, tool diameter and tool offset, select Setup Library from the Tools Menu.

Multiple tool programing can be set to either wait for a manual change of the tool, or can be set to move to another tool which is already mounted on a separate tool post, based on offset values. If a tool turret is installed, the machine will automatically change tools.

There are four basic steps in setting up the Turning Center for multiple tool operation:

1. Placing the appropriate codes in your NC program,
2. Establishing a reference tool,
3. Establishing the offsets for other tools from that reference, and
4. Testing your program.

## Using Multiple Tool Codes

The T code is used in the NC program to offset the cutter so that the NC program becomes independent of the cutter length, which is set up in Tool Definitions. This means you can replace a worn tool with a tool of a different length without changing the NC program, just by entering new offsets. The T code can be located anywhere within the block of NC code, but it is normally placed after any G codes.

When you place T codes in your program for tool changes, you should also use the M06 code to shut off the spindle and to retract the cross slide. The M06 code is placed after the T code. This code instructs the Turning Center to shut off the spindle, while it pauses until you manually change the tool. Pressing the Return key will turn the spindle back on to continue with the NC program.

The tool that is shown in the Machine Info window is the default tool for the start of a program. The Control Program uses this tool unless another tool is specified with a T code. Once a tool is specified, any X and Z coordinates are applied to that tool until another T code is encountered in the NC program.

## Back Side Cutting

All of the X coordinates for a tool cutting on the back side of a workpiece should be negative unless the tool crosses the centerline into a positive quadrant, in which case the value of the point would be positive. When switching to a tool that cuts on the back side of a workpiece within an NC program which is primarily frontside orientation, make sure that your program reflects the change from a positive quadrant to a negative quadrant.

For example, if T2 is a cut-off tool on the back side of a 3" long, 0.375" diameter workpiece, the codes

```
T2Z1
X1
```

would tell the tool to cut right through the middle of the workpiece and stop at 1 inch on the other side of the centerline in the positive quadrant, which is beyond the necessary cut. For a more precise cut, the codes that should have been used are:

```
T2Z1   or   T2Z1
X.1         X0
```

# Establishing the Reference Tool

When using multiple tools for manual changing, a reference tool, normally Tool #1, is set to zero for the Z axis. This establishes a reference tool position which is used as a reference point for additional tools. For demonstration purposes, we will use Tool #1 as the reference tool and Tool #2 as the additional tool.

To set the reference tool:

1. Decide on a reference point (a point on the workpiece, or on a gauge, where you will jog the tip of each tool).
2. Open the Jog Control Panel (select Jog Control from the View Menu). With the tool installed in the spindle, jog Tool #1 to the reference point. The tip of the tool should barely touch the workpiece, or zero-out a reference gauge.
3. Select **Set Position** from the Setup Menu. Set the current position of Tool #1 to zero on the Z and X axes.

Tool #1 is now established as the reference tool.

## CAUTION

Before any tool change, be sure to jog the current tool away from the workpiece to avoid a crash in the program and damage to your workpiece.

# Establishing Tool Offsets

Now that the reference tool is established, additional tools can be assigned offsets. You can move the tool and accept its current Z and X axes position as the offset value, or you can manually enter offset values.

To set the offset for Tool #2:

1. The reference tool must first be set to the reference point as previously described.
2. With tool #2 selected, jog the tool turret to touch the surface of the workpiece, or offset sensor, at the previously determined reference point. In the **Position Window**, the current Z and X coordinates will be displayed. These are your offset values from the reference tool.
3. To enter these values as the offset values, select **Setup Library** from the Tools Menu and select the **Use Current Postion** bar. This will automatically enter the ofset values for you. Alternatively, you may manually enter the offset values for the Z and X axes by copying the values from the **Postion Window**.

The offset for Tool #2 is now established.

# Testing Your Multiple Tool Program

After setting all of the tool offsets, test run your program without a workpiece mounted.

1. After installing Tool #1, close the safety shield, put on your safety glasses, and complete the safety checklist.
2. Select the Run/Continue command from the Program Menu. Select line one as the start line and select Start. Throughout the test, be prepared to press the emergency stop button on the Turning Center in case of a tool crash. The computer will run the program until it reaches the M06code. The M06 stops the Turning Center.
3. When the spindle has completely stopped and the Pause message appears on the screen, push in the emergency stop button on the Turning Center.
4. Open the safety shield.
5. Remove Tool #1 and install Tool #2, making certain it is securely fastened to the tool post.
6. Close the shield, and pull out the emergency stop button. Press the Return key on the computer keyboard. The spindle turns on and the cross slide moves to the previous position. Operation continues as programmed until the next M06 code is encountered.

7. At each pause, repeat Steps 3 through 6, installing the appropriate tool at the appropriate points in the program.

# Running a Sample Multiple Tool NC Program

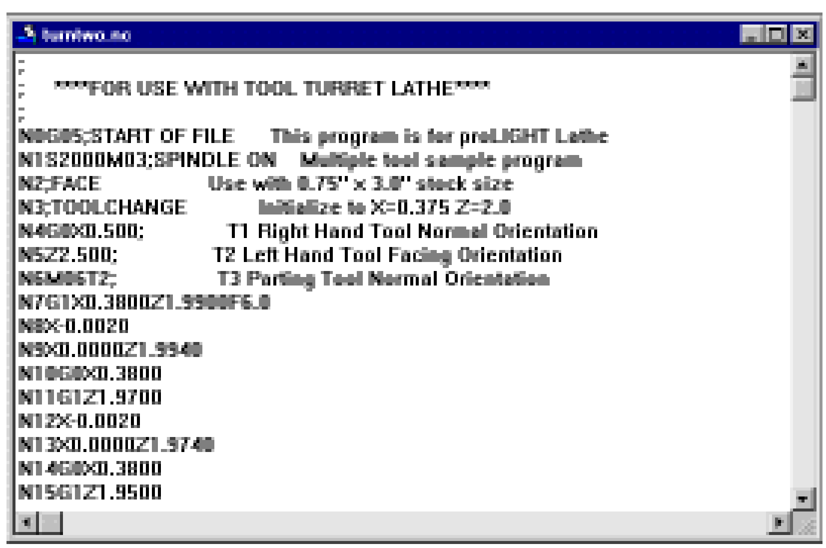
When you installed the spectraLIGHT Control Program an NC part program file, named turntwo.nc, was copied into the WSLT directory along with the other files. The turntwo.nc program is meant to turn a 3" (length) x 0.75" (diameter) cylindrical piece of machinable brass, aluminum, Delrin or wax. You will be using this file to create your a workpiece on the Turning Center using multiple tool programing.

**WARNING**

**Do not attempt to operate the spectraLIGHT Turning Center without reviewing all of the safety precautions set forth in the Reference Guide: Section J.**

## Open Turntwo.nc

1. Select the **Open** command from the File Menu, or click on the Open button on the Standard Tool Bar. The Open dialog box appears.
2. Double-click on the **turntwo.nc** file name, or click on the file name then click on the **Open** button. The edit window for turntwo.nc appears.



## Define the Tools

To turn this part, you will use three tools. To define the tool parameters, first select **Setup Library** from the Tools menu.

1. Select Tool 01, a Right-Hand tool, for Tool #1. The specifications are as follows:

| | |
|---|---|
| Description: | Left Hand Diamond |
| Tool Type: | Diamond |
| Station: | 1 (May be different with tool turret installed) |
| Material Type: | High Speed Steel |
| Radius: | 0.0001 |
| Angle: | 30 |
| Orientation: | Outside |
| Reference: | Tool Radius Center |
| Cut Direction: | Left/Down |

2. Select Tool 02, a Left-Hand tool, for Tool #2. The specifications are as follows:

| | |
|---|---|
| Description: | Facing Grooving |
| Tool Type: | Grooving |
| Station: | 2 (May be different with tool turret installed) |
| Material Type: | High Speed Steel |
| Radius: | 0.0001 |
| Width: | 0.01 |
| Orientation: | Facing |
| Reference: | Tool Radius Center |
| Cut Direction: | Right/Up |

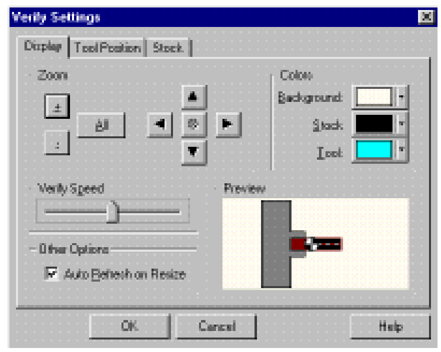
3. Select Tool 03, a Parting Tool, for Tool #3. The specifications are as follows:

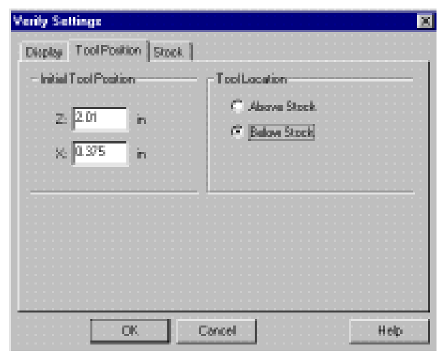
| | |
|---|---|
| Description: | Backside Grooving |
| Tool Type: | Grooving |
| Station: | 3 (May be different with tool turret installed) |
| Material Type: | High Speed Steel |
| Radius: | 0.0001 |
| Width: | 0.0313 |
| Orientation: | Outside |
| Reference: | Tool Radius Center |
| Cut Direction: | Left/Down |

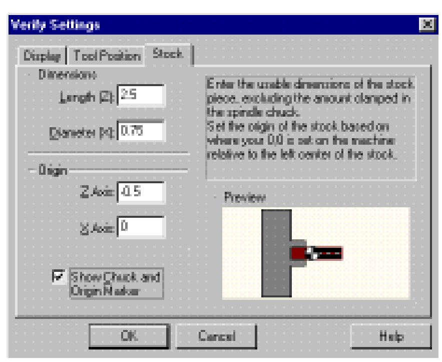
## Adjust the Verify Settings

After opening the NC program, you need to adjust the Verify Settings for the part you are about to turn, as you did in the turntwo.nc program. To view the Verify Settings dialog box, select Verify Window from the View Menu; double click on the Verify window when it appears.

You may also select Verify from the Program Menu, or from the Standard Toolbar, then click on the Verify Settings button. The Verify Settings dialog box appears. Adjust the verify settings for the Display, Tool Position and Stock. The Initial Tool Position is Z=2.01, X=0.375. The Stock Dimensions are Z2.5, X0.75. The Origin is Z0.5, X0.



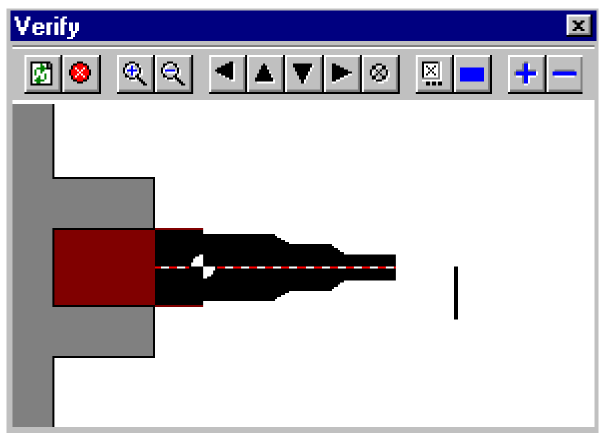



## Verify turntwo.nc

Tool path verification allows you to check for programming errors before actually running the part program on the Turning Center.

Since we supplied this program, you shouldn't encounter any errors during verification. You should, however, make a habit of verifying your programs more than once to ensure there are no errors before running them on the Turning Center.

1. Select Verify from the Program Menu or from the Standard Tool Bar. The Verify Program dialog box appears. The default starting line for the program is Line 1. When verifying a program for the first time, you should begin on Line 1.
2. Click on the Verify Program button, then watch the Verify Window. You will see the turntwo.nc program executed on the cylindrical workpiece.



# Hardware Set Up for Turntwo.nc

CAUTION

When programming with multiple tools, be aware that any tool that is not currently selected may run into the headstock or workpiece.

Check the program first with the spindle turned off and with the workpiece removed, using the single-step mode to make sure no collisions occur.

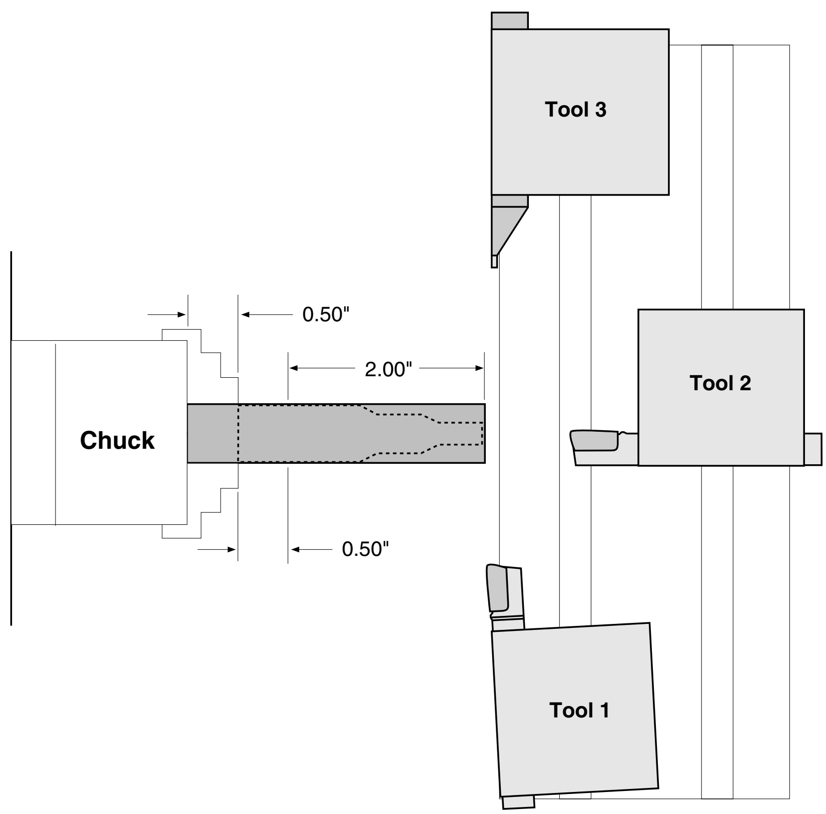
Now that you've got the NC program ready to run, you are ready to get the workpiece and stock set up.

## Mount the Tools and Workpiece

You should have a Right-Hand tool (tool 1), a Left-Hand tool (tool 2) and a Parting tool (tool 3) mounted in posts as shown in the following illustration. You should also mount a 3"x.75" workpiece in the chuck with approximately .5-inch of the workpiece inside the chuck.

Note: If you are using the optional Automatic Tool Turret, mount the required tools in accordance with the instructions in Section A of this guide.

Select one of the tools on the turret as the reference tool and offset the other tools from that reference, just as you would if the tools were mounted on the cross slide.

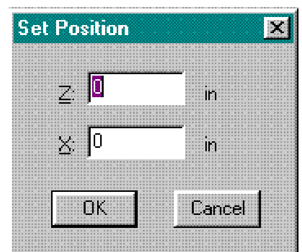



## Set the Reference Tool

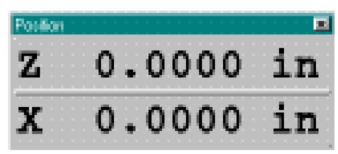
When using multiple tools, a reference tool, normally Tool #1 is set to zero for both axes. This established position is then used as a reference point for the other tools.

Begin with the Emergency Stop button pressed in, and the spindle speed turned all the way down. The tool should be mounted in the tool post.

1. Mount the workpiece in the chuck.
2. Close the Safety Shield and pull out the Emergency Stop button.
3. Select Jog Control from the View Menu (or the Standard Toolbar). The Jog Keypad appears.
4. Use the Jog Keypad to jog the tool to the bottom of the front right corner of the workpiece (the end of the stock furthest from the chuck and at the edge of the stock). You can also select another point on the workpiece as a reference point. However, this reference point must be consistent for all tools.
5. Select **Set Position** from the Setup Menu. The Set Position dialog box appears.




6. Enter zero in the Z and X boxes to establish work origin.
7. Click on **OK**. The values in the Position Readout all change to zero.




8. Jog the tool up and away from the workpiece. Press the Emergency Stop button, open the Safety Shield and remove the workpiece.

## Establish the Tool Offsets for Tool 2 and Tool 3

The second tool for this program is a Left-Hand tool used for performing the facing cuts. The third tool is a cutoff tool. To set the multiple tool offsets for these two tools:

1. Jog the second tool to the same reference point as Tool #1. Jog in small increments to move the tool to the exact position of the reference point.
2. Open the **Tool Library**. For Tool #2, check Use Current Position. Select **OK** to accept the new offset.
3. Repeat these two steps for Tool #3.

## Initialize the Workpiece Origin

To initialize the workpiece origin, you must do one of the following:

In the **Setup** menu, select **Set Position**. Enter the desired values for X and Z.

Or  use the **G92**  code,Preset position. This code works like the Set Position command under the Setup Menu. The X and Z coordinates following a G92 code define the new current position of the tool.

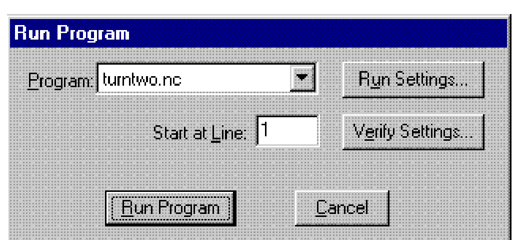
Or use the **G54** code, Use Coordinate System One, to use this preset position.

## Dry Run the NC Program

Before you run this multiple tool part program for the first time, you should perform a dry run (run the program with no stock mounted). This will ensure that all the movements of the Turning Center make sense and that none of the tools are in no danger of striking any fixtures of the Turning Center such as the headstock or workpiece. Although you should dry run the program with no stock mounted, you must first set the point of origin using the workpiece as previously instructed, and then remove it.

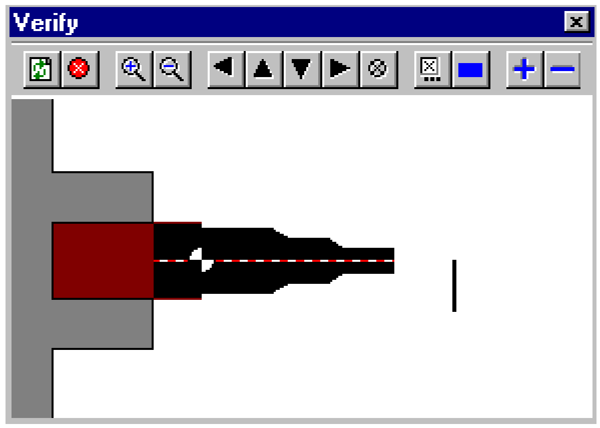
Begin with the Emergency Stop button pressed in, and the spindle speed turned all the way down. The tool should still be mounted in the tool post.

1. Jog the tools up and away from the workpiece. Press the Emergency Stop button, open the Safety Shield and remove the workpiece.
2. Return the Safety Shield to the closed position and pull out the Emergency Stop button.
3. Put on a pair of safety glasses and complete the Safety Checklist (refer to the Reference Guide: Section J).
4. Select **Run/Continue** from the Programs Menu. The Run Program dialog box appears.



12. Click on the **Run Program** button.

13. As the part program runs, observe the tool motion in relation to the chuck (and eventually the workpiece). Look for signs of a possible tool crash and be prepared to press the Emergency Stop switch on the Turning Center. Edit the program as required. When you are satisfied that the tool motions are correct, you can mount the workpiece and run the program.




## Mount the Workpiece

1. Using the Jog Keypad, jog the tool post up and out of the way.
2. Before mounting the workpiece, push the Emergency Stop button in.
3. Mount the 3"x.75" workpiece in the chuck. Take care to position the workpiece perpendicular to the tool post.
4. Pull the Emergency Stop button out.
5. Jog Tool #1 to position the center of the tool tip at the top of the front, right corner of the workpiece or which ever reference point you have set for each of the three tools. Jog Tool #1 to the corner of the workpiece.
6. Select the **Set Position** command from the Setup Menu and enter Z= 2.0", the length of the workpiece, and X= 0.375", the radius of the workpiece.

The workpiece is now correctly mounted.

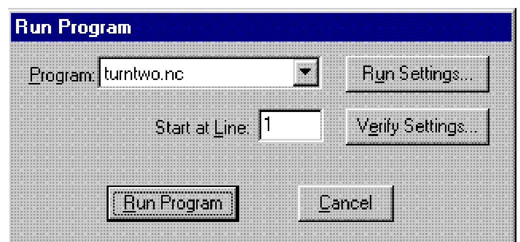
# Run the Program

Before executing the turntwo.nc program, check that all safety precautions have been taken. The Turning Center safety shield should be closed, and you should be wearing safety glasses.

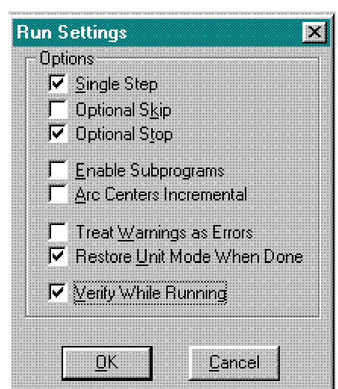
If anything goes wrong, immediately press the Emergency Stop button on the Turning Center to stop the operation. A safety checklist has been provided in the Reference Guide: Section J of this guide. Post a copy of this checklist near the Turning Center and review it before you run any NC program.

To run the program:

1. **After reviewing the Safety Checklist**, select the **Run/Continue** command from the Program Menu. The Run Program dialog box appears.




2. Make sure that the **Start Line** box is set to line 1 of the program.
3. Click on the **Run Settings** button. The Run Settings dialog box appears.




4. Make desired changes in the Run Settings dialog box, then select **OK**.
5. Click on the **Run Program** button to begin running your program.
6. After the part is turned, press the Emergency Stop button before opening the safety shield and removing the finished part.

### To change any of the Run Settings:

Click on an item's checkbox. The Run Settings include:

**Single Step** Allows you to run the NC part program one line at a time, pausing after each line is executed.

**Optional Skip** Recognizes the optional skip code (/ ).

**Optional Stop** Pauses the NC program at any M01 code.

**Enable Subprograms** Must be on if the NC program uses subprograms.

**Arc Centers Incremental** Recognizes the % code indicating that the center of an arc is an incremental value relative to the start of the arc.

**Treat Warnings as Errors** Halts the NC program at a warning as though it were an error.

**Restore Unit Mode When Done** Restores the original unit mode (inches or metric) regardless of the units used in the current NC program.

**Verify While Running** Allows tool path verification to occur while the NC program is running on the Turning Center.

# Using the Optional Automatic Tool Turret

When using the Automatic Tool Turret for multiple tool operations, use the Configure Turret button on the Turret Control Toolbar or the Configure Turret command in the Tool menu to define each tool in its current position in the Turret. To establish tool offsets, use the same procedure used for tools mounted on the cross slide; make one tool the reference tool and offset all other tools from that reference. Another option is to enter the offsets values from the Z axis in the Setup Library from the Tools Menu.

You must use an M06 code with the T codes for multiple tool operations with the Tool Turret. If you do not use an M06 code, the machine will use the tool currently selected in the turret and interpret the T code as an offset reference only.

When used with an M06, the T code simply indicates, by number, which tool and which offset to use. For example, M06T2 tells the turret to rotate to tool #2 and use offset #2.

## To make the turret rotate to a specific tool...

Select the tool you would like to use by selecting the proper Tool Station button from the Turret Control Toolbar.

You can also select the proper tool in the Select Tool From command from the Tool menu which will show you a list of each of the defined tools that are on the Turret Control Toolbar.

## To home the turret position...

**Important!**

Make sure the tools have clearance from the stock and chuck before selecting the moving the turret.

Whenever starting up the machine, you should home the turret to the Tool #1 position. The turret can then move correctly to any other tool position. To home the turret, you can use the Set/Check Home command to establish an initial reference point. See section E for information on using the Set/Check Home Command.

# Threading

Note:

Threading on the spectraLIGHT Turning Center should be performed at low spindle speeds (0 to 200RPM, depending on the threads per inch).

Note:

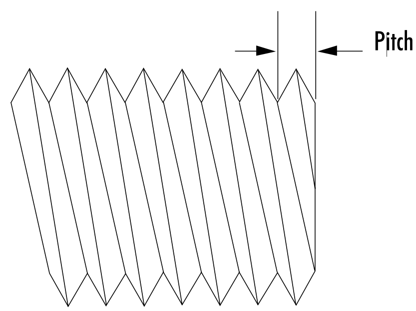
Always position the Spindle Speed Switch on the Turning Center in the "Computer" position for thread turning.

Generally, machine screws used in the United States conform to the American National, or Unified Thread System. Threads are specified by the outside diameter of the thread, then the threads per inch, such as 1/4-20.

Threading with a single point tool is accomplished by taking a series of cuts in the same helix of the thread (also known as chasing a thread).

The *pitch* of the thread is the distance that the tool moves along the Z axis for each revolution of the spindle. The pitch is specified by the F code in the NC program as the inverse of the number of threads per inch. For example, the F code parameter for a 1/4-20 UNC thread is specified as 1/20 or 0.05.

A direct ratio exists between the spindle speed, the feed rate, and the *pitch* of the thread.




Pitch = inch/thread

The pitch of the thread multiplied by the spindle speed determines the Z axis feed rate of the Turning Center.

F (IN/REV) x S(REV/MIN) = Feed Rate (IN/MIN)

So, cutting a 1/4-20 thread with a spindle speed of 200 RPM results in a feed rate of F0.05 x S200 = Feed Rate 10 ipm.

It is important to specify an appropriate spindle speed for the pitch of the thread to keep the required Z axis feed rate below the machine's maximum cutting feed rate of 12 ipm. Specifying an inappropriate spindle speed will result in a bad thread being cut.

On many lathes the gear ratio is altered by manually changing gears or by using a quick change gear box. These gear changes increase or decrease the rate at which the lead screw turns, thereby changing the feed rate of the tool. The spectraLIGHT Turning Center, however, utilizes an F code in the NC program to instruct the Z axis drive motor, which turns the lead screw.

**Note:**

It is important that the speed of the spindle rotation has time to slow to the designated value before a threading motion is made. To accomplish this, specify the spindle speed for threading then dwell for several seconds.

The spectraLIGHT Turning Center uses electronic gearing. An optical encoder on the spindle measures the speed of the spindle and instantaneously varies the speed of the Z axis to keep a constant ratio for cutting a thread. An index sensor, which triggers once per revolution of the spindle, is also used, so the thread is started at the same point on the rotation of the part for multiple passes to rough out the thread.

## Programming for Threading

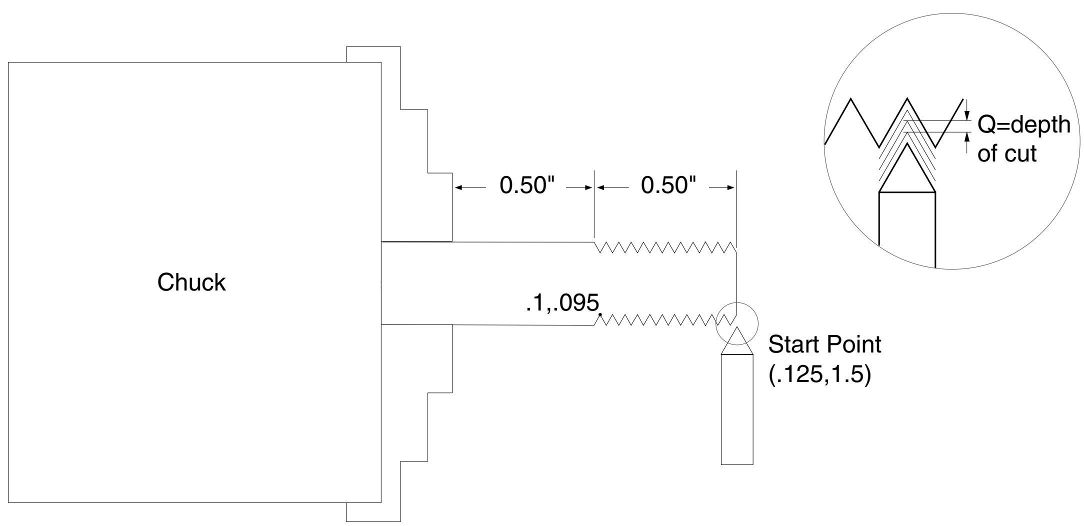
Here's a typical NC program for cutting a 20-pitch thread on a .25" diameter rod:

This NC program is used to turn a 1/2"-long 1/4-20 thread.

You can find a copy of this program "THREAD20.NC" in the SLATHE/programfiles/LMC/WSLT directory.

```
G0X.125Z1.5;TO SET THE START POINT
S200M03;SPINDLE SPEED 200RPM
G04F5;DWELL FOR 5 SECONDS
G32X.095Z1Q.002F.05;X & Z ENDPOINTS, Q=DEPTH, F=PITCH
G80;END CANNED CYCLE
```

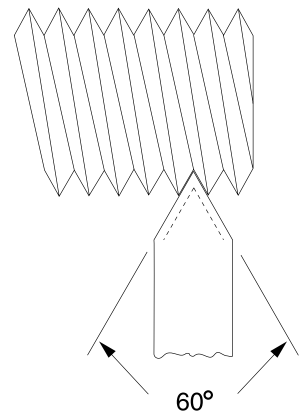
The recommended Q depth for threading is very small; from .001 to .003.



## Tooling

Most threads are cut with a 60° threading tool. For fine threads, the tool can be ground to a sharp point. For coarse threads, you may wish to radius the point.

A threading tool is used for external threading operations. An inside threading tool is used for internal threading, like on a nut.

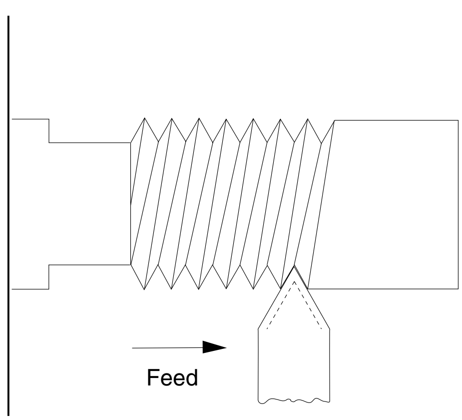



## Cutting Left-Hand Threads

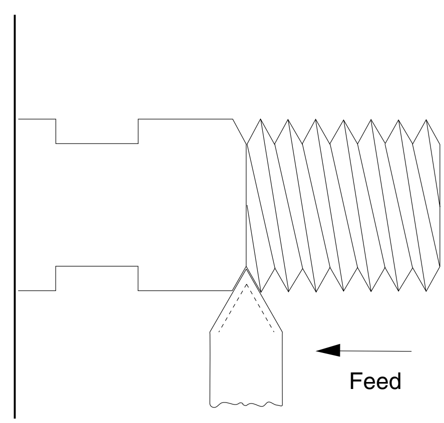
Most threaded parts are made with right-hand threads. The Turning Center can turn both right-hand and left-hand threads. The same tool is used for both left- and right-handed threading; the direction of the cut is altered.

The procedure for cutting left-hand threads on the Turning Center is the same as for right-hand threads, except the feed is reversed so the cut is made from the left to the right.

Make sure the workpiece is tightly secured in the chuck or collet since the tool will be pulling the workpiece away from the holding device.

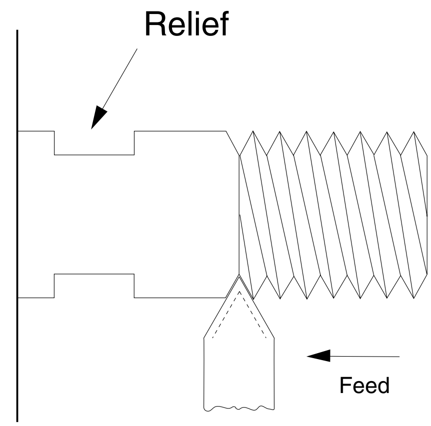



Left-Hand Threading




Right-Hand Threading

You still have to cut a relief groove to provide a starting place for the tool. Make sure you have sufficient relief on the right side of the tool as well.




## Internal Threading

Most of the same rules that apply to external threading also apply to internal threading. You have to remember that your starting X value will be smaller than the ending X value. Also, you won't be able to see the tool, so use caution.

To perform single point internal threading, a hole is first drilled to 1/16 inch diameter less than the minor diameter. A boring bar is then used to bore the hole to the minor diameter of the thread. Holes that are smaller than the minimum diameter the boring bar can turn can not be tapped.

## Before You Start...

Before cutting a thread, you should cut a groove, or a relief behind the end point of the thread. This is so the tool isn't abruptly removed from the workpiece after the final thread is cut, making the end of the helix straight.

Make sure the start point is away from the workpiece on the Z axis because you want the tool and workpiece to be in motion before cutting begins.

## Setting Up for Threading

The workpiece should be mounted in a chuck or in a collet. Set the tool height exactly on the centerline of the workpiece for a correct thread angle. Locate the zero point of the workpiece just as you would for other turning operations.

Reference Guide: Section I

# General Machining Information

Understanding Coordinate Systems

Feed Rate and Depth of Cut

Spindle Speeds

Feed Rate and Spindle Speed Selection

Lubricants and Coolants

Tool Types

Mounting the Cutting Tool

Sharpening the Tools

# Understanding Coordinate Systems

For a beginning user, understanding coordinate systems can be difficult. The first thing to remember is, before performing most machining operations, you are required to set the machine to "home" position. This returns the machine to the machine zero point, and acts as a reference point for all operations. It is a good idea to home the machine every time you run the control software.

## How Coordinates Relate to the Turning Center

Note:

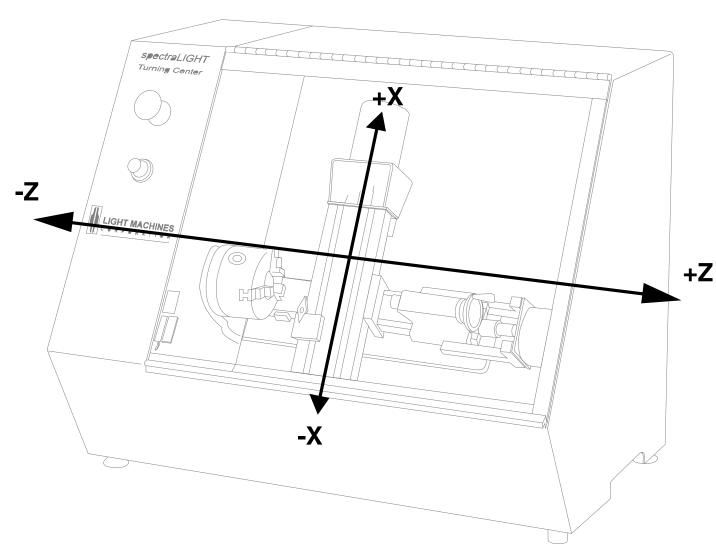
As explained in Section A, with the use of the Tool Turret, the polarity of the X axis is reversed.

In machining, the Z axis is always the spindle axis. The Z axis on a machine tool can be horizontal (as on a lathe) or vertical (as on a vertical mill).

On the spectraLIGHT Turning Center the Z axis is horizontal (parallel to the spindle axis). The X axis is also considered horizontal, but perpendicular to the Z axis and parallel to the cross slide. The Y axis is not used because the tool does not move vertically.

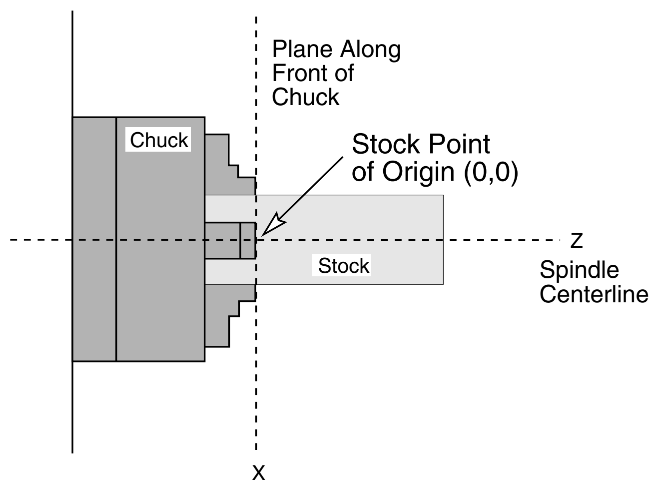
In NC programming, the programs are written as though the workpiece is stationary and the tool is moving. The motion of the tool from right to left or left to right, is along the Z axis. The motion of the tool from front to back (in and out of the workpiece) is along the X axis.

The point of origin can be located anywhere on the workpiece, but is usually located on the Z axis centerline on the plane along the front of the chuck. This way, all the work is done in Quadrant 1, with positive values for X and Z. What this means is that all tool motions will be determined by positive values. This does not mean that the tool will always move in the positive direction, only that the numbers which describe the coordinates will be positive. For example, when the tool move into the workpiece, it moves in the negative X direction.



## Machine Coordinates

Machine Zero is usually located on the Z axis centerline on the plane along the front of the chuck. This is a fixed point on the machine, and can not be changed. The machine uses this as a reference point for all operations. If the machine is not homed (set to the machine zero) it can not accurately locate the workpiece on the cross slide. The machine is homed by using the **Set/ Check Home** command under the Setup menu. First, set a measured piece of stock in the chuck so that you can set the position of the tool at the end of the stock, which is the X,Z distance from zero. If using a stock piece measuring 3" long and .75" wide, by setting the tool position at the top edge of the front of the workpiece, above the Z axis centerline with coordinates X= 0.375, Z= 2.5, the Turning Center knows where the Machine Zero is.




## Work Coordinates

Once home is set, open an NC program using the Control Program. The NC program will need a point of origin to start from. Setting a point of origin will establish the work coordinates. Work coordinates relate to the workpiece, and are usually set at the top edge of the front of the workpiece, above the Z axis centerline of the workpiece. Once the stock is mounted in the chuck, jog the tool tip to this position on the workpiece. From the Setup Menu, select **Set Position**. The tool coordinates appear in the dialog box. Set the coordinates to X= 0.375, Z= 2.5 and click on OK to set the current tool position as the point of origin on the workpiece.

# Multiple Coordinate Systems

For more advanced operations, such as machining multiple partson the same piece of stock, you can set up multiple coordinate systems. For example, if you have an NC program that machines a complex shape and you want to machine that shape in multiple places on the same part, use the following procedure.

Move the tool tip to the work coordinate point of origin (0,0) using the **Set Position** command from the Setup Menu. After setting this location on the workpiece as the origin, select the **Coordinate Systems** command from the Setup Menu. The Setup Coordinate Systems dialog box appears. Select one of the **G54** to **G59** codes, enter the coordinates for the first workpiece, and click on **Apply**. Repeat this procedure for as many coordinate systems as necessary by setting up a coordinate system for each point on the part that corresponds to the zero point of the shape you are turning. Then in your program, you move to each coordinate system and run the subprogram that contains the NC code for the complex shape. (For more information about the Setup Coordinate Systems procedure, see Section E.)

# Feed Rate and Depth of Cut

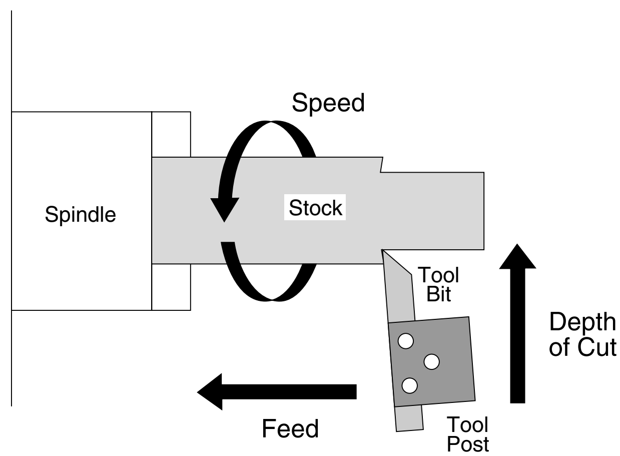
Two terms used in general machining are *feed* and *cut*.

Normal machining on the Turning Center involves reducing the diameter of a workpiece. This is accomplished by advancing the cutting tool into the workpiece by an appropriate amount (depth of cut).

The rate of tool travel is called the feed rate. On machines like the spectraLIGHT Turning Center, the tool advances into the workpiece and the workpiece is rotated on its centerline by the spindle.

The depth of cut and feed rate you select should depend on the turning speed of the spindle, the type of material and lubricant used, and the type of cutting tool used for the operation.

Excessive depth of cut and high feed rates place greater strain on the spindle, may bind the tool and workpiece, or produce a poor surface finish on the part.



## Spindle Speeds

Turning speed is inversely proportional to the diameter of the workpiece; the larger the workpiece, the slower the recommended turning speed. The relative hardness of the material also affects turning speed; the harder the material is, the slower the speed should be.

High spindle speeds may produce excess heat which causes the workpiece to expand. If the workpiece expands, the cutting tool will *rub* rather than cut the material, resulting in a poor surface finish. Slow spindle speeds cause no harm, but may be inappropriate for finishing certain types of materials.

The load put on the spindle motor must also be taken into account. Heavy cuts at low speeds will make the motor run hotter than lighter cuts at higher speeds. The selected feed rate and depth of cut should not cause the spindle motor to greatly lose speed or cause the tool to *chatter* against the workpiece.

## Feed Rate and Spindle Speed Selection

Feed rate selection for turning parts on the spectraLIGHT Turning Center depends on factors such as type of material, type of cut, diameter of rough stock, depth of cut, and spindle speed. The type of tool chosen to make the cuts also affects the depth of cut and, therefore, the feed rate. Consult your machinist's handbook for selecting a feed rate based on spindle speed and material type. Experience and experimentation will enable you to establish feed rates best suited to particular applications.

## Lubricants and Coolants

Lubricants remove heat from the tool and workpiece and are often used when high production rates are required or when cutting very hard materials, such as stainless steel. A mixture of one part soluble oil to six parts water may be used on steel to assist in producing a smoother finish and to reduce tool chatter. Aluminum and aluminum alloys may require the use of paraffin, oil or kerosene to prevent chips from welding to the tool's cutting edge. Brass and cast iron are always turned dry.

When lubrication is necessary, small amounts of water-soluble cutting fluids are recommended for use on the spectraLIGHT Turning Center. Lubricants should be wiped from the machine after use, because some petroleum-based fluids may deteriorate the electrical wiring insulation, the plastic safety shield, or the computer enclosure.

Note:

The spectraLIGHT Turning Center is not designed for flood cooling. Small amounts of coolant may be applied to the tool tip and workpiece before a program is run.

Note:

Short-run, small part machining in Delrin or aluminum on the spectraLIGHT Turning Center does not require the use of coolant.

# Tool Types

Cutting tools are usually made from hardened steel and are ground to various shapes. The clearances ground behind cutting edges are adjusted for the type of material the tool will cut and the direction the tool will be fed along the workpiece. Tools are often ground to shape by the operator to suit a particular cutting requirement.

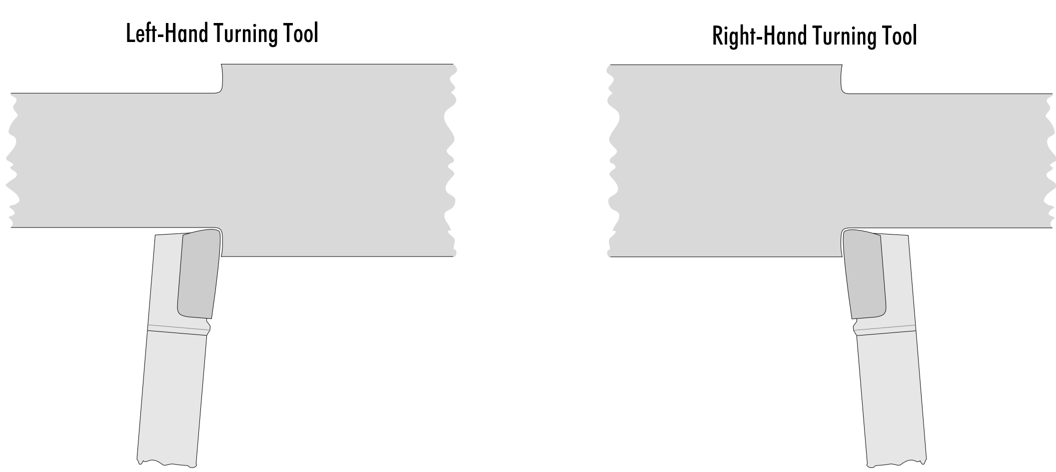
Insufficient clearance behind the cutting edge will cause the tool to rub. Excessive clearance will produce a ridged or wavy finish due to the small length of tool edge in contact with the workpiece. Standard tool types are: turning tools, side tools, parting tools and boring tools. Carbide tooling has become popular in recent years. Carbide is more brittle than steel, but has a longer tool life.

# Side Tools

Side tools are used to face-off the ends of shoulders or to make facing cuts in the surface of a workpiece held in a chuck. They may also be used as turning tools.

Right side (or right-hand) tools feed from right to left and are used to reduce the workpiece to a desired diameter. The shape of the cutting edge and the clearance (behind the point between the end of the tool and the workpiece) determine the surface finish of the workpiece. Rough cuts are made in small increments until the tool is within approximately 0.010 inch (0.25mm) of the desired diameter. Final cuts are made at slow feed rates with a very shallow depth of cut.

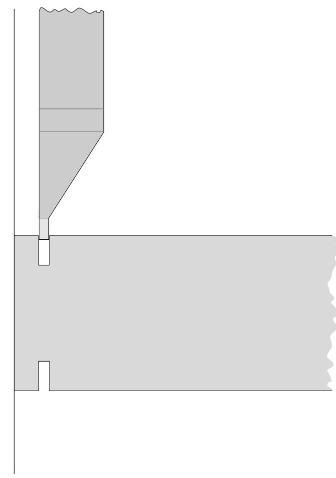
Left side (or left-hand) tools feed from left to right. Side tools cut very flat surfaces and can be used to produce a part with an exact thickness.



# Parting Tools

A parting tool has a dual-edged, dovetail-shaped cutting end which is used to cutoff workpieces. The parting tool is plunged into the workpiece and the cross slide is moved across the lathe bed until the workpiece is severed.

A Parting, or Cutoff tool at work

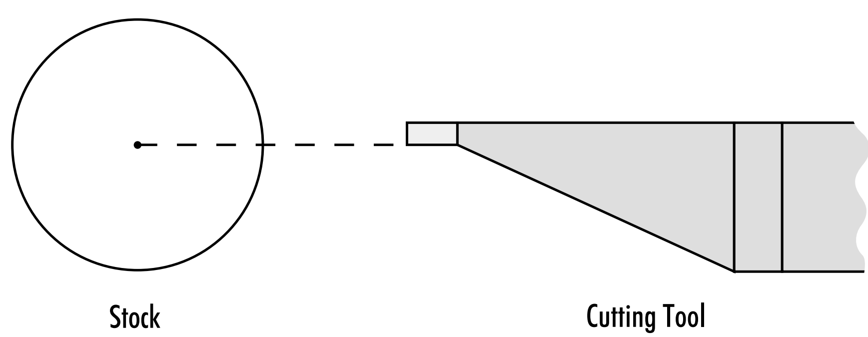


Parting tools are clamped to a special tool post with a minimum of over-hang for maximum rigidity and chatter-free operation. The cutoff point should be located as close to the spindle center as possible. The cutting end of the parting tool should be perpendicular to the workpiece to avoid sideward drift. A small square is useful in aligning the tool perpendicular to the workpiece.

Also, the height of the tip of the tool should be positioned so it is vertically aligned with the center of the stock.

Cutoff operations are performed at a slow turning speed because the parting tool has a large amount of cutting edge in contact with the workpiece. If the tool chatters or produces noise, the turning speed and feed rate should be reduced.

Make sure to align the tip of the cutoff tool with the center of the stock

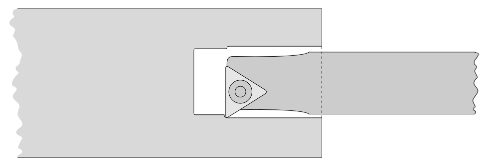


## Boring Tools

Boring tools are used to enlarge or modify a drilled or cored hole in a work-piece. Clearance must be maintained behind the cutting point of the tool.

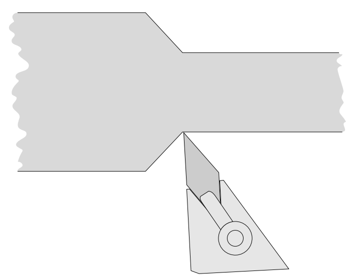
A slow feed rate and frequent tool withdrawals are required with boring tools because chips cannot freely escape from the hole. Depth of cut and feed rates must be reduced to avoid chatter. The tool should not be driven deeply into a hole. When boring a hole where a flat bottom is required, stop the feed at least 0.002 inch from the desired depth of the smaller hole being bored out.

A cutaway view of a Boring tool at work



## Profiling Tools

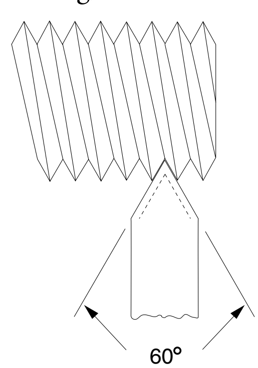
Profiling tools are quite popular in CNC applications because they can cut on both sides and in both directions. A profiling tool cuts in the same way as a turning tool.



## Threading Tools

Most threads are cut with a 60° threading tool. For fine threads, the tool can be ground to a sharp point. For coarse threads, you may wish to radius the point.

A threading tool is used for external threading operations. An inside threading tool is used for internal threading, like on a nut.



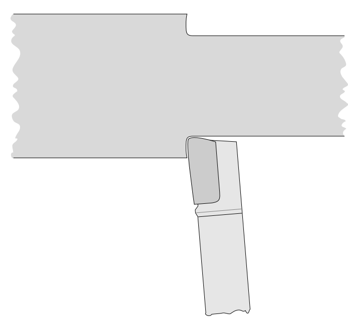
# Mounting the Cutting Tool

Each cutting tool used in the machining operation must be sharp and tightly inserted in the tool post. The cutting edge of the tool must be on the centerline or just below the centerline (0.004  inch or 0.1mm maximum) of the axis of rotation of the lathe.

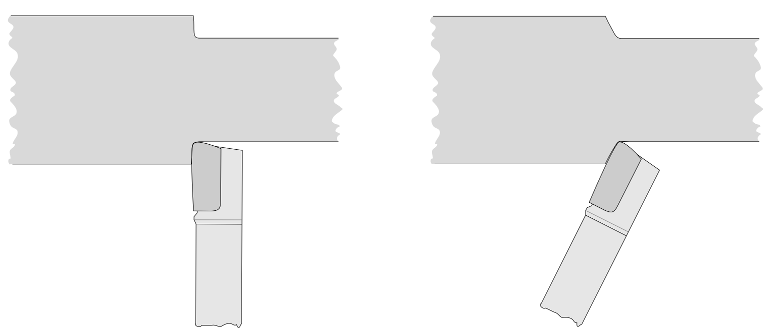
The cutting tool is mounted by loosening the mounting screws at the top of the tool post and positioning the tool in the slot beneath the mounting screws. To assure a rigid mounting and to avoid chatter, the cutting edge of the tool should not protrude more than necessary from the tool post. The further the tool extends from the tool post, the more chatter will occur.

You must make sure that only the very tip of the tool is doing the cutting. All tools must be positioned in a manner that allows them to access all areas they are to cut. The exception to this is the cutoff tool, which should always be perpendicular to the stock. Backside tools should be mounted upside-down and on the centerline or just above.

This angle is correct for a right-hand turning tool.



These angles are incorrect for a right-hand turning tool.



The position of the cutting edge relative to the centerline of rotation can be checked at the headstock center. Loosen the bolt holding the tool post to the cross slide and slide the tool post as close as possible to the center. Check the location of the cutting edge position.

Tighten the tool holding bolts, reposition the tool post on the cross slide and tighten the cross slide bolt.

## Sharpening the Tools

A cutting tool must be sharpened regularly to preserve its original cutting angle and shape. Longer tool life will be obtained from cutting edges if they are finished with a small oilstone. Only the cutting end and sides of the tool should be ground as required. Never grind the top face of the tool.

Reference Guide: Section J

# Safe Turning Center Operation

Safety Rules

Safety Checklist

Lista de Seguridad

Emergency Stops

# Safety Rules

Feel free to copy these rules, or the Safety Checklist, and post them in your work area for quick reference.

The following safety rules should be reviewed and practiced by all operators of the spectraLIGHT Turning Center.

## Wear Safety Glasses

Foresight is better than no sight. During operation any power tool can throw foreign objects and harmful chemicals into your eyes. Always put on safety glasses or eye shields before starting up the Turning Center. Safety glasses or shields should provide full protection at the sides, as well as the front of the eyes.

## Know Your Machine Tool

Read this guide carefully before you use the Turning Center and keep it readily accessible for quick reference. Know the intended applications and limitations of the Turning Center as well as its hazards.

## Ground All Tools

The Turning Center has an AC power cord terminated by a three-prong plug. The power cord should be plugged into a three-hole, *grounded* receptacle. If a grounding adapter is used to accommodate a two-prong receptacle, the adapter wire *must* be attached to a known ground. *Never* remove the third prong from the plug on the AC power cord.

## Keep the Safety Shield in Place

The safety shield should remain in place whenever the spindle motor is on or the cross slide is moving.

## Remove Adjusting Keys and Wrenches

Make it a habit to check that keys and adjusting wrenches are removed from the Turning Center before turning on the machine.

## Keep the Work Area Clean

Cluttered work areas and bench tops invite accidents.

## Avoid a Dangerous Environment

Don’t use the Turning Center in damp or wet locations. Never operate electrical equipment in the presence of volatile and flammable petroleum-based solvents and lubricants.

## Keep Untrained Visitors Away from the Equipment

Children and visitors unfamiliar with the hazards of rotating machinery should always be kept away from the work area.

## Prevent Unauthorized Users from Operating the Turning Center

Lock and remove the key from the Turning Center control panel when the system is not in use.

## Do Not Force a Tool

Select the feed rate and depth of cut best suited to the design, construction and purpose of the cutting tool. It is always better to take too light a cut than too heavy a cut.

## Use the Right Tool

Select the type of cutting tool best suited to the turning operation. Don't force a tool or attachment to do a job it wasn't designed to do.

## Dress Appropriately

Don't wear loose clothing or jewelry which can get caught in moving parts. Wear a hat or hair net, or tie your hair back to keep it away from moving parts.

## Secure the Workpiece

Be certain that you have firmly secured the workpiece in the spindle and the cutting tool to the cross slide before turning on the spindle motor.

## Do Not Overreach

Keep your footing and balance at all times so you won’t fall into or grab the moving machine.

## Maintain Cutting Tools In Top Condition

Keep cutting tools sharp and clean. Lubricate and clean Turning Center components on a regular basis.

## Disconnect Tools Before Servicing

Always use the emergency stop switch to disconnect power and disable the spindle motor before mounting or removing the workpiece, or changing tools. *Do not* rely solely on a programmed Pause command to disable Turning Center operation.

## Avoid Accidental Starting

Make sure the power switch on the control box is off before plugging in the Turning Center power cord.

## Use Recommended Accessories

To avoid stressing the Turning Center and creating a hazardous machining environment, use only those accessories designed for use with the spectraLIGHT Turning Center, available through Light Machines Corporation.

## Tighten All Holding, Locking and Driving Devices

Do not overtighten tool holding devices. Overtightening may damage threads or warp parts, thereby reducing accuracy and effectiveness.

## Keep Coolant Away from Electrical Components

Do not allow coolant to splash into or near the computer.

## Do Not Operate the Machine Under the Influence of Alcohol or Drugs

Alcohol or drugs may impair your judgement and reaction time, which could contribute to an on-the-job accident.

## Avoid Distractions While Running the Machine

Use simple common sense and pay attention while operating any piece of machinery.

# Safety Checklist

**IMPORTANT!**

Post copies of this checklist in the work area. Verify that all items are checked-off prior to each operation of the spectraLIGHT Turning Center.

**Before you enter the work area:**

- ❏ Put on safety glasses.
- ❏ Tie back loose hair and clothing.
- ❏ Remove jewelry including rings, bracelets and wristwatches.

**Before machining a part:**

- ❏ Make sure you have the correct tool for the job.
- ❏ Secure the tool properly.
- ❏ Make sure all tool positions have been properly initialized.
- ❏ Verify the NC program on the computer before machining.
- ❏ Remove all loose parts and pieces from the machine.
- ❏ Remove adjusting keys and wrenches from the machine.
- ❏ Close the safety shield.
- ❏ Only operate the machine after being properly trained in its use.
- ❏ Perform a dry run:
  - ❏ Set the spindle motor switch to lowest manual speed setting.
  - ❏ Make certain there is no workpiece in place.
  - ❏ Run the NC program to make sure all the moves make sense before running the program with a workpiece in place.
- ❏ After completing the dry run, properly secure the workpiece to the machine.
- ❏ Keep fluids away from all electrical connections, electronic or electrical devices, the computer and nearby electrical outlets.

**While machining a part:**

- ❏ Do not touch moving or rotating parts.
- ❏ Press the Emergency Stop button before opening the safety shield.
- ❏ Only open the safety shield after the spindle has stopped rotating.
- ❏ Press the Emergency Stop button whenever changing tools or mounting or removing a workpiece.
- ❏ Pull the Emergency Stop button out only after closing the safety shield.
- ❏ Keep all unauthorized persons away from the work area.

## ¡IMPORTANTE!

Pegue copias en el área de trabajo. Verifique que todos los puntos esten checados antes de cada puesta en marcha de la máquina.

# Lista de Seguridad

**Antes de entrar en el área de trabajo:**

- ❏ Use sus lentes de seguridad.
- ❏ Procure recogerse el cabello y no usar ropa floja.
- ❏ No use joyería como: anillos, pulseras y relojes.

**Antes de trabajar a máquina una pieza:**

- ❏ Utilize la herramienta correcta para el trabajo. Asegurela de forma correcta en el husillo con una boquilla.
- ❏ Asegurese que la posición de la herramienta de corte ha sido inicializada correctamente.
- ❏ Remueva todas las partes sueltas y coloquelas lejos de la Fresadora. Limpie todos los residuos de la Fresadora después de cada corrida.
- ❏ Cierre la guarda de seguridad antes de ejecutar cualquier operación en la Fresadora.
- ❏ Corra los programas por primera vez con el motor del husillo apagado y sin pieza de trabajo. Asegurese que todos los movimientos sean correctos.
- ❏ Asegure la pieza de trabjo a la mesa. Quite las herramientas y llaves antes de cerrar la guarda de seguridad.
- ❏ Asegurese que todos los contactos de corriente A.C. esten aterrizados.
- ❏ Mantenga los líquidos refrigerantes lejos de la Caja de Control, Computadora y cualquier Suministro Eléctrico.

**Mientras trabaja a máquina una pieza:**

- ❏ Nunca levante la guarda de seguridad mientras que la Computadora este ejecutando un programa. Presione siempre primero el botón de “Paro de Emergencia”.
- ❏ Presione siempre el botón de “Paro de Emergencia” cuando se cambie una herramienta, se coloque o remueva una pieza de trabajo. Jale el botón de “Paro de Emergencia” después de haber puesto la guarda de seguridad.
- ❏ Mantenga fuera del área de trabjo a toda persona no autorizada.

# Emergency Stops

All spectraLIGHT operators must be fully aware of how to shut down the machine quickly, should the need arise.

There are two ways an emergency stop can be initiated on the spectraLIGHT Turning Center:

- By pressing the emergency stop switch on the Turning Center.
- By pressing a key on the computer keyboard.

# G and M Codes Listed by Group

# G Codes by Group

### Interpolation Group

| | |
|---|---|
| **G00** | Rapid traverse. |
| **G01** | Linear interpolation. |
| **G02** | Circular interpolation (clockwise). |
| **G03** | Circular interpolation (counterclockwise). |

### Programming Mode Group

| | |
|---|---|
| **G90** | Absolute coordinate programming: All X and Z axes coordinates are relative to a (0,0) location on a lathe. |
| **G91** | Incremental coordinate programming: Each command is relative to the one before it in the program. |

### Units Group

| | |
|---|---|
| **G70** | Inch: Used to instruct the lathe that inches are the unit of measure for the part program. (Fanuc G20) |
| **G71** | Metric: Used to instruct the lathe that millimeters are the unit of measure for the part program. (Fanuc G21) |

### Wait Group

| | |
|---|---|
| **G04** | Dwell (wait): Equals the value of the feed rate (F code) in seconds (used primarily for robotic operations). G04 excludes motion commands with a new feed rate on the same line (block). |
| **G05** | Pause: Used for operator intervention. The order of action for the pause and dwell codes in one NC block is G05, G04 (pause, dwell). |
| **G25** | Wait for robot input #1 to be high: Used in conjunction with H code, which specifies input number. The default is H5. Used for robot synchronization (see Section L). |
| **G26** | Wait for robot input #1 to be low: Used in conjunction with H code, which specifies input number. The default is H5. Used for robot synchronization (see Section L). |

### Polar Programming Group

| | |
|---|---|
| **G15** | Polar programming cancel. |
| **G16** | Begin polar programming. |

### Coordinate System Group

| | |
|---|---|
| **G53** | Rapid traverse to specified machine coordinate in Absolute programming mode. (e.g. G53X0Z0 rapids to machine reference point) |
| **G54** | Use coordinate system one. |
| **G55** | Use coordinate system two. |
| **G56** | Use coordinate system three. |
| **G57** | Use coordinate system four. |
| **G58** | Use coordinate system five. |
| **G59** | Use coordinate system six. |

## Canned Cycle Group

| | |
|---|---|
| **G32** | Canned cycle thread cutting |
| **G72** | Canned cycle arc turning, clockwise |
| **G73** | Canned cycle arc turning, counterclockwise |
| **G77** | Canned cycle side turning |
| **G79** | Canned cycle end turning |
| **G80** | Canned cycle cancel. |
| **G81** | Canned cycle drilling. |
| **G83** | Canned cycle peck drilling. |

## Preset Position Group

| | |
|---|---|
| **G27** | Check reference point: This code moves the tool to its home position on the Machining Center to perform automatic calibration of the axes. |
| **G28** | Set reference point: Sets machine position to 0,0. |
| **G29** | Return to reference point: Moves the tool to a coordinate specified by XZ. Typically used after a G27 or G28 code. |
| **G92** | Set position: This code works like the Set Position function under the Setup Menu (see Section E). The X and Z coordinates following a G92 code define the new current position of the tool. |
| **G98** | Rapid move to initial tool position after canned cycle complete (Section G). |
| **G99** | Rapid move to point R (surface of material or other reference point) after canned cycle complete (Section G). |

## Scaling Group

| | |
|---|---|
| **G50** | Cancel scaling. |
| **G51** | Invoke scaling. Use this code to scale axes by a single factor or by independent factors around a fixed origin. The default scale factor is 1. |

## Rotation Group

| | |
|---|---|
| **G68** | Invoke rotation. Use this code to rotate geometry by an arbitrary angle around a specified point. |
| **G69** | Cancel rotation. |

# M Codes by Group

## Program Stop/End Group

**M00** *Pause:* Allows you to place a pause in your code. Acts like a G05 pause.

**M01** *Optional Stop:* Allows you to place an optional stop in your code. Place an M01 in the block of code where you would like to pause. With Optional Stop on, the M01 works like a G05. With Optional Stop off, the M01 code is ignored, the other codes on the block are executed as usual.

**M02** *End of Program:* Takes effect after all motion has stopped; turns off drive motors, spindle and accessory outlets.

**M30** *Program stop:* Same as M02.

## Spindle Group

**M03** *Spindle Motor On:* Activated concurrently with motion specified in the program block; remains in effect until superseded by M05.

**M05** *Spindle Motor Off:* Activated after the motion specified in the program block; remains in effect until superseded by M03.

## Tool Change Group

**M06** *Tool Change:* Used in conjunction with a T code to perform multiple tool operations. See Section H.

## I/O Group

**M08** *ACC1 On:* Turns on accessory ACC1 outlet concurrently with the motion specified in the program block; remains in effect until superseded by M09.

**M09** *ACC1 Off:* Turns off accessory ACC1 outlet after the motion specified in the program block; remains in effect until superseded by M08.

**M10** *Clamp ACC2:* Turns on ACC2. Closes air vise accessory concurrently with the motion specified in the program block; remains in effect until superseded by M11.

**M11** *Unclamp ACC2:* Turns off ACC2. Opens air vise accessory after the motion specified in the program block; remains in effect until superseded by M10.

**M25** *Set robot output #1 on:* Used for robot synchronization. Used in conjunction with H code to specify output number. The default is H4.

**M26** *Set robot output #1 off:* Used for robot synchronization. Used in conjunction with H code to specify output number. The default is H4.

**M35** *Set robot output #2 on*: Used for robot synchronization.

**M36** *Set robot output #2 off:* Used for robot synchronization.

## Program Management Group

**M20** *Chain to Next Program*: This code appears at the end of a part program and is followed on the next line by the file name of another program which is executed when all motion stops. Here's an example of a part program chain to another program:

```
N37Z.2
N38M20
PROGRAM TWO
```

If the two programs you are chaining are not in the same directory, you must specify the full pathname for each file. If the specified file is not found, the Open dialog box appears so you can locate it.

**M22** *Output to file:* Outputs information to a file. The first time the Control Program encounters an M22 code, it opens the specified file. You must enclose the name of the file in parentheses for the Control Program to recognize it.

The proper format for using this code is: M22([filename.ext [,A]]) [text and macros]. Items in brackets [ ] are optional, except that a filename is required for the first M22. If no text is specified to be output to the file, the current axis positions are output. M22 automatically adds a tilde (~) to the output text, so the next M22 starts on a new line in the file.

If you use more than one M22, only the first occurrence must have the filename in the parentheses. The remaining M22's may have empty parentheses, ( ). If you want to generate more than one file at a time you must include the filename each time you specify M22. If a filename is not specified, the first file opened is used.

Following is a list of special codes that can be used with M22 to generate run-time reports.

| | |
|---|---|
| @X | Current X position (in current coordinate system) |
| @Z | Current Z position (in current coordinate system) |
| ~ (tilde) | New line (starts a new line in the file) |
| @TD | Time of day (12hour): "11:59:59AM" |
| @TC | Time (elapsed) for cycle: "99:11:59" (0's trimmed from left) |
| @TT | Time total (of program run): "99:11:59" |
| @TA | Time Average (per cycle): "99:11:59" ("??:??" if first part) |
| @TL | Current tool number: "5" |
| @C | Cycle number (current pass): "3" |
| @D | Date: "12/31/94" |
| @FN | Current file (without path): "PART.NC" ("UNTITLED.NC" if untitled) |
| \t | Tab |
| \\ | Outputs a single backslash character to the file |

The M22 code supports multiple output files. The first occurrence of a filename opens the file. In the name M22(FILE.OUT,A) TEXT... the output is appended to the file (if it exists). Each unique filename opens a separate file. For backward compatibility, empty parentheses M22( ) TEXT... cause the M22 output to go to the first file that was opened with an M22.

| | |
|---|---|
| **M47** | *Rewind:* Restarts the currently running program; takes effect after all motion comes to a stop. Use the L code to repeat a finite number of times. The L code defines the number of times to run. For example, M47L2 rewinds once. |
| **M98** | *Call to subprogram* |
| **M99** | *Return from subprogram:* Returns you to the block following the initial M98 command.<br><br>*Go to:* Used with P code. P code defines N code destination. Goes to first occurrence of N code within the main program. The N code can not follow any subprogram (O code). |
| **M105** | *Operator message:* A nonstandard Light Machines code used to display messages in the Control Program. |
| **M111** | *Home the X axis* |
| **M112** | *Home the Z axis* |

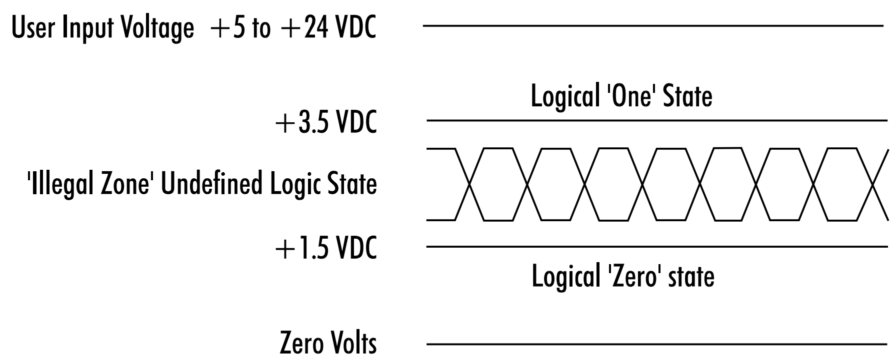
Reference Guide: Section L

# Robotic Integration

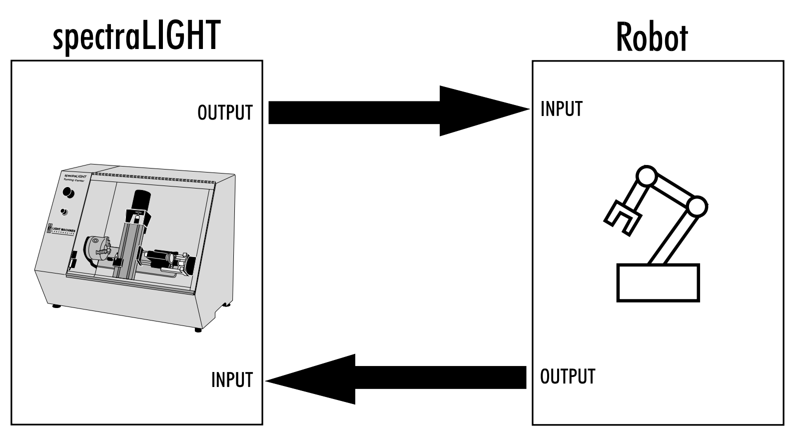
# How Robotic Integration Works

The spectraLIGHT Turning Center has a simple interface for interacting with common robots, like those used for automatic part loading between turning operations. The Turning Center and the robot communicate by way of an interface connector (labeled TTL I/0) located on the rear panel of the Turning Center's Controller Box.

The method of communication between the Turning Center and robot is very basic. They are both able to transmit and receive *high* or *low* signals. Since communication signals are typically 0 and 5 volts, a high signal is 5 volts (or 3.5 volts or greater) which turns the Turning Center or robot on; and a low signal is 0 volts (or 1.5 volts or less) which turns the Turning Center or robot off. See the figure below.



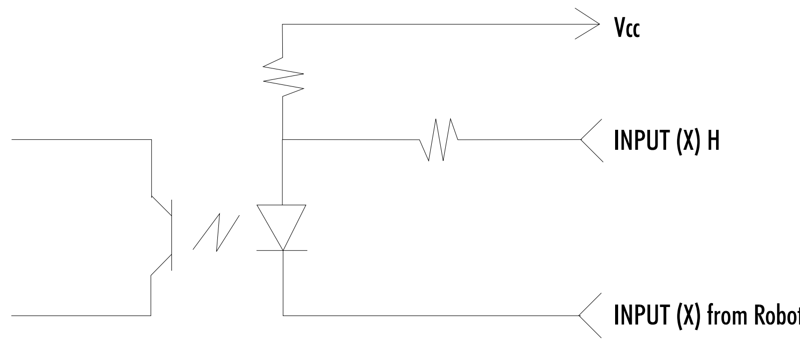

There are NC codes that you can place in a part program to instruct the Turning Center to *transmit* a high or low signal to the robot. There are also codes that instruct the Turning Center to *wait* for a high or low signal from the robot. The Turning Center and robot are set up to recognize any change in signal state; a low signal must go high, or a high signal must go low for the change to be detected by either machine.



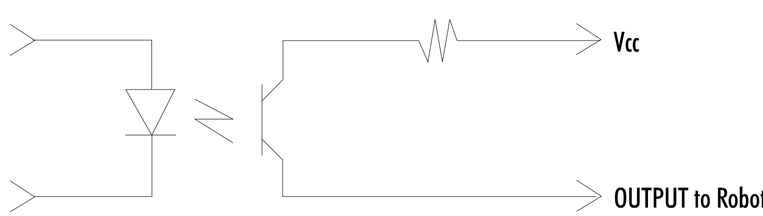
Signals sent out are referred to as *outputs*, while signals coming in are called *inputs*. Any signals that the Turning Center transmits to the robot are transmitted through an *output pin* on the interface connector. Outputs can drive a maximum load of 1mA. Any signals that the Turning Center receives from the robot come in through an *input pin* on the interface connector.

Examples of input and output wiring are shown in the schematic below.

## Input Configuration




## Output Configuration

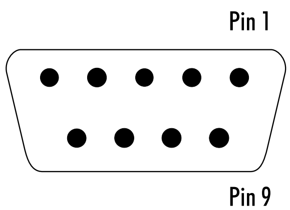


# The Interface Connector

The 9-pin I/O connector on the interface connector provides a number of ways for the Turning Center to communicate with robots and other external mechanisms such as an additional limit switch.

The inputs and outputs on the Turning Center are designated as follows:

The Robotic Interface Connector




| Pin | Function | I/O Levels | Name |
|---|---|---|---|
| 1 | Input 1 | +5V | INP 1L |
| 2 | External Limit | 0 or +5V | ELim |
| 3 | Output 2 | +1V to +3.5V | OUTPUT 2 |
| 4 | Output 1 | +1V to +3.5V | OUTPUT1 |
| 5 | Common | | GND |
| 6 | Input 2 | 0 or +5V | INPUT 2 |
| 7 | Input 1H | See Note | INPUT 1H |
| 8 | Input 2H | See Note | INPUT 2H |
| 9 | Common | | GND |

**Notes:**

Grounding pins 1 and 6 will cause the input to be true.

Inputs 1H and 2H are only used in isolated input mode, and the Controller Box must be specifically configured.

**Note:**

You can check the status of each input and output by using the Input and Output Toolbars in the Control Program.

Other Input/Output Notes:

- All User outputs are disabled by an ESTOP condition.
- Never apply a negative voltage or an AC voltage to an input.
- Outputs are set to source current only. The maximum load on each output is 1mA.
- Do not operate relays, motors or other inductive loads with the TTL I/O outputs. This will damage the machine. You may however, run an optical relay.
- The external limit signal can be used to detect motion problems and shut off the Turning Center. Use a switch that sets that signal (Pin 2) to Ground (Pins 5 or 9). Grounding this signal has the same affect as when a limit switch is tripped on the Turning Center.

## Pin Assignments for the Robotic Connector

**Note:**

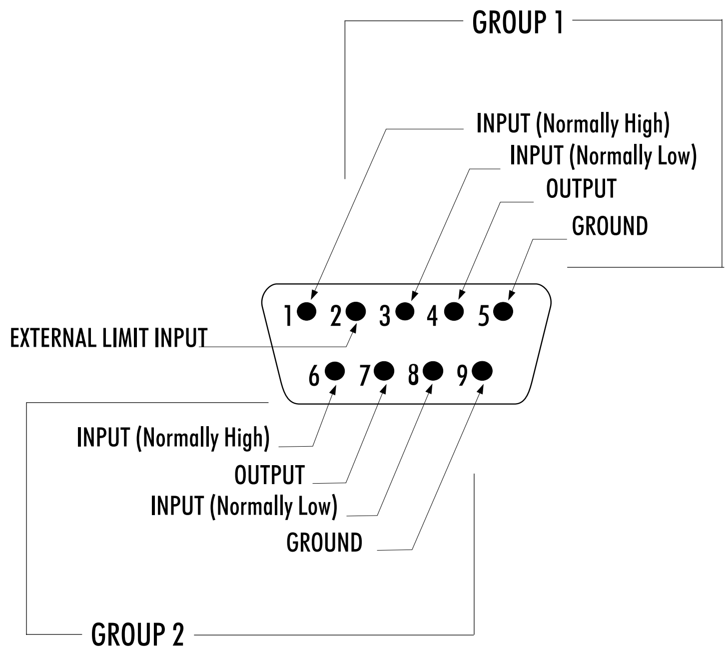
The optional Air Chuck (see Section A) includes the appropriate cable for interfacing the robotic connector with common robots.

| Pin No. | Function | Cable Color |
|---|---|---|
| 1 | Robot Input No. 1 | Purple |
| 2 | External Limit Input | White |
| 3 | Robot Output No. 2 | Yellow |
| 4 | Robot Output No. 1 | Orange |
| 5 | Ground | Blue |
| 6 | Robot Input No. 2 | Black |
| 7 | Robot Input No. 1 | Red |
| 8 | Robot Input No. 2 | Brown |
| 9 | Ground | Green |

# Multiple Robot Use

There are two groups of input and output pins (group #1 includes pins 1, 4, 5 and 7, and group #2 includes pins 3, 6, 8 and 9). Having two pin groupings allows you to interface the Turning Center with more than one robot (or other device) at a time. Each pin grouping is made up of two input pins, one output pin, and one ground pin.

In most cases, input signals coming from the robot are in a *normally-high* state (input pins 1 and 6). The robot must make the signal go low for the Turning Center to recognize a change of state. There are cases where a *normally-low* state is preferred. That is the reason for the existence of second input pins for each grouping (pins 7 and 8); they are normally-low and the robot must make them go high for the Turning Center to recognize the change.



# The NC Codes

**Special Note:**

You should not use Cycle Start when digitizing (G131).

You should use caution, if not entirely avoid using any G code which requires the activation of the Cycle Start button.

You may wish to follow a 'G26H8' block with a 'G25H8' command, so that once you press the Cycle Start button, for the G26, the G25 waits until you release the Cycle Start button before continuing.

The NC codes used in robotic communication are:

G25 - wait for High signal
G26 - wait for Low signal
M25 - transmit High signal
M26 - transmit Low signal
H# - specifies the input or output (default is H1)

The H code is used in conjunction with the wait codes and transmit codes. For example, G25H3 tells the Turning Center to wait until the state at input #3 goes high.

Assuming the robot's initial output state is low, if you place this line of code at the beginning of your program, the Turning Center waits until input #3 goes high, then executes the next line of code.

If the external device sends a low signal, nothing happens because the Turning Center is waiting for a high signal. If there's no change to a high signal, the Turning Center does not execute the next line of NC code in the program.

When the robot sends a high signal, the Turning Center responds by continuing with the NC program. If the Turning Center does not respond to the robot as you have programmed it to, check that you have correctly wired the robot to the interface, and that the robot's initial output state was not changed to high during the connection of the robot.

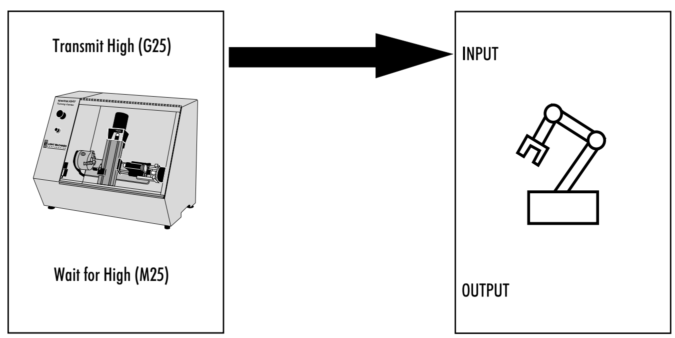
# A Sample of Lathe/Robot Communication

As previously explained, communication between the Turning Center and robot is very simple. They only use and recognize high or low signals. Once the Turning Center sends a signal to the robot, it goes into a wait state and continues to wait until the robot sends the appropriate signal back. When the signal is recognized, the Turning Center executes the next instruction in the part program. After executing the instruction, it sends a signal to the robot. The robot, which has been in a wait state since its last transmission, receives the signal, performs as it has been programmed, then again signals the Turning Center.

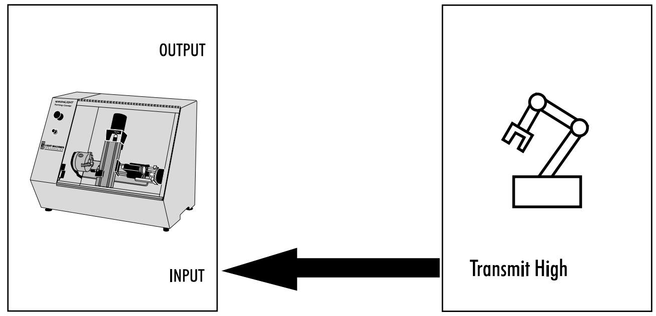
This reciprocal system of communication is made possible by the placement of G and M codes in the part program. The G codes provide the wait instructions to the inputs, while the M codes provide the transmit instructions to the outputs.

The following sequence is typical of communication between the Turning Center and a robot. Since the robot's initial output state is low, the lathe's inputs have already been pulled down to a low state. This sequence assumes the Turning Center is equipped with an air chuck.

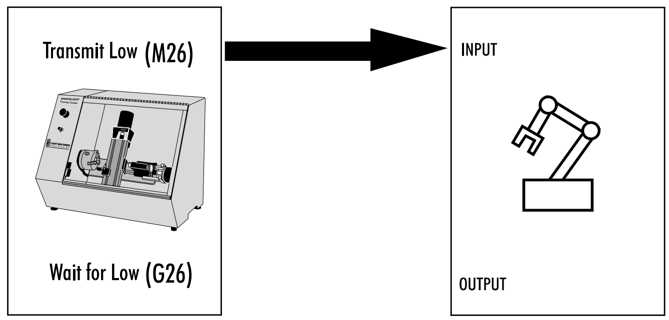
1. The Turning Center opens the air chuck (M11), transmits a high signal (M25) to the robot, and waits for a high (on) signal (G25).



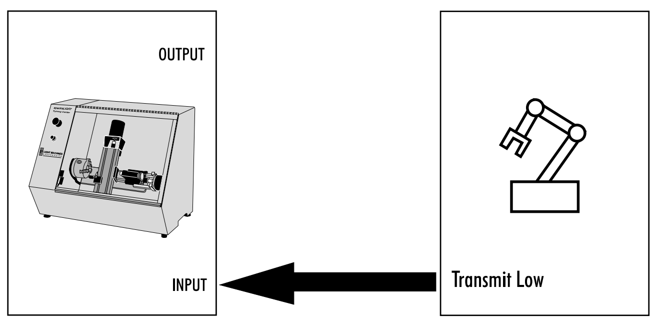
2. The robot places a work piece in the chuck and transmits a high signal to the Turning Center indicating that it is okay to close the chuck.



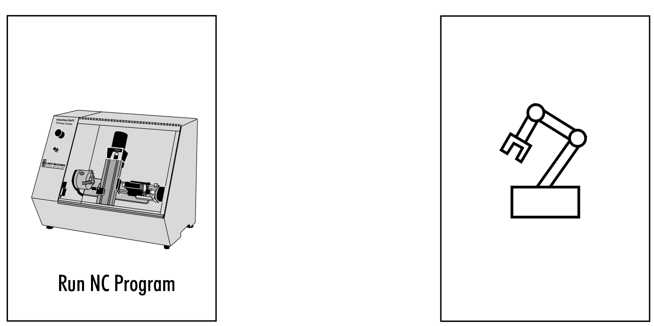

3. The Turning Center receives the signal, closes the air chuck (M10), then signals (M26) the robot that the chuck is closed. The Turning Center then waits (G26) for a low signal from the robot.



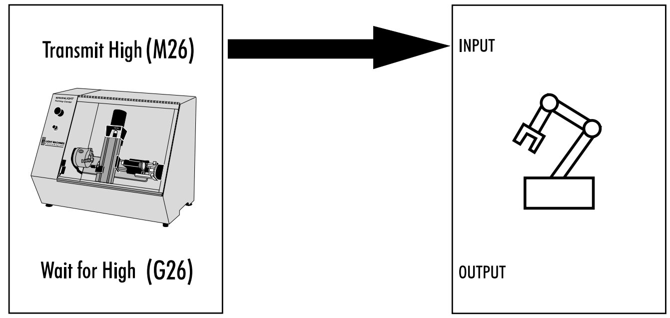

4. The robot releases the work piece and leaves the work area. It transmits a low signal when it is away from the work area.



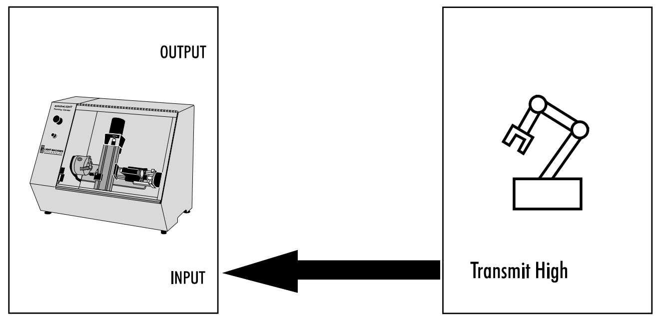

5. The Turning Center receives the low signal from the robot and begins to machining the part.



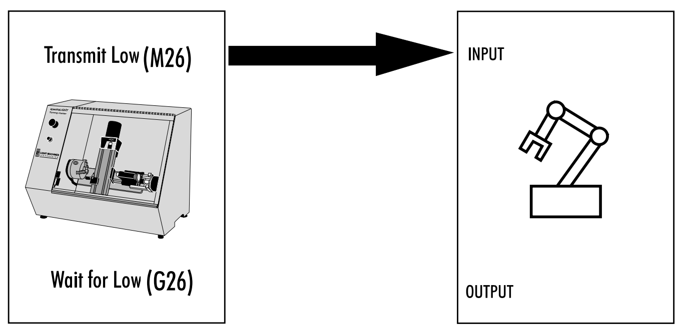
6. When the Turning Center has finished the part, it signals the robot (M25) to come take the part and waits (G25) for a high signal.



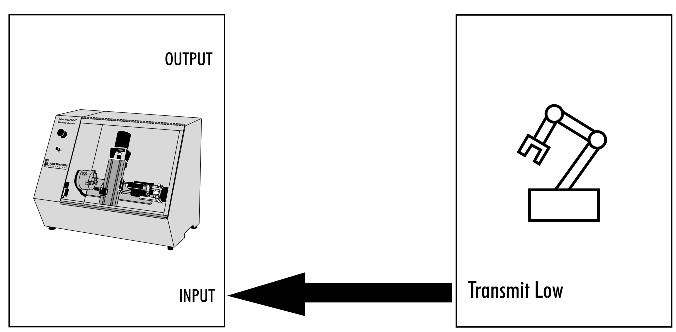

7. When the robot receives the signal, it approaches the Turning Center, grasps the part and sends the high signal to the Turning Center.




8. When the Turning Center receives the high signal, it opens the chuck (M11) to release the finished part, signals the robot (M26) and waits (G26, low).




9. The robot removes the finished part from the work area, and places the part in another area according to its own program. When the robot has cleared the lathe's work area, it sends a low signal to the Turning Center.




10. At this point, the cycle begins again (M47).

# A Sample Robotic Integration NC Program

This program illustrates one way to use G, M and H codes to interface with a robot. The codes inserted for robot interfacing are in bold.

**Robot Handshaking**

N0M11;OPEN CHUCK
N1**M25H2**;TRANSMIT HIGH SIGNAL THROUGH OUTPUT 2
N2**G25H1**;WAIT FOR HIGH SIGNAL THROUGH INPUT 1
N3**M10**;CLOSE CHUCK
N4**M26H2**;TRANSMIT LOW SIGNAL THROUGH OUTPUT 2
N5**G26H1**;WAIT FOR LOW SIGNAL THROUGH INPUT 1

**NC Program**

```
N0G0G90X.26Z1.01;MOVE TOOL AWAY FROM WORKPIECE
N1M3;TURN ON SPINDLE
N2X.25;MOVE TOOL TO START POINT
N3G1X.23F2;FIRST ROUGH CUT, SET FEED RATE
N4Z.4
N5G0X.25;BACK OUT AND RETURN
N6Z1.01
N7G1X.21;SECOND ROUGH CUT
N8Z.44
N9G0X.23;BACK OUT AND RETURN
N10Z1.01
N11G1X.19;THIRD ROUGH CUT
N12Z.48
N13G0X.21;BACK OUT AND RETURN
N14Z1.01
N15G1X.17;FOURTH ROUGH CUT
N16Z.52
N17G0X.19;BACK OUT AND RETURN
N18Z1.01
N19G1X.15;FIFTH ROUGH CUT
N20Z.66
N21G0X.17;BACK OUT AND RETURN
N22Z1.01
N23G1X.13;SIXTH ROUGH CUT
N24Z.7
N25G0X.15;BACK OUT AND RETURN
N26Z1.01
N27G1X.11;SEVENTH ROUGH CUT
N28Z.74
N29G0X.13;BACK OUT AND RETURN
N30Z1.01
N31G1X.1;FINISH CUT
N32Z.75
N33X.16Z.63
N34Z.53
N35X.25Z.35
N36G0Z1.01;RAPID TRAVERSE TO START POINT
N37M05;END OF TURNING PORTION OF CYCLE
```

**Robot Handshaking**

N38**M25H2**;TRANSMIT HIGH SIGNAL THROUGH OUTPUT 2
N39**G25H1**;WAIT FOR HIGH SIGNAL THROUGH INPUT 1
N40**M11**;OPEN CHUCK
N41**M26H2**;TRANSMIT LOW SIGNAL THROUGH OUTPUT 2
N42**G26H1**;WAIT FOR LOW SIGNAL THROUGH INPUT 1
N43M47L3;CYCLE BACK TO N0
N44M2;END OF PROGRAM

# Index

# Index

## S

## T



## U



## V



## W

## X



## Z

# spectraLIGHT Turning Center Accessory List

## Accessories

### Automatic Tool Turret

This computer-controlled 8-station tool turret (ACC-5601) provides automatic tool changing ability. The turret accommodates four 1/4" square shank tools, four 1/4" round shank tools and a parting tool.

This package also includes all items listed below in the tool Turret Tooling Package.

### Tool Turret Tooling Package

The Tool Turret Tooling Package (ACC-5605) includes:

- A profiling tool holder
- Two profiling carbide inserts (.016" and .031")
- A cut off tool (.045")
- A threading tool
- A boring tool
- A hex key

### Turning Center Fixture Kit

The fixture kit (ACC-5610) includes:

- A collet set (1/16", 1/8", 3/16", 1/4", and 5/16")
- One live center for the tailstock
- One dead center for the tailstock
- One dead center for the headstock
- One faceplate
- One lathe dog

### Tool Posts and Tools

This tool and post set (ACC-5650) includes:

- Two two-position tool posts
- One carbide tool set with a right-hand cutting tool, a left-hand cutting tool and a threading tool
- One high-speed steel tool set with a right-hand cutting tool, a left-hand cutting tool and a boring tool
- One cutoff tool holder
- One cutoff tool
- One hex key driver (5/32")

### Carbide Insert Set

This set (ACC-5630) contains three carbide inserts for use with the Carbide Indexable Tool Holder.

### Carbide Indexable Tool Holder with Insert

This set (ACC-5640) includes one carbide indexable tool holder and one carbide insert.

### Aluminum Turning Stock Package

The aluminum stock package (PKG-9320) is made from 2001 Series aluminum and includes fourteen pieces each of 1/2" x 2", 1/2" x 3", 3/4" x 2", and 3/4" x 3" stock.

### Brass Turning Stock Package

The brass stock package (PKG-9330) is made from 360 free machining brass and includes:

- Four 1/2" x 3" pieces
- Four 3/4" x 3" pieces
- Six 1/2" x 2" pieces
- Six 3/4" x 2" pieces

### Brass Turning Stock Package

The brass stock package (PKG-9140) is made from 360 free machining brass and includes:

- Thirty 3/4" x 3" pieces

### Machinable Wax Turning Stock Package

The machinable wax stock package (PKG- 9350) contains twenty pieces of blue machinable wax 1" diameter x 12" long.

### Portable Air Compressor

The portable air compressor (PNU-4535) is an intermittent duty, 3/4 hp, oil-less air compressor with a built-in color-coded regulator and pressure selection chart. It delivers 2.1 CFM at 90 psi.

### Air Chuck Robotic Interface

This pneumatic air chuck (PNU-4615) allows automatic securing of a workpiece for production applications. It is ideal for use in a computer-integrated manufacturing system (CIM), a flexible manufacturing system (FMS), or for automated factory simulation with robotics.

### Pneumatic Shield Opener

This is a computer-controlled shield opener that provides automatic opening and closing of the Turning Center shield (PNU-4625). It is ideal for use in a computer-integrated manufacturing system (CIM), a flexible manufacturing system (FMS), or for automated factory simulation with robotics.

## Curriculum/Software

### Chess Set

The Chess Set software package (DOC-7561) includes a profiling tool, a two-position tool post, software with NC programs, and documentation to machine a complete chess set. Software supplied on 3.5" disks.

### Chess Set Project

The Chess Set Project (DOC-7552) is a series of NC programs used for the manufacture of a chess set. The project consists of a floppy disk containing NC programs, a special high-speed steel profiling tool, pre-cut blank brass and aluminum stock and documentation. Software supplied on 3.5" disks. (Requires the use of a self-centering industrial grade chuck.)

### Introduction to CNC Workbook (Student Edition)

A student workbook (DOC-7631) presenting topics such as using the lathe, coordinate systems, NC programming, editing part programs, and verifying and running part programs. Includes CNC turning projects.

### Introduction to CNC Workbook (Teacher Edition)

This teacher's workbook (DOC-7641) is similar to the student workbook, but includes teacher notes and instructions to assist in the presentation of the topics. Answers to post-test reviews are provided. Includes sample programs for turning projects on 3.5" disks.

## Teklink Curriculum Series

### Computer Aided Manufacturing for Turning

The fifteen activity curriculum guide includes versions for both the students and the teachers. These guides provide instruction for using the CAM-6741 spectraCAM Turning for Windows software.

### CNC Technology for Turning

This fifteen activity curriculum guide includes versions for both the students and the teachers. These guides provide instruction for using the spectraLIGHT Turning Center with the Windows 95 control software.

## CAM Software

### spectraCAM Turning

spectraCAM is an industrial CAD/CAM package that generates part programs that can be run on the spectraLIGHT Turning Center. spectraCAM (CAM-6601) can import part geometry from CAD programs that export DXF files (such as AutoCAD).

### spectraCAM Turning for Windows

This Windows 95 version of spectraCAM (CAM-6741) is an industrial CAM package that incorporates CAD into the CAM program to provide the most comprehensive and user friendly package available. Solid model tool path verification is featured.