proLIGHT™ 2000 Machining Center User's Guide

for MS-DOS®-compatible computers

Manchester, New Hampshire, U. S. A. April 1996

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 (34-7211-0001, April 1996) corresponds to proLIGHT 2000 and proLIGHT 2500 Machining Center software version 1.4

Printed in U.S.A.

WARNING

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

Read the entire contents of this guide before running NC programs on the proLIGHT Machining Center.

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

Contents

Section 1: Introduction

Section 2: Installation

Section 3: The proLIGHT 2000 Control Program

Section 4: Tutorial

Section 5: CNC Programming Codes

Section 6: Optional Machining Capabilities

Section 7: proLIGHT Machining Center Maintenance

Appendices

Index

Accessory List

proLIGHT™ 2000 Machining Center

User's Guide

for MS-DOS®-compatible computers

34-7211-0001

Section 1: Introduction

What is the proLIGHT Machining Center?

System Requirements

Machining Center Options

Using This Guide

The Readme File

What is the proLIGHT Machining Center?

The proLIGHT Machining Center is a three-axis tabletop milling machine you can run directly from your personal computer.

The Machining Center uses your MS-DOS-compatible computer as its controller. The proLIGHT Control Program, which you load onto your computer, accepts standard EIA RS-274D, ISO, and Fanuc G&M code CNC programs.

The Machining Center can machine a large variety of parts in plastics, aluminum and steel. If used in concert with Light Machines' spectra-CAMsoftware, it's easy to machine parts directly from drawings produced with CAD programs, such as AutoCAD, that generate DXF files. Linear, circular and helical interpolation are available on all three axes.

The proLIGHT Machining Center is shipped fully assembled, and is easy to install. Complete instructions for installation of the Machining Center, as well as software installation instructions, are included in Section 2.

Features

Some of the proLIGHT Machining Center's most notable features include:

- ❏ A one-horsepower brushless DC spindle motor
- ❏ An R8 industry-standard spindle taper for the proLIGHT 2000 and an EX16 spindle taper for the proLIGHT 2500.
- ❏ Spindle speeds from 200 to 5,000 RPM (500 to 10,000 RPM optional) for the proLIGHT 2000, and from 200 to 42,000 RPM for the proLIGHT 2500.
- ❏ Rapid traverse rates up to 150 ipm
- ❏ EIA RS-274D standard G&M code programming
- ❏ Multiple tool programming
- ❏ Feed rate and spindle speed override functions
- ❏ A built-in full-screen NC program editor
- ❏ An on-line help utility

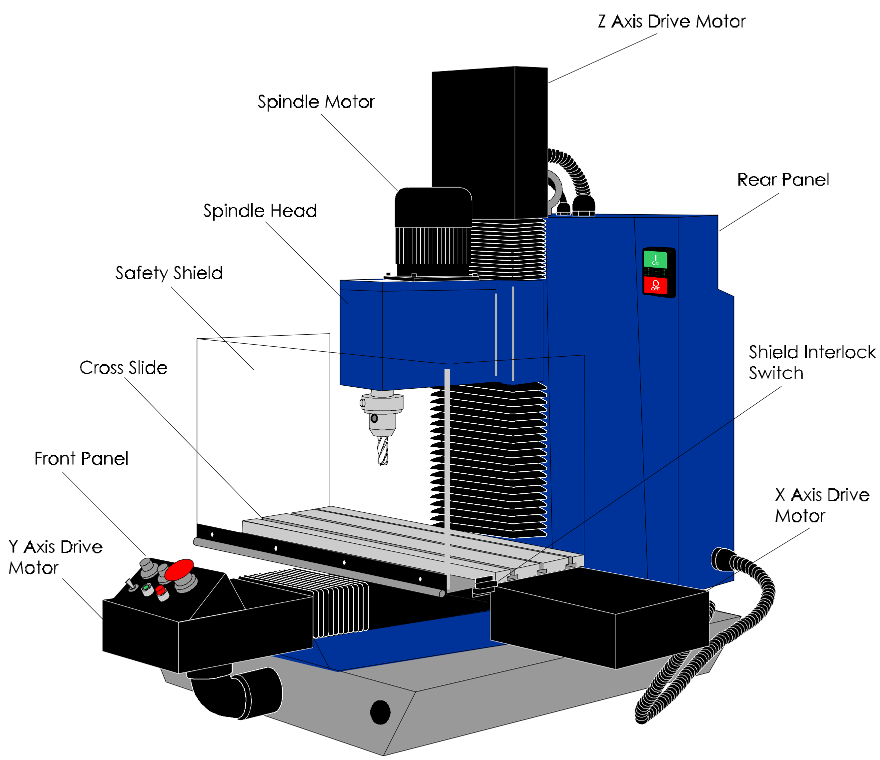
The proLIGHT Machining Center Components

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

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

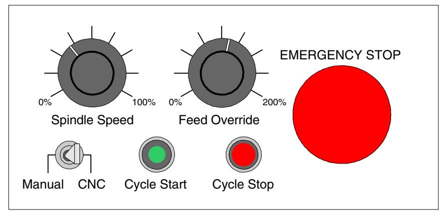
The X, Y and Z motion of the machine is performed by closed-loop DC Servo Drive Motors on each axis. There are also Limit Switches (beneath the way covers, next to the drive motor on each axis) to prevent the machine from traveling beyond its limits on each axis.

The Spindle Head supports a 1hp Spindle Motor.



The Front Panel

The Front Panel provides the operating controls shown here.



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

The **Spindle Speed** knob is used to establish the spindle speed when the system is in Manual mode. On the proLIGHT 2000, the minimum and maximum positions on the knob are equivalent to 0 RPM (min) to approximately 5,000 (max) RPM (or 10,000 RPM, depending on the option). On the proLIGHT 2500, the minimum and maximum positions on the knob are equivalent to 0 RPM (min) to approximately 42,000 RPM (max).

Select Spindle Speed mode with the **Manual/CNC** mode switch. The CNC setting on this switch gives spindle control to the computer. There must be an S code, or codes, in the NC program to regulate the spindle speed when using the CNC setting.

Use the **Feed Override** knob to override the feed rate values in the NC program.

The **Cycle Start** button makes an input on the I/O connector on the rear panel go from a normally low state to a high state. With the appropriate code in the NC program, you can use Cycle Start to perform repetitive machining tasks.

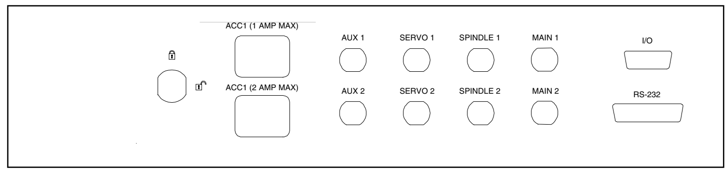
Also, whenever you are running an NC program, any time you are required to press the Enter key on the computer keyboard, you may press the Cycle Start button instead. The Cycle Start button works like the Enter key unless it is expecting an I/O signal, for instance when you use G26H8 (in) or G25H8 (out) in your NC program.

The **Cycle Stop** button works like the Emergency Stop button.

The Rear Panel

The Rear Panel houses the power and interface related controls for the machine.

Note: The ACC1 and ACC2 receptacles on the rear panel of the proLIGHT 2500 are 230 VAC.



The ACC1 and ACC2 receptacles allow you to connect options and accessories to your machine, such as the optional Air Vise Robotic Interface. Use the accessory port adapter cables (included) to connect these options and accessories.

The Key Lock switch keeps unauthorized persons from turning on the machine.

When the Key Lock is in the unlocked position, the Power switch can be turned on. The Power switch lights up when power is turned on.

Note: If circuit breakers trip before running the machine, check that the machine is wired for the correct voltage.

The machine has eight resettable circuit breakers. If a circuit breaker trips, attempt to determine the fault and correct it. After the fault has be found and corrected, wait five minutes for the machine to cool before resetting. Press to reset. Some possible causes:

> If a breaker trips at Power On, verify that the machine is connected to the correct voltage.
>
> If the Spindle breaker trips, check that the machine is not working too hard.
>
> If the ACC trips, there is an accessory that is drawing too much power.
>
> For ongoing problems with circuit breakers tripping, contact Customer Support.

The I/O connector provides the inputs and outputs required for interfacing with external devices such as robots. See Appendix F for information on robotic interfacing.

The RS-232 receptacle is used to interface the machine controller electronics with your computer. A cable is provided in the Accessory Kit for this interface. Connect the cable to the RS-232 receptacle and the mating serial port connector on the computer only! Refer to Section 2 for correct installation procedures.

The Control Program Software

The heart of the proLIGHT Machining Center is the Control Program software that is run by the computer. Using industry standard *EIA RS-274D* NC codes, the Control Program provides for three-axis CNC programming and milling.

The menu-driven Control Program prompts you for the data required to run a part program. The Control Program offers features such as a Help utility, program editing, and manual machine control.

Section 3 of this guide provides instructions on how to select commands from the Control Program menus. Some menus display other options (dialog boxes or other menus) from which you can initiate operations or enter data. Messages are displayed as necessary to prompt you on how to proceed or to inform you of errors.

The Accessory Kit

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

System Requirements

The proLIGHT Control Program runs on MS-DOS-compatible computers. The computer must have:

- ❏ MS-DOS version 3.1 or higher.
- ❏ At least 640KB RAM.
- ❏ A floppy drive.
- ❏ A hard drive. There must be at least 600KB of available space on the hard drive to install the Control Program software.
- ❏ A CGA, EGA, or VGA color monitor is recommended, but you can also use a Hercules-compatible monitor.
- ❏ A Microsoft-compatible mouse.

You may wish to consider the following optional components for your computer.

- ❏ A math coprocessor is highly recommended to maximize the Control Program's computation speed.
- ❏ A printer is useful for producing hard copy records of part programs.

Machining Center Options

Note: The following options are not available for the proLIGHT 2500:

- Machining Center Machinist Kit (ACC-5110)
- Quick-Change Tooling (ACC-5141)
- Digitizing Package (ACC-5261)
- Vacuum System (ACC-5730)

The following options are available for the proLIGHT Machining Center from Light Machines Corporation.

❑ **Machining Center Machinist Kit (ACC-5110)** The Machinist Kit includes a 3" milling vise with hold-down clamps, a 4-piece R8 collet set, a 9-piece high-speed end mill set (including 2 ball end mills), a 52-piece hold-down set, a Jacob's chuck, an R8 to 2JT arbor, and a 3/8" x 12" cushion-grip T-handle hex key. *Not available for the proLIGHT 2500.*

❑ **End Mill Package (ACC-5120)** The ACC-5120 End Mill Package consists of the following end mills, all with 3/8" shanks: two 1/8" HSS center cutting 4 flute end mills, two 3/16" HSS center cutting 4 flute end mills, two 1/4" HSS center cutting 4 flute end mills, two 5/16" HSS center cutting 4 flute end mills, two 3/8" HSS center cutting 4 flute end mills, two 1/8" HSS ball 2 flute end mills, and two1/4" HSS ball 2 flute end mills

❑ **Quick-Change Tooling (ACC-5141)** Quick-Change Tooling provides an easy way of integrating multiple tools within one NC program. It comes with the tool body, five tool holders, a tool holder stand, three allen keys and an electronic tool height offset sensor. The Quick-Change Tooling Kit also has optional positive drive tool holders. *Not available for the proLIGHT 2500.*

❑ **Low Profile Clamping Kit (ACC-5180)** This kit includes a custom rail and stop with four clamps for work holding.

❑ **4th Axis 5C Rotary Positioner (ACC-5222)** The ACC-5222 4th Axis 5C Rotary Positioner is a complete rotary positioning device that is mounted vertically on the proLIGHT 2000 Machining Center and provides simultaneous 4-axis machining capability. The rotary positioner incorporates a 5C collet system with manual collet closer. When installed: Available X axis travel is 7", Z axis travel is 5-3/4".

- **Digitizing Package (ACC-5261)** The ACC-5261 Digitizing Package includes a digitizing probe and software that lets you digitize existing parts and models to capture 3-D surface data. The easy-to-use PC-driven package provides reverse engineering capabilities with your proLIGHT series machining center. It can generate files in three different formats; an NC part program, an ASCII text file with XYZ data points, and a DXF 3-D mesh file.

  The Digitizing Package includes a touch signal probe, a probe interface cable with 15 pin and 9 pin adapters, a 20 mm long stylus with 3 mm diameter head, a 20 mm stylus extension, the Digitizing Package User's Guide, the Digitizing Package software on 3-1/2" and 5-1/4" diskettes, and an 8 mm R8 Collet. *Not available for the proLIGHT 2500.*

- **Vacuum System (ACC-5730)** The ACC-5730 Vacuum System is designed for use with the proLIGHT Series Machining Centers. It includes a vacuum with noise reduction features, a GFT hose, a 1-1/2" diameter nozzle mounting assembly, and an electrical relay. *Not available for the proLIGHT 2500.*

- **spectraCAM (CAM-6601)** spectraCAM is Light Machines' CAD/CAM package. The spectraCAM Milling software generates G&M code NC programs that can be used on the proLIGHT Machining Center. spectraCAM Milling can import part geometries from CAD programs, such as AutoSketch, that export DXF files.

- **CAD Engraver Program (DOC-7671)** The CAD Engraver is a 2-1/2-axis CNC milling program for machining text and CAD drawings.

- **Air Vise (PNU-4115)** The Air Vise has jaws that open to a maximum of 3.0 inches. The vise comes with a solenoid valve and all the tubing required to interface with 1/4" pipe fittings from filtered and regulated shop air (50-125 psi).

Using This Guide

In addition to this introductory section, this guide is divided into seven sections plus appendices.

Section 2
Follow the procedures in Section 2 to assemble and connect the Machining Center, to install the Control Program on your computer, and to install accessories.

Section 3
Section 3 is an in-depth reference that explains Control Program menus, commands, dialog boxes and other screens.

Section 4
Section 4 is a tutorial that takes you step-by-step through a sample NC program that introduces you to menu functions and Machining Center operation.

Section 5
The proLIGHT Machining Center supports standard EIA RS-274D G&M NC programming codes, along with an array of other codes. All of the supported codes are reviewed in Section 5.

Section 6
Section 6 explains two of the proLIGHT Machining Center's more popular optional machining capabilities: multiple tool programming and machining with the 4th Axis Rotary Positioner. If you plan to use either of these options you should read this section.

Section 7
Good maintenance practices assure a longer life for the proLIGHT Machining Center. Section 7 tells you how to care for the Machining Center and Controller Box, and provides some valuable information on using a computer in a shop environment.

Appendices
The appendices provide background information on troubleshooting, machining practices, safe machining operation, G and M codes, working with DOS, and robotic integration.

The Readme File

There is a README.NC file on the proLIGHT master disk. The Readme file contains information that may not be covered in this guide. The Readme file has been created in NC file form so you can read it by loading it into the proLIGHT Control Program. You can also read it using a standard text editor, or by using the **type** command in DOS.

To view the Readme file using the Control Program, run the Control Program, then use the **Open** command to load the README.NC file.

Section 2: Installation

Getting Ready

Hardware Installation

Software Installation

Getting Ready

Check Your Shipment

The first thing you should do after receiving your proLIGHT Machining Center is locate the packing slip in the Accessory Kit. This slip lists all of the items you should have received with your Machining Center. Check all of the items on the list.

Inspect the Machining Center for signs of visible damage. If you notice any damage, or if you find any discrepancies between the packing slips and the items you have received, call Light Machines' Customer Service Department (800/221/2763) or 603-625-8600.

Register Your Machining Center

You'll find a registration card in the small box with the documentation and software disks. It's important that you clearly print all the requested information and return this card to Light Machines as soon as possible.

Prepare Your Work Place

Finally, you have to make sure you have all the items on hand that you'll need to perform the installation. To install the proLIGHT Machining Center, you must have ready:

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

- ❏ A sturdy table on which you'll place the Machining Center and your computer. Placing the table against a wall provides more stability. For the proLIGHT 2000, make sure the wall has a 120VAC, 20-amp polarized outlet. International machines and the proLIGHT 2500 require a 230VAC, 10-amp polarized outlet.
- ❏ An MS-DOS-compatible computer with a minimum of: 640KB of RAM; one 1.2MB, 5.25" floppy disk drive or one 720KB, 3.5" floppy disk drive; a CGA, EGA, VGA or Hercules-compatible graphics card; a video monitor; and a keyboard.
- ❏ Your computer owner's manual or equivalent documentation.

Unpack the Machining Center

IMPORTANT: Be sure to keep the pallet and all of the original cartons in which the proLIGHT Machining Center was shipped. Should any components need to be returned to the factory, re-pack them exactly as they were received.

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

1. Position the pallet near the table on which you'll set the Machining Center. The table should be located against a wall for maximum support.
2. Remove the staples that attach the bottom of the cardboard container to the pallet.
3. Lift the cardboard container off the pallet.
4. Inspect the Machining Center chassis for signs of visible 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 slip and the items received, call Light Machines' Customer Service Department (800/221/2763 or 603-625-8600).
5. Remove the four bolts holding the Machining Center base to the pallet. Keep the bolts and other packaging materials.

Set Up the Machining Center

CAUTION:

The proLIGHT Machining Center weighs approximately 355 pounds, so be very careful when lifting it.

1. Lift the Machining Center off of the pallet and onto the table.
2. Once on the table, position the Machining Center correctly for safe, convenient machining.
3. Remove the protective paper from the safety shield.

Hardware Installation

The following paragraphs review the procedures for interconnecting your computer with the proLIGHT Machining Center. You should have already set up your personal computer in accordance with the directions in the owner's guide that came with your computer.

The diagram on the following page has been provided as a visual aid for the recommended interconnection.

WARNING

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

Positioning the Machining Center and Computer

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

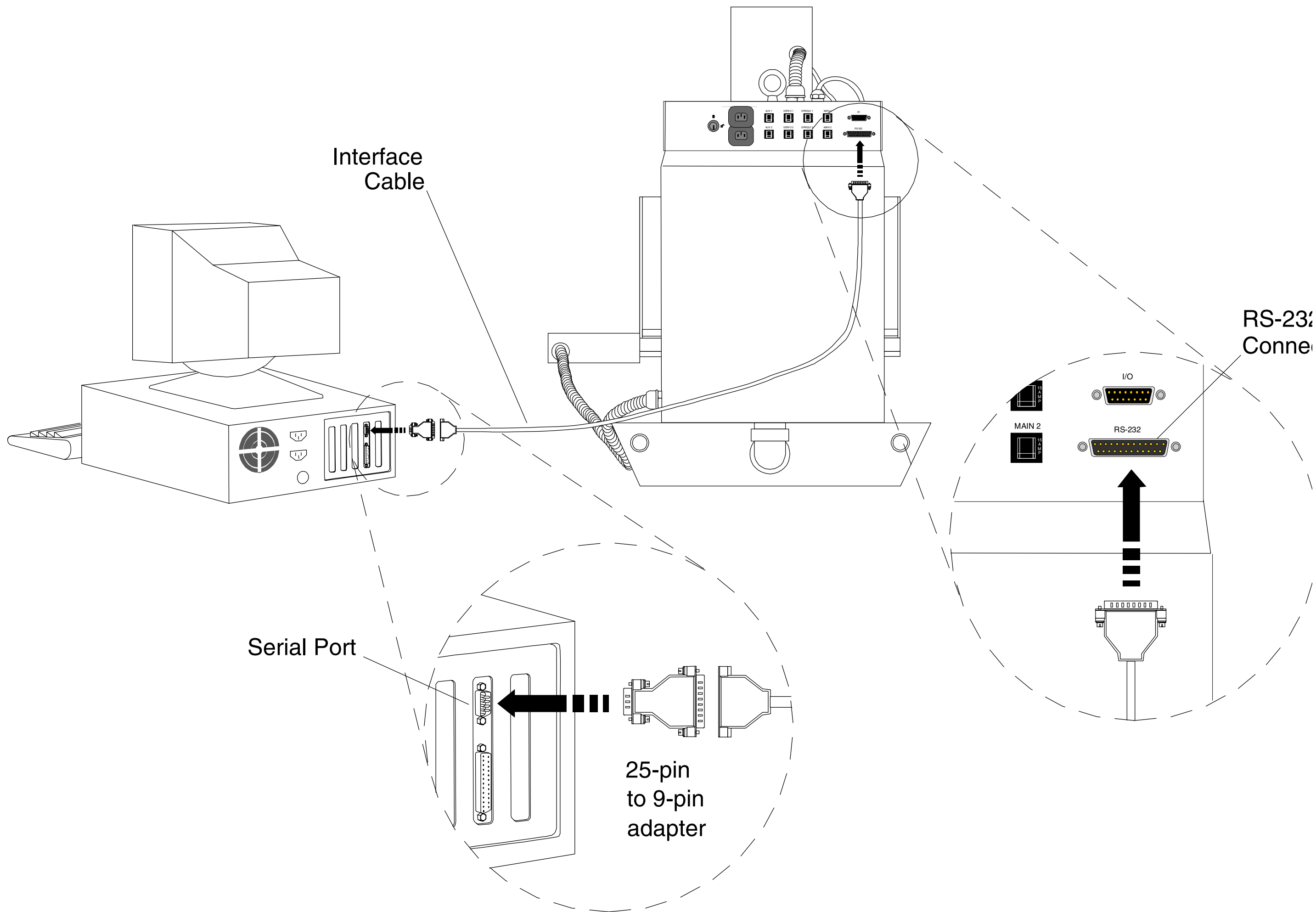
Keep in mind that when troubleshooting, you may need to check the circuit breakers that are mounted on the rear panel of the Machining Center. Place the computer in an area where it will not be exposed to metal chips or cutting fluid.

WARNING

Never connect or disconnect cables with power on!

This will cause damage to the internal controller drive components. Always operate the Machining Center with all cables firmly secured.

Rear View



proLIGHT Machining Center Interconnection with Computer

CAUTION:

Do not allow the AC power cords to cross the interface cable. The power cords can create noise problems with the signal lines in the interface cable.

Connecting the Computer to the Machining Center

The computer interfaces with the Machining Center by way of a single interface cable. The interface cable is terminated with a 25-pin connector at each end. Connect one end of the cable to the 25-pin receptacle on the rear panel of the Machining Center. Connect the other end to the 25-pin serial port connector on the back of your computer. If your computer serial port has a 9-pin connector, use the special 25-to-9-pin adapter from the Accessory Kit (see the illustration on the previous page).

Important! You must instruct the Control Program to communicate with serial ports other than COM1. This requires setting an environment variable or running the SETUP program from the PLM2000 prompt.

Note: You have to remove the protective cap from the plug end of the accessory port adapter cable before inserting it into the machine.

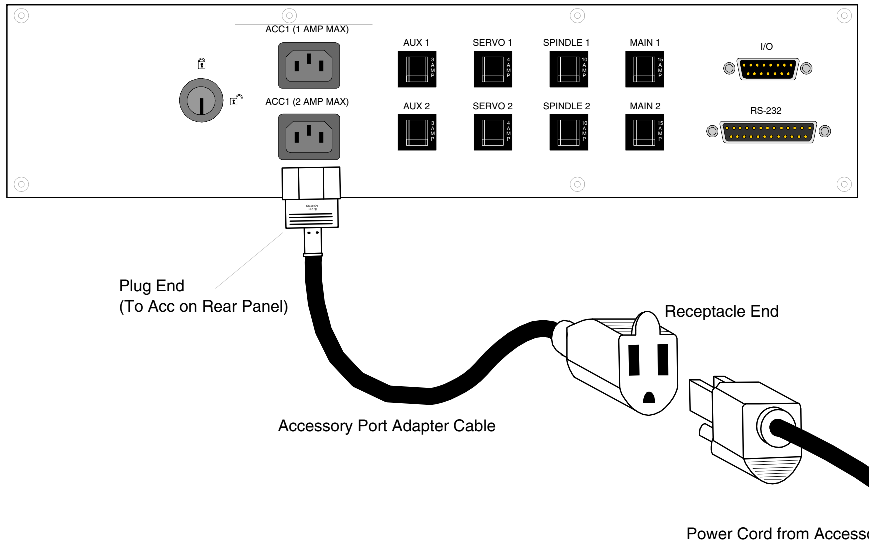
Optional Connections

If you have purchased the optional air vise, plug the power cord from the solenoid valve on the vise into the receptacle end of the Accessory Port Adapter Cable. Insert the plug end of the cable into one of the ACC receptacles on the rear panel of the Machining Center.

You can attach another 120VAC accessory to your Machining Center. 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 other receptacle on the rear panel of the Machining Center; the current draw for such accessories, however, is limited to 3 amps on 120VAC machines (1.5 amps on 230VAC proLIGHT 2500 and international machines).

A 15-pin connector is provided on the rear panel for interfacing to an I/O device such as a robot. See Appendix F of this guide for information on interfacing with robots.

Connecting an accessory to the Controller Panel using the Accessory Port Adapter Cable. Note that the accessory receptacles on proLIGHT 2500's are 230 VAC.



A Note to International Users:

You need to install the plug on the end of the electrical cord to accommodate 230V, 50Hz, 10A service, the wiring is color coded as:
Line - black
Neutral - white
Ground - green

If your power outlet only provides two lines, Line and Neutral, you **must** provide a separate Ground!

Connecting Power to the Components

Plug both the proLIGHT 2000 and the computer into 120VAC, 60Hz wall outlets. Plug the proLIGHT 2500 into a 230VAC, 50-60Hz, 10A, single phase wall outlet. The Machining Center has an attached power cord terminated with the appropriate plug. Insert the plug end of this cord into the correct wall outlet.

Software Installation

After turning on your computer, you will most likely see the DOS system prompt. The prompt often appears as C:\>.

C: is the hard drive designation.

\ indicates that you are at the root directory (the top level) of the system.

> with a blinking cursor indicates the computer is awaiting input.

The proLIGHT Control Program is shipped on both a 5.25" 1.2MB disk and a 3.5" 720KB disk. The Control Program must be installed on a hard drive containing a copy of DOS (version 3.1 or higher). You must have at least 600KB of free space on your hard drive to perform this installation. (See Section 1 for system requirements.)

CAUTION: The proLIGHT master disk is shipped write-protected (5.25 disks have a tab over the notch in the sleeve, and 3.5 disks have the write-protect window open) to prevent accidental destruction of the software. *Never* remove the write protection! Create and use a working copy of the software. Always store the master disk 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 Machining Center for the first time.

Installing the Control Program

The proLIGHT Control Program must be copied to the hard drive on your computer. 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 the system prompt (C:\>) to appear.
2. Insert the proLIGHT disk in the floppy drive of your computer.
3. Type A:INSTALL then press Return. (You may have to type B:INSTALL if your computer has multiple floppy drives.)

   This command creates a new directory called PLM2000 on your hard disk. It then copies all the files from the master disk in the floppy drive to the PLM2000 directory on the hard drive. When the copying is completed, the computer prompts you to press Return to run the SETUP program.

```
a:\millone.mc => c:\plm2000\millone.mc
a:\milltwo.mc => c:\plm2000\milltwo.mc
a:\rotary.mc => c:\plm2000\rotary.mc
a:\setup.com => c:\plm2000\setup.com
a:\setup.map => c:\plm2000\setup.map
a:\setup.plm => c:\plm2000\setup.plm
a:\accel.map => c:\plm2000\accel.map
a:\spindle.map => c:\plm2000\spindle.map
a:\setup.pl4 => c:\plm2000\setup.pl4
    25 file(s) copied
Press RETURN to run the setup program
```

4. Press Return to run the SETUP program. (See the following page for instructions on running the SETUP program.)

Running the SETUP Program

The SETUP program automatically configures the control software for your machining center's specifications. When you press Return after running the INSTALL program, the computer begins the SETUP program.

Setup for the proLIGHT 2500

1. Enter Y for yes if you have purchased a proLIGHT 2500 Machining Center with a 42,000 RPM Spindle.

2. Enter 2 if your machine is connected to serial port #2 on your computer. The default is 1. After you answer this question, the computer completes the SETUP process.

```
Setup for PLM-2000 software

Do you have the proLIGHT 2500 with 42000 RPM spindle (Y or Default=N)?:y

Do you have the machine hooked up to serial port #1 or serial port #2?
(if you have a mouse it may be hooked up to port#1)  (2 or default=1):1

Processing...
Finished.

C:\PLM2000>
```

Setup for the proLIGHT 2000 with the 10,000 RPM Spindle Option

1. Enter Y for yes if you have purchased a machine with a 10,000 RPM Spindle.

2. Enter 2 if your machine is connected to serial port #2 on your computer. The default is 1.

3. Enter 3 if your machine's serial number begins with three digits followed by a space. Enter 4 if your machine's serial number begins with four digits followed by as space. If you are unsure of this information, look at the serial number on your machine.

   If you answer 3 to this question, the computer completes the SETUP process. If you answer 4 to this question, the computer prompts you with two further questions.

4. Enter Y for yes if you have a machine with 500 line encoders. If you are unsure of this information, look at the serial number on your machine. To answer yes to this question, the last five digits of your serial number should be less than 20053.
5. Enter Y for yes if you have a revision 1 spindle speed board. If you are unsure of this information, look at the serial number on your machine. To answer yes to this question, the last five digits of your serial number should be less than 20066. After you answer this question, the computer completes the SETUP process.

```
Setup for PLM-2000 software

Do you have the proLIGHT 2500 with 42000 RPM spindle (Y or Default=N)?:

Do you have the 10000 RPM spindle option (Y or Default=N)?:y

Do you have the machine hooked up to serial port #1 or serial port #2?
(if you have a mouse it may be hooked up to port#1)  (2 or default=1):1

Does your machine serial number start with 3 digits followed by a space
or 4 digits followed by a space?(4 or Default=3):4

Do you have a machine with 500 line encoders (The last five numeric
 digits of your serial number should be less than 20053. Y or Default=N):N

Do you have a revision 1 spindle speed board (The last five numeric
 digits of your serial number should be less than 20066. Y or Default=N):N

Processing...
Finished.

C:\PLM2000>
```

Setup for the proLIGHT 2000 without the 10,000 RPM Spindle Option

1. Enter N for no (or press Enter to select the default: no) if your machine is not equipped with the 10,000 RPM spindle option.
2. Follow steps 3 through 6 above. After you answer these questions, the computer completes the SETUP process.

Section 3: The proLIGHT 2000 Control Program

Starting the proLIGHT System

Note: Do not attempt to operate the proLIGHT 2000 Machining Center without reading and practicing the safety instructions contained in this Section and Appendix C.

After you have set up your machine and have copied the Control Program to the hard drive of your computer, you can start the proLIGHT System.

The procedure for starting the proLIGHT System is:

1. Turn the computer on. After the C:\> prompt appears, you can turn on the machining center.
2. At the C:> prompt, change from the main directory to the directory containing the proLIGHT files. Type: CD\PLM2000 and press the Enter (or Return) key.

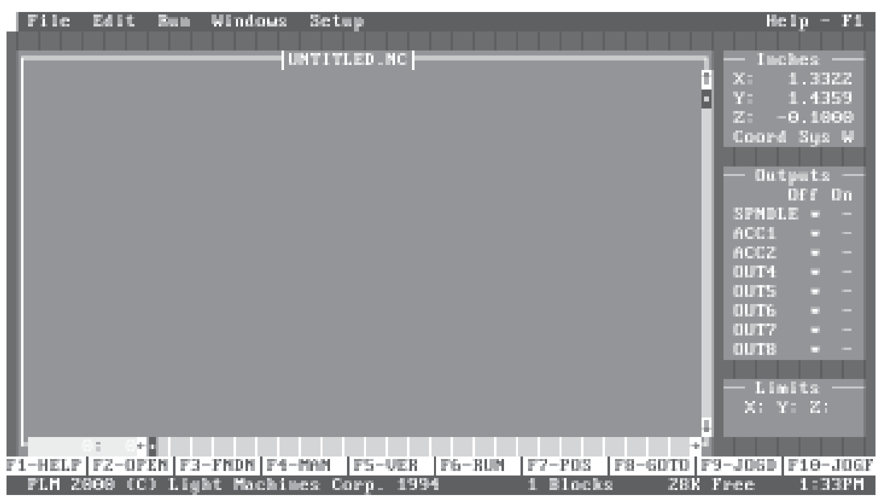
   (If you created your own directory name when you installed the proLIGHT Control Program, type CD\ then enter your directory name and press Enter.)
3. When you're in the proper directory, type PLM2000 at the C:> prompt and press Return. When the proLIGHT Control Program is running, the main screen appears.

Note to 4th Axis Rotary Positioner users:

The start up procedure is slightly different for starting the Control Program when using a Rotary Positioner.

At the C:> prompt, type CD\PLM2000 and press Enter. When you are in the correct directory, type PLM2000 /4.

See Section 6 for further details.

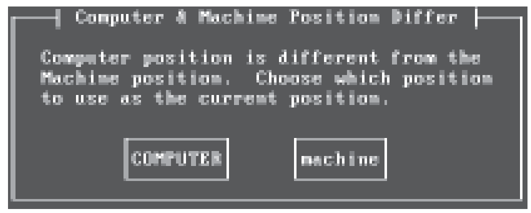


This is how the main screen looks when you first start up the Control Program.

Having a Problem?

Computer and Machine Position Differ

Your computer's position and the Machining Center's position may differ if you home the machine then exit the Control Program without saving parameters. When you restart the Control Program, the following screen appears:



Select Computer to use the last saved computer position as the current position. Select Machine to use the machine's current position as the current position.

Error Initializing Control

If the following screen appears when you start up the Control Program, check your installation connections and make sure the Machining Center is turned on.

After checking the connections, you can select Retry to start the program. Otherwise, you can select Simulate to use the Control program without communicating with the Machining Center, or select Quit to exit the Control Program and return to DOS.

If you are experiencing other start-up problems, check the items listed below.

- Are the computer and monitor plugged into a wall outlet? Is there power to the wall outlet?
- Is the monitor correctly connected to the computer?
- Are all interface connections secure?
- Are the monitor and computer turned on?
- Have you specified the correct directory?
- If you are using a mouse, was the mouse driver software loaded before the proLIGHT Control Program software?
- Do you have enough memory on the computer to run the Control Program? Make sure there are no other programs running before starting up the Control Program.

If you still experience start up problems after checking the above items, refer to your *PC Owner's Manual* and the *DOS Reference Guide* for additional information on loading software.

Selecting Commands

The proLIGHT Control Program has pull-down menus like many other MS-DOS-based applications. Selecting a menu item may execute a command or produce a *dialog box* on the screen.

A dialog box may require an ok or cancel, or a yes or no response from you. You also may be required to enter numerical data or accept current values before selecting ok or cancel. You always have the option of canceling a function and exiting the dialog box without changing anything.

Selecting Menu Options

Note: Pressing the ESC key allows you to exit any menu or dialog box.

You may select desired functions from a menu using any of the following techniques, or a combination of these techniques.

Menu Titles

You can click on the titles at the top of the screen (File, Edit, Run, Windows, Setup and Help) to pull down the menus. You can also access the menus by pressing the Alt key on your keyboard, then pressing the key that corresponds with the first letter of the menu title. For instance, to open the File Menu, press Alt+F.

Menu Commands

Once you've pulled-down a menu, there are two ways to select a command. When using a mouse, you can execute any command on a menu simply by clicking on it. You can also press the key on your keyboard that corresponds to the highlighted character in the command. For instance, to execute the Save **A**s command on the File Menu, press the A-key.

Hot Keys and F Keys

You can use a series of keyboard key strokes and the keyboard function (F) keys to select certain menu functions. For example, press Ctrl + S to save your current NC program. Press the F2 key to bring up the Open NC File dialog box. Menu options with Hot Key combinations are displayed on the menu with the combination. The carat character (^) represents the Ctrl key on the keyboard. The F key functions appear along the bottom of the program screen.

Selecting with the Arrow Keys

You can use the arrow keys on the computer keyboard to highlight a menu item or an item on a list in a dialog box. Once the item is highlighted, press the Enter key to complete the selection.

Selecting with Tab and Shift+Tab

Note: There are a few dialog boxes that do not allow you to tab through buttons. These dialog boxes contain requirements related to safety. E.g., the Continue/Quit dialog box that appears while running a program.

To move from one data field to another within a dialog box, you may press the Tab key to move forward to the next field, or press the Shift and Tab keys simultaneously to move back to the previous field. Once a field is highlighted, enter data by typing. Press the Enter key when you are finished entering data, or use the Tab or Shift+Tab keys to move the cursor to another field.

If the dialog box contains push-buttons, such as ok, cancel, yes or no, the selected push-button will change case when selected. For example, when *not* selected, the ok button is displayed in lower-case letters. When it *is* selected, the OK button is displayed in upper-case, or capital letters.

ok	OK
Not Selected	*Selected*

Selecting with a Mouse

Note: Refer to the User's Guide supplied with your mouse for additional information on using the mouse.

The proLIGHT Control Program is designed to run on a personal computer equipped with a Microsoft-compatible mouse driver. When the mouse driver is loaded, the cursor becomes a block-shaped pointer. Move the pointer on the screen by moving the mouse over your desktop.

Important: You must load the mouse driver software **before** loading the proLIGHT Control Program software.

With the cursor positioned over the area of the screen you wish to select, press and release the left button on the mouse. This action is called *clicking*. Clicking selects an item or field on the screen. You can also click on buttons (like ok and cancel) to press them. To enter data in data fields, click on the field, type in the data and click on ok. You can use the mouse to select a menu, a menu command, a data field, or an area of the screen.

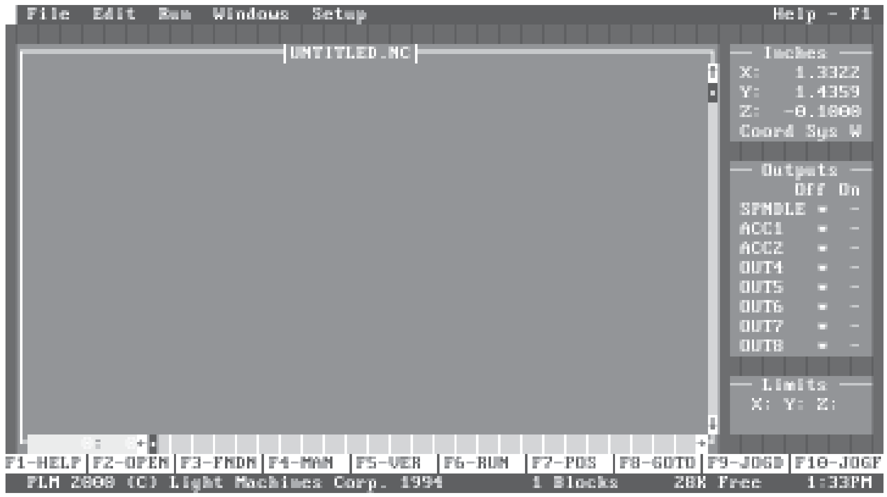
You may click twice (*double-clicking*) to select some items, for instance the drives, directories or files in the Load NC File dialog box.

The Control Program Screen

The screen that appears when you run the proLIGHT Control Program is dependent on the Startup Mode you select (see the Set Preferences Command later in this section). Here we show the screen in the default Editor mode.

This is the first screen you'll see when you start the Control Program. It is constructed of a series of separate "windows" that perform individual functions.

The Inputs Window (not shown) appears only while the Manual Control Panel and Run Program Windows are active.



The Edit Window

The Edit Window is where you enter your NC code. When you open an existing NC program, it also appears in the Edit window. When you are working with large programs, you can scroll through the Edit Window by clicking on the scroll bars along the right and bottom edges of the Edit Window.

The numeric display in the lower left corner of the Edit Window indicates the number of the block the cursor is on, and the number of the character in the block. For instance, if the display reads 20:3, the cursor is on the third character in the 20th block of the program.

Once you have opened a program, you can use both the mouse and the keyboard to edit the NC code.

Always Save NC programs after editing to make sure your changes are not lost before turning off power to the PC. Periodic saving can also preserve the work you've done should the power to your PC be inadvertently lost.

The Position Window

The Position Window displays the currently selected programming mode (inch or metric) and the current X, Y and Z position of the cutting tool.

If you change the coordinate system using the Coordinate Systems command from the Setup menu, the currently selected coordinate system is also displayed. The default is Coordinate System W (which stands for *world*, meaning no offsets are being used). The coordinate system is always W, except when running an NC1 program using G54 through G59.

The Outputs Window

The Outputs Window contains on/off switches for enabling or disabling the eight system outputs. An output is on if an asterisk appears in its "on" column.

You can control the outputs as long as the Manual Control Panel is active. To turn an output on or off, click on the On or Off column with the mouse. You can also use a Shift+F-Key combination. For instance, press Shift+F1 to toggle output #1 (Spindle) on and off.

The first three outputs have dedicated assignments to the spindle, ACC1 and ACC2. The remaining outputs correspond to certain outputs on the TTL I/O connector on the rear panel of the Machining Center (see Robotic Integration, Appendix F).

The Limits Window

The Limits Window alerts you when the machine has reached its maximum positive or negative limit of travel. When a limit is hit, the window lights up and displays the word LIMIT. Also, a plus (+) or minus (-) is displayed next to the offending axis. For instance, if the Limits Window changes color and there is a small white plus sign next to the X, it means that the positive X axis limit has been hit.

The Limits Window also alerts you to other causes of program interruption. For instance, it changes color and displays ESTOP if you press the Emergency Stop or Cycle Stop buttons, and displays Door Open messages.

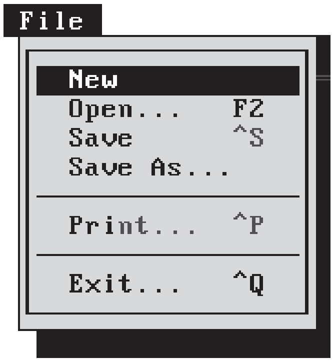
The Inputs Window

The Inputs Window appears only when the Manual Control Panel is open, or when you are running an NC program. This window displays the status of the inputs on the TTL I/O connector on the rear panel of the Machining Center.

All inputs are normally low (at a low voltage state). The inputs will go high under certain circumstances. For instance, the Cycle Start button on the front panel of the Machining Center will cause Input 8 to go into a high state. This can be used in conjunction with certain G codes for robotic interfacing and for performing repetitive part machining. For more information on the use of the inputs, see Appendix F.

The Menu Bar and Message Bar

The menu bar at the top of the screen provides the pull-down menus, and the message bar at the bottom of the screen provides information on the number of blocks in the open program, the amount of remaining memory on your computer, and the time.



The File Menu

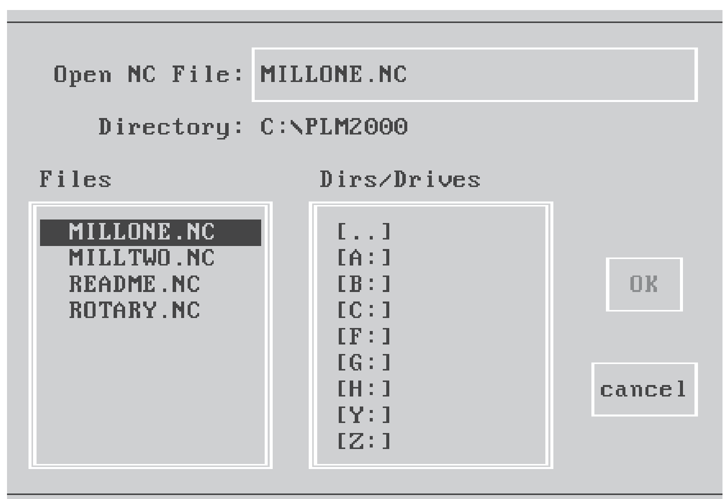
The File Menu provides general file handling commands.

The New Command

You can use existing NC programs, or you can create new NC programs by typing the code in the Edit window. To create a new NC program, select **New** from the File Menu. You may be prompted to save changes to any currently open NC program. Once the current program is saved, a new "Untitled.NC" window appears and you can begin typing NC code.

The Open Command

When you select **Open** from the File Menu, the Open NC File dialog box appears.



Note: If you are unable to open an NC file because it is too large, use SPLIT.EXE to divide the program into smaller units.

See "Splitting NC Files" in Appendix E for more information.

The name of the last program loaded into RAM is the default filename in the Open NC File: box. If you type in a different filename, remember, you must follow DOS conventions for identifying the full DOS pathname: [drive]:\[directory]\filename]

For example:

- B:CASTLE.NC (an NC program file in drive B named "castle")
- C:\PROGRAMS\CASTLE.NC (an NC program file named "castle" in the PROGRAMS directory of drive C)

The default drive is the drive from which the proLIGHT Control Program was started. The default directory is the PLM2000 directory.

You may open directories or select a file in the window using a mouse, or using the Arrow keys on the keyboard; the selected filename appears in the Open NC File: box. Use the Tab key to move the cursor through each window and button.

If there are more files or directories than the window can show, a scroll bar will appear on the right side of the window. To view additional files or directories, click on the scroll bar.

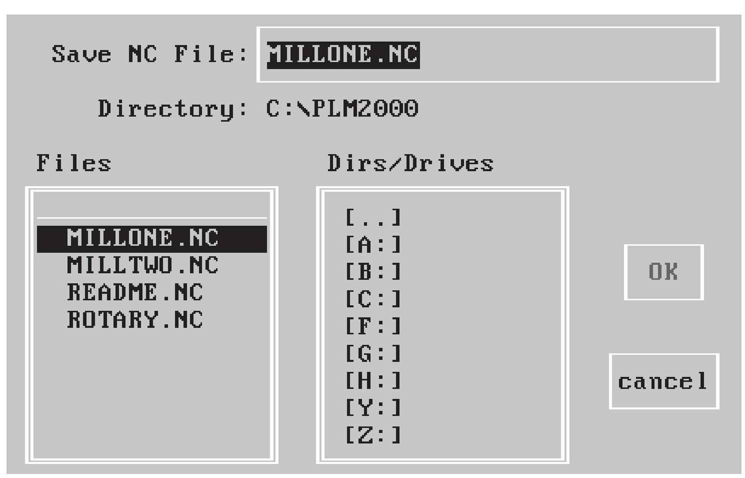
To view the contents of the parent directory to the directory you are currently in, select the dual periods (..).

To open the program file shown in the Open NC File: box, select ok. (You can also use the mouse to double-click on the filename in the file window.)

Select cancel to exit the Open NC file dialog box.

The Save and Save As Commands

The Save command allows you to save the latest changes to your NC program under its current filename. If you have written a new program and it is still "untitled," only the Save As command is available from the File Menu. The Save command is not available until changes have been made to the file.



IMPORTANT:

Save your NC program often as you work, especially if it's a long program. Until a program has been saved, it could be lost due to a power interruption or a software problem. Always save your program before running it on the Machining Center.

The name of the last program loaded is the default filename in the Save As File: box. If you type in a different filename, remember, you must follow DOS conventions for identifying the full DOS pathname.

Saving an NC program under its original filename over-writes previous copies of the file with any changes that have been made. You can save several versions of a single part program by saving the program under different filenames.

If you want to save your program under another name, or save it to another directory or drive, select the Save As command. Enter a new DOS pathname for the program before selecting ok. If you do not add the .NC file extension, the Control Program automatically adds it when you select ok.

The default drive is the drive from which the proLIGHT Control Program was started. The default directory is the PLM2000 directory.

Use a mouse or the Arrow keys on the keyboard to open a directory or change drives. Use the Tab key to move the cursor through each window and button.

If there are more directories than the window can show, a scroll bar appears on the right side of the window. To view additional directories, click on the scroll bar.

To view the contents of the parent directory to the directory you are currently in, select the dual periods (..).

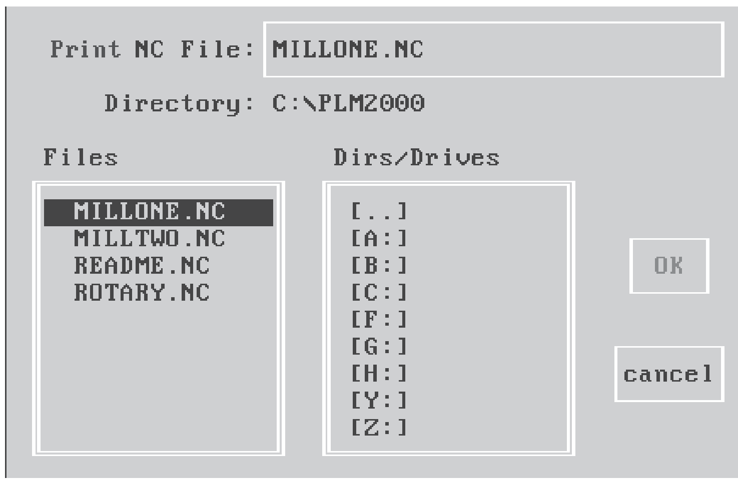
To save the file shown in the Save NC File: box, select ok. Select cancel to exit the dialog box.

The Print Command

The Print command allows you to make a hard copy of the currently loaded program. It's a good idea to print out a hard copy of new or edited programs as insurance against losing them on damaged disks.

When you select Print, the following dialog box appears. The currently opened file is automatically placed in the Print NC File: field. Select ok to print the NC program. Select cancel to cancel printing.

Before selecting Print, the printer should be loaded with paper and turned on.



If your file does not print, you may get a time-out error. This occurs when the printer is not on, or is not properly connected. If the printer is properly connected and running, and your file still does not print, refer to the manuals supplied with your computer and printer for troubleshooting procedures.

The proLIGHT Machining Center is set to print to the standard printer port LPT1. If you are printing a very large file, there may be a short delay before a print progress dialog box appears.

The Exit Command

To exit the Control Program and return to DOS, select Exit. If you have not saved the program you have been working on, you will be prompted to do so. Once that is done, the Control Program closes and you are returned to DOS.

The Edit Menu

The Edit Menu provides the NC program editing commands. After opening a program, or typing your NC code in the Edit window, you can use these commands to make modifications.

The Cut Command

Use the Cut command to delete a selected portion of your NC program and keep it available to paste elsewhere in the program. When you select a portion of NC code, then select Cut, the code is kept in memory for you to place somewhere else in the program. If you select and cut again, the previously cut portion is erased from memory. Until you use the mouse to select the NC code you wish to cut, the Cut command is grayed-out and you can not select it. Once you have selected the code you wish to cut, the Cut command is available.

The Copy Command

Note: You may use the mouse, or the Shift+Arrow key combination to select a portion of your NC program for cutting, pasting or copying.

Use the Copy command to copy a selected portion of your NC program and keep it available to paste elsewhere in the program. When you select a portion of NC code, then select Copy, the code is kept in memory for you to place somewhere else in the program. If you copy a different portion of the program, the previously copied portion is erased from memory. Until you select the NC code you wish to copy, the Copy command is grayed-out and you can not select it. Once you have selected the code you wish to copy, the Copy command is made available.

The Paste Command

Use the Paste command to paste a cut or copied portion of your NC program elsewhere in the NC program. The Paste command can be used again and again once a portion of code is cut or copied to memory. Until you cut or copy a portion of the NC program, the Paste command is grayed-out and you can not select it. Once you have cut or copied a section of NC code, the Paste command is made available. Before selecting Paste, you must move the cursor to the point in the NC program where you would like to paste the code.

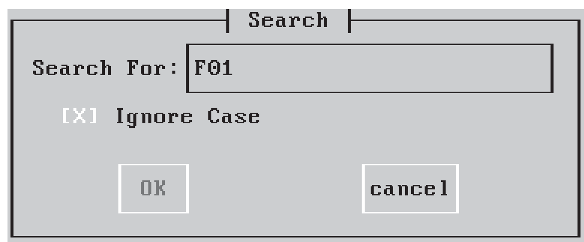
The Clear Command

The Clear command deletes NC code that has been selected with the mouse.

The Search Command

Note: The Search command begins a search from the current line of your NC program. To search the entire program, be sure to place your cursor at the beginning of the program.

The Search command allows you to look for particular character strings within the NC file, eliminating the need to scroll through the file and search by sight. This is especially handy if your NC file is large. When you select Search, a dialog box appears. Enter the character string (up to 20 characters) you'd like to look for.



Ignore Case allows you to search only for those character strings that have the same case attributes (upper case or lower case), or ignore any difference in case.

Ok initiates the search operation. The cursor will move to the first occurrence of the character string in the program. Press F3 or select the Find Again command to find the next occurrence.

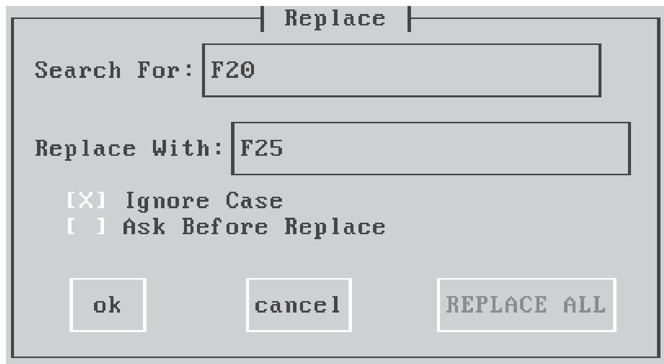
Cancel cancels the search operation and closes the dialog box.

The Find Again Command

Find Again finds the next occurrence of the character string specified using the Search command.

The Replace Command

The Replace command will look for a character string in the NC program and replace it with another character string. For instance, if your CAM program placed a feed rate of 20 ipm throughout the program, and you want to change all occurrences of 20 ipm to 25 ipm, enter the appropriate values in the Replace dialog box.



Ignore Case allows you to search only for those character strings that have the same case attributes (upper case or lower case), or ignore any difference in case.

Ask Before Replace will stop at the string and prompts you for permission to replace the string. The prompts appear in the message bar at the bottom of the screen.

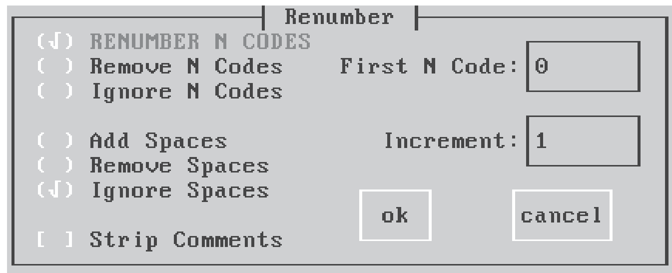
Ok initiates the search and replace operation.

Cancel cancels the search and replace operation and closes the dialog box.

Replace All changes all occurrences of the specified string. If Ask Before Replace is not selected, all occurrences of the string are replaced automatically.

The Renumber Command

The Renumber command brings up a dialog box that allows you to alter the N codes in the program.



N codes specify the sequential number of the blocks in an NC program. Using N codes is optional; however, when you do use N codes, they must be the first characters of each block in the program.

Renumber allows you to:

❑ Renumber all the N codes. (This is handy if you have added a number of blocks and must renumber the program.)

❑ Remove all the N codes.

❑ Ignore all the N codes.

❑ Increment the block numbers by whatever number spacing you wish to use. (For instance, if you enter a number spacing of 2, the block numbers will read N0, N2, N4, N6, N8, and so on.)

❑ Add spaces between all the NC codes in the program.

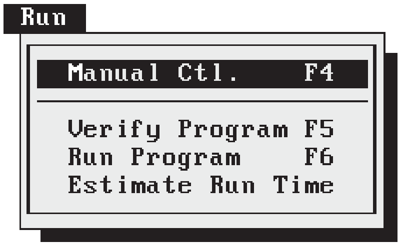
❑ Remove any spaces between NC code in the program.

❑ Remove any comments in the program.

The Locked Command

Note: Although the Copy command is available when the NC file is locked, the code you select and copy can only be pasted into an unlocked NC file.

The Locked command alternates your NC file between an editable format and a read-only format. The Locked command prevents the NC program from being altered. When an NC program is locked, you can not enter new codes, or use the Cut, Paste, Replace or Renumber commands. When a file is locked, the file name at the top of the Edit screen becomes red. Also, a check mark appears next to the Locked command on the Edit Menu. The file name of an unlocked file appears in white.

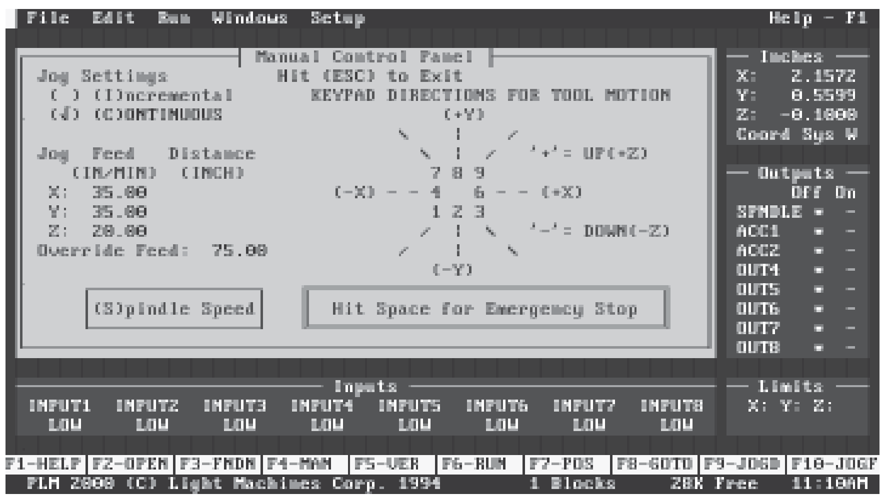


The Run Menu

The Run Menu provides commands that can directly effect the running of an NC program. These commands include Manual Control, Run Program and Verify Program. Although the proLIGHT Control Program provides graphic tool path verification, we recommend that you perform a dry run of all of your NC programs before you actually try to machine the parts.

The Manual Control Command

When you select Manual Control from the Run Menu, the Manual Control Panel appears. Using the Manual Control Panel you can: jog the cross slide to a new position, control robotic inputs, and turn the spindle and accessory outlets on or off.



Jog Settings

To select a mode of motion, use the mouse to click on the mode, or press the I-key for Incremental or the C-key for Continuous. You can also press the Insert key to toggle between the two modes. A check mark appears next to the selected mode.

Jog motion can be either continuous or incremental.

With Incremental motion selected, the tool moves in increments equal to the value specified in the Set Jog Distance dialog box (see the Setup Menu).

Continuous (or traverse) motion allows you to move the tool at the current jog speed by pressing a key (see Keypad Directions). The tool moves until you press the Space Bar, or any key on the numeric keypad.

Note: As long as Use Feedrate Override is selected in the Run Preferences dialog box (see Setup Menu), you can use the Feed Override switch on the Machining Center to override the X, Y and Z feed rates as well as the Override Feed on the Manual Control Panel.

When you select Continuous, another parameter, Override Feed, appears on screen. Use Override Feed to override the specified X, Y and Z feed rates. Specify all feed rates (X, Y and Z feeds and the Override Feed) using the Set Jog Feed command under the Setup Menu. To use Override Feed, hold down the shift key while jogging with the keypad (see Keypad Directions).

Just below Jog Settings, the current X, Y and Z Jog speed and distance are displayed. Jog distance appears only when Incremental mode is selected. Override Feed is displayed only when Continuous mode is selected.

Inputs Window

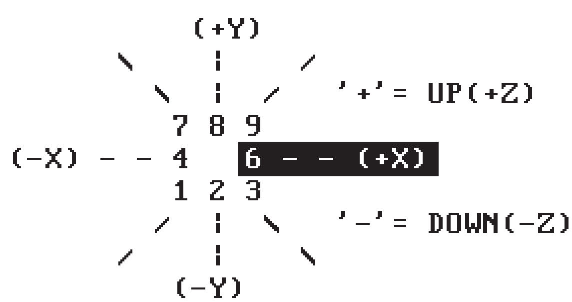
When the Manual Control Panel is active, you can view the status of the eight system Inputs. These inputs correspond to pins on the I/O connector on the rear panel of the proLIGHT Machining Center. The Robotic Inputs appear Low or High depending on the signal state at the robotic input. For more information on using the Inputs, refer to Robotic Integration, Appendix F.

Keypad Directions

Tool movement can be controlled by clicking the mouse on the keypad on the Manual Control Panel, by pressing the corresponding number keys on your keyboard (on the right-hand side of the computer keyboard) or by pressing the corresponding Arrow keys.

For example, if you click the mouse on this portion of the screen keypad (or press the 6-key)...

Note: The keypad on the Tandy 1000 keyboard works differently than other MS-DOS-compatible keyboards. If you are using a Tandy 1000, use the arrow keys for tool movement instead of the numeric keypad.



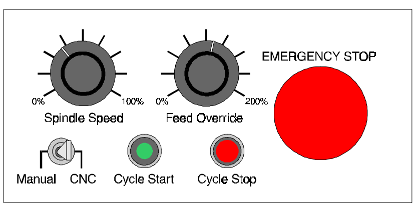
...the tool moves in the -X direction at the speed you set with Set Jog Speed.

If you use Continuous motion, the tool moves at the set speed until any key is pressed. If you use Incremental motion, the tool moves in the X direction for the specified distance.

If you click on another portion of the screen keypad, for instance +Y, the tool moves in that direction at the set speed and distance.

The plus and minus signs will make the spindle (Z axis) move up or down.

Note: The spindle speeds specified in the NC program and in the Spindle Control dialog box can be overridden by the spindle speed controls on the Machining Center.



The mode switch on the Machining Center allows you to alternate between computer control (CNC) and manual control (Manual). The mode must be set to Manual before the spindle speed knob will override the software commands.

Spindle Speed

If you have not used an S code in your NC program to specify a spindle speed, you can use the Spindle Control dialog box to enter a speed. To open the dialog box click on the Spindle Speed button, or press the S key. This dialog box allows you to alter the current spindle speed as well as turn the spindle on or off.

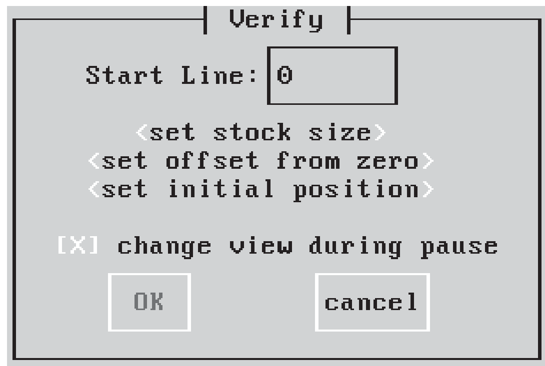
When running a program, the spindle speed is primarily determined by the Mode Switch on the front panel of the Machining Center. If the switch is set to Manual Mode, spindle speed is controlled by the Spindle Speed Control knob, also on the front panel. If the Mode is set to CNC, spindle speed is determined by an S code in the NC file. If an S code is not present in the NC file, the Control Program uses the speed entered in the Spindle Control dialog box. The Spindle Control setting does not override the S code in the NC file.

The Verify Program Command

The Verify Program Command cannot verify 4 axis programs. The Verify Command works only on the X, Y, and Z axes.

The Verify Program command generates a graphic computer simulation of the machining process for NC programs. You can only simulate the currently open NC program. When you select Verify Program, a dialog box appears. This dialog box allows you to set up parameters for the simulation.

If verification encounters an error in the program, exit Verify, and use the Edit command to correct your program. When the Edit window appears, the cursor should already be located on the block that caused the error. After correcting the error, save your program, then try verifying it again.

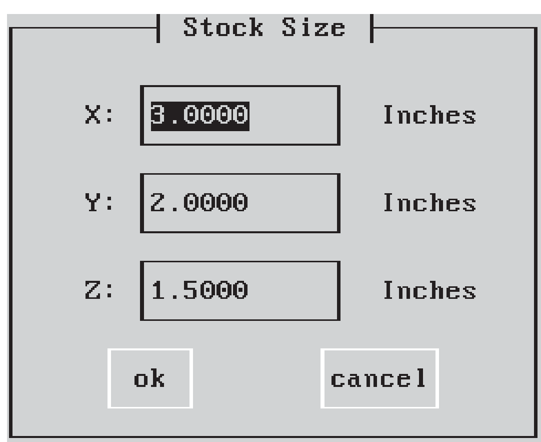


Start Block

If you wish to start verification at an NC block other than block zero, enter the block number in the Start Block box.

Set Stock Size

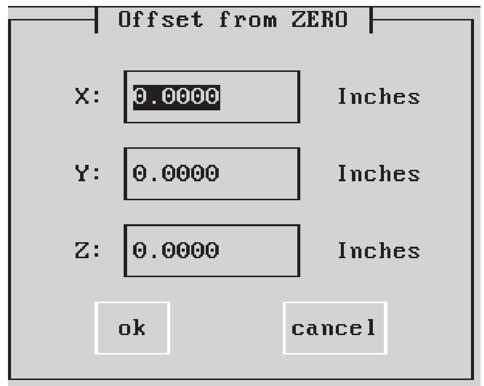
You must tell the Control Program the dimensions of the piece of stock you intend to machine. To set the size of the stock, select set stock size. A dialog box appears to allow you to enter the stock size for all three axes. Enter the desired dimensions, then select ok.



Set Offset from Standard Zero

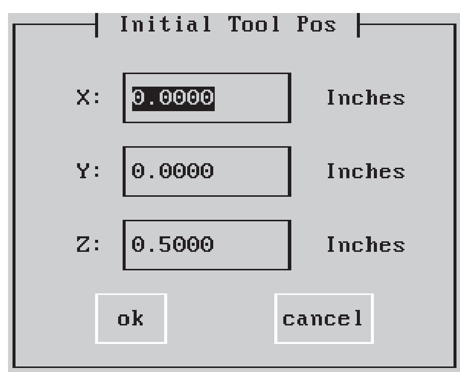
For more information on using tool offsets, refer to the discussion on machining with multiple tools in Section 6.

Set Offset from Standard Zero allows you to adjust the verification for different coordinate systems. Most NC programs set the 0,0,0 point at the left corner of the stock. In these cases the offset is zero. Occasionally, however, you may want to use a different origin (the center of the stock, for example). In these cases, you must enter an offset to properly verify your program.



Set Initial Position

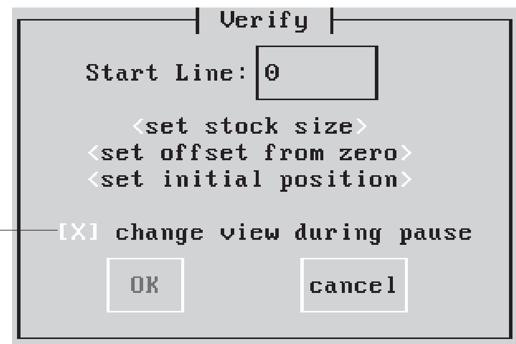
You can position the tool where you wish in relation to the workpiece by using the set initial position command to enter a tool start point. When the dialog box appears, enter the coordinates for all three axes, then select `ok`. You only use Set Initial Position for graphic verification.



Change View During Pause

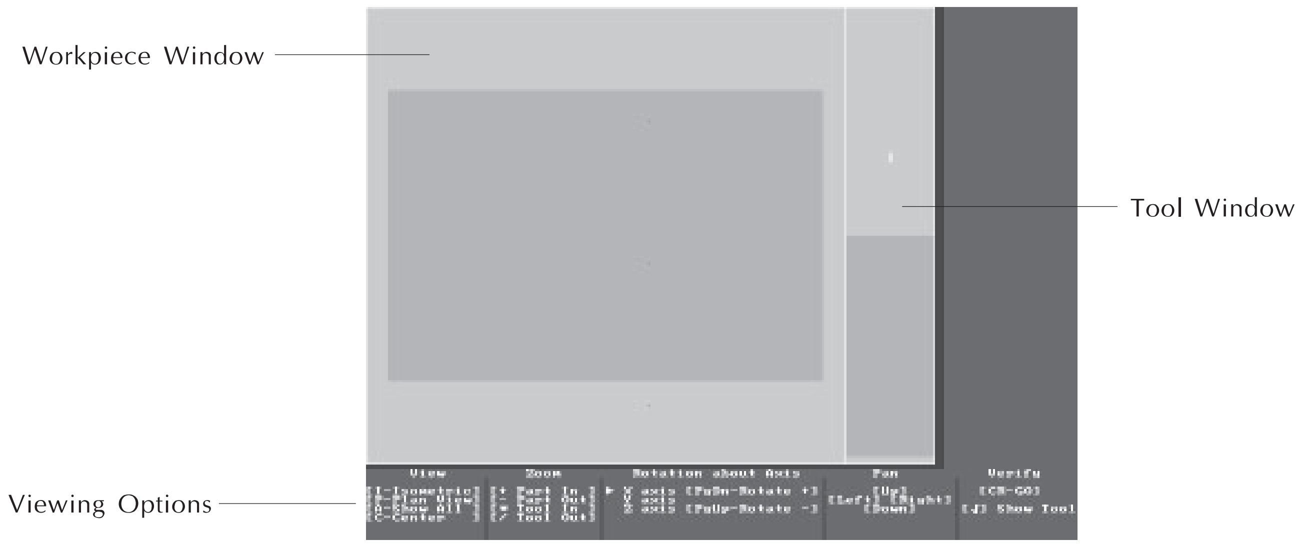
You can view the machining simulation in either *plan* view (two-dimensional) or *isometric* view (three-dimensional). If you enable change view during pause, you can pause the NC program during simulation (by pressing the P-key) and change from one view to another (by pressing the V-key to display the Viewing Options). If you do not enable change view during pause you may not change the view of the part during simulation.

When enabled, an X appears next to the Change View During Pause command.

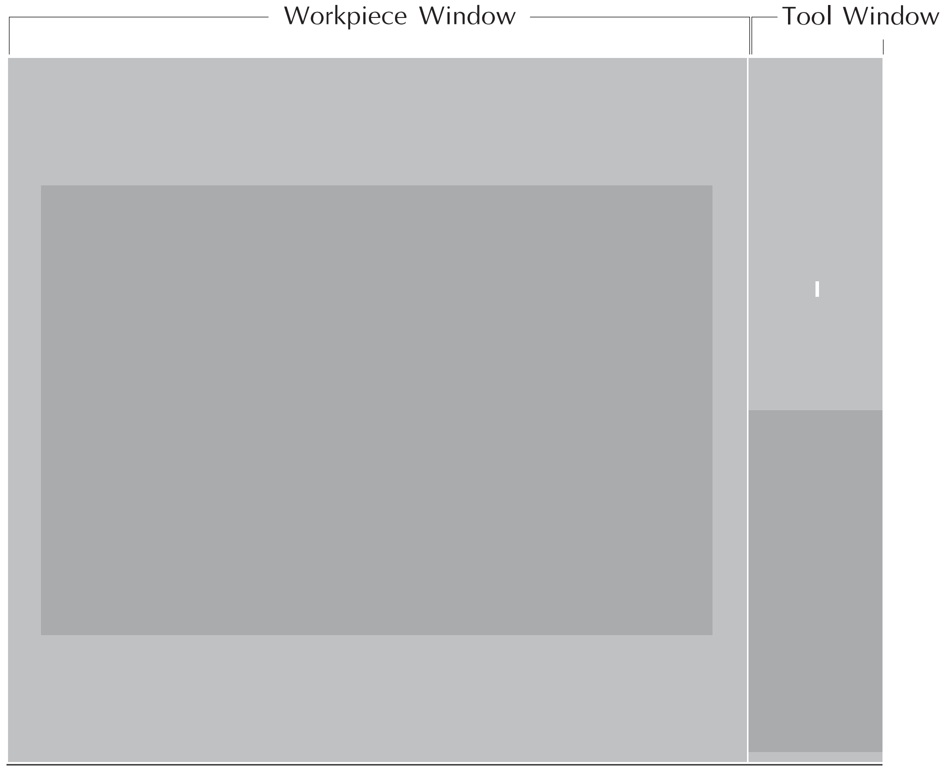


The Verify Screen

Once you have set the parameters for verification, select `ok` to bring up the Verify screen. When the Verify screen first appears, the Workpiece Window, Tool Window and Viewing Options are displayed.



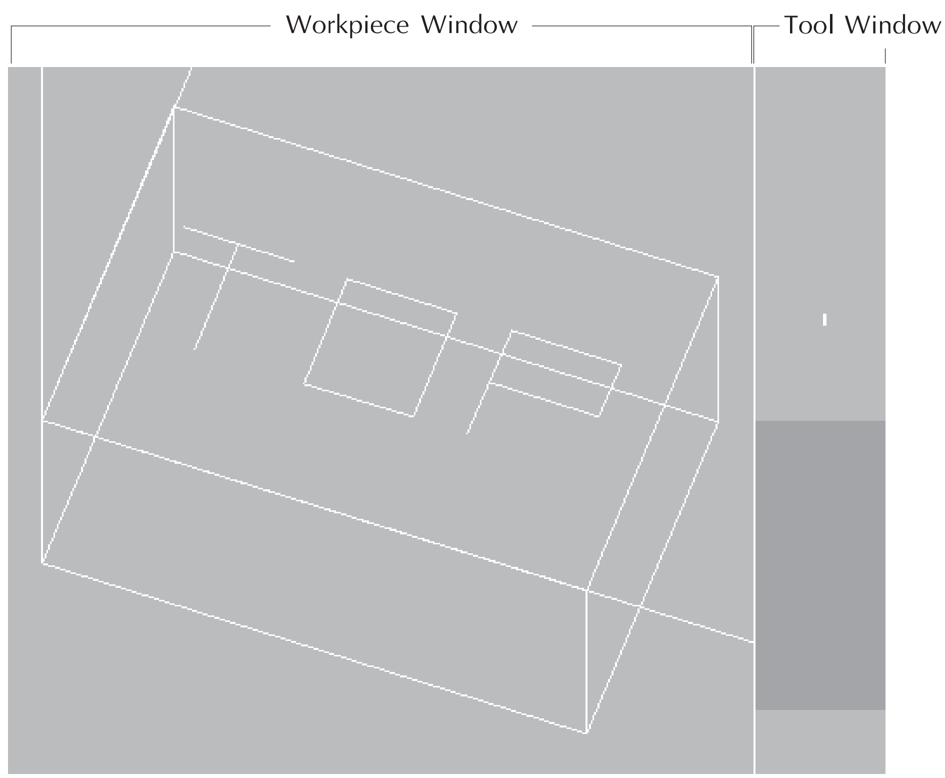
Plan View



As you can see, the difference between a workpiece shown in Plan View and a workpiece shown in Isometric View is quite obvious.

However, there is no difference in the Tool Window.

Isometric View



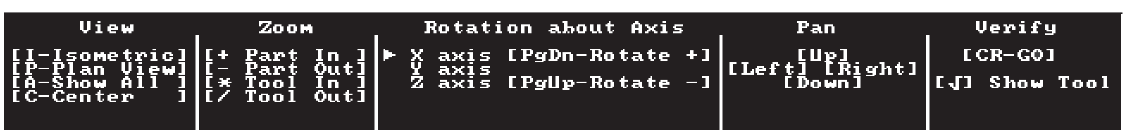
The **Workpiece Window** shows the workpiece in its entirety. The first time you verify your part program, the workpiece is shown in Plan View. Plan View is a two-dimensional view. The aspect is from above the workpiece. You can change to a three-dimensional view by selecting Isometric View (see Viewing Options).

See The **Set Tools Command** later in this section for information on setting the radius of the tool for verification.

The **Tool Window** shows a side view of the workpiece as a solid object. It can not provide a three-dimensional view (or wire frame image) like the Workpiece Window can. During verification, this window shows the tool moving up and down as the Z axis codes in the NC program are executed. When the tool moves down, you can see how far the tool is cutting into the stock.

Small, blue, horizontal lines appear on the right side of the Tool Window whenever a G00 is used in an NC program. These lines indicate that a rapid traverse motion has been made. If the tool makes a rapid motion into the stock at an inappropriate place, you'll be able to tell by the position of these lines.

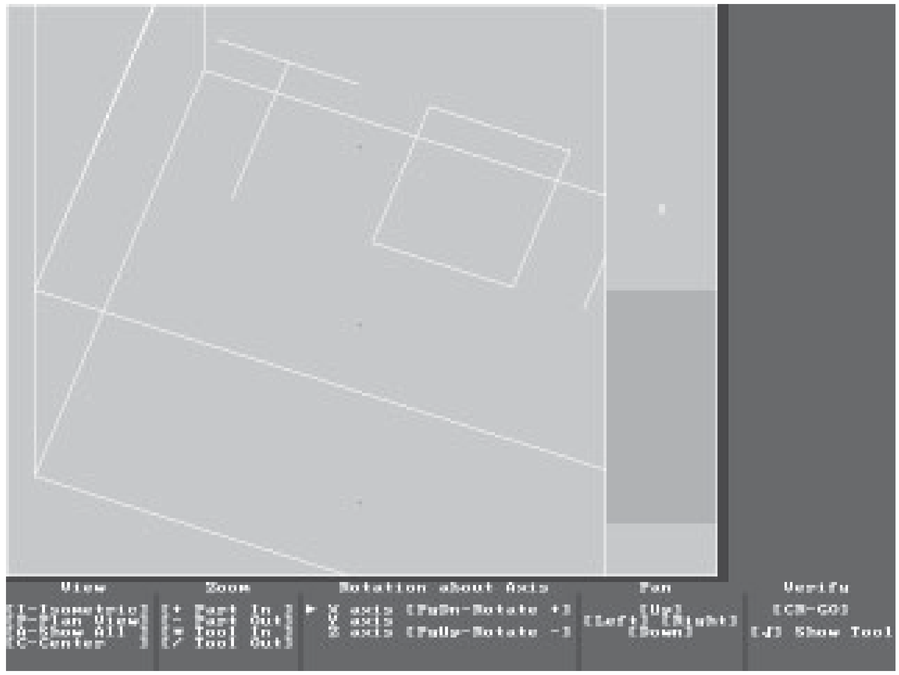
The **Viewing Options** (in the blue bar across the bottom of the screen) control the picture in the Workpiece Window. The Viewing Options include View, Zoom, Rotation and Pan commands.



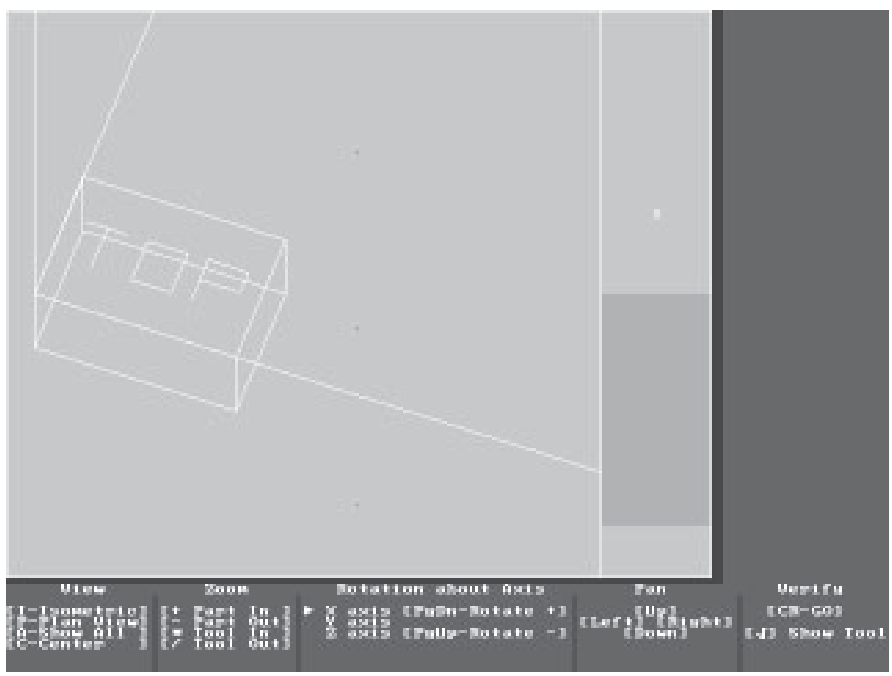
View Commands

You can watch the simulation in Plan View, which shows the workpiece in two dimensions, or Isometric View, which shows the workpiece in three dimensions. When you select Plan View, the workpiece is depicted as a solid two-dimensional object. When you select Isometric View, the workpiece is depicted as a wire frame three-dimensional object.

Here, we pressed the plus (+) key to zoom the Workpiece Window, and the asterisk (*) key to zoom the Tool Window.



Then we pressed the minus (-) key and forward slash (/) key to zoom out.



After manipulating the Workpiece Window by panning, zooming, or rotating, you can use the Show All command to resize the workpiece to fill the window. Or, you can use Center to bring the workpiece back to the center of the window after panning.

To select a View command, press the indicated key (I for Isometric, P for Plan, A for show All, or C for Center). You can also click on the command with the mouse.

Zoom Commands

The Zoom commands effect both the Workpiece and Tool Windows. To zoom in to the Workpiece Window, press the plus (+) key. To zoom back out, press the minus (-) key.

To zoom into the Tool Window, press the asterisk (*) key. To zoom back out, press the forward slash (/) key.

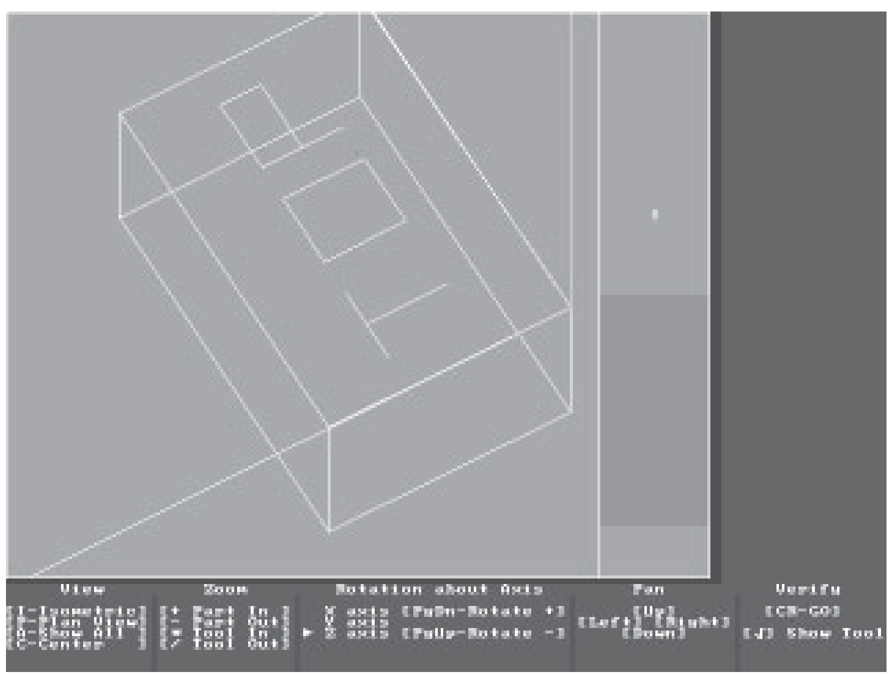
Rotation Commands

If you press the Page Down key while in Plan View, the workpiece changes from a solid image to a wire frame image which you can Rotate. You can also rotate the three-dimensional wire frame image in Isometric View.

Here, we selected the Z axis (by pressing the Z key) then we rotated the workpiece by pressing the Page Up key.

If you want to start over, press the I key to go back to the original Isometric View, then select the X, Y or Z axis and press Page Up or Page Down to rotate the workpiece.

If you rotate the workpiece out of view, use Show All or Center to bring it back into view.



Note: If you use any of the Rotation commands while in Plan View, the workpiece will change from a solid object to a wire frame object. This wire frame image is not the same as the object you'll see if you select Isometric View. An isometric object is projected at a 30° angle. The simple wire frame used in Plan View may be projected at any angle.

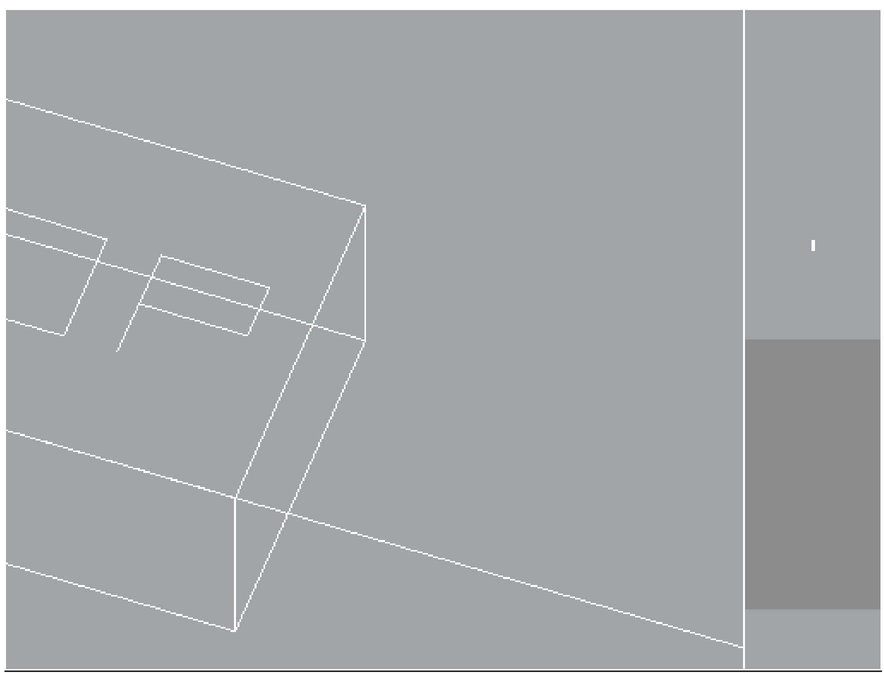
Rotations occur in 10° increments. You can select an axis of rotation for the workpiece by pressing the X, Y or Z keys, or by clicking on the command with the mouse. An arrow appears next to the selected axis and, in the Workpiece Window, the selected axis is defined by a red line.

To rotate the workpiece around the selected axis, press the Page Down or Page Up keys. Page Down rotates the workpiece in a positive direction. Page Up rotates the workpiece in a negative direction. You can also click on PgUp or PgDn with the mouse.

Pan Commands

The Pan commands move the workpiece to the left, right, up or down. To pan, press the arrow key that corresponds to the direction in which you wish to move the workpiece. For example, if you wish to view more of the right side of the workpiece, move it to the left by pressing the left arrow. (You can also click on the Pan commands with the mouse.

Press the left arrow key to move the workpiece to the left, like this, or press the right, up or down keys to pan in other directions.



Verify

In Plan View:

The tool in the Workpiece Window is depicted as a circle as it moves over the workpiece. Any rapid traverse motion is depicted by dotted lines. It is obvious when cuts are made.

In Isometric View:

The tool is represented by a small dotted line. This line represents the center point of the tool. Rapid traverse motions are depicted by larger dotted lines. Cuts on the workpiece are depicted by solid lines.

Under Verify there are two commands, [CR-GO] and Show Tool.

Clicking the mouse on [CR-GO] has the same result as pressing the Enter key to begin verification. You should select the Viewing Options before clicking on to [CR-GO] begin verification.

The Show Tool command controls the image of the tool in the Workpiece Window in both Isometric and Plan View. To turn the tool display on or off, press the S-key or click on the Show Tool command before beginning simulation. When the tool is on, a check mark appears next to the Show Tool command.

The Simulation

Press Enter to begin the simulation. As soon as the simulation begins, the Position and Outputs Windows appear in the upper-right corner of the screen, and the Viewing Options bar, along the bottom of the screen, changes into a Status Display.

The Status Display runs along the bottom of the verification screen. During verification, the Status Display is continually updated (as are the Position and Outputs Windows).

The Status Display shows the NC program name and path, plus the current block, tool number, pass, feed rate, display speed (see Hidden Commands) and spindle speed.

Each block of NC code is displayed as it is executed. The immediately preceding and following blocks are also displayed.

If you pause the simulation (press the P-key), a Pause message appears on the bottom of the Status Display until you resume verification by pressing the Enter key. Other messages, such as tool change instructions and error messages appear here as well.

The Status Display

The Position and Outputs Windows that appear during verification are the same windows you see in the main screen. If you turn them off in the main screen (see The Windows Menu) they will not appear here.

Hidden Commands

There are a number of hidden quick-key commands available to enhance the verification simulation.

Display Speed

The Display Speed commands allow you to place a delay between each block of NC code (or arc segment). The speed settings range from 0 to 9, with 0 being the slowest setting. The default setting is 9.

To alter the Display Speed, press the number for the speed you wish to run the verification. For example, to verify at 5, press the 5 key while the verification is running.

View Change

If you wish to change the view of the Workpiece Window while the NC program is being verified, pause the simulation by pressing the P-key, then press the V-key to display the Viewing Options bar. Change the view as desired, then press Enter to resume the simulation.

The data required to perform this function is saved to disk while you are verifying. Make sure you have adequate space on your hard drive. (Typically, you should have at least half the size of the NC program available.) The data file is deleted as soon as you are through verifying, so it doesn't continue to consume disk space each time you verify. You may verify from a floppy, though it may be a little slower and you may notice some pauses while it accesses the drive.

View Right and View Front

To quickly view either the right side or the front of the workpiece, use the View Right and View Front commands. These commands work in both Plan and Isometric Views.

View Right displays a wire frame view of the right end of the workpiece. To select View Right, press the R-key while the Viewing Options bar is active.

View Front displays a wire frame view of the front end of the workpiece. To select View Front, press the F-key while the Viewing Options bar is active.

Single Step

This command pauses the simulation after each NC block so you can view the execution of the program a block at a time. Set single step mode in the Set Run Settings dialog window. To move on to the next block in the program, press Enter.

Stopping

Both the Q-key and Space Bar can be used to initiate emergency stops of the verification. If you press the Space Bar or Q-key to initiate an emergency stop, a small dialog box appears.

A similar dialog box appears when the M02 code is encountered at the end of the NC program.

This box gives you the opportunity to either exit verification, or change the view to examine where an error occurred. If you elect to exit, you may exit verification by pressing the ESC key, the Enter key, the Space Bar, or by clicking on Done in the dialog box.

Pausing

Use the P-key to pause the simulation. (You must pause before using the V-key for Viewing Options.) When an NC program is paused, you can press the ESC key to exit verification.

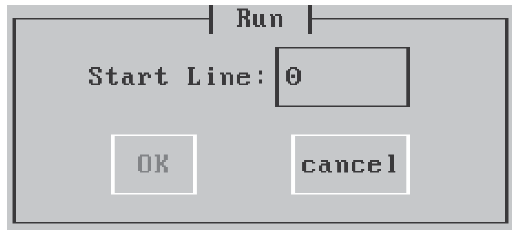
CAUTION:

Always wear safety glasses and close the safety shield before running a program.

Always observe the set-up and safety precautions in Appendix C of this guide.

The Run Program Command

The Run Program command from the Run Menu executes the current part program. When you select Run Program from the Run Menu, a small dialog box appears. You are given the option of running the program from block zero, or specifying the starting block of the program.

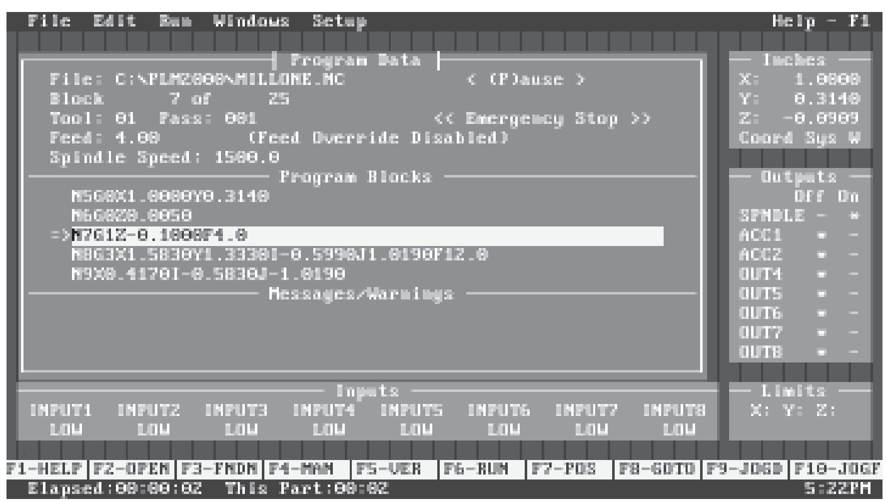


Program Settings

Before running an NC program, the initial X, Y and Z axis coordinates of the cutting tool must be set. Use the Jog function on the Manual Control Panel to position the cutting tool. Use the Set Position command to enter the tool position.

When you start at a block other than zero, 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.

With the start point specified, the Run Program dialog box appears. This box has three major areas: Program Data, Program Blocks, and Messages/Warnings.



Program Data tells you the name (full pathname) of the running file, which block is currently being executed and how many blocks are in the program, which tool is being used, and the program loop counter. If Use Feedrate Override is selected in the Run Preferences dialog box, a "Feed Override Enabled" message appears. If Use Feedrate Override is not selected, a "Feed Override Disabled" message appears.

```
                            Program Data
 File: C:\PLM2000\MILLONE.NC              < (P)ause >
 Block      7 of     25
 Tool: 01  Pass: 001                   << Emergency Stop >>
 Feed: 4.00        (Feed Override Disabled)
 Spindle Speed: 1500.0
```

Program Blocks shows the block being executed along with the two previous blocks and the next two blocks.

```
                            Program Blocks
    N5G0X1.0000Y0.3140
    N6G0Z0.0050
  =>N7G1Z-0.1000F4.0
    N8G3X1.5830Y1.3330I-0.5990J1.0190F12.0
    N9X0.4170I-0.5830J-1.0190
```

Messages/Warnings provides messages such as "Pause at Block 0" and prompts for tool changes. It also alerts you to errors or operational problems that you may encounter while running a program.

While the Program is Running . . .

While the system is running a program, only certain keys are active; they are the P-key, the Q-key, and the Space Bar.

Pause

Pressing the P-key will pause the program after the current NC block has been executed. To resume running the program after a pause, press the Return key. To abort the program and return to the Main Menu, press the Space Bar.

Q and Space Bar

Press the Q-key or the Space Bar to stop the program. Cutting halts immediately and the current tool position is stored by the computer. When you press the Return key you are automatically returned to the Edit mode.

To restart a program after an Emergency Stop from the keyboard, manually jog the tool above the workpiece to avoid a tool crash. Then, select the Run Program command from the File Menu. You will not have to reset the initial tool position (assuming the first block of your program moves the tool to the start position).

An NC program can also be stopped by pressing the Emergency Stop Switch on the front panel of the Machining Center, or by tripping a limit switch. Refer to "Safely Running the Machining Center" in Section 4 for instructions on how to recover from these Emergency Stop conditions.

Quitting the Program

When your program has come to a normal program stop (M02), press Return or ESC, or click on `ok`.

The Estimate Run Time Command

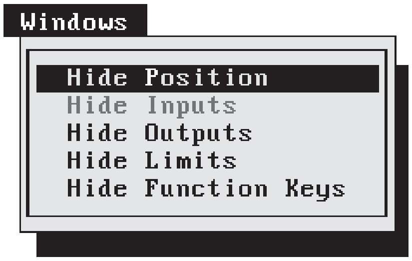
The Estimate Run Time command from the Run Menu calculates the approximate amount of time the proLIGHT 2000 needs to machine your part and the approximate distance the machine travels while machining your part.

The Estimate Run Time command does include Dwell times and subprograms when calculating the estimated run time, but it can not include program stops that have indefinite lengths of stop time. These program stops include:

- ❑ Pause (G5/M00)
- ❑ Chain (M20)
- ❑ Skip/Digitize (G31/G131)
- ❑ Wait for input high/low (G25/G26)
- ❑ Write to file/Write to file with Digitize coordinates (M22/M122)
- ❑ Rerun (M47/M47Ln)

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

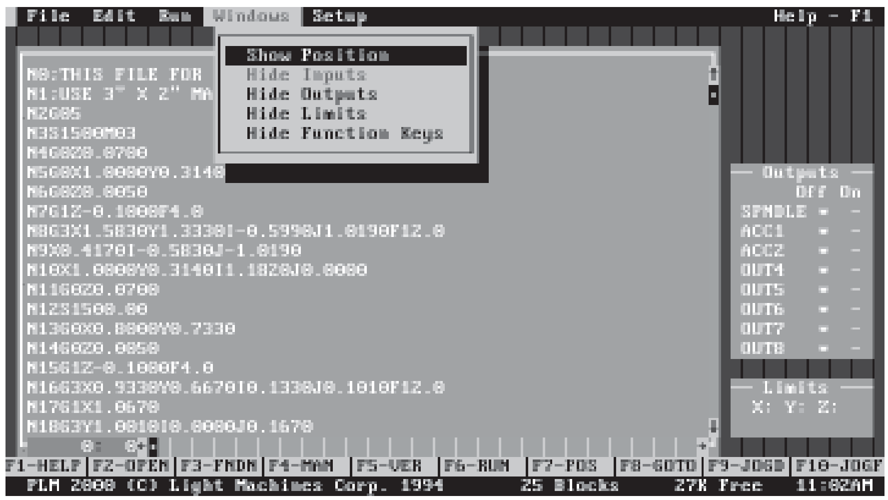
This command also verifies the syntax of your NC programs while calculating the estimated run time. If an error is found, the Estimate Run Time Command alerts you with a dialog box.



The Windows Menu

The Windows Menu controls the appearance of the Position, Outputs, Limits and Inputs windows. For instance, if you select Hide Position, the Position box in the upper right corner of the screen disappears.

If you pull down the Windows Menu again, the Hide Position command has changed to Show Position. If you select Show Position, the Position box reappears and the command reverts to Hide Position.



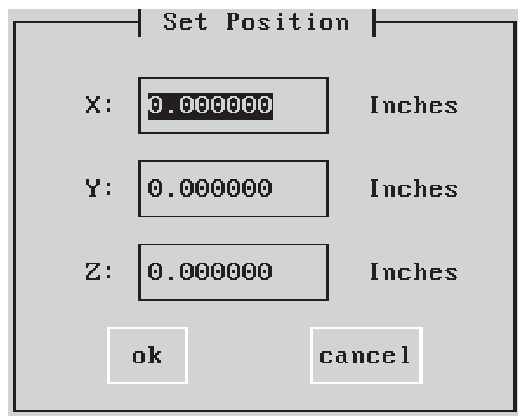
The Hide Inputs command is grayed-out and can not be selected unless the Manual Control Panel is active.

The Setup Menu

The Setup Menu provides a number of commands that allow you to set up system parameters.

The Set Position Command

Selecting Set Position brings up a dialog box for setting new X, Y and Z positions. The Set Position command establishes a *world coordinate* system in relationship to the machine coordinate system, or machine zero. (Each time you turn on the Machining Center, the current machine position is stored as the machine coordinate system, X0,Y0,Z0.)



Setting the X, Y and Z coordinates for the tool also defines the zero point of the coordinate system for absolute motion.

When you save system parameters (see Setup Preferences) the current X, Y and Z position settings are saved. If you save the current parameters, you will not have to initialize the position the next time you run the Control Program (assuming the job set up is not altered and the machine is not shut off).

The Zero Position Command

The proLIGHT Machining Center features a *floating zero format*. This means you can set the X, Y and Z zero points anywhere on the workpiece by moving the tool tip to the coordinates. Since the tool length and the workpiece position on the cross slide may change from one tooling set-up to another, the machine zero position needs to be initialized each time the set-up is changed.

The zero point for the Z axis should be set when the tip of the cutting tool just touches the surface of the workpiece. Motion into the workpiece is in the negative direction from the zero point (-Z); motion away from the workpiece is in the positive direction (+Z).

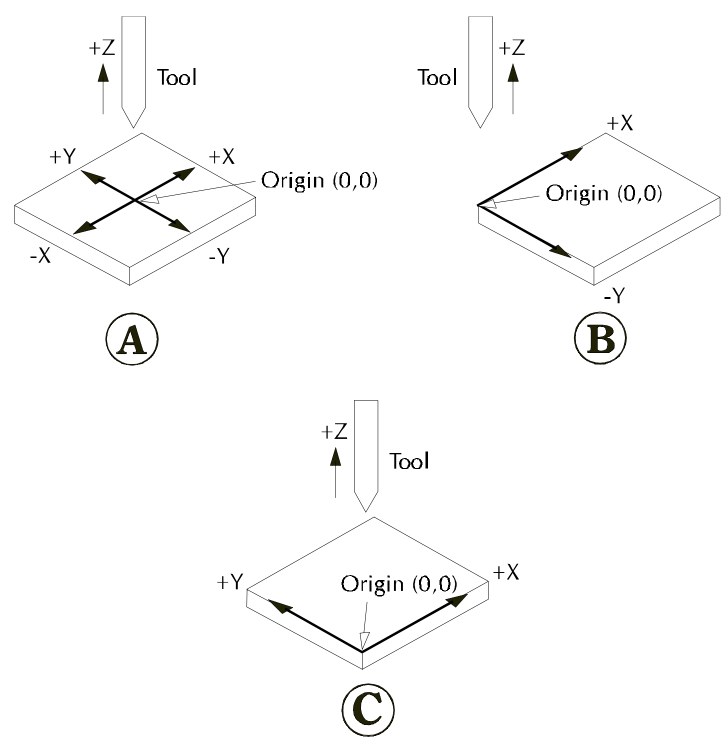
The X, Y zero point can be located anywhere on the workpiece. Illustrations A, B and C show various locations for initializing the zero point on the workpiece.

Locating the X, Y Zero Point on the Workpiece (Isometric View)

Locating the zero point at the center of the workpiece (A) will produce four quadrants of motion with combinations of negative and positive coordinates.

Specifying the zero point in the upper left corner of the workpiece (B) produces a single quadrant of motion, but still requires negative and positive coordinates for motion.

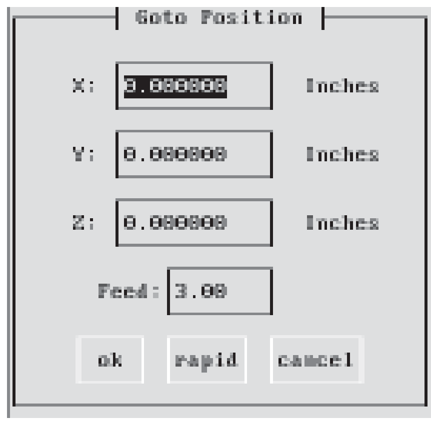
Locating the zero point at the lower left corner of the workpiece (C) produces a single quadrant of motion with positive coordinates for motion.



Once you have located the zero point on the workpiece, you can select Zero Position to establish the current tool position as the zero point.

The Goto Position Command

The current position coordinates of the tool are shown in the dialog box that appears when you select Goto Position from the Setup Menu. Goto Position allows you to enter coordinates to move the tool to any point on the workpiece. The coordinates are relative to the zero point established using the Set Position command. Changing the coordinates causes a move to occur.



Selecting ok causes the machine to make a linear motion to the specified coordinates at the programmed feed rate. Selecting rapid causes the machine to move to the new coordinates at the rapid traverse rate.

The Set Jog Distance & Set Jog Feed Commands

Note: The jog function can be used for cutting, but be careful. If the cutting distance is too large, the tool could jam or the workpiece could break before the jog move is completed.

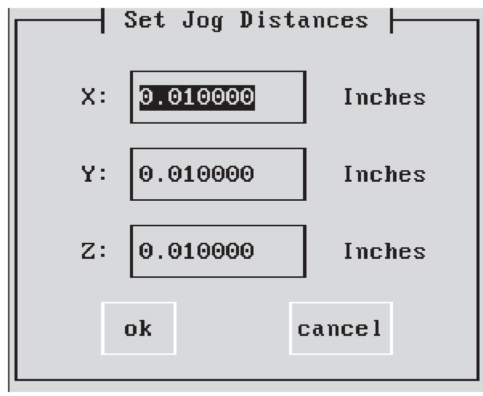
Use the Set Jog Distance and Set Jog Feed functions to control tool motion in the manual control panel.

Jog motions are controlled according to the values you enter for the jog distance and the jog feed rate along the X, Y and Z axes. Distance and feed rate values are separately entered for each axis. Jog Settings for the type of motion (Incremental or Continuous), X, Y and Z feeds and distances are shown on the Manual Control Panel.

Set Jog Distance

Distance values determine how far the tool moves each time a key is pressed while using the Incremental Jog Setting. This distance can be set at a low value (for instance 0.005 inch) to move the tool for a precise cut or at a high value (e.g. 5 inches) to position a tool. To change the current distance values, select Set Jog Distance from the Setup Menu. The following dialog box appears.

Enter desired values in each field and select ok. The new values appear in the Jog Settings area of the Manual Control Panel.

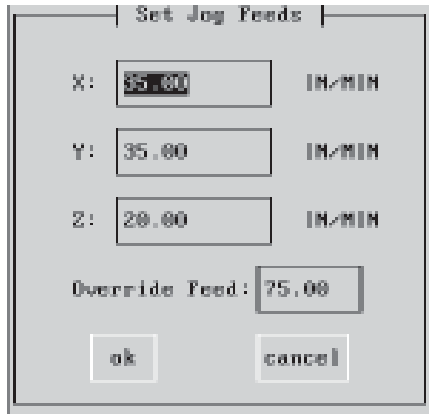


Set Jog Feed

The jog rate for the X and Y axes is the feed rate at which the tool moves away from or toward the workpiece along the X or Y axes. The Jog Feed for the Z axis is the feed rate at which the tool moves toward or away from the workpiece along the Z axis. These feed rates can be set as high as 150 inches per minute.

Override Feed is used to override the X, Y and Z feeds. To use Override Feed, press and hold the Shift key while pressing an Arrow key.

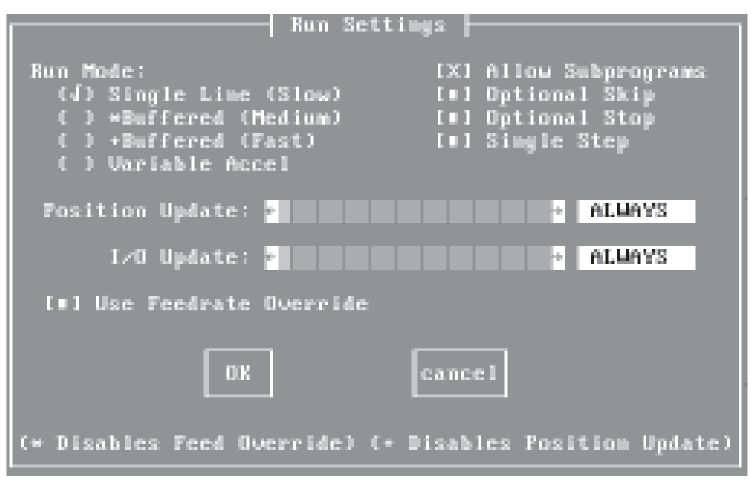
To change the current feed values select Set Jog Feed from the Setup Menu. The following dialog box appears.



Enter desired values in each field and select ok. The new values appear in the Jog Settings area of the Manual Control Panel.

The Set Run Settings Command

The Run Settings allow you to set certain parameters for running an NC program. When you select Set Run Settings from the Setup Menu, the following dialog box appears.



Run Mode

The Run Mode determines how the computer transmits NC code to the controlling electronics on the Machining Center. There are four modes: Single Line, Buffered (Medium), Buffered (Fast), and Variable Acceleration. The *buffer* is the onboard memory on the machine. The buffer has a memory capacity of 32 kilobytes where it holds the NC code sent by the computer.

Single Line mode allows you to maintain maximum control of the machining center. The machine can use Feed Rate Override and can adjust Position and I/O updates while in operation. A pause occurs immediately in Single Line mode.

Buffered (Medium) mode allows for faster downloading of NC programs to the machine while still allowing the machine to adjust Position and I/O updates. The machine puts user requested pauses in the queue; therefore, there is a delay while the machine executes the codes still in the buffer. There is no Feed Rate Override in Buffered (Medium) mode.

Buffered (Fast) mode provides continuous buffering of the NC file for faster running of the program. There is no communication (no Position and I/O updates) between the machining center and the computer until a problem is encountered. Feed Rate Override is available in Buffered (Fast) Mode.

Variable Acceleration mode changes the acceleration rate based on the line segment length in an attempt to achieve the commanded feed rate. The Accel.Map file determines the acceleration rate based on line segment length. Use the Variable Acceleration Mode when your part tolerances are not critical or when speed of machining is more important than precision.

Other Run Settings

Allow Sub-Programs allows you to run NC programs that contain subprograms. If you wish to run an NC program that contains a subprogram, this option *must* be enabled.

Optional Skip allows you to execute or ignore any optional skips (/ codes) you have embedded in the NC program.

Note: Decreasing the Position and I/O Update intervals allows programs to run more smoothly by decreasing the amount of communication between the mill and the computer.

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. Single Step is particularly useful after changing the workpiece size.

Position Update allows you to set a time interval for updating the position readout in the Position Window. Just move the cursor to Position Update with the Tab key, then use the arrow keys to adjust the time. Or, just click on the line with the mouse.

I/O Update allows you to set time intervals for updating the input status in the Inputs Window. Just move the cursor to I/O Update with the Tab key, then use the arrow keys to adjust the time. Or, just click on the line with the mouse.

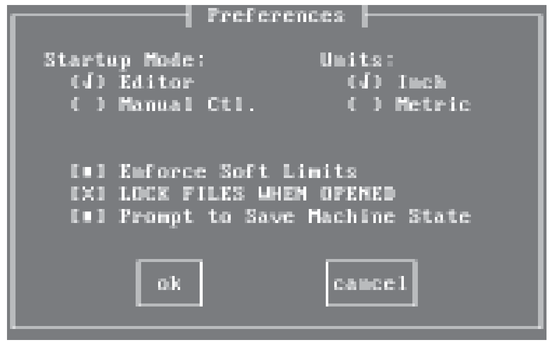
Update Current Line in Block Mode controls how the blocks of NC code are displayed when a program is running in Block mode. If this command is selected, each line of code is highlighted as it is executed, even though an entire block of code was transmitted to the machine. If this command is not selected, only the final line of code in the transmitted block is highlighted.

Feedrate Override gives you an alternative to using the feeds specified in the NC program. With this command selected, you can turn the Feed Override knob on the Machining Center to increase or decrease the feed rate while the NC program is running. The switch will override the X, Y and Z feeds, as well as the Override Feed specified in the Set Jog Feed dialog box.

Before running a program with Feedrate Override selected, make sure the Feed Override switch on the machining Center is set at approximately mid-range. This will closely approximate the feed specified in the NC code. Adjustments can then be made, either up or down in value. Feed Override allows adjustment of the feed rate from 0% to 200% of the programmed feed rate.

The Set Preferences Command

This command provides general system preferences. When you select Set Preferences from the Setup Menu, the Preferences dialog box appears.



Startup Mode

This allows you to select which screen you'd like the software to start up with. You may select either the Edit window or the Manual Control Panel. This will not take affect until you restart the program.

Units

The proLIGHT Machining Center supports both inch and metric measurements. The currently selected measurement mode is displayed in the Position window.

Other Preference Options

Enforce Soft Limits gives you the option of using or ignoring the Soft Limits you set up with the Set Limits command.

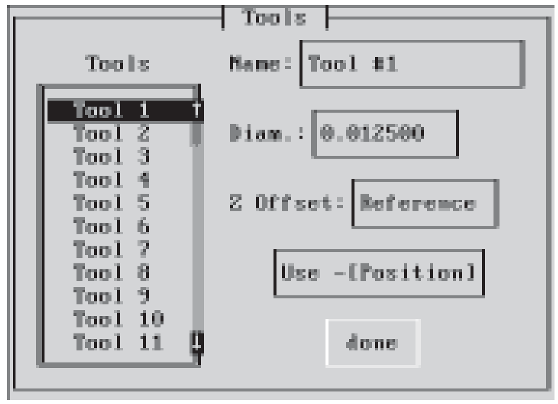
Lock Files When Opened protects newly opened files from being changed. This is handy if you want your NC files to be used as *read only* files.

Current parameters are written to the directory containing the Control Program files. The next time you run the Control Program, the parameters and preferences will be unchanged from the previous session.

Prompt to Save Machine State will prompt you to save the current parameters, such as the Jog Feed and Distance, and the current X, Y and Z position, as well as the preferences you set. If you do not select this command, the system will automatically save all the parameters and preferences for you. When you select this command, the computer prompts you to save the parameters when exiting the software.

The Set Tools Command

When you select Set Tools from the Setup Menu, the following dialog box appears.



The Tools dialog box allows you to specify tools and establish individual Z axis offsets for each tool. You may assign offsets for up to 20 tools.

Name allows you to enter a description for this tool. Your description can be up to 15 characters long.

Diam is used to specify the diameter of the tool displayed during tool path verification.

Z Offset is used by the computer to compensate for different tool lengths while running a program.

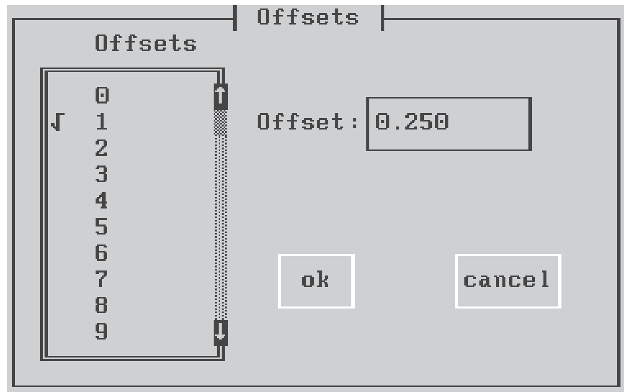
When you select a tool from the Tools list, the name of the tool appears in the Name box. You can type in any name you like. For instance, if the first tool you'll use for your NC program is a 1/4-inch end mill, select Tool 1 then select the Name box and type 1/4 End Mill.

Use -[position] establishes tool offsets. For more information on tool offsets and multiple tool programming, refer to Section 6.

The Set Offsets Command

When you select Set Offsets from the Setup Menu, the following dialog box appears:

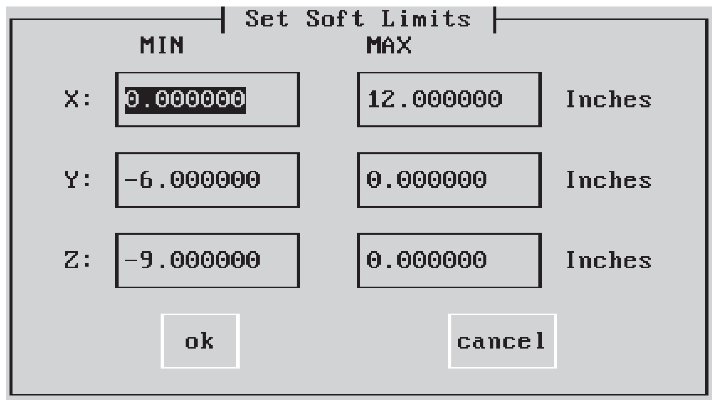
Note: You cannot assign a value to offset number 0. The offset value for offset number 0 is always zero.



Select an offset number or set the offset value in the Offset Box. To set the offset value, double-click on an offset number from the scrolling Offset window. A check marks appears to the left of the offset number. Enter the offset value in the Offset Box and select ok.

The Set Limits Command

This command allows you to establish limits for each axis within the software that are different than the actual fixed hardware limits on the machine. Soft limits are defined in relation to the machine coordinates; therefore, you must home the machine before using soft limits. The soft limits confine the travel of the tool to an area smaller than the normal maximum travel of the tool.



The machine shuts down if it trips a soft limit. 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.

WARNING

Soft limits are based on Machine Coordinates. You should only enable soft limits if you Home the machine, establishing the proper Machine Coordinate system. Soft limits do not function properly if the machine has not been homed.

The Coordinate Systems Command

This command allows you to use the code for one part to machine multiple parts based on multiple coordinate systems. You can do this by inserting the appropriate coordinate system G codes (G54 through G59) in the NC program and entering offset values using the Coord. Systems command. You can also use subprograms to perform the actual machining while the main body of your NC program alternates coordinate systems.

Caution:

Be careful when mixing use of Coordinate Systems and G92 codes. If you use a G92, your coordinate systems *will* be offset.

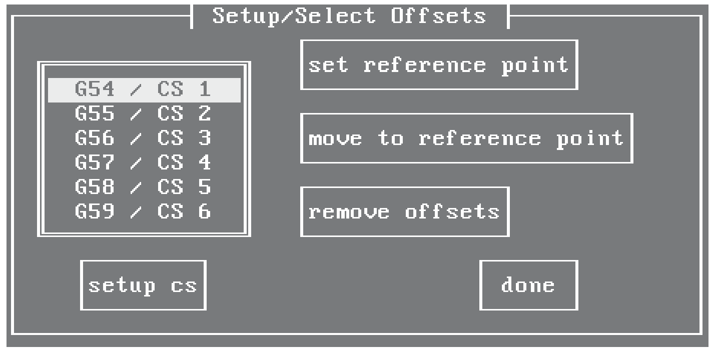
Here's an example of an NC program using multiple coordinate systems and a subprogram:

```
G54;USE COORDINATE SYSTEM ONE
M98P1000;GO TO SUBPROGRAM AT O1000
G55;USE COORDINATE SYSTEM TWO
M98P1000;GO TO SUBPROGRAM AT O1000
G56;USE COORDINATE SYSTEM THREE
M98P1000;GO TO SUBPROGRAM AT O1000
M2;END OF MAIN PROGRAM
O1000;SUBPROGRAM TO MILL SQUARE
G90X.125Y1Z.1;RAPID TO .125, 1, .1
G1Z-.1F2;PLUNGE AT CURRENT LOCATION
X1
Y.125
X.125
Y1
G90G0Z.1
M99;RETURN FROM SUBPROGRAM
```

To machine the first part, we called out a G54 (coordinate system one) at the beginning of the main program. We used an M98 to call the subprogram that does the actual machining. When the machining is finished, we are returned to the main program by the M99.

After returning to the main program, we execute a G55 (coordinate system two command). We called out the subprogram again to machine the same part, but at a different position on the workpiece. You can repeat this cycle in your part program for each part (up to six parts, using six coordinate systems).

You must set up multiple coordinate systems before you run your NC program. When you select Coord. Systems from the Setup Menu, the following dialog box appears.



Note: There are G codes that perform similar functions:

•A G27 performs a "check reference point."

•A G28 performs a "set reference point."

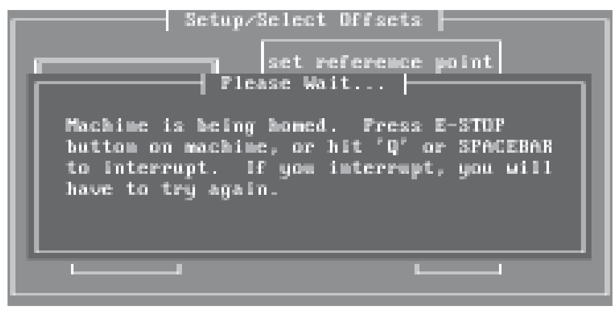
•A G53 performs a rapid to the coordinates specified, in machine coordinates (e.g. G53X0Y0Z0 rapids to reference point).

The **set reference point** button homes the machine to the machine position to 0,0,0. When you select set reference point, the following message appears:

If you select OK, the machine moves to the negative X axis limit, positive Y limit, and the positive Z limit. The PLM2000 uses a fixed home position (0,0,0) as a reference point. You must set the reference point before setting up any alternate coordinate systems or before using soft limits.

The **move to reference point** button checks the reference point, similar to using a G27 code. It compares the reported position against zero to see if any position has been lost. When you select move to reference point, the following dialog box appears:

Select OK to move the machine to the reference point. The following dialog box appears:



The **remove offsets** button sets all coordinate systems back to the reference point of 0,0,0 (machine coordinates).

The **setup cs** button brings up a second dialog box that allows you to enter offsets for alternate coordinate systems.

Setting the Coordinate Systems

Basically, the process for setting up multiple coordinate systems is similar to setting up multiple tools. First you'll want to establish the reference point from which the alternate coordinate systems are offset. Select the set reference point button. A dialog box appears, asking if you wish to move the tool to the home position, 0,0,0. Select ok. Unless it is already there, the machine moves to its fixed home position.

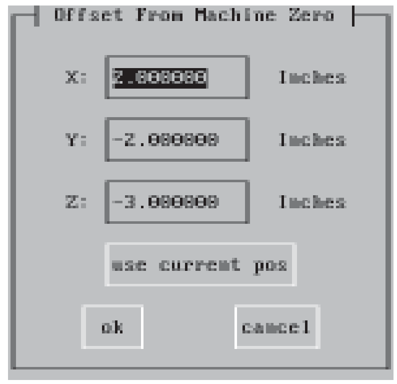
To open the Offset from Machine Zero dialog box you can:

•select a coordinate system with the mouse, then select the **setup cs** button,

•double-click on the coordinate system, or

•select a coordinate system with the Tab key, then press Enter.

Select coordinate system one, G54/CS 1. Then select setup cs. The Offset from Machine Zero dialog box appears.



Note: You can also move the machine to a specific position using the Manual Control Panel, then use the **use current pos** button to automatically enter offsets for a coordinate system.

Enter offset values for all three axes. When G54 is executed in the NC program, the point of origin is offset from the machine's home position by the amount you enter in this dialog box. Select ok. The first dialog box reappears. Repeat this process for each successive coordinate system you wish to offset.

Using Multiple Coordinate Systems

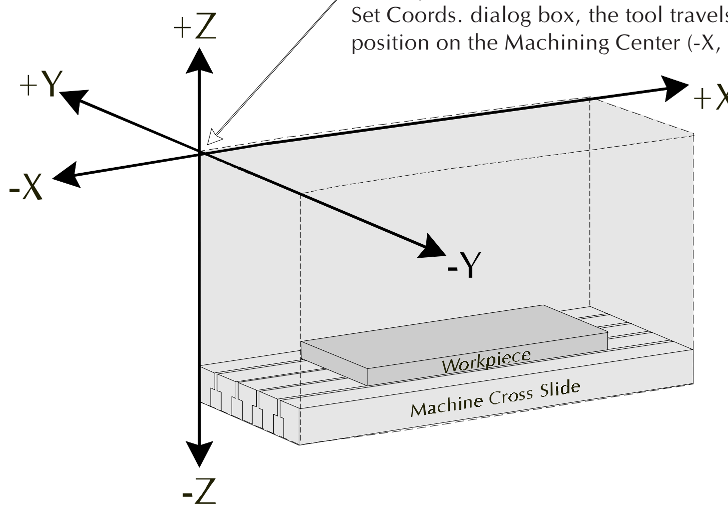
There are actually three coordinate systems for the proLIGHT Machining Center:

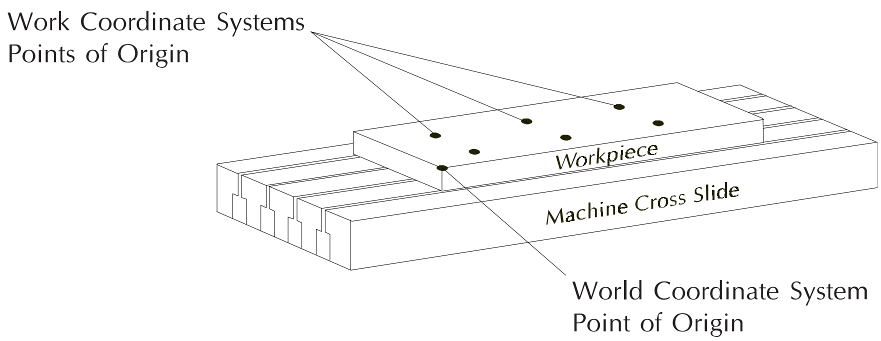
Machine Coordinates

Machine coordinates are established by Homing the system to establish a point of origin at the ends of travel on the Machining Center. Once established, these coordinates remain fixed. When the Remove Offsets button is pressed in the Coordinate Systems dialog box, all offsets from this fixed position are canceled.

World Coordinates

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. The coordinate system established with this offset point of origin is known as the World coordinate system. World





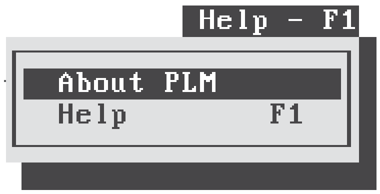
coordinates are not fixed; they can be established anywhere else on the system by using the Set Position command or by using the G92 code.

Work Coordinates

The coordinates established by using the G54 through G59 codes along with the Coordinate Systems command, are the Work coordinates. Work coordinates are offset from the World coordinates. If you do not use World coordinates (if you do not establish a point of origin using either the Set Position command or a G92 code), the Work coordinates are offset from the fixed Machine coordinates.

Setting Reference Point/Establishing Machine Coordinate System

If you plan on using any of the following commands . . .
G27, G28, G29, G53, G54, G55, G56, G57, G58, G59, and Soft Limits
. . . you should perform a Set Reference Point from within the Coordinate Systems dialog box every time you start up the Control Program, especially after the machine has been shut off.



The Help Menu

The Help Menu provides helpful information about the Control Program's menus and commands. It also provides tips for using the Control Program.

The About PLM Command

When you select About PLM from the Help Menu, a box appears with the program name, version number, copyright information and the name of the selected machine.

The Help Command

The Help command allows you to quickly call up information about specific features of the Control Program. When you select Help from the Help Menu (or press F1), a dialog box appears.

The Help information is presented in hierarchy form. The topics that have subtopics are indicated by a small dot to the left. Click on a topic for general information on the main topic. Click on the Right Arrow (or double-click on the topic) to move downward in the hierarchy to view subtopics. Click on the Left Arrow to move back to the Main Topic.

Section 4: Tutorial

Starting the Control Program

Note: Do not attempt to operate the proLIGHT 2000 Machining Center without reading and practicing the safety instructions contained in this Section and Appendix C.

The correct sequence for running the proLIGHT 2000 Machining Center with your computer is:

1. Make sure the Machining Center is turned off.
2. Turn your computer on and allow it to boot up fully.
3. Turn on the Machining Center.
4. Run the Control Program.

The procedure for running the Control Program is:

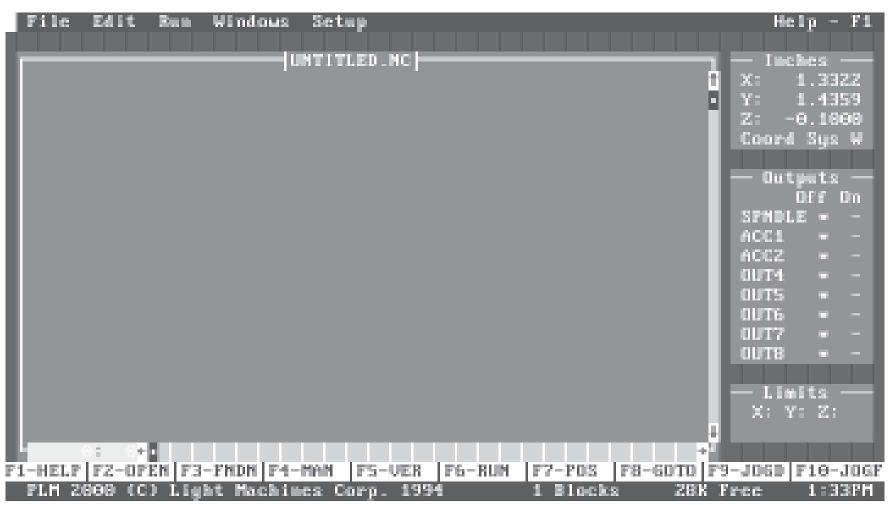
1. At the C:> prompt, change from the main directory to the directory containing the proLIGHT files. Type: CD\PLM2000 and press the Enter (or Return) key.

   (If you made up your own directory name when you installed the proLIGHT Control Program, type CD\ then enter your own directory name and press Enter.)

Note to 4th Axis Rotary Positioner users:
The start up procedure is slightly different for starting the Control Program when using a Rotary Positioner. See Section 6 for details.

2. When you're in the proper directory, type PLM2000 at the C:> prompt and press Return.
3. When the proLIGHT Control Program is running, the main screen appears.

This is how the main screen looks when you first start up the Control Program.



Having a Problem?

If the following screen appears when you start up the Control Program, check your installation connections and make sure the Machining Center is turned on.

After checking the connections, you can select Retry to start the program. Otherwise, you can select Simulate to use the Control program without communicating with the Machining Center, or select Quit to exit the Control Program and return to DOS.

If you are experiencing other start-up problems, check the items listed below.

- Are the computer and monitor plugged into a wall outlet? Is there power to the wall outlet?
- Is the monitor correctly connected to the computer?
- Are all interface connections secure?
- Are the monitor and computer turned on?
- Have you specified the correct directory?
- If you are using a mouse, was the mouse driver software loaded before the proLIGHT Control Program software?
- Do you have enough memory on the computer to run the Control Program? Make sure there are no other programs running before starting up the Control Program.

If you still experience start up problems after checking the above items, refer to your *PC Owner's Manual* and the *DOS Reference Guide* for additional information on loading software.

Safely Running the Machining Center

Like any other power tool, the proLIGHT Machining Center is a potentially dangerous machine if operated in a careless manner. The importance of safely operating the proLIGHT Machining 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 Appendix C.

Safety Rules

All operators of the proLIGHT 2000 Machining Center should practice the following safety rules.

Remove Adjusting Keys and Wrenches
Make a habit of checking that keys and adjusting wrenches are removed from the Machining 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.

For the proLIGHT 2500:

Use only tools rated for use with 42,000 rpm spindles or higher.

Never use shank tools or any tool with a diameter larger than 3/8" with the proLIGHT 2500.

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

Secure the Workpiece
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 Machining Center bed, cross slide, or stock on start up.

Tighten All Holding, Locking and Driving Devices
Tighten the work holders and tool holders. Do not over-tighten these devices. Over-tightening may damage threads or warp parts, thereby reducing accuracy and effectiveness.

Making Emergency Stops

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

You'll have to place a Pause command in your part program. Once the pause is executed, push in the Emergency Stop button and open the Machining Center shield.

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

Before you run the proLIGHT Machining 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 proLIGHT Machining Center: by pressing the emergency stop button, by pressing a key on the computer keyboard, by activating one of the limit switches, or by activating the safety shield interlock switch.

Stopping with the Emergency Stop Button

There is an emergency stop button located on the front panel of the proLIGHT Machining Center; it has an oversized red cap. Before power can be applied to the proLIGHT Machining 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 Machining Center.

In the event that a tool crashes into the workpiece, you can immediately kill power to the Machining Center by pushing in the emergency stop button. Pushing in the emergency stop button terminates the part program. Wait until the Machining 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), press the Return (or Enter) key on the computer keyboard to exit the program. 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 (see Section 3).

Stopping with the Computer Keyboard

To stop the part program with the keyboard, press the Space Bar, or the Q key. In the Manual Control Panel, press the ESC key or the Space Bar to stop machining. The machining stops immediately and the cutting tool remains in position. To restart the program from a keyboard-generated emergency stop, press Return on the keyboard. Select Run from the Main Menu to begin the operation again.

Stopping with a Limit Switch

The proLIGHT Machining Center is equipped with limit switches to sense the end of travel. If the table travel exceeds the end of travel on any axis, a limit switch is activated and shuts down machine operation.

Once a limit switch is activated, the tool must be jogged away from it using the Manual Control function of the Control Program (see Section 3). To move the cross slide away from the limit switch, you must jog it in the opposite direction. Each axis has a positive and a negative limit. If a *positive* limit is hit, you must jog away from it in a *negative* direction. If a *negative* limit is hit, you must jog away from it in a *positive* direction. If both a positive and a negative limit are hit, you can jog off them in any direction on any axis.

If the cross slide comes close enough to the end of travel on any axis to activate a limit switch, the following procedure must be followed to restore normal operation.

1. Select Manual Control from the Run Menu.
2. Press the appropriate jog key on the keyboard, or click on the jog keypad, to move the cross slide away from the triggered limit switch.
3. Press the ESC key to return to Edit Mode.
4. Check your initial machine set up to make sure it was done correctly.

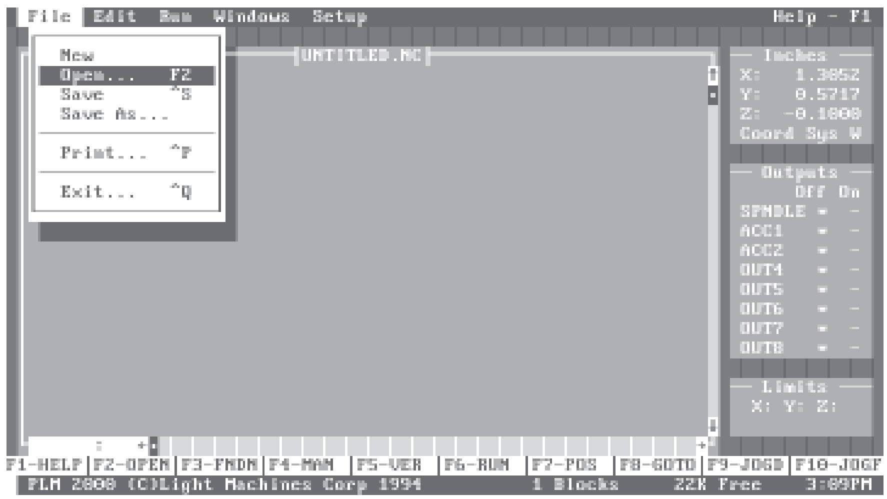
Running a Sample NC Program

> **WARNING**
>
> Do not attempt to operate the proLIGHT Machining Center without reviewing all of the safety precautions set forth in Appendix C.

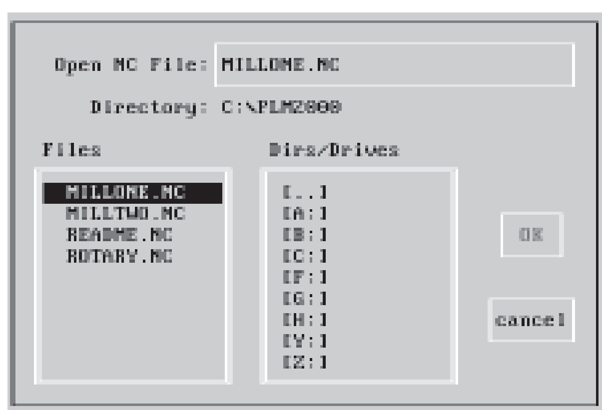
There is a sample program, MILLONE.NC, on your proLIGHT master disk. When you installed the proLIGHT software it should have been copied into the PLM2000 directory along with the other files. This program is meant to machine a 3"x2" piece of machinable wax.

Opening the Sample Program

1. Select Open from the File Menu.

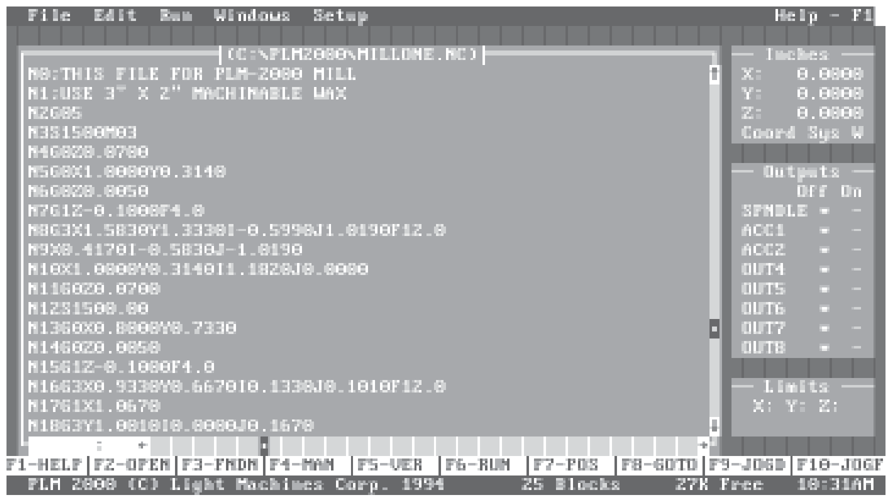


A dialog box appears with a list of the files in the PLM2000 directory.



2. Select and open the MILLONE.NC file.

   The NC program appears in the Edit window. The message bar on the bottom of the screen shows you the number of blocks in the program and how much memory is free.



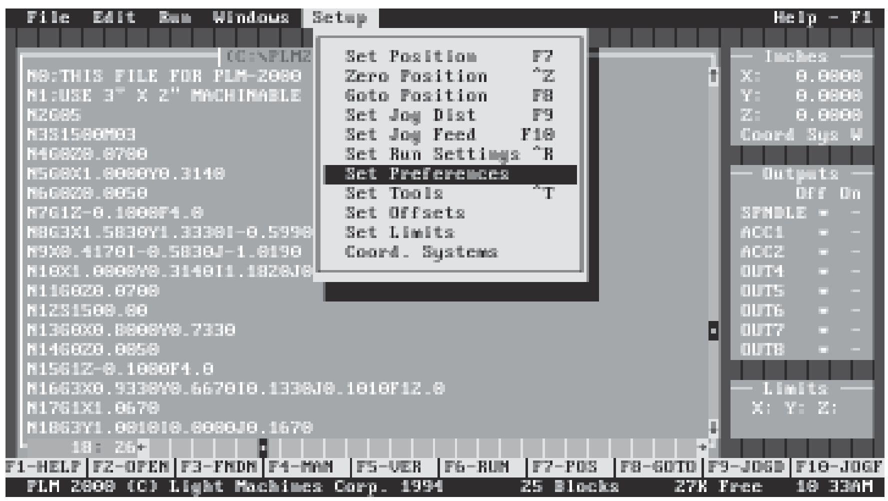
Performing a Dry Run of the Program

Before you run your part program for the first time, you should perform a dry run (run the program with no stock mounted) to make sure that all of the Machining Center movements make sense and that the tool is in no danger of striking any fixtures or crashing into the cross slide.

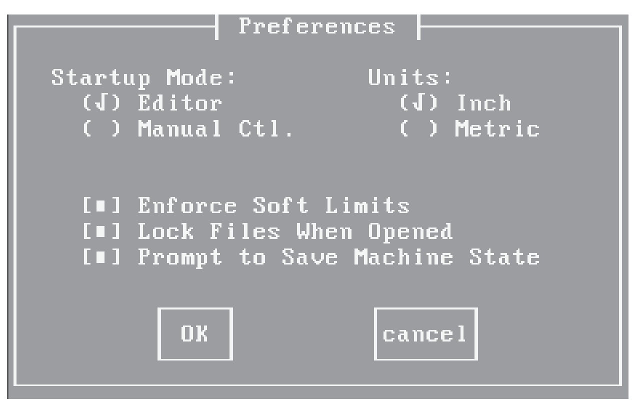
Begin with the spindle speed turned down. The vise or other work holding device should be mounted to the cross slide and the tool should be mounted in the spindle.

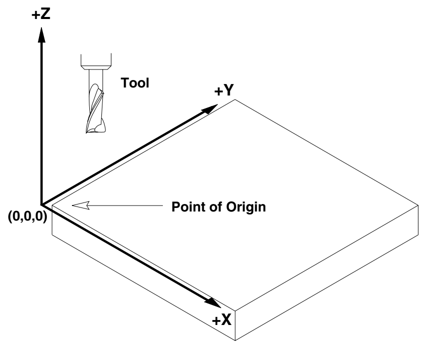
1. If MM appears in the Position window, select Set Preferences from the Setup Menu and change the Units to Inch.

To change the measurement mode from millimeters to inch (or vice versa) select Set Preferences form the Setup Menu.



When the Preferences dialog box opens, simply click on the measurement mode you'd like to use, then click on ok.

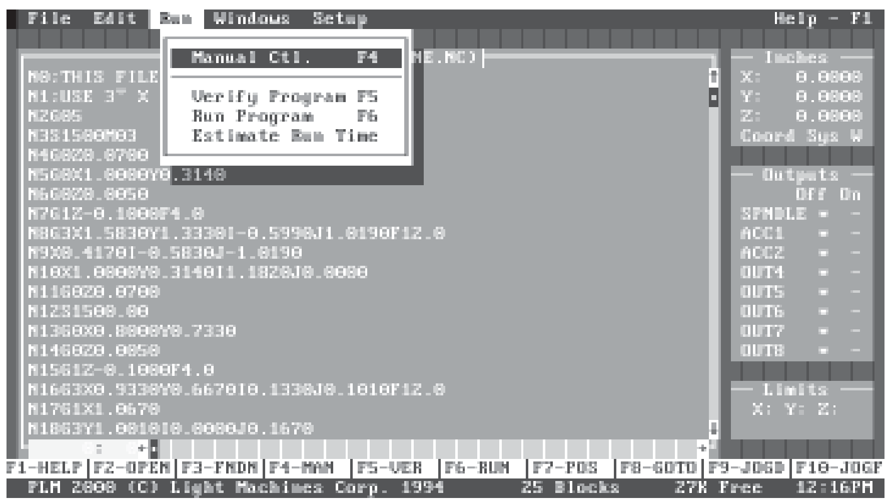




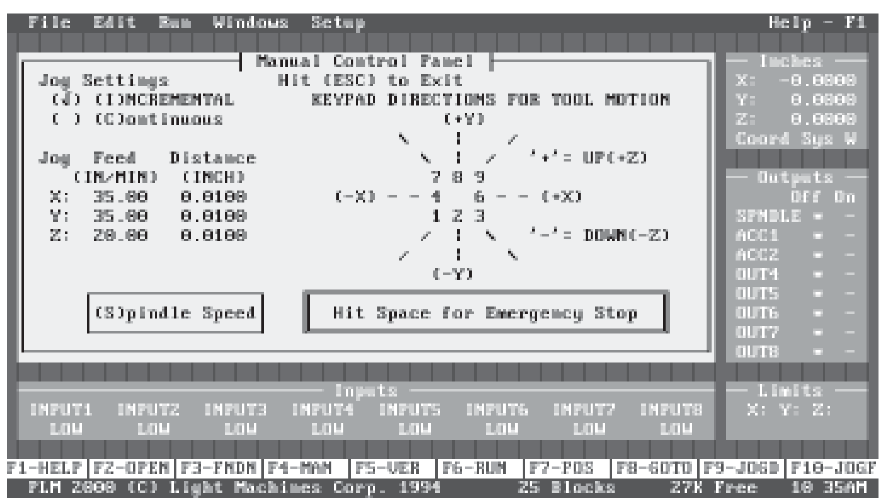
The zero point of origin on the X, Y, and Z axes.

2. Select Manual Control from the Run Menu and jog the spindle to the approximate X and Y zero points, then jog to approximately the zero point of the Z axis. (See Section 3 for instructions on using the Manual Control Panel.)

To move the tool to the 0,0,0 point on the workpiece, first select Manual Control from the Run Menu.

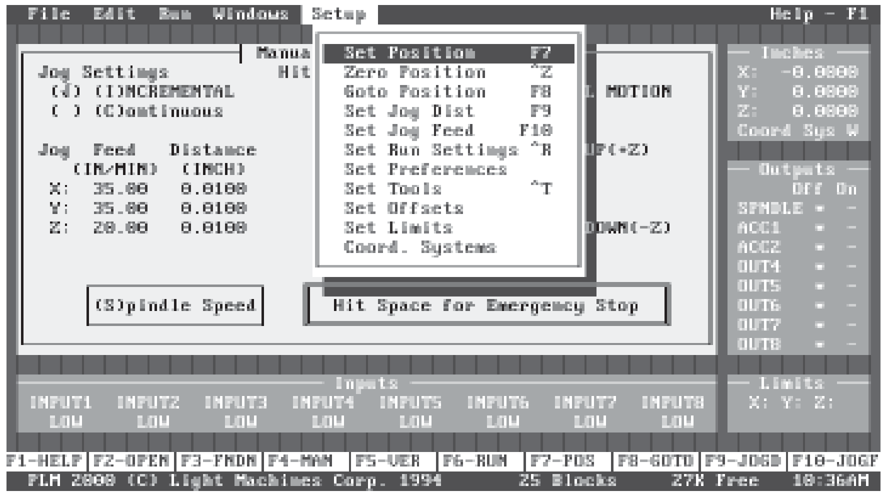


Click on the Jog Keypad with the mouse or use the corresponding keys on the keyboard to move the tool.

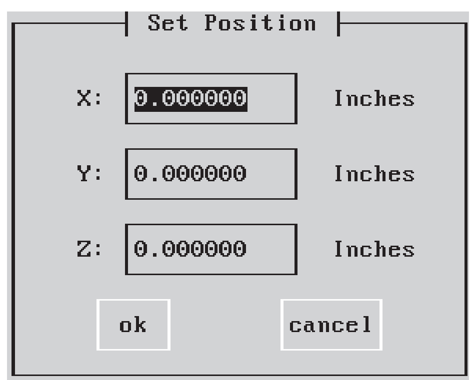


3. Select Set Position from the Setup Menu and set the X, Y and Z positions to zero.

To set the position for all three axes to zero, first select Set Position from the Setup Menu.

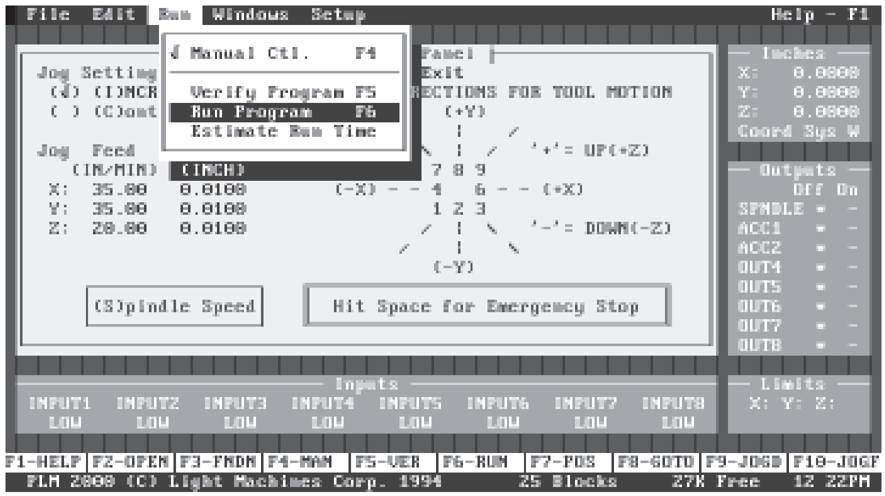


When the Set Position dialog box appears, enter zero for each of the axes, then click on Ok.

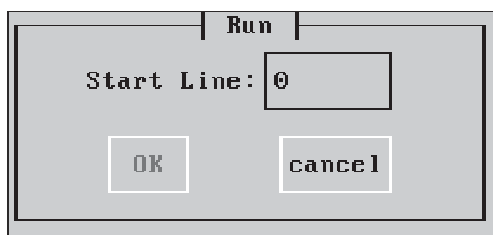


4. Review the safety checklist in Appendix C of this guide. Make sure that all of the safety guidelines have been met.

5. Select Run Program from the Run Menu.

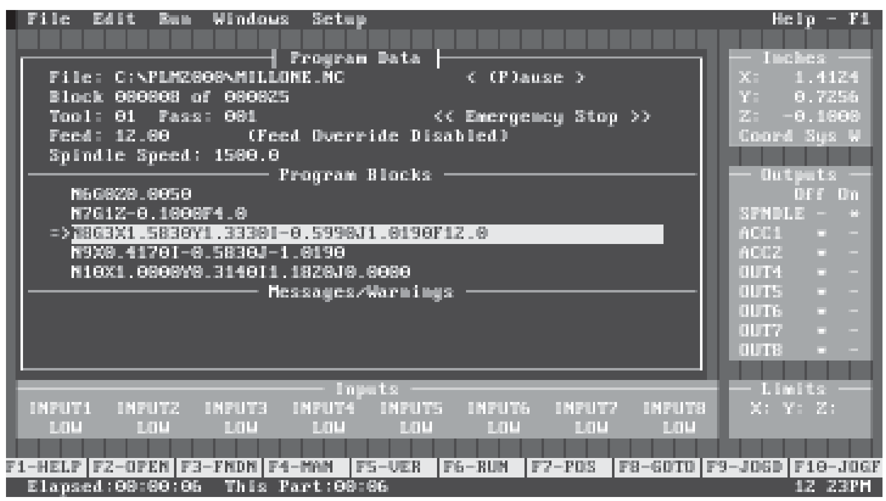


7. A dialog box appears. The default starting block is block zero, so just press Return.



8. The Run Program screen appears and the program pauses with the G05 code in block N2. Press Return to continue the program.

Watch the Run Program screen and the Machining Center motions while the program runs. Be prepared to press the Space Bar or the Emergency Stop button on the Machining Center if necessary. If any errors occur, you must stop the program, check your set up and dry-run it again.



Press Return to run the program.

9. When the program has been through a successful dry run, press the Emergency Stop button.

10. Open the safety shield and mount the 3" x 2" machinable wax stock.

Running the Program

CAUTION:

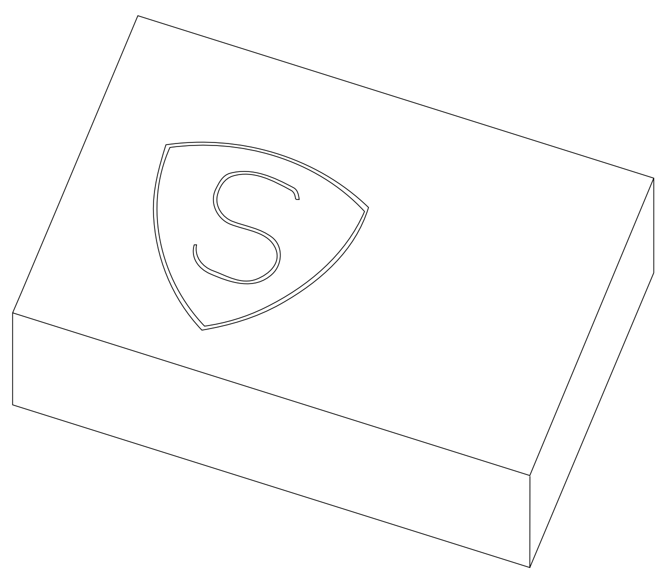
Allow the proLIGHT 2500 to warm up before machining. Run the spindle between 4200-6000 rpm (70-100Hz) for ten minutes before attempting higher spindle speeds. Failure to allow the machine to warm up may result in premature bearing failure.

Before executing the MILLONE.NC program, check that all safety precautions have been taken. The Machining Center safety shield should be in the upright position, and you should be wearing safety glasses. If anything goes wrong, immediately press the Space Bar on the computer keyboard, or the Emergency Stop button on the Machining Center to stop the operation. A safety checklist has been provided in Appendix C of this guide. Post a copy of this checklist near the Machining Center and review it before you run any NC program.

To run the program:

1. After reviewing the Safety Checklist, select Run Program from the Run Menu.
2. Make sure that zero is selected as the start block, and press Return to begin running your program.
3. After the part is machined, press the Emergency Stop button before opening the safety shield and removing the finished part.

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



Section 5: CNC Programming Codes

The Elements of an NC Part Program

Part programs generally incorporate two types of instructions: those which define the tool path (such as X, Y 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*). There are many categories of address characters used in NC part programs for the proLIGHT Machining Center (see Categories of NC Code).

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

```
N0G90G01X.5Y1.5Z0F1
```

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

N0 This is the block sequence number for the program. Block 0 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".

Y1.5 This specifies the Y axis destination position as 1.5".

Z0 This specifies the Z axis destination position as 0". The cutting tool will move to the absolute coordinate position (0.5,1.5,0).

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

There are many categories of NC code used for programming the proLIGHT Machining Center. Each category is identified by an address character that has a special meaning. Here is a list of the address characters supported by the proLIGHT Machining Center (in order of their appearance in a block).

/ Optional skip

\ Skip

% Incremental arc centers (Fanuc)

N Block number (for user reference only)

O Subprogram starting block number

G Preparatory codes

X X axis motion coordinate

U Incremental linear axis parallel to the X axis (for absolute dimensioning)

Y Y axis motion coordinate

V Incremental Y motion dimension for absolute dimensioning

Z Primary Z motion dimension

W Incremental Z motion dimension for absolute dimensioning

A A axis coordinate (see Rotary Positioner information in Section 6)

I Arc center, X axis dimension (circular interpolation)

J Arc center, Y axis dimension (circular interpolation)

K Arc center, Z axis dimension (circular interpolation)

R Arc radius for circular interpolation, and drilling start location

Q Peck depth for pecking cycle

H Input selection number/Tool length offset

D Compensation offset value

L Loop counter for subprograms and program cycle counter

F Feed rate in inches per minute, or dwell time in seconds

S Spindle speed

T Tool specification

M Miscellaneous functions

P Subprogram reference number/Uniform scale multiplier

; Comment (open parenthesis can also be used)

$ Run program with absolute arc centers

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

Optional Skip (/)

The optional skip code allows you to optionally skip blocks of code as the NC program executes. Make sure to activate the Optional Skip parameter in the Run Settings dialog box, then place a forward slash (/) in front of the block you want to skip. With Optional Skip off, the slash is ignored and the block of code is executed. With optional skip on, the slash is recognized and the block of code is skipped. If you place a number after the optional skip code, the program executes that block every nth pass. For example: /5G28; Homes every fifth pass

Skip (\)

The skip code works like the optional skip, except that it works even if the Optional Skip parameter from the Run Settings dialog box is off.

Block Number (N Code)

The N code specifies the sequential number of the block in the NC program. Using the N code is optional; however, when you do use the N code, it must be the first character of each block in the program. Select Renumber from the Edit Menu to renumber and remove and N codes.

Subprogram Block Number (O Code)

The O code replaces the N code in a block when a subprogram begins, and marks the start of a subprogram. *Only the first block in the subprogram should contain an O code.*

Note: 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.*

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 supplied, whether to pause for operator intervention, and so on.

The G codes supported by the proLIGHT Control Program fall into a number of groups: the Interpolation Group, the Units Group, the Plane Selection Group, the Wait Group, the Canned Cycles Group, the Programming Mode Group, the Preset Position Group, the Compensation Functions Group, the Coordinate System Group, the Feed Functions Group, and the Polar Programming Group.

The Interpolation Group

These G codes are retained until superseded in the program by another code from the same group. Four interpolation G codes are supported:

G00 Rapid traverse
G01 Linear interpolation (default)
G02 Circular interpolation (clockwise)
G03 Circular interpolation (counterclockwise)

Units Group

There are two G codes to denote units of measure, G70 (inch) and G71 (metric). These codes tell the Machining Center which unit of measure you are using. They are placed at the beginning of the program before any moves are made. Only one of the two codes can be used per block. You can also use the Fanuc equivalents, G20 (inch) and G21 (metric).

Plane Selection Group

This group of codes allows you to select different planes for circular interpolation, rotation, and cutter compensation. G17 is the proLIGHT Control Program default. The supported Plane Selection Group codes are:

G17 Select the X, Y plane for circular interpolation. Use this code if you are switching back to the X,Y plane after circular moves on the X, Z or Y, Z planes. The arc center coordinates are given by I for the X axis and J for the Y axis.

G18 Select the X, Z plane for circular interpolation. Use this code to perform circular interpolation on the X, Z plane. The arc center coordinates are given by I for the X axis and K for the Z axis.

G19 Select the Y, Z plane for circular interpolation. Use this code to perform circular interpolation on the Y, Z plane. The arc center coordinates are given by J for the Y axis and K for the Z axis.

> **WARNING**
>
> **Do not** use G04 for tool changes in the middle of a program; use G05 or M06 for this purpose.

Wait Group

Wait Group codes apply only to the block in which they appear. Multiple Wait Group G codes should be placed on several different lines. The supported Wait Group codes are:

G04 Dwell (wait): Equals the value of the time (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.

G25 Wait until robot input goes high. Used in conjunction with H code, which specifies the input number. Used for robot synchronization.

G26 Wait until robot input goes low. Used in conjunction with H code, which specifies the input number. Used for robot synchronization.

G31 Linear move to specified coordinate. Stop short if specified input goes High (if H is positive) or Low (if H is negative). Move until an input is triggered or until a coordinate is reached.

G131 Stop Z axis motion on input (INROB1 ON) during digitizing. You should only use this code with Light Machines' digitizing package.

Canned Cycle Group

Canned cycle codes allow you to perform a number of tool motions by specifying just one code. For detailed information on using these codes, refer *Canned Cycle Programming* in this section. Canned cycles are retained until superseded in the program by another canned cycle code. These are the canned cycle codes supported by the proLIGHT Machining Center:

G80 Canned cycle cancel.
G81 Canned cycle drilling.
G82 Canned cycle straight drilling with dwell.
G83 Canned cycle peck drilling.
G84 Canned cycle tapping.
G85 Canned cycle boring.
G86 Canned cycle boring with spindle off (dwell optional).
G89 Canned cycle boring with dwell.

Programming Mode Group

Programming mode G codes tell the Machining Center which programming mode to use; G90 for absolute, or G91 for incremental. These codes remain in effect until superseded by each other. The default code on program start up is G90.

With absolute programming, all X, Y and Z coordinates are relative to the 0,0 point on the Machining Center. With incremental programming, each motion to a new coordinate is relative to the previous coordinate.

Preset Position Group

The preset position G codes move the tool to a predetermined position. They apply only to the current programming block. These are the preset position codes supported by the proLIGHT Machining Center:

G27 Check reference point: Compares reported position against zero to see if any position has been lost.

G28 Set reference point: This code moves the machine to its home position and sets the machine position to 0,0,0. The G28 code performs an automatic calibration of the axes.

G92 Preset position: This code works like the Set Position function under the Setup Menu (see Section 3). The X, Y 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 (see *Canned Cycle Programming* in this section.)

G99 Rapid move to point R (surface of material or other reference point) after canned cycle complete.

Compensation Functions Group

G39 Corner offset circular interpolation.

G40 Cancels cutter compensation.

G41 Left cutter compensation: Enables cutter compensation to the left of programmed tool path.

G42 Right cutter compensation: Enables cutter compensation to the right of programmed tool path.

G43 Tool length offset: Shifts Z axis in a positive direction by a value specified by an H code.

G44 Tool length offset: Shifts Z axis in a negative direction by a value specified by an H code.

G45 Increases the movement amount by the value stored in the offset value memory.

G46 Decreases the movement amount by the value stored in the offset value memory.

G47 Increases the movement amount by twice the value stored in the offset value memory.

G48 Decreases the movement amount by twice the value stored in the offset value memory.

G49 Cancels tool length offsets.

G50 Cancels scaling.

G51 Invokes scaling.

G68 Invokes rotation.

G69 Cancels rotation.

Coordinate System Group

Coordinate system codes allow you to create multiple parts by establishing multiple coordinate systems on one work piece. For instance, you can run a part program using a typical coordinate system (with the point of origin on the surface of the front left corner of the workpiece), then select another coordinate system which has its origin at a different point on the surface of the work piece.

There are seven coordinate system codes. One of these codes (G53) is used to rapid to specified coordinates, while the other six allow you to make up to six individual parts on the same work piece by specifying different coordinate systems for each part. The coordinate system codes are G54 (coordinate system 1), G55 (coordinate system 2), G56 (coordinate system 3), G57 (coordinate system 4), G58 (coordinate system 5) and G59 (coordinate system 6). For a more detailed explanation on using coordinate system codes, refer to Section 3.

Note: Do not place absolute and incremental commands in the same block. For example:

G90X1V1

will not produce the expected motions.

Polar Programming Group

These two codes, G15 and G16, allow you to perform operations based on polar coordinates. The polar coordinates are defined by X (radius) and Y (angle in degrees) when programming for the X,Y plane. For more information on using these G codes, refer to *Polar Programming* in this section.

X Axis Coordinate (X or U Code)

An X code specifies the coordinate of the destination along the X axis. The default measurement value is set using the Set Preferences command under the Setup Menu (refer to Section 3). You can use a U code while in absolute dimensioning to specify an incremental X motion.

Note: If no I, J or K code is specified, the system assumes a value of zero.

Y Axis Coordinate (Y or V Code)

An Y code specifies the coordinate of the destination along the Y axis. The default measurement value is set using the Set Preferences command under the Setup Menu (refer to Section 3). You can use a V code while in absolute dimensioning to specify an incremental Y motion.

Z Axis Coordinate (Z or W Code)

A Z code specifies the coordinate of the destination along the Z axis (spindle axis). The default measurement value is set using the Set Preferences command under the Setup Menu. You can use a W code while in absolute dimensioning to specify an incremental Z motion.

Note: Full circles (360º arcs) cannot be performed with an R code. Split the arc into two arcs or use center point (I, J and K) values for full 360º circles.

A Axis Angular Dimension (A Code)

An A code specifies the coordinate of the destination (in degrees) along the A axis. This code is used only when machining with a fourth axis (refer to Section 6).

X Axis Coordinate of Center Point (I Code)

The I code specifies the X axis distance from the start point of motion to the center point of the arc for circular interpolation.

Y Axis Coordinate of Center Point (J Code)

The J code specifies the Y axis distance from the start point of motion to the center point of the circle for circular interpolation.

Z Axis Coordinate of Center Point (K Code)

The K code specifies the Z axis distance from the start point of motion to the center point of the circle for circular interpolation.

Radius of Arc, Drilling Start Location (R Code)

As an alternative to using the center point of an arc (I,J,K), you can use the arc radius can. Use the same value for radius in both absolute and incremental modes. G02 or G03 specifies the direction of motion.

Positive values for R (radius) are specified for arcs up to 180 degrees. Negative values are used for arcs greater than 180 degrees.

The R code is also used in all canned cycles to specify a starting reference point for peck drilling. The point can be at the material surface or at another reference point.

Peck Depth (Q Code)

The Q code is used in canned cycles to specify the depth of each peck.

Input Selection Number (H Code)

Use the H code is in conjunction with the wait codes G25, G26; the G31 code; the tool length offset codes (G43, G44); and transmit codes M25 and M26 for interfacing with robots or other external devices. The default value is H1.

CAUTION:

Using multiple tools is an advanced operation, and should not be attempted by persons unfamiliar with using the proLIGHT Machining Center.

Compensation Offset Value (D Code)

The D code specifies a number from the Control Program's Offset Table. This number has a corresponding value from the table. Use the D code with cutter compensation and tool offset adjust codes.

Loop or Program Cycle Counter (L Code)

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 separate block from the motion commands.

The L code is a loop counter for subprograms, as a program cycle counter, and to specify tolerance with homing commands and arc resolution.

Feed Rate (F Code)

An F code specifies the rate of speed at which the tool is moved (feed rate) in inches per minute (ipm). You should set the feed rate to a low value for cutting operations (for example, F3 equals three inches per minute). You can assign the feed rate a value up to 150 ipm. Feed rate values are in millimeters per minute (mpm) when in metric mode.

Spindle Speed (S Code)

The S code is used to set the spindle speed from within the NC program. Spindle speed is specified in a block of code by the address character S followed by a parameter that represents the speed in RPM's. For example, S750 is the designation for a spindle speed of 750 RPM's.

Tool Selection (T Code)

T codes specify tool offset in multiple tool machining operations. Tools are specified in a block of code 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.

Miscellaneous Codes (M Codes)

M codes control Machining Center functions while the part program is running. They control the spindle On/Off switch, and clamp or unclamp the air vise. M codes can also be used to *chain* a second program to the end of a part program, or to repeat the program. Only one M code can be specified per NC block. These are the M codes supported by the proLIGHT Machining Center:

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 an M02 stop. 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.

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:* Used in conjunction with a T code to perform multiple tool operations. See Section 6.

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:* Closes air vise accessory concurrently with the motion specified in the program block; remains in effect until superseded by M11.

M11 *Unclamp ACC2:* Opens air vise accessory after the motion specified in the program block; remains in effect until superseded by M10.

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 path name for each file.

M22 *Write to file:* Outputs machine coordinates to a file. The proper format for using this code is: M22(*filename*). The first time the Control Program encounters an M22 code, it opens your specified file. You're basically naming the file as you create it. You must enclose the name of the file in parentheses for the Control Program to recognize it.

If the file name is followed by ",A" (e.g., test.nc,A), the Control Program does not delete previous information from the file, rather it appends the information to the end of the existing information

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, ( ).

M25 *Set robot output on (High):* Use for robot synchronization. Used in conjunction with H code to specify output number.

M26 *Set robot output off (Low)*: Use for robot synchronization in conjunction with H code to specify output number.

M30 *Program stop:* Same as M02.

M47 *Rewind:* Restarts the currently running program; takes effect after all motion comes to a stop.

M98 *Call to subprogram*

M99 *Return from subprogram:* Returns you to the block following the initial M98 command.

Go to: Used with P code. P code defines N or O code destination. Goes to first occurrence of N or O code within the main program.

M111 *Arc as Linear Segments:* Causes the proLIGHT 2000 to make arcs as linear segments. Use with Helical Interpolation.

M112 *True Arcs:* Causes the proLIGHT 2000 to make true arcs. Opposite of M111.

M122 *Output current position to file:* Writes current X, Y, and Z position to a data file. Used with G131 for digitizing with Light Machines' digitizing package. Similar to M22.

Subprogram Reference Number (P Code)

The P code is used to reference a subprogram and immediately follows an M98 command. See Section 3 for more detailed information.

Comments (; or ( )

In addition to the other NC codes, the proLIGHT system allows you to add *comments* to your NC blocks. A semicolon ";" or open parenthesis "(" at the end of an NC block indicates that a comment follows. The comment must come after all other NC codes in the block. Comments are ignored when the part program is run.

Here is an example of an NC block with a comment:

```
X0Y0Z0;MOVE TO ZERO POINT
```

The comment tells us that the X, Y and Z codes in this block command the cutting tool to move to the zero point (coordinate 0,0,0).

Comments can be combined with the G05 pause code and M06 tool change code to display messages to the operator during program execution. Here is an example of an NC block with a pause coded comment: M06T2(INSTALL T2, 1/4" END MILL

This NC block would cause a program pause with a message to the operator to change the tool in the spindle.

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.

Run Program with Absolute Arc Centers ($)

The proLIGHT Machining Center interprets arc centers as incremental values. If you place a dollar sign ($) at the beginning of your NC program, the software will interpret arc centers as absolute instead of incremental. The command only applies to the NC program in which you place it.

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), Y (V), Z (W), A, I, J, K, R, Q, H, 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

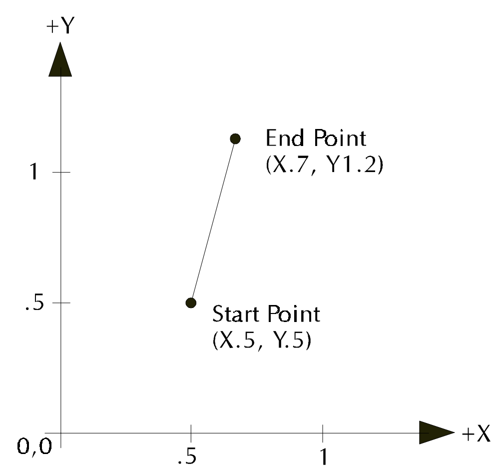
unless a new code appears.

This does *not* apply to: N words, I, J, 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.

Typical Tool Movement Using Linear Interpolation

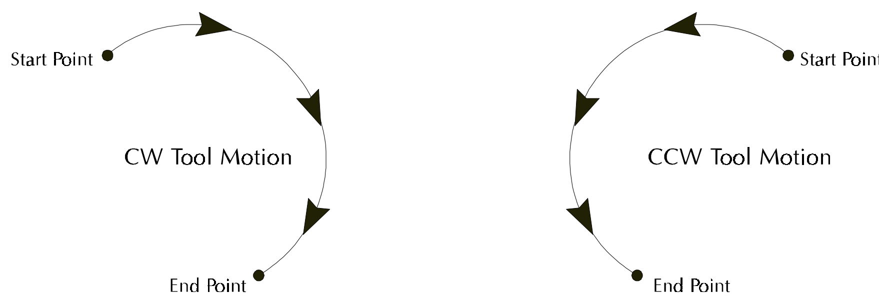


7. Part programs should reference the zero point with Z0 at

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, two Y 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 Ma-

G02 (CW) and G03 (CCW) Cutting Paths



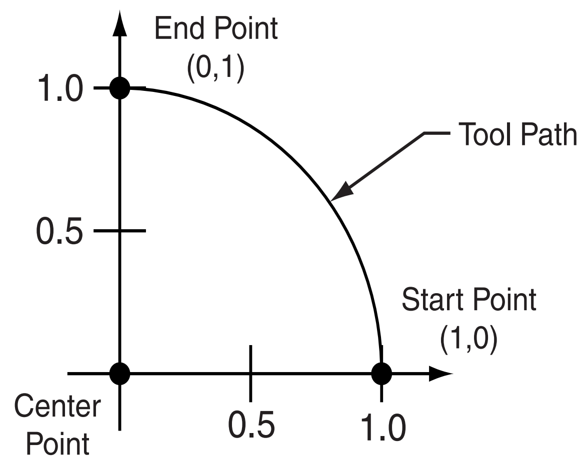
chining Center to see if the tool movements are logical.

9. The first portion of a part program should turn on the spindle and establish the feed rate.
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.

Linear Interpolation Programming

Linear interpolation is the movement of the tool in a straight line from its current position to a coordinate location specified by an NC block. Here's a typical block of NC code using lin-

Typical Tool Movement Using Circular Interpolation



ear interpolation: N5G90G01X.7Y1.2F2

Broken down into individual words:

N5	The block sequence number is 5
G90	Coordinates are given using absolute dimensioning
G01	Linear interpolation is specified
X.7	X axis coordinate of end point = .7
Y1.2	Y axis coordinate of end point = 1.2
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.5, Y.5, the tool movement generated by the above block is something like this:

An equivalent movement can be achieved with incremental dimensioning (G91): N5G91G01X.2Y.7F2

Circular Interpolation Programming

Circular interpolation moves the cutting tool along an arc

Note: The Control Program automatically switches to M111 when it encounters helical interpolation.

from the starting point specified in one block, to an end point specified in the next block. The curvature of motion is determined by the location of the center point (I, J, or K), which must also be specified in the second NC block.

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.

Here are two typical blocks of NC code using circular interpolation:

```
N9G90X1Y0;SET START POINT
N10G03X0Y1I-1J0F2;COUNTERCLOCKWISE TO X0,Y1
```

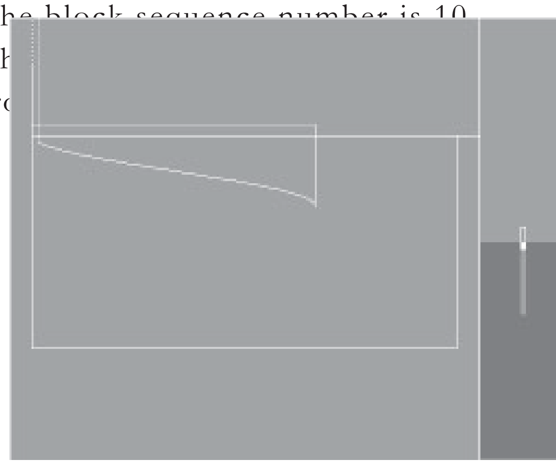
The first block defines the starting point. The second block defines the end point and the center of the arc. Broken down into individual words, the second block reads:

In the example program above, the tool plunges into the workpiece then makes the helical interpolation move to the back corner of the stock (X0Y2Z0).

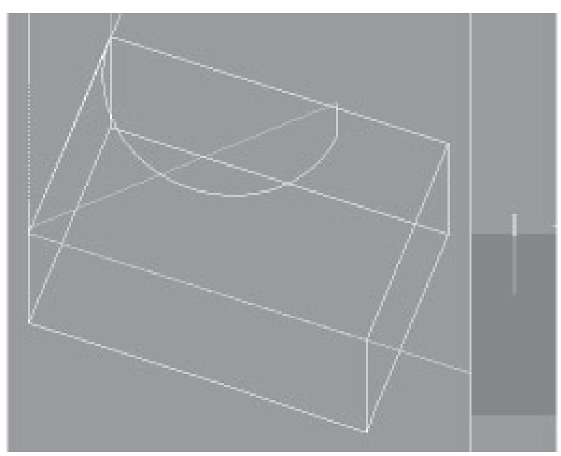
Here the movement is shown in Front View

N10 The block sequence number is 10

G03 Th … wise direction fr …) coordi-



Here the movement is shown in Isometric View



nates; center point of arc is specified by (I, J) coordinates

X1 X axis coordinate of end point = 0

Y0 Y axis coordinate of end point = 1

I-1 I coordinate of center point of arc = -1

J0 J coordinate of center point of arc = 0

F2 Feed rate is 2 inches per minute

The tool path generated by the preceding block is something like this:

An equivalent movement can be achieved with incremental dimensioning (G91):

```
N9G91X1Y0;SET START POINT
N10G03X-1Y1I-1J0F2
```

In this NC block, the X and Y values are the distance the tool is to move from its current position. In both cases, the I and J values are equal to the X and Y distance from the start point to the center point.

Circular Interpolation on Other Planes

To perform circular interpolation on a plane other than the X, Y plane, use a G18 code to select the X, Z plane, or use a G19 code to select the Y, Z plane. This feature is rarely used in manual part programming, but may be used by CAM systems to generate surfaces of revolution. The G17 code is used to return to the X, Y plane. An example of circular interpolation on the X, Z plane is:

```
N9X0Z0
N10G90G18G03X0Z1I0K.5F2
```

In this NC block, the X and Z values are the destination position of the tool. The I and K values are the incremental location of the center point of the curvature of motion.

Helical Interpolation Programming

Helical interpolation is performed when the axis not used in circular interpolation is commanded to move. For example (assuming a start point of 0,0,0): N10G90G03X0Y1Z1I0J.5F2

This block would cause the Z axis to move at a constant feed to Z1 while the X and Y axes move in a circular path, resulting in a helical motion. Helical interpolation works with circular motion on the X,Z and Y,Z planes as well.

Here is an example of an NC program using helical interpolation.

```
G90M03S1500
G0X0Y0Z0.070
G0X2Y2
G1Z-0.5F10
G02X0Y2Z0I-1J0F10
M02
```

Rapid Traverse Programming

On the proLIGHT Machining Center, the rapid traverse code (G00) can move the tool at the maximum available feed rate (150 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.

Here's a sequence of typical NC blocks using rapid traverse:

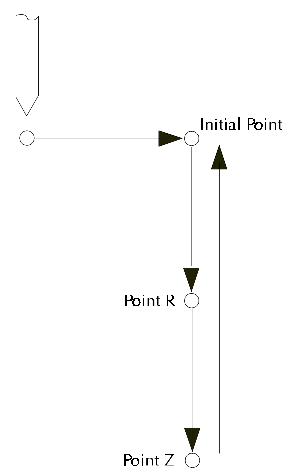
G90G01X1F2; MOVE IN A STRAIGHT LINE TO X = 1 AT 2 IPM

G00X2; RAPID TRAVERSE TO X=2

X3; RAPID TRAVERSE TO X=3

G01X4; MOVE IN A STRAIGHT LINE TO X=4 AT 2 IPM

If we specified a G99 here instead of a G98, the tool would rapid to point R instead of the initial point.



Canned Cycle Programming

Canned cycle commands allow you to put many tool movements together in one code instead of using many codes. Canned cycles are typically used for repetitive operations to reduce the amount of data required in an NC program.

The proLIGHT Machining Center recognizes the following canned cycles codes:

G80 Canned cycle cancel

G81 Straight drilling

G82 Straight drilling with dwell at bottom

G83 Peck drilling

G84 Tapping cycle

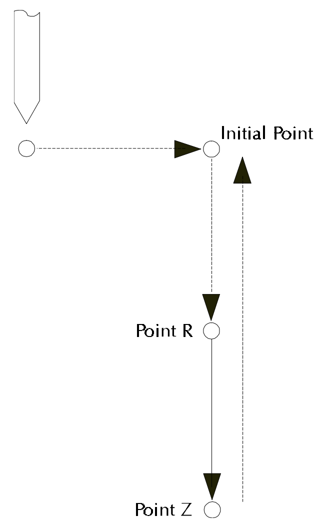
G85 Boring cycle

G86 Boring cycle with spindle off (dwell optional)

G89 Boring cycle with dwell

These codes are used in conjunction with canned cycle

If we used a G99 with the G82 instead of a G98, the tool would rapid to point R instead of the initial point.



codes:

G98 Rapid to initial position after canned cycle complete; this is the system default

G99 Rapid to point R after canned cycle complete

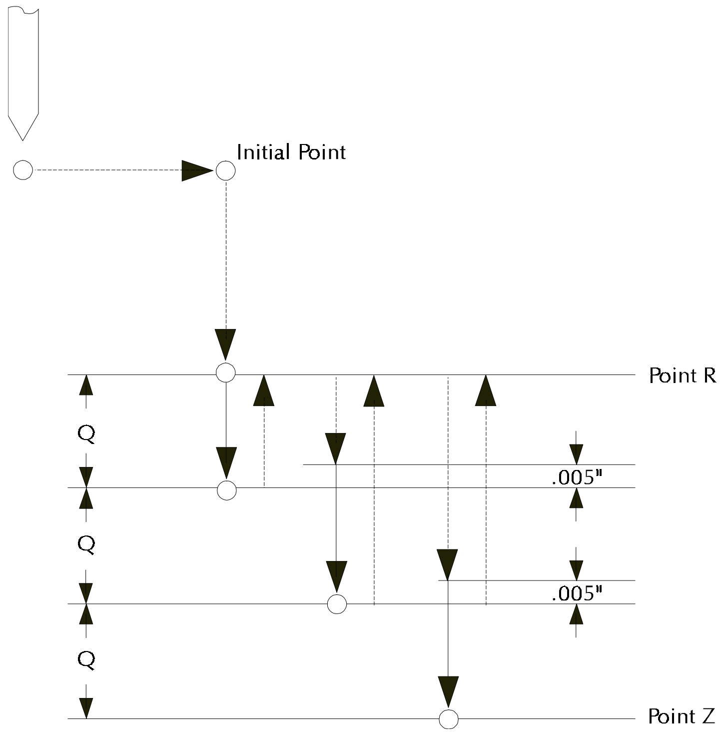
K Code Specifies the number of repeats. The default is 1. When K=0, drilling data is stored.

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.

R Code Used for specifying a starting reference point for peck drilling. The point can be at the material sur-

G99 returns the tool to point R. Use G98 with the G83 to return the tool to the initial point at the end of the canned cycle.



face or at another reference point.

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 drilling 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

The G81 code performs straight drilling operations. By specifying an R value of zero, the tool will return to the initial point after drilling to point Z. Here is a sample G81 program.

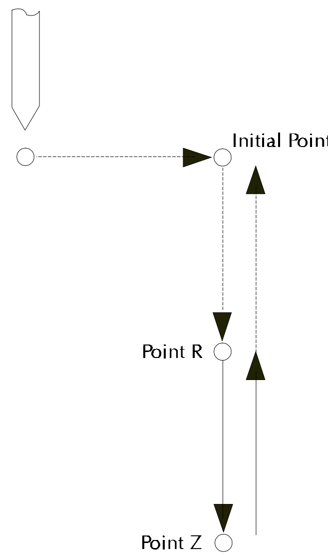
```
G0X1Y1Z.1;RAPID TO 1, 1, .1
G81G98Z-.5R0F2;DRILL TO DEPTH OF -.5, RAPID TO INITIAL POINT
G80;CANCEL CANNED CYCLE
M2;END PROGRAM
```

This program will generate tool motions similar to this:

More than one canned cycle can be accomplished by specifying only X and Y coordinates. For example:

```
G0X1Y1Z.1;RAPID TO 1, 1, .1
```

If G99 were specified instead of G98, the tool would not rapid back to the initial point. It would remain at point R.




```
G81G98Z-.5R0F2;DRILL TO DEPTH OF -.5, RAPID TO INITIAL POINT
X.5Y1;PECK AT NEW X,Y COORDINATES
X.25Y1;PECK AT NEW X,Y COORDINATES
G80;CANCEL CANNED CYCLE
M2;END PROGRAM
```

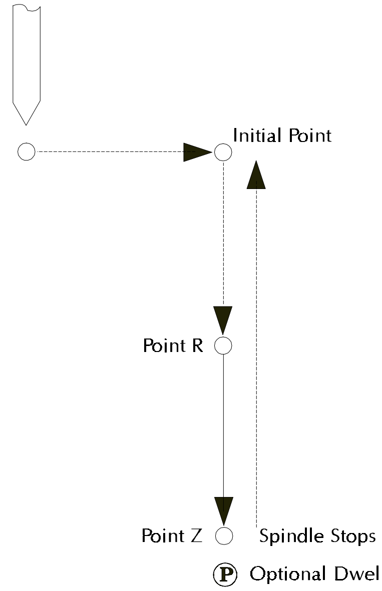
If a dwell is not specified (P code not used), a G05 pause is executed after the spindle stops at the bottom of the hole.

Using G82

A G82 works just like a G81, except it is used when you wish to incorporate a dwell (P code) at the bottom of the hole (point Z). A block of code utilizing the G82 and P code, and the tool motion it creates are shown below.

```
G0X1Y1Z.1;RAPID TO 1, 1, .1
```

If G99 were used with the G86, the tool would not rapid back to the initial point. It would go to point R.



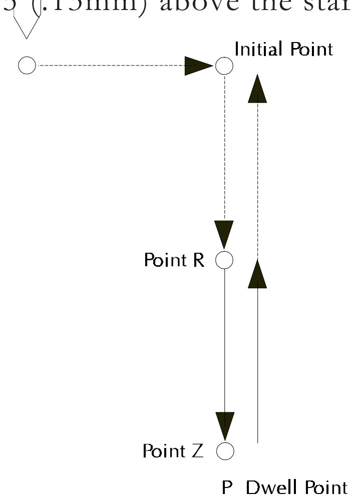
G82G98Z-.5R0P5F2;DRILL TO DEPTH OF -.5, RAPID TO INITIAL POINT AFTER A DWELL OF FIVE SECONDS

G80;CANCEL CANNED CYCLE

M2;END PROGRAM

Using G83

The G83 code is used for peck drilling cycles. By adding a Q depth to the code block, you can specify drilling increments. For instance, the following code will peck drill to a depth of -.5 in .1 increments. The tool will rapid back to point R after each peck drill. Also, before each peck the tool will rapid to .005 (.13mm) above the start point.



G0X1Y1Z.1;RAPID TO 1, 1, .1

G83G99Z-.5R0Q.1F3;PECK DRILL TO DEPTH OF -.5, RAPID TO R

G80;CANCEL CANNED CYCLE

M2;END PROGRAM

Using G84

A G84 is used for tapping threads. You specify the depth of the tapped hole. When the tap reaches that depth, it is pulled out in at a rate 1.6 times the rate of insertion (60% faster). The G84 tells the computer to calculate the insertion/extraction ratio. You must use a tapping head with a reversing mechanism when using G84.

Note: The L code is also used as a program cycle counter. For instance, if the last block 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.

G0X1Y1Z.1;RAPID TO 1, 1, .1

G84G98Z-.5R0F2;TAP TO DEPTH OF -.5, RAPID TO INITIAL POINT

G80;CANCEL CANNED CYCLE

M2;END PROGRAM

Using G85

A G85 specifies a boring cycle. After the tool plunges, it retracts at the same feed to point R. This sometimes gives a better surface finish on the hole. Then the tool rapids to the initial point.

G0X1Y1Z.1;RAPID TO 1, 1, .1

G85G98Z-.5R0F2;BORE TO DEPTH OF -.5, RAPID TO INITIAL POINT FROM POINT R

G80;CANCEL CANNED CYCLE

M2;END PROGRAM

Note: You can also perform an M99P"block number" at the end of the subprogram. This returns to the main program at the specified block. It's like a return with a go to.

Using G86

A G86 works like a G82 except the spindle stops at the bottom of the hole. The dwell (optional) allows the spindle to come to a complete stop before the tool rapids back to the initial point.

G0X1Y1Z.1;RAPID TO 1, 1, .1

G86G98Z-.5R0P5F2;DRILL TO DEPTH OF -.5, SHUT OFF SPINDLE, RAPID TO INITIAL POINT AFTER A DWELL OF FIVE SECONDS

G80;CANCEL CANNED CYCLE

M2;END PROGRAM

Using G89

The M98 calls the subprogram that begins on block O1000 (referenced by P1000) and tells it to repeat five times.

The G89 code works like the G85 except it utilizes a dwell at the bottom of the hole.

G0X1Y1Z.1;RAPID TO 1, 1, .1

G89G98Z-.5R0P5F2;BORE TO DEPTH OF -.5, PAUSE FOR FIVE SECONDS THEN RAPID OUT FROM POINT R

G80;CANCEL CANNED CYCLE

M2;END PROGRAM

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 proLIGHT Machining Center are:

M98 Call to subprogram.

M99 Return from subprogram.

P Code The P code is used to reference the first block 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 sub-programming. The computer executes the subprogram as many times as defined by the L code. For instance, if the code L5, the subprogram is executed five times. (optional)

O Code The O code replaces the N code on the first block of a subprogram.

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 block of the subprogram). The first block of the subprogram uses an O code instead of an N code for block numbering.

When the M99 is executed, the main portion of the NC program continues to execute from the block after the subprogram was called.

Subprograms can also be *nested* within other subprograms. This means that while a subprogram is being executed, it can call another subprogram. Subprograms can be nested up to 20 levels deep.

A Sample NC Subprogram

G05

M03S1000

; SAMPLE OF SUBPROGRAM

; USE 7.25 X 3.00 STOCK FOR VERIFY

G0X1Y1Z.1; RAPID TO 1, 1, .1

M98P1000L4; RUN SUBPROGRAM 1000 FIVE TIMES

G90G0X0Y0Z.1

M2; END OF MAIN PROGRAM

O1000; SUBPROGRAM TO MILL SQUARE AND MOVE TO NEXT POINT

G90G1Z-.1F2; PLUNGE AT CURRENT LOCATION

G91; INCREMENTAL COORDINATE

X1F5; FIRST MOVE, FEED RATE 5

Y1; SECOND MOVE

X-1; THIRD MOVE

Y-1; FOURTH MOVE

G90G0Z.1; RAPID UP ABOVE WORK

G91X1.5; RAPID TO START OF NEXT SQUARE

M99; RETURN FROM SUBPROGRAM

G28 **sets** a machine reference point, similar to the "set reference point" button in the "Setup/Select Offsets" dialogue box from the Setup Menu.

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 Y codes. The X code specifies the radius. The Y 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:

```
G0X0Y0Z0.07
M03
G16
G91X2Y0
M98P1L12
M2
O1Y30
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
G91X2Y0
M98P1L360
G15
M2
```

```
O1Y1
M99
```

In polar programming, the center point is the origin if you specify G90 and the radius (X code). The center point is the current center if you specify only the angle (X and Y codes). The center point is the current point if you specify G91 and the radius (X code).

Using the Homing Commands

Note: The 4th axis is not included in reference point related functions. Because the 4th axis has no limit or index switches, there is no way to determine a specific reference location.

The Homing commands allow you to establish a point of origin at the ends of travel on the Machining Center. The Machining Center then uses this point as a reference for all machine coordinate movements. The Machining Center can then move consistently to the same location when programmed to do so.

Using G28

The G28 code "homes" the machine: it moves the Machining Center's spindle, cross slide, and saddle to the ends of travel along each axis and sets the Machine Coordinate System to (0,0,0). This zero point is located at the positive limits of the Y and Z axes, and the negative limit of the X axis. Use G28 to automatically initialize the machine every time it's turned on. See the sample NC program below for one use of the G28 code.

Using G28 in an NC Program

Here is an example of an NC program using Homing Commands (the Homing Commands and related commands are in bold):

G28; HOMING THE MACHINE
M3S1000; SPINDLE MOTOR ON SPEED 1000
G54; USE COORDINATE SYSTEM ONE
G0X0Y0z0; RAPID TO O,O,O
G1z-0.070F5; CUTTING THE PIECE

```
G1X3
G1Y4.25
G1X0
Y0
G0Z0.070
M2
```

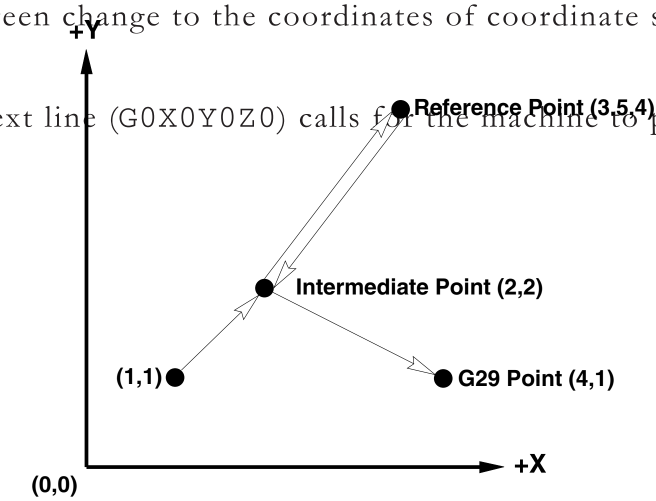
The G28 Code homes the machine and sets the Machine Coordinates and World Coordinates to zero. Use a G27 if you want to check the home position, but do not want to set that position to zero.

The G54 calls up coordinate system one, which contains the offset values relative to the machine's home position. These are the values you 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 (G0X0Y0Z0) calls for the machine to perform

In block N1, the tool moves from its current point to the intermediate point, then to the reference point.

In block N2, the tool moves from the reference point, through the intermediate point, to the new point established by the G29 code.



a rapid traverse motion from the Home position to point 0,0,0 in coordinate system one.

The remaining lines of code instruct the machine to cut a square in a piece of stock.

Using G28 Before Setting Soft Limits

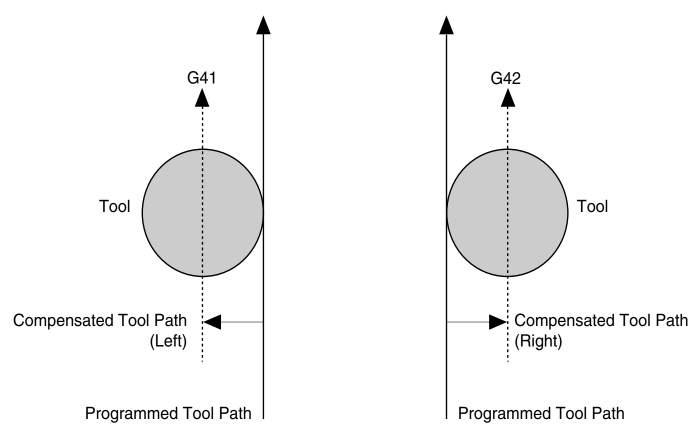
Remember, Soft Limits are based on Machine Coordinates. You should only enable Soft Limits after you have set the machine to the Home position using a G28 command. Otherwise, the Machine Coordinate system origin remains as it was the last time the machine was turned on, and the soft limits you have set will not be correct for this coordinate system.

IMPORTANT:

The Offset Table you use for Cutter Compensation D values is the same table you use for Tool Offset Adjust D values and Tool Length Offset H values.

Using G27

After you have set a reference point (using either the Set Reference Point button in the Setup Coord. System dialog box or a G28 code, you can use the G27 code to check the ac-



tual machine position against the expected machine position. This command causes the machine to perform a homing-like function, moving each axis independently from its current position to the reference point. The Control Program then

Cutter compensation mode begins when the following are met:

- A G41 or G42 code is commanded.
- The specified offset number is not 0.
- A move in any of the axes in the offset plane is commanded. The move is not 0.
- No arc commands are commanded in the start up block.

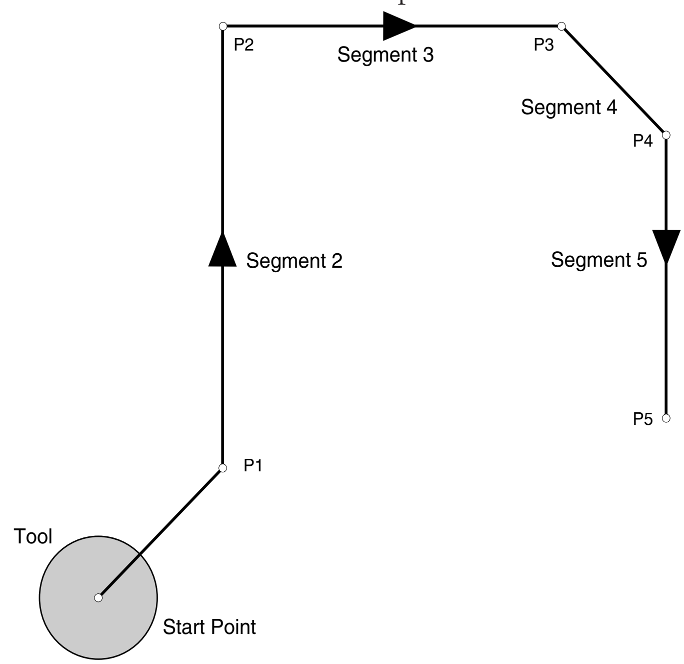
compares the current position to the one set by the G28. If the deviation is larger than a threshold you can specify in the SETUP file, the Control Program reports an error is after all axes are checked. If there is no deviation, the program continues.

G27 also takes an optional position specified by XYZ. This position is called the *intermediate position.* You do not need to specify all axes for the intermediate point, but for each axis that you specify the current coordinate for the intermediate point is updated to that value. Only axes that have specified coordinates move when you specify an intermediate point. For example, if the first intermediate point commanded is G27Z.6, the intermediate point motion is only to move the Z axis to .6. The machine first moves to the current intermediate point at rapid traverse, then performs the reference point check.

Using G29

The G29 code moves the tool at a rapid traverse rate to a co-

1. This is the programmed tool path before cutter compensation is enabled. The tool sits at the start point waiting for the first motion command.

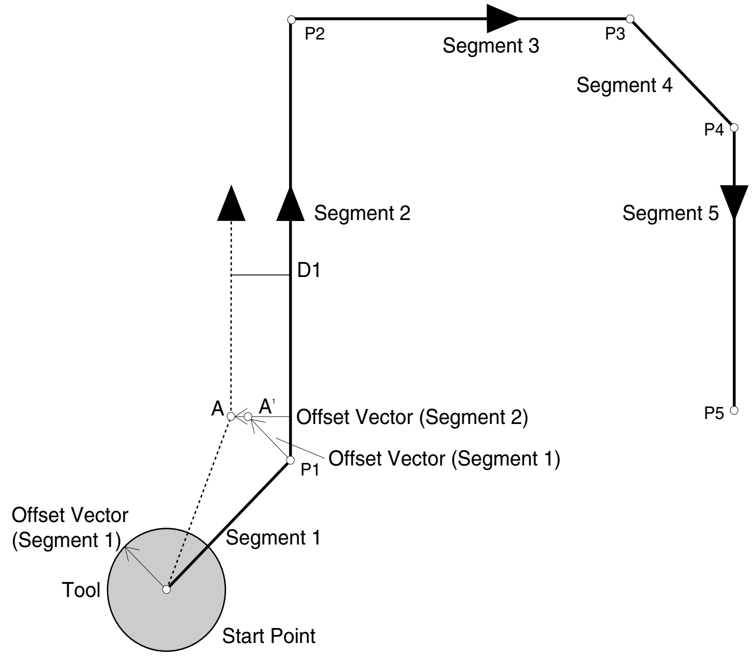


2. Because segment 1 begins before cutter compensation is enabled, the start point of segment 1 is on the original tool path. When cutter compensation is enabled, the Control Program creates Offset Vectors perpendicular to each segment of the programmed tool path to determine the compensated tool path.

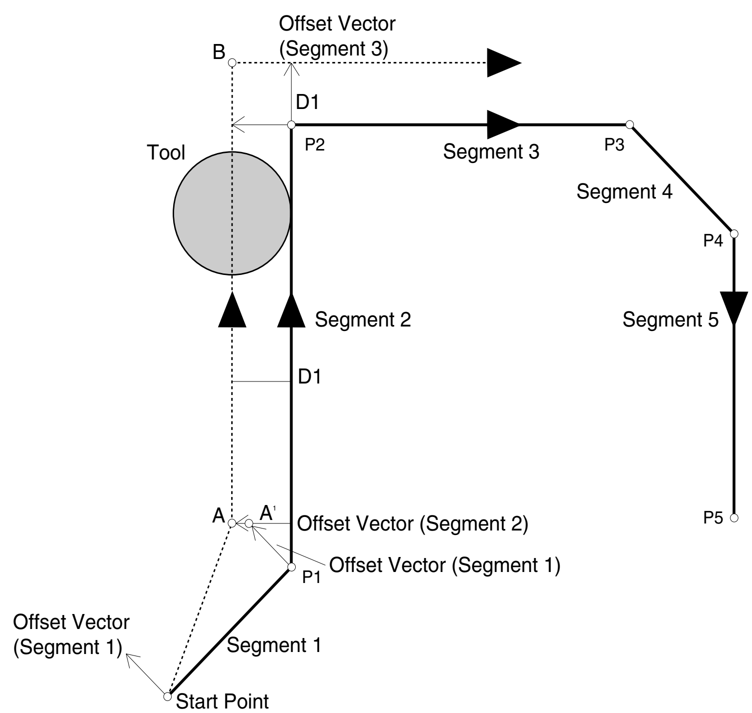
Point A[1] is located at the intersection of the Offset Vector of segment 2 and a point offset by the value of D on the Offset Vector of segment 1. Point A is located on a path parallel to the tool path, offset by the value of D.

Before beginning each motion, the Control Program looks ahead to the next motion in the NC code to determine the compensated end point of the first motion.

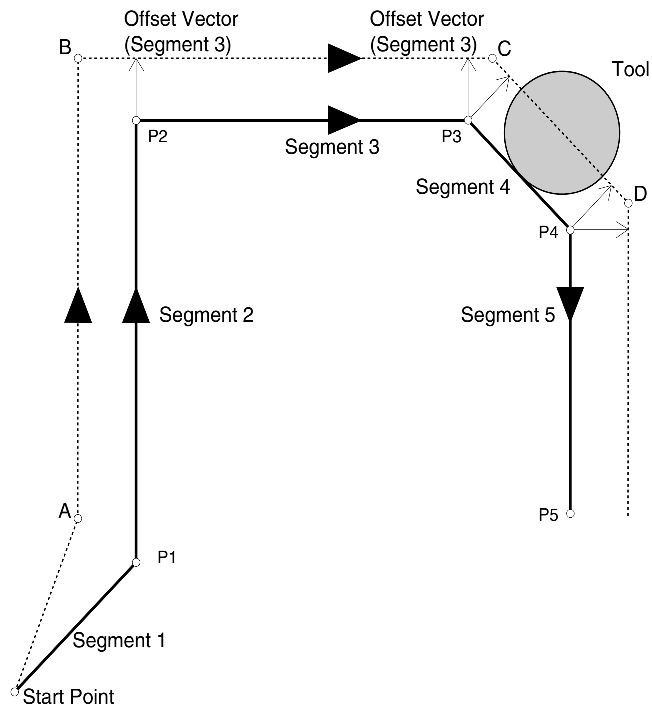
Using this method, the Control Program determines that point A is the end point of the first motion. The tool moves to point A on the compensated tool path.



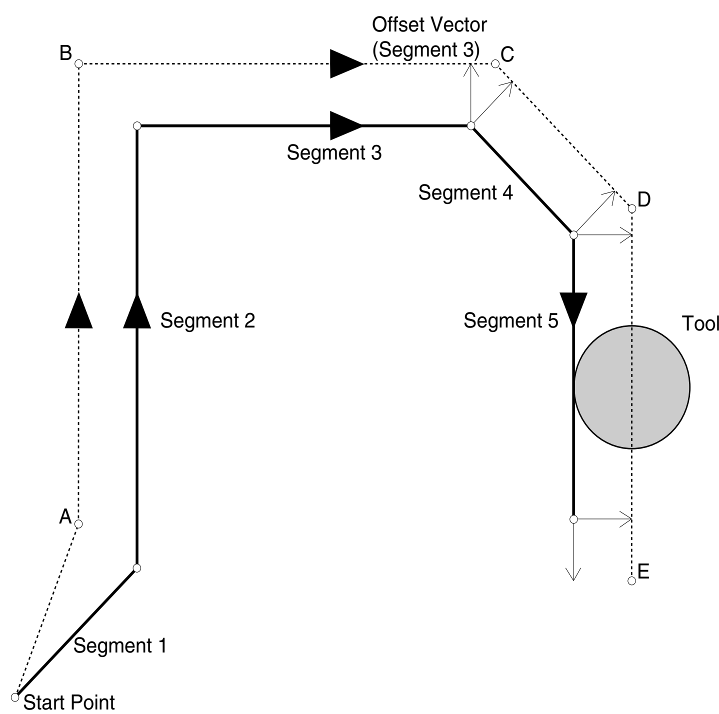
3. The tool moves to the end point of segment 2 (point B) on the compensated tool path. Point B is the intersection of lines drawn perpendicular to the Offset Vectors of segments 2 and 3 with an offset value of D from the programmed tool path.



4. The tool moves to the end point of segment 3 (point C), which is located at the intersection of lines drawn perpendicular to the Offset Vectors of segments 3 and 4 with an offset value of D. The tool then begins travel towards point D, the end point of segment 4.

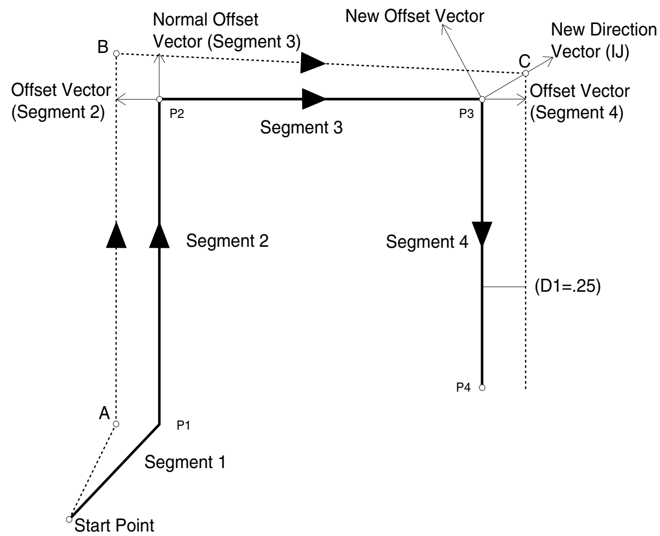


5. The tool moves to the end point of segment 5 (point E) on the compensated tool path. Point E is a point on the compensated path. (See *Canceling Cutter Compensation* in this section for more information.)



Note: When specifying an IJK vector, you must include the G41 or G42 code on the same line.

ordinate specified by XYZ. 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 XYZ point is relative to the intermediate point. If you have not specified an intermediate point, your specified XYZ point is relative to the current position. Use the G29 code after a G28 command 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.



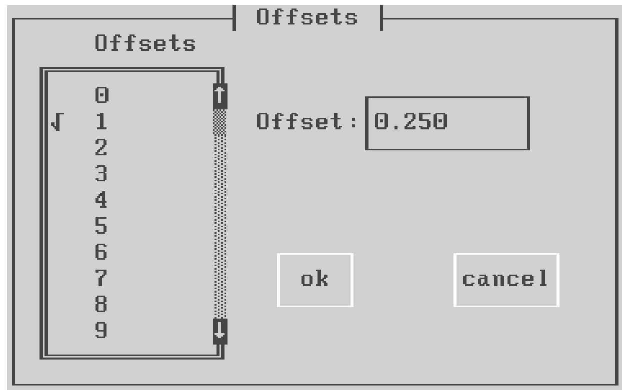
N1G28X2Y2Z-1; INTERMEDIATE POINT THEN HOME

N2G29X4Y1Z1; GO TO G29 POINT

Using Cutter Compensation

Note: You cannot assign a value to offset number 0. The offset value for offset number 0 is always zero.

Cutter compensation automatically adjusts the proLIGHT 2000 to compensate for variations in a cutting tool's radius. It



uses values from the Offset Table (tool radius values) to determine the compensation offset value. Use the following codes for cutter compensation:

G39 Inserts an arc at the corner of compensated path

G40	Cancels cutter compensation
G41	Invokes left cutter compensation
G42	Invokes right cutter compensation
D	Compensation offset value (Tool Radius Value)

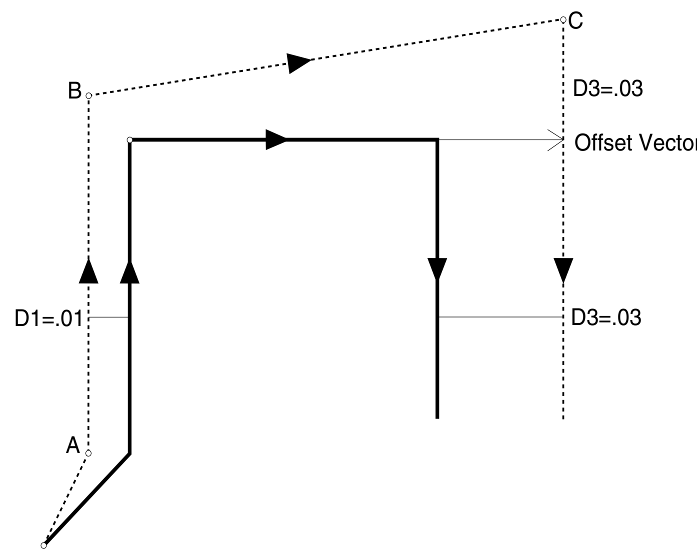
Determine left and right cutter compensation in relation to the direction of the tool path. Use left compensation when you need to move the tool to the left of the programmed tool path. Use right compensation when you need to move the tool to the right of the programmed tool path.

Select compensation offset values for D from the Offset Table. You can store 200 offset values in the table. Remember, however, that these offset values are the same values you use for adjusting tool length offsets.

In this example:

D1=.01

D3=.03

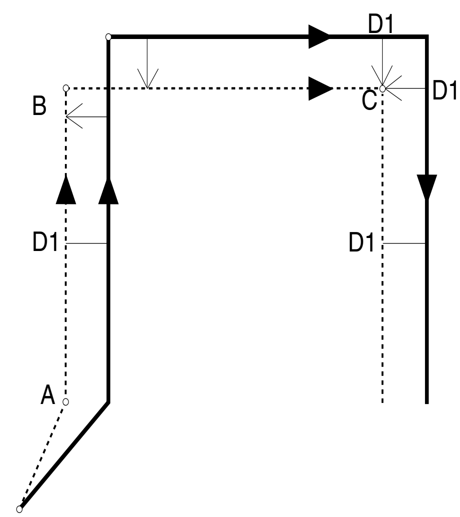


Starting Cutter Compensation (G41/G42)

You can start cutter compensation by inserting a G41 (for left compensation) or G42 (for right compensation) into your NC program.

In the example below, left cutter compensation is enabled and the compensation value is equal to offset value 1 from the Offset Table.

```
G0X0Y0
G91; INCREMENTAL
G41D1; CUTTER COMP ON
G1X.25Y.25; MOVE TO P1
G1X0Y1; MOVE TO P2
G1X.75Y0; MOVE TO P3
G1X.25Y-.25; MOVE TO P4
```




```
G1X0Y-.75; MOVE TO P5
```

The following illustrations show how the Control Program constructs the compensated tool path for the NC code above:

Cutter Compensation with IJK Vectors

Any G41 or G42 command can include an IJK vector, which defines the end point direction vector and the end point offset vector. For example, if, for segment 3 of the previous illustration, we had specified

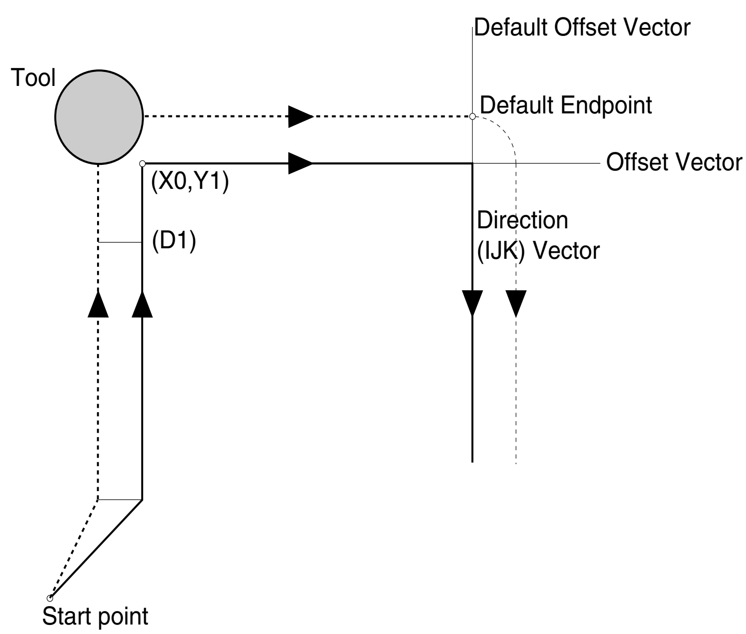
```
G41X.75I1J1
```

rather than just

```
X.75,
```

the compensated tool path would look like this:

The IJK Vector represents an incremental direction (the length of the vector is not important. For example, I1J2<=>I3J6. By default, the end direction vector is tangent to the segment.



Setting Cutter Compensation Offsets (D)

Select Set Offsets from the Setup Menu to set the value for D (the cutter compensation offset value). The following dialog box appears:

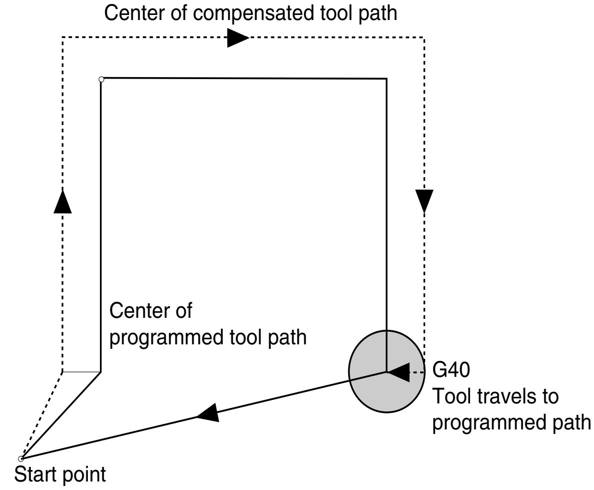
Select an offset number or set the offset value in the Offset Box. To set the offset value, double-click on an offset number from the scrolling Offset window. A check marks appears to the left of the offset number. Enter the offset value in the Offset Box.

Changing Offset Values

You can change the D numbers that represent values from the Offset Table while in cutter compensation. For example:

```
N1G91
N2G41D1X.25Y.25
N3Y.25
N4X.25D3; USE OFFSET #3
N5Y-.25
```

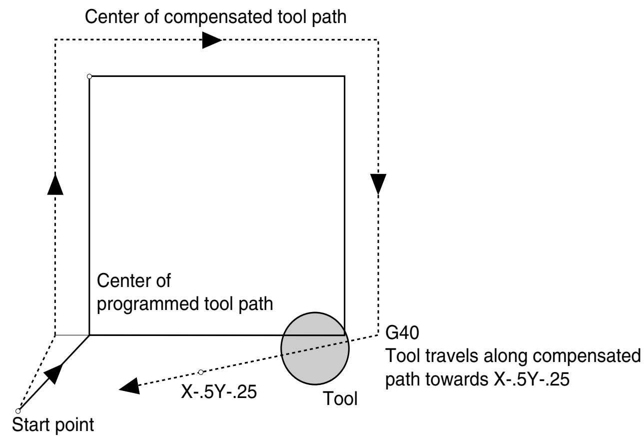
```
G91G41D1
...
X.25
Y-.25
Z.2; RETRACT
G40; OR D0
X-.5Y-.25
M2
```



...

In this example, the D number changes from 1 to 3 in line N4. Because the value of D3 is greater than the value of D1, the compensated path moves farther away from the programmed path and is at the new D value by the time the tool reaches point C.

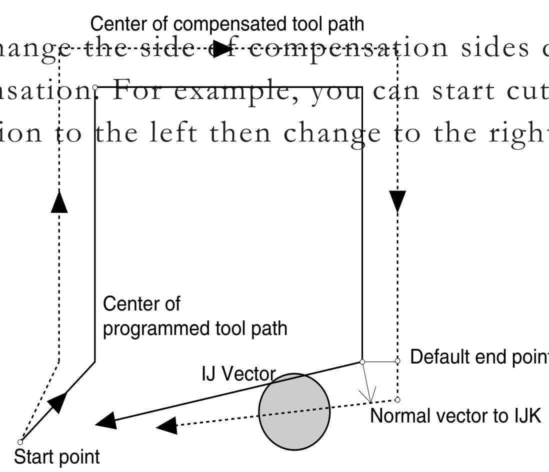
G91G41D1
...
X.25
Y-.25
Z.2;RETRACT
G40X-.5Y-.25
M2



Changing Offset Sides

You can change the side of compensation sides during cutter compensation. For example, you can start cutter compensation to the left then change to the right while in

G91G41D1
...
X.25
Y-.25
Z.2; RETRACT
G40X-.5Y-.25I-.5J-.25
M2



cutter compensation:

```
G91
G41D1X.25Y.25; LEFT CC ON
Y.25
G42X.25; RIGHT CC ON
 Y-.25
...
```

In this example, left cutter compensation is on at point A, but right cutter compensation begins as the tool moves towards point B. In this case, the offset value (D) is the same for both left and right cutter compensation.

The same situation would occur if you made the offset value negative. For the example above, changing the D value from .01 to -.01 would produce the same result as changing from G41 to G42.

CAUTION:

Using a P Code to scale an entire piece may affect the Z axis, which may affect your programmed depths of cuts. Use caution when performing scaling operations.

Using Corner Offset Circular Interpolation (G39)

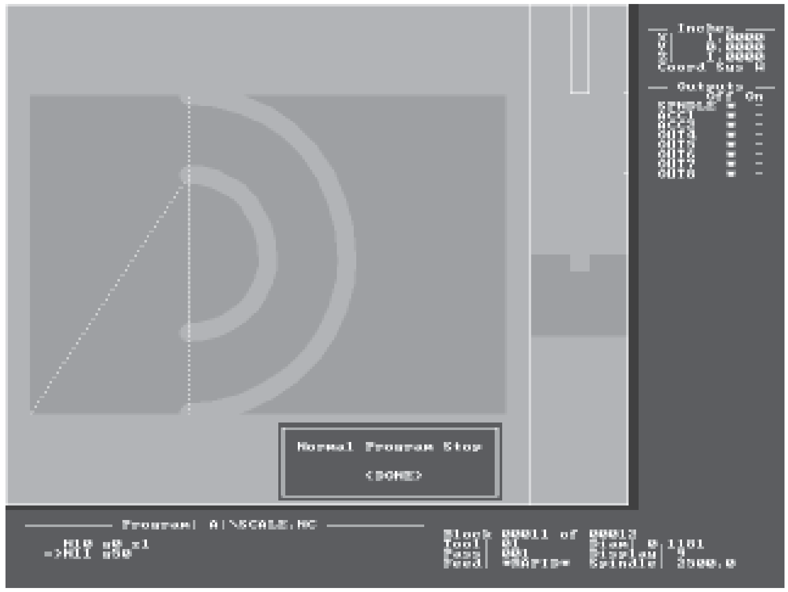
The G39 code inserts an arc at the corner of a cutter compensated tool path. The G39 instructs the cutter compensation function to complete the current segment by moving to its default endpoint (the endpoint of the Offset Vector). It then creates an arc (with a radius equal to the offset value), starting at the buffered segment's default endpoint, and ending at the endpoint of the offset vector (IJ). Here is an example of an NC program using G39:

```
G91
G41D1...
...
Y.25
X.25
G39I0J-1;  CORNER OFFSET
```

Canceling Cutter Compensation

Use the G40 code to cancel cutter compensation. G40 is effective for only one move. There are six ways to cancel cutter

In this example, a uniform scaling factor for all axes produces a shape scaled from the original.



Note: If you do not specify any of the coordinates for the scaling center, the current position for unspecified axes becomes the scaling center coordinate.

compensation.

1. G40
2. G40XYZ
3. G40XYZIJK
4. D0
5. D0XYZ
6. G41/42D0XYZIJK

In cases 4 through 6 above, setting the offset number to zero is the same as cancelling cutter compensation.

Cases 1 and 4

In case 1 the G40 code cancels cutter compensation. The cutter moves from the offset path to the programmed end point. The same occurs in case 4, where you set the D value to zero.

Cases 2 and 5

In cases 2 and 5, the G40 (or the D0) cancels the cutter compensation, but a subsequent motion is included in the program. The tool moves towards the programmed path in the direction of X-.5Y-.25.

Cases 3 and 6

In these cases an IJK vector specifies the direction of movement after cutter compensation is cancelled.

CAUTION:

Performing Z axis mirroring is an advanced operation. Use extreme caution when machining negative Z values.

Using Scaling and Rotation Codes

Scaling codes and rotating codes can be used separately or they can be combined. Each of these functions is described in the following paragraphs.

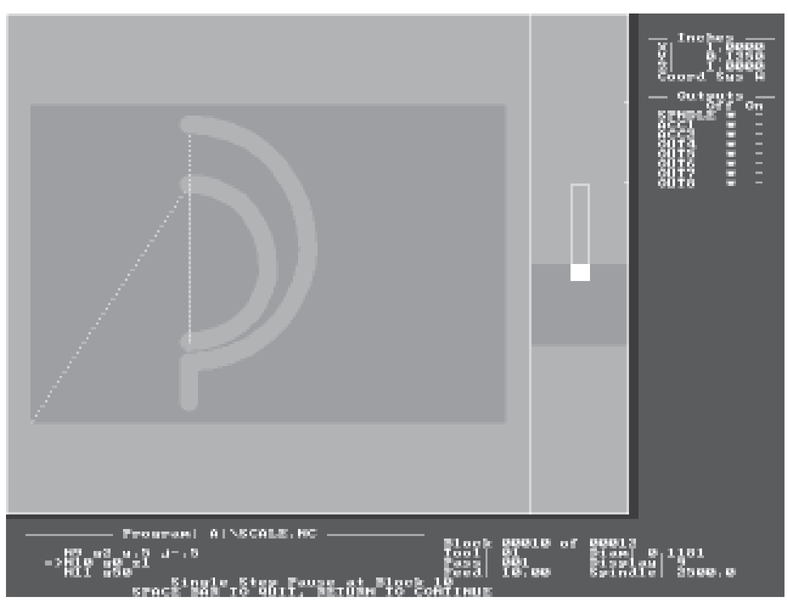
Scaling

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
P	Uniform scale multiplier

In this example, different scaling factors for the X and Y axes produce a shape similar to the one shown on the right.

When an arc is scaled differently in the two plane axes (X and Y in G17), an ellipse results. Because the Control Program cannot construct ellipses, it interprets the shape as an arc followed by a line segment move to the desired end point. The Control Program determines the arc by using the smaller of the two scale factors to determine its radius.

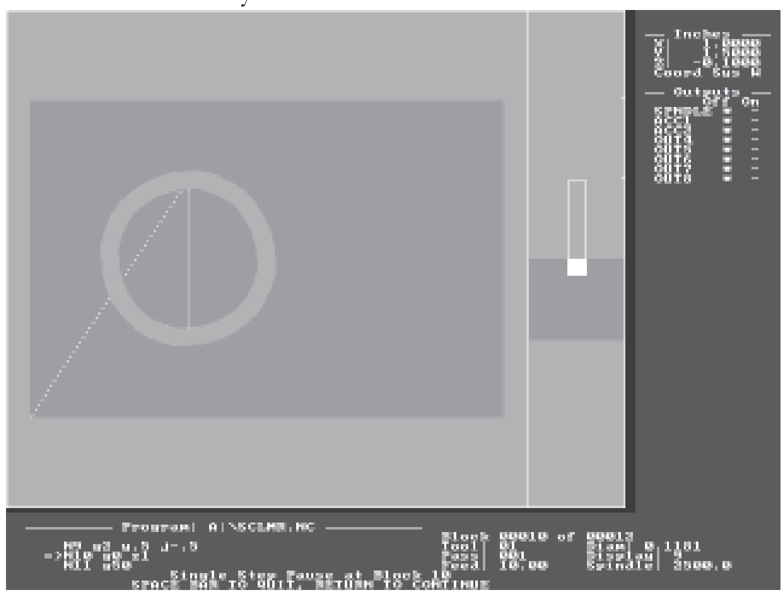


Note: If you do not specify a scale factor for an axis, the value of that axis defaults to a factor of 1

Uniform Scaling

Use a P Code to scale an entire piece uniformly along each axis. When you specify a value for P, subsequent motions are scaled by that value, starting from the scaling center. The Control Program measures the distance from the scaling center to the start and end points of the shape, then multiplies those values by the P value.

In this example, negative I and J values create a mirror image of the original shape.



The NC program below creates a half-circle then scales those motions by two to create a larger, uniformly-scaled half-circle (the scaling codes are in bold print):

```
N0G0Z.5
N1X1Y1.5
N2G1Z-.1F10
N3G2Y.5J-.5
N4G0Z1
```

N5G51X1Y1Z0P2; SUBSEQUENT MOTIONS SCALED BY 2

```
N6G0Z.5
N7X1Y1.5
N8G1Z-.1F10
N9G2Y.5J-.5
N10G0Z1
```

N11G50; CANCEL SCALING

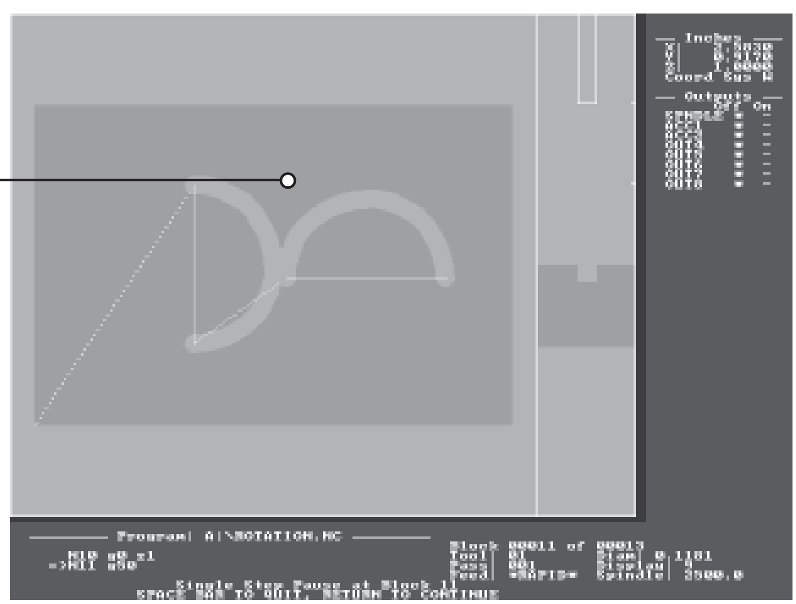
The values for X, Y, and Z in line N5 represent the absolute position of the scaling center. The P value represents the scale factor. In this example, the entire part is scaled by two. The G50 in line N11 cancels the scaling.

Scaling Each Axis

You can scale each axis by different magnifications. The following NC program uses scaling to change the proportions of

In this example, G68 rotates a shape 90^0 from the original.

Rotation Origin
(1.583, 1.5)



a motion by scaling each axis separately (the scaling codes are in bold print):

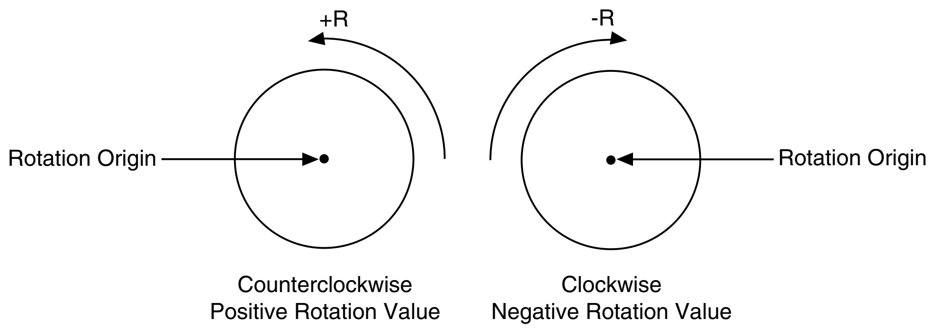
N0G0Z.5

N1X1Y1.5

N2G1Z-.1F10

N3G2Y.5J-.5

N4G0Z1



IMPORTANT:

When combining the scale and rotate features, always **scale** the part **first**, before rotating it.

N5G51X1Y1Z0I1.5J1.75K1; SCALING ON

N6G0Z.5

N7X1Y1.5

N8G1Z-.1F10

N9G2Y.5J-.5

N10G0Z1

N11G50; CANCEL SCALING

The values for X, Y, and Z in line 5 represent the absolute position of the scaling center. The I, J, and K values represent the scale factors for the X, Y, and Z axes respectively. When scaling each axis individually, you do not use a P code.

Creating Mirror Images with Scaling

Notice that in this program the P Code is used to reference the first block of the subprogram (not shown in this example). The subprogram contains the code that creates each part.

You can create mirror images of shapes by specifying negative values for I, J, and K. The NC program below uses negative I and J values to create a mirror image of the original shape on the XY plane (the scaling codes are in bold print):

N0G0Z.5

N1X1Y1.5

N2G1Z-.1F10

N3G2Y.5J-.5

N4G0Z1

N5G51X1Y1Z0I-1J-1K1; SCALING ON

N6G0Z.5

N7

In this example, scaling and rotation are combined to produce different sized shapes that are scaled and rotated from the original. Only three of the four parts are shown in the example at right; the fourth part is created with the call to subprogram in line N16.

IMPORTANT:

If your Machining Center is equipped with an Automatic Tool Changer, refer to the ATC Supplement for instructions on using Tool Length Offsets.

N8G1Z-.1F10

N9G2Y.5J-.5

N10G0Z1

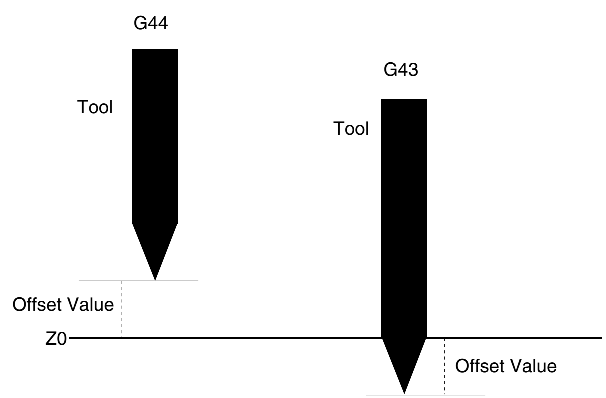
N11G50; CANCEL SCALING

The values for X, Y, and Z in line 5 represent the absolute position of the scaling center. The I, J, and K values represent the scale factors for the X, Y, and Z axes respectively. Notice the negative I and J values for mirroring on the XY plane. Remember that performing Z axis mirroring is an advanced operation. Use caution when machining negative Z values.

Rotation Codes

Rotation codes allow you to rotate a programmed shape around a rotation origin. You can rotate a shape on any plane, one plane at a time. Use the Rotation code to modify

Longer tools need G43 to retract the spindle from the cross slide, while shorter tools need G44 to move the spindle closer to the cross slide.



an NC program when a work piece has been rotated from the programmed position on the machine.

G68	Invokes rotation
G69	Cancels rotation

Here is an example of an NC program using Rotation (the Rotation codes are in bold):

N0G0Z.5

N1X1Y1.5

N2G1Z-.1F10

N3G2Y.5J-.5

N4G0Z1

N5G68X1.583Y1.5R90; ROTATION ON ROTATION ORIGIN XY

```
N6G0Z.5
N7X1Y1.5
N8G1Z-.1F10
N9G2Y.5J-.5
N10G0Z1
```
N11G69; CANCEL ROTATION

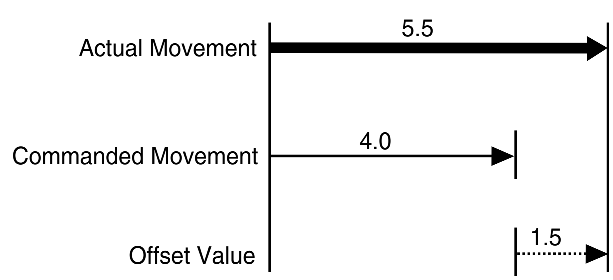
The X and Y values in line N5 are the coordinates of the rotation origin; the rotation occurs around this point. The R value represents the absolute value of the rotation angle. The G69 in line N11 cancels the rotation.

Note that positive R values represent counterclockwise rotation angles; negative R values represent clockwise rotation angles.

Combining Scaling and Rotation Codes

You can combine scaling and rotation in the same NC program. The portion of the NC program below combines scaling and rotation codes to machine a part:

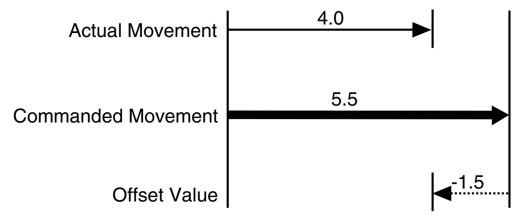
```
N1; DO THE ORIGINAL FIRST
N2M98P100
N3; NOW SCALE BY 1/2 AND ROTATE 90 CCW
N4G51X1.583Y0Z0I.5J.5K1; SCALING ON
N5G68X1.583Y1.5R90; ROTATION ON
```



N6M98P100; SECOND PART

N7; NOW MIRROR AND SCALE BY 1/2, AND ROTATE 90 CCW

N8G69G50



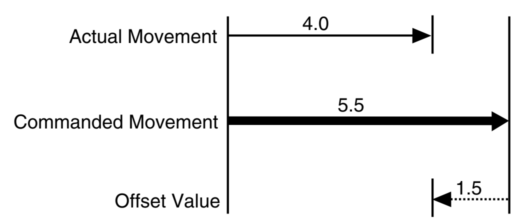
N9G51X1.583Y0Z0I-.5J.5K1

N10G68X1.583Y1.5R90

N11M98P100; THIRD PART

N12; NOW SCALE BY 1/2 AND ROTATE BY 90 CW

N13G69G50

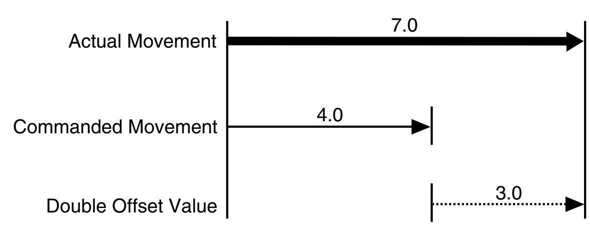


N14G51X1.583Y0Z0I.5J.5K1

N15G68X1.583Y1.5R-90

N16M98P100; FOURTH PART

N17;END OF PROGRAM

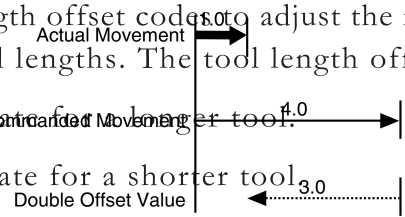


Using Tool Length Offset Codes

Use the tool length offset codes to adjust the machine for variations in tool lengths. The tool length offset codes are:

G43 Compensate for a longer tool.

G44 Compensate for a shorter tool.



G49 Cancel tool length offset.

H Specifies the offset number from the offset table. (The Offset Table you use for Tool Length Offset H values is the same table you use for Cutter Compensation and Tool Offset Adjust D values.)

The G43 compensates for a longer tool by retracting the spindle away from the cross slide. The G44 compensates for a shorter tool by moving the spindle closer to the cross slide.

The T code normally specifies the tool, its diameter, and offset value. When you include a G43 or G44, the computer ignores the T code offset value and uses the offset you assign to H.

Here is an example of the tool length offset code used in an NC program:

```
M06
G43T2H1
```

The M06 code stops the machine. The G43 compensates for a tool that is longer than the reference tool. The T2 refers to the tool number and the tool diameter, but not the offset value. The H1 represents the offset value set in the Offset Table.

Using Tool Offset Adjust Codes

Use these codes for making critical dimension adjustments to the offset values. When you specify tool offset adjust

Section 6: Optional Machining Capabilities

Quick Change Tooling Installation

Multiple Tool Programming

Machining with the 4th Axis Rotary Positioner

Quick Change Tooling Installation

Note: Before using Quick Change Tooling, clean the taper of the tool holder and the bore of the tool body.

Clean and coat the surface of the tool holder with oil once a week.

There are two Quick Change Tooling Models available from Light Machines for the proLIGHT 2000, ACC-5141 and ACC-5140 (Quick Change Tooling is not available for the proLIGHT 2500). The installation procedure for each is described below.

Installing the Quick Change Tooling Option on the proLIGHT 2000 Machining Center is a three-step process; installing the tool body in the machine spindle, attaching the cutting tool to the tool holder, and mounting the tool holder to the tool body.

Installing Quick Change Tooling Model ACC-5141

Note: Quick Change Tooling is not available for the proLIGHT 2500.

Refer to the drawing on the following page for the location of the parts referenced in these instructions.

Installing and Removing the Tool Body

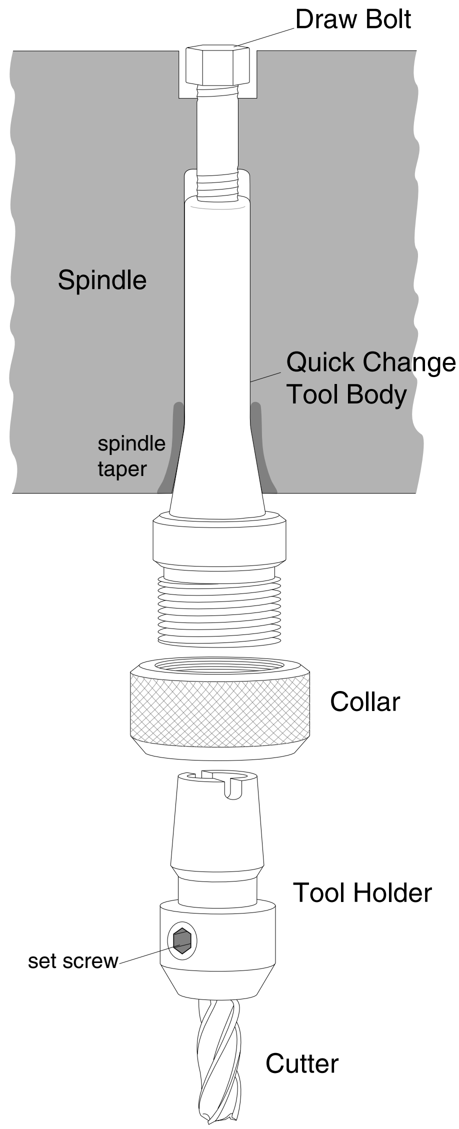
To install the tool body into the spindle:

1. Insert the draw bolt and washer (from the Machining Center Accessory Kit) into the top opening of the spindle shaft.
2. Insert the spindle locking pin (from the Accessory Kit) into the opening on the side of the spindle to keep it from rotating.
3. Insert the tool body into the bottom opening of the spindle until it makes contact with the spindle taper.
4. Screw the draw bolt into the tool body until secure.
5. Remove the spindle locking pin.

To remove the tool body from the spindle:

1. Insert the spindle locking pin into the opening on the side of the spindle.
2. Loosen the draw bolt approximately two turns.
3. Use a hammer (preferably brass) to hit the top of the draw bolt to release it from the spindle taper.

CAUTION: End mills and other cutters are very sharp. Use a shop rag or similar material to hold the tool while mounting it to avoid severe cuts.



4. While holding the tool body in place, loosen the draw bolt completely and allow the tool body to drop into your hand.
5. Remove the spindle locking pin.

Attaching the Cutting Tool to the Tool Holder

1. Insert the tool into the bottom opening of the tool holder.
2. Tighten the set screw on the side of the tool holder while holding the tool in place. Make sure the set screw is pressing against the flat of the tool.

Installing and Removing the Tool Holder

To mount the tool holder to the tool body:

1. Insert the spindle locking pin (from the Accessory Kit) into the opening on the side of the spindle to keep it from rotating.
2. Screw the collar all the way onto the tool body.
3. Now unscrew the collar approximately 1-1/2 turns.
4. Press the tool holder up into the tool body until it seats into place (This occurs when the two indents on the top of the tool holder align with the locking pins inside the tool body.)
5. Hand-tighten the collar. *Do not overtighten!*
6. Remove the spindle locking pin.

To remove the tool holder from the tool body:

1. Insert the spindle locking pin (from the Accessory Kit) into the opening on the side of the spindle to keep it from rotating.
2. Hold the tool holder, then unscrew the collar approximately 1-1/2 turns. Remove the tool holder.

3. Remove the spindle locking pin.

Installing Quick Change Tooling Model ACC-5140

Note: Before using Quick Change Tooling, clean the taper of the tool holder and the bore of the tool body.

Clean and lubricate the tool holder once a week.

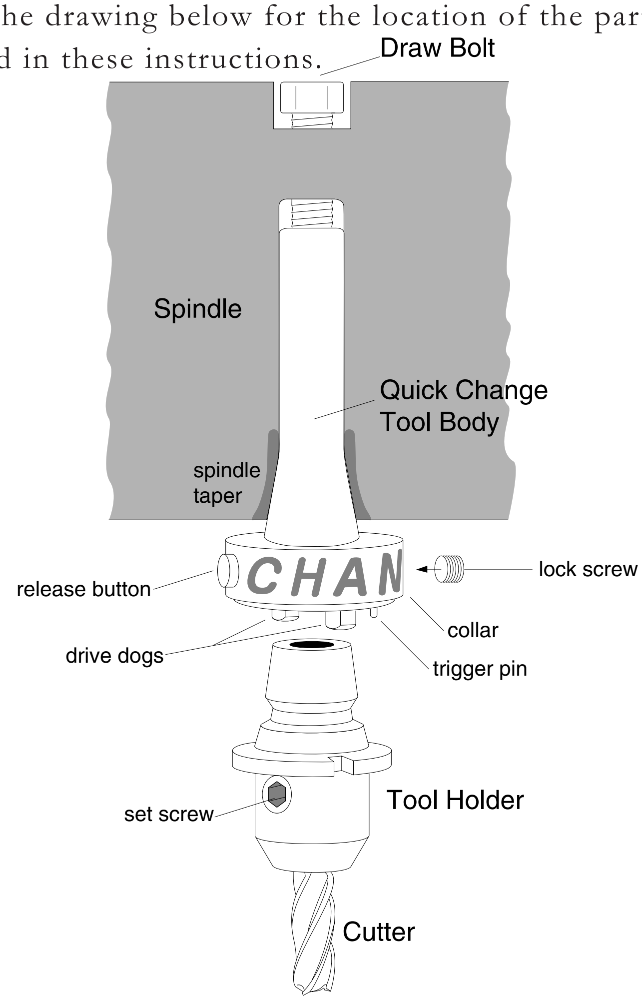
WARNING!

The lock screw must be tightened for all operations (single or multiple tool) in excess of 3,000 RPM to prevent ejection of the tool body from the machine tool.

This will also reduce wear on the holder and may enhance the performance of the machine.

CAUTION: End mills and other cutters are very sharp. Use a shop rag or similar material to hold the tool while mounting it to avoid severe cuts.

Refer to the drawing below for the location of the parts referenced in these instructions.



Installing and Removing the Tool Body

To install the tool body into the spindle:

1. Insert the draw bolt and washer (from the Accessory Kit) into the top opening of the spindle shaft.
2. Insert the spindle locking pin (from the Accessory Kit) into the opening on the side of the spindle to keep it from rotating.
3. Insert the tool body into the bottom opening of the spindle until it makes contact with the spindle taper.
4. Screw the draw bolt into the tool body until secure.
5. Remove the spindle locking pin.

To remove the tool body from the spindle:

1. Insert the spindle locking pin into the opening on the side of the spindle.
2. Loosen the draw bolt approximately two turns.
3. Use a hammer (preferably brass) to hit the top of the draw bolt to release it from the spindle taper.
4. While holding the tool body in place, loosen the draw bolt completely and allow the tool body to drop into your hand.
5. Remove the spindle locking pin.

Attaching the Cutting Tool to the Tool Holder

To attach the cutting tool to the tool holder:

1. Insert the tool into the bottom opening of the tool holder.
2. Tighten the set screw on the side of the tool holder while holding the tool in place. Make sure the set screw is

pressing against the flat of the tool.

Installing and Removing the Tool Holder

To mount the tool holder to the tool body:

1. Insert the spindle locking pin (from the Accessory Kit) into the opening on the side of the spindle to keep it from rotating.
2. Press the release button on the collar of the tool body. While the button is depressed, rotate the collar in a clockwise direction, approximately 1/8 turn, and hold it.
3. Align the indentations on the tool holder with the drive dogs on the tool body, then press the tool holder up into the tool body until it snaps into place.
4. Release the collar. The collar must return to its original position after release.
5. Make sure the tool holder is properly seated. If properly seated, the collar will not rotate in either direction.
6. Remove the spindle locking pin.

To remove the tool holder from the tool body:

1. Insert the spindle locking pin (from the Accessory Kit) into the opening on the side of the spindle to keep it from rotating.
2. Loosen the lock screw.
3. Press the release button on the collar of the tool body. While the button is depressed, rotate the collar in a counterclockwise direction, approximately 1/8 turn, and hold it.
4. Wiggle the tool holder out of the tool body.
5. Remove the spindle locking pin.

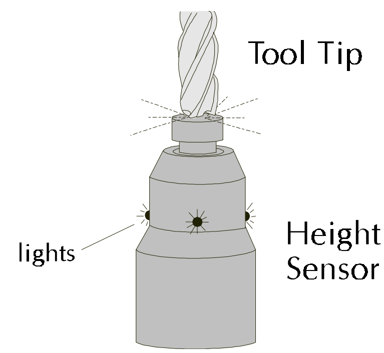
Using the Tool Height Sensor

A tool height offset sensor is included with every Quick Change Tooling unit. This sensor helps to establish a constant reference point for setting tool offsets for multiple tools.

Note: The tool height offset sensor does not work if placed on a wax, wood, or plastic surface. You must place the sensor on a conductive surface for it to work.

The battery powered sensor uses the machine's frame to provide an electrical circuit. When a tool contacts the sensor the circuit closes, energizing the sensor's LED indicators. You must place the sensor on a conductive surface, such as the cross slide, for this to take place.

To establish a reference point:



1. Place the sensor on the cross slide or other conductive surface.
2. Jog tool #1 to the tip of the sensor until the lights come on.
3. Use the Set Position function in your Control Program software to set the Z axis value for this point to zero.

To establish offsets:

1. Install tool #2 in the Quick Change Tooling unit.
2. Jog tool #2 to the tip of the sensor until the lights come on.
3. Use the Tool Definitions function in your Control Program to enter an offset value for tool #2.
4. Repeat this process for each additional tool.

   You can use this procedure to establish the Z position of the reference tool to the stock. For example, if the length of the reference tool is 2.000", set the Z position equal to 2.000.

Multiple Tool Programming

CAUTION:

You should become thoroughly familiar with NC programming for a single tool before attempting to do multiple tool programming.

Note: Multiple Tool Programming is not available for the proLIGHT 2500

The proLIGHT 2000 Machining Center allows you to designate up to 20 different tool offsets for tool changes during milling operations. To designate the tool number and Z axis offset, select Set Tools from the Setup Menu.

There are four basic steps in setting up the Machining Center for multiple tool operation:

- ❑ Place the appropriate codes in your NC program.
- ❑ Establish a reference tool.
- ❑ Establish the offsets for other tools from that reference.
- ❑ Test your NC program.

Using Multiple Tool Codes

The T code is used in the NC program to offset the cutter so the NC program becomes independent of the cutter length. 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. (Any actual tool change is performed manually.) The T code can be located anywhere within the block of NC code, but it is normally placed after the G code.

Note: The T code and M06 command can be used independently since they perform different functions. The T code offsets the coordinate system for the tool specified. The M06 code simply retracts the spindle and pauses so you can change the tool.

When you place T codes in your program for tool changes, you should also use the M06 code to retract and shut off the spindle. The M06 code is placed after the T code. This code instructs the Machining Center to shut off and retract the spindle to the top of the vertical column, where it pauses until you manually change the tool. Pressing the Return key turns the spindle back on and move it back to the previous position to continue with the NC program.

Establishing the Reference Tool

When using multiple tools, 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 work piece, or on a gauge, where you will jog the tip of each tool).
2. Select Manual Control from the Run Menu. With Tool #1 installed in the spindle, jog the tool to the reference point. The tip of the tool should barely touch the work piece, 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 axis.

Tool #1 is now established as the reference tool.

IMPORTANT:

If your Machining Center is equipped with an Automatic Tool Changer, refer to the ATC Supplement for instructions on using Tool Length Offsets.

Establishing Tool Offsets

Now that the reference tool is established, you can assign offsets to additional tools. You can move the tool and accept its current Z axis position as the offset value, or you can manually enter offset values.

To set the offset for Tool #2:

1. Jog the spindle up, insert Tool #2 then jog it down just to touch the surface of the work piece, or offset sensor, at the previously determined reference point.
2. Select Set Tools from the Setup Menu. The Z axis position will be off by the difference in the length between this tool and the reference tool.
3. Select Tool #2. Select the Use -[Position] button. The Control Program assigns the offset value of Tool #2 as the opposite sign of the current Z position.

The offset for Tool #2 is now established.

Testing Your Multiple Tool Program

Note: There is a sample multiple tool NC program on the proLIGHT Control Program disk that came with your Machining Center. The sample program should have been copied to your hard drive along with the other files during the software installation procedure (see Section 2). Look in the PLM2000 directory for MILLTWO.NC.

Note: A Quick Change Tooling option is available from Light Machines Corporation (part number ACC-5140). This package makes manual tool changing quicker and easier.

After setting all of the tool offsets, dry run your program without a work piece mounted and with the spindle speed turned down (just as you did with the sample single-tool program in Section 4).

1. After installing Tool #1, close the safety shield, put on your safety glasses, and complete the safety checklist.
2. Select Run Program from the Run Menu. Select zero as the start block and select ok. Throughout the test, be prepared to press the emergency stop switch on the Machining Center, or the space bar on the computer keyboard in case of a tool crash. The computer runs the program until it reaches the M06 code. The M06 stops and retracts the spindle.
3. When the spindle has completely stopped and the Pause message appears on the screen, press the emergency stop switch on the Machining Center.
4. Open the safety shield.
5. Remove Tool #1 and install Tool #2, making certain it is securely fastened to the spindle.
6. Close the shield, and pull out the emergency stop switch. Press the Return key on the computer keyboard. The spindle turns on and 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.
8. Edit the program, if required. When you are satisfied that the program works correctly, mount the work piece, set the spindle speed, and run your multiple tool program.

CAUTION: Do not use collets for tool changing. Use an end mill holder or the Quick Change Tooling option to make sure the tool length protruding from the spindle does not vary each time you load the tool into the spindle.

Machining with the 4th Axis Rotary Positioner

For Rotary Positioner maintenance information, refer to Section 7.

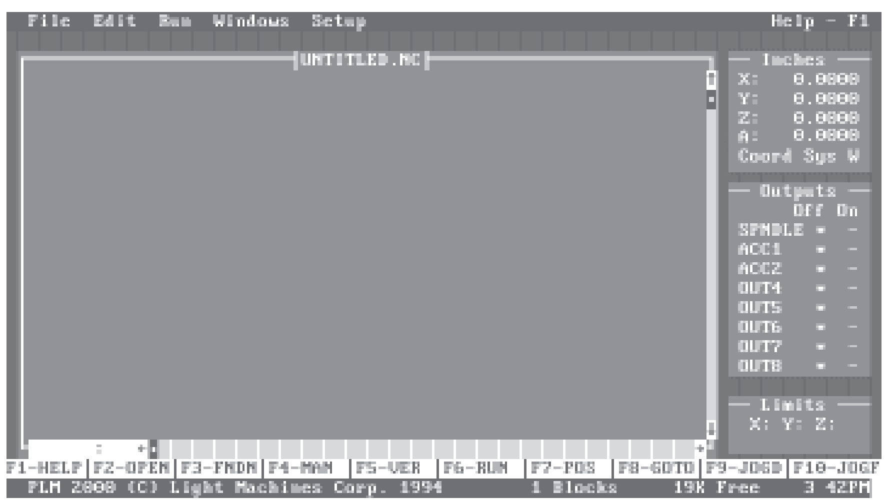
The 4th Axis Rotary Positioner is a factory-installed option available from Light Machines Corporation. This rotary positioning device uses a 5C collet system with a manual collet closer and mounts vertically on the Machining Center table. With the Rotary Positioner, you can perform numerous machining tasks, such as simultaneous 4-axis machining, wrapping geometries around cylinders as in cylindrical engraving, and rotary indexing to mill on the sides of a part.

When working with a fourth axis there are, of course, differences in both the Control Program software, and the NC programming method to accommodate the additional axis.

The Control Program

Note: You can type SET LMC=/4 at the C:> prompt to set up a permanent command line switch for 4th axis machining. Add the command to your AUTOEXEC.BAT file if you do not want to supply the /4 switch every time you run the Control Program.

The ability to accommodate a fourth axis is embedded in the PLM2000 Control Program software. To invoke the fourth axis version of the software, type PLM2000/4 once you are in the PLM2000 directory. The Control Program



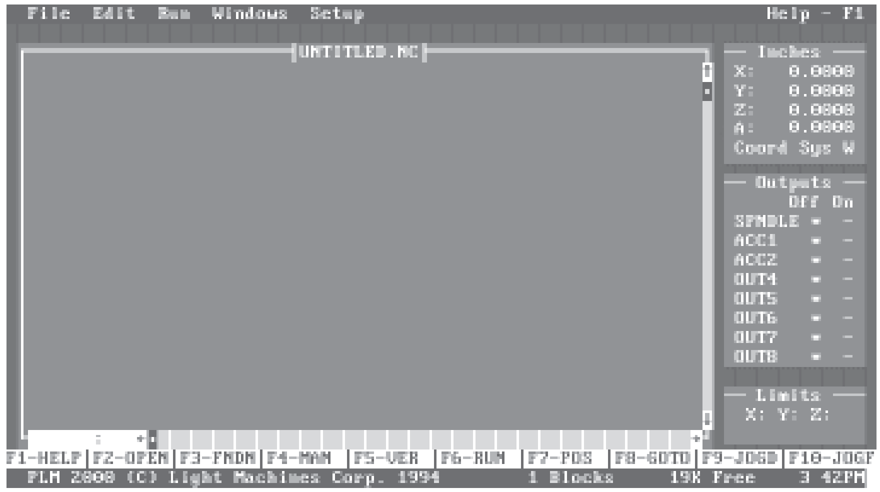
screen appears.

You can see right away that the Position and Limits Windows have the additional A axis. In fact, wherever the other

Note: To rotate the 4th axis with Keypad, use the Page Up and Page Down keys. Both keys use the current Continuous or Incremental jog settings.

three axes are displayed, you see the A axis displayed.

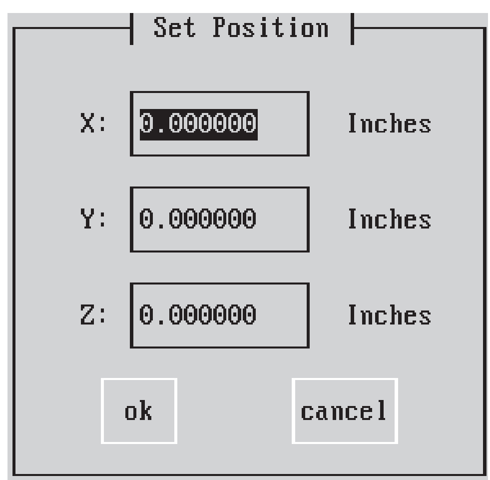
For instance, when you open the Manual Control Panel, the A axis Jog Feed and Distance is displayed just below the Z



axis Jog Feed and Distance.

Under the Setup Menu, the commands Set Position, Goto Position, Set Jog Distance, Set Jog Feed, Set Limits and Coordinate. Systems now accommodate the fourth axis in their dialog boxes.

This is the Set Position dialog box for the 4th Axis Rotary Positioner. Other dialog boxes with information or data entry pertaining to the axes look similar to this.



Programming

Things to keep in mind when using A codes:

•You can specify more than 360 degrees using the A code.

•You can perform helical interpolation on the A axis

•You can't home the A axis.

•You can't perform G02 or G03 moves with A codes.

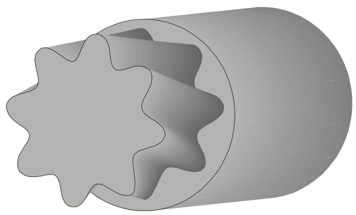
The most obvious difference in programming for a fourth axis is in the use of the A code. As with the X, Y and Z codes, the A code follows incremental and absolute programming. If an A code is placed on the same block as an X, Y or Z code, the A axis feed is calculated after the X, Y or Z axis feed so the A motion, in inches per minute, stops at the same place each time (similar to helical interpolation).

When an A code is placed on a block without X, Y, and Z codes, the accompanying F code specifies revolutions in degrees per minute, while the A code represents the amount of rotation in degrees. Depending on their position in the NC program, the values for F codes can represent both revolutions in degrees per minute and inches per minute. For example:

```
N1M3S1500
N2G0X0.8Y0Z0.05A0
N3G1Z-0.125F3; IPM
N4G1A360F30; 4TH AXIS MOTION IS 30 DEGREES PER MINUTE
N5G0Z0.05
N6G0X0.2A0
N7G1Z-0.125F3; IPM
N8G1A360F30; 4TH AXIS MOTION IS 30 DEGREES PER MINUTE
N9G0Z0.05
N10G0A0
N11G1Z-.125
N12G1X.8A360F5; 4TH AXIS MOTION IS 5 INCHES PER MINUTE
N13G0Z0.050
N14M02
```

The F codes that accompany the A codes in blocks N4 and N8 represent revolutions in degrees per minute. The F code that accompanies the A codes in block N12 represents revolutions in inches per minute.

The sample program ROTARY.NC generates a part similar to the one shown here.



Running the ROTARY.NC Sample Program

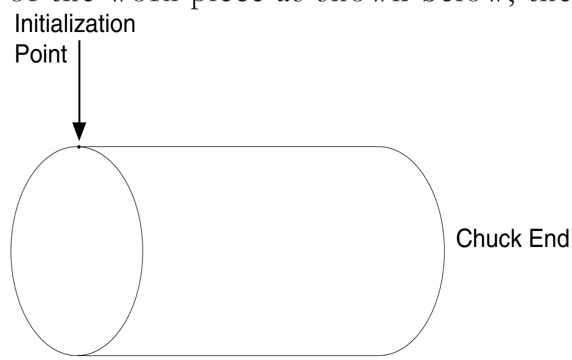
A sample NC program called ROTARY.NC is included on the Control Program installation disk. It should have been copied to your hard drive along with the other sample programs when you performed the software installation. *This program only works on systems that have the 4th Axis Rotary Positioner installed.*

What You Need...

To make the part, you need a 2-inch-diameter by 2-inch-long workpiece, preferably made of wax due to the depth of the cuts. You'll also need a 1/4-inch ball end mill.

To Initialize the Software...

To initialize the software, you must jog the tool to the top right edge of the work piece as shown below, then set the

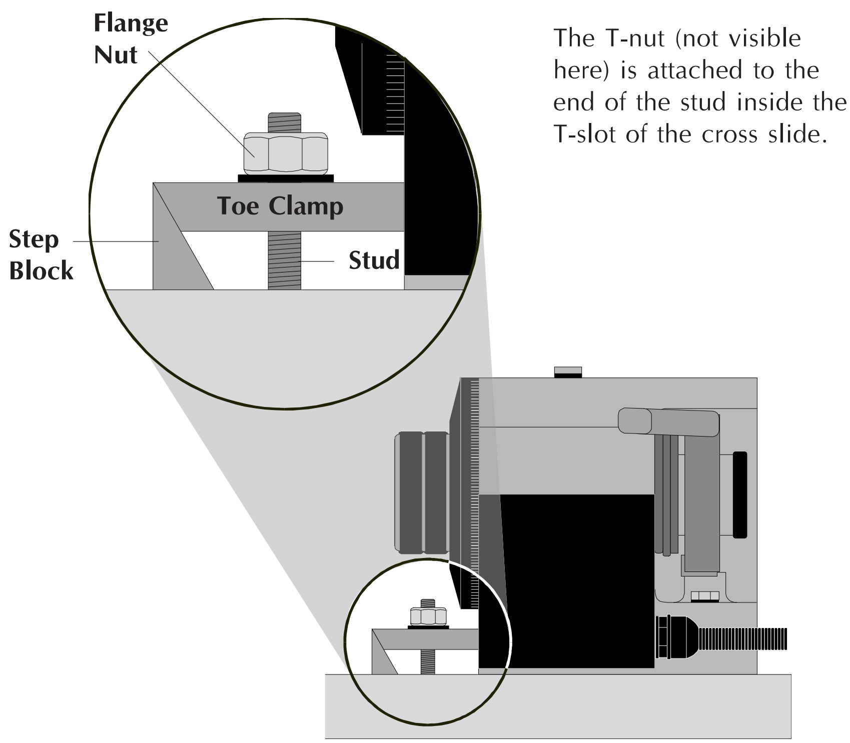


tool position to X0, Y0, Z0, A0.

Installing and Removing the Rotary Positioner

To install the Rotary Positioner:

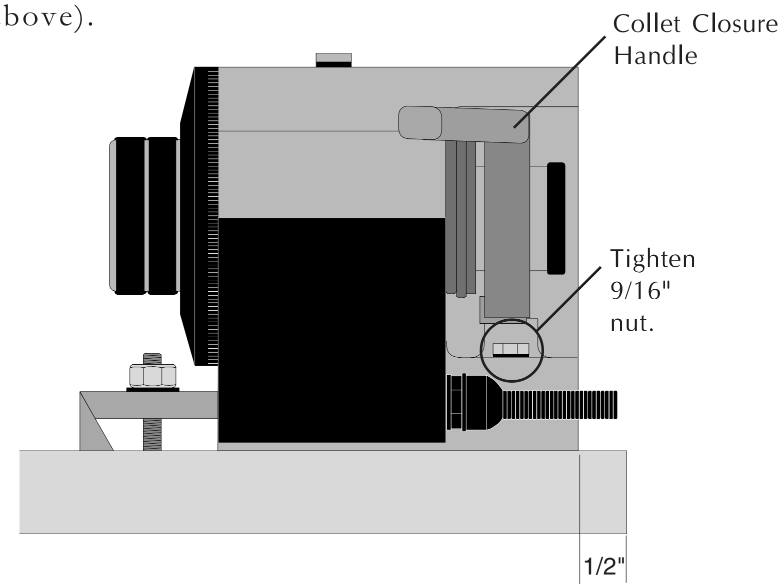
1. Make sure the PLM2000 is disconnected from all power



sources.

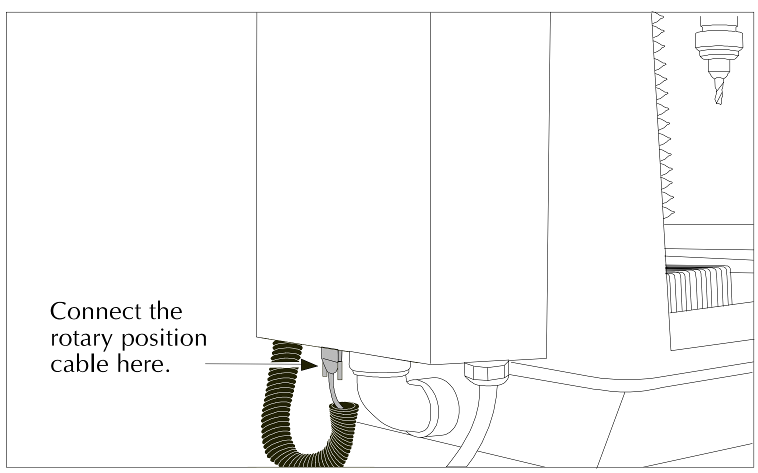
2. Loosely assemble the stud, flange nut, toe clamp and a T-nut as shown below. Slide the assembly into position on the cross slide (with the T-nut in the center T-slot of the cross slide).
3. Slide the Rotary Positioner into position (with the T-nut in the T-slot of the cross slide) on the cross slide. The edge of the Positioner should be approximately 1/2" away from the edge of the cross slide. Make sure the base of the Positioner is parallel with the edge of the cross slide.

4. Place the right end of the toe clamp all the way in to the recess on the base of the Rotary Positioner (as shown above).



5. Insert the step block under the left edge of the toe clamp. Make sure the toe clamp is level with the cross slide and tighten the flange nut with an 11/16" wrench.

6. Tighten the nut beneath the collet closure handle (shown



below) with a 9/16" wrench.

7. Remove the Loop back plug from the Rotary Position connector on the rear of the Machine.
8. Connect the rotaty position cable (CPC-16 connector) to the mating connector on the back of the Machining Center.

To remove the Rotary Positioner:

1. Turn off the PLM2000 and disconnect it from all power sources.
2. Disconnect the encoder cable (9-pin D-subminiature connector) and the servo motor power cable (4-pin CPC connector) from the back of the Machining Center.
3. Loosen the two nuts on the studs holding the T-nuts in the T-slots of the cross slide.
4. You can now slide the Rotary Positioner off the Machining Center cross slide.

Using 5C Collets and 5C Chucks

Use standard 5C collets and 5C chucks with your Rotary Positioner. To install a collet:

1. Insert the collet or chuck into the spindle. Be sure to align the keyway on the collet with the pin inside the

spindle.

2. Hold the collet in the spindle while adjusting the collet draw bar.
3. Pull the collet closure handle away from the spindle. Check for appropriate tightness.
4. Repeat steps 2 to 4 until you find the appropriate tightness.

Rotary Positioner Specifications

Available X-axis travel:	6.5 inches
Available Z-axis travel:	5.25 inches
Spindle Runout:	0.0004 T.I.R.
Spindle Backlash:	+/- 1 minute
Spindle Speeds:	120 deg/sec
Collets:	Standard 5C
Spindle Nose Thread:	2.19-10 Thd.
Indexing Accuracy:	+/- 15 seconds in one direction
Repeatability:	4 seconds
Resolution:	0.001 deg
Motor:	40 oz. in. continuous torque DC Servo
Gear Ratio:	90:1 self-locking worm gear with preload
Duty Cycle:	50% maximum duty cycle
Weight:	45 lbs.
Spindle Height:	4.300 +/- .001 from spindle center to base
Encoder:	1000 line

Section 7: proLIGHT Machining Center Maintenance

Maintaining the proLIGHT 2000

Performing preventative maintenance on your proLIGHT 2000 Machining Center ensures a longer, trouble-free life for the machine. We provide instructions for preventative maintenance in the following paragraphs.

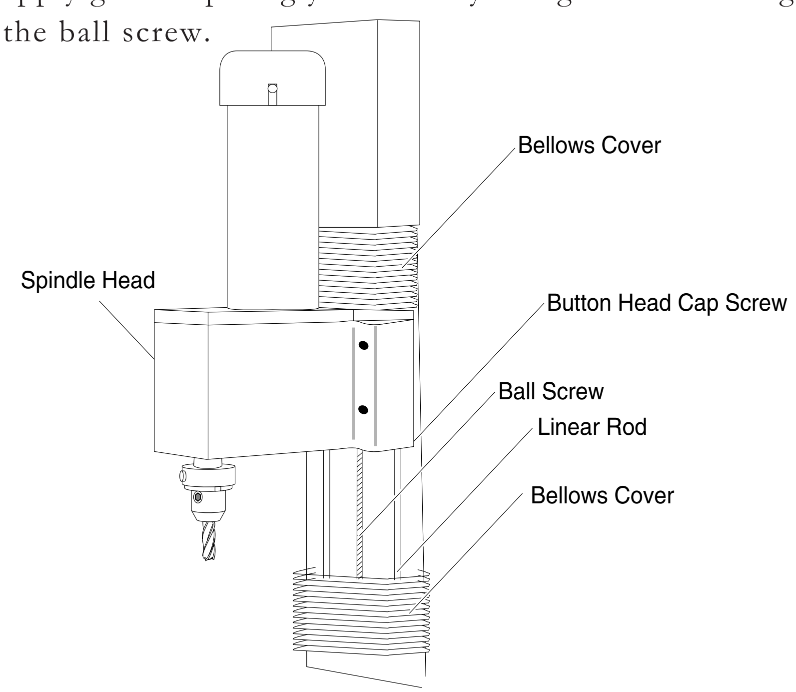
Ball Screws

Note: If you run your machine continuously (8 hours per day) at high feed rates, you should lubricate your machine every 16 hours of use.

The proLIGHT uses pre-loaded ball screws on all three axes. The screws are lubricated at the factory with a special long-life, waterproof ball screw lubricant. This lubricant should last for at least 200-250 hours of machine use. After 200 hours, you can apply more lubricant to the screws in a thin film over the length of the screw. The ball screw lubricant (part number M2006) is available from Light Machines Corporation.

To gain access to the ball screws, jog the cross slide and spindle to the extreme negative end of travel on all axes just before the limits are tripped. Remove the bellows cover by removing the two button head cap screws from the bracket holding the bellows at one end. Use a small brush to apply grease sparingly but evenly along the entire length of the ball screw.

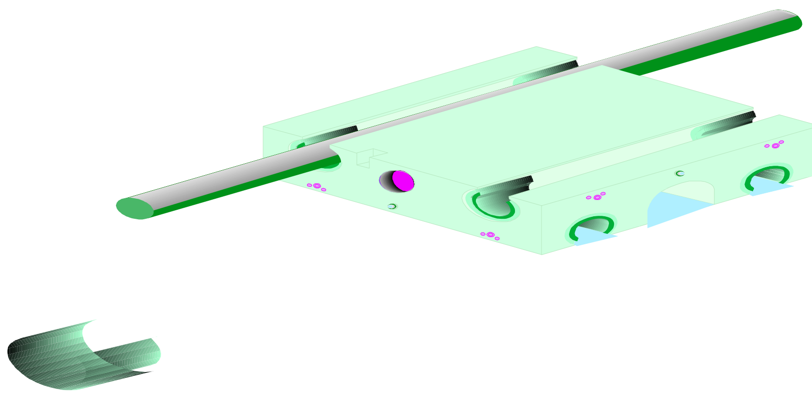
Here is the proLIGHT 2000 with the Z axis bellows cover removed to expose the Z axis linear rods and ball screw.



The Saddle

The saddle engages the linear rods that are attached to the base of the machining center. A ball screw moves the saddle along the Y axis. The linear rods running through the top of the saddle engage the cross slide. A ball screw moves the cross slide along the X axis. The oil ports for the X axis and Y axis linear rods are located on the saddle.

The Machining Center's Sadd



Linear Slides

The linear slides on the proLIGHT consist of linear rods and linear bearings. It is *very important* that a thin film of lubricant be maintained on the surface of the linear rods to minimize wear. The linear rods will wear very quickly with no lubrication.

Lubrication

Light Machines provides a small oil can with the Machining Center. Use 10W engine oil for lubricating the linear rods.

There are thick oil-impregnated felt pads between the bearings in the saddle which keep a thin film of oil on the linear rods. Each rod (there are six) can be lubricated through its own oil port which is located on the saddle for the X and Y axes, and on the spindle head for the Z axis. You should lubricate the rods every 30 days or 100 hours of use, whichever comes first. When applying oil to the oil ports, pull the trigger on the oil gun very slowly to allow the oil to be soaked up by the felt.

CAUTION:

Do not remove the portion of the motor cover that has the wires coming out of it.

Adjustment

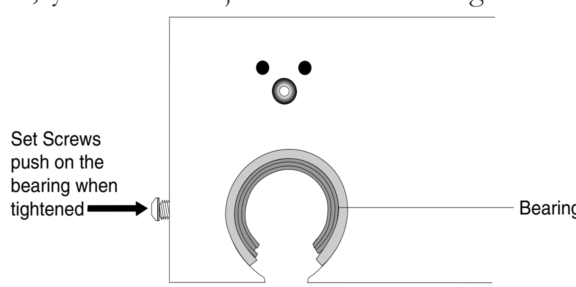
You can adjust the linear bearings to remove any play in the saddle or spindle. The bearings are factory-adjusted and should not require adjustment for at least 250 hours of use. Be very careful not to over-tighten the bushings because over-tightening can cause overworking and overheating of the motor, and excessive wear to the rods, bushings and ball screws. To adjust the linear bushings, follow the steps below.

WARNING:

Always unplug the machine before making adjustments.

1. Jog to the extreme positive end of motion on the axis you are adjusting. Stop just before the limit switch trips. Shut off power to the system. Unplug the machine.
2. Remove part of the servo motor cover for the axis you are adjusting. Loosen the screws but do not remove the portion of the cover that has the wires coming out of it. Do not remove the whole cover at once or you may damage the wiring. If you need to use excessive force to turn the lead screw, you have adjusted the bearing too tightly.

A side view of the Saddle.



3. Loosen all eight recessed allen setscrews on the axis you are adjusting. They are located on the four sides of the saddle for the X and Y axes, and on the sides of the spindle for the Z axis.
4. Tighten the two setscrews for each bearing while checking the force it takes to turn the ball screw. Tighten the setscrews (applying approximately 4-8 in/lb of torque) until the force required to turn the ball screw pulley in-

CAUTION:

Adjusting the spindle head is a very precise procedure. Do not adjust the spindle head.

creases when turning it by hand. Then back the set-screws off by 1/8-turn. Repeat this procedure for all four bushings on each axis.

5. Replace the servo motor cover.

Spindle

Two Piece Spindle Head

Light Machines ships the proLIGHT Machining Center with a factory-aligned two piece spindle head. You should not attempt to align the spindle head without first contacting Light Machines.

CAUTION:

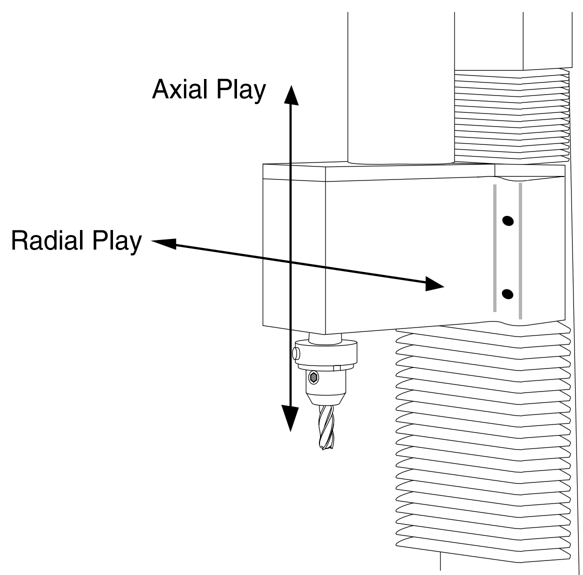
Adjusting the spindle shaft pre-load is a very precise procedure. Do not attempt to adjust the pre-load without first contacting Light Machines for information.

Spindle Motor

The spindle motor on the proLIGHT Machining Center is a 1hp DC permanent magnet motor. The wearing parts on the motor are the ball bearings on the motor shaft and the brushes. The brushes, although replaceable, have a long lifespan. The ball bearings are sealed, lifetime-lubricated bearings that do not require special maintenance.

Checking for Spindle Shaft Play

The spindle shaft is pre-loaded against sealed ball bearings that do not require lubrication or user maintenance. You should, however, check the spindle shaft for both radial play (side to side) and axial play (up and down). If the



spindle shaft starts to develop play (or begins to make unusual noises while in operation), contact Light Machines (800-221-2763).

To check for play, grip the spindle shaft and push and pull it in each direction along both axes. The spindle shaft should be firm against your pressure.

Belts

Spindle Belt

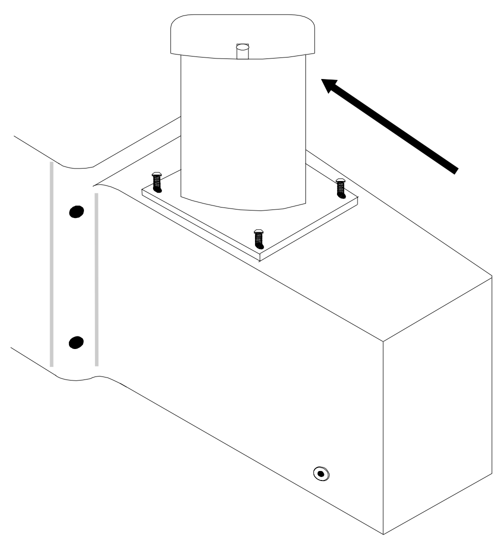
WARNING:

Always unplug the machine before making adjustments.

The spindle motor drives the spindle shaft with a serpentine belt. If the belt becomes loose, it will wear out quickly. If the belt is loose, you can feel backlash between the spindle motor shaft and the spindle shaft. To check for backlash, rotate the spindle shaft back and forth slightly and observe the fan inside the motor. Make sure the fan rotates as soon as you reverse the direction of the spindle shaft. If not, there is backlash. The adjustment is a simple operation.

To adjust the spindle timing belt:

1. Make sure all power to the system is shut off. Unplug the machine.



Loosen the four cap screws and push the motor backwards

2. Loosen the four socket head cap screws on the base of the spindle motor.
3. Push the spindle motor backwards to remove play from the belt.

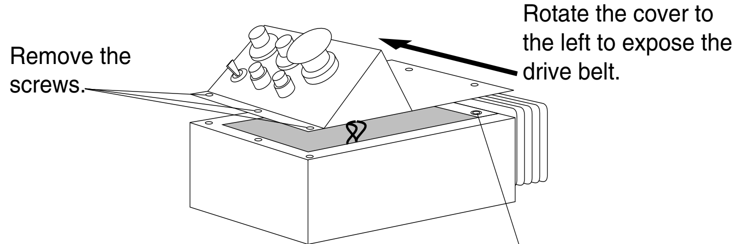
4. While holding the motor in place, tighten the four socket head cap screws.
5. Turn the spindle shaft by hand to make sure the spindle moves freely and the spindle motor turns.
6. If everything appears all right, plug the machine back in and turn on the spindle motor. Slowly bring the spindle up to speed. Check for excessive vibration.

Axis Drive Belts

The axis drive belts are located between the servo motors and ball screws on each axis. Normally, they should not need

Warning!

Disconnect Power Cord before removing cover. Dangerous voltages are present when machine is connected.



adjustment, but you should check every 250 hours.

Checking and Adjusting the Y Axis Drive Belt

CAUTION:

Do not over-tighten the drive belt. Excessive tension will damage the motor shaft.

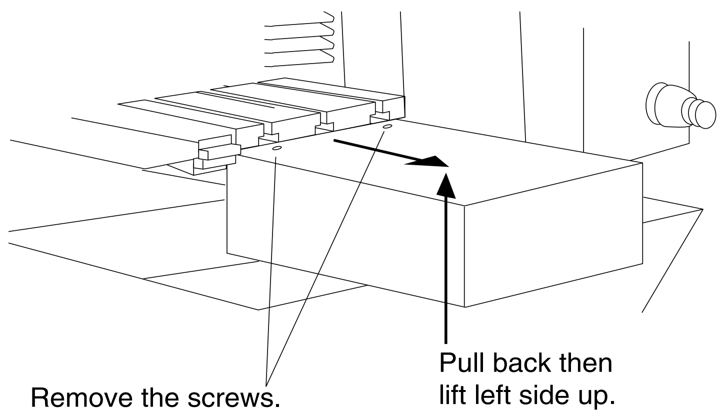
1. Remove the 7 button head cap screws on the motor box cover.
2. Lift the cover and rotate it 90 degrees. Be aware of the two #10 washers under the cover. These washers are used as spacers in the motor box. Be sure not to lose these washers and remember to re-install them when you are finished checking the belt.
3. Apply approximately 3 pounds of force on the center point of the belt. It should deflect no more than 1/8-inch (3mm).

If the belt needs adjustment:

1. Remove the two button head cap screws from the bot-

tom of the motor box.

2. Gently move the box up and down while pulling back until the box comes off the motor mount.
3. Slide the motor box to the left and down until you have access to the four screws holding the servo motor onto the motor mount.



4. Loosen the four screws.
5. Slide the motor to increase tension on the belt and re-tighten the screws.
6. Check the deflection again.

CAUTION:

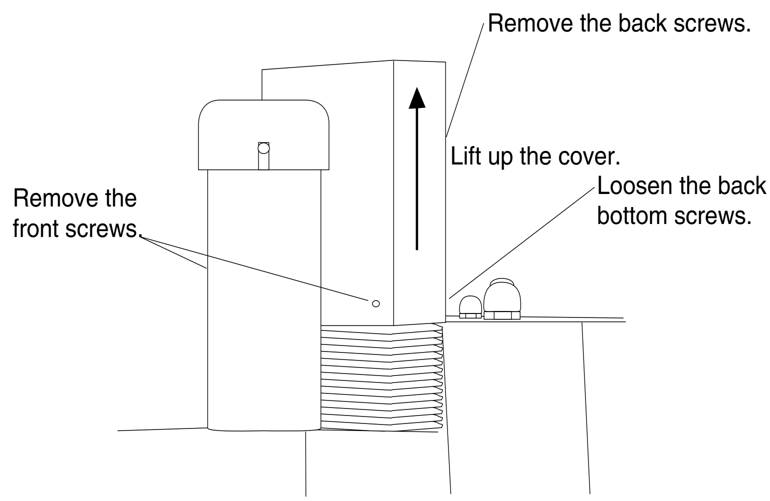
Do not over-tighten the drive belt. Excessive tension will damage the motor shaft.

Checking and Adjusting the X Axis Drive Belt

1. Remove the two button head cap screws from the top of the motor box and the seven button head cap screws from the bottom of the motor box cover.
2. Pull the motor box back approximately 1 inch until it

comes off the motor mount. Lift the left side of the motor box up and over the motor to expose the drive belt.

3. Apply approximately 3 pounds of force on the center point of the belt. It should deflect no more than 1/8-inch (3mm).



If the belt needs adjustment:

CAUTION:

Do not over-tighten the drive belt. Excessive tension will damage the motor shaft.

1. Loosen the four screws holding the servo motor onto the motor mount.
2. Slide the motor to increase tension on the belt and re-tighten the screws.
3. Check the deflection again.

Checking and Adjusting the Z Axis Drive Belt

1. Remove the two button head cap screws on the front of the motor box and five of the seven button head cap

screws on the back. Loosen but leave in place the two bottom screws on the back of the motor box cover.

2. Gently move the box front and back while pulling it up until it is completely off of the motor mount.

If the belt needs adjustment:

1. Loosen the four screws holding the servo motor onto the motor mount.
2. Slide the motor to increase tension on the belt and re-tighten the screws.
3. Check the deflection again.

Note: If you run your machine continuously (8 hours per day) at high feed rates, you should lubricate your machine every 16 hours of use.

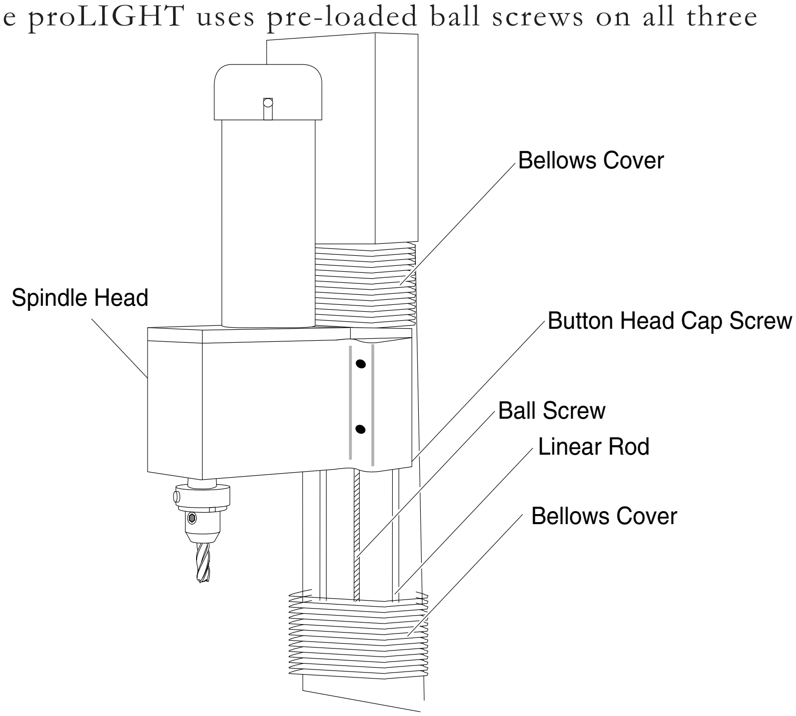
Maintaining the proLIGHT 2500

Performing preventative maintenance on your proLIGHT 2500 Machining Center ensures a longer, trouble-free life for the machine. We provide instructions for preventative maintenance in the following paragraphs.

Ball Screws

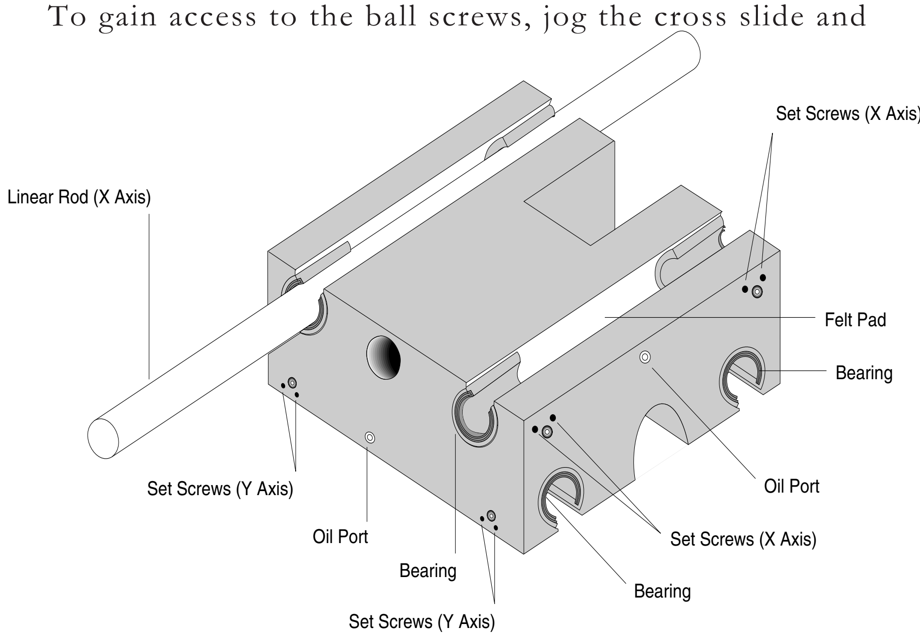
The proLIGHT uses pre-loaded ball screws on all three

Here is the proLIGHT 2500 with the Z axis bellows cover removed to expose the Z axis linear rods and ball screw.



axes. The screws are lubricated at the factory with a special long-life, waterproof ball screw lubricant. This lubricant should last for at least 200-250 hours of machine use. After 200 hours, you can apply more lubricant to the screws in a thin film over the length of the screw. The ball screw lubricant (part number M2006) is available from Light Machines Corporation.

To gain access to the ball screws, jog the cross slide and



The Machining Center's Saddle.

spindle to the extreme negative end of travel on all axes just before the limits are tripped. Remove the bellows cover by removing the two button head cap screws from the bracket holding the bellows at one end. Use a small brush to apply grease sparingly but evenly along the entire length of the ball screw.

The Saddle

The saddle engages the linear rods that are attached to the base of the machining center. A ball screw moves the saddle along the Y axis. The linear rods running through the top of the saddle engage the cross slide. A ball screw moves the cross slide along the X axis. The oil ports for the X axis and Y axis linear rods are located on the saddle.

Linear Slides

The linear slides on the proLIGHT consist of linear rods and

CAUTION:

Do not remove the portion of the motor cover that has the wires coming out of it.

WARNING:

Always unplug the machine before making adjustments.

linear bearings. It is *very important* that a thin film of lubricant be maintained on the surface of the linear rods to minimize wear. The linear rods wear very quickly with no lubrication.

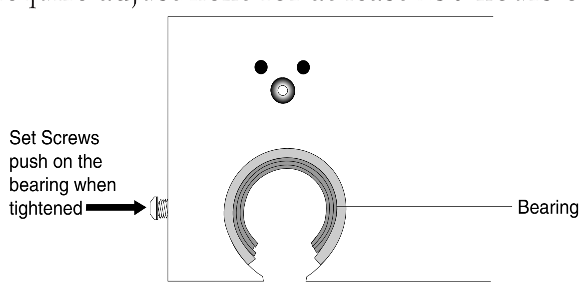
Lubrication

We provided a small oil can with your Machining Center. Use 10W engine oil for lubricating the linear rods.

There are thick oil-impregnated felt pads between the bearings in the saddle which keep a thin film of oil on the linear rods. Each rod (there are six) can be lubricated through its own oil port which is located on the saddle for the X and Y axes, and on the spindle head for the Z axis. You should lubricate the rods every 30 days or 100 hours of use, whichever comes first. When applying oil to the oil ports, pull the trigger on the oil gun very slowly to allow the oil to be soaked up by the felt.

Adjustment

You can adjust the linear bearings to remove any play in the saddle or spindle. The bearings are factory-adjusted and should not require adjustment for at least 250 hours of use. Be



A side view of the Saddle.

very careful not to over-tighten the bushings because over-tightening can cause overworking and overheating of the motor, and excessive wear to the rods, bushings and ball screws. To adjust the linear bushings, follow the steps below.

1. Jog to the extreme positive end of motion on the axis you are adjusting. Stop just before the limit switch trips. Shut off power to the system. Unplug the machine.

CAUTION:

Adjusting the spindle head is a very precise procedure. Do not adjust the spindle head.

2. Remove part of the servo motor cover for the axis you are adjusting. Loosen the screws but do not remove the portion of the cover that has the wires coming out of it. Do not remove the whole cover at once or you may damage the wiring. If you need to use excessive force to turn the lead screw, you have adjusted the bearing too tightly.
3. Loosen all eight recessed allen setscrews on the axis you are adjusting. They are located on the four sides of the saddle for the X and Y axes, and on the sides of the spindle for the Z axis.
4. Tighten the two setscrews for each bearing while checking the force it takes to turn the ball screw. Tighten the setscrews (applying approximately 4-8 in/lb of torque) until the force it takes to turn the ball screw pulley increases when turning it by hand. Then back the setscrews off by 1/8-turn. Repeat this procedure for all four bushings on each axis.
5. Replace the servo motor cover.

Spindle

Two Piece Spindle Head

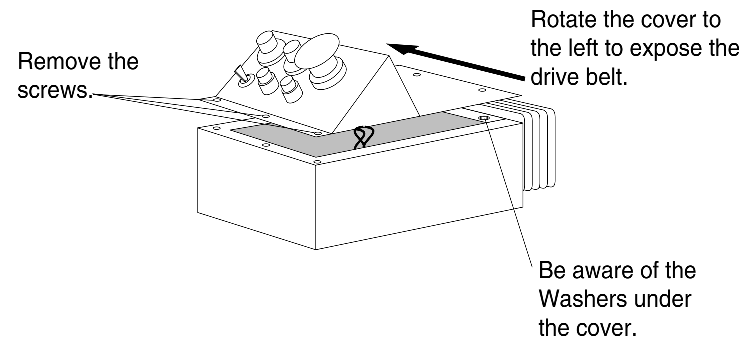
Light Machines ships the proLIGHT Machining Center with a factory-aligned two piece spindle head. You should not attempt to align the spindle head without first contacting Light Machines.

Spindle Assembly

Your proLIGHT 2500's spindle components have been specially lubricated for maintenance free operation. Never attempt to lubricate or otherwise service the proLIGHT 2500's

Warning!

Disconnect Power Cord before removing cover. Dangerous voltages are present when machine is connected.



CAUTION:

Do not over-tighten the drive belt. Excessive tension will damage the motor shaft.

spindle assembly. Only qualified specialists should service your machine's spindle assembly.

Axis Drive Belts

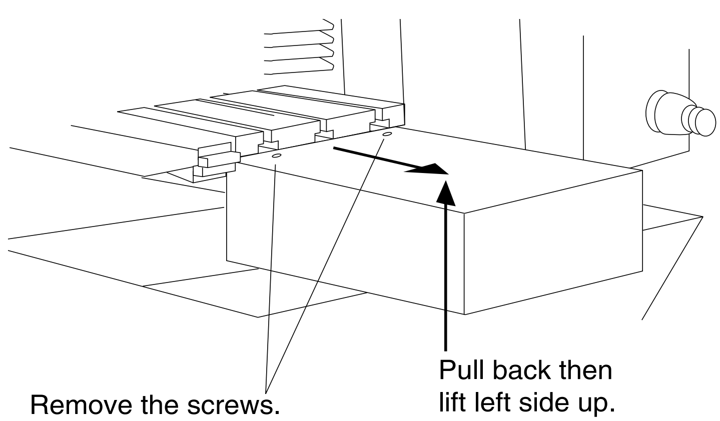
The axis drive belts are located between the servo motors and ball screws on each axis. Normally, they should not need adjustment, but you should check every 250 hours.

Checking and Adjusting the Y Axis Drive Belt:

1. Remove the seven button head cap screws on the motor box cover.
2. Lift the cover and rotate it 90 degrees. Be aware of the two #10 washers under the cover. These washers are used as spacers in the motor box. Be sure not to lose these washers and remember to reinstall them when you are finished checking the belt.
3. Apply approximately three pounds of force on the center point of the belt. It should deflect no more than 1/8-inch (3mm).

If the belt needs adjustment:

1. Remove the two button head cap screws from the bottom of the motor box.
2. Gently move the box up and down while pulling back

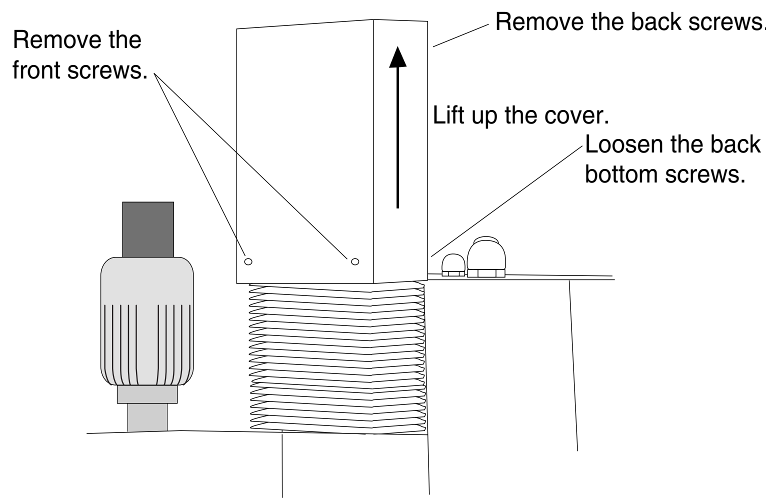


until the box comes off the motor mount.

3. Slide the motor box to the left and down until you have access to the four screws holding the servo motor onto the motor mount.
4. Loosen the four screws.
5. Slide the motor to increase tension on the belt and re-tighten the screws.
6. Check the deflection again.

Checking and Adjusting the X Axis Drive Belt

1. Remove the two button head cap screws from the top of the motor box and the seven button head cap screws from the bottom of the motor box cover.



2. Pull the motor box back approximately 1 inch until it comes off the motor mount. Lift the left side of the motor box up and over the motor to expose the drive belt.
3. Apply approximately three pounds of force on the center point of the belt. It should deflect no more than 1/8-inch (3mm).

proLIGHT Machining Center Routine Maintenance Schedule

MAINTENANCE PERIOD

MAINTENANCE ACTION

	After Every Use	Every 50 Hours or Every 2 Months	After 100 Hours	After 250 Hours
Lubricate Ball Screws		✓		
Lubricate Linear Rod Pads			✓	
Adjust Linear Bushings				✓
Check Spindle Shaft for Play				✓
Check Spindle Belt for Play				✓
Clean Chips Out of Machine	✓			
Coat Exposed Surfaces with Light Oil	✓			
Check Limit Switches and Wiring				✓

If the belt needs adjustment:

1. Loosen the four screws holding the servo motor onto the motor mount.
2. Slide the motor to increase tension on the belt and re-tighten the screws.
3. Check the deflection again.

Checking and Adjusting the Z Axis Drive Belt

1. Remove the two button head cap screws on the front of the motor box and five of the seven button head cap screws on the back. Loosen but leave in place the two bottom screws on the back of the motor box cover.
2. Gently move the box front and back while pulling it up until it is completely off of the motor mount.

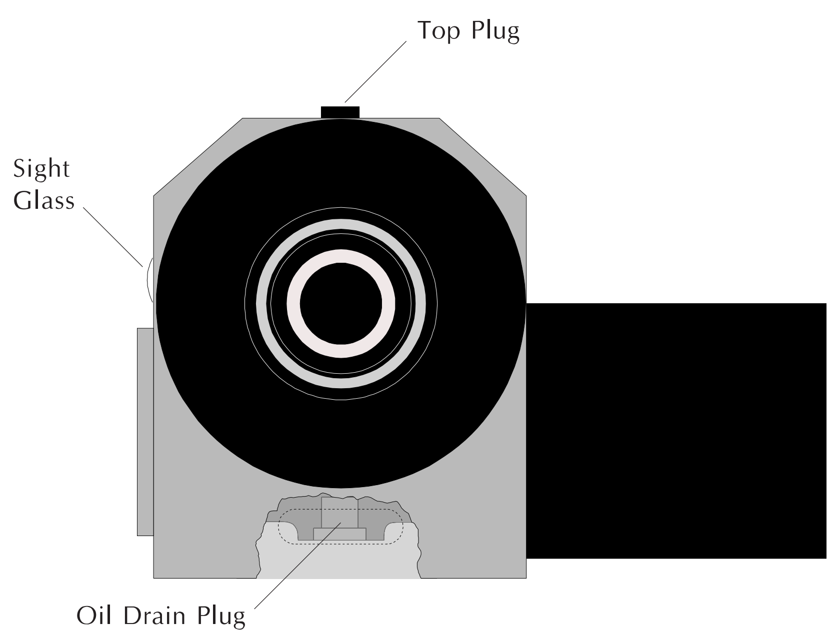
If the belt needs adjustment:

1. Loosen the four screws holding the servo motor onto the motor mount.
2. Slide the motor to increase tension on the belt and re-tighten the screws.
3. Check the deflection again.

Recommended oils or equivalents:
Texaco, Meropa 150
Shell, Omala 100
Mobil, Mobilgear 629

Characterisitics of the above oils:

Agma Grade:	4-EP
Flash Point:	400°F
ISO Grade:	100-150
SUS Viscosity:	725 @ 100°F
	75.5 @ 210°F



Maintaining the Rotary Positioner

If you purchased the 4th Axis Rotary Positioner option for your PLM2000 Machining Center, you must keep it properly lubricated to extend its life and to ensure maximum operational performance.

Lubricating the Rotary Positioner

Oil level and cleanliness should be checked regularly. Drain the oil once or twice a year and refill with new oil. If there is any evidence of contamination, the gear box should be flushed with clean solvent, the seals checked, and replaced if required, and more frequent oil changes considered.

The lubricant must be a premium quality heavy-duty industrial gear oil for enclosed gear sets. The oil must provide good rust and corrosion protection, oxidation stability, foaming resistance, and pressure characteristics which minimize temperature rise.

Since some oil could be lost through the seals, the oil level should be checked weekly or even more frequently should you find it necessary to add much oil at the weekly checks.

Oil is added through the top of the unit after removing the top plug. As oil is added, check the oil level through the sight glass on the side of the unit. When draining oil, loosen the top plug to vent. Replace the drain plug before adding oil.

Maintaining the PC in a Shop Environment

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

Caring for the Computer

Follow these general rules for computer care.

1. 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).
2. Keep liquids (soda, coffee, cutting fluid, grease) away from the computer and peripherals.
3. Keep oil, grease, metal chips and excess dust or cigarette ash away from the computer, keyboard and floppy disks. You should consider erecting a clear plastic shield between the computer and the mill to keep chips off the computer.
4. 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.
5. 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.

1. 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.
2. Keep disks in a disk box or special disk container in-

Appendix A: Control Program Messages

The proLIGHT Control Program displays messages to present important information or, if a problem is encountered during normal operation, to inform you of an error. Usually any problem can be readily corrected without affecting the operation currently in progress.

The following messages are provided (in alphabetical order) with their meaning and, if necessary, a remedy.

4th axis not valid in Verify or 3-axis.
This message appears if you attempt to run a 4-axis part program on a 3-axis machine, or if you try to verify it.

Additional data required.
The canned cycle you have placed in your NC code requires a code other than P, Q, R, or Z.

An error occurred.
This generic error message occur for a number of reasons. You may have to discontinue running your NC program and begin again.

An error occurred while printing.
This message may appear because of a communications error between your computer and your printer. Check the connections between the computer and the printer and try again.

Bad T code.
The currently chosen tool number is not valid. Currently, the Control Program supports up to 20 tools. If your T code is less than 1 or greater than 20, you will get this error.

Bad address.
An invalid character was encountered in your NC code. For example, the exclamation mark shown in the following NC code is invalid: `N5 X1 E40 !3`

Bad code.
An invalid number followed a valid address was encountered in your NC code. Typically, this involves a bad G or M code. For example, the M code in this block makes no sense. N5 M1234

Can't perform circle with radius.
You have entered bad endpoints for the given radius. For example, you may have entered the same start and end points.

Door open.
The safety shield was opened while a program was running, halting the program.

Drive overheat error.
The Machining Center drive motors overheated. Allow the motors some time to cool down, then try running the program again. If this error recurs, call Light Machines Technical Support.

Duplicate subprogram value.
There is more than one block in your subprogram that has an O code of the same value assigned to it.

Emergency stop - keyboard.
You hit the Q-key or the Space Bar on the computer keyboard while the NC program was running, or you hit Quit while the program was paused.

Encoder Error

Error chaining to file.
This will occur if the Control Program is unable to find the file to chain to, and you cancel while it's trying to find the correct file. You may also see this message if an error occurs which prevents the file from loading.

Error parsing for subprograms.
An error occurred while the Control Program was parsing for M98, M99 and O codes. If subprograms are enabled, the Control Program scans your NC program for these codes before you run a program and before you renumber a program.

Error! Subprograms are not enabled!
This error message appears if you attempt to run an NC program containing an M98 or M99 while subprograms are not enabled. You can enable subprograms with the Run Settings command under the Setup Menu.

Fatal error! Can't read state file.
If you see this message call Light Machines Technical Support for assistance.

File not found! Please find it.
This means the file you are attempting to chain to can't be found in the current file's directory. Select ok. The Open File dialog box appears so you can locate the file.

File too large. Aborting.
There is not enough memory available to open the NC file. You must either free up some memory or use a text editor to break the large file into two or more smaller files.

GOTO destination not found.
You specified an invalid block number for M99. There is no block number in your NC program that matches the specified P code.

Hardware ESTOP hit.
This message appears when you press the Emergency Stop button on the Machining Center while running an NC program.

Invalid canned cycle/'G' code.
If you see this message, call Light Machines Technical Support.

Invalid subprogram value.
You specified an invalid subprogram number for M98. There is no O code in your NC program that matches the specified P code.

M22 command format error.
This error message appears if an M22 code is not followed by a filename enclosed by parentheses.

M99P?? must return to main program.
If you perform a GOTO, the block specified by the P code must be before any subprogram (O codes).

Missing numbers.
You have left out a parameter following an address. For example, in the following code, the Y parameter is missing.
N5 X1 Y Z1

Motion will trigger %c%c soft limit. Continuing will disable soft limits.
If software limits are enabled, the Control Program examines each tool movement to see if it will exceed these limits. If limits will be exceeded, you are warned before the move is executed. There will only be one warning. You can continue, which will disable software limits, or you can quit and edit your program or the limits.

Negative limit hit.
Use the Manual Control Panel to move the tool away from the stock, then check your set up and make sure the software was properly initialized.

Normal program stop.
The Control Program detected an M02 code in the NC program and ceased running the program accordingly.

P code required.
G82 and G89 codes require a P code with a dwell value.

Position WRONG! - Select 'CONTINUE' to do a linear move to correct position or 'QUIT' to stop.
You will get this message if you enter a starting block other than zero before running your part program, and the actual position of the machine at that block isn't the same as the position it would be in if the program had started at block 0.

Positioning error.
This occurs if the motors are unable to follow the correct position. This is typically encountered when the feed rate is too high, or if something prohibits machine motion (like plunging the tool through a hunk of steel).

Positive limit hit.
Use the Manual Control Panel to move the tool away from the stock, then check your set up and make sure the software was properly initialized.

Printer is out of paper.
Check the paper level in the printer and try again.

Printing cancelled.
This message appears if you cancel the printing operation.

Q code required.
A G83 code requires a Q code for a peck depth.

Q value is too small.
The peck depth for a Q code must be larger than .001.

R code required.
Canned cycle commands require an R code as a reference point.

Software limit hit.
The machine position exceeds the specified software limits. You will only see this message if software limits are enabled.

Specified radius is too small.
No such arc exists with the given radius. This will occur, for instance, if your start point and end point are the same.

Subprogram max depth exceeded.
Subprograms can have an M98 code imbedded within them to call another subprogram. You may embed subprograms up to 20 levels deep, but the depth cannot exceed 20.

Subprogram not found.
You have specified an M98 with a P code that references an O code that doesn't exist in the NC program.

Time out occurred while printing.
This means either a printing error occurred, no printer is connected, or the printer isn't ready.

Too many subprograms.
You have used more than 20 subprograms (O codes) in your NC program.

Appendices

Appendix B: General Machining Information

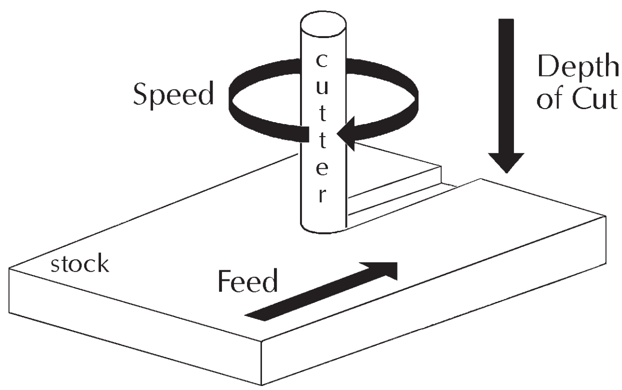
General machining requirements include: setting the feed rate and depth of cut appropriate to the type of stock being used; setting the spindle speed; selecting a lubricant (if necessary); and selecting the proper cutting tool(s) and accessories.

Feed Rate and Depth of Cut

Two terms used in general machining are *feed* and *cut*.

Normal machining on the mill involves removing material from the surface of the 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 proLIGHT Machining Center, the tool does not advance into the workpiece; instead, the cross slide moves the



workpiece beneath the tool. However, the same feed rate principle applies.

The depth of cut is set by the vertical column (Z axis) drive motor and the feed is controlled by the drive motors on the cross slide (X axis) and the end of the bed (Y axis). 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 rapid 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

The relative hardness of the material and the type of cutting tool (end mill or drill) affect spindle 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 machining parts on the proLIGHT Machining Center depends on factors such as: type of material, type of cut, 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.

Note: The proLIGHT Machining Center is not designed for flood cooling. Small amounts of coolant may be applied to the tool tip before a program is run.

Lubricants and Coolants

Lubricants remove heat from the tool and workpiece and are often used when high production rates are required or

Note: Short run small part machining in Delrin or aluminum on the proLIGHT Machining Center does not require the use of coolant.

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 coolant to prevent chips from welding to the tool's cutting edge. Brass and cast iron are always machined dry.

When lubrication is necessary, small amounts of water-soluble cutting fluids are recommended for use on the proLIGHT Machining Center. Lubricants should be wiped from the mill after use, because some petroleum-based fluids may deteriorate the electrical wiring insulation, the plastic safety shield, or the computer enclosure.

Tool Types

CAUTION: Always be careful when handling end mills. They have sharp edges which can easily cut your hands. Use a thick cloth rag when handling tool bits to protect your hands.

Cutting tools are made from hardened steel and are ground to various shapes. Tools are often ground to shape by the operator to suit a particular cutting requirement. 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.

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: end mills, center drills, drills and boring tools.

End Mills

End mills come in two types: flat end mills and ball end mills. They should be sharp and must run true. Holding end mills in a drill chuck is a poor practice; use collets instead. Be certain the spindle speed is set correctly for the type of material being machined. An end mill can be instantly damaged if a cut is attempted at excessive speed.

When plunging into the workpiece, use a center cutting end mill. Center cutting end mills have teeth at the end of the mill going into the center of the mill.

Begin with light cuts and progressively increase cuts until satisfactory results are obtained. End mills should not be used for drilling holes, but they can be used to enlarge holes.

Center Drills

Use small amounts of cutting lubricant with center drills. Clear the drill frequently, otherwise the tip may clog and twist off even in soft materials.

Boring Tools

Use boring tools to enlarge or modify a drilled or cored hole in a workpiece. Mount the workpiece in a vise or on the cross slide. The machine must maintain clearance 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.

Do not drive the tool deeply into a hole. When boring a hole where a flat bottom is required, stop the down feed at least 0.002 inch above the desired depth of the hole being bored out.

Sharpening 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.

Attachments and Accessories

You can equip the proLIGHT Machining Center with a variety of standard attachments and accessories available from industrial supply companies. In addition, Light Machines Corporation offers a Machinist Kit with various tools and tool holding devices.

A machinist should become familiar with the possibilities offered by the selection and use of different mill attachments to take advantage of the mill's versatility.

proLIGHT 2000 Accessories

The following accessories come standard with the proLIGHT 2000 Machining Center.

Hold Down Set

Hold downs are versatile clamps that can be used to secure odd-shaped workpieces to the cross slide or mill bed. The hold down set available for the proLIGHT 2000 Machining Center includes the bolts and nuts that fit the T-slots in the cross slide and mill bed.

End Mill

The proLIGHT 2000 Machining Center comes with a 1/4-inch high-speed steel end mill with a 3/8" shank.

Collet

A collet is used to hold an end mill in the spindle. The proLIGHT 2000 Machining Center comes with a 3/8-inch R8 collet.

proLIGHT 2500 Accessories

The accessories that come standard with the proLIGHT 2500 Machining Center are:

Hold Down Set

Hold downs are versatile clamps that can be used to secure odd-shaped workpieces to the cross slide or mill bed. The hold down set available for the proLIGHT 2500 Machining Center includes the bolts and nuts that fit the T-slots in the cross slide and mill bed.

End Mill

The proLIGHT 2500 Machining Center comes with a 1/32-inch, 1/16-inch, 60-degree high-speed steel end mill with a 1/8" shank.

Collet

A collet is used to hold an end mill in the spindle. The proLIGHT 2500 Machining Center comes with a 1/8-inch EX16 collet.

Appendix C: Safe Machining Center Operation

Safety Rules

The following safety rules should be reviewed and practiced by all operators of the proLIGHT 2000 Machining 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 Machining 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 Machining Center and keep it readily accessible for quick reference. Know the intended applications and limitations of the Machining Center as well as its hazards.

Ground All Tools
The Machining 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 Machining 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 Machining 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 proLIGHT Machining Center
Lock and remove the key from the Machining 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 milling operation. Don't force a tool or attachment to do a job it wasn't designed for.

Dress Properly
Don't wear loose clothing or jewelry which can get caught in moving parts. Wear a hat or 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 vise and the cutting tool to the collet 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 Your Tools In Top Condition
Keep cutting tools sharp and clean. Lubricate and clean Machining Center components on a regular basis.

Disconnect Power 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 Machining Center operation.

Avoid Accidental Starting
Make sure the power switch on the Machining Center is off before plugging in the Machining Center power cord.

Use Recommended Accessories
To avoid stressing the Machining Center and creating a hazardous machining environment, use only those accessories designed for use with the proLIGHT 2000 Machining Center, available through Light Machines Corporation.

Tighten All Holding, Locking and Driving Devices
Tighten the vise. Do not over-tighten tool holding devices. Over-tightening may damage 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.

IMPORTANT!
Post copies of this checklist in the work area. Verify that all items are checked-off prior to each operation of the proLIGHT Machining Center.

Safety Checklist

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 □rea de trabajo. Verifique que todos los puntos esten checados antes de cada puesta en marcha de la m□quina.

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

There are four ways an emergency stop can be initiated on the proLIGHT Machining Center:

- ❑ Press the emergency stop switch on the Machining Center
- ❑ Press a key on the computer keyboard
- ❑ Click the mouse
- ❑ Activate one of the limit switches on the Machining Center.

Refer to Section 3 for detailed information on using the emergency stop methods.

Appendix D: proLIGHT G and M Codes

This appendix provides quick-reference lists of the G and M codes supported by the proLIGHT 2000 Machining Center.

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 (Fanuc uses U,W): All X, Y and Z axes coordinates are relative to a (0,0) location on a mill.

G91 Incremental coordinate programming: Each command is relative to the one before it in the program.

Units Group

G70 Inch: Used to instruct the mill that inches are the unit of measure for the part program. (Fanuc G20)

G71 Metric: Used to instruct the mill 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 to be high: Used in conjunction with H code, which specifies input number. Used for robot synchronization (see Appendix F).

G26 Wait for robot input to be low: Used in conjunction with H code, which specifies input number. Used for robot synchronization (see Appendix F).

G31 Linear to specified coordinate.Stop short if specified input goes High (if H is positive) or Low (if H is negative).

Polar Programming Group

G131	Same as G31. Used for digitizing.
G15	Polar programming cancel.
G16	Begin polar programming.

Coordinate System Group

G53	Rapid traverse to specified coordinates in Absolute programming mode. (e.g. G53X0Y0Z0 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

G80	Canned cycle cancel.
G81	Canned cycle drilling.
G82	Canned cycle straight drilling with dwell.
G83	Canned cycle peck drilling.
G84	Canned cycle tapping.
G85	Canned cycle boring.
G86	Canned cycle boring with spindle off (dwell optional).
G89	Canned cycle boring with dwell.

Preset Position Group

G27	Home: This code moves the tool to its home position on the Machining Center to perform automatic calibration of the axes.
	Check reference point: Compares reported position against zero to see if position has been lost.
G28	Set reference point: Sets machine position to 0,0,0.
G29	Return to reference point: Moves the tool to a coordinate specified by XYZ. Typically used asfter a G27 or G28 code.
G92	Preset position: This code works like the Set Position function under the Setup Menu (see Section 3). The X, Y 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 5).

Plane Selection Group

G99 Rapid move to point R (surface of material or other reference point) after canned cycle complete (Section 5).

G17 Select the X,Y plane for circular interpolation. This is the default plane for circular interpolation. Use this code to switch back to the X,Y plane after circular moves on the X,Z or Y, Z planes. The arc center coordinates are given by I for the X axis and J for the Y axis.

G18 Select the X,Z plane for circular interpolation. Use this code to perform circular interpolation on the X,Z plane. The arc center coordinates are given by I for the X axis and K for the Z axis.

G19 Select the Y,Z plane for circular interpolation. Use this code to perform circular interpolation on the Y,Z plane. The arc center coordinates are given by J for the Y axis and K for the Z axis.

Scaling Group

G50 Cancel scaling.

G51 Invoke scaling. Use this code to scale axes with independent factors around a fixed origin. The default is 1.

Rotation Group

G68 Invoke rotation. Use this code to rotate a geometry from its origin by an arbitrary angle. Rotation works on any plane, one plane at a time.

G69 Cancel rotation.

Cutter Compensation Group

G39 Corner offset in circular interpolation.

G40 Cancel cutter compensation.

G41 Invoke cutter compensation left.

G42 Invoke cutter compensation right.

D Specifies the offset number from the Offset Table.

Tool Length Offset Group

G43 Shifts Z axis in a positive direction by a value specified by H.

G44 Shifts Z axis in a negative direction by a value specified by H.

G49 Cancel Tool Length Offsets.

H Specifies the offset number from the Offset Table.

Tool Length Adjust Group

G45 Increases the movement amount by the value of D.

G46 Decreases the movement amount by the value of D.

G47 Increases the movement amount by twice the value of D.

G48 Decreases the movement amount by twice the value of D.

D Specifies the offset number from the Offset Table.

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 an M02 stop. 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 6.

Accessories On/Off 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:* Closes air vise accessory concurrently with the motion specified in the program block; remains in effect until superseded by M11.

M11 *Unclamp ACC2:* Opens air vise accessory after the motion specified in the program block; remains in effect until superseded by M10.

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.

M22 *Output to file:* Outputs machine coordinates to a file. The proper format for using this code is: M22(*filename*). You're basically naming the file as you create it. You must enclose the name of the file in parentheses for the Control Program to recognize it.

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, ( ).

M47 *Rewind:* Restarts the currently running program; takes effect after all motion comes to a stop.

M98 *Call to subprogram*

M99 *Return from subprogram:* Returns you to the block following the initial M98 command.

Go to: Used with P code. P code defines N or O code destination. Goes to first occurrence of N or O code within the main program.

I/O Group

M25 *Set robot output:* Used for robot synchronization. Used in conjunction with H code to specify output number.

M26 *Set robot output*: Used for robot synchronization. Used in conjunction with H code to specify output number.

M111 *Linear segments:* Causes the proLIGHT 2000 to make arcs as linear segments. Use with Helical Interpolation.

M112 *Arc as linear segments:* Causes the proLIGHT 2000 to make arcs as linear segments. Use with Helical Interpolation.

M122 *Output current position to file:* Writes current X, Y, and Z position to a data file. Use with G131 for digitizing.

M195 *Set part radius:* defines the radius for a four axis part. Used only if the feed per rotation (G95) mode active. The default radius is 1.0. This code is not valid in Verify mode or in 3 Axis mode.

Appendix E: Working in DOS

There are a number of functions in DOS that can be used to alter or create NC programs, or alter the Control Program. You should be thoroughly familiar with DOS before attempting to use the commands and techniques in this chapter.

Environment Variables

Before running the Control Program you can set environment variables to increase efficiency or to establish a more appropriate environment in which to run the Control Program. The environment variables can be typed in at the C> prompt in any directory as long as you haven't started the application.

Graphics Card

The Control Program automatically senses what kind of graphics adapter you have installed in your computer (CGA, EGA, VGA or Hercules). However, in some cases the Control Program may not recognize the graphics adapter you have installed. If this happens, you can force the Control Program to recognize your adapter by using the environmental variable LMGR. For example, to set the environment variable to EGA color, (at the C> prompt) type: SET LMGR=EGA

Unfortunately, you must do this every time you start your computer, before you run the Control Program. You might want to place this command in the AUTOEXEC.BAT file so the computer automatically performs this function for you when you turn it on. Here are some other LMGR settings you can use.

SET LMGR=VGA	Force VGA color mode
SET LMGR=VGAMONO	Force VGA monochrome mode
SET LMGR=EGAMONO	Force EGA monochrome mode
SET LMGR=CGA	Force CGA color mode (only for EGA card with CGA monitor, or some Tandy 1000's with 16-color CGA at 640 x 200 resolution)

SET LMGR=CGAMONO	Force CGA monochrome mode
SET LMGR=HERCMONO	Force Hercules compatible monochrome mode

Allocating Expanded Memory

If you wish to allocate expanded memory to create larger NC programs, you can use the environment variable LMCEMS. You can specify a number of 16K blocks, or pages, to be added up to the amount of available RAM on you computer.

To allocate EMS, type: SET LMCEMS=*number of 16K blocks*
Or use "*All*," "*A*," or "*-1*" to use all avaiable memory.

Setting Permanent Command Line Switches

You may use the SET LMC= command to set up permanent command line switches for the Control Program. When you set this environment variable, it is appended to the command line every time you start the Control Program. For example, 4th axis users can use the command: SET LMCOPTIONS=/4

If you place this command in your AUTOEXEC.BAT file, you won't have to supply the /4 switch every time you run the Control Program.

Managing Machine Communications

Communications Port
The Machining Center communicates to the controlling computer through a serial port on the computer. The default communications port for the Control Program is COM1. If the Machining Center has been connected to a serial port other than the default, an environment variable needs to be set to advise the control software of the new communications port.

To set the environment variable, at the DOS prompt type:

SET COMPORT=*the number of the comport*

This command forces the Control Program to communicate through the designated port when running the machine tool. For example, if the Machining Center were connected to COM2, the environment variable needs to be set as: SET COMPORT=2

The computer then uses COM2 whenever the Control Program is used. This command may be added to the AUTOEXEC.BAT file so the computer automatically performs the function when the computer is switched on.

Baud Rate

The Machining Center communicates with the controlling computer through a serial port on the computer. When communications between the machine tool and the computer break down, a Communications Error results. The Control Program allows for some fine tuning of the communications parameters to allow for slight differences in serial communications to be accommodated.

If communications errors occur when you are attempting to run the Machining Center, fine adjustment of the software may be required. The first thing to adjust when communications problems occur is the Baud Rate.

Note:Using SET BAUD overrides the SETUP file value.

The Machining Center communicates with the controlling computer at a default baud rate of 19,200 baud. For most applications this setting works very well and need not be changed. If you are having persistent problems, the baud rate may be reset to 9,600 baud.

To set the baud to the lower value, type SET BAUD=9600 at the DOS prompt (make sure to insert a space after *SET*). Try running the machine tool again. Communication will have been slowed down slightly, allowing for deviations from a standard serial port.

Other Commands

If problems still persist, other parameters may also be adjusted. Particularly, slight delays can be incorporated into the communications routines when running the machine.

For example, if errors only occur when circular interpolation is being performed, a pause can be incorporated when cutting the arc. This will give the system more time for communicating. The following is a list of variables that may be set:

SET MV_DELAY= Set delay for straight line motion.

SET ARC_DELAY= Set delay for arc motion.

SET POS_DELAY= Set delay for position update.

SET STAT_DELAY= Set delay for status update

SET IOSTAT_DELAY= Set delay for I/O update

These commands are looking for arguments in milliseconds. So, to set a delay of 200 milliseconds when performing arc motions, type SET ARC_DELAY=200 at the DOS prompt.

These commands may be placed in the AUTOEXEC.BAT file, or you may wish to create a separate batch file for fine tuning the machine communications.

Command Line Switches

When starting up the Control Program, you can add the following commands to disable certain functions or perform other tasks automatically.

- Type PLM2000 /DS to disable the spindle.
- Type PLM2000 /BW to set the Control Program to black and white mode.
- Type PLM2000 /4 to set the Control Program for fourth axis machining.
- Type PLM2000 /Bbaud to set the baudrate to baud, where baud=9600 or 19200.
- Type PLM2000 /En to override Set LMCEMS=n.
- Type PLM2000 /Pn to specify the comport on the command line.
- Type PLM2000 /Mmachine to set the machine equal to TEST or PLM.
- Type PLM2000 /T to enable terminal mode as a menu item.

You can use multiple switches at once by typing them in succession.

Splitting NC Files

SPLIT.EXE is an executable DOS program that you can use to divide large NC programs into smaller units. Use SPLIT.EXE when the computer is unable to process your large NC programs. When you type SPLIT at the PLM2000 prompt, the following screen appears:

At the PLM2000 prompt, enter the maximum number of lines for each section of the program you are dividing, followed by the complete path name for the file. The filename, not including the subdirectory, must be 6 characters or less. For example, to split a program called NUPART every 750 lines, enter:

```
NC File Splitter.                    Version 1.21
Copyright (C) Light Machines Corporation, 1988,94   All rights reserved.

Usage: split <max nc lines> <filename>

<max nc lines> is optional. It is normally set to 1000 lines
<filename> must be 6 characters or less (not including subdirectory)

example:   SPLIT 500 C:\NCPROGS\PART
example:   SPLIT \NC\PROGNO.NC

C:\PLM2000>
```

Note: Entering the maximum number of lines is optional. The default is 1000 lines.

SPLIT 750 C:\PLM2000\NCPROGS\NUPART.NC

Press Return. The File Splitter divides the NC program into sections, each containing the number of lines you specify, and labels each section with a number. For example:

C:\PLM2000\NCPROGS\NUPART1.NC

C:\PLM2000\NCPROGS\NUPART2.NC

C:\PLM2000\NCPROGS\NUPART3.NC

The File Splitter inserts an M20 code at the end of each section of the program, followed by the filename of the next section. For example:

M20

C:\PLM2000\NCPROGS\NUPART2.NC

The M20 code automatically “chains” each section of the program together. Now you can run NEWPART.NC by opening and running the first section. The Control Program then loads and executes all of the subsequent chained sections until the entire program is complete.

Using BASIC to Generate NC Code

You can write BASIC programs that generate NC code for use on the proLIGHT Machining Center. A program can be designed to generate NC part programs for polynomial curve cuts, or to accept input from graphics tablets. This is very easily done using Microsoft BASIC and the DOS operating system. The *IBM PC Owner's Manual* and the *Microsoft BASIC Reference Manual* contain more information about the BASIC programming language.

The BASIC program opens an NC program file, prints lines of NC code to that file, and closes the file. This file can then be accessed by loading it into the proLIGHT Control Program. A template for the BASIC program is listed below. The template is stored on your system disk under the name TEMPLATE.BAS.

Note: Any NC file with the same name as the one you specify in line 10 will be deleted when you run this program.

```
10      INPUT "HELLO, WHAT IS THE FILE NAME "; F$
30      OPEN F$ + ".NC" FOR OUTPUT AS #1
40      X = -1.2345:  Z = 0.4321
50      REM
60      REM
70      REM ____ PUT YOUR NC PRINT STATEMENTS HERE
80      PRINT #1, USING "G90X+#.####Y+#.####F2"; X; Z
90      REM
100     REM
900     CLOSE
```

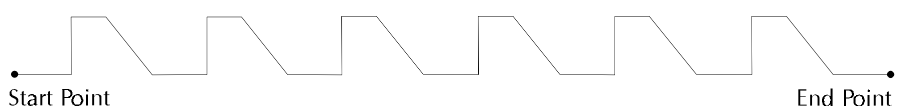
Line 30 opens the file, and line 900 closes it. The bulk of the program should be written between lines 40 and 900. Use print statements like the one shown in line 80 above.

Using BASIC for Repetitive Cuts

REPEAT.BAS is a BASIC program that performs a single cut six times. You enter only four blocks, not the 24 blocks necessary for the NC program. The REPEAT.BAS program is listed below.

```
10      INPUT "HELLO, WHAT IS THE FILE NAME "; F$
30      OPEN F$ + ".NC" FOR OUTPUT AS #1
50      REM
60      X = 6
70      FOR  I = 1 TO X
80      PRINT #1, "G91X.25Y0"
90      PRINT #1, "G91X0Y.2"
100     PRINT #1, "G91X.1Y0"
110     PRINT #1, "G91X.2Y-.2"
120     NEXT I
130     REM
200     CLOSE
```

These are the repetitive cuts generated by REPEAT.BAS.



Line 60 sets the number of repetitions required. In this case, a For-Next loop from line 70 to line 120 is used to repeat the cutting instructions six times. Lines 80 through 110 are the four print statements that contain the NC blocks required to program the cut. Remember that the NC block must be put inside quotation marks.

While in DOS, run REPEAT.BAS. Answer the prompt HELLO, WHAT IS THE FILE NAME? by giving an NC program file name that does not match any existing files. The BASIC program opens a file under that name and writes the print statements (lines 80 through 110) into it. Now you can go into the proLIGHT Control Program and load the program using the filename you entered.

Appendix F: Robotic Integration

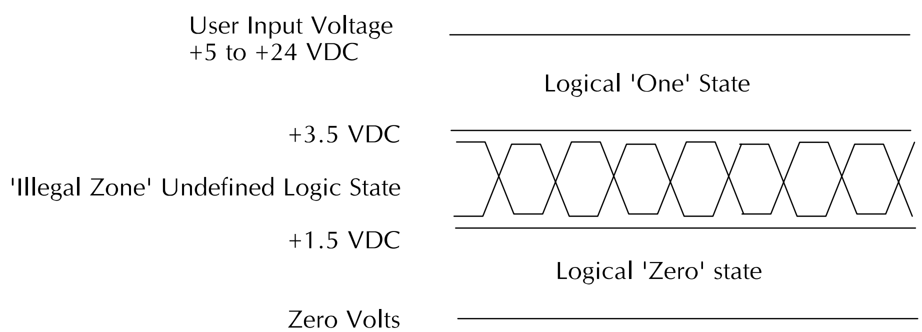
Note: Older versions of the PLM2000 are not compatible with some of the information in this Appendix. Please note that the newer model of the PLM2000 requires a User Supplied input voltage of up to +24 VDC for the robotic interface. Older versions supply their own and will be damaged if any voltage is applied.

The easiest way to determine whether you have a new or old version of the PLM2000 is by the location of the On/Off switch. Older models have the On/Off switch on the rear panel. Newer models have the green and red On/Off switch on the front of the machine.

The proLIGHT Machining Center has a simple interface for interacting with common robots, like those used for automatic part loading between machining operations. The Machining Center and the robot communicate by way of an interface connector (labeled
I/0) located on the rear panel of the Machining Center.

How it Works

The method of communication between the Machining Center and robot is very basic. They are both able to transmit and receive *high* or *low* signals. Since communcation signals are typically 0 and 5 volts, a high signal is 5 volts (or 3.5 volts or greater); and a low signal is 0 volts (or 1.5 volts or less). See the figure below.



There are NC codes that you can place in a part program to instruct the Machining Center to *transmit* a high or low signal to the robot. There are also codes that instruct the Machining Center to *wait* for a high or low signal from the robot.

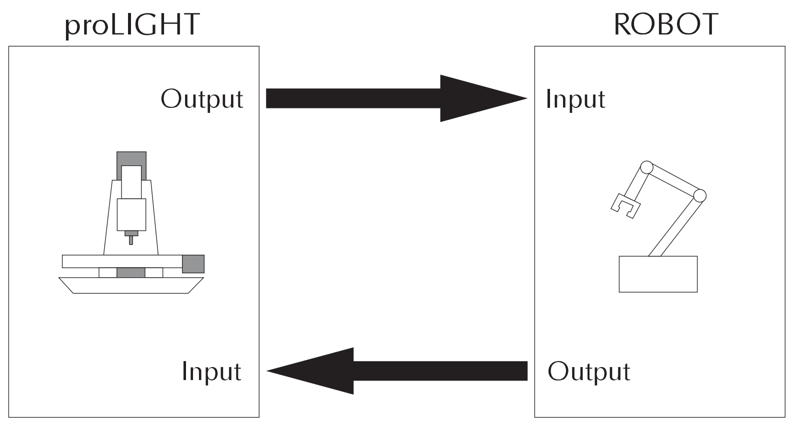
Signals sent out are referred to as *outputs*, while signals coming in are called *inputs*. Any signals that the Machining Center transmits to the robot are transmitted through an *output pin* on the interface connector. Outputs can drive a maximum load of 8mA. Any signals that the Machining Center receives from the robot come in through an *input pin* on the interface connector.

Examples of the input and output wiring are shown in the schematic below.

The Interface Connector

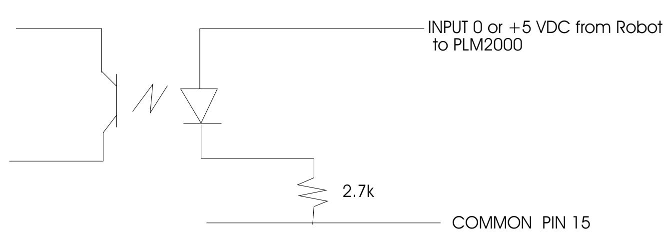
The 15 pin I/O connector on the rear panel provides a number of ways for the Machining Center to communicate with robots and other external mechanisms, such as an additional limit switch. Most of the Machining Center Controller inputs and outputs are located on this connector.

Some of the inputs and outputs are not free for robotic interfacing, they already have dedicated uses; inputs 7 and 8 for

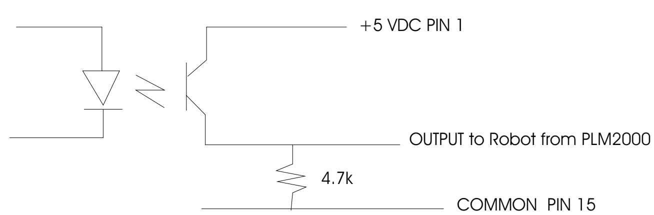


example. The Cycle Start button on the front panel of the Machining Center sends signals to input #8, while the Cycle Stop button sends signals to inputs #7. The inputs and outputs on the Machining Center are designated as follows.

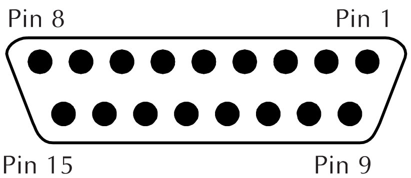
Input Configuration



Output Configuration



The proLIGHT Robot Interface Connector



You can check the status of each input and output by looking at the Inputs Window (active in Manual Control and Run Program only) and Outputs Window in the Control Program.

Other Input/Output Notes:

- ❑ User must supply +5 to +24 VDC between Pins 1 and 15.
- ❑ All User outputs are disabled by an ESTOP condition.
- ❑ All inputs and outputs are optically isolated.
- ❑ Never apply a negative voltage or an AC voltage to an input.
- ❑ Do not connect to pins 7, 8, or 10. These are for special applications only.
- ❑ Outputs are set to source current only. Max load on each output is 8mA.
- ❑ Do not operate relays, motors or other inductive loads with the I/O outputs. This will damage the machine. You may however, run an optical relay.
- ❑ Inputs 4, 5, and 6 are for the high speed spindle and not available on the PLM2500.
- ❑ Input 3 may be configured as an external ESTOP input allowing the machine to be interlocked with a light curtain or other safety device.

Pin	Function	I/O Levels	Name
1	User I/O: Voltage Input	+5 to +24 VDC	Supply Voltage
2	User Output 5	0 or Supply	OUTPUT5
3	User Output 7	0 or Supply	OUTPUT7
4	User Input 1	0 to Supply	INPUT1
5	User Input 3	0 to Supply	INPUT3
6	User Input 5	0 to Supply	INPUT5
7	UNUSABLE		
8	UNUSABLE		
9	User Output 4	0 or Supply	OUTPUT4
10	User Output 6	0 or Supply	OUTPUT6
11	UNUSABLE		
12	User Input 2	0 to Supply	INPUT2
13	User Input 4	0 to Supply	INPUT4
14	User Input 6	0 to Supply	INPUT6
15	User I/O: Common		COMMON

Note: +5 to +24 VDC should only be used on newer models. More than +5 VDC at pins 1 and 15 will damage older models of the PLM2000. See note at beginning of this Appendix for more information.

The NC Codes

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 Machining 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 Machining 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 Machining Center is waiting for a high signal. If there's no change to a high signal, the Machining Center does not execute the next line of NC code in the program.

When the robot sends a high signal, the Machining Center responds by continuing with the NC program. If the Machining 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.

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.

If you do mix the two, be sure not press and hold the Cycle Start button, as it causes any NC program resident on the controller to start running (there is a resident program only when digitizing).

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.

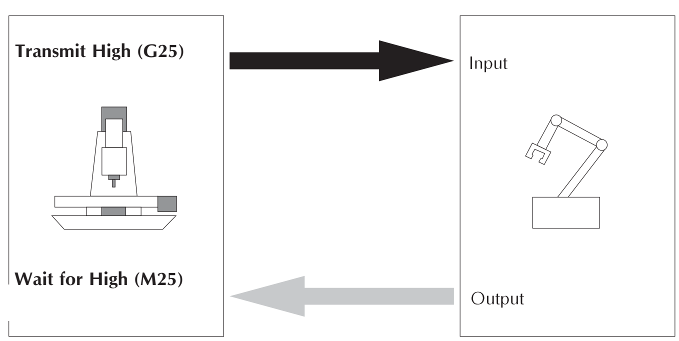
A Sample of Mill/Robot Communication

As previously explained, communication between the Machining Center and robot is very simple. They only use and recognize high or low signals. Once the Machining 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 Machining 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 Machining Center.

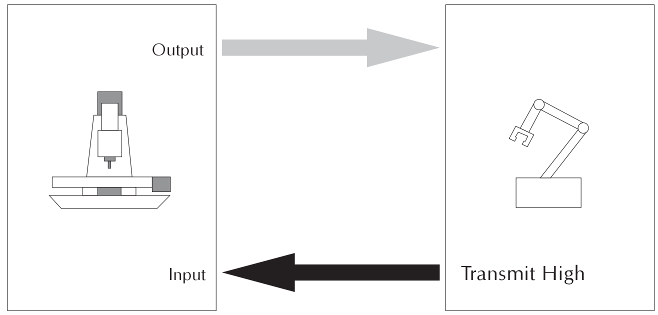
This reciprocal system of communication is made possible by the placement of G and M codes in the part program. The G codes provide the wait instructions to the inputs, while the M codes provide the transmit instructions to the outputs.

The following sequence is typical of communication between the Machining Center and a robot. Since the robot's initial output state is low, the mill's inputs have already been pulled down to a low state. The Machining Center is equipped with an air vise.

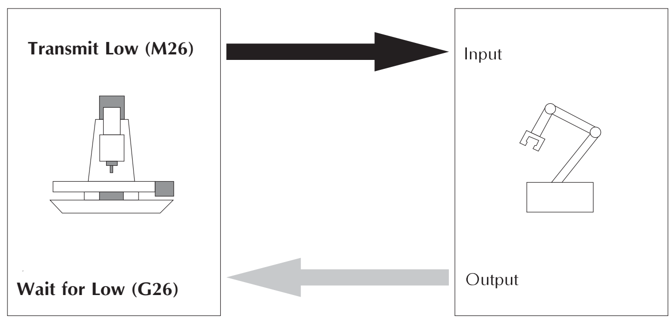
1. The mill opens the air vise (M11), transmits a high signal (M25) to the robot, and waits for a high (on) signal (G25).



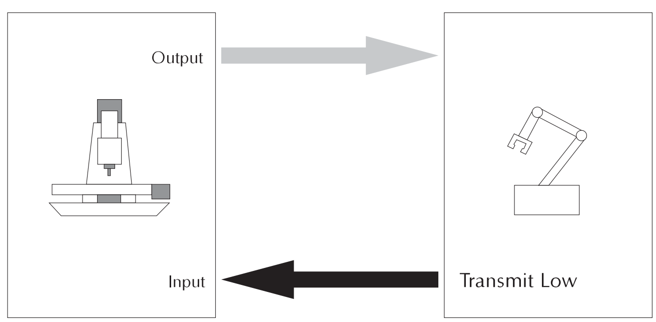
2. The robot places a work piece in the vise and transmits a high signal to the mill indicating that it is okay to close the vise.



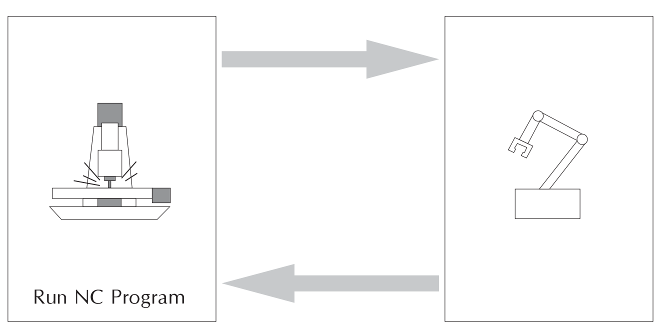
3. The mill receives the signal, closes the air vise (M10), then signals (M26) the robot that the vise is closed. The mill then waits (G26) for a low signal from the robot.



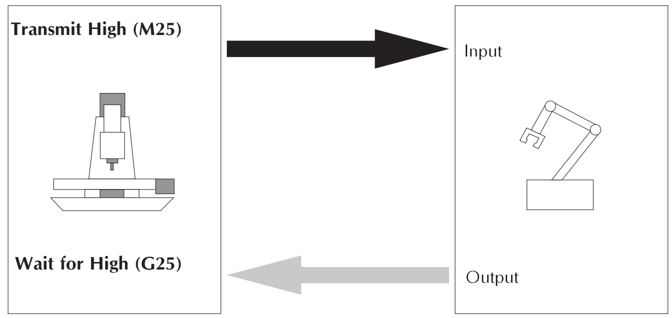
4. The robot releases the work piece and leaves the work area. It transmits a low signal when it is away from the work area.



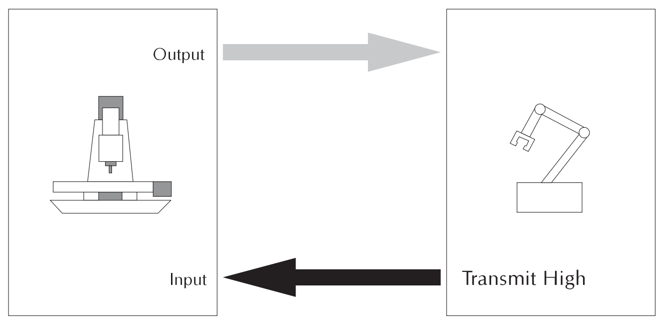
5. The mill receives the low signal from the robot and begins to mill the part.



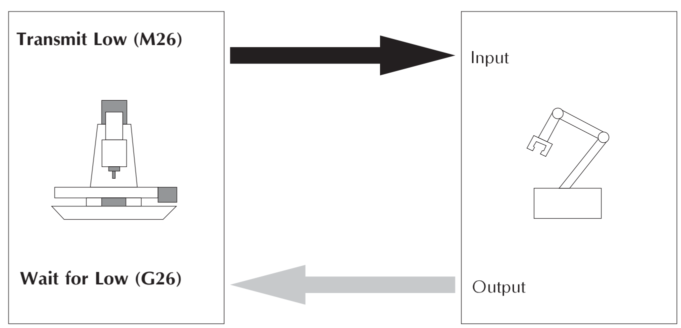
6. When the mill has finished the part, it signals the robot (M25) to come take the part and waits (G25) for a high signal.



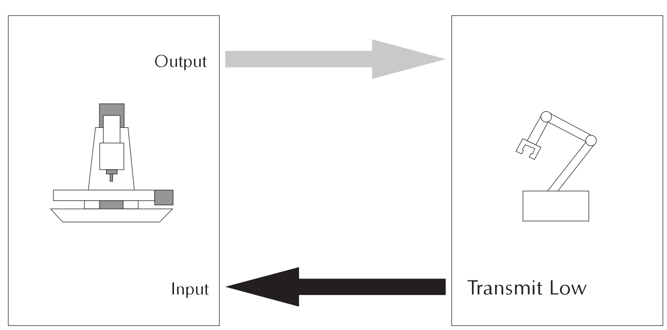
7. When the robot receives the signal, it approaches the mill, grasps the part and sends the high signal to the mill.



8. When the mill receives the high signal, it opens the vise (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 mill's work area, it sends a low signal to the mill.



10. At this point, the cycle begins again (M47).

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 VISE

N1**M25H4**;TRANSMIT HIGH SIGNAL THROUGH OUTPUT 4

N2**G25H3**;WAIT FOR HIGH SIGNAL THROUGH INPUT 3

N3**M10**;CLOSE VISE

N4**M26H4**;TRANSMIT LOW SIGNAL THROUGH OUTPUT 4

N5**G26H3**;WAIT FOR LOW SIGNAL THROUGH INPUT 3

NC Program

```
N0;THIS FILE FOR PLM-2000 MILL
N1;USE 3" X 2" MACHINABLE WAX
N2G05
N3S1500M03
N4G0Z0.0700
N5G0X1.0000Y0.3140
N6G0Z0.0050
N7G1Z-0.1000F4.0
N8G3X1.5830Y1.3330I-0.5990J1.0190F12.0
N9X0.4170I-0.5830J-1.0190
N10X1.0000Y0.3140I1.1820J0.0000
N11G0Z0.0700
N12S1500.00
N13G0X0.8000Y0.7330
N14G0Z0.0050
N15G1Z-0.1000F4.0
N16G3X0.9330Y0.6670I0.1330J0.1010F12.0
N17G1X1.0670
N18G3Y1.0010I0.0000J0.1670
N19G1X0.9330Y1.0000
N20G2Y1.3340I0.0000J0.1670
N21G1X1.0670Y1.3330
N22G2X1.1997Y1.2668I0.0000J-0.1660
N23G0Z0.0700
N24M02
```

Robot Handshaking

N26**M25H4**;TRANSMIT HIGH SIGNAL THROUGH OUTPUT 4

N27**G25H3**;WAIT FOR HIGH SIGNAL THROUGH INPUT 3

N28**M11**;OPEN VISE

N29**M26H4**;TRANSMIT LOW SIGNAL THROUGH OUTPUT 4

N30**G26H3**;WAIT FOR LOW SIGNAL THROUGH INPUT 3

N31M47L3;CYCLE BACK TO N0

N32M2;END OF PROGRAM

Index

Symbols

A

B

C

D

E

F

G

H

S

Y

Z

proLIGHT 2000 Machining Center Accessory List

Accessories

Automatic Tool Changer
This four-station ATC transports tools directly to the spindle, increasing productivity. The ATC includes four tool holders and a set of ER-20 collets: one each of 1/16", 3/16", 1/4", 1/2", and two 3/8". (Factory installed.)

Machining Center Machinist Kit
This option (ACC-5110) provides a 52-piece hold down set, a 3-inch vise for a cross slide with hold down provisions, four R8 collets (1/8", 1/4", 1/2", and 3/4"), seven 4-flute high-speed steel center cutting end mills (1/8", 3/16", 1/4", 5/16", 3/8", 7/16" and 1/2"), two 2-flute high-speed steel ball end mills (1/8" and 1/4"), a zero to 3/8" J2 Jacob's chuck with a key, an R8 to 2JT drill chuck arbor, ball screw lubricant, a 12-inch-long T-handle hex wrench (3/8"), and an edge finder.

End Mill Package
The End Mill Package (ACC-5120) includes: five high-speed steel center-cutting end mills in 1/8", 3/16", 1/4", 5/16", and 3/8" sizes and two high-speed steel ball end mills in 1/8" and 1/4" sizes. All end mills have 3/8" shanks.

Long Flute End Mill Package
This end mill package (ACC-5122) includes five four-flute HSS in 1/8", 3/16", 1/4", 3/8", and 1/2" sizes and three four-flute HSS centercutting ball single end mills in 1/16", 1/4", and 3/8" sizes.

Jacob's Chuck and Arbor
The Jacob's J6 Drill Chuck with R8 to J6 Arbor (ACC-5130) consists of a Jacob's chuck which opens from 0" to 1/2". The chuck comes with a key and an R8 to J6 arbor.

Quick Change Tooling
Quick Change Tooling (ACC-5141) allows easy integration of multiple tools when running an NC program. This package includes a quick change tool body for an R8 spindle, a tool holder stand for up to ten tools, two 3/8-inch end mill tool holders, a 1/4-inch end mill tool holder, a 1/2-inch end mill tool holder, a J2 holder, and an electronic height offset sensor.

Quick Change Tool Holders
There are five sizes of Quick Change Tool Holders to choose from: 1/8" (ACC-5143), 3/16" (ACC-5145), 1/4" (ACC-5147), 3/8" (ACC-5149), and 1/2" (ACC-5151). There is also a J2 Taper Quick Change Tool Holder (ACC-5155).

Low Profile Clamping Kit
This clamp (ACC-5180) adjusts to unusually shaped parts and offers quick cam action clamping for fast set ups and repeatable location. It includes a custom ground and hardened rail with a stop, four clamp base assemblies with bolts, T-nuts and hex keys.

Hold Downs and Vise
The Hold Downs and Vise (ACC-5185) for use with the proLIGHT Series Machining Centers, includes a 52 Piece hold-down set, and a 3" vise with capacity of opening up to 4"

4th Axis 5C Rotary Positioner
This is a complete 4th axis rotary positioner (ACC-5222) that mounts directly onto the machine table. This servo driven rotary positioner incorporates a 5C collet system with a manual collet closer. (Factory installed.)

Digitizing Package
The Digitizing Package (ACC-5261) captures 3-D surface data on existing pieces with depths up to 1 1/2 inches. The package includes a touch signal probe, digitizing software, a 20mm-long stylus with a 3mm- diameter head, a 20mm-long stylus extension, and an 8mm R8 collet.

5C Collet Set
This set (ACC-5370) contains four 5C collets for use with the 4th Axis 5C Rotary Positioner (1/4", 1/2", 3/4" and 1").

5C 3" 3-Jaw Self-Centering Chuck
This industrial grade chuck (ACC-5372), for use with the 4th Axis 5C Rotary Positioner, comes with two sets of hardened steel jaws and a chuck key.

Mobile Workstation with Storage Cabinets
The Mobile Workstation (ACC-5590) provides a laminated 60" wide by 30" deep work surface. The storage cabinets are mounted on a mobile base with four 6" diameter heavy-duty ball-bearing casters. The overall height of the workstation is 31.25".

CAD Engraver Tool Set
The CAD Engraver Tool Set (ACC-5710) consists of one carbide conical nose 60° end mill with 1/8" shank, one carbide 4 flute end mill 1/16" with 1/8" shank, and one carbide 4 flute end mill 1/32" with 1/8" shank.

Vacuum System
The Vacuum System (ACC-5730) consists of a vacuum with noise reduction features, a 1-1/2-inch-diameter 6-foot hose, a nozzle, a mounting assembly, and an electrical relay for NC program control.

Air Vise Robotic Interface
This computer-controlled air vise (PNU-4115) allows automatic holding of work pieces for production applications. The vise opens to a maximum of 3 inches when used with the replaceable vise jaws, or 3-1/4 inches maximum when used without the jaws. The vise comes complete and ready to interface with a standard shop air supply, filtered and regulated from 50 to 125psi.

Portable Air Compressor
The portable air compressor (PNU-4535) is an intermittent duty, 3/4hp, oil-less air compressor with a built-in color-coded regulator and pressure selection chart. It delivers 2.1 CFM at 90 psi.

Machinable Wax Milling Stock
There are two Machinable Wax Stock Packages available for the proLIGHT Machining Centers. PKG-9110 includes 8 large blocks and PKG-9120 includes 50 small blocks of blue machinable wax.

CAM Software

spectraCAM Milling and Turning
spectraCAM is an industrial CAD/CAM package that generates part programs that can be run on the proLIGHT 2000. spectraCAM can import part geometry from CAD programs that export DXF files (such as AutoCAD). spectraCAM is available for both MS-DOS-compatibles (CAM-6601) and Macintosh computers (CAM-6602).

spectraCAM Milling for Windows
spectraCAM for Windows (CAM-6721) is a CAD/CAM part programming package designed for use with all Light Machines' CNC machining centers.

The included CAD program, spectraCAD, uses the simplicity of graphic buttons and tool bars. A variety of drawing and editing tools assist in constructing geometry easily, quickly, and precisely. spectraCAD can display the geometry in a number of different ways and provide printed output for inspection. Bidirectional DXF file transfer capability makes spectraCAD and spectraCAM compatible with other applications like AutoCAD.

Curriculum/Software

enLIGHT Milling Volume 1
An introductory curriculum package (CUR-7111) that takes students through the CAD/CAM/CNC process to produce a machined part. Includes a computer program, a Student Edition of AutoSketch, Light Machines' CAD to NC program, Instructor's and Student's Resource Guides and three machining part projects.

The projects included with the enLIGHT packages are based on core curriculum subjects to reinforce basic disciplines such as math, science, and engineering, with an emphasis on problem-solving skills.

Introduction to CNC Video Package
The Introduction to CNC Video Package (Milling, DOC-7911) is one of many curriculum packages that Light Machines offers as part of the Manufacturing Technology Interactive (MTI) series of products.

This comprehensive package describes and defines CNC technology, reviews its history, examines its uses, studies manufacturing operations, presents career opportunities, and applies problem solving skills through activities and projects.

Introduction to CNC Workbook (Student Edition)
A student workbook (DOC-7531) presenting topics such as using the mill, coordinate systems, NC programming, editing part programs, and verifying and running part programs. Includes CNC milling projects.

Introduction to CNC Workbook (Teacher Edition)
This teacher's workbook (DOC-7541) 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 milling projects on 3.5" disks.

N-See Tutor
N-See Tutor (CUR-6581) provides an interactive environment for learning to write instructions for NC machine tools. Each time you enter a line of NC code, N-See Tutor simulates the tool moving through the workpiece as though the part is actually being machined.

N-See Tutor includes computer-based interactive lessons that teach NC programming structure, G&M codes, part planning, and coordinate systems.