

**Benchtop CNC Router
User's Guide
For Windows**

**© 1997 Light Machines Corporation
All Rights Reserved.**

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 (34-7751-0000, February, 1997) corresponds to the LMR-8800 Control Program software.

Printed in U.S.A.

All Register marks or trademarks are of their respective holders

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 Benchtop CNC Router.

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

Benchtop CNC Router
User's Guide
For Windows

**Benchtop CNC Router
User's Guide
For Windows**

**© 1997 Light Machines Corporation
All Rights Reserved.**

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 (34-7761-0000 April, 1997) corresponds to the LMR-8800 Control Program software.

Printed in U.S.A.

All Register marks or trademarks are of their respective holders

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 Benchtop CNC Router.

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

Table of Contents

User's Guide

Section A: Installation

Getting Ready for Installation	A-2
Check Your Shipment	A-2
Register Your Benchtop Router	A-2
Prepare Your Work Place	A-3
Unpack the Benchtop Router	A-3
Set Up the Benchtop Router	A-3
Hardware Installation	A-4
Installing the Interface Card in the PC	A-4
Opening the PC Chassis	A-5
Unpacking the Interface Card	A-5
Inserting the Interface Card	A-5
Checking Your Installation	A-6
Connecting the Benchtop Router	A-6
Interface Connections	A-7
Hardware Connections	A-8
Optional Connections	A-8
Computer System Requirements	A-9
Software Installation	A-9
Installing the Control Program	A-9
Uninstalling the Control Program	A-10
The Setup Program	A-10
Technical Support	A-11
Before Calling	A-11
Warranty	A-11

Section B: System Hardware

The Benchtop CNC Router	B-2
Features	B-2
Machine Specifications	B-2
The Accessory Kit	B-3
The Router Components	B-4
The Front Panel	B-5
The Rear Panel	B-5
Maintaining the Benchtop Router	B-6
Cleaning and Lubrication	B-6
Ballscrews	B-6
The Dust Collection System	B-8
Maintaining the Controller Box	B-9
Maintaining the PC in a Shop Environment	B-10
Caring for the Computer	B-10
Caring for Floppy Disks	B-11
Benchtop Router Options	B-12

Section C: Getting to Know the Control Program

Starting the Control Program	C-2
Starting the Control Program in Simulate Mode	C-2
If You Need Help.....	C-3
Exploring the Control Program Screen	C-4
Menu Bar	C-4
Standard Tool Bar	C-4
Outputs Tool Bar	C-6
Inputs Tool Bar	C-7
Edit Window	C-8
Status Bar	C-9
Position Readout	C-10
Machine Info Panel	C-10
Verify Window	C-11

Section D: Tutorial

Safely Running the Benchtop Router	D-2
Safety Rules	D-2
Making Emergency Stops	D-3
Stopping with the Emergency Stop Button	D-3
Stopping with the Computer Keyboard	D-3
Stopping with a Limit Switch	D-4
Stopping with the Safety Interlock Switch	D-4
Running a Sample NC Program	D-5
Open ROUTER1.NC	D-5
Adjust the Verify Settings	D-6
Adjust the View	D-6
Adjust the Stock	D-7
Define the Tool	D-8
Add the Tool to the Library	D-8
Select the Tool for Verification	D-9
Installing a Tool	D-10
Verify ROUTER1.NC	D-11
Dry Run the NC Program	D-12
Mount the Workpiece	D-14
Run the Program	D-14

Reference Guide

Section E: Control Program Reference

The Benchtop Router Interface	E-2
Using the Message Bar	E-3
Using Windows	E-3
Program Edit Windows	E-3
Verify Window	E-4
Machine Info Window	E-4
The Position Window	E-5

Using Tool Bars	E-6
Standard Toolbar	E-6
Inputs Toolbar	E-7
Outputs Toolbar	E-8
Panels	E-9
Jog Control Panel	E-9
Operator Panel	E-10
Status Bar	E-11
Menu Bar	E-12
File Menu	E-12
New Command	E-12
Open Command	E-13
Close Command	E-13
Save Command	E-14
Save As ... Command	E-14
Print Command	E-15
Print Setup Command	E-15
Opening a Recent Program	E-16
Exit Command	E-16
Edit Menu	E-17
Undo Command	E-17
Redo Command	E-18
Cut Command	E-18
Copy Command	E-18
Paste Command	E-18
Clear Command	E-19
Delete Line Command	E-19
Find Command	E-19
Replace Command	E-20
Goto Line Command	E-20
Renummer Command	E-21
Insert N Codes	E-21
Insert or Remove Spaces	E-22
Remove Comments	E-23
Remove N Codes	E-23
Lock Command	E-23
Select Font Command	E-24

Font Settings	E-24
View Menu	E-24
Position Command	E-24
Machine Info Command	E-25
Jog Control Command	E-25
Operator Panel Command	E-25
Verify Window Command	E-25
Toolbars Command	E-25
Program Menu	E-26
Run/Continue Command	E-26
Verify Command	E-27
Estimate Runtime Command	E-28
Pause Command	E-29
Feedhold Command	E-29
Stop Command	E-29
Tools Menu	E-30
Setup Library Command	E-30
Setup Tool Wizard Command	E-32
Select Tool Command	E-32
Setup Menu	E-33
Set Position Command	E-34
Zero Position Command	E-34
Jog Settings Command	E-35
Run Settings Command	E-36
Verify Settings Command	E-38
Verify Type Command	E-41
Set/Check Home Command	E-41
Goto Position Command	E-42
Units Command	E-42
Coordinate Systems Command	E-42
Offsets Command	E-44
Spindle Command	E-45
Backlash Command	E-46
Soft Limits Command	E-46
Preferences Command	E-47
Window Menu	E-49
Cascade Command	E-49
Tile Command	E-49

Arrange Icons Command	E-49
Window List Command	E-50
Help Menu	E-50
Help Command	E-50
Index Command	E-50
Using Help Command	E-50
Save Settings Command	E-50
Restore Settings Command	E-51
Tip of the Day Command	E-51
About BenchRouter ... Command	E-51
Selecting Commands	E-52
Select a Command Using Pop-Up Menus	E-52
Program Edit Window Pop-up Menu	E-52
Position Window Pop-up Menu	E-53
Verify Window Pop-up Menu	E-54
Jog Control Panel Pop-up Menu	E-55
Select a Command Using Hot Keys	E-56
Select a Command Using Toolbars	E-57
Positioning Screen Components	E-58
Positioning Toolbars	E-58
Positioning Information Areas	E-58
Positioning Program Edit Windows	E-59
Saving the Window Positions	E-59
Docking and Floating Windows and Toolbars	E-60
Docked Screen Components	E-60
Floating Screen Components	E-60
The Setup Program	E-61
Welcome Panel	E-61
Interface Card Panel	E-62
General Panel	E-62
Offset Table	E-63
Working in Simulation Mode	E-63

Section F: Basic CNC Programming

The Elements of an NC Part Program.....	F-2
Categories of NC Code.....	F-3
Incremental Arc Center (%).....	F-4
Absolute Arc Centers (\$).....	F-4
Skip (\) and Optional Skip (/).....	F-4
Compensation Offset Value (D Code).....	F-5
Feed Rate (F Code).....	F-6
Preparatory Codes (G Codes).....	F-6
The Interpolation Group.....	F-7
The Units Group.....	F-7
The Plane Selection Group.....	F-7
The Wait Group.....	F-8
The Canned Cycle Group.....	F-9
The Programming Mode Group.....	F-9
The Preset Position Group.....	F-9
The Compensation Functions Group.....	F-10
The Coordinate System Group.....	F-11
The Polar Programming Group.....	F-11
Input Selection Number/Tool Length Offset (H Code).....	F-12
X Axis Coordinate of Center Point (I Code).....	F-12
Y Axis Coordinate of Center Point (J Code).....	F-13
Z Axis Coordinate of Center Point (K Code).....	F-13
Angle of Arc Resolution, Loop Counter (L Code).....	F-13
Miscellaneous Codes (M Codes).....	F-14
M22: Output Current Position to File.....	F-16
M99: Return from Subprogram, Goto.....	F-17
M105: Operator Message.....	F-18
Block Number (N Code).....	F-19
Subprogram Block Number (O Code).....	F-19
Subprogram Reference Number (P Code).....	F-20
Peck Depth (Q Code).....	F-20
Radius of Arc, Drilling Start Location (R Code).....	F-20
Spindle Speed (S Code).....	F-20
Tool Selection (T Code).....	F-21

X Axis Coordinate (X or U Code)	F-21
Y Axis Coordinate (Y or V Code)	F-21
Z Axis Coordinate (Z or W Code)	F-21
Comment Codes	F-22
General Programming Suggestions	F-23

Section G: More CNC Programming

Linear Interpolation Programming	G-2
Circular Interpolation Programming	G-3
Circular Interpolation on Other Planes	G-4
Helical Interpolation Programming	G-5
Rapid Traverse Programming	G-6
Canned Cycle Programming	G-7
Using G80	G-7
Using G81	G-8
Using G82	G-9
Using G83	G-9
Using G84	G-10
Using G85	G-11
Using G86	G-11
Using G89	G-12
Subprogram Programming	G-13
A Sample NC Subprogram	G-14

Section H: Advanced CNC Programming

Using Polar Programming	H-2
Using the Homing Commands	H-3
Using G28	H-3
Using G28 in an NC Program	H-3
Using G28 Before Setting Soft Limits	H-4
Using G27	H-4
Using G29	H-5

Using Cutter Compensation	H-6
Starting Cutter Compensation (G41/G42)	H-7
Cutter Compensation with IJK Vectors	H-10
Setting Cutter Compensation Offsets (D)	H-11
Changing Offset Values	H-12
Changing Offset Sides	H-13
Using Corner Offset Circular Interpolation (G39)	H-14
Canceling Cutter Compensation	H-15
Methods 1 and 4	H-15
Methods 2 and 5	H-16
Methods 3 and 6	H-17
Using Scaling and Rotation Codes	H-18
Scaling	H-18
Uniform Scaling	H-18
Scaling Each Axis	H-19
Creating Mirror Images with Scaling	H-20
Rotation Codes	H-21
Combining Scaling and Rotation Codes	H-23
Multiple Tool Programming	H-24
Using Multiple Tool Codes	H-24
Establishing the Reference Tool	H-25
Establishing Tool Offsets	H-25
Testing Your Multiple Tool Program	H-26
Understanding Coordinate Systems	H-27
Machine Coordinates	H-27
Work Coordinates	H-27
Multiple Coordinate Systems	H-27
Using Tool Length Offset Codes	H-28
Using Tool Offset Adjust Codes	H-29
G45	H-29
G46	H-30
G47	H-30
G48	H-30

Section I: Safe Router Operation

Safety Rules	I-2
Emergency Stops	I-4
Safety Checklist	I-5
Lista de Seguridad	I-6

Section J: G and M Codes

G Codes by Group	J-2
M Codes by Group	J-5

Section K: Robotic Integration

Robotic Integration	K-2
How it Works	K-2
The Interface Connector	K-4
The NC Codes	K-5
A Sample of Router/Robot Communication	K-6

Index

User's Guide

Section A: Installation

Section B: System Hardware

Section C: Getting to know the Control Program

Section D: Tutorial

Reference Guide

Section E: Control Program Reference

Section F: Basic CNC Programming

Section G: More CNC Programming

Section H: Advanced CNC Programming

Section I: Safe Router Operation

Section J: G and M Codes

Section K: Robotic Intergration

**Benchtop CNC Router
User's Guide
For Windows**

Installation

Getting Ready for Installation

Hardware Installation

Computer System Requirements

Software Installation

Technical Support

Getting Ready for Installation

Before you connect and run your new benchtop router, you should:

1. Check your shipment to make sure you received everything you need.
2. Register your benchtop router so you are covered by your warranty.
3. Prepare a workspace for your benchtop router and controller.
4. Unpack and set up your benchtop router.
5. Install the interface card into the computer.

Once all of these procedures are complete, you can connect the benchtop router and controller box, then install the Control Program.

Check Your Shipment

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

Register Your Benchtop Router

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

Note:

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

IMPORTANT!

Be sure to keep the pallet and all of the original cartons in which the benchtop router 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 the original cartons.

CAUTION

The router weighs approximately 300pounds (170 kg), so be very careful when lifting it.

Prepare Your Work Place

Make sure you have all the items on hand necessary to perform the installation. To install the benchtop router, you must have:

- ◆ A sturdy bench on which you'll place the benchtop router and your computer. Placing the bench against a wall provides more stability. Make sure the wall has a 120VAC, 15 amp polarized outlet.
- ◆ A personal computer running Windows 95 or Windows NT version 3.51 (or greater). See page A-10 for a complete list of the necessary computer equipment.
- ◆ Your *PC Owner's Manual* or equivalent documentation.

Unpack the Benchtop Router

1. Position the pallet near the bench on which you'll set the benchtop router. The bench 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 of the pallet.
4. Inspect the benchtop router chassis for visible signs of 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).

5. Using a 9/16" wrench, remove the four bolts holding the two 2x4 braces holding the router on the pallet. Keep the bolts and other packaging materials in case you need to return the router to the factory.

Set Up the Benchtop Router

1. Using the four handles, carefully lift the benchtop router off of the pallet and onto the table.
2. Once on the table, position the benchtop router correctly for machining.
3. Remove the protective paper from the safety shield.

Hardware Installation

The following paragraphs review the procedures for installing the hardware components of the benchtop router. You should already have your computer system set up in according to the directions in the computer owner's guide.

The first step is to install the Interface Card in your personal computer.

WARNING

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

Note:

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

Installing the Interface Card in the PC

The following paragraphs describe the procedure for installing the Interface Card in the chassis of your PC. You should have already set up your PC according to the instructions in your Owner's Manual. The Interface Card can be installed in any slot designated for expansion card use. Refer to your computer owner's manual to determine particular expansion card restrictions.

WARNING

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

Opening the PC Chassis

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

WARNING

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

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

Set the cover aside and locate the slot where the card is to be mounted. Remove the blank slot cover (if any). Removing the slot cover requires removing a screw at the top rail of the rear panel. You may choose to discard the cover, but save the screw for installing the Interface Card.

Unpacking the Interface Card

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

Inserting the Interface Card

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

1. Grasp the Interface Card at the front and back.
2. Position the card above the bus connector at the chosen slot. The interface connector on the end of the card should face the rear panel of the computer chassis.

3. Slide the card into the bus slot. The connector on the card should protrude from the rear panel of the computer. Carefully wiggle the card back and forth to assure its tightness in the bus. Components on the Interface Card should not touch adjacent cards or other components.
4. Secure the card to the top rail of the rear panel with the screw you saved when removing the blank slot cover.
5. Make note of which slot the Interface Card is plugged into. Do not get it mixed up with the parallel port which uses the same type of connector.

Checking Your Installation

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

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

When the Windows program is running, select shut down from the start menu, turn off the power, and install the other hardware components.

Connecting the Benchtop Router

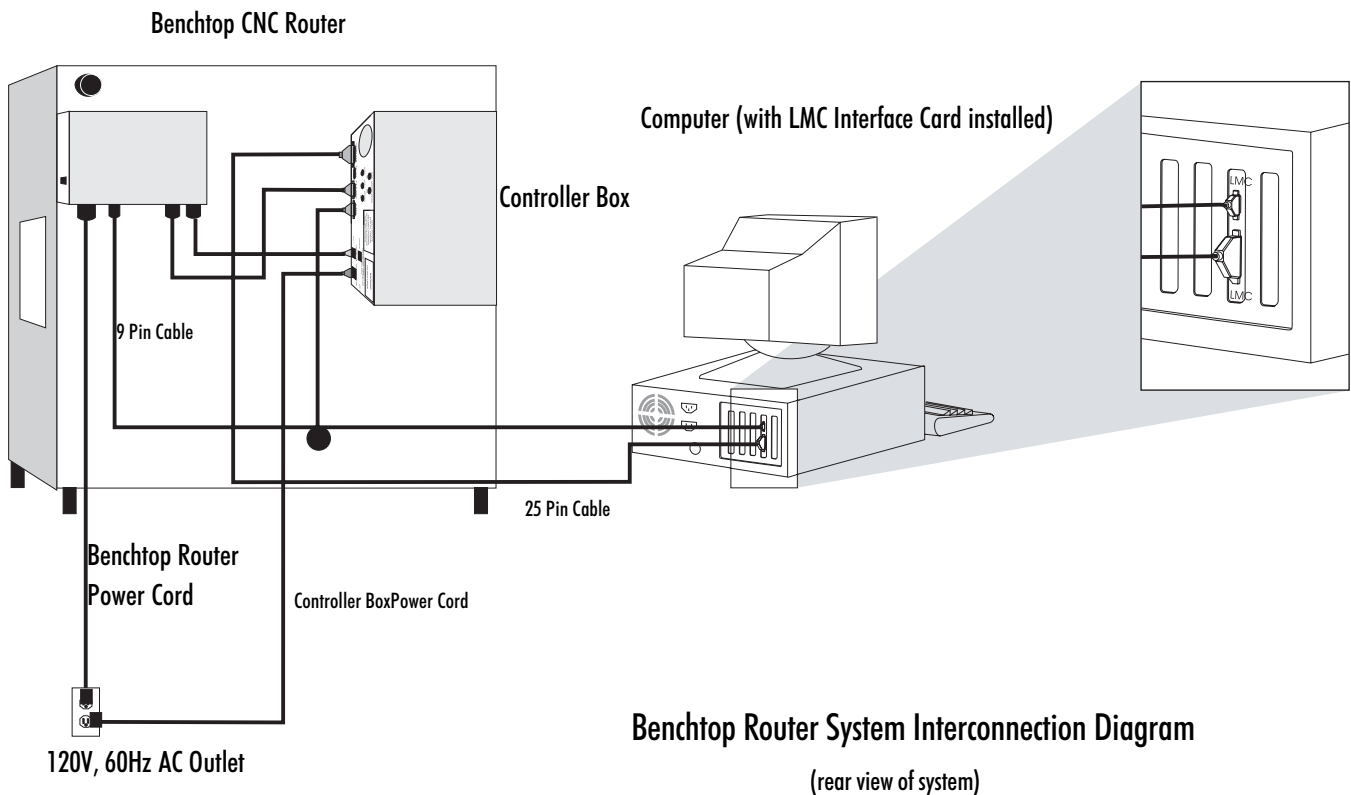
The following paragraphs review the procedures for connecting your computer to the benchtop router controller box. The diagram on the following page has been provided as a visual aid for the recommended connections.

WARNING

Do not connect power to the Benchtop Router until instructed to do so in the following procedures.

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

Operate the Benchtop Router with all cables firmly secured.



Interface Connections

There are four connections that must be made in order to operate the Benchtop Router; two interface connections and two power cords. The interface cables are long enough to allow the computer to be located up to five feet away from the router. Make sure the router is placed on a stable, flat surface and leveled properly.

1. Route the interface cables from the router to the computer.
2. Insert the 9-pin plug into the 9-pin receptacle protruding from the LMC Interface Card in the rear of the computer.
3. Connect the end of the cable marked COMPUTER to the 25-pin connector protruding from the interface card in the rear of the computer. Be sure you are plugging the cable into the interface card connector, not the parallel port.
4. Make sure all connectors are secured with screw locks.

CAUTION

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

International Users:

If you need to change the plug on the end of the electrical cord to accommodate 220V, 50 Hz service, the wiring is color coded as follows:

- Line: brown
- Neutral: blue
- Ground: green w/ yellow stripe

Note:

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

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

Hardware Connections

Both the Benchtop Router and the Controller Box must be plugged into grounded 120VAC, 60Hz, 15A polarized outlets. These outlets must be capable of supplying up to 12 amps of power to the machine and controller box.

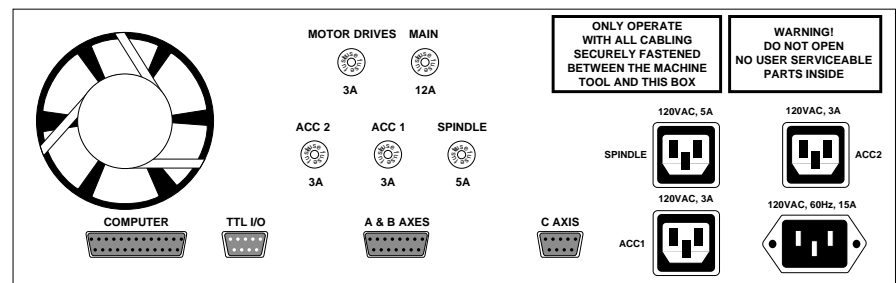
Locate the power cord for the Controller Box and the power cord for the Router. Insert the plug ends of these cords into a grounded, three-hole, 120VAC, 60Hz wall outlet.

International users should connect to a grounded 220VAC, 50Hz, 8A polarized outlet.

Optional Connections

You can attach a 120VAC accessory to your machine. Plug the accessory's power cord into the receptacle end of another accessory port adapter cable. Insert the plug end of the cable into the receptacle labelled ACC 1 on the rear of the Controller Box; the current draw for such accessories, however, is limited to 3 amps.

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



Computer System Requirements

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

- ◆ Windows 95 or Windows NT 3.51 or higher. (Windows NT users: see the readme file for the latest information regarding available accessories.)
- ◆ 8 MB RAM minimum for Windows 95 (16 MB is recommended for best performance). 16 MB RAM minimum for Windows NT.
- ◆ A 3.5" floppy drive.
- ◆ A hard drive with at least 5MB of available space.
- ◆ An available expansion slot.
- ◆ A VGA graphics controller and monitor.
- ◆ A Windows-compatible mouse.

CAUTION

The master disks are shipped write-protected (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 disks. Always store disks in a safe place away from heat, sunlight and static.

CAUTION

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

Software Installation

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

Installing the Control Program

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

1. Turn on the computer. Wait for it to go through its internal checks and for it to complete the start up process.
2. When your Windows desktop appears, insert the disk in the computer floppy drive.
3. Using the Windows Explorer, (Start Menu>Programs>Windows Explorer) open the floppy drive. Note: If you are installing on Windows NT, use either the File Manager to access the floppy drive, or select "Run" from the Program Manager.
4. Double click on Setup.EXE to start the installation.
5. The Welcome screen appears warning you to exit all other running programs. If no other programs are running, click Next.

6. The next screen is the destination directory for the control program. If you would like it put somewhere other than the default, click on Browse. Otherwise, click Next.
7. A window showing the progress of the installation will appear, and when ready, will prompt you to install the second disk. After the installation is complete, a message will appear asking if you would like to view the Readme file. It is suggested that you do view this file. It contains important information about the software and the machine that may not be included in the manual.
8. After reading the Readme file, remove the disk from the floppy drive. In Windows 95, you may run the control program by double clicking the program icon. From Windows NT, you will need to reboot the system.

Note:

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

Uninstalling the Control Program

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

To uninstall the control program software, just double click the Uninstall icon in the control program folder. A message will appear asking if you are sure you want to remove the program and all its files. Click on Yes to uninstall, or Cancel to exit the Uninstall program.

The Setup Program

The Control Program software automatically sets most variables for you. If you need to access the Setup Program, see the Reference Guide: Section E in this manual.

Technical Support

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

Before Calling

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

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

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

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

IMPORTANT!

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

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

Warranty

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

System Hardware

The Benchtop CNC Router

Maintaining the Benchtop Router

Maintaining the Controller Box

Maintaining the PC in a Shop Environment

Benchtop CNC Router Options

The Benchtop CNC Router

The Benchtop CNC Router is a three-axis wood router you can run directly from your personal computer.

The Windows-based control program, which you install on your computer, accepts standard EIA RS-274D, ISO, and Fanuc G&M code CNC programs.

If used with Light Machines' spectraCAM software, it's easy to create designs 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.

Features

Some of the Benchtop CNC Router's features include:

- A 1.75 horsepower motor
- Spindle speeds from 10,000 to 25,000 RPM
- Rapid traverse rates up to 100 ipm
- EIA RS-274D standard G&M code programming
- Aluminum machine base
- Feed rate and spindle speed override functions
- A built-in full-screen NC program editor
- An on-line help utility

Machine Specifications

Axis Travel

X Axis	22.50" (571mm)
Y Axis	24.00" (610mm)
Z Axis	4.00" (100mm)
Open Height	4.75" (120mm)

Router Work Area

Table Size	24" x 32" (610mm x 812mm)
Table load capacity	100 lbs. (45 kg)

Spindle

Drive motor	1.75 hp (1300 watts)
Speed Range	10,000 - 25,000 RPM
Spindle Nose	1/2"
Collet Capacity	1/4" and 1/2"

Feed Motors

Linear Feed Rate	0.1 - 60 ipm (2 - 1525mm/min)
Circular Feed Rate	0.1 - 40 ipm (2 -1015mm/min)
Rapid Feed Rate	100 ipm (2540mm/min)

Power Requirements

120 VAC +10% - 15% (U.S. only)
47-63 Hz single phase, 3 wire
15 Amps / 1.8 KVA

Dimensions

Width	32.5" (825mm)
Depth	47.0" (1194mm)
Height	34.0" (864mm)

Weight (Approximate)

Net	420 lbs (190 kg)
Shipping	520 lbs (236 kg)

The Accessory Kit

The Accessory Kit included with the Benchtop Router contains all the tools and hardware necessary for installation and maintenance.

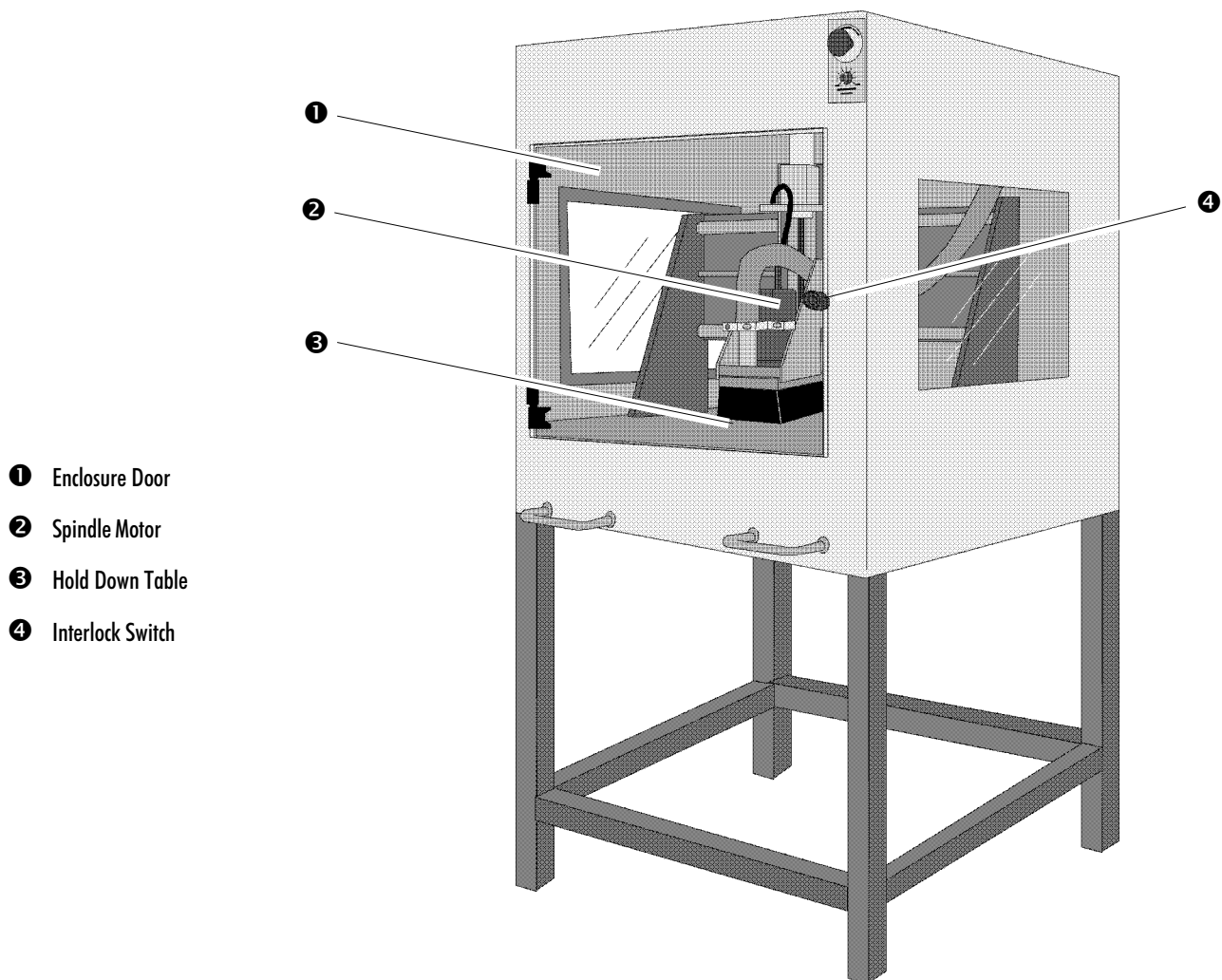
The Router Components

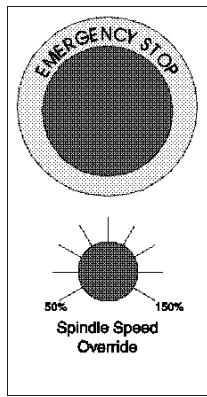
There are, of course, more components on the Benchtop CNC Router than those shown here, but to begin, you need only be concerned with the major components.

A 1 3/4 **hp Spindle Motor** supplies spindle speeds from 10,000 to 25,000 RPM. The **Enclosure** helps protect the operator from flying chips, and helps keep the area clean. The enclosure door **interlock switch** prevents the router from operating with the door open.

The X, Y and Z motion of the machine is performed by Drive Motors on each axis. There are also limit switches to prevent the machine from traveling beyond the limits on each axis.

The mechanical clamps on the **Hold Down Table** keep the stock firmly in place during a cutting operation.





The Front Panel

The Front Panel provides the operating controls shown here.

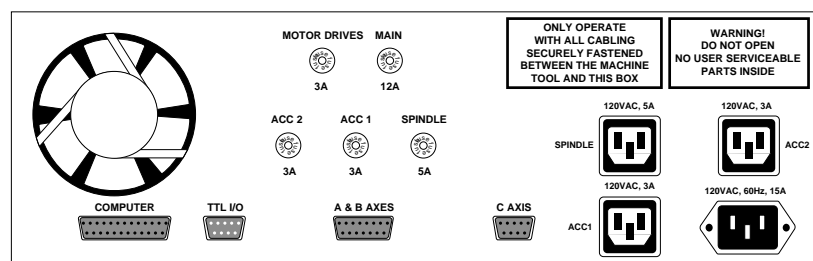
You can't miss the most important control on the router; the **Emergency Stop** button. When pressed, this bright red palm button immediately halts router operation. To resume operation, twist the button and pull it back out. It is important that this button be pressed before performing any manual operations,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

The **Spindle Speed Override** knob is used to establish the spindle speed. On the CNC Router, the minimum and maximum positions on the knob are equivalent to 50% to 150% of the programmed RPM.

The Rear Panel

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

On the rear panel are the power and interface connectors, and the fuses. The box has five fuses for main power, the spindle motor, the stepper motors, and the two accessory outlets (labeled ACC 1 and ACC 2). The values for these fuses are clearly printed next to the fuse receptacles. Do not use fuses with amperage ratings other than those shown.



The interface connectors are used for communication between the controller box, the router, and your computer. All connections between the router and controller box are made at the factory. The only connections you will make are from the router and controller box to the computer, and the two power cords. Refer to Section A for correct installation procedures.

Maintaining the Benchtop Router

Performing preventative maintenance on your Benchtop CNC Router ensures a longer, trouble-free life for the machine. We provide instructions for preventative maintenance in the following paragraphs.

Cleaning and Lubrication

Because dust and wood chips find their way into the base area that houses the Y axis ballscrews, it makes sense to perform cleaning and lubrication at the same time.

Ballscrews

The Benchtop Router uses pre-loaded ballscrews on the X and Y axes. The screws are lubricated at the factory with a special long-life, waterproof lubricant. The ballscrews should be lubricated every 6-8 weeks. The lubricant (part number M2006) is available from Light Machines Corporation.

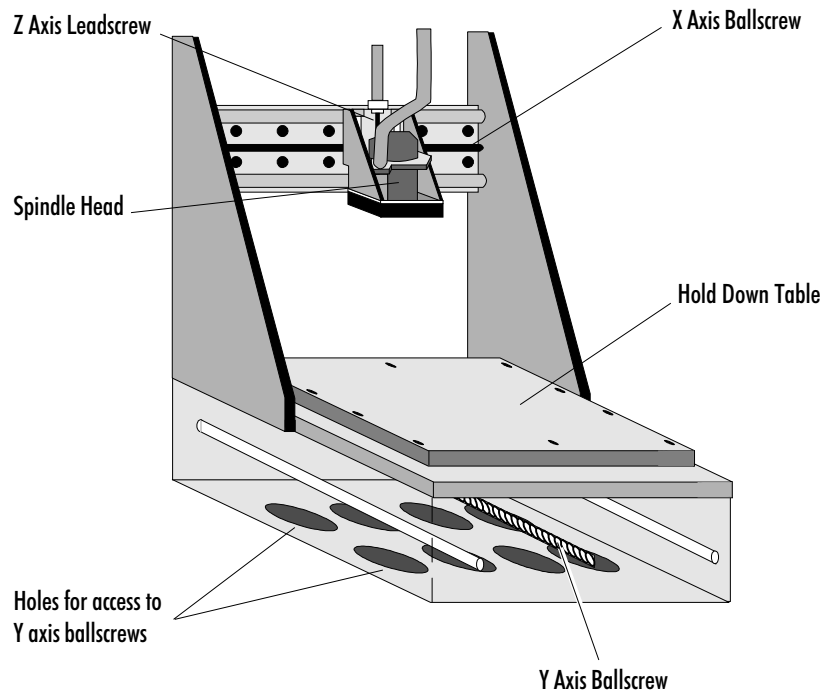
The Z axis has a leadscrew that needs no lubrication. Lubricating the Z axis leadscrew may cause damage.

The X axis ballscrew is easy to lubricate. Press the Emergency Stop button to remove power to the router. Simply reach inside the enclosure, wipe the dirt off, and apply grease to the exposed ballscrew. Because of the constant exposure to dirt and sawdust, it should be cleaned and lubricated once a month.

If you purchased the Optional Router Stand (ACC-5598), lubricating the Y axis ballscrew is easy. Reach underneath the router and access the ballscrew through the holes in the machine base. Be careful of the dust building up in the area beneath the Hold Down Table. The dust can cause the Y axis ballscrew to bind, stopping movement on the Y axis. Using a small vacuum, reach through the holes used to access the ballscrew and remove any sawdust in that area. It is a good idea to check for dust buildup every time you lubricate the ballscrew. After cleaning the base area, apply a thin film of lubricant along the length of each of the ballscrew.

Note:

The router is shown here with the enclosure removed for easy identification. It is not necessary to remove the enclosure to perform maintenance.



If your machine is on a bench where you cannot access the underside of the machine, you will need to follow this procedure:

1. Jog the router head to the rear of the machine and the top of the Z axis, just before the limits are tripped.
2. Press the Emergency Stop button and disconnect the power cord to the router.
3. Using the handles at the front and rear of the router, raise the machine up high enough to access the holes in the bottom of the base.
4. Set blocks **under all 4 corners** of the machine. Be sure that: a) the base is high enough to safely reach under the machine; b) the router is resting solidly on the blocks at **all 4 corners**.
5. Reach up through the holes in the base to vacuum out the dust, and then use a small brush to apply lubricant to the ballscrew.
6. After cleaning and lubrication, lower the machine back to the benchtop and reconnect power.

CAUTION

Lift the Benchtop Router very carefully. Be sure that the machine is resting completely on the blocks before reaching underneath. The Benchtop Router weighs approximately 300 lbs (135kgs).

The Dust Collection System

If you purchased the optional Dust Collection System (ACC-5734), you will need to empty the canister at the end of each day. You should also verify that the collector is turned on before running a program on the router. The power switch for the collection system is located on the top of the unit.

Some sawdust will escape the Dust Collection System. Be careful of the dust building up in the area beneath the Hold Down Table. The dust can cause the Y axis ballscrew to bind, stopping movement on the Y axis. Using a small vacuum, reach through the holes used to access the ballscrew and remove any sawdust in that area. It is a good idea to check for dust buildup every time you lubricate the ballscrew.

Maintaining the Controller Box

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

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

1. Turn the power off, and disconnect the power cords to the Controller Box and the Benchtop CNC Router.

WARNING

Changing fuses with the AC power connected can cause electric shock!

Always disconnect the main AC power before servicing the Controller Box or router.

2. The fuse holders are externally mounted on the rear panel of the Controller Box and are labelled to correspond with each AC outlet. Locate the correct fuse holder.
3. Remove the fuse by turning the fuse cap counterclockwise while pressing slightly inward. The fuse is removed with the cap.
4. Visually inspect the fuse. If the fuse element appears broken, it is blown. You can also use an ohm meter to check continuity across the fuse.
5. Replace the blown fuse with a standard slow-blow fuse of the appropriate rating. Three-amp (3A) fuses are used for the motor drives, ACC 1, and ACC 2. A five-amp (5A) fuse is used for the spindle, and a twelve-amp (12A) fuse is used for main AC power.
6. Replace the fuse holder cap by pushing inward and turning clockwise.
7. Reconnect power.

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.

- 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.
- Keep liquids (soda, coffee, cutting fluid, grease) away from the computer and peripherals.
- Keep oil, grease, wood chips and excess dust away from the computer, keyboard and floppy disks.
- Use grounded, three-prong outlets for the computer and peripherals. Take precautions against current overload. Use a line-surge suppression unit to help alleviate this problem.
- Don't block the vent holes in the computer or drives; they are required for air circulation.

Caring for Floppy Disks

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

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

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

Benchtop Router Options

The Benchtop Router has the following available options:

Dust Collection System (ACC-5734)

Router Support Stand (ACC-5598)

spectraCAM Milling for Windows (CAM-6721)

Getting to know the Control Program

Starting the Control Program

If You Need Help...

Exploring the Control Program Screen

Starting the Control Program

To start the Control Program:

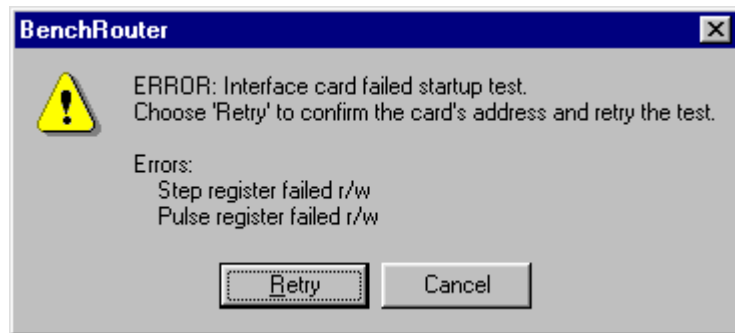
From **Windows 95** select Start >Programs >BenchRouter >BenchRouter.EXE. You can also create a Windows 95 shortcut by opening the BenchRouter directory and dragging BenchRouter.EXE onto your desktop.

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

You should have the controller box connected and powered up before starting the control software, unless you are going to be working in the Simulate Mode.

Starting the Control Program in Simulate Mode

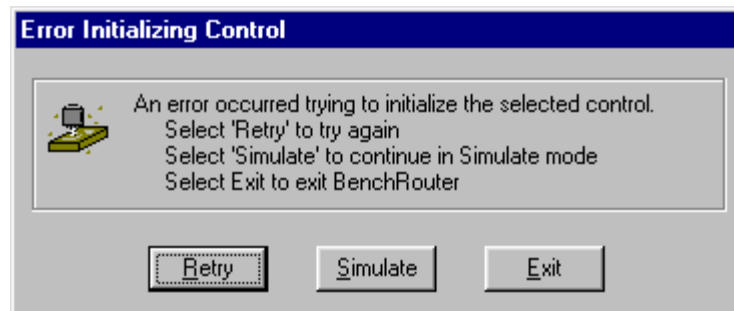
If this dialog box appears when you start the Control Program, the interface card that came with your router has not been properly installed.



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

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

When the next dialog box appears, select Simulate.



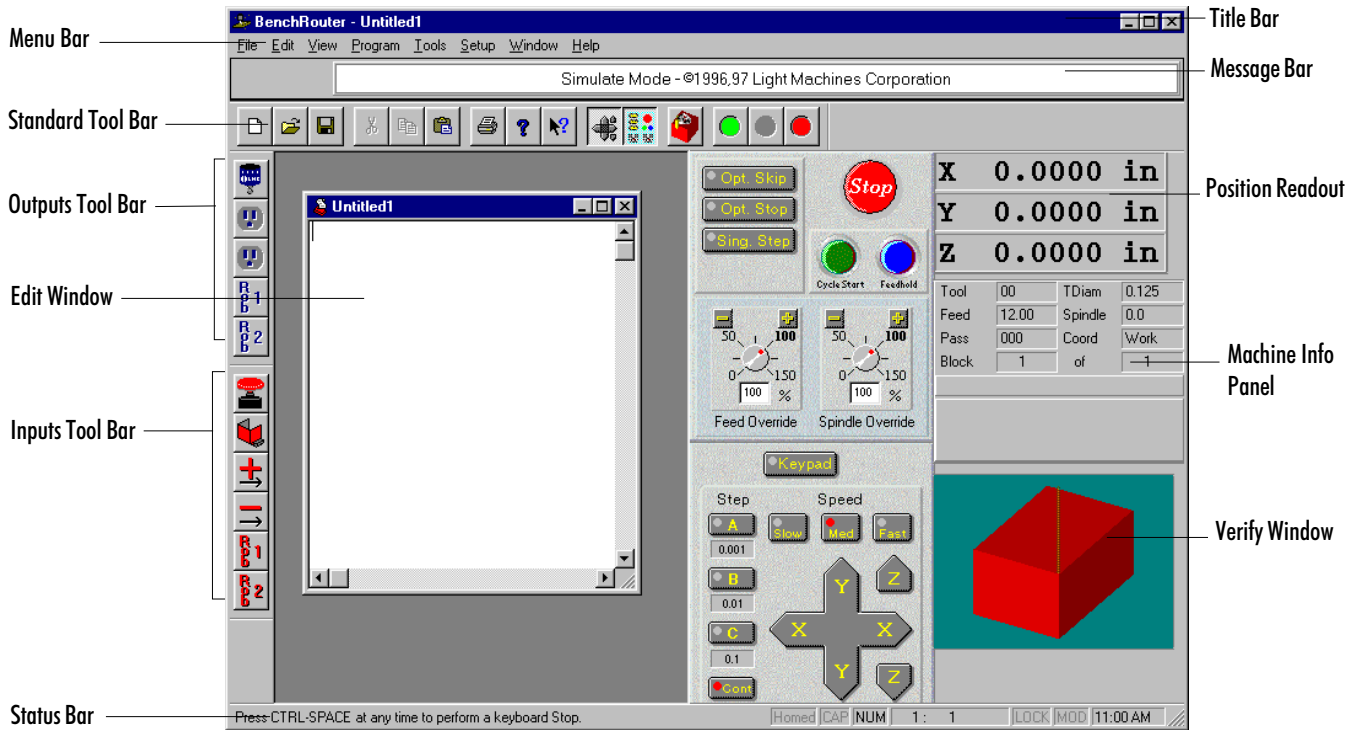
If You Need Help...

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

For more information on many of the functions and screens in the Control Program software, you can also see Reference Guide: Section E, Control Program Reference.

Exploring the Control Program Screen

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


















Menu Bar

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






Standard Tool Bar

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

Tool	Function
 New File	Begin a new NC part program file.
 Open File	Open an existing NC part program file.
 Save File	Save current NC part program file to disk or drive.
 Cut NC Code	Cut selected code from program and place on clipboard.
 Copy NC Code	Copy selected code to clipboard.
 Paste NC Code	Paste code from clipboard to cursor position in program.
 Print File	Send program to printer.
 Help	Access Help program.
 Context Help	Obtain help on selected object.
 Jog Control	Access Jog Control Panel.
 Operator Panel	Access Operator Control Panel.
 3-D Verification	Access 3-D tool path verification.
 Run (green)	Run the current NC part program.
 Pause (yellow)	Pause the currently running NC part program.
 Stop (red)	Halt the currently running NC part program.







Outputs Tool Bar

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

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

Inputs Tool Bar

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

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

IMPORTANT

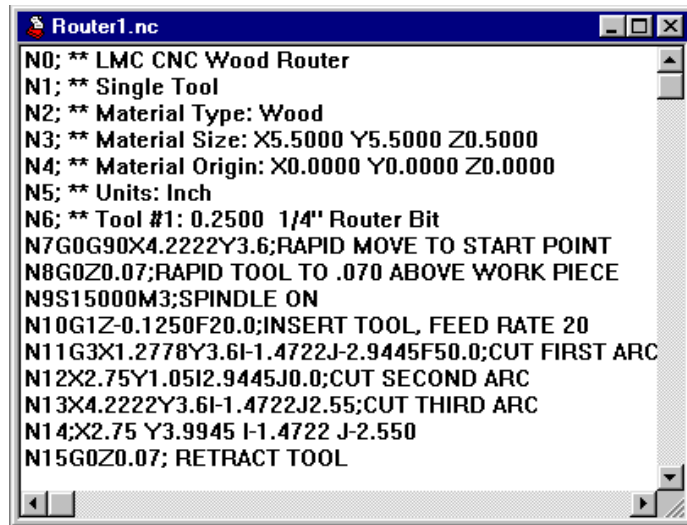
Always verify NC programs after editing to ensure that your changes will not cause a tool crash!

Here is what the edit window for the NC program Router1.NC looks like.

Edit Window

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

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



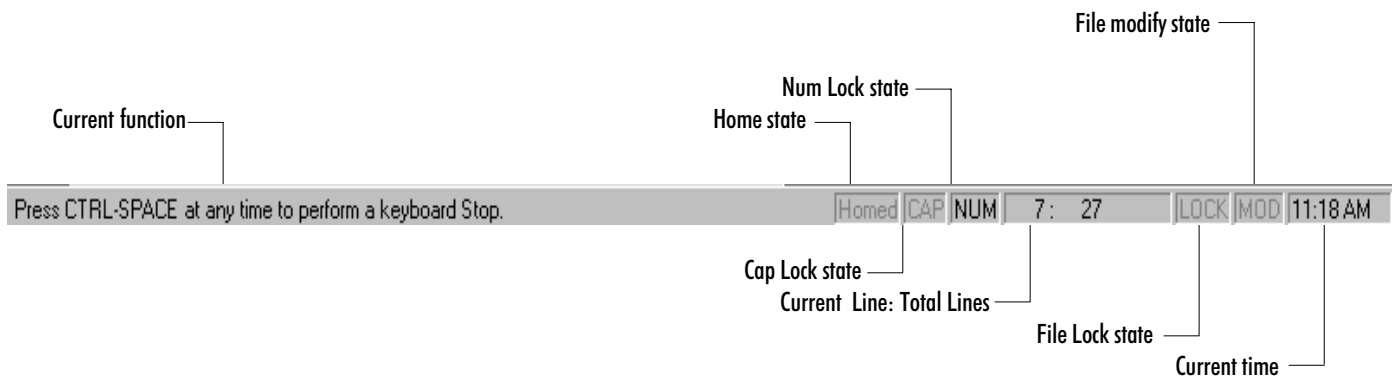
```
Router1.nc
N0; ** LMC CNC Wood Router
N1; ** Single Tool
N2; ** Material Type: Wood
N3; ** Material Size: X5.5000 Y5.5000 Z0.5000
N4; ** Material Origin: X0.0000 Y0.0000 Z0.0000
N5; ** Units: Inch
N6; ** Tool #1: 0.2500 1/4" Router Bit
N7G0G90X4.2222Y3.6;RAPID MOVE TO START POINT
N8G0Z0.07;RAPID TOOL TO .070 ABOVE WORK PIECE
N9S15000M3;SPINDLE ON
N10G1Z-0.1250F20.0;INSERT TOOL, FEED RATE 20
N11G3X1.2778Y3.6I-1.4722J-2.9445F50.0;CUT FIRST ARC
N12X2.75Y1.0512.9445J0.0;CUT SECOND ARC
N13X4.2222Y3.6I-1.4722J2.55;CUT THIRD ARC
N14;X2.75 Y3.9945 I-1.4722 J-2.550
N15G0Z0.07; RETRACT TOOL
```

Status Bar

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

- ◆ Whether or not the router is homed
- ◆ Whether or not the Caps Lock key is activated
- ◆ Whether or not the Num Lock key is activated
- ◆ The line on which the cursor is located, and total lines in the program
- ◆ Whether or not the current NC part program is locked
- ◆ Whether or not the current NC file has been modified
- ◆ The current time according to your computer

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



Tips:

- ◆ If you wish, you can move the Position Readout to another part of the screen. Hold the Control key down, then click and drag the Position Readout. Once you release the Control key and the mouse key, the Position Readout will stay in the new position. If you move it back to the docking area, it will automatically dock.

- ◆ If you want it to remain floating, press the right mouse button in the window, and uncheck "dockable." The window can only be resized when it is undockable.

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

- ◆ For more information on moving and resizing windows, see the Control Program Reference section of this manual.

Tips:

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

- ◆ If you want it to remain floating, press the right mouse button in the window, and uncheck "dockable." The window can only be resized when it is undockable.

- ◆ For more information on moving and resizing windows, see the Control Program Reference section of this manual.

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

Position Readout

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

X	0.0000 in
Y	0.0000 in
Z	0.1000 in

Machine Info Panel

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

Tool	10	TDiam	1.5
Feed	1.00	Spindle	1500.0
Pass	000	Coord	Work
Block	1	of	1

Tool	10	TDiam	1.5
Feed	1.00	Spindle	1500.0
Pass	000	Coord	Work
Block	1	of	1
Part Time 03:10			
4 N3G1 Z-.02F3;INSERT TOOL, FEED RA			
5 N4G2X1.583I1.000J.314;CUT FIRST A			
6 N5X1.000Y.314I.401J1.333;CUT SEC			

Elapsed time indicator

Current, previous and next block

Verify Window

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

Tool path verification can be performed in centerline view or solid view. Centerline view is based on the centerline of the tool. Solid view is a solid representation of the tool and workpiece.

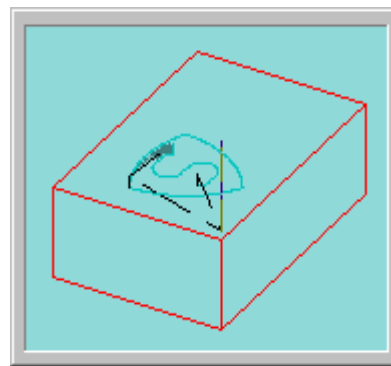
Tips:

- ◆ If you wish, you can move the Verify Window to another part of the screen. Hold the Control key down, then click and drag the Verify Window. Once you release the Control key and the mouse key, the Verify Window will stay in the new position. If you move it back to the docking area, it will automatically dock.

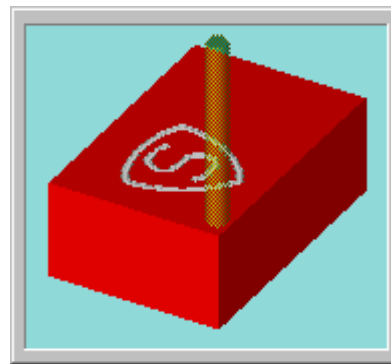
- ◆ If you want it to remain floating, press the right mouse button in the window, and uncheck "dockable." The window can only be resized when it is undockable.

- ◆ You can double-click on the Verify Window to bring up the Verify Setup dialog box.

- ◆ For more information on moving and resizing windows, see the Control Program Reference section of this manual.



Centerline View



Solid View

Tutorial: Cutting a Sample Part

Safely Running the Router

Running a Sample NC Program

Safely Running the Benchtop Router

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

Safety Rules

The following safety rules should be practiced by all operators of the Benchtop Router.

Remove Adjusting Keys and Wrenches

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

Do Not Force a Tool

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

Use the Right Tool

Select the type of cutting tool best suited to the 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 hold down table and the cutting tool in the spindle before turning on the spindle motor.

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.

Note:

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

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

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

Making Emergency Stops

Before you run the Benchtop Router 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 Router: by pressing the Emergency Stop button, by simultaneously pressing the Control and Space Bar keys on the computer keyboard, by activating one of the limit switches, or by activating the safety shield interlock switch.

Stopping with the Emergency Stop Button

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

In the event that an error occurs, such as a tool crashing into the workpiece, you can immediately kill power to the Router by pushing in the Emergency Stop button. Pushing in the Emergency Stop button terminates the part program. Wait until the Router has completely stopped moving before opening the safety shield.

When the tool crash or other error has been cleared and the Emergency Stop button is released (pulled back out), press Enter on the keyboard to close the error dialog box that appeared when the Emergency Stop button was pressed. Edit the part program to remove the cause of the tool crash before running the program again. You may need to reinitialize (reset the point of origin for all axes) the Router using the Set Position command from the Setup Menu, and you may need to home the machine again.

The Emergency Stop button should be your first target when a critical error occurs. You do not need to use it in situations where safety or reaction time is not an issue. For example, if you start to run a program but notice that the tool is moving towards the wrong corner of the workpiece, use a software stop to halt the program and reset the origin of the workpiece.

Stopping with the Computer Keyboard

The execution of the part program can be interrupted by pressing keys on the computer keyboard. To stop the part program with the keyboard, press the Control key and Space Bar simultaneously. The cutting stops immediately and the cutting tool remains in position.

To restart the program from a keyboard-generated stop, select the Run/Continue command from the Program Menu. In the Start At Line box, enter the number of the last line executed, then click on the Run Program button. (When you stopped the program, the last line executed is displayed on the Machine Info Panel, and the cursor is placed on that line in the file.)

Stopping with a Limit Switch

The Benchtop Router is equipped with limit switches to sense the end of travel on each axis. 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 Jog Control keypad (selected from the View Menu).

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 must jog away in a negative direction.

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 Jog Control from the Standard Tool Bar.
2. Press the appropriate jog key on the jog keypad, to move the cross slide away from the triggered limit switch.
3. Check your initial machine set up to make sure it was performed correctly.

Note:

Never open the Enclosure Door while a program is running. Always push the Emergency Stop button in before making any adjustment to the Router or workpiece.

Stopping with the Safety Interlock Switch

Opening the enclosure door activates a safety interlock switch which interrupts the NC program and stops machine motion. The cutting stops immediately and the cutting tool remains in position. To restart the program, select Run/Continue from the Program Menu to begin the operation again. However, this is not recommended as the proper method of stopping the machine in an emergency situation. Use the Emergency Stop button.

Running a Sample NC Program

When you installed the Control Program, an NC part program file named ROUTER1.NC, was copied into the BenchRouter directory along with the other files. This program is meant to cut a 5.5" x 5.5" piece of wood. You will be using this file to create your first workpiece on the Router.

WARNING

Do not attempt to operate the Benchtop Router without reviewing all of the safety precautions set forth in Section J.

Open ROUTER1.NC



1. Select the Open command from the File Menu, or click on the Open button on the Standard Tool Bar. The Open dialog box appears.
2. Double-click on the filename , or select the filename then select the Open button. The edit window for Router1.NC appears.

```
Router1.nc
N0; ** LMC CNC Wood Router
N1; ** Single Tool
N2; ** Material Type: Wood
N3; ** Material Size: X5.5000 Y5.5000 Z0.5000
N4; ** Material Origin: X0.0000 Y0.0000 Z0.0000
N5; ** Units: Inch
N6; ** Tool #1: 0.2500 1/4" Router Bit
N7G0G90X4.2222Y3.6;RAPID MOVE TO START POINT
N8G0Z0.07;RAPID TOOL TO .070 ABOVE WORK PIECE
N9S15000M3;SPINDLE ON
N10G1Z-0.1250F20.0;INSERT TOOL, FEED RATE 20
N11G3X1.2778Y3.6I-1.4722J-2.9445F50.0;CUT FIRST ARC
N12X2.75Y1.05I2.9445J0.0;CUT SECOND ARC
N13X4.2222Y3.6I-1.4722J2.55;CUT THIRD ARC
N14;X2.75 Y3.9945 I-1.4722 J-2.550
N15G0Z0.07; RETRACT TOOL
```

Adjust the Verify Settings

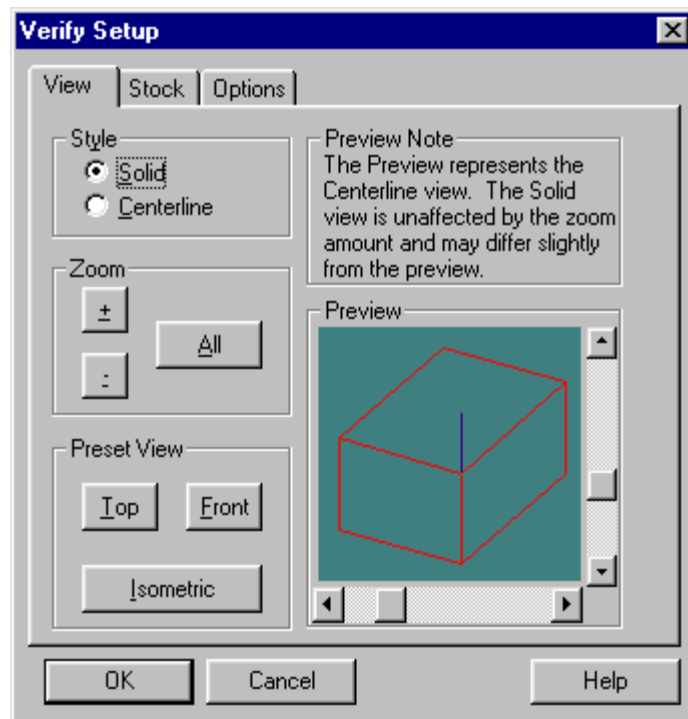
After loading the NC program, you need to adjust the Verify Settings for the stock you are about to cut. To view the Verify Setup dialog box, double click on the Verify window. You may also select the Program Menu and choose Verify from the pull down menu, or Select Verify from the Standard Toolbar and choose Verify Settings. The Verify Setup dialog box appears.

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

The View panel allows you to:

- ◆ Alter the Style (Solid or Centerline)
- ◆ Zoom in, zoom out or fill the window by selecting All
- ◆ Look at the workpiece in two or three dimensions

You can also alter the view of the part by adjusting the sliders on the Preview box.



Adjust the View

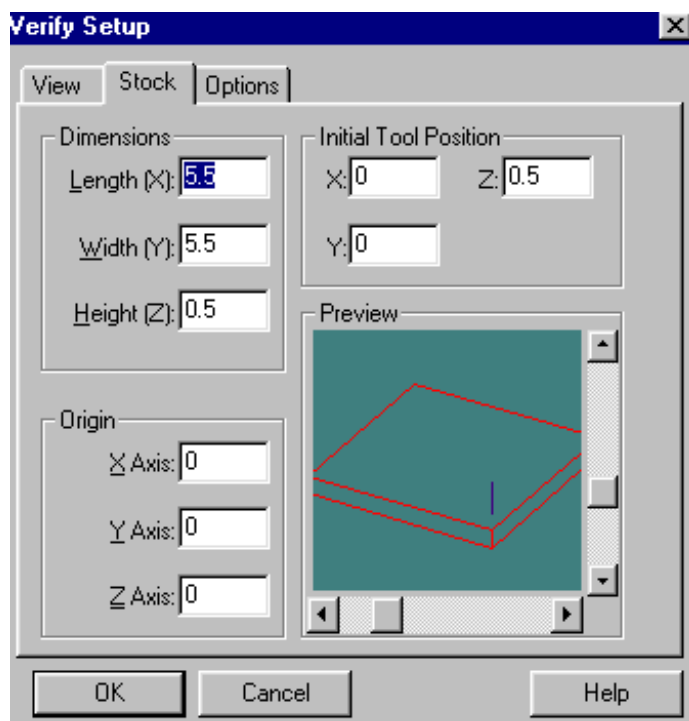
1. Select View.
2. Select either Solid or Centerline (for a solid representation or a centerline representation of the tool and workpiece).
3. Select Isometric for a three dimensional view of the part.

Adjust the Stock

1. Select the Stock tab.
2. Enter the stock Dimensions for the ROUTER1.NC program. The stock dimensions are X=5.5", Y=5.5" and Z=0.5".
3. Set the Initial Tool Position to X=0, Y=0 and Z=0.5
4. Set the point of Origin to zero on all three axes.
5. Select OK. The dialog box will close, and you should notice a change in the shape of the workpiece in the Verify Window.

The Stock panel allows you to:

- ◆ Enter the dimensions of the workpiece
- ◆ Set the initial position of the tool
- ◆ Set the point of origin for the workpiece



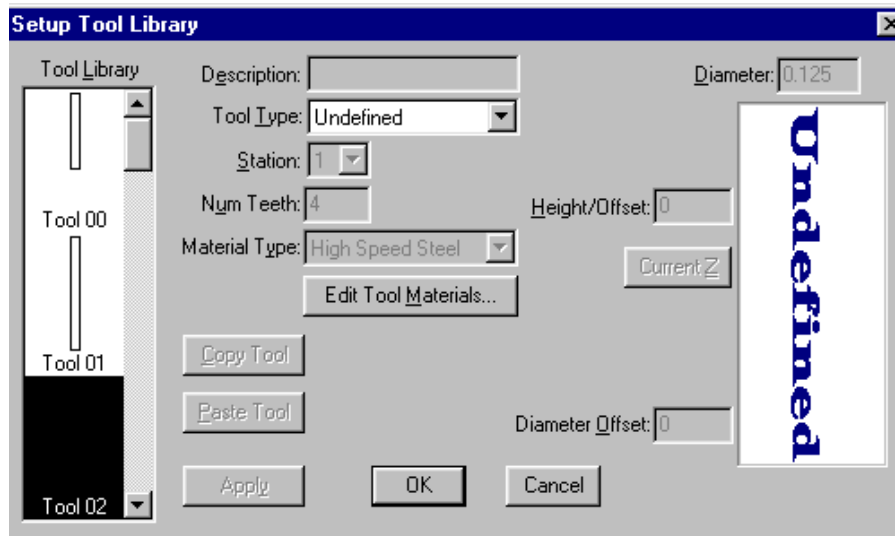
Define the Tool

You will be using an 1/4" router bit for this program. You will be using the parameters for this particular bit for the tool path verification as well. To define the tool parameters, first add the tool to the tool library, then select the tool for verification.

Add the Tool to the Library

1. Select Setup Library from the Tools Menu. The Setup Tools dialog box appears. There is no 1/4" Router bit defined, so we will need to create one.

To define a new tool, you first have to select an undefined tool, as we have here, then select a Tool Type.



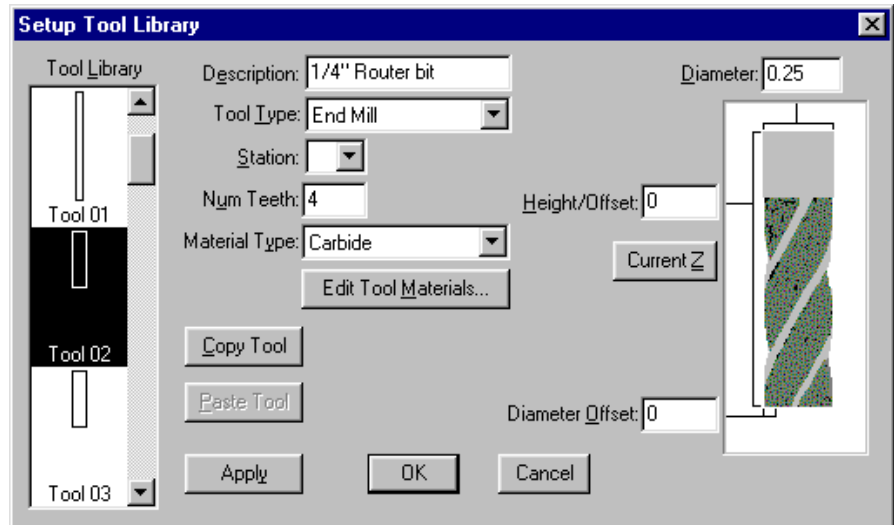
2. In the Tool Library scroll box, scroll down to tool 2, which is undefined at this point. Highlight Tool 2 by clicking on it with your mouse.
3. In the Tool Type pull-down list, select End Mill.
4. Enter "Router Bit" in the Description box.
5. In the Material Type pull-down list, select Carbide.
6. Enter 0.25 in the Diameter box.

Note: You can also enter fractions and let the program calculate the decimal equivalent.

- Click on the Apply button. You have just defined a new tool in the library. From now on, whenever you need an 1/4" Carbide Router Bit, it will be there.
- Click on OK to exit the Tool Library.

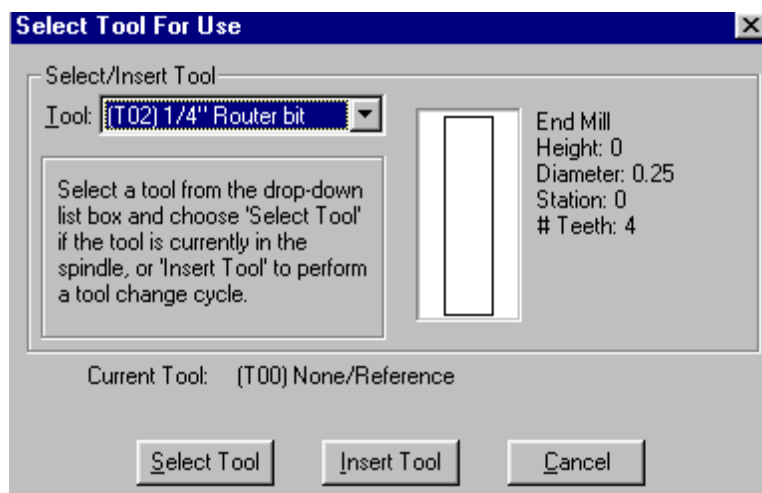
Here is the newly defined tool, an 1/4" Carbide Router Bit that has four teeth.

These are the same tool parameters you will use when actually run the Router1.NC program.



Select the Tool for Verification

- Select Tool from the Tools Menu. The Select Tool for Use dialog box appears.
- Select Tool 2 from the pull-down list.
- Click on the Select Tool button.



Installing a Tool

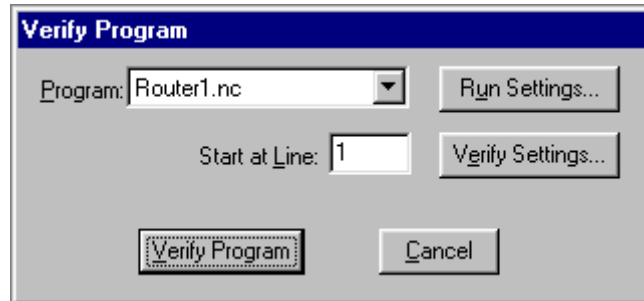
There are two wrenches included in the Accessory Kit shipped with the router. The collet wrenches are used for installation and removal of tools. To install a tool into the spindle:

1. Using the Jog Control Panel, jog the spindle head high enough away from the table to allow you to access the collet.
2. Press the Emergency Stop button, and open the enclosure door.
3. Remove the brush from the spindle. There is a single thumbscrew on the front of the brush. Loosen this screw and swing the brush down. Pull the brush forward slightly and then down again to remove it.
4. Using the collet wrenches, loosen the collet nut.
5. Install the tool into the collet, and hand tighten the collet nut.
6. Using the wrenches, tighten the collet nut. **Do not overtighten!** Overtightening the collet nut will cause damage to the nut.
7. Replace the brush around the spindle head.
8. Close the door and release the Emergency Stop button.
9. From the Tools menu, choose Select Tool.
10. Choose the tool number and description that correspond to the tool you installed in the Router. If the tool you installed is not already defined in the list, you must define the tool (see Define the Tool).
11. Choose Select Tool to close the window.

Verify ROUTER1.NC

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

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



2. Click on the Verify Program button, then watch the Verify Window. You will see ROUTER1.NC executed on the graphic workpiece.



Dry Run the NC Program

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

Begin with the Emergency Stop button pressed in, and the spindle speed turned all the way down. The work holding devices should be mounted to the table and the tool should be mounted in the spindle.

1. Mount the workpiece on the table.
2. Close the enclosure door and pull out the Emergency Stop button.
3. Select Jog Control from the View Menu (or the Standard Toolbar). The Jog Keypad appears.

To jog the tool:

Click on the appropriate axis buttons on the Jog Keypad.

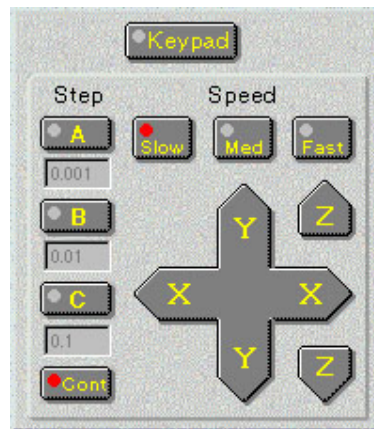
The tool moves at the speed and distance selected using the Speed and Step buttons.

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

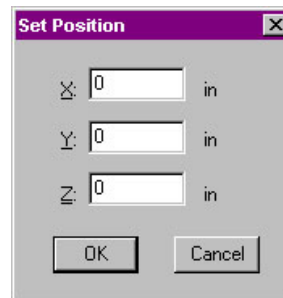
To move the tool in a continuous motion, select Cont. The tool will continue to move as long as the axis button is depressed.

Note:

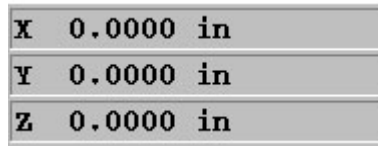
Double-clicking on the Jog Keypad opens the Jog Parameters dialog box.



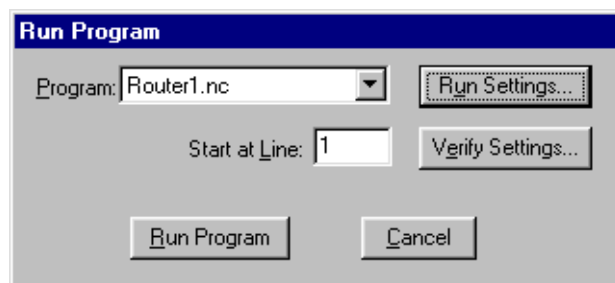
4. Use the Jog Keypad to jog the tool to top front left corner of the workpiece.
5. Select Set Position from the Setup Menu. The Set Position dialog box appears.
6. Enter zero in the X, Y and Z boxes.



7. Click on OK. The values in the Position Readout all change to zero.
8. Jog the tool up and away from the workpiece, press the Emergency Stop button, open the Enclosure Door and remove the workpiece.



9. Return the Enclosure Door to the closed position and pull out the Emergency Stop button.
10. Put on a pair of safety glasses and complete the Safety Checklist (refer to Reference Guide: Section J).
11. Select Programs, and choose Run/Continue.



12. The Run Program dialog box appears. Click on the Run Program button.
13. As the part program runs, observe the tool motion in relation to the table (and eventually the workpiece). Look for signs of a possible tool crash and be prepared to press the Emergency Stop switch on the Router. Edit the program as required. When you are satisfied that the tool motions are correct, you can mount the workpiece.

Mount the Workpiece

1. Using the Jog Keypad, jog the spindle up and out of the way.
2. Before mounting the workpiece, push the Emergency Stop button in.
3. Mount the workpiece on the hold down table using either the clamps provided, or a vise.
4. Pull the Emergency Stop button out.
5. Select the Operator Panel and check that the spindle speed is set to 100%.

The workpiece is now correctly mounted.

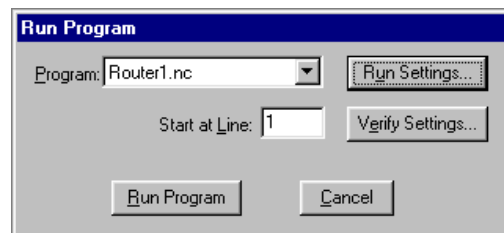
Run the Program

Before executing the ROUTER1.NC program, check that all safety precautions have been taken. The Router enclosure door should be closed, and you should be wearing safety glasses.

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

To run the program:

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



To change the Run Settings, click on an item checkbox. The Run Settings include:

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

Optional Skip When off does not recognize the optional skip code (/).

Optional Stop When on, will pause the NC program at any M01 code.

Enable Subprograms Must be on if the program uses subprograms.

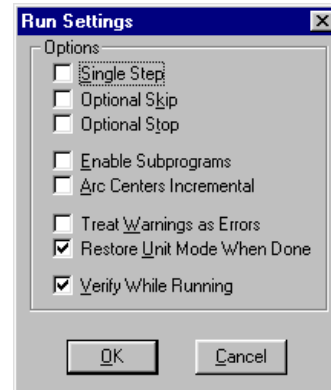
Arc Centers Incremental When off, does not recognize the % code indicating that the center of an arc is an incremental value relative to the start of the arc.

Treat Warnings as Errors When on, will halt the NC program at a warning as though it were an error.

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

Verify While Running When on, tool path verification occurs while the NC program is running on the Router.

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



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

Control Program Reference

The Benchtop Router Interface

Using the Message Bar

Using Windows

Using Toolbars

Using Panels

Using the Menu Bar

Selecting Commands

Positioning Screen Components

The Setup Program

The Offset Table

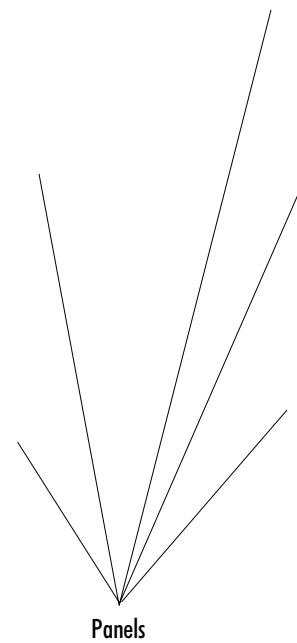
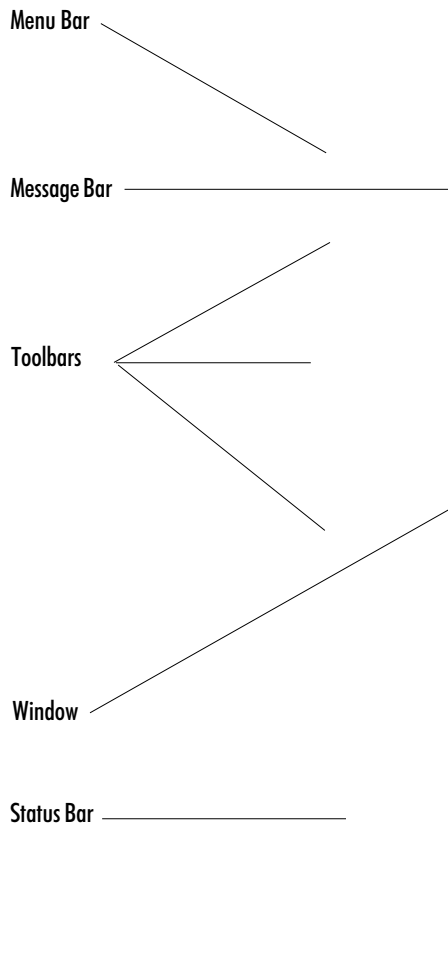
Working in Simulation Mode

The Benchtop Router Interface

The Benchtop Router interface (the screen) is composed of several components that allow you to create NC part programs and interact with the router.

The Benchtop Router interface components include:

- u The Message Bar
- u Windows
- u Toolbars
- u Panels
- u The Status Bar
- u The Menu Bar



Using the Message Bar

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

Using Windows

Windows are used to display information. Some windows can be docked or they can be floating windows. Windows are activated or hidden using the commands under the View Menu.

The following windows are available:

- u Program Edit Windows
- u Verify Window
- u Machine Info Window
- u Position Window

Program Edit Windows

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

Program edit windows appear in the Edit Area (the large central area) of the screen. The Edit Area is fixed in position; you cannot close it or move it.

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

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

The Edit Area can contain multiple program windows. The Windows Menu has several commands for managing program windows. You may select a particular window from a list of all current program windows, identified by file names.

Tips:

- If you wish, you can move the Verify Window to another part of the screen. Hold the Ctrl key down, then click and drag the Verify Window. If you hold the Ctrl key, the window will not dock, so you can place it anywhere on the screen. If you release the Ctrl key, the window will try to dock to the main window, depending on where you drop the window. To prevent the window from ever docking, right-click on the window and uncheck the Dockable command.
- Pressing the right mouse key while in the Verify window displays a pop-up menu with Verify-related commands.
- When the Verify Window is floating (not docked) you can resize it just like any other window.
- You can double-click on the Verify Window to bring up the Verify Setup dialog box.

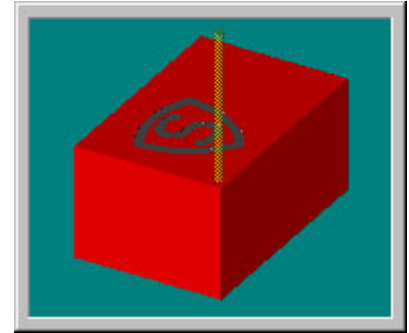
Tips:

- If you wish, you can move the Machine Info Window to another part of the screen. Hold the Ctrl key down, then click and drag the Machine Info Window. Once you release the Ctrl key, the Machine Info Window becomes a floating window which you can place anywhere on the screen.
- When the Machine Info Window is floating (not docked) you can resize it just like any other window.

Verify Window

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

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



Tool path verification can be performed in centerline view or solid view. Centerline view is based on the centerline of the tool.

Solid view is a solid representation of the tool and workpiece.

Machine Info Window

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

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

- u Current tool number and diameter,
- u Feed rate and spindle speed,
- u Number of passes made for the current program
- u Coordinate system being used
- u Current block number, and the number of blocks in the program.

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

Tool	11	TDiam	0.125
Feed	10.00	Spindle	1500.0
Pass	001	Coord	Work
Block	5	of	20

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

Tool	11	TDiam	0.125
Feed	*RAPID*	Spindle	1500.0
Pass	001	Coord	Work
Block	2	of	20
Part Time 00:13			
1 N0G0Z.1;RAPID TOOL AWAY FROM STOCK			
2 N1G90X.417Y1.333;MOVE TO START POINT			
3 N2S1500M3;SPINDLE ON			

When a program is being verified, the Machine Info Window displays the current line of code, plus the previous and next lines. The elapsed time is not indicated.

Tool	11	TDiam	0.125
Feed	10.00	Spindle	1500.0
Pass	001	Coord	Work
Block	20	of	20
19 N18X0Y0;MOVE TOOL BACK TO START			
20 N19M2;END OF PROGRAM			

The Position Window

Note:

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

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

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

X	0.0000 in
Y	0.0000 in
Z	0.1000 in

Using Tool Bars

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

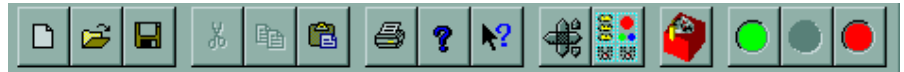
The toolbars include:

Standard Toolbar

Outputs Toolbar

Inputs Toolbar

Standard Toolbar



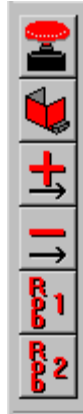
The Standard Toolbar provides quick-access buttons for the following commands:

Command:	Used to:
New File	Create a new program edit window
Open File	Open an existing NC program file
Save File	Save an NC program file
Cut	Cut text from a program
Copy	Copy text from a program
Paste	Paste text into a program
Print	Print an NC program
Help	Access Help
Context Help	This button can help you instantly find information on the objects you see on the screen. For instance, click on the Context Help button, then click on a menu item, toolbar button, window or other screen element. The Help file for that particular item appears. Note that toolbars, windows, and panels must be docked for Context Help to work.
Jog Control Panel	Show or hide the Jog Control Panel

Note:

Context-sensitive means the results of your action are dependent on the item you click, or on the operation you are currently performing.

Operator Panel	Show or hide the Operator Panel
Verify	Clicking on this button performs the same operation as selecting the Verify command under the Program Menu.
Run	Run the current NC part program
Pause	Pause the currently running NC part program
Stop	Immediately halt the currently running NC program



Inputs Toolbar

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

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

The inputs on the Inputs Toolbar include:

- u The **Emergency Stop** condition. This input is in the "on" condition (depressed) if the Emergency Stop button on the Router is pushed in.
- u The **Enclosure Door** condition. This input is in the "on" condition (depressed) if the Enclosure Door on the Router is open.
- u The **Positive Limit** condition. This input is in the "on" condition (depressed) if one of the positive axis limits has been hit.
- u The **Negative Limit** condition. This input is in the "on" condition (depressed) if one of the negative axis limits has been hit.
- u The **Robot Input 1** condition. This input is in the "high" condition (depressed) if the secondary robotic input on the TTL I/O connector on the Controller Box is currently in a high state. Refer to Section K of this manual for more information on robotic interfacing.
- u The **Robot Input 2** condition. This input is in the "high" condition (depressed) if the robotic input on the TTL I/O connector on the Controller Box is currently in a high state. Refer to Section K of this manual for more information on robotic interfacing.



Outputs Toolbar

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

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

The output controls on the Outputs Toolbar include:

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

Panels

Use the Jog Control Panel and the Operator Panel to control router operation.

Tips:

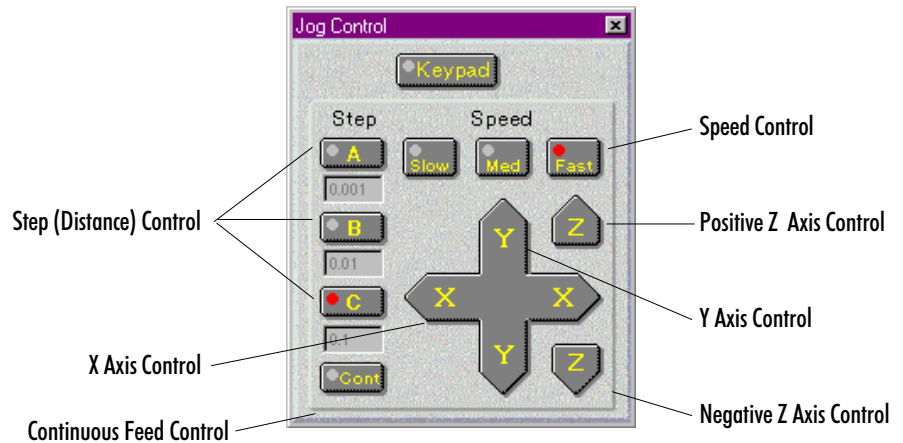
- If you wish, you can move the Jog Control Panel to another part of the screen. Hold the Ctrl key down, then click and drag the Jog Control Panel. Once you release the Ctrl key, the Jog Control Panel becomes a floating window which you can place anywhere on the screen.
- Double-clicking on the Jog Control Panel opens the Jog Settings dialog box.

Jog Control Panel

The Jog Control Panel is accessed by selecting the Jog Control command under the View Menu, or by clicking on the Jog Control button on the Standard Toolbar. The Jog Control Panel allows you to manually move (jog) the spindle on the router. You may also use the numeric keypad on the computer keyboard to jog the machine.

Each axis on the machine is represented by buttons. The X and Y axes are represented by the crosspad. The crosspad follows the Cartesian coordinate system standard; -X to +X is left to right, while -Y to +Y is bottom to top. The Z axis is represented by two buttons, one for positive motion and one for negative. Pressing any of the axis buttons moves the tool in the indicated direction, as long as the system is not in Simulate Mode.

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



To jog the router head:

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

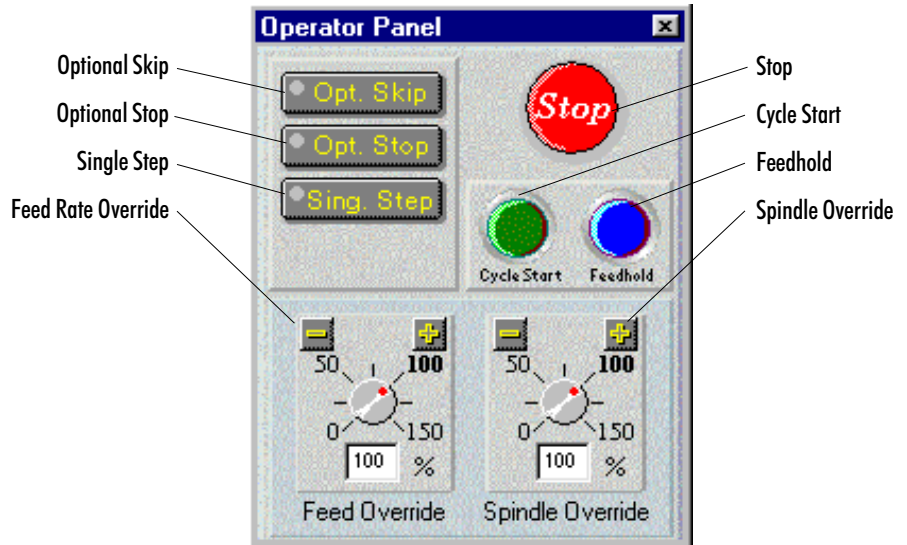
1. Define the Speeds and Steps (distances) for jogging by selecting the **Jog Settings** command under the Setup Menu.
2. Click on the **Axis** button on the Jog Keypad to move the tool in the desired direction. The tool moves at the speed and distance indicated by the Speed and Step buttons.

Tip:

If you wish, you can move the Operator Panel to another part of the screen. Hold the Ctrl key down, then click and drag the Operator Panel. Once you release the Ctrl key, the Operator Panel becomes a floating window which you can place anywhere on the screen.

Operator Panel

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



The Operator Panel controls include:

- u **Optional Skip**, when on, allows you to **execute or ignore** any optional skips (/ codes) you have embedded in the NC program.
- u **Optional Stop**, when on, allows you to **execute or ignore** any optional stops (M01 codes) you have embedded in the NC program.
- u **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.
- u **Stop**, when pressed, immediately halts the currently running NC program. This button works the same as the Ctrl + Space Bar combination.
- u **Cycle Start** begins running the current NC program from the beginning or from a paused condition.
- u **Feedhold** pauses the currently running NC program. To continue running the program from a Feedhold, press the Feedhold button again or press the Cycle Start button.
- u **Feed Rate Override** overrides the programmed feed rate.
- u **Spindle Speed Override** overrides the programmed spindle speed.

Status Bar

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

Press CTRL-SPACE at any time to perform a keyboard Stop.

Homed CAP NUM 1 : 1 LOCK MOD 1:41 PM

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

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

Menu Bar

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

The available menus are:

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

File Menu

The File Menu provides typical file management commands and the Exit command.

Command:	Used to:
New	Create a new program window.
Open	Open an existing file.
Close	Close an open program window.
Save	Save a program.
Save As	Save a program under a different filename or location.
Print	Print an open NC program.
Print Setup	Set up your printer for printing.
Recently opened files	Open one of the eight most recently used files.
Exit	Exit the Control Program.

New Command

Use the New command under the File Menu to create a new program edit window.

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

To create a new program window, select **New** from the File Menu, or press **Ctrl+N**.

A new program window is created. The filename on the Title Bar is “Untitled,” indicating that this is a new program. The program will remain untitled until you save it. You should save your new programs before they are run or verified.

Open Command

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

To open an existing NC program:

1. Select **Open** from the File Menu, or press **Ctrl+O**. The File Open dialog box appears.
2. In the dialog box, locate and highlight the desired NC file.
3. Click **OK** or press **Enter**. The selected NC program file is opened. The Title Bar displays the name of the file.

To select a file that is already open:

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

Close Command

You can close a program window that is not in use at any time. Unless you have already done so, you will be prompted to save any changes made to the program file.

To close a program window:

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

Note:

The title bar displays the name of the NC program file. It is located across the top of each Program Edit Window.

Tip:

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

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

Save Command

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

If the current NC program was previously saved, selecting Save saves the changes to the same file. If the current program is new (and still has the name “Untitled”), selecting Save brings up the Save As dialog box, in which you name, choose a location for, and save the new program.

To save a program, select **Save** from the File menu, or press **Ctrl+S**. The current program is saved to a file. If this is a new program, the Save File As dialog box appears. Choose a name and location for the new file.

Save As ... Command

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

Save As Dialog Box

The Save As dialog box is the same in the Control Program as in other Windows applications.

To use the Save As dialog box:

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

Note:

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

Print Command

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

To print the program:

1. There are several ways to print an NC program:
 - u Click the **Print** button on the toolbar.
 - u Select the **Print** command from the File Menu.
 - u Press **Ctrl+P**.
2. The Print dialog box appears. Choose the desired print options in the dialog box. Clicking **Setup** opens the Print Setup dialog box.
3. Click **OK** to print, or click **Cancel** to exit the Print dialog box without printing the NC program.

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

Print Setup Command

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

To choose print settings:

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

Opening a Recent Program

You can use the numeric (1, 2, 3, ... 8) commands under the File menu to open any of the eight most recently opened files.

The names and paths of the eight most recent files appear as file 1 (the file last opened) through file 8 (the eighth most recent).

To open one of the eight most recently opened programs, select 1, 2, 3, 4, 5, 6, 7 or 8 from the File menu. The recent program you selected is opened. The Title Bar displays the name of the program file.

Exit Command

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

To exit the Control Program:

Select the **Exit** command from the File Menu.

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

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

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

Edit Menu

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

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

Undo Command

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

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

- u If your last editing action was to delete selected text, the text is restored.
- u If your last editing action was to delete a character, the character is restored.
- u If your last editing action was to paste text, the text is removed.
- u If your last editing action was to type a character, the character is removed.
- u If Undo is grayed-out in the Edit menu, no changes can be undone.

Redo Command

The Redo command redoes the edit command performed before the undo command. For example, if you deleted a part of the text, and then decided to undo that deletion, the Redo command performs the original delete again.

Cut Command

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

To cut text to the Clipboard:

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

Copy Command

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

To copy text to the Clipboard:

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

Paste Command

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

To paste text from the Clipboard:

1. Place the cursor at the point in the NC program where you wish to insert text that has been previously cut or copied to the Windows clipboard.
2. Select **Paste** from the Edit menu, or press **Ctrl+V**. The contents of the clipboard are inserted into the program. If this menu command is grayed-out, there is no text on the clipboard to paste.

Selecting Text

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

To select text using the editing keys:

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

To select text using the mouse:

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

Clear Command

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

To delete text using the Clear command:

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

Delete Line Command

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

To delete a program line using the Delete Line command:

1. Place the cursor anywhere on the line of code you wish to delete.
2. Select the **Delete Line** command from the Edit Menu, or press **F2**. The line of code is deleted.

Find Command

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

To use the Find command:

1. Select **Find** from the Edit Menu, or press **Ctrl+F**.
2. Enter the character sequence you are looking for in the **Find:** box.
3. Check off the **Match Case** box to restrict the search to finding only those text strings that match the case (upper or lower) of the text that you entered.
4. Select **Up** or **Down** from the Direction box to search through the text before or after the cursor position, respectively.
5. Click **Find Next** or press **Alt+S** to begin the search. Click **Cancel** or press **ESC** to exit the Find dialog box without performing the search.

To find the same character string again, use the **Find Next** button.

Replace Command

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

To use the Replace command:

1. Select **Replace** from the **Edit** menu.
2. Enter the existing character string in the **Find What:** box.
3. Enter the new character string in the **Replace With:** box.
4. If you select **Match whole word only** the program will only find and replace text that matches your entry. When you select **Match case**, it will find and replace those text strings that match the case (upper or lower) of the text you entered.
5. Select one of the buttons depending on how you wish to replace text.
 - u **Find Next** will find the next occurrence of the text.
 - u **Replace** will replace the selected text with the new text.
 - u **Replace All** will replace all occurrences of the text with the new text.
 - u **Close** will close the dialog box.

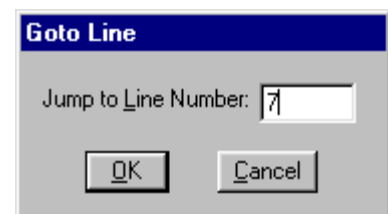
Goto Line Command

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

To use the Goto Line command:

1. Select **Goto Line** from the Edit Menu, or press **Ctrl+G**. The Goto Line dialog box appears.
2. Enter a line number in the Jump To Line box and click on OK. The cursor moves to the specified line.

If the line number entered is larger than the number of lines in the program, the cursor moves to the end of the program.

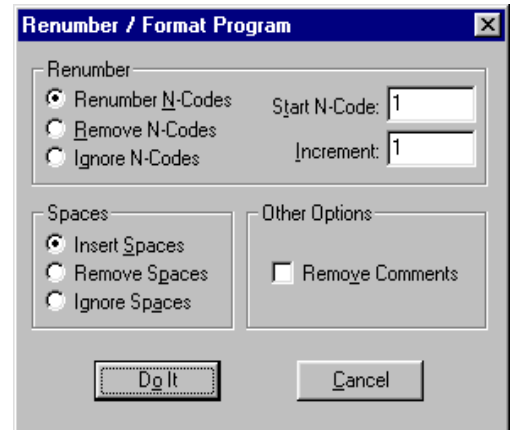


Renumber Command

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

The Renumber command can be used to:

- u Insert N codes in a program that doesn't have any.
- u Remove N codes from a program.
- u Renumber N codes in a program.
- u Insert, remove or ignore spaces between NC commands.
- u Remove comments from the program.



Insert N Codes

Note:

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

To insert or renumber the N codes in your program:

1. Select **Renumber** from the Edit Menu. The Renumber/Format Program dialog box appears.
2. Select **Renumber N Codes** or press **Alt+N**.
3. Click on the **Start N Code** box (or press **Alt+T**), then enter the number of the first N code. The default starting block number is N1.
4. Click on the **Increment** box (or press **Alt+I**), then enter the increment you wish to use.

For instance, if you wish to have each N code numbered in increments of 5, enter 5 in the Start N Code box and enter 5 in the Increment box. The N code sequence will then be: N5, N10, N15, N20...and so on.

This option is useful if you are renumbering a portion of the program to be inserted into another program. Using increments greater than 1 allows you to insert additional numbered lines without having to renumber the whole program.

5. Select **Do It**, or press **Alt+O**, to execute the Renumbering options you selected. Select **Cancel**, press **Alt+C**, or press **Esc**, to exit the Renumber/Format Program dialog box without altering the NC program.

Renumbering and Subprograms

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

```
For instance,  N41X  
               N42X...  
               O25G...  
               N44...
```

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

Insert or Remove Spaces

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

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

Note:

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

Note:

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

IMPORTANT

None of the Renumber actions can be undone!

Remove Comments

To remove comments from your program:

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

Remove N Codes

To remove the N codes from your program:

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

Lock Command

Use the Lock command under the Edit Menu to prevent or allow changes to your NC programs. If a check mark appears next to this command, the current NC program is locked.

When an NC program is unlocked, it can be modified by the commands on the Edit Menu. When an NC program is locked, the program cannot be changed by any commands. By default, when you open a file it is automatically locked to prevent accidental changes. You can change this in the Preferences dialog box.

If you have multiple program edit windows open, each is individually locked or unlocked. The state of the currently selected NC program can be easily checked by looking at the Lock Indicator on the Status Bar, or by the color of the edit window background. Locked files have a gray background.

To use the Lock command to lock or unlock your NC program, select **Lock** from the Edit Menu, press **Ctrl+L**, or double-click the **Lock Indicator** on the Status Bar.

Select Font Command

Use the Select Font command under the Edit Menu to change the font settings for open NC programs. Font Settings are intended for viewing and printing purposes only. They do not affect the NC program in any way and are not stored within the program file. All open program windows use the same font settings.

To use the Select Font command:

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

Note:

The Undo command does not undo a font settings change.

Font Settings

The font settings control the font and font size used in the program edit window. The fonts listed are the fonts already installed on your system.

View Menu

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

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

Position Command

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

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

Machine Info Command

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

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

Jog Control Command

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

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

Operator Panel Command

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

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

Verify Window Command

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

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

Toolbars Command

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

To show or hide a toolbar:

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

Program Menu

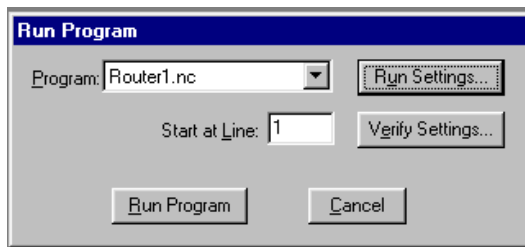
The Program Menu commands allow you to Run, Verify, or Stop an NC program.

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

Run/Continue Command

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

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



1. Select an NC Program.

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

2. Select a starting line.

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

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

3. Set the Run Settings

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

4. Set the Verification Settings

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

5. Run the Program

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

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

- u The name of the NC program.
- u Which block is currently being executed.
- u How many blocks are in the program.
- u Which tool is being used.
- u The number of passes made.
- u The tool diameter.
- u The spindle speed.
- u Operator messages such as which block paused the program or the error that caused the program to stop.

CAUTION

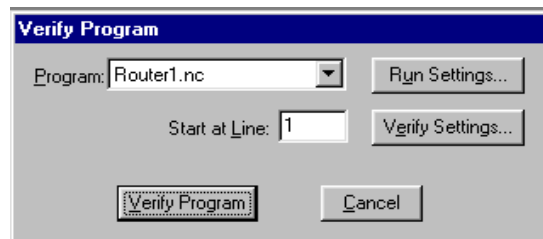
Always wear safety glasses and close the safety shield before running an NC program on the router. Always observe set up and safety precautions.

Verify Command

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

Note:

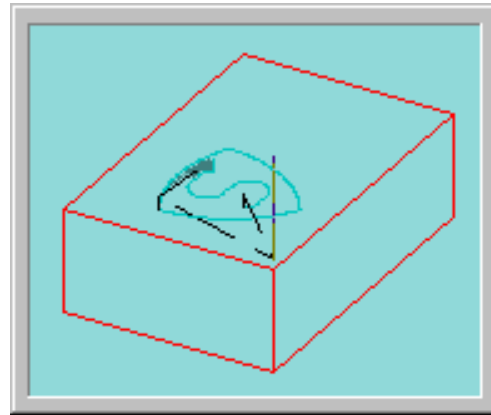
If you are verifying a part program for the first time, you should begin the verification at line one.



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

Begin the verification by pressing the Verify Program button. If the Verify Window is not already open, it will open automatically.

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



Estimate Runtime Command

Note:

The Calculated Distance is the total distance that the tool and the workpiece move in relation to each other. Every linear and rapid motion (no arcs) has a length ($\text{length} = \sqrt{dx^2 + dy^2 + dz^2}$). These lengths are added together, yielding the total distance. The feed rate of each segment is divided into the length to calculate the "ideal" time for each segment (assuming zero-communications overhead and infinite acceleration and deceleration). These times are added together to yield the total estimated time.

Use the Estimate Runtime command to calculate the approximate amount of time the router requires to run your program, and the approximate distance the machine travels while running.

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

- u Pause (G05/M00)
- u Chain (M20)
- u Skip (G31)
- u Wait for input high/low (G25/G26)
- u Write to file (M22)
- u Rerun (M47)

The Estimate Runtime 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, and places the cursor near the error.

Pause Command

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

To use the Pause command, select **Pause** from the Program menu, or click the Pause button on the Standard Toolbar.

To resume running a program after a **Pause**, press **F5**, click the **Run** button on the Standard Toolbar, or click the **Go** button on the Message Bar.

Feedhold Command

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

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

To use the Feedhold command, select **Feedhold** from the Program menu.

To resume running a program after a Feedhold, press **F5**, click the **Run** button on the Standard Toolbar, or click the **Go** button on the Message Bar.

Stop Command

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

To use the Stop command:

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

To restart a program after a Stop performed through the Control Program:

1. Manually jog the tool so it is above the workpiece to avoid a tool crash.
2. Select the **Run/Continue** command to restart the NC program. You will not have to reset the initial tool position (assuming the first block of your program moves the tool to the start position).

Note:

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

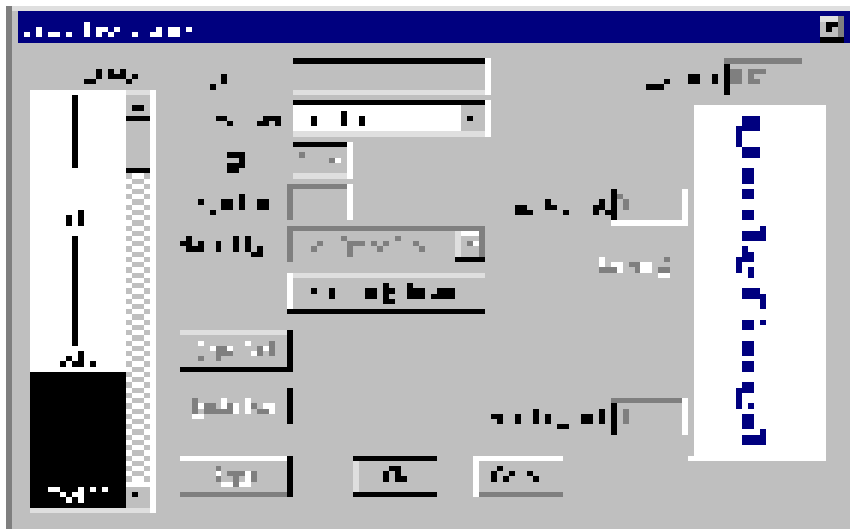
Tools Menu

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

Command:	Used to:
Setup Library	Define tools used with the router.
Setup Tool Wizard	Aid in establishing tool lengths for use with multiple tool programs.
Select Tool	Select a tool for use on the router.

Setup Library Command

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



You can create a new tool two ways:

- u Use the Copy and Paste buttons to copy an existing tool in the Tool Library box and paste it into the Tool Library box under an unassigned tool number. Assigned tool numbers are displayed with a tool icon. Unassigned tool numbers have no tool icon.
- u Manually create a tool using the features available in the Setup Tool Library dialog box.

To create a new tool:

1. Select an unassigned tool number from the **Tool Library** list.
2. Select a tool type, such as End Mill, from the **Tool Type** drop-down menu.
3. Enter a name for the tool in the **Description** field.
4. Enter the number of teeth (**Num Teeth**) on the tool.
5. Enter the **Material Type** from which the tool is made.
6. There is a secondary library for tool materials. You can use this library to create new materials or edit existing materials. To do this, click on the **Edit Tool Materials** button.
 - a) Enter a material **Name**.
 - b) Select a **Material Class**.
 - c) Enter a **Multiplier**. This should be set to “1” for now. This will be used when integrating with CAM for calculating feed rates and spindle speeds when generating tool paths.
 - d) Click on the **Add** button.
 - e) Click on the **Delete** button to remove tool materials you no longer need.
 - f) Press **Enter** or click on **OK** to accept the new material. Click on **Cancel** to exit the Tool Material Type dialog box without changing the material library.
7. Enter a tool **Diameter**.
8. Enter a tool **Height/Offset** value. You may also click on the **Current Z** button to establish the current Z axis position of the tool as the Height/Offset.
9. Enter a **Diameter Offset**. The Diameter Offset is the amount of deviation from the actual cutter diameter to the programmed cutter diameter. This parameter is very important when using tool diameter cutter compensation. For example, a 0.125” diameter end mill may be specified for a job. When loading tools, the diameter of the end mill may actually measure 0.1248”. The deviation in diameter is the Diameter Offset.
10. Apply the new parameters to the selected tool number by clicking the **Apply** button.
11. Press **Enter** or click on **OK** to accept the new tool information. Click on **Cancel** to exit the Tool Library dialog box without changing the tool library.

To alter an existing tool:

1. Select an existing tool from the **Tool Library** list.
2. Make the desired changes to the tool parameters, then click on the **Apply** button.
3. Press **Enter** or click on **OK** to accept the new tool information. Click on **Cancel** to exit the Tool Library dialog box without changing the tool library.

Setup Tool Wizard Command

Use the Setup Tool Wizard as an aid in establishing tool heights for use with multiple tool programs.

To use the Setup Tool Wizard:

1. Select **Setup Tool Wizard** from the Tools Menu. This starts the Wizard.
2. Follow the Wizard's instructions very carefully!

Select Tool Command

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

To select a tool:

1. Select the **Select Tool** command from the Tools Menu. The Select Tool for Use dialog box appears.
2. Select a tool from the drop-down **Tool** list. The tool parameters appear in the window to the right of the list.
3. Select an action to exit the dialog box:
 - u Click **Select Tool** button if the tool is already in the mounted spindle.
 - u Click **Insert Tool** to perform a tool change cycle. The router moves to its tool change position (maximum Z axis height) and you are prompted to insert a tool into the spindle. After you insert the tool and press F5, the router returns to its original position.
 - u Click **Cancel** to exit the Select Tool for Use dialog box without selecting a tool.

Setup Menu

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

Command:	Used to:
Set Position	Establish the X, Y and Z position of the tool.
Zero Position	Set the current tool position to X0, Y0, Z0.
Jog Settings	Establish speed and distance parameters for jogging the tool.
Run Settings	Establish options for running an NC part program.
Verify Settings	Establish options for verifying an NC part program.
Verify Type	Select centerline or solid view for tool path verification.
Set/Check Home	Establish or check a fixed known position on the machine.
Goto Position	Automatically move the tool to a specific set of coordinates on the router.
Units	Select Inch or Metric units of measure.
Coordinate Systems	Define multiple coordinate systems or select a new coordinate system.
Offsets	Modify the table of Offset values used for certain NC codes.
Spindle	Specify a spindle speed if you have not used an S code in your NC program.
Backlash	Define the amount of play in the Router turning screws.
Soft Limits	Establish software limits for each axis that are different than the actual fixed hardware limits on the router.
Preferences	Establish defaults for saving files and security features.

Set Position Command

Note:

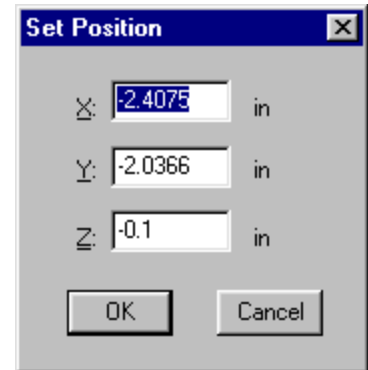
This command sets the position of the tip of the tool to the specified values. It takes the tool's defined height into account.

Use the Set Position command under the Setup Menu to set new X, Y and Z positions for the tool. This command establishes a Work Coordinate System in relationship to the Machine Coordinate System. Setting the X, Y and Z coordinates for the tool also defines the zero point of the coordinate system for absolute motion.

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

To set a new position:

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



The new position is displayed in the Position Window.

Zero Position Command

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

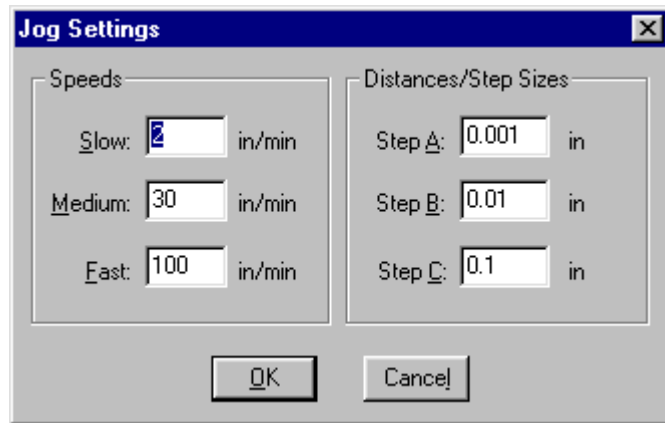
This command is also available under the Position Window Pop-up Menu.

To set the zero point:

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

Jog Settings Command

Use the Jog Settings command to enter speed and distance values for the Jog Control Panel. This command is also available through the Jog Control Panel Pop-up Menu.



To set jog parameters:

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

Jog Speed

The jog speed is the rate at which the tool moves along the X, Y, or Z axes. Select the speed by pressing the appropriate button. The speed can be Slow, Medium or Fast.

Default Values for Speed

1 ipm for Slow

12 ipm for Medium

50 ipm for Fast

These feed rates can be set as high as 50 inches per minute.

Jog Distance (Steps)

Distance values determine how far the tool moves each time a key is pressed. Referred to as Steps, the distance is selected by pressing the A, B or C buttons. The distance can be set at a low value (for instance 0.0005 inch) to move the tool for a precise cut, or at a high value (e.g. 0.5 inches) to position a tool.

Default Values for Steps

0.001 inch for Step A

0.01 inch for Step B

0.1 inch for Step C

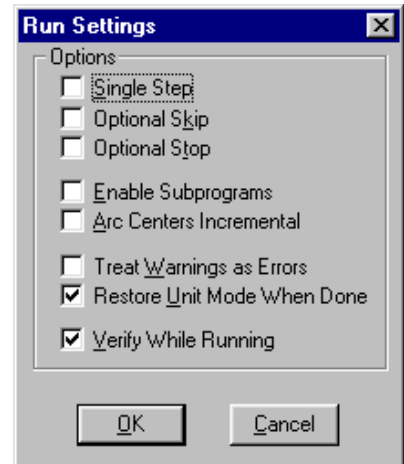
Note:

Pressing the Continuous (Cont.) button moves the tool continuously as long as the axis button is depressed. Once the axis button is released, the tool stops.

Run Settings Command

Use the Run Settings dialog box to set or clear options for running your NC program. The available options are:

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



Single Step

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

- u Click the **Run** button on the Standard Toolbar.
- u Click the **GO** button on the Message Bar.
- u Press **F5**.
- u Press **Enter**.
- u Select **Run/Continue** from the Program menu.

Optional Skip

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

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

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

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

Optional Stop

This enables or disables the optional stop code (M01). The optional stop code allows you to place an optional stop in your NC program. To activate the optional stop, select the Optional Stop box in the Run Settings dialog box or activate the Optional Stop button on the Operator Panel. Place an M01 on the line of code where you would like to pause.

With the Optional Stop option on, the M01 works like an M00 or a G05. With Optional Stop off, the M01 code is ignored, and the other codes on the line are executed as usual.

Enable Subprograms

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

Arc Centers Incremental

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

Treat Warnings as Errors

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

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

Restore Unit Mode When Done

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

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

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

Note:

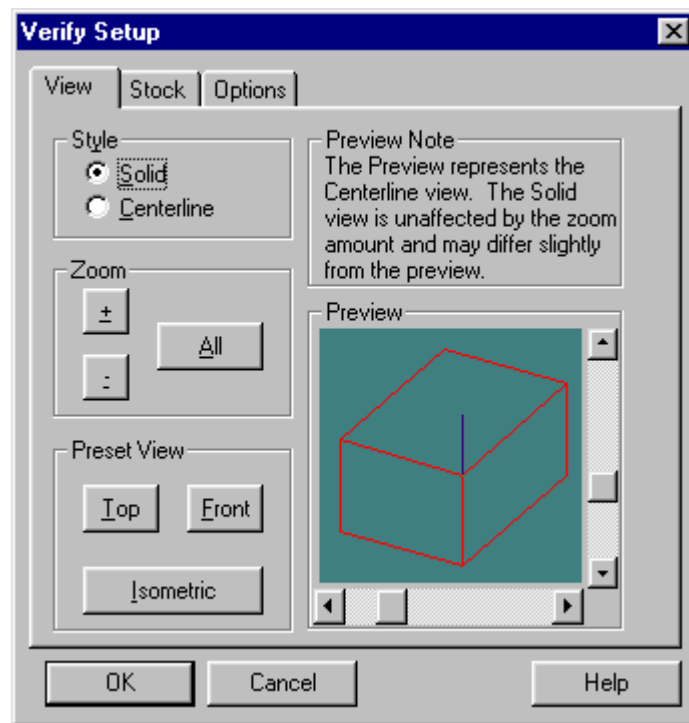
The NC program itself can override the default by placing the Incremental Arc Centers (I) or Absolute Arc Centers (A) codes in the first line of the file.

Verify While Running

If this box is checked, the Verify window will display the program verification while the program is running. The verification does not show exactly what is happening on the router. There is a delay between each tool motion. You will see each tool motion on the screen, but the *verification screen will pause* until the machine finishes the motion and the next program block is read.

Verify Settings Command

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



The Verify Settings dialog box is tabbed, with the settings organized into three panels: View Panel, Stock Panel and Options Panel.

View Panel

The View panel allows you to control the view style and zoom factor. It also offers a selection of preset views.

Style

Use these radio buttons to select between Solid and Centerline views of the stock. Solid View is a solid three-dimensional view of the workpiece and tool. Centerline View is a view that depicts the centerline of the tool, instead of a solid model of the tool. The workpiece is shown in wireframe.

Zoom

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

Button	Function
+	Zoom in on the stock.
-	Zoom away from the stock.
All	Fit the stock into the window.

Preset View

Use these buttons to select a perspective, then use the Preview Window to fine-tune the angle:

Button	Function
Front	View the stock directly from the front. The stock appears as a rectangle along the X, Z axes.
Top	View the stock directly from above. The stock appears as a rectangle along the X, Y axes.
Isometric	View the stock at an angle, in three dimensions.

Stock Panel

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

Stock Dimensions

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

Origin

Use this area to adjust the verification for different workpiece setups. Most NC programs set the 0,0,0 point at the top of the front-left corner of the stock. Occasionally, however, you may want to use a different origin (the center of the stock, for example). In this case, you must enter a different origin to properly verify your program. The values entered should correspond to the coordinates of the left corner of the stock relative to the 0,0,0 point for the program.

For instance, if the origin for the program is the center of a 3x2x1 piece of stock, set the origin to -1.5, -1.0.

Initial Tool Position

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

Options Panel

The Verify Options control certain aspects of the solid and centerline verification display.

Solid Options

Auto-refresh automatically refreshes the solid display of the workpiece if something changes during verification, such as resizing the verify window, changing the view, or changing the stock dimensions. When Auto-refresh is disabled, you will need to manually cause the verify window to update by selecting the Redraw command from the window's context menu. This setting only affects refreshing the window when no verification is in progress.

Centerline Options

Auto-refresh automatically refreshes the centerline display of the workpiece if something changes during verification.

Show Tool controls whether or not the tool is displayed in the Verify Window.

Show Rapid Moves displays the rapid moves made between the end point of one cut to the start point of another.

The Preview Window

The Preview Window appears in each of the above groups. The Preview Window shows you approximately what the Verify Window will look like. The Preview Window always depicts the stock in Centerline view.

In addition to depicting how the Verify Window will look, you can use the preview box to change the orientation of the stock:

- u Use the slider bars on the side of the Preview Window to rotate the stock along the X, Y plane and along the Z plane. There are two ways to control the sliders:
 1. Select the slider button by clicking on it with the mouse. Hold the left mouse button down while sliding the button along the bar until the stock is in the desired position.
 2. Select the slider button by tabbing to it. The button will blink to indicate it is selected. Use the arrow keys on the keyboard to move the slider button along the bar until the stock is in the desired position.
- u Use the mouse to rotate the stock in all planes simultaneously. “Grab” the stock by clicking and holding it with the mouse. Move the stock with the mouse until the stock is in the desired position.

Buttons

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

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

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

Verify Type Command

The Verify Type allows you to choose between a solid view and a centerline view in the Verify Window.

Set/Check Home Command

Use the Set/Check Home command under the Setup Menu to establish or check a fixed known position on the machine. This command is also available through the Jog Control Panel Pop-Up Menu.

Home

This option moves the router head to the ends of travel along each axis and sets the Machine Coordinate System to X0, Y0, Z0.

Quick Home

This option is for homing when you have lost position slightly, or when you start the machine at the beginning of the day and the machine hasn't moved. Using this option, the Control Program assumes that it is close to being homed (that it knows approximately what the machine position is). The machine rapids to a short distance from the limits before homing; this is much faster than feeding in at the normal slow rate. The Quick-Home feature is useful for homing the machine after hitting a limit, or after pressing the Emergency Stop while the machine is moving at a high speed.

Check Home

This option checks the reference point, identical to using a G27 code. It compares the reported position against zero to see if any position has been lost.

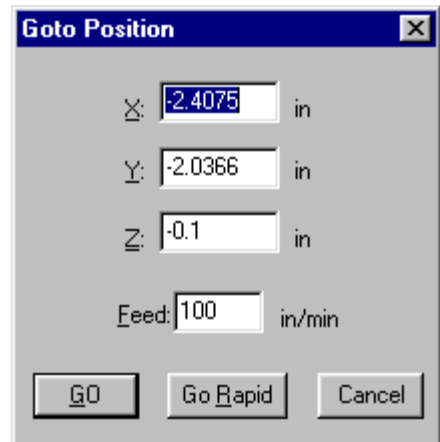
Goto Position Command

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

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

To use the Goto Position command:

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



Note:

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

Units Command

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

Coordinate Systems Command

Use the Coordinate Systems command under the Setup Menu to define multiple coordinate systems for machining more than one workpiece. This is often done for production runs of the same part. This command is also available under the Position Window Pop-up Menu. For an overview of Coordinate Systems, see page H-27.

To select an existing coordinate system:

1. Select **Coordinate Systems** from the Setup Menu.
2. Select an existing coordinate system from the fly-out menu. The coordinate systems available, CS1 through CS6, are equivalent to using the codes G54 through G59 in your NC program.
3. OR select the Work Coordinates command to cancel the Coordinate System offsets and return to Work Coordinates.

To define a new coordinate system:

1. Select **Coordinate Systems** from the Setup Menu.
2. Select **Setup** from the fly-out menu.
3. Select a coordinate system, then enter and apply the offsets.

Important Note:

Machine coordinates are established by Homing the system to establish a point of origin at the ends of travel on the Router. Once established, these coordinates remain fixed. Each time you turn on the router, the current machine position is stored as the machine coordinate system X0, Y0, Z0.

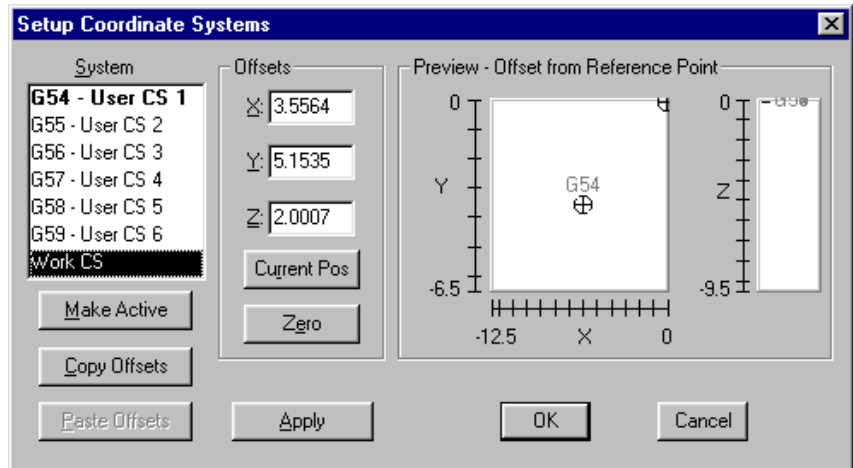
Work coordinates are different. When you use the Set Position command to set a point of origin on the workpiece, you are actually entering a position that is offset from the fixed machine position. The same thing occurs when you use a G92 code.

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

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

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

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



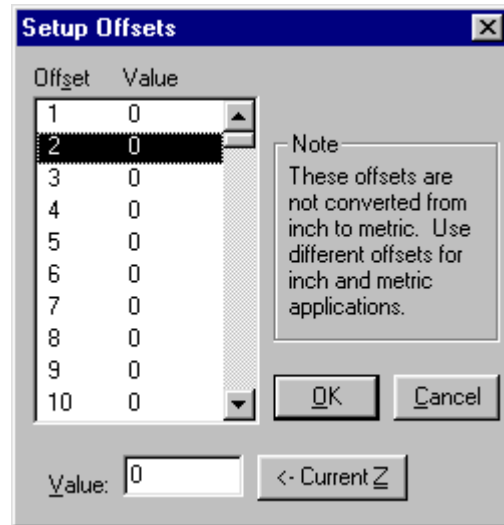
- a) Select a CS from the **System** box.

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

- b) Select the offsets for a user coordinate system by:
 - u Entering X, Y and Z offset values in the **Offsets** boxes.
 - u Clicking the **Current Pos** button to establish offset values based on the current tool position.
 - u Selecting a Marker in the **Preview Area** for the coordinate system and dragging it to the desired position.
 - u Copying offsets from one CS to another using the **Copy Offsets** and **Paste Offsets** buttons.
- c) Make the currently selected CS the active CS by selecting the **Make Active** button.
- d) Apply the coordinates or exit the dialog box:
 - u The **OK** button applies the changes you have made and closes the dialog box.
 - u The **Cancel** button closes the dialog box without applying any of the changes you have made since clicking the Apply button.
 - u The **Apply** button applies the changes you have made to the selected CS (you can still cancel changes once they have been applied by selecting the Cancel button).

Offsets Command

Use the Offsets command under the Setup Menu to compensate for variations in the cutting tools being used. The offset values are used for tool length offset, cutter compensation, and tool offset adjustment NC codes. For information on setting tool length offsets for multiple tool programming, see the Tool Setup Wizard in this section.



To enter a compensation offset:

1. Select **Offsets** from the Setup Menu.
2. Click on an Offset Number (the numbers 1 through 199 are available). This number only acts as a designation (a name) for the offset.
3. Enter an Offset Value in the **Value** box, or select the **Current Z** button to use the current Z position (also useful for defining tool heights if you are using G43 or G44 to specify tool heights).
4. Press **Enter** or click on **OK**. The Offset Value has been associated with the Offset Number. The next time you open the Offsets table, you will see the new offset.

The Offset Table

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

Set the values in the Offset Table using the Offsets command under the Setup Menu. The Offsets are stored in the LMR.INI file when running with a machine, and in TESTCTL.INI when running in simulation mode.

Spindle Command

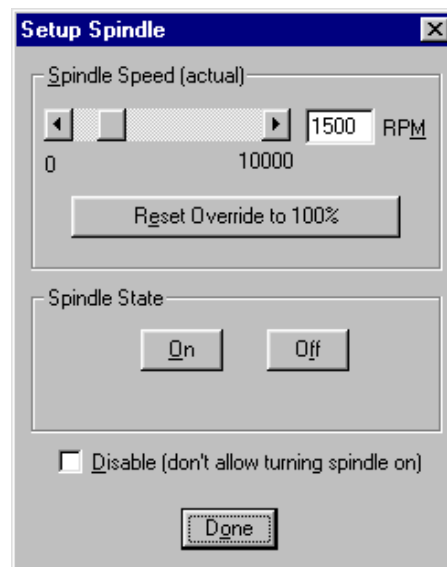
Note:

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

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

To specify a spindle speed:

1. Select the **Spindle** command from the Setup Menu. The Spindle Setup dialog box appears.
2. Select a spindle speed by entering a value in the RPM box, or by using the slider and arrow buttons.



Also in this dialog box:

- u Reset the spindle override value to 100% by clicking on the **Reset Override to 100%** button.
- u Change the On/Off state of the spindle using the **On** and **Off** buttons in the Spindle State area.
- u Disable spindle operation by selecting the **Disable** option. This is useful if you have a device mounted in the spindle that cannot withstand being rotated, such as a wired probe.
- u Select **Done** to close the dialog box.

Note:

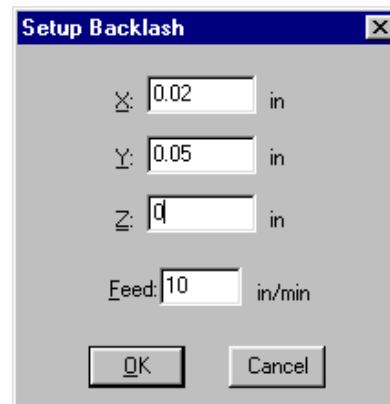
You should not specify too slow of a backlash feed rate or you will notice delays each time an axis with backlash changes direction.

Backlash Command

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

To establish new backlash parameters:

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



IMPORTANT!

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

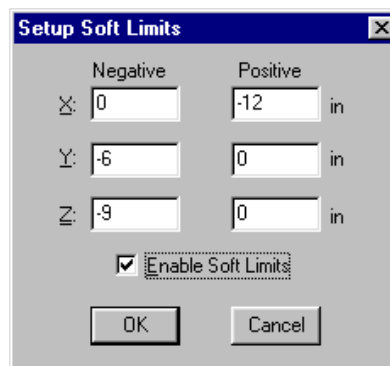
Soft Limits Command

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

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

To establish software limits on the router:

1. Select **Soft Limits** from the Setup Menu.
2. Enter the coordinates that define the software perimeter you wish to establish.
3. Click on the **Enable Soft Limits** option to enable soft limits. Use this option to turn soft limits on or off as you need to use them.
4. Press **Enter** or click on **OK** to accept the new soft limit parameters, or click on **Cancel** to exit the Setup Soft Limits dialog box without setting new soft limit parameters.



Note:

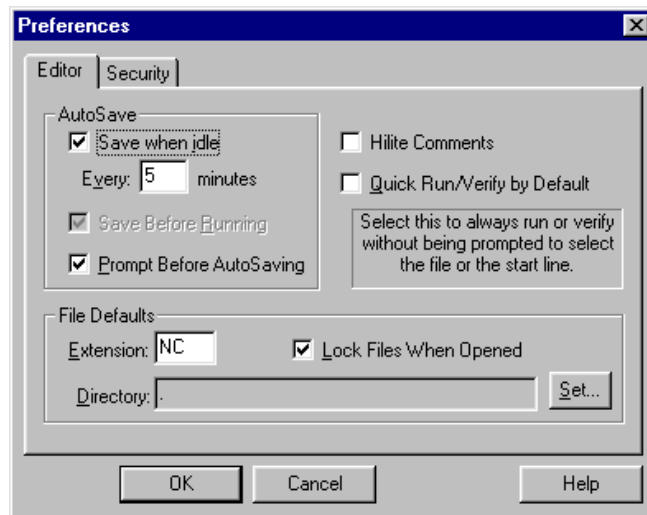
You must be within the Soft Limit range in order to Enable the Soft Limits. If you are outside of the Soft Limit range and Soft Limits are enabled when you close the dialog box, you will be instructed to jog the machine to within the Soft Limits. After you do so, you can open the Soft Limits dialog box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

Preferences Command

Use the Preferences command under the Setup Menu to establish defaults for saving files and setting security features.

To alter the system preferences:

1. Select the **Preferences** command from the Setup Menu.
2. Select either the Editor preferences tab or the Security preferences tab.



Editor Preferences

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

To select AutoSave features:

- u Select **Save when idle** and enter a value in the **Every:___ minutes** box to save your NC programs automatically at the specified time increment.
- u Select **Save Before Running** to save changes to your NC program prior to running it for the first time with the changes.
- u Select **Prompt Before AutoSaving** if you wish to be prompted by the Control Program before it automatically saves the NC program at the specified time increment.

To select File Default features:

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

The **Hilite Comments** box will highlight comments in the NC program in a different color.

The **Quick Run/Verify by Default** box suppresses the messages and dialog boxes that appear asking for the file name and starting line when you run a program.

To exit the Preferences dialog box:

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

To access Help for this panel:

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

Security Preferences

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

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

To secure the software using Administrator Mode:

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

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

To return to Administrator Mode:

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

To change your password:

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

To exit the Preferences dialog box:

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

To access Help for this panel:

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

Window Menu

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

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

Cascade Command

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

Tile Command

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

Arrange Icons Command

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

Window List Command

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

Help Menu

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

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

Help Command

Use the Help command to access the Help contents. You can also press the F1 key to get information about the currently highlighted command on a drop-down or pop-up menu.

Index Command

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

Using Help Command

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

Save Settings Command

The Save Settings command allows you to retain current library, security, screen and control settings as defaults.

- u Click on a particular setting(s) to tag it, then click on the **Save** button to save the selected items and exit the dialog box.
- u Click on the **Save All** button to save all settings without having to select each one.
- u Click on the **Cancel** button to exit the Save Defaults dialog box without changing the existing defaults.

Restore Settings Command

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

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

Tip of the Day Command

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

About BenchRouter ... Command

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

Selecting Commands

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

Select a Command Using Pop-Up Menus

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

To select a pop-up menu command:

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

The following windows have pop-up menus:

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

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

Program Edit Window Pop-up Menu

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

Cut Command

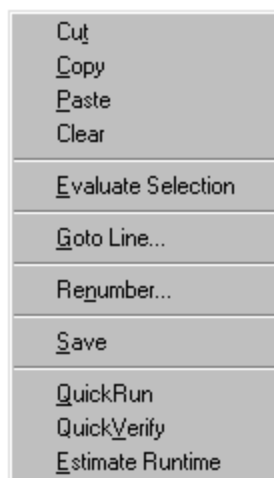
Cut is the same as selecting the Cut command from the Edit Menu.

Copy Command

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

Paste Command

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



Clear Command

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

Goto Line Command

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

Renumber Command

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

Save Command

Saves the current program.

QuickRun Command

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

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

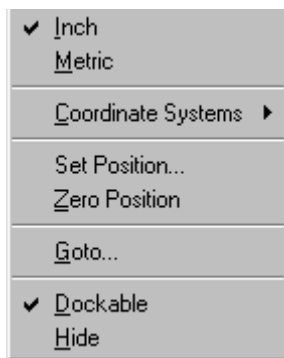
QuickVerify Command

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

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

Estimate Runtime

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



Position Window Pop-up Menu

The Position Window Pop-Up Menu contains these commands:

Inch Command

Automatically switches the units of measure for the application to inch units.

Metric Command

Automatically switches the units of measure for the application to metric units.

Coordinate Systems Command

Produces a fly-out menu that allows you to set up and select coordinate systems.

Set Position Command

Opens the Set Position dialog box.

Zero Position Command

Sets the current tool position to zero on all axes.

Goto Command

Opens the Goto Position dialog box.

Dockable Command

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

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

Hide Command

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

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



Verify Window Pop-up Menu

The Verify Window Pop-Up Menu contains these commands:

Solid Command

Switches the verification to a solid view.

Centerline Command

Switches the verification to a centerline view.

Redraw Command

This command repeats the most recent tool path verification simulation.

Stop Redraw Command

This command will interrupt a redraw currently in progress.

Reset Command

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

Dockable Command

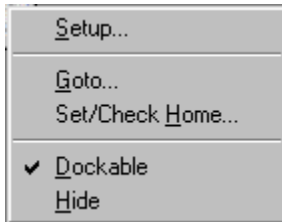
The Dockable command toggles the window between being a dockable window and being undockable. See *Docking and Floating Windows* for more information.

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

Hide Command

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

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



Jog Control Panel Pop-up Menu

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

Setup Command

This command opens the Jog Settings dialog box.

Goto Command

This command opens the Goto Position dialog box.

Set/Check Home Command

This command opens Machine Home/Reference Point dialog box.

Dockable Command

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

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

Hide Command

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

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

Select a Command Using Hot Keys

Some menu commands have one or more key designations next to them; those are the *hot keys* for that command. Pressing the hot key(s) selects the corresponding command.

To select a command using hot key, press the hot key for the desired command. For example, press **Ctrl+S** to save your program. Here's a list of the available hot keys:

Key(s):	Menu/Command:
Ctrl+C	Edit Menu/Copy
Ctrl+F	Edit Menu/Find
Ctrl+G	Edit Menu/Goto Line
Ctrl+H	Setup Menu/Set/Check Home
Ctrl+L	Edit Menu/Lock
Ctrl+N	File Menu/New
Ctrl+O	File Menu/Open
Ctrl+P	File Menu/Print
Ctrl+R	Setup Menu/Run Settings
Ctrl+S	File Menu/Save
Ctrl+Space	Program Menu/Stop
Ctrl+T	Tools Menu/Setup Library
Ctrl+V	Edit Menu/Paste
Ctrl+X	Edit Menu/Cut
Ctrl+Y	Edit Menu/Redo
Ctrl+Z	Edit Menu/Undo
Ctrl+Shift+Z	Setup Menu/Zero Position
F1	Help Menu/Help
F2	Edit Menu/Delete line
F5	Program Menu/Run/Continue
F6	Program Menu/Verify
F8	Setup Menu/Goto Position
Ctrl+KeyPad+	Increase Feed rate Override
Ctrl+KeyPad-	Decrease Feed rate Override
Ctrl+Backspace	Edit Menu/Undo

Shift+Delete	Edit Menu/Cut
Shift+F1	Help Menu/Context Help
F4	Activate Jog Control
Ctrl+F5	QuickRun
Ctrl+F6	QuickVerify
Ctrl+TAB	Next Edit Window
Ctrl+Insert	Edit Menu/Copy
Shift+Insert	Edit Menu/Paste
Pause	Edit Menu/Pause

Select a Command Using Toolbars

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

Positioning Screen Components

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

Positioning Toolbars

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

Note:

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

Positioning Information Areas

The information areas (Position, Machine Info, Jog Control, Operator Panel) are initially docked on the docking area on the right side of the screen. If these items are moved away from the docking area, they become floating windows or panels until they are moved back to the docking area.

The Verify window is initially not dockable.

To move an information area (create a floating window):

1. Press the Ctrl key.
2. Click on the information area with the left mouse button.
3. Hold and drag the area off the docking area.
4. Release the Ctrl key.
5. When the floating window is over its new location, release the mouse button. You can move and close the window as you would any other window.

To return a floating window to its docking area (the window must be dockable):

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

More information about dockable and undockable windows is provided in this section.

Positioning Program Edit Windows

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

Saving the Window Positions

After arranging the application's windows on the screen, you can save their positions. Use the Save Settings command on the Help menu to save the window positions (and any other selected items). To restore your saved window positions, or to restore the default window positions (the factory preset positions), use the Restore Settings command on the Help menu.

The Control Program automatically stores the current window positions when you exit the software, and automatically restores them the next time you run it.

Docking and Floating Windows and Toolbars

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

Docked Screen Components

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

Floating Screen Components

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

To select the Docking/Floating state of a toolbar:

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

To select the Docking/Floating state of a window:

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

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

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

The Setup Program

The Setup Program allows you to set program and hardware defaults. In order to access the Setup Program, you must first exit the Benchtop Router program. Locate the Benchtop Router folder on your hard drive, (default install is in the C:\>Windows\Programs folder) and double click the Setup Icon. The program will start and you will see the Welcome screen. You may choose from the file tabs to view the defaults for each category.

The Setup Program provides defaults under the following categories:

- u Welcome
- u Interface Card
- u General

Welcome Panel

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

Interface Card Panel

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

General Panel

This panel allows to you alter several software defaults.

- u **Run Offline** starts the Control Program in Simulate Mode without checking for a machine connection.
- u **Decimal Places for Inch Values** controls the display of values in dialog boxes. When in Metric mode, the software displays 1 less than the specified number of decimal places.
- u **NC Programming Settings** controls several programming options:
 - u The maximum number of subprograms.
 - u The maximum depth of subprograms; the number of nested subprograms.
 - u The default L-factor; the angle at which a line segment approximates a portion of an arc.
 - u The default feed rate; used when no feed rate is specified in the NC program.

Note:

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

Note:

There are two Offset Tables, one for Simulate Mode, and one for Normal Mode. If your off-line NC program development includes using values from the offset table, you must ensure that they are present in the Normal Mode offset table in the same positions as in the Simulation Mode offset table.

Offset Table

The Offset Table stores 200 values which are used in several operations including tool offset adjustment, cutter compensation, and tool length compensation to ensure uniform application of an offset value. The numbers are stored as unit-less values; their interpretation depends on the Unit mode currently in effect. Offset zero is always 0.0 and cannot be modified. Using Offset zero for most compensations cancels that compensation.

Set the values in the Offset Table using the Offset command under the Setup Menu. The Offsets are stored separately for the machine, and for Simulation Mode.

Working in Simulation Mode

Simulation mode is provided to allow the Control Program to be used when your computer is not connected to the router. This is primarily to support the off-line development of NC programs.

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

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

Basic CNC Programming

The Elements of an NC Part Program

Categories of NC Code

General Programming Suggestions

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 or value*). There are many categories of address characters used in NC part programs for the Benchtop Router (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:

```
N1G90G01X.5Y1.5Z0F1
```

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

- N1** This is the block sequence number for the program. Block 1 is the first block in the program.
- G90** This indicates absolute coordinates are used to define tool position.
- G01** This specifies linear interpolation.
- X.5** This specifies the X axis destination position as 0.5".
- 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 Benchtop Router. Here is a list of the NC codes (designated by the address character) supported by the router.

Code:	Function:
%	Incremental Arc Centers (Fanuc).
\$	Absolute Arc Centers (EIA-274).
\	Skip.
/	Optional skip.
D	Compensation offset value.
F	Feed rate in inches per minute; with G04, the number of seconds to dwell.
G	Preparatory codes.
H	Input selection number; Tool length offset.
I	Arc center, X axis dimension (circular interpolation).
J	Arc center, Y axis dimension (circular interpolation).
K	Arc center, Z axis dimension (circular interpolation).
L	Loop counter; Program cycle (repeat) counter for blocks and subprograms; homing tolerance, arc interpolation control.
M	Miscellaneous codes.
N	Block number (user reference only).
O	Subprogram starting block number.
P	Subprogram reference number (with M98 or M99); Uniform scale multiplier (with G51).
Q	Peck depth for pecking canned cycle.
R	Arc radius for circular interpolation (with G02 or G03); Starting reference point for peck drilling (with canned cycle codes); Rotation angle for coordinate rotation.
S	Spindle speed.
T	Tool specification.
U	Incremental X motion dimension.
V	Incremental Y motion dimension.
W	Incremental Z motion dimension.
X	X axis motion coordinate.
Y	Y axis motion coordinate.
Z	Z axis motion coordinate.
;	Comments.

Incremental Arc Center (%)

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

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

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

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

Absolute Arc Centers (\$)

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

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

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

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

Skip (\) and Optional Skip (/)

Note:

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

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

To use the Skip code (\):

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

To use Skip code (\) with a parameter:

Use the Skip code with a parameter to instruct the Control Program to execute the line of code every nth pass. Place the code at the beginning of the line you wish to skip.

The syntax is: `\n`, where **n** is the number of passes between executions.

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

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

1. Place the code at the beginning of the line you wish to skip.
2. Select the Optional Skip option from the Run Settings dialog box or the Operator Panel.

When you run the NC program, the specified line will be skipped. If you do not select the Optional Skip option in the Run Settings dialog box, the skip code is ignored and the line is executed normally.

To use the Optional Skip code (`/`) with a parameter:

Use the Optional Skip code with a parameter to instruct the Control Program to execute the line of code every *n*th pass. Place the code at the beginning of the line you wish to optionally skip.

The syntax is: `/n`, where **n** is the number of passes between executions.

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

Compensation Offset Value (D Code)

The D code is used to select a value from the Control Program's Offset Table. For example, D1 selects entry number 1 from the Offset Table. Selecting offset 0 (D0) cancels cutter compensation.

Use the D code with:

- u Cutter compensation codes to specify the tool radius.
- u Tool offset adjust codes to specify a consistent increase or decrease in the commanded movement.

Use the Offsets command under the Setup Menu to view and manage the Offsets Table.

Feed Rate (F Code)

Use the F code to:

- u Specify the rate of speed at which the tool moves (feed rate) in inches per minute (ipm). For example, F3 equals 3 ipm.

The feed rate should be set to a low value (up to 50 ipm) for cutting operations. Feed rate values are in millimeters per minute (mpm) when using metric units. The Control Program limits the programmed feed rate so it doesn't exceed the maximum allowed by the machining center.

- u Specify the number of seconds to dwell when used with the G04 code.

Preparatory Codes (G Codes)

G codes take effect before a motion is specified. They contain information such as the type of cut to be made, whether absolute or incremental dimensioning is being used, whether to pause for operator intervention, and so on.

More than one G code from different groups can appear in each NC block. However, you may not place more than one G code from the same group in the same block.

The G codes supported by the Control Program fall into the following groups:

- u The Interpolation Group
- u The Units Group
- u The Plane Selection Group
- u The Wait Group
- u The Canned Cycle Group
- u The Programming Mode Group
- u The Preset Position Group
- u The Compensation Functions Group
- u The Coordinate System Group
- u The Polar Programming Group

The Interpolation Group

The Interpolation Group allows you to specify the type of motion for interpolation. These G codes are retained until superseded in the NC program by another code from the Interpolation Group.

The supported interpolation G codes are:

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

The Units Group

By default, an NC program is interpreted using the units of measure (inch or metric) specified using the Units command on the Setup Menu.

The codes in the Units group, G70 (inch) and G71 (metric), are used to override the Units command for the entire program.

If the code is placed at the beginning of the program before any tool motions are made, that unit of measure is assumed for the entire program. Otherwise, it affects the rest of the program following the code. You can use these codes to switch between inch and metric modes throughout your program at your convenience.

The Fanuc equivalents, G20 (inch) and G21 (metric), can also be used.

The Plane Selection Group

This group of codes allows you to select different planes for circular interpolation. G17 is the Control Program default.

The supported Plane Selection Group codes are:

- G17** Select the X,Y plane for circular interpolation. 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. 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. The arc center coordinates are given by J for the Y axis and K for the Z axis.

The Wait Group

Wait Group codes apply only to the block in which they appear. The program does not continue until the wait conditions are satisfied.

The supported Wait Group codes are:

- G04** Dwell (wait): Pause between motions on all axes for the number of seconds specified by the F code, then continue the program. Because the F code is used to specify the number of seconds, you cannot also specify a new feed rate in the same block.
Example: G04F10;Wait for 10 seconds
- G05** Pause: Used for operator intervention. Stop motion on all axes until the operator manually resumes program execution using the Run/Continue command.
- G25** Wait until TTL input #1 (Robot 1 or user input 5) goes high before executing the operations on this block. Used for robot synchronization (see Section K for more information). Use the H code to specify an input other than the default, H5.
- G26** Wait until TTL input #1 (Robot 1 or user input 5) goes low before executing the operations on this block. Used for robot synchronization (see Section K for more information). Use the H code to specify an input other than the default, H5.
- G31** Linear move to specified coordinate; used with H code to specify both input number and the High or Low condition for stop (designated by the operator input , + or -). The move occurs until an input is triggered or until the specified end point is reached. The move stops short if specified input goes High (if H is positive) or Low (if H is negative). The default is input 5 (Robot 1) High.
You can have the control program go to a specified block (N Code number) if the input meets the required condition. Use a P code to specify the destination, as with the M99 code.
For example, **G31X5Y5H6P50000** instructs:
- u Move (using the current programming mode) to the X5 and Y5.
 - u If input 6 (Robot 2) goes low during the move, stop the motion and jump to block number 50000.
 - u If input 6 (Robot 2) doesn't go low, complete the move and continue with the next block in the program.
- G35** Wait until TTL input #2 (Robot 2 or user input 6) goes high before executing the operations on this block. Used for robot synchronization (see Section K for more information). This is the same as using the codes G25 H6.
- G36** Wait until TTL input #2 (Robot 2 or user input 6) goes low before executing the operations on this block. Used for robot synchronization (see Section K for more information). This is the same as using the codes G26 H6.

The Canned Cycle Group

Canned cycle codes allow you to perform a number of tool motions by specifying just one code. Canned Cycle codes are typically used for repetitive operations to reduce the amount of data required in an NC program. Canned cycle codes are retained until superseded in the program by another canned cycle code.

The supported Canned Cycle codes are:

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

Refer to Section G for more information on these functions.

The Programming Mode Group

Programming mode G codes select the programming mode, absolute (G90) or incremental (G91). These codes remain in effect until superseded by each other. The default code on program start up is G90.

With absolute programming, all X, Y and Z coordinates are relative to origin of the current coordinate system. With incremental programming, each motion to a new coordinate is relative to the previous coordinate.

The supported Programming Mode codes are:

- G90** Absolute programming mode
- G91** Incremental programming mode

The Preset Position Group

The preset position G codes move the tool to a predetermined position, or affect how future motions will be interpreted.

The supported Preset Position codes are:

- G27** Check reference point: This code moves the machine to its home position and compares the reported position against zero to see if any position has been lost. The difference between the reported position and zero is compared to a tolerance value specified using the Setup Program. Use the L code in this block to override the tolerance value from the Setup Program.
- 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** Set position: This code works like the Set Position command under the Setup Menu. 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.
- G99** Rapid move to point R (surface of material or other reference point) after canned cycle complete.

The Compensation Functions Group

Use the cutter compensation NC codes to automatically compensate for the variations in a cutting tool's radius and length. Refer to Reference Guide: Section H for more information on using cutter compensation.

The supported Compensation codes are:

- G39** Corner offset circular interpolation.
- G40** Cancel 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 in the Offset Table, specified by an H code.
- G44** Tool length offset: Shifts Z axis in a negative direction by a value in the Offset Table, specified by an H code.
- G45** Tool offset adjust: Increases the movement amount by the value stored in the offset value memory.
- G46** Tool offset adjust: Decreases the movement amount by the value stored in the offset value memory.
- G47** Tool offset adjust: Increases the movement amount by twice the value stored in the offset value memory. Refer to the User's Guide for more information.
- G48** Tool offset adjust: Decreases the movement amount by twice the value stored in the offset value memory.
- G49** Cancels tool length offset.
- G50** Cancels scaling.
- G51** Invokes scaling.
- G68** Invokes rotation.
- G69** Cancels rotation.

The Coordinate System Group

Use the coordinate system codes to establish multiple coordinate systems on one or more workpieces to create multiple parts.

For instance, you can run a part program using a typical coordinate system (with the point of origin on the 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 workpiece.

There are seven coordinate system codes. One of these codes (G53) is used to rapid to specified machine coordinates. The other six codes allow you to make up to six individual parts on the same workpiece by specifying different work coordinate systems for each part.

The coordinate system codes are G54 through G59, referring to coordinate systems 1 through 6 respectively. These coordinate systems may be set through the Coordinate Systems command on the Setup Menu.

The Polar Programming Group

The polar programming codes allow you to perform polar programming 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. Refer to Section H for more information on using polar programming.

The supported Polar Programming codes are:

- G15** Polar programming ON
- G16** Polar programming OFF

Input Selection Number/Tool Length Offset (H Code)

The H code has multiple uses. It can be used to specify inputs, input state changes, outputs, and offset amounts.

H Code Inputs

H1	Emergency Stop
H2	Safety Shield
H3	+Limit
H4	-Limit
H5	Robot 1
H6	Robot 2

H Code Outputs

H1	Spindle
H2	Acc1
H3	Acc2
H4	Robot 1
H5	Robot 2

Use the H code in conjunction with:

- u The wait codes G25 and G26, to specify the input number. If the H code is NOT used with these G codes, input 5 is assumed.
- u The wait code G31, to specify input change to high or low. If the H code is NOT used with this G code, input 5 High is assumed.
- u The tool length offset codes G43 and G44, to specify the amount of Z axis shift. 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.
- u The transmit codes M25 and M26 for interfacing with robots or other external devices, to specify the output number. If the H code is not used with these M codes, output 4 (Robot 1) is assumed.

X Axis Coordinate of Center Point (I Code)

In absolute programming mode (G90), the I code specifies the X axis coordinate of the center point of an arc or circle when using circular interpolation. In incremental mode (G91), the I code specifies the X axis distance from the start point of motion to the center point of the arc for circular interpolation.

If no I code is specified, the system uses the current X axis location as the X axis center of the arc. If in Fanuc mode, all arc centers are incremental.

The I code is also used with the G51 code to specify the scale factor for the X axis when performing scaling functions, including scaling each axis and mirror scaling. Refer to Reference Guide: Section H for more information on using scaling.

Y Axis Coordinate of Center Point (J Code)

In absolute programming mode (G90), the J code specifies the Y axis coordinate of the center point of an arc or circle when using circular interpolation. In incremental mode (G91), the J code specifies the Y axis distance from the start point of motion to the center point of the arc for circular interpolation.

If no J code is specified, the system uses the current Y axis location as the Y axis center of the arc. If in Fanuc mode, all arc centers are incremental.

The J code is also used with the G51 code to specify the scale factor for the Y axis when performing scaling functions, including scaling each axis and mirror scaling. Refer to Reference Guide: Section H for more information on using scaling.

Z Axis Coordinate of Center Point (K Code)

In absolute programming mode (G90), the K code specifies the Z axis coordinate of the center point of an arc or a circle when using circular interpolation. In incremental mode (G91), the K code specifies the Z axis distance from the start point of motion to the center point of the arc for circular interpolation.

If no K code is specified, the system uses the current Z axis location as the center of the arc. If in Fanuc mode, all arc centers are incremental.

The K code is also used with the G51 code to specify the scale factor for the Z axis when performing scaling functions, including scaling each axis and mirror scaling. Refer to Reference Guide: Section H for more information on using scaling.

Angle of Arc Resolution, Loop Counter (L Code)

The L code specifies the angle of arc resolution in circular interpolation programming. When the system executes a circular motion, it actually splits the arc into a series of line segments to approximate the circle. The L code specifies the angle in degrees which a line segment approximates a portion of the arc. The smaller the angle, the smoother the cut. A negative value for L will generate a *normalized* L factor (degrees x radius {in inches}) so larger radii have smaller degree values. For example, with the default L factor of -1:

- u An arc with a radius of 1 inch will have line segments approximating every 1 degree of the arc.
- u An arc with a radius of .5 inches will have line segments approximating every 2 degrees of the arc (Degrees = $-L/R$, or Degrees = $-(-1) / .5$, which is 2).
- u Note that the line segments generated by a normalized L code are always approximately the same length regardless of the arc's radius (The length of the arc segment being represented by each line segment is exactly the same).

The default setting for the Router is -1, and typically this will work quite well. You may notice the Router hesitating on arcs if the resolution is too fine. The L code can be a fraction of a degree (such as L.5), but it must be large enough so the Router will move at least the minimum motion (.0005") on each of the straight line motions.

Use the L code with:

- u The M98 code as a loop counter for subprograms.
- u The M47 code as a program cycle counter, to repeat a program a finite number of times.
- u The G27 code to specify tolerance with homing commands (this is an LMC-specific NC language extension). The difference between the current position and 0 is compared to a tolerance value specified using the Setup Program; use the L code to override this tolerance value.
- u An L code that is not used by one of these codes is used as an Arc Resolution factor.

Miscellaneous Codes (M Codes)

M codes control a variety of Router functions while a part program is running. Only one M code should be specified per NC block. M codes and motion commands should be placed on separate blocks to avoid confusion over whether an M code is activated during or after a motion command.

The supported M codes are:

- M00** Pause: Allows you to place a pause in your code. Acts like a G05 pause.
- M01** Optional Stop: Allows you to place an optional pause in your code. Place an M01 in the block of code where you would like to pause. There are switches to activate or deactivate the Optional Stop code in the Run Settings dialog box and on the Operator Panel. With Optional Stop on, the M01 works like a G05 pause. With Optional Stop off, the M01 code is ignored, and the other codes on the block are executed as usual.

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.

- M02** End of Program: Takes effect after all motion has stopped; turns off drive motors, and all outputs, including the spindle and the accessory outlets.
- M03** Spindle Motor On: Activated concurrently with motion specified in the program block; remains in effect until superseded by M05.
- M05** Spindle Motor Off: Activated after the motion specified in the program block; remains in effect until superseded by M03.
- M06** Tool Change: Pauses all operations, turns off spindle, retracts spindle for tool change. If you are using an ATC, this performs the complete tool change cycle.
- M08** Accessory #1 On: Turns on ACC 1 accessory AC outlet concurrently with the motion specified in the program block; remains in effect until superseded by M09. This is the same as using the M25 H2 codes.
- M09** Accessory #1 Off: Turns off ACC 1 accessory AC outlet after the motion specified in the program block is completed; remains in effect until superseded by M08. This is the same as using the M26 H2 codes.
- M10** Accessory #2 On: Turns on ACC 2 accessory AC outlet concurrently with the motion specified in the program block; remains in effect until superseded by M11. This is the same as using the M25 H3 codes.
- M11** Accessory #2 Off: Turns off ACC 2 accessory AC outlet after the motion specified in the program block is completed; remains in effect until superseded by M10. This is the same as using the M26 H2 codes.
- M20** Chain to Next Program: This code is used to chain several NC files together. It appears at the end of a part program and is followed on the next line by the file name of another program which is executed when all motion stops. Here's an example of a part program chain to another program:
- ```
N37 Z.2
N38 M20
PROGRAM2.NC; Chain to PROGRAM TWO
```
- If the two programs you are chaining are not in the same directory on your computer, you must specify the full path name for the next program file. If the software cannot locate the specified file, you will be prompted to find it.
- M22** Output current position and other information to file.
- M25** Set TTL output #1 (Robot 1 or Output 4) On: Used for robot synchronization (see Section K for more information). Use the H code to specify an output other than the default, H4.
- M26** Set TTL output #1 (Robot 1 or Output 4) Off: Used for robot synchronization (see Section K for more information). Use the H code to specify an output other than the default, H4.

- M30 End of program: Same as M02.
- M35 Set TTL output #2 (Robot 2, or Output 5) On: Used for robot synchronization (see Section K for more information).
- M36 Set TTL output #2 (Robot 2, or Output 5) Off: Used for robot synchronization (see Section K for more information).
- M47 Rewind: Restarts the currently running program; takes effect after all motion comes to a stop. Typically used with an L code to repeat a program a set number of times.
- M98 Call to subprogram. Use the P code to specify the subprogram starting block number. Use the L code to specify the number of times the subroutine is executed. You can nest subprogram calls to a depth of 20.
- M99 Return from Subprogram; Goto
- M105 Operator Message (LMC)

**Note:**

All text on the same block, after the closing parenthesis, is output to the file, with all valid macros being replaced as it is written.

## M22: Output Current Position to File

The M22 code is used to write information to a file while a program is running. Typically, this code is used when digitizing to write the current X, Y, and Z machine coordinates to a file. The proper format for using this code is:

M22(filename) Data to Write to File. The first time the Control Program encounters an M22 code, it opens the specified file. You must enclose the name of the file in parentheses for the Control Program to recognize it. If you do not specify any DataToWriteToFile text, the default data is output. This default is the current position, equivalent to specifying 'X@X Y@Y Z@Z'. Notice that the @X,@Y,@Z 'macros' are replaced by the actual machine position when the data is written. Each M22 code automatically adds a linefeed to the end of its output so the next M22 starts on a new line.

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

If you use more than one M22, only the first occurrence must have the file name in the parentheses. The remaining M22's may have empty parentheses, ( ), or may specify a different file.

If you want to generate more than one file at a time, you must include the filename each time you specify M22. If a filename is not specified, the first file opened is used.

**Example:**

```

. . . ; code to move to position
; Open my1.xyz, discard contents, write coordinates
M22(my1.xyz)
. . . ; code to move to next position
; Append to currently open data file
M22()

```



```
. . . ; code to move to next position
;Open my2.xyz and append coordinates
M22(my2.xyz,A)
```

Information about digitizing is provided with the digitizing package. For additional information, please call Light Machines Technical Support.

Special codes that can be used with M22 to generate run-time reports.

```
@X Current X position (in current coordinate system).
@Y Current Y position (in current coordinate system).
@Z Current Z position (in current coordinate system).
~ (tilde) New Line (starts a new line in the file).
@TD Time of Day (12hour) "11:59:59AM"
@TC Time (elapsed) for cycle "99:11:59" (0's trimmed from left)
@TT Time Total (program run) "99:11:59"
@TA Time Average (per cycle) "99:11:59" ("??:??" if first part)
@TL Current Tool #. "5"
@C Cycle # (Current Pass) "3"
@D Date "12/31/94"
@FN Current File (w/o path) "Part.NC" ("Untitled.NC" if untitled)
\t TAB
\\ Outputs a single '\ character.
```

**Example:**

```
; Start of file
... ; Process a single part
; Output part time statistics to file c:\Reports\Stats.txt (c:\Reports directory must exist)
M22 (c:\Reports\Stats.txt,A) Part #@C processed in @TC.
M47 L50 ; We want to process 50 parts.
```

## **M99: Return from Subprogram, Goto**

The M99 code has two specific uses; it can be used as a command to return from a subprogram or it can be used as a goto command.

Using M99 with subprograms:

When used in a subprogram, this code returns you to the block following the last M98 (Call to Subprogram) command. If the M98 used an L code to specify multiple calls to the subprogram, the M99 will return to the block containing the M98 until all the specified number of subprogram calls have been made; then it will proceed to the block following the M98.

You can use the P code plus a block number to override the block returned to; however, if this feature is used from a nested subprogram call, all return targets are discarded. The rules for a Goto target block apply to this use as well.

Using M99 as a Goto command:

This command can be used in the main NC program as a Goto command to jump to a block on a line before the first subprogram (as denoted by the O code). You may not jump to a block that is after the first subprogram.

Use the P code to identify the block number being jumped to. Control is transferred to the first occurrence of this N code; it cannot be used to transfer control between chained programs (see M20).

This command can be used anywhere in the program to change the flow of program execution. It is good programming practice to place this command on a line by itself to improve the program's readability.

## M105: Operator Message

This command is used to display messages in the Control Program. It provides a way to display messages to the operator on the message bar while an NC program is running. You can also pause the program with a custom message. This is a non-standard, Light Machines code.

By default, the message is centered, displayed as a Normal Message, and is persistent (not cleared until the program clears it or until the next message is displayed).

The correct format for using this code is:

`M105(the message);comment`

For instance, the following line of code displays a simple message:

`M105(End of Roughing Segment);Normal Message; doesn't pause`

Messages can be altered by using the following alternate characters:

`^` Displays the message and performs a pause requiring operator intervention to continue.

`~` Displays the message as a Warning Message.

`\b` Beeps when the message is shown.

The format for using the M105 code with an alternate character is:

`M105(alternate character plus the message) ;comment`

For example:

`M105(~WARNING);Warning Message , doesn't pause`

Here are some other examples of how to use this code:

`M105( ) ; clears current message`

`M105(^Please stop and read this!) ; Normal Message, pauses`

`M105(~^I MEAN IT!) ; Warning Message, pauses`

`M105(\b\b\b) ; Clears current message, beeps 3 times, and doesn't pause`

---

## Block Number (N Code)

N codes have two uses:

- u To provide destinations for Gotos (M99) elsewhere in the program.
- u To clearly show the organization of the code and improve readability.

Using the N code is optional; however, when you do use the N code, it must be the first character in the block. Other than for the above stated uses, N codes are ignored by the Control Program. Their presence, absence, or sequential value does not affect the execution of the NC program in any way (unless the target of a goto is missing).

You may have N codes on some blocks and not on others. N code sequence numbers do not have to be in order, but regular sequential order does make it easier to follow and reference sections of the program. The Control Program can change the N codes in a program by inserting, removing, or renumbering them.

---

## Subprogram Block Number (O Code)

The O code is used to indicate the start of a subprogram, and must be followed by a number which identifies the subprogram. The O code replaces the N code in the first block of the subprogram.

To call a subprogram, use the M98 code; use the P code to specify which subprogram to execute. To return from the subprogram, use the M99 code.

Only the first block in the subprogram contains the O code. The remaining blocks may contain N codes. The O and N code numbers may be used to help identify and set apart the subprogram to improve readability, for example:

```
M98 P50000 ;call to first subprogram
...;after first subprogram is finished, M99 code returns to this point
...
M98 P60000 ;call to second subprogram
...;after second subprogram is finished, M99 code returns to this point
...
O50000 ;start of subprogram
N50010 ;first line of subprogram
N50020 ;second line of subprogram
N50030 M99 ; last line of subprogram
...
O60000 ;second subprogram
N60010 ;first line of second subprogram
N60020 ;second line of second subprogram
N60030 M99 ;last line of second subprogram
```

---

## Subprogram Reference Number (P Code)

Use the P code with:

- u The G31 code to reference a goto target block.
- u The G51 code to specify a uniform scaling factor.
- u The M98 code to reference a subprogram using the subprogram block number.
- u The M99 code to specify a return block number as a goto target.

---

## Peck Depth (Q Code)

The Q code is used with the G83 code in canned cycle peck drilling to specify the incremental depth of each peck.

---

## Radius of Arc, Drilling Start Location (R Code)

As an alternative to specifying the center point of an arc (I Code, J Code, or K Code) you can specify the arc radius. Use the same value for the radius in both absolute and incremental programming modes.

G02 or G03 specifies the direction of motion.

Positive values for R (radius) are specified for arcs up to 180°. Negative values are used for arcs greater than 180°. Full circle arcs cannot be performed with an R code. Split the circle into two arcs, or use center point (I, J, and K) values for full circles.

Use the R code in canned cycles to specify a Z axis reference point for peck drilling. The point can be at the material surface or at another reference point. The R code is also used to specify the rotation angle, in degrees, with the G68 code.

### Note:

Full circles (360° 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 circles.

---

## Spindle Speed (S Code)

Use the S code to set the spindle speed from within the NC program. Spindle speed is specified by the address character “S” followed by a parameter that represents the speed in RPMs. For example, S750 is the designation for a spindle speed of 750 RPM. For the S code to have an effect, the spindle must be turned on by the M03 command. If the spindle is off, the spindle speed is stored and used when the spindle is turned on again within the program. Use the M05 command to turn the spindle off.

---

## CAUTION

---

Using multiple tools is an advanced operation, and should not be attempted by persons unfamiliar with using the Benchtop CNC Router.

---

## Tool Selection (T Code)

T codes specify the tool offset (number) in the Tool Library in multiple tool machining operations. Tools are specified by the address character “T” followed by a parameter that represents the number of the tool. For example, T3 is the designation for tool number three.

---

## X Axis Coordinate (X or U Code)

An X code specifies the coordinate of the destination along the X axis. A U code is used in absolute programming mode (G90) to specify an incremental X motion. You cannot use the U code to mix incremental and absolute programming in the same block.

---

## Y Axis Coordinate (Y or V Code)

A Y code specifies the coordinate of the destination along the Y axis. A V code is used in absolute programming mode (G90) to specify an incremental Y motion. You cannot use the V code to mix incremental and absolute programming in the same block.

---

## Z Axis Coordinate (Z or W Code)

A Z code specifies the coordinate of the destination along the Z axis (spindle axis). A W code is used in absolute programming mode (G90) to specify an incremental Z motion. You cannot use the W code to mix incremental and absolute programming in the same block.

### Note:

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

`G90X1V1`

will not produce the expected motions.

## Comment Codes

The Control Program allows you to add *comments* to your NC blocks. The Control Program recognizes two comment codes:

- u A semicolon “;”
- u An open parenthesis “(“

These two comment codes are equivalent. The use of either of these codes within an NC block indicates that a comment follows.

Comments must follow all other NC codes in the block. Comments are ignored when the part program is executed. Comments can be placed on a block without any NC codes to document what is occurring within a program. NC programmers use these comments to annotate their programs.

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

```
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 and the M06 Tool Change codes to display messages to the operator during program execution. Here is an example of an NC block with a pause coded comment:

```
G05(ROUGH DIAMETER SHOULD BE 0.5 in.)
```

When the program pauses, the comment is displayed on the message bar, telling the operator to verify the diameter of the workpiece before continuing. The M105 code provides a more versatile and powerful message facility.

The Control Program can strip the comments from a program with a single command; however, comments cannot subsequently be replaced automatically.

# General Programming Suggestions

The following rules should be followed when writing NC part programs.

- u 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, D, L, F, S, T, M, P, ; A different order may cause unpredictable results.
- u 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 all codes. Some of the codes 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.
- u 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.
- u 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.

- u The first instruction in a part program should move the tool to the starting position. This makes restarts much easier.
- u 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.
- u 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.
- u 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 Router to see if the tool movements are logical.
- u The first portion of a part program should turn on the spindle and establish the feed rate and spindle speed.
- u M codes should be placed on separate blocks to avoid confusion over whether an M code is activated during or after a motion command.
- u Double-check all program blocks against your coding sheet to locate and correct typographical errors.





# More CNC Programming

**Linear Interpolation Programming**

**Circular Interpolation Programming**

**Rapid Traverse Programming**

**Canned Cycle Programming**

**Subprogram Programming**

# Linear Interpolation Programming

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

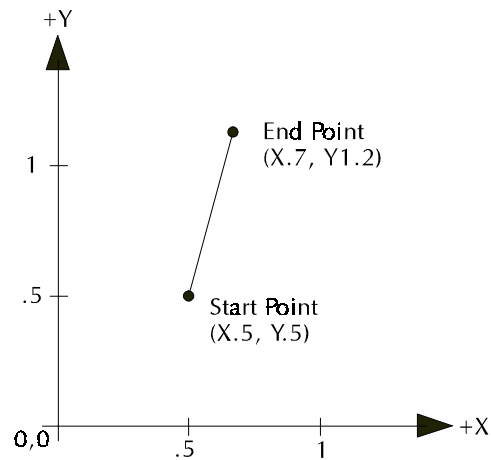
```
N5G90G01X.7Y1.2F2
```

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:

Typical tool movement using linear interpolation



An equivalent movement is achieved with incremental dimensioning (G91):

```
N5G91G01X.2Y.7F2
```

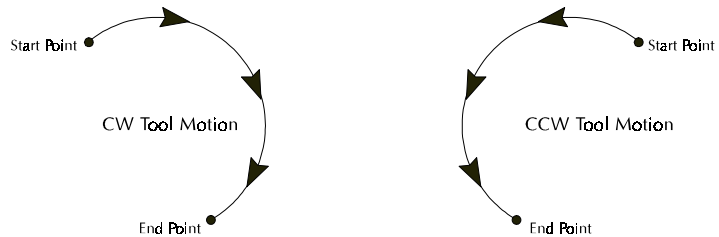
# Circular Interpolation Programming

Circular interpolation moves the cutting tool along an arc 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:

G02 (CW) and G03 (CCW) cutting paths



```
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:

**Note:**

This block is written for incremental arc center mode.

**N10** The block sequence number is 10

**G03** The tool will proceed in a counterclockwise direction from the starting point to specified (X, Y) coordinates; 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

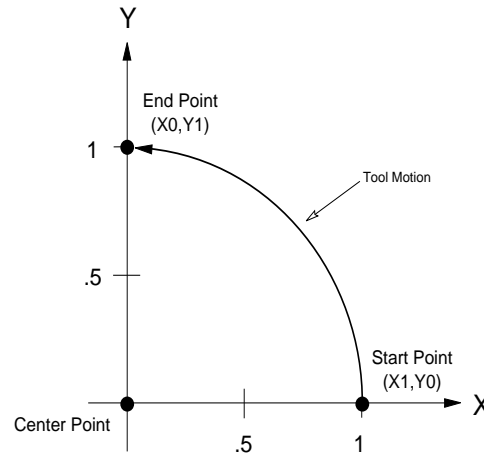
**I-1** I coordinate of center point of arc = -1 (relative to start point)

**J0** J coordinate of center point of arc = 0 (relative to start point)

**F2** Feed rate is 2 inches per minute

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

Typical tool movement using circular interpolation



An equivalent movement is 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
```

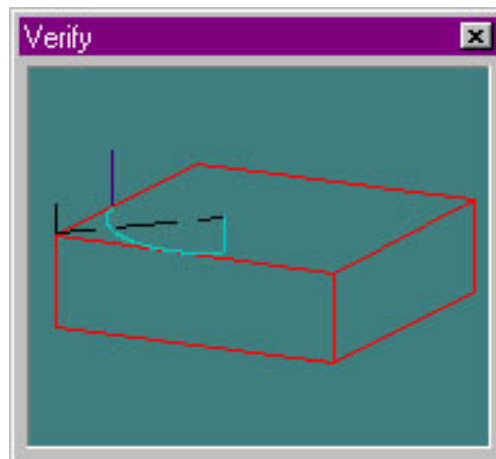
This block would cause the Z axis to move at a constant feed to Z1 while the X and Y axes move in a circular path, resulting in a helical motion. Helical interpolation works with circular motion on the X,Z and Y,Z planes as well.

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

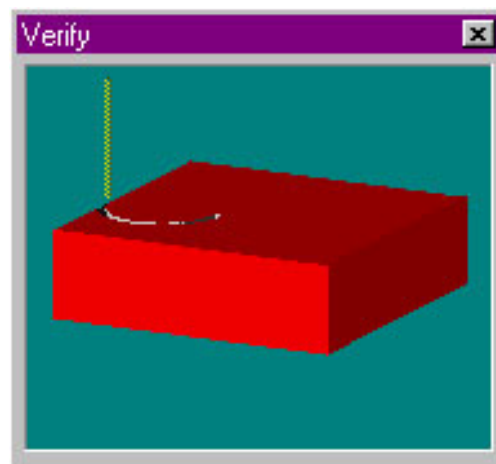
This program uses incremental arc centers. Include the % code if you have changed the Control Program default to absolute.

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

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



Centerline View



Solid View

## Rapid Traverse Programming

On the Benchtop Router, the rapid traverse code (G00) can move the tool at the maximum available feed rate (50 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 TO X=4 AT 2 IPM**

---

## Canned Cycle Programming

Canned cycle commands allow you to perform drilling operations by specifying just a few codes. They are typically used for repetitive operations to reduce the amount of data required in an NC program. Canned cycle codes are retained until superseded in the program by another canned cycle code. The supported canned cycles codes are:

- G80** Canned cycle cancel
- G81** Straight drilling
- G82** Straight drilling with dwell at bottom
- G83** Peck drilling
- 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 codes:

- G98** Rapid to initial position after canned cycle complete; this is the system default
- G99** Rapid to point R after canned cycle complete
- Q** 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** Used for specifying a starting reference point for peck drilling. The point can be at the material surface 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, G01, G02, or G03.

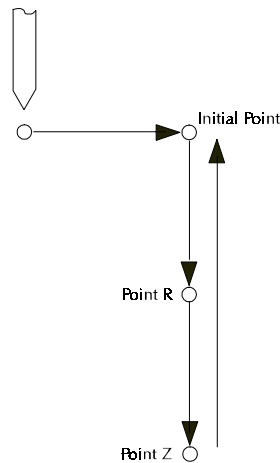
## Using G81

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:

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



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

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



---

## 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.

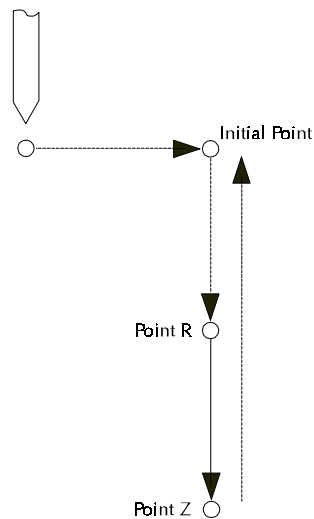
```
G0X1Y1Z.1;RAPID TO 1, 1, .1
```

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

```
G80;CANCEL CANNED CYCLE
```

```
M2;END PROGRAM
```

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



---

## Using G83

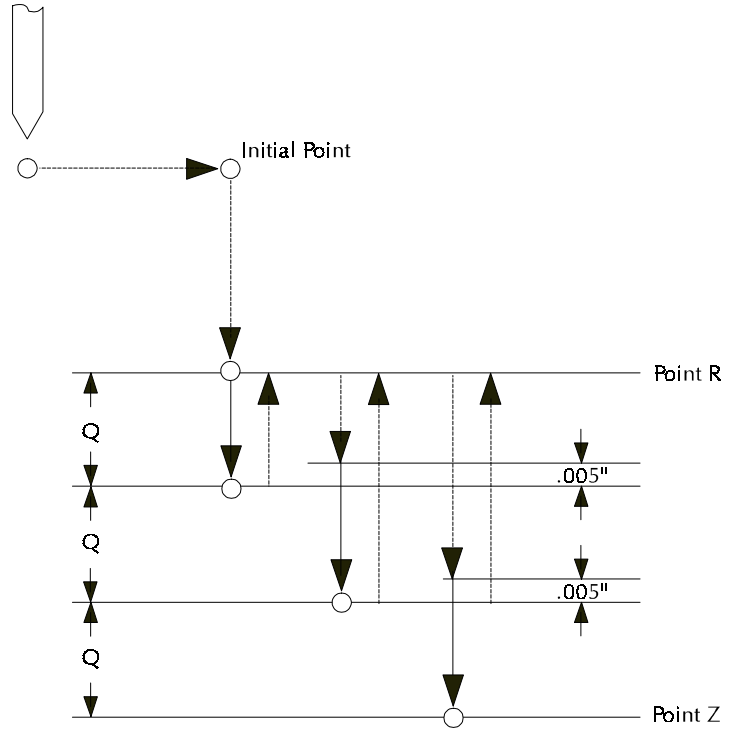
The G83 code is used for peck drilling cycles. By adding a Q depth to the code block, you can specify drilling increments. For instance, the following code will peck drill to a depth of -.5 in .1 increments. The tool will rapid back to point R after each peck drill. Also, before each peck the tool will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

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.



## 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.

```
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.

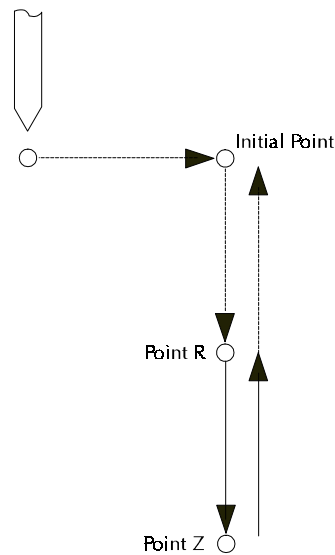
```
G0X1Y1Z.1;RAPID TO 1, 1, .1
```

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

```
G80;CANCEL CANNED CYCLE
```

```
M2;END PROGRAM
```

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



---

## 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
```

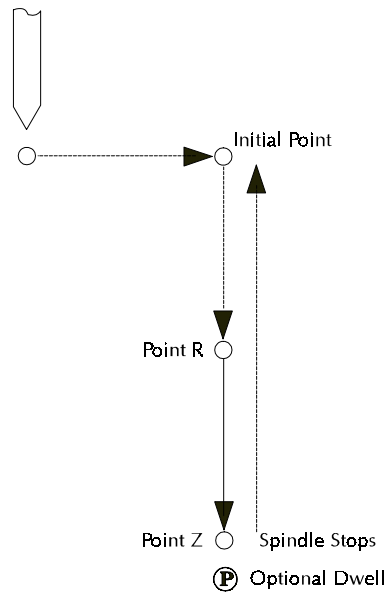
```
G86G98Z-.5R0P5F2;DRILL TO DEPTH OF -.5, SHUT OFF SPINDLE, RAPID TO INITIAL POINT AFTER A DWELL OF FIVE SECONDS
```

```
G80;CANCEL CANNED CYCLE
```

```
M2;END PROGRAM
```

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.

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



## Using G89

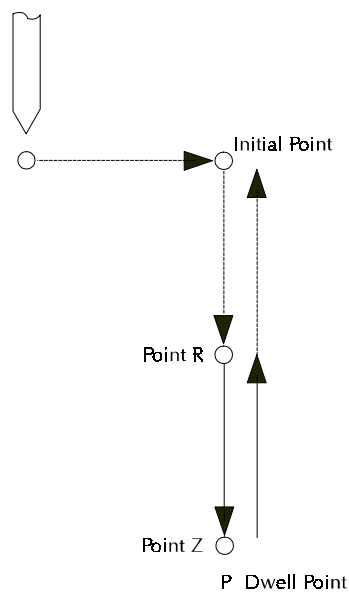
The G89 code works like the G85 except it utilizes a dwell at the bottom of the hole.

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 Benchtop Router 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 subprogramming. The computer executes the subprogram as many times as defined by the L code. For instance, if the code is L5, the subprogram is executed five times. (Optional)

**O Code** The O code replaces the N code on the first 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.

Subprogram calls can also be *nested* within other subprograms. This means that while a subprogram is being executed, it can call another subprogram. The default number of levels that subprograms can be nested is 20 levels deep. You can change the default by using the Setup Program (click on the Setup icon in the BenchRouter program group).

### 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 program ends.

### 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.

## 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 FOUR 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

---

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

# Advanced CNC Programming

**Using Polar Programming**

**Using the Homing Commands**

**Using Cutter Compensation**

**Using Scaling and Rotation Codes**

**Multiple Tool Programming**

**Understanding Coordinate Systems**

**Using Tool Length Offset Codes**

**Using Tool Offset Adjust Codes**

## Using Polar Programming

Using polar coordinates allows you to specify a radius and an angle by specifying a G16 code (polar programming on), then X and 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). If you specify G91 and the radius (X code), the center point is the current point. If you specify only the angle (Y code), the center point is the current center.



---

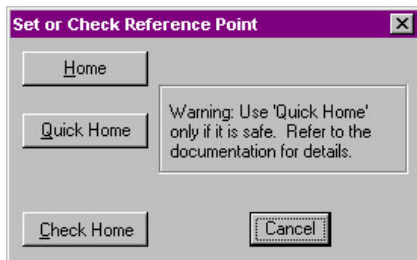
## Using the Homing Commands

The Homing commands allow you to establish a point of origin at the ends of travel on the Benchtop Router. The Benchtop Router uses this point as a reference for all machine coordinate movements. This allows the machine to move consistently to the same location.

---

## Using G28

The G28 code “homes” the machine: it moves the Benchtop Router’s head 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 negative limits of the X and Y axes, and the positive limit of the Z 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.



G28 sets a machine reference point, similar to the Set/Check Home command from the Setup Menu.

## 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 0,0,0
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 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 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 can not use Soft Limits until you have set the machine to the reference point using the G28 code or pressing the Home button in the Set or Check Reference Point dialog box (access using the Set/Check Home command under the Setup Menu).

---

## Using G27

After you have set a reference point (using either the Set Reference Point button in the Setup Coordinate System dialog box or a G28 code), you can use the G27 code to check the actual 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 compares the current position to the one set by the G28. If the deviation is larger than the threshold you specify using the L code (i.e. G27L.001), the Control Program reports an error 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.

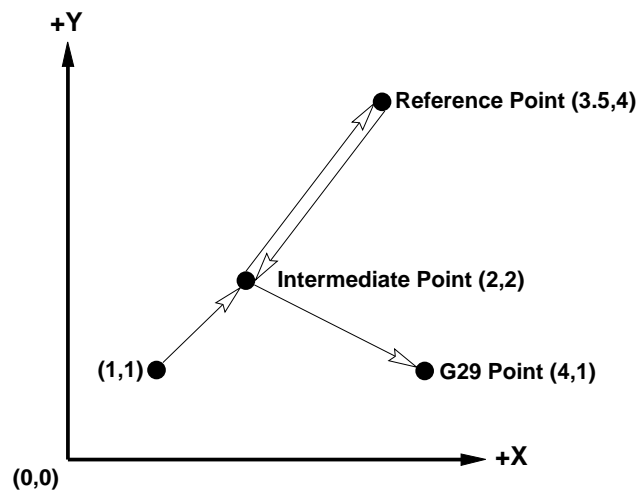
## Using G29

The G29 code moves the tool at a rapid traverse rate to a coordinate 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.

```
N1G28X2Y2Z1; INTERMEDIATE POINT THEN HOME
N2G29X4Y1Z1; GO TO G29 POINT
```

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 point specified with the G29 code.

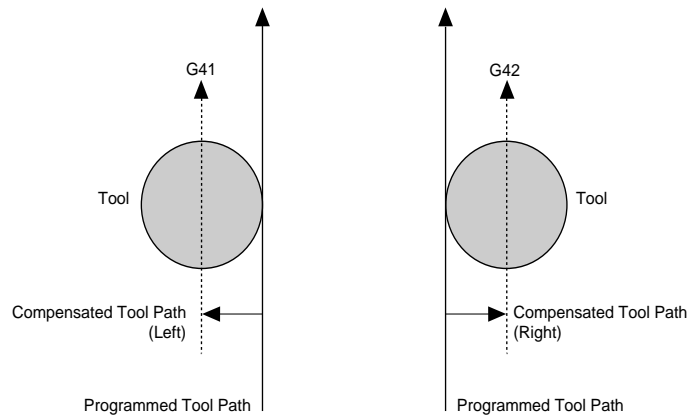


## Using Cutter Compensation

Cutter compensation automatically adjusts the Benchtop Router to compensate for variations in a cutting tool's radius. It uses values from the Offset Table 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.



---

### 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.

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.

## Starting Cutter Compensation (G41/G42)

Cutter compensation mode begins when the following are met:

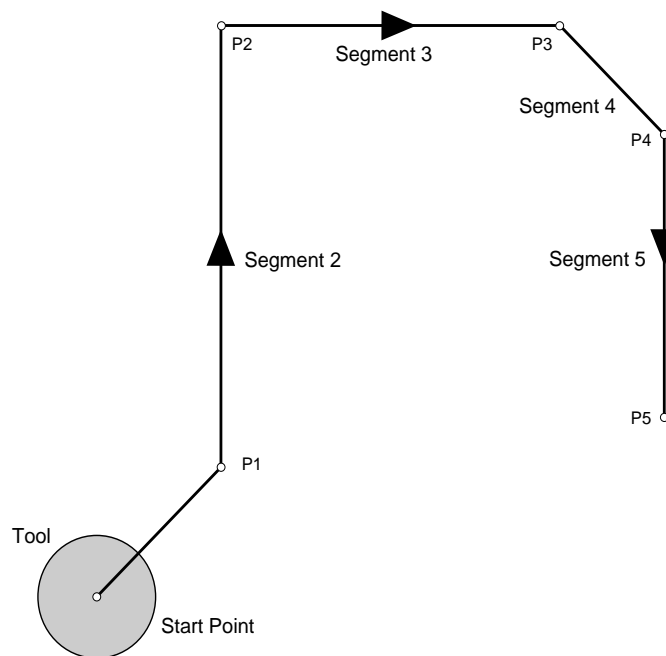
- ◆ A G41 or G42 code is commanded.
- ◆ The specified offset number is not 0.
- ◆ A move in any of the axes in the offset plane is commanded. The move is not 0.
- ◆ No arc commands are commanded in the start up block.

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:

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' 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.

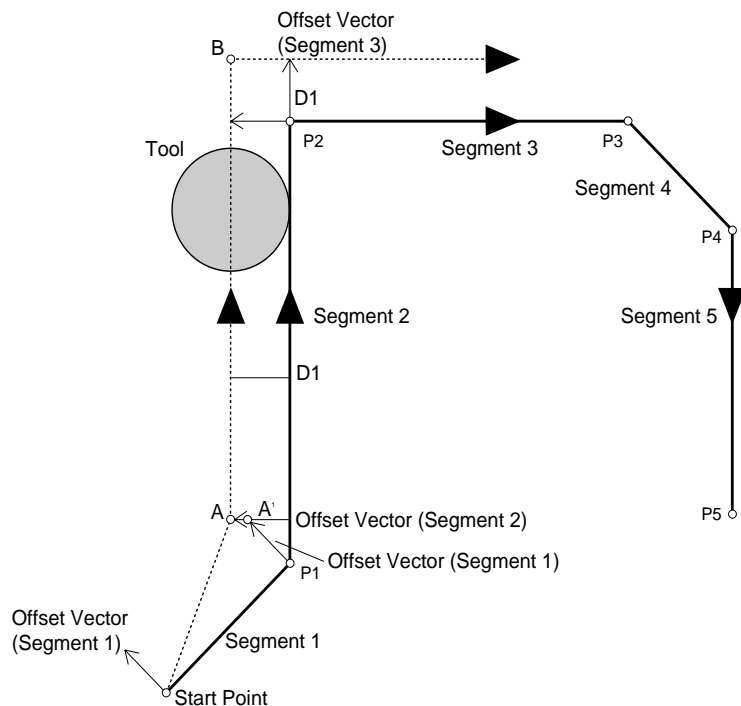
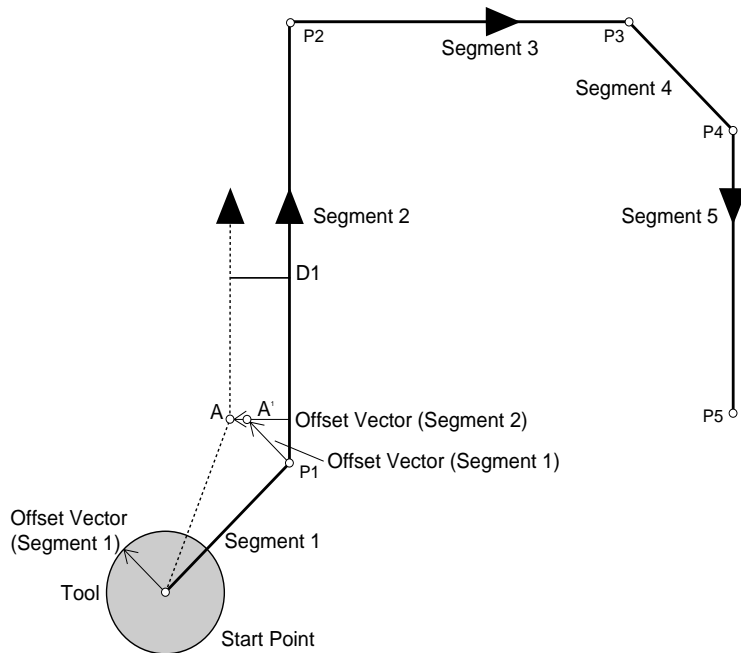
**Note:**

*Cutter Compensation Startup now moves to the vector normal (perpendicular) to the start point of the first segment specified after startup, rather than calculating the intersection between the startup segment and the first complete segment. The diagrams shown do not reflect this change.*

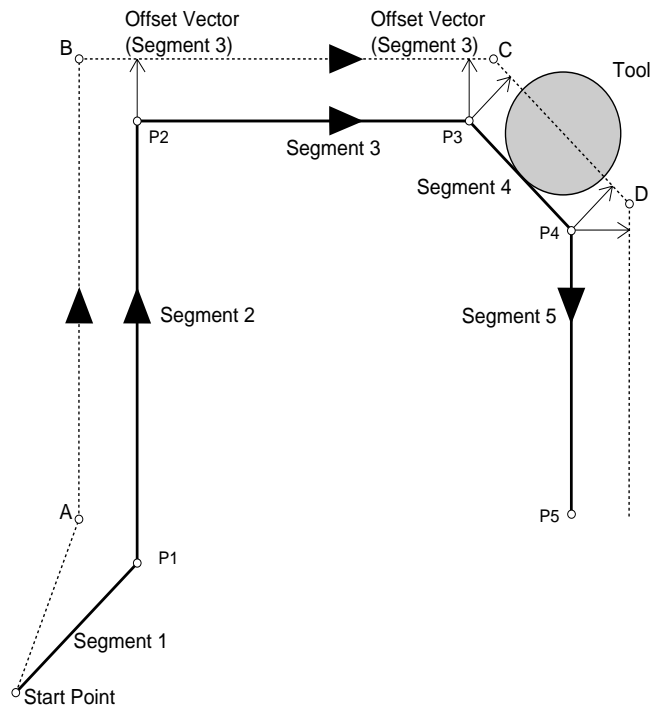
Before beginning each motion, the Control Program looks ahead to the next motion in the NC code to determine the compensated end point of the first motion.

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.

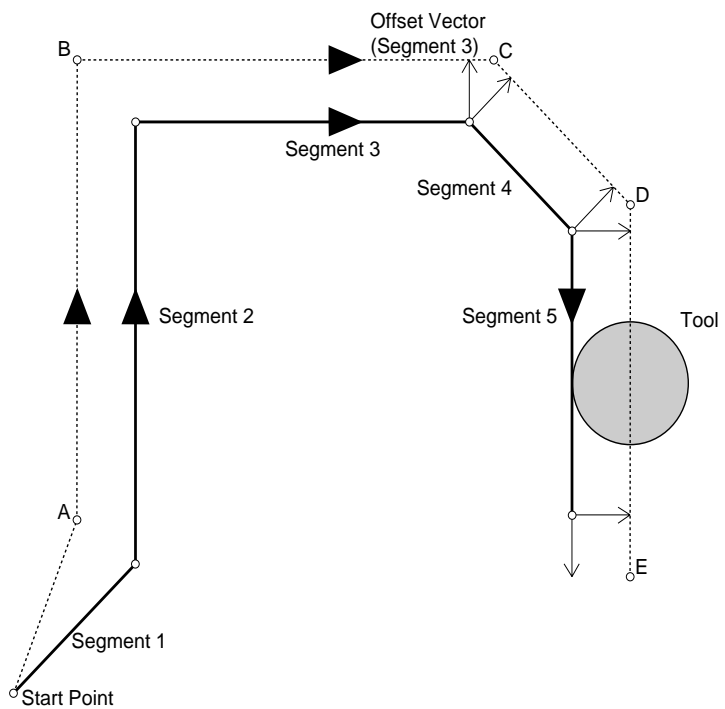
3. The tool moves to the end point of segment 2 (point B) on the compensated tool path. Point B is the intersection of lines drawn perpendicular to the Offset Vectors of segments 2 and 3 with an offset value of D from the programmed tool path.



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.)



## 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

**Note:**

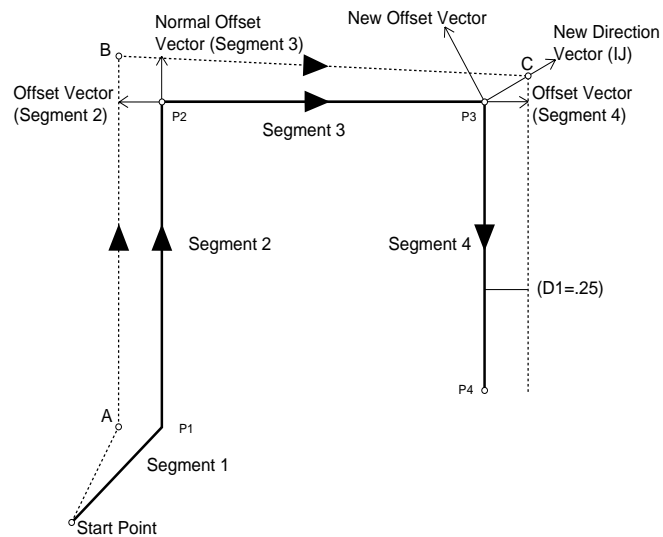
When specifying an IJK vector, you must include the G41 or G42 code on the same line.

G41X.75I1J1

instead of 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 $\Leftrightarrow$ I3J6. By default, the end direction vector is tangent to the segment.



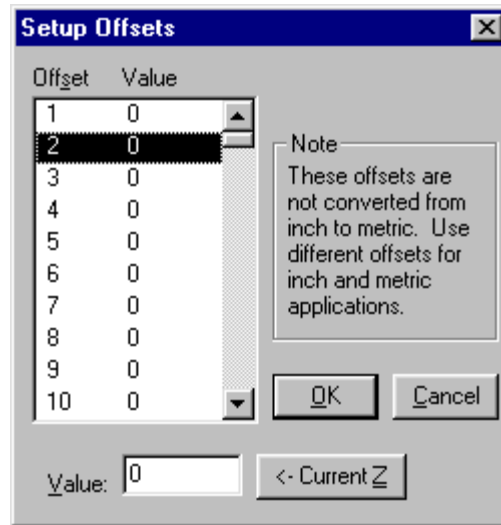
## Setting Cutter Compensation Offsets (D)

Select Setup Offsets from the Setup Menu to set the values referenced by the D codes (the cutter compensation offset value). The following dialog box appears:

Select an offset from the list to modify. Change its value in the Value Edit box, or use the Current Z button to use the current Z position.

### Note:

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



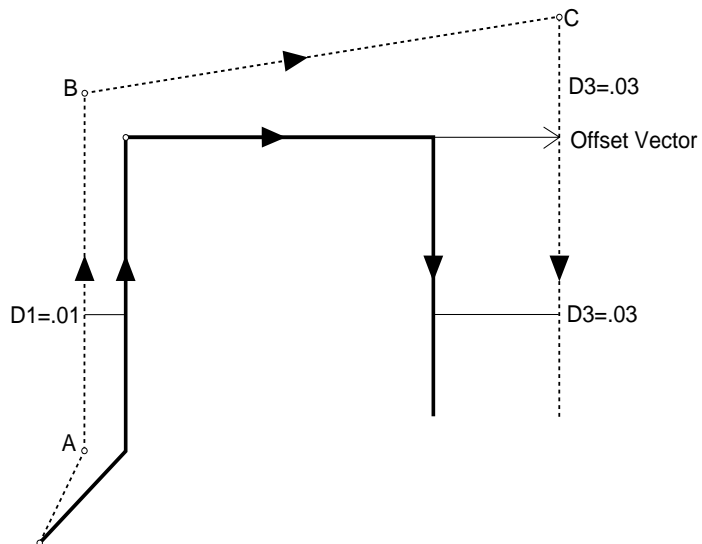
## Changing Offset Values

While cutter compensation is active, you can change offsets by specifying a new offset number with the D-code. For example:

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

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

In this example: D1=.01 and  
D3=.03

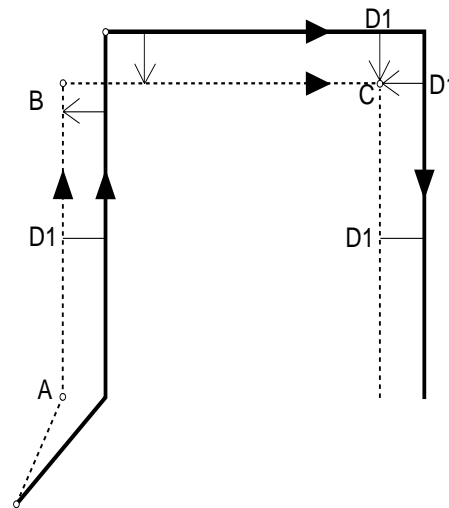


## Changing Offset Sides

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 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.

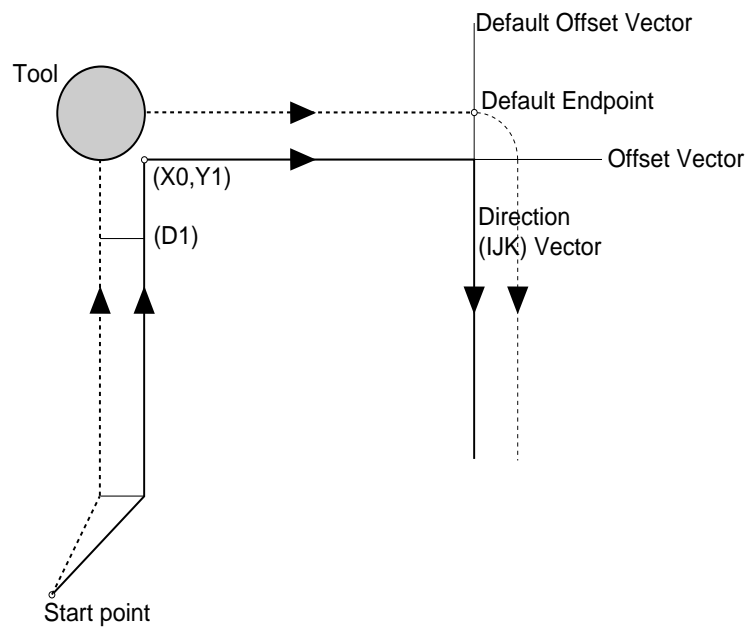


The same situation would occur if you made the offset value negative. For the example above, changing the D value from .01 to -.01 would produce the same result as changing from G41 to G42.

## 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 methods for cancelling cutter compensation.

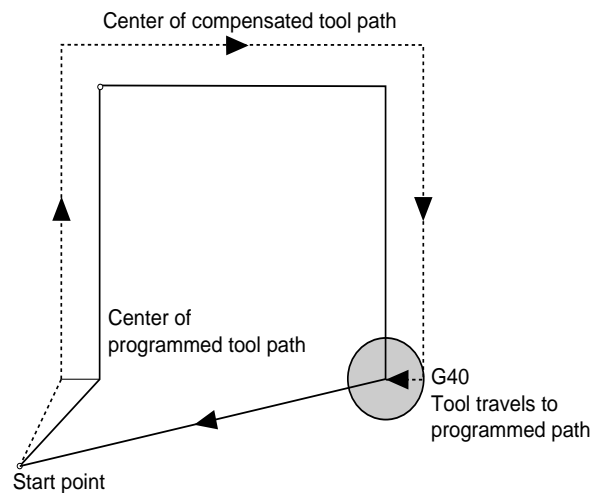
1. G40
2. G40XYZ
3. G40XYZIJK
4. D0
5. D0XYZ
6. G41/42D0XYZIJK

With methods 4 through 6 above, setting the offset number to zero has the same effect as cancelling cutter compensation. However, cutter compensation is still active.

### Methods 1 and 4

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

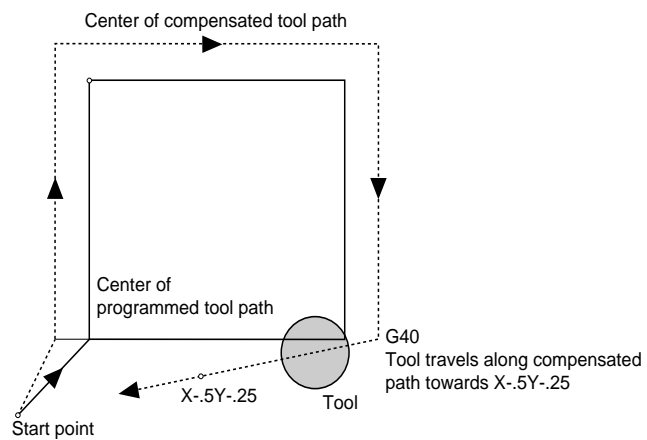
```
G91G41D1
...
X.25
Y-.25
Z.2; RETRACT
G40; OR D0
X-.5Y-.25
M2
```



## Methods 2 and 5

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

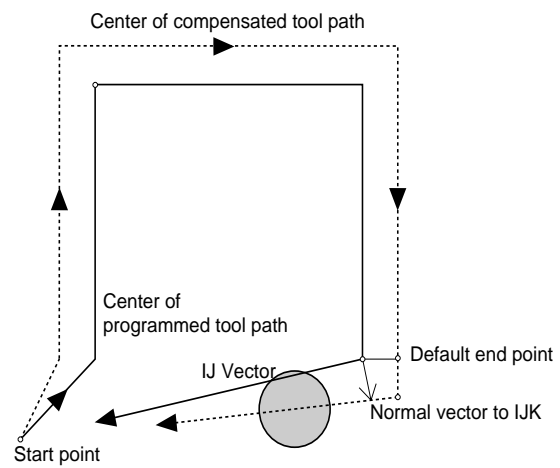
```
G91G41D1
...
X.25
Y-.25
Z.2;RETRACT
G40X-.5Y-.25
M2
```



## Methods 3 and 6

With these methods an IJK vector specifies the direction of movement after cutter compensation is cancelled.

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



# 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

---

### CAUTION

---

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

## 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.

The NC program below creates a half-circle then scales those motions by two to create a larger, uniformly-scaled half-circle (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
```

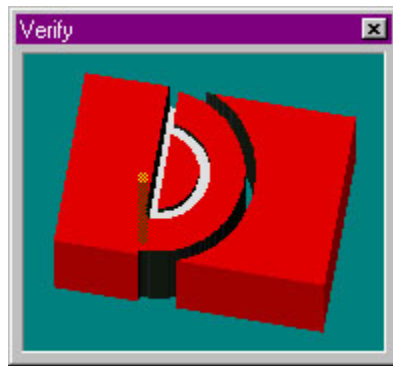
#### Note:

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

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.



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



## Scaling Each Axis

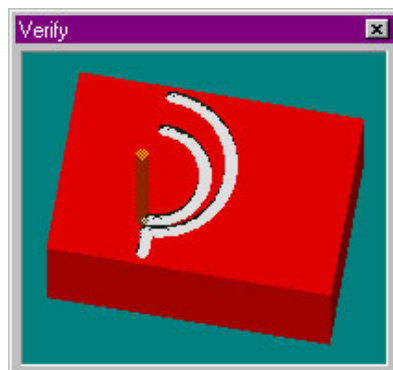
You can scale each axis by different magnifications. The following NC program uses scaling to change the proportions of a motion by scaling each axis separately (the scaling codes are in bold print):

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

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.



## Creating Mirror Images with Scaling

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
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. 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 with a negative Z scaling factor.

---

### CAUTION

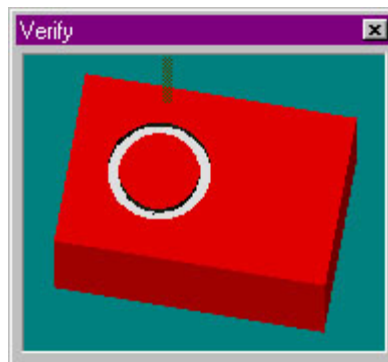
---

Performing Z axis mirroring is an advanced operation. Use extreme caution when machining with a negative Z scaling factor.

#### Note:

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

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



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

**G68** Invokes rotation

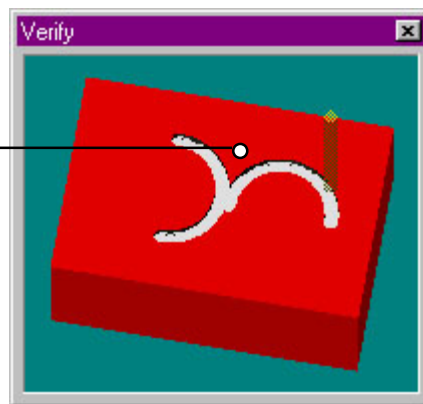
**G69** Cancels rotation

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
```

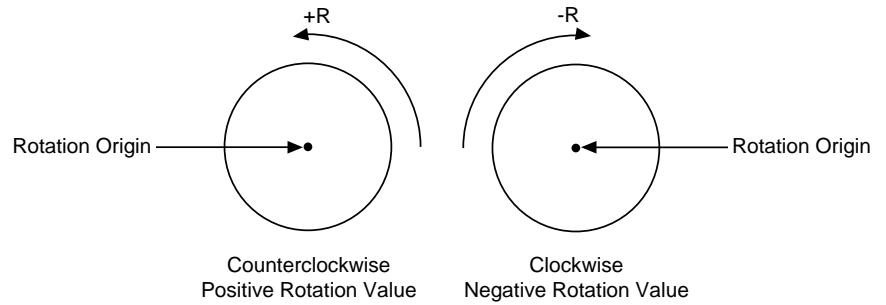
In this example, G68 rotates a shape 90° from the original.

Rotation Origin  
(1.583, 1.5)



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

### IMPORTANT!

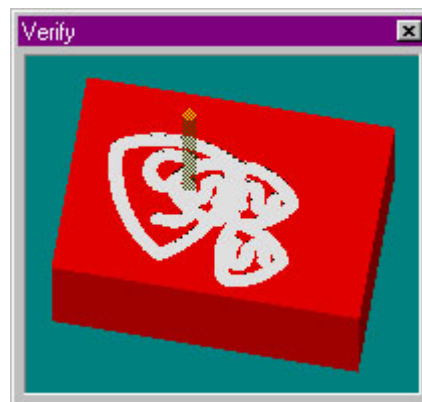
When combining the scale and rotate features, always scale the part first, before rotating it.

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 combine scaling and rotation in the same NC program. The portion of the NC program below combines the scaling and rotation codes necessary to machine the following part (the actual subprogram to cut the shield (O100) is not listed here).

```
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
```

In this example, scaling and rotation are combined to produce different sized shapes that are scaled and rotated from the original.



---

## Multiple Tool Programming

The Benchtop Router allows you to define up to 20 different tools for use during router operations.

You will need to define tools and write the tool changes into the NC program. For defining tool length, the easiest method is to use the Setup Tool Wizard under the Tools menu.

Defining tools is explained in the tutorial in Section D of this guide. Programming codes, reference tools, offsets, and testing are described below.

---

## Using Multiple Tool Codes

The T code is used in the NC program to offset the cutter so that the NC program becomes independent of the cutter length, which is set up in Tool Definitions. This means you can replace a worn tool with a tool of a different length without changing the NC program, just by entering new offsets. (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.

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 router 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 will turn the spindle back on and 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 workpiece, or on a gauge, where you will jog the tip of each tool).
2. Install a tool into the spindle.
3. From the Tools Menu, choose Select Tool. If the tool is not already defined in the pull down menu, see page D-8 to define the tool.
4. Choose the tool you are using, then click Select Tool.
5. If you are using a sensor, place it on the hold down table.
6. Using the Jog Control Panel, jog tool #1 to the tip of the sensor until the lights come on.
7. Select Set Position from the Setup Menu, set the Z axis value to zero, and click OK.

Tool #1 is now established as the reference tool.

---

## Establishing Tool Offsets

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

To set the offset for Tool #2:

1. Install tool #2 in the spindle.
2. From the Tools Menu, choose Select Tool. If the tool is not already defined in the pull down menu, see Section D to define the tool.
3. Choose the tool you are using, then click Select Tool.
4. Jog tool #2 to the tip of the sensor until the sensor lights come on.
5. Select Setup, and then Offsets from the pull down menu.
6. Choose the correct tool (the description that matches the tool you installed) from the list, and click on the Current Z button at the bottom of the window. Select OK to save the value.
7. Repeat this process for each additional tool.

## Testing Your Multiple Tool Program

After setting all of the tool offsets, test run your program without a workpiece mounted and with the spindle speed turned down.

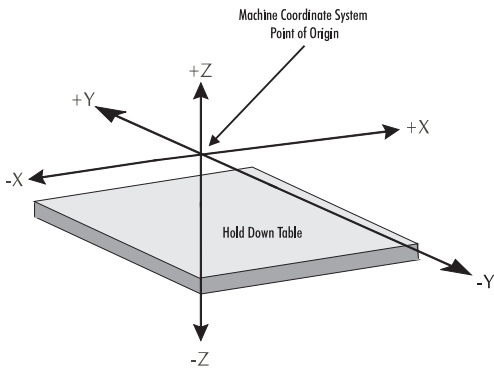
1. After installing Tool #1, close the safety shield, put on your safety glasses, and complete the safety checklist.
2. Select the Run/Continue command from the Program Menu. Enter zero as the start line and select Start. Throughout the test, be prepared to press the emergency stop switch on the Benchtop Router, or the space bar on the computer keyboard in case of a tool crash. The computer will run the program until it reaches the M06 code. If your machine is equipped with an ATC, the tool change will take place automatically, and you may ignore steps 3-7. The M06 stops and retracts the spindle.
3. When the spindle has completely stopped and the Pause message appears on the screen, push in the emergency stop button on the Benchtop Router.
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 button. 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 workpiece, set the spindle speed, and run your multiple tool program.



# Understanding Coordinate Systems

For a beginning user, understanding coordinate systems can be difficult. The first thing to remember is, before performing most machining operations, you are required to set the machine to “home” position. This returns the machine to the machine zero point, and acts as a reference point for all operations. It is a good idea to home the machine every time you run the control software.

Set/Check Home, or G28



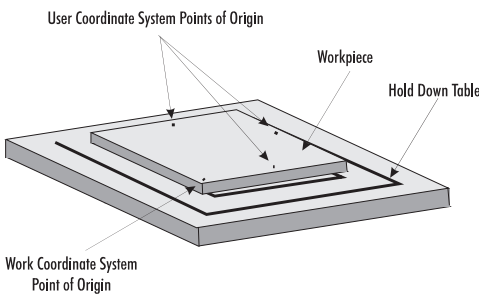
## Machine Coordinates

Machine Zero positions the router head at the extreme negative end of travel on the X and Y axis, and the extreme positive end of travel on the Z axis. This is a fixed point on the machine, and cannot be changed. The machine uses this as a reference point for all operations. If the machine is not homed (set to the machine zero) it can not accurately locate the workpiece on the hold down table. The machine is homed by selecting Set/Check Home under the Setup menu, and choosing the Home button; or by using a G28 code in the NC program.

## Work Coordinates

Once home is set, you load an NC program into the computer. Your program will need a point of origin to start from. Setting a point of origin will establish the work coordinates. Work coordinates relate to the workpiece, and are usually set from the top of the front left corner of the workpiece mounted on the table. Once the stock is mounted on the table, jog the spindle to the top front left corner of the workpiece. From the Setup Menu, select Set Position. The tool coordinates appear in the dialog box. Enter 0,0,0 and click on OK to set the current tool position as the point of origin on the workpiece.

Setting Coordinate Systems



## Multiple Coordinate Systems

For more advanced operations, you can set up multiple coordinate systems. For example, three or four designs can be cut from the same piece of stock. First, secure the stock to the hold down table. Then, set the work coordinate point of origin (0,0,0) at the corner of the stock using the Set Position dialog box from the Setup Menu. After setting the corner of the stock as the origin, select Coordinate Systems from the Setup menu. The Setup Coordinate Systems dialog box appears. Select a “User CS”, enter the coordinates for the first design, and click on Apply. Repeat this procedure for as many User CS’ as necessary. (For a detailed description of the Setup Coordinate Systems procedure, see Section E.)

## Using Tool Length Offset Codes

Tool length offsets are included in the Tool Setup procedure in Section E. When you command a tool change in the program and the tool is loaded, the offset is automatically read as part of the tool information that was entered when the tool was defined.

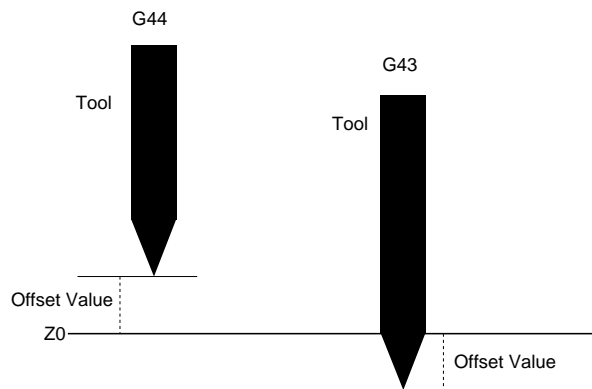
Fanuc mode is entered by placing a % at the beginning of your program. This does not affect how the software interprets T codes.

However, if you are programming in Fanuc-compatible mode, 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 examples on this page assume positive offset values.

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.



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 (for verify only), 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:

```
M06G43T2H1
```

The M06 initiates the tool change cycle. Once the tool change is complete, the G43 adds the value from the offset table to the Z offset. 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 codes, you can increase or decrease the movement distance of the specified axis by the offset value (D). The tool offset adjust codes are:

- G45** Increases the movement amount by the value stored in the offset table
- G46** Decreases the movement amount by the value stored in the offset table
- G47** Increases the movement amount by twice the value stored in the offset table
- G48** Decreases the movement amount by twice the value stored in the offset table
- D** Offset value

You must command a motion for tool offset adjust codes to adjust the offset values. Following are examples of the motions caused by tool offset adjust codes.

### G45

1. The offset value increases the actual movement by 1.5 beyond the commanded movement.

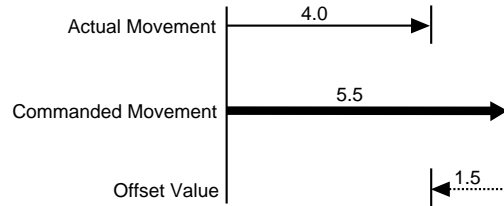


2. The negative offset value decreases the actual movement by 1.5 from the commanded movement. Note that a G45 code with a negative offset value is the same as a G46 code with a positive value.



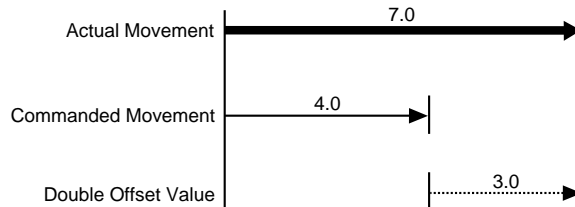
## G46

The G46 code decreases the commanded movement by the offset value. Note that the G46 code is the same as reversing the positive or negative direction specified by the offset value of a G45 code.



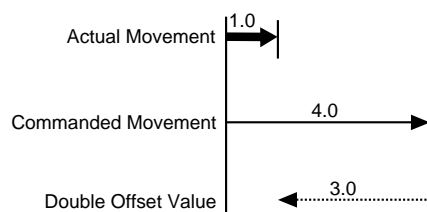
## G47

The G47 code increases the commanded movement by twice the offset value.



## G48

The G48 code decreases the commanded movement by twice the offset value. Note that the G46 code is the same as reversing the positive or negative direction specified by the offset value of a G47 code.



# Safe Router Operation

**Safety Rules**

**Emergency Stops**

**Safety Checklist**

**Lista de Seguridad**

---

## Safety Rules

The following safety rules should be reviewed and practiced by all operators of the Benchtop Router.

### 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 Router. 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 Router and keep it readily accessible for quick reference. Know the intended applications and limitations of the Router as well as its hazards.

### Ground All Tools

The Router 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.

### Remove Adjusting Keys and Wrenches

Make it a habit to check that keys and adjusting wrenches are removed from the Router 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 Router in damp or wet locations. Never operate electrical equipment in the presence of volatile and flammable petroleum-based solvents and lubricants.

Feel free to copy these rules, or the Safety Checklist, and post them in your work area for quick reference.

## **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 Router**

Lock and remove the key from the Router 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 to do.

## **Dress Appropriately**

Don't wear loose clothing or jewelry which can get caught in moving parts. Wear a hat, a hairnet, 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 router motor.

## **Do Not Overreach**

Keep your footing and balance at all times so you won't fall into or grab the moving machine.

## **Maintain Cutting Tools In Top Condition**

Keep cutting tools sharp and clean. Lubricate and clean Router components on a regular basis.

## **Disconnect Tools Before Servicing**

Always use the emergency stop switch to disconnect power and disable the motor before mounting or removing the workpiece, or changing tools. *Do not* rely solely on a programmed Pause command to disable Router operation.

## **Avoid Accidental Starting**

Make sure the power switch on the control box is off before plugging in the Router power cord.

## **Use Recommended Accessories**

To avoid stressing the Router and creating a hazardous machining environment, use only those accessories designed for use with the Benchtop Router, available through Light Machines Corporation.

## **Tighten All Holding, Locking and Driving Devices**

Tighten the collet. Do not over-tighten tool holding devices. Over-tightening may damage threads or warp parts, thereby reducing accuracy and effectiveness.

## **Do Not Operate the Machine Under the Influence of Alcohol or Drugs**

Alcohol or drugs may impair your judgement and reaction time, which could contribute to an on-the-job accident.

## **Avoid Distractions While Running the Machine**

Use simple common sense and pay attention while operating any piece of machinery.

---

## **Emergency Stops**

All operators must be fully aware of how to shut down the machine quickly, should the need arise.

- ◆ In an emergency, you should always use the red Emergency Stop button on the Router.
- ◆ You can also stop it by pressing the Control and Space keys on the computer keyboard, or by clicking on the stop buttons on the screen with the mouse. In an emergency, always use the Emergency Stop button on the machine.



---

## IMPORTANT!

---

Post copies of this checklist in the work area. Verify that all items are checked-off prior to each operation of the Benchtop Router.

# 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 área de trabajo.  
Verifique que todos los puntos estén  
checados antes de cada puesta en  
marcha de la máquina.

## Lista de Seguridad

### Antes de entrar en el área de trabajo:

- Use sus lentes de seguridad.
- Procure recogerse el cabello y no usar ropa floja.
- No use joyería como: anillos, pulseras y relojes.

### Antes de trabajar a máquina una pieza:

- Utilice 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 trabajo a la mesa. Quite las herramientas y llaves antes de cerrar la guarda de seguridad.
- Asegurese que todos los contactos de corriente A.C. estén 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 trabajo a toda persona no autorizada.

# G and M Codes

**G Codes by Group**

**M Codes by Group**

# G Codes by Group

## Interpolation Group

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

## Programming Mode Group

|     |                                                                                                                                |
|-----|--------------------------------------------------------------------------------------------------------------------------------|
| G90 | Absolute coordinate programming (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

|     |                                                                                                          |
|-----|----------------------------------------------------------------------------------------------------------|
| G20 | (Fanuc) Inch: Used to instruct the mill that inches are the unit of measure for the part program         |
| G21 | (Fanuc) Metric: Used to instruct the mill that millimeters are the unit of measure for the part program. |
| G70 | Inch: Used to instruct the mill that inches are the unit of measure for the part program.                |
| G71 | Metric: Used to instruct the mill that millimeters are the unit of measure for the part program.         |

## 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 Section K).                                          |
| G26 | Wait for robot input to be low: Used in conjunction with H code, which specifies input number. Used for robot synchronization (see Section K).                                           |
| G31 | Linear to specified coordinate. Stop short if specified input goes High (if H is positive) or Low (if H is negative).                                                                    |
| G15 | Polar programming cancel.                                                                                                                                                                |

|                                |     |                                                                                                                                                                                                              |
|--------------------------------|-----|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>Polar Programming Group</b> | G16 | Begin polar programming.                                                                                                                                                                                     |
| <b>Coordinate System Group</b> | G53 | Rapid traverse to specified coordinates in the Machine Coordinate System. (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.                                                                                                                                                                                   |
| <b>Canned Cycle Group</b>      | 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.                                                                                                                                                                              |
| <b>Preset Position Group</b>   | G27 | Check reference point: This code moves the tool to its home position on the Machining Center to check the calibration of the axes. Compares reported position against zero to see if position has been lost. |
|                                | G28 | Set reference point: This code moves the tool and calibrates the axes. The Machine Coordinate System origin is located at this home position.                                                                |
|                                | G29 | Return to reference point: Moves the tool to a coordinate specified by XYZ. Typically used after a G27 or G28 code.                                                                                          |
|                                | G92 | Set position: This code works like the Set Position function under the Setup Menu (see Section E). The X, 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 G).                                                                                                                                 |
|                                | G99 | Rapid move to point R (surface of material or other reference point) after canned cycle complete (Section G).                                                                                                |
|                                | 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 cir-                                              |

|                                  |            |                                                                                                                                                                                                   |
|----------------------------------|------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>Plane Selection Group</b>     |            | cular 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.                                                                             |
|                                  | <b>G18</b> | 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. |
|                                  | <b>G19</b> | 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. |
| <b>Scaling Group</b>             | <b>G50</b> | Cancel scaling.                                                                                                                                                                                   |
|                                  | <b>G51</b> | Invoke scaling. Use this code to scale axes by a single factor, or by independent factors around a fixed origin. The default scale factor is 1.                                                   |
|                                  | <b>G68</b> | Invoke rotation. Use this code to rotate geometry an arbitrary angle around a specified point. Rotation works on any plane, one plane at a time.                                                  |
| <b>Rotation Group</b>            | <b>G69</b> | Cancel rotation.                                                                                                                                                                                  |
| <b>Cutter Compensation Group</b> | <b>G39</b> | Corner offset in circular interpolation.                                                                                                                                                          |
|                                  | <b>G40</b> | Cancel cutter compensation.                                                                                                                                                                       |
|                                  | <b>G41</b> | Invoke cutter compensation left.                                                                                                                                                                  |
|                                  | <b>G42</b> | Invoke cutter compensation right.                                                                                                                                                                 |
|                                  | <b>D</b>   | Specifies the offset number from the Offset Table.                                                                                                                                                |
|                                  | <b>G43</b> | Shifts Z axis in a positive direction by a value specified by H.                                                                                                                                  |
| <b>Tool Length Offset Group</b>  | <b>G44</b> | Shifts Z axis in a negative direction by a value specified by H.                                                                                                                                  |
|                                  | <b>G49</b> | Cancel Tool Length Offsets.                                                                                                                                                                       |
|                                  | <b>H</b>   | Specifies the offset number from the Offset Table.                                                                                                                                                |
| <b>Tool Length Adjust Group</b>  | <b>G45</b> | Increases the movement amount by the value of D.                                                                                                                                                  |
|                                  | <b>G46</b> | Decreases the movement amount by the value of D.                                                                                                                                                  |
|                                  | <b>G47</b> | Increases the movement amount by twice the value of D.                                                                                                                                            |
|                                  | <b>G48</b> | Decreases the movement amount by twice the value of D.                                                                                                                                            |
|                                  | <b>D</b>   | Specifies the offset number from the Offset Table.                                                                                                                                                |

## M Codes by Group

### Program Stop/End Group

|     |                                                                                                                                                                                                                                                                                                 |
|-----|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| M00 | <i>Pause:</i> Allows you to place a pause in your code. Acts like a G05 pause.                                                                                                                                                                                                                  |
| M01 | <i>Optional Stop:</i> Allows you to place an optional stop in your code. Place an M01 in the block of code where you would like to pause. With Optional Stop on, the M01 works like a G05. With Optional Stop off, the M01 code is ignored, the other codes on the block are executed as usual. |
| M02 | <i>End of Program:</i> Takes effect after all motion has stopped; turns off drive motors, spindle and accessory outlets.                                                                                                                                                                        |
| M30 | <i>Program stop:</i> Same as M02.                                                                                                                                                                                                                                                               |

### Spindle Group

|     |                                                                                                                                        |
|-----|----------------------------------------------------------------------------------------------------------------------------------------|
| M03 | <i>Spindle Motor On:</i> Activated concurrently with motion specified in the program block; remains in effect until superseded by M05. |
| M05 | <i>Spindle Motor Off:</i> Activated after the motion specified in the program block; remains in effect until superseded by M03.        |

### Tool Change Group

|     |                                                                                                           |
|-----|-----------------------------------------------------------------------------------------------------------|
| M06 | <i>Tool Change:</i> Used in conjunction with a T code to perform multiple tool operations. See Section H. |
|-----|-----------------------------------------------------------------------------------------------------------|

### Accessories On/Off Group

|     |                                                                                                                                                                     |
|-----|---------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| M08 | <i>ACC1 On:</i> Turns on accessory ACC1 outlet concurrently with the motion specified in the program block; remains in effect until superseded by M09.              |
| M09 | <i>ACC1 Off:</i> Turns off accessory ACC1 outlet after the motion specified in the program block; remains in effect until superseded by M08.                        |
| M10 | <i>Clamp ACC2:</i> Turns on ACC2. Closes air vise accessory concurrently with the motion specified in the program block; remains in effect until superseded by M11. |
| M11 | <i>Unclamp ACC2:</i> Turns off ACC2. Opens air vise accessory after the motion specified in the program block; remains in effect until superseded by M10.           |
| M25 | <i>Set robot output:</i> Used for robot synchronization. Used in conjunction with H code to specify output number.                                                  |
| M26 | <i>Set robot output:</i> Used for robot synchronization. Used in conjunction with H code to specify output number.                                                  |

## Program Management Group

- M20** *Chain to Next Program:* This code appears at the end of a part program and is followed on the next line by the file name of another program which is executed when all motion stops. Here's an example of a part program chain to another program:
- ```
N37Z.2  
N38M20  
PROGRAM TWO
```
- If the two programs you are chaining are not in the same directory, you must specify the full pathname for each file. If the specified file is not found, the Open dialog box appears so you can locate it.
- M22** *Output to file:* Outputs information to a file. The first time the Control Program encounters an M22 code, it opens the specified file. You must enclose the name of the file in parentheses for the Control Program to recognize it.
- M47** *Rewind:* Restarts the currently running program; takes effect after all motion comes to a stop. Use the L code to repeat a finite number of times. The L code defines the number of times to run. For example, M47L2 rewinds once, to run the program twice.
- 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 code destination. Goes to first occurrence of N code within the main program. The N code can not follow any subprogram (O code).

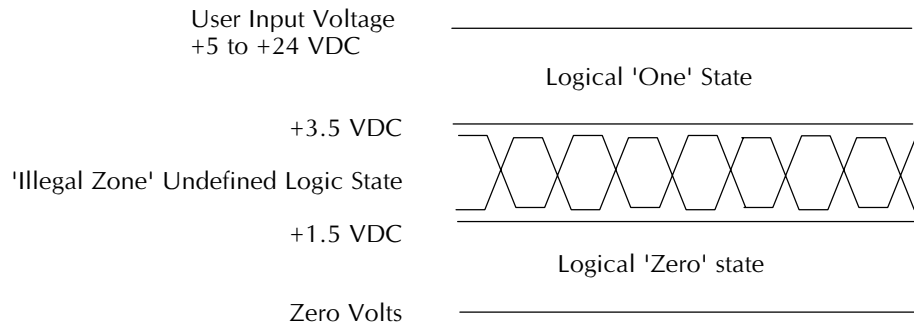
Robotic Integration

Robotic Integration

The Benchtop CNC Router has a simple interface for interacting with common robots, like those used for automatic part loading between machining operations. The machine and the robot communicate by way of an interface connector (labeled I/O) located on the rear panel of the router.

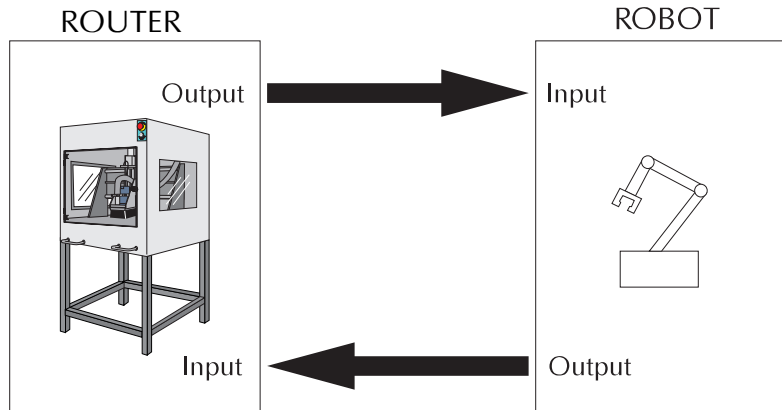
How it Works

The method of communication between the router and robot is very basic. They are both able to transmit and receive *high* or *low* signals. Since communication signals are typically 0 and 5 volts, a high signal is 5 volts (or 3.5 volts or greater); and a low signal is 0 volts (or 1.5 volts or less). See the figure below.



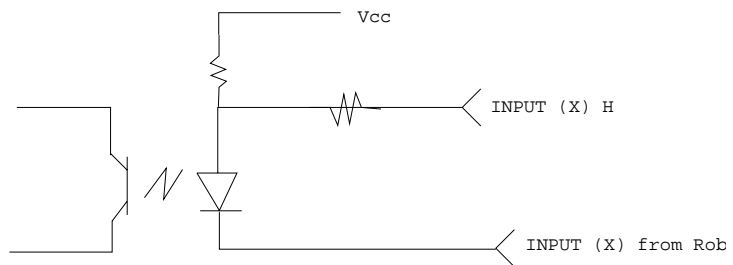
There are NC codes that you can place in a part program to instruct the router to *transmit* a high or low signal to the robot. There are also codes that instruct the router to *wait* for a high or low signal from the robot.

Signals sent out are referred to as *outputs*, while signals coming in are called *inputs*. Any signals that the router transmits to the robot are transmitted through an *output pin* on the interface connector. Outputs can drive a maximum load of 1mA. Any signals that the router receives from the robot come in through an *input pin* on the interface connector.

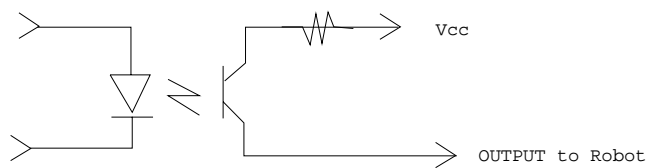


Examples of input and output wiring are shown in the schematic below.

Input Configuration

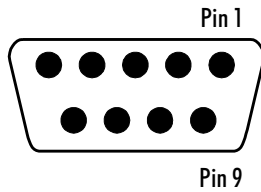


Output Configuration



The Interface Connector

The Robotic Interface Connector



You can check the status of each input and output by using the toolbars in the Control Program.

Notes: Grounding pins 1 and 6 will cause the input to be true.

Inputs 1H and 2H are only used in isolated input mode, and the Controller Box must be specifically configured.

The 9 pin I/O connector on the controller box rear panel provides a number of ways for the router to communicate with robots and other external mechanisms, such as an additional limit switch. Most of the 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 inputs and outputs on the router are designated as follows.

Pin	Function	I/O Levels	Name
1	Input 1	+5V	INP 1L
2	External Limit	0 or +5V	ELim
3	Output 2	+1V to +3.5V	OUTPUT 2
4	Output 1	+1V to +3.5V	OUTPUT1
5	Common		GND
6	Input 2	0 or +5V	INPUT 2
7	Input 1H	See Note	INPUT 1H
8	Input 2H	See Note	INPUT 2H
9	Common		GND

Other Input/Output Notes:

- u All User outputs are disabled by an ESTOP condition.
- u Never apply a negative voltage or an AC voltage to an input.
- u Do not connect to pins 7 or 8. These are for special applications.
- u Outputs are set to source current only. Max load on each output is 1mA.
- u 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.

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 H5)

The H code is used in conjunction with the wait codes and transmit codes. For example, G25H3 tells the router 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 router 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 router is waiting for a high signal. If there's no change to a high signal, the router does not execute the next line of NC code in the program.

When the robot sends a high signal, the router responds by continuing with the NC program. If the router does not respond to the robot as you have programmed it to, check that you have correctly wired the robot to the interface, and that the robot's initial output state was not changed to high during the connection of the robot.

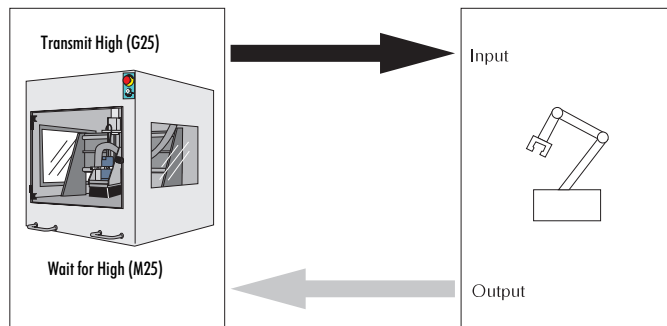
A Sample of Router/Robot Communication

As previously explained, communication between the router and robot is very simple. They only use and recognize high or low signals. Once the router 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 router 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 router.

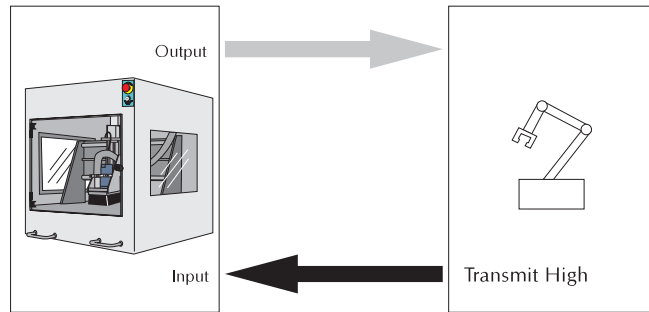
This reciprocal system of communication is made possible by the placement of G and M codes in the part program. The G codes provide the wait instructions to the inputs, while the M codes provide the transmit instructions to the outputs.

The following sequence is typical of communication between the router and a robot. Since the robot's initial output state is low, the routers inputs have already been pulled down to a low state.

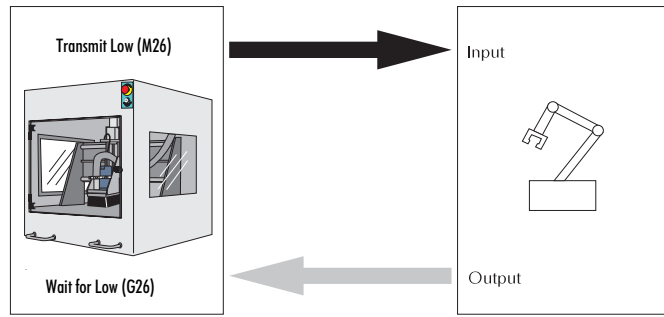
1. The router opens the air vise (M11), transmits a high signal (M25) to the robot, and waits for a high (on) signal (G25).



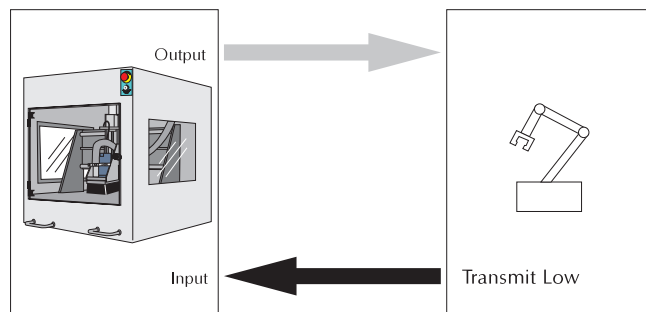
2. The robot places a work piece in the vise and transmits a high signal to the router indicating that it is okay to close the vise.



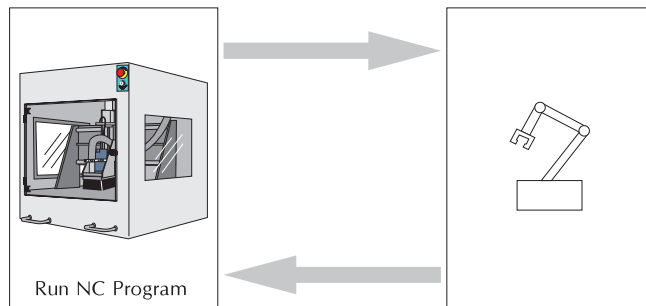
3. The router receives the signal, closes the air vise (M10), then signals (M26) the robot that the vise is closed. The router then waits (G26) for a low signal from the robot.



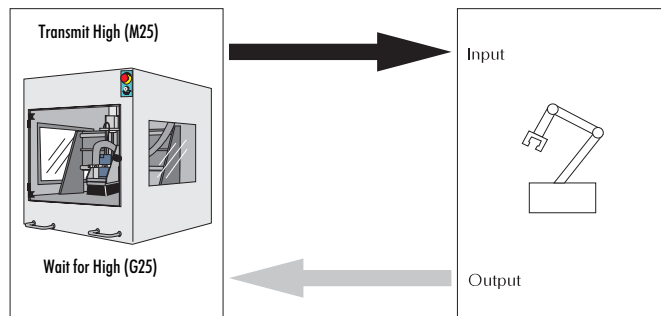
4. The robot releases the work piece and leaves the work area. It transmits a low signal when it is away from the work area.



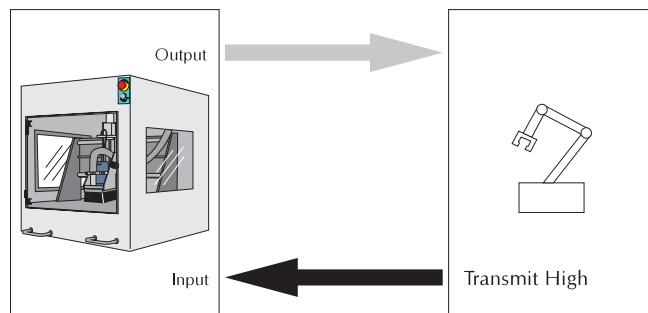
5. The router receives the low signal from the robot and begins to router the part.



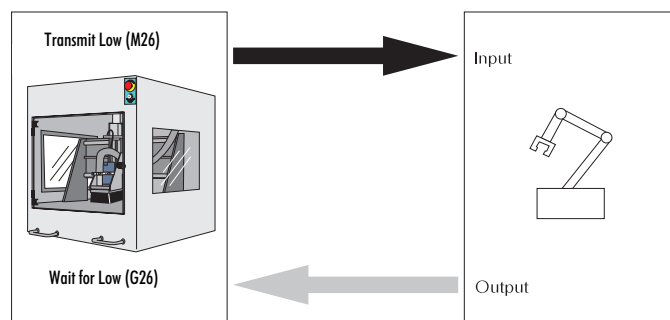
6. When the router has finished the part, it signals the robot (M25) to come take the part and waits (G25) for a high signal.



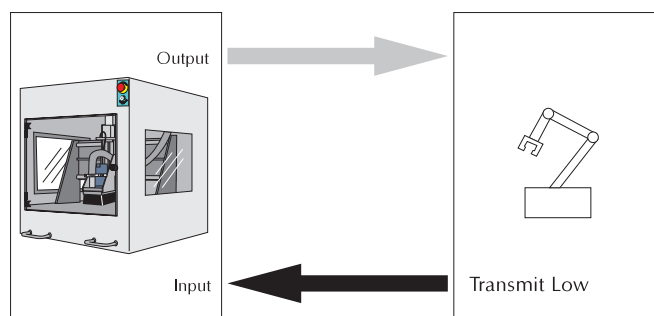
7. When the robot receives the signal, it approaches the router, grasps the part and sends the high signal to the router.



8. When the router 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 router's work area, it sends a low signal to the router.



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  
N1M25H5;TRANSMIT HIGH SIGNAL TO ACC2  
N2G25H5;WAIT FOR HIGH SIGNAL FROM ACC2  
N3M10;CLOSE VISE  
N4M26H5;TRANSMIT LOW SIGNAL TO ACC2  
N5G26H5;WAIT FOR LOW SIGNAL THROUGH INPUT 1
```

NC Program

```
N0;THIS FILE FOR BENCHTOP ROUTER  
N1;USE 5.5" X 5.5" PINE  
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

```
N26M25H5;TRANSMIT HIGH SIGNAL TO ACC2  
N27G25H5;WAIT FOR HIGH SIGNAL FROM ACC2  
N28M11;OPEN VISE  
N29M26H5;TRANSMIT LOW SIGNAL TO ACC2  
N30G26H5;WAIT FOR LOW SIGNAL FROM ACC2  
N31M47L3;CYCLE BACK TO N0  
N32M2;END OF PROGRAM
```


Index

A

- About BenchRouter E-51
- Absolute coordinate programming code J-2
- Accessories I-4
- Accessory Kit B-3
- Accessory off code J-5
- Accessory on code J-5
- Address character F-2
- Administrator Mode E-48
- Alter existing tool E-32
- Arc Centers Incremental E-37
- Arrange Icons E-49
- Auto-refresh E-40
- AutoCAD B-2. *See also* CAD programs
- AutoSave E-47

B

- Backlash E-46
- Ball screw
 - lubricant B-6
- Benchtop CNC Router
 - components B-2
 - Control Program B-2
- Benchtop Router Interface E-2
- Blocks
 - maximum number per program F-2
 - of NC code F-2
- Boring cycle G-11
- Boring cycle code G-7
- Boring cycle with dwell code G-7
- Boring cycle with spindle off code G-7
- Buttons E-41

C

- CAD
 - programs B-2
- Call to subprogram code G-13
- Canned cycle
 - boring G-11
 - cancel G-7
 - codes J-3
 - dwell G-9
 - G codes G-7
 - peck drilling G-9
 - programming G-7
 - straight drilling G-8
 - tapping threads G-10
- Cascade E-49
- Centerline Options E-40
- Chain to next program J-6
- Check Home E-41
- Circular interpolation B-2, G-4, H-14

- clockwise, code for J-2
- counterclockwise, code J-2

- Circular interpolation programming G-3

- Clamp ACC2 code J-5

- Clear E-19

- Close Command E-13

- CNC. *See* NC programming

- compensation offset E-44

- Computer System Requirements A-9

- Context Help E-6

- Control Program

- Installation A-9

- screens C-4

- starting C-2

- Controller Box B-9

- Coordinate System

- codes J-3

- Command E-42

- Coordinate Systems, understanding H-27

- Copy E-18

- Corner offset circular interpolation H-14

- Corner offset code J-4

- Crosspad E-9

- Ctrl+C E-56

- Ctrl+F E-56

- Ctrl+G E-56

- Ctrl+H E-56

- Ctrl+L E-56

- Ctrl+N E-56

- Ctrl+O E-56

- Ctrl+P E-56

- Ctrl+R E-56

- Ctrl+S E-56

- Ctrl+Shift+Z E-56

- Ctrl+Space E-56

- Ctrl+T E-56

- Ctrl+V E-56

- Ctrl+X E-56

- Ctrl+Y E-56

- Ctrl+Z E-56

- Customer Service Department A-3

- Cut E-18

- Cutter compensation H-6

- canceling H-15

- codes J-4

- IJK vectors H-10

- setting offsets H-11

- starting H-6, H-7

D

- Default Values for Speed E-35

- Delete E-19

- Diameter Offset E-31

- docked screen component E-60

- Door Interlock Switch D-3

- Door, safety I-2
- Dwell code J-2
- DXF files B-2
- E**
- Edit Menu E-17
- Edit Tool Materials E-31
- Edit Window C-8
 - moving E-59
- Editor Preferences E-47
- EIA RS-274D NC codes B-2
- Emergency Stop D-3
 - button
 - location of B-5
- Emergency Stop Button D-3
- Enable Soft Limits E-46
- Enable Subprograms E-37
- Enclosure
 - Door, Interlock Switch B-4
- End of program J-5
- Estimate Runtime E-28, E-53
- Exit command E-16

F

- F1 E-56
- F3 E-56
- F5 E-56
- F6 E-56
- F8 E-56
- Feed Rate Override E-10
- Feedhold E-29
- File Default E-47
- File Menu E-12
- Find E-19
- floating screen component E-60
- Font Settings E-24
- fuses B-5

G

- G92 E-43
- General Panel E-62
- Glasses I-2
- Go to code J-6
- Goto Line E-20
- Goto Position E-42

H

- H code K-5
- Hardware Installation A-4
- Helical interpolation G-5
- Help Index
 - Menu, Command E-50
- Home E-41
- Homing
 - commands H-3
- Hot Keys E-56

I

- Inch programming code J-2
- Incremental coordinate programming code J-2
- Initial Tool Position E-40
- Inputs Toolbar C-7, E-7
- Insert N Codes E-21
- Insert or Remove Spaces E-22
- Interface Card
 - Installing A-4
- Interface Card Panel E-62
- interlock switch D-3
- Interpolation B-2
 - circular G-3
 - linear G-2

J

- Jog Control E-25
- Jog Control Panel E-9
 - Pop-up Menu commands E-55
- Jog Distance E-35
- Jog Settings E-35
- Jog Speed E-35

K

- Key Lock switch B-5

L

- L code G-13
- Limit switches
 - location of B-4
- limit switches D-3
- Linear interpolation
 - code J-2
 - programming G-2
- Linear motion code J-2
- Lock E-23
- Loop counter code G-13

M

- Machine Coordinates H-27
- Machine Info Window E-4
- Maximum blocks per program F-2
- Menu Bar C-4, E-12
- Message Bar E-3
- Metric code J-2
- Multiple Coordinate Systems H-27
- Multiplier E-31

N

- NC code
 - for robotic communication K-5
- NC part programs F-2
- negative limit D-4

New Command E-12
Num Teeth E-31

O

O code G-13
Offset
 number E-44
 number code J-4
 sides, changing H-13
 values, changing H-12
Offset Table E-44, J-4
Open Command E-13
Opening a Recent Program E-16
Operator Panel E-10, E-25
Optional
 Skip E-36
 Stop E-10, E-37, J-5
Options Panel E-40
Origin E-39
Output to file code J-6
Outputs Toolbar C-6, E-8

P

P code, calling subprograms G-13
Password
 changing, creating E-49
Paste E-18
Pause E-29
 G code for J-5
Pause code J-2
Peck drilling code G-7
Peck drilling cycles G-9
Plane selection codes J-4
Polar programming H-2
 cancel code J-3
polar programming
 code J-3
Pop-Up Menu E-52
Position E-24
Position Readout C-10
Position Window E-5
 Pop-up menu commands E-53
Positioning Information Areas E-58
Positioning Toolbars E-58
positive limit D-4
Preferences E-47
 exiting E-48
 Security E-48
Preset View E-39
Preview Window E-40
Print E-15
Print Setup E-15
Program Edit Window E-3
 Pop-up Menu commands E-52
Program Menu E-26
Program stop code J-5

Programming
 suggestions F-23

Q

Quick Home E-41
QuickRun E-53
QuickVerify E-53

R

Rapid motion codes J-3
Rapid to initial position code G-7
Rapid to point R code G-7
Rapid traverse
 code J-2, J-3
 programming G-6
Redraw E-54
Remove Comments E-23
Remove N Codes E-23
ReNUMBER E-21
Renumbering and Subprograms E-22
Replace E-20
Reset Override E-45
Restore Settings E-51
Restore Unit Mode E-37
Return from subprogram code G-13
Return to reference point code J-3
Rewind J-6
robot synchronization codes J-2
Rotation
 and scaling H-23
 codes for H-21
Rotation codes J-4
Rules, Safety I-1, I-2
Run Settings E-27, E-36
Run/Continue E-26, E-29
Running a Sample NC Program D-5, D-14
Running a sample NC program B-8

S

Safety D-2
Safety Door I-2
 interlock switch D-3
Safety Glasses I-2
Safety Rules D-2, I-1, I-2
Save As ... E-14
Save As Dialog Box E-14
Save Command E-14
Save Settings E-50
Scaling
 and rotation H-23
 codes for H-18
 creating mirror images with H-20
 each axis H-19
Scaling codes J-4
select an existing coordinate system E-42
Select Font E-24

- Select Tool E-32
- Selecting Commands
 - Pop-up Menus E-52
- Set
 - robot output J-5
- Set Position E-34
- Set reference point code J-3
- Set/Check Home E-41
- Setting
 - offsets H-11
 - soft limits H-4
- Setup Library E-30
- Setup Menu E-33
- Setup Program E-61
- Setup Tool D-8
- Shift Z axis code J-4
- Show Tool E-40
- Simulation Mode E-63
- Single Step E-10, E-36
- slider bars E-40
- slider button E-40
- Soft Limits E-46
- Solid Options E-40
- Specify depth of cut code G-7
- Specify starting reference point code G-7
- spectraCAM B-2
- Spindle E-45
 - motor off code J-5
 - motor on code J-5
 - speed
 - control switch, location of B-5
- Spindle Speed Override E-10
- Standard Toolbar C-4, E-6
- Starting Block E-26
- Status Bar E-11
- Stock Dimensions E-39
- Stock Panel E-39
- Stop E-10, E-29
- Stop Redraw E-54
- Straight drilling G-8
- Straight drilling code G-7
- Straight drilling with dwell code G-7
- Style E-38
- Subprogram
 - call to J-6
- Subprogram reference block code G-13
- Subprogram, return from code J-6
- Subprograms G-13
 - nested G-13

T

- Tapping threads G-10
- Technical Support A-11
- Tile E-49
- Tip of the Day E-51
- Tool

- change code J-5
- length offset H-29
- length offset code J-4
- offset adjust H-29

- Tool Library E-31
- Tool Type E-31
- Tool Wizard E-32
- Toolbars E-25
- Tools I-3
 - Creating E-30
- Tools Menu E-30

U

- Unclamp air chuck code J-5
- Undo E-17
- Uniform scaling H-18
- Units E-42
- Unpacking A-3
- Using Help E-50

V

- Verify E-27
 - While Running E-38
- Verify Settings D-6, E-27, E-38
- Verify Type E-41
- Verify Window C-11, E-4, E-25
 - Pop-up Menu commands E-54
- View Menu E-24
- View Panel E-38

W

- Warnings as Errors E-37
- Warranty A-11
- Window List E-50
- Window Menu E-49
- Words, NC F-2
- Work Area B-2
- Work Coordinates H-27
- Wrenches I-2

Z

- Zero Position E-34
- Zoom E-39