BenchTurn 7000 Turning Center

USER'S GUIDE

Catalog #200063 Rev D

intelitek

BenchTurn 7000 Turning Center User's Guide

Catalog #200063 Rev D

August 2016

website: http://www.intelitek.com

email: info@intelitek.com

Tel: (603) 625-8600

Fax: (603) 437-2137

intelitek
Dynamic Solutions, Inspired Classrooms

Warnings

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

Read the entire contents of this guide before running the BenchTurn 7000 Turning Center.

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

The following icons indicate important information throughout this User’s Guide.

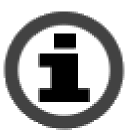
 **Safety**	Provides essential safety instructions that must be followed to prevent operator injury or death.
 **Product Care**	Provides recommendations for reducing the chance of machine damage.
 **Take Note**	Provides important information about your product.

Table of Contents

intelitek
Dynamic Solutions, Inspired Classrooms

Using this Guide

Welcome to the BenchTurn 7000 User's Guide.

This guide is designed to help you install and begin using the BenchTurn 7000 hardware and software. The later chapters provide an NC programming reference.

We recommend that you use the guide as follows.

1. Read chapter 1 Safety Guidelines. Review this chapter often.
2. Read chapter 2 Introducing the BenchTurn 7000.
3. Install the hardware and software as described in chapter 3 Installing the Hardware and Software.
4. Read chapter 4 Maintaining the BenchTurn 7000.
5. Read chapter 5 Using the Control Software.
6. Follow the instructions in the tutorial presented in chapter 6 Tutorial: Turning a Sample Part.
7. Use the remaining chapters as a reference guide for NC programming.
   a. Chapter 7 Basic CNC Programming presents guidelines for writing basic NC programs, and lists and describes the use of all codes available for use with the BenchTurn 7000.
   b. Chapter 8 NC Programming Routines provides instructions with examples for advanced NC programming routines.
   c. Chapter 9 Multiple Tool Programming provides instructions for configuring the control software and writing NC code for programs that require the use of more than one cutting tool. The chapter also presents step-by-step instructions for turning a sample part using multiple tools.
   d. Chapter 10 An Introduction to CNC Turning provides a basic introduction to the fundamentals concept in CNC turning.
   e. Chapter 11 Automation Integration provides instructions for integrating the BenchTurn 7000 in a robotic environment.

1. Safety Guidelines

The safety rules presented here should be reviewed and practiced by all operators of the BenchTurn 7000 turning center.

This section presents the following information:

Section Contents: Safety Guidelines		
Section	**Name**	**Page**
1.1	Detailed Safety Guidelines	1
1.2	Safety Checklist	5

1.1. DETAILED SAFETY GUIDELINES

The table below provides detailed safety instructions.

Info Table: Safety Guidelines		
Category	**Guideline**	**Comment**
Operator knowledge and authorization	Review the User's Guide.	Read this guide carefully before you use the turning center and keep it readily accessible for quick reference. Know the intended applications and limitations of the turning center as well as its hazards.
	Keep untrained visitors away from the equipment.	Children and visitors unfamiliar with the hazards of rotating machinery should always be kept away from the work area.
	Prevent unauthorized users from operating the turning center.	Ensure that unauthorized users can not gain access to the room in which the machine is stored. Ensure that the computer to which the machine is attached is password protected.
Behavior	Do not overreach.	Keep your footing and balance at all times so you won't fall against or clutch at the moving machine.
	Do not operate the machine under the influence of alcohol or drugs.	Alcohol or drugs may impair your judgment 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.
Work Area	Keep the work area clean.	Cluttered work areas and bench tops invite accidents.
	Avoid a dangerous environment.	Don't use the turning center in damp or wet locations. Never operate electrical equipment in the presence of volatile and flammable petroleum-based solvents and lubricants.

Info Table: Safety Guidelines		
Category	**Guideline**	**Comment**
	Keep coolant away from electrical components.	Do not allow coolant to splash into or near the computer.
Clothing and Hair	Avoid loose hair and clothing.	Don't wear loose clothing or jewelry that can get caught in moving parts. Wear a hat or hair net, or tie your hair back to keep it away from moving parts.
Safety Equipment	Wear safety glasses.	During operation any power tool can throw foreign objects and harmful chemicals into your eyes. Always put on safety glasses or eye shields before starting up the turning center. Safety glasses or shields should provide full protection at the sides, as well as the front of the eyes.
	Ground all tools.	The turning center has an AC power cord terminated by a three-prong plug. The power cord should be plugged into a three-hole, grounded receptacle. If a grounding adapter is used to accommodate a two-prong receptacle, the adapter wire must be attached to a known ground. Never remove the third prong from the plug on the AC power cord.
	Keep the safety door closed while machine is in motion.	The safety door should remain in place whenever the spindle motor is on or the cross slide is moving.
	Remove adjusting keys and wrenches.	Make it a habit to check that keys and adjusting wrenches are removed from the turning center before turning on the machine.
Emergency Stop	Stopping the machine.	Before you run the BenchTurn 7000 for the first time, you should know how to stop the machine should an emergency situation arise. There are a number of ways an emergency stop can be initiated on the turning center: • Press the Emergency Stop button. • Press the Control and Space Bar keys on the computer keyboard simultaneously. • Activate one of the limit switches. • Activate the safety door interlock switch.
	When to use the Emergency Stop.	You should use the Emergency Stop button to disconnect power to the turning center when changing tools or when mounting or removing a workpiece.
	Using the machine-mounted emergency stop button.	There is an Emergency Stop button located on the front panel of the turning center; it has an oversized red cap. To engage: Press the button in. To release: Turn the button clockwise; it will pop out on its own.

intelitek
Dynamic Solutions, Inspired Classrooms

Info Table: Safety Guidelines		
Category	**Guideline**	**Comment**
	Using the software stop button.	The execution of the part program can be interrupted by pressing the Control and Spacebar buttons on the computer keyboard. Unlike using the Emergency Stop button, this method of stopping the turning center does not cause the software to lose track of the tool position.
Operation Rules	General	Proper setup of the turning center is essential for safe turning. These procedures must be followed each time a new tool is mounted. General setup requirements for the turning center include checking components for cleanliness and lubrication, mounting the cutting tool, mounting the workpiece, and setting the spindle rotation speed. The following safety rules should be practiced by all operators of the BenchTurn Turning Center for each use.
	Avoid accidental starting.	Make sure the power switch is off before plugging in the turning center power cord.
	Check lathe components.	Always examine the bed saddle, cross slide and lead screw to be sure they are free of shavings and particles from previous operations. Remove such debris from the lathe to avoid possible binding of components which may result in possible damage to the lathe, the workpiece, or the operator. Always make sure the machine is properly lubricated.
	Do not force a tool.	Select the feed rate and depth of cut that are best suited to the design, construction, and purpose of the cutting tool. It is always better to take too light a cut than too heavy a cut.
	Use the right tool.	Select the type of cutting tool best suited to the turning operation. Don't force a tool or attachment to do a job it wasn't designed for.
	Maintain cutting tools in good condition.	Keep cutting tools sharp and clean. Lubricate and clean turning center components on a regular basis.
	Mount the cutting tool correctly.	Each cutting tool used in the turning operation must be sharp and tightly inserted in the tool turret. The cutting edge of the tool must be on the centerline or just below the centerline (0.004 inch or 0.1mm maximum) of the axis of rotation of the lathe
	Secure the workpiece.	Be certain that you have firmly secured the workpiece in the spindle and the cutting tool to the cross slide before turning on the spindle motor.
	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.
	Turn the spindle by hand before starting.	Manually turning the spindle allows you to safely determine that the tool will not hit the chuck or stock on start up.

Info Table: Safety Guidelines		
Category	**Guideline**	**Comment**
	Set the spindle rotation speed.	The BenchTurn Turning Center is equipped with an electronically controlled spindle motor which produces a comprehensive range of spindle rotation speeds. Speed can be set with the Control Software or by using an S code in the NC program. Always use a safe spindle speed.
Accessories	Use recommended accessories only.	To avoid stressing the turning center and creating a hazardous turning environment, use only those accessories designed for use with the BenchTurn7000, available through Intelitek Corporation.

1.2. SAFETY CHECKLIST

Safety

Post copies of this checklist in the work area. Verify that all items are checked-off prior to each operation of the BenchTurn turning center.

Before you enter the work area:

- ☐ Put on safety glasses.
- ☐ Tie back loose hair and clothing.
- ☐ Remove jewelry including rings, bracelets and wristwatches.

Before turning 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.
- ☐ Remove all loose parts and pieces from the machine.
- ☐ Remove adjusting keys and wrenches from the machine.
- ☐ Close the safety door.
- ☐ Only operate the machine after being properly trained in its use.
- ☐ Perform a dry run:
    - ☐ 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 turning a part:

- ☐ Do not touch moving or rotating parts.
- ☐ Press the Emergency Stop button before opening the safety door.
- ☐ Only open the safety door after the spindle has stopped rotating.
- ☐ Press the Emergency Stop button whenever changing tools or mounting or removing a workpiece.
- ☐ Release the Emergency Stop button only after closing the safety door.
- ☐ Keep all unauthorized persons away from the work area.

2. Introducing the BenchTurn 7000

The BenchTurn 7000 is a versatile PC-based benchtop CNC turning center that enables you to deliver robust instruction in computer numerical control and advanced manufacturing.

The BenchTurn 7000 comes equipped with 2-axis stepper motors, ball screws, a variable speed brushless spindle motor, limit/home switches, and an MT3 taper spindle with MT2 taper tailstock.

This benchtop CNC system requires no assembly and is ready to run on an Ethernet port on a standard PC, and fits comfortably into any classroom without sacrificing features.

Like larger industrial machines, the BenchTurn 7000 uses EIA, ISO, and Fanuc-compatible G&M code programs to cut parts in a variety of materials.

This section presents the following information:

2.1. OVERVIEW OF STANDARD FEATURES

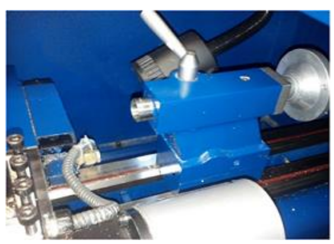
Some of the BenchTurn Turning Center's most notable hardware and software features are listed in the table below:

Info Table: Standard Features	
Network and software features	Ethernet-based control
	PC-based CNC software
	EIA RS-274D standard G&M code programming
	A built-in full-screen NC program editor with graphic tool path verification
	Multiple tool programming
	Help functions on screen
Standard hardware features	No assembly required
	Brushless spindle motor
	Tailstock
	Work light
Standard turning specifications	X-axis travel of 2.96 inches (75mm)
	Z-axis travel of 9.84 inches (250mm)

Info Table: Standard Features	
	Feed rates up to 20 IPM (500mm/min) Rapid traverse up to 79 IPM (2000mm/min)
	Computer-controlled spindle speeds from 100 to 3,000 RPM
Safety features	Full enclosure with automatic safety door lock
	Automatic diagnostics and power cut off protection
	Safety door and limit switches
	Emergency stops from the turning center and computer keyboard
Machine ready optional accessories	Coolant ready
	Jog pendant ready
	Robotic integration ready with 6 inputs, 6 outputs

2.2. BENCHTURN 7000 COMPONENTS

This section shows the location of major components of the BenchTurn 7000, arranged by the view from which they are visible:

Section Contents: Components	
 2.2.1 External View, pg. 8	  2.2.2 Left Side Panel, pg. 8
  2.2.3 Enclosure (Front View), pg. 9	 2.2.4 Enclosure (Right Side View), pg. 11

2.2.1. External View

The external view is shown below.

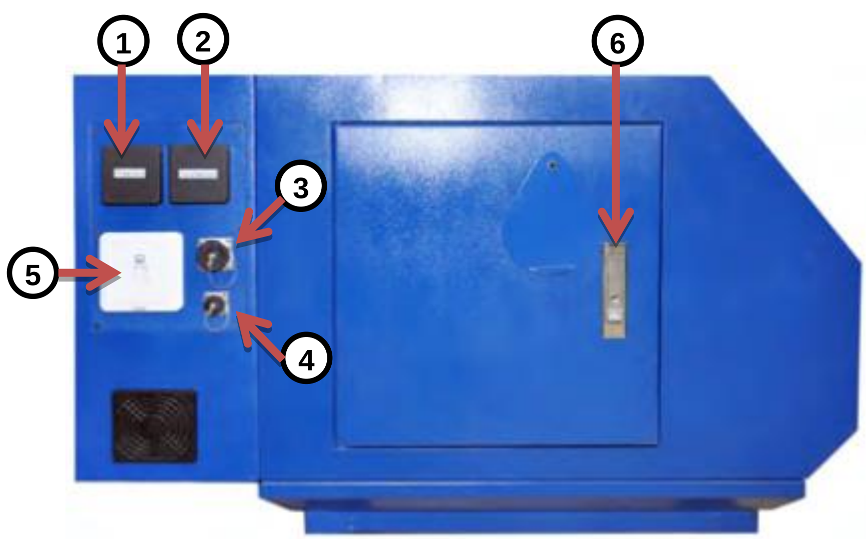
1. Enclosure door
2. Emergency stop switch
3. Enclosure release button
4. I/O ports
5. On/Off switch
6. Right access door

Notes:

- The Safety door encloses the turning area to help protect the operator from flying chips. A magnetic shield interlock switch prevents the machine from operating with the shield open.
- The Emergency Stop button is used to halt machine operation. When pushed, machine operation stops immediately. To resume operation, the button must be rotated clockwise and will then pop out on its own. It is important that this button be pushed in (i.e. engaged) before performing any manual operation, such as changing the stock or tooling.

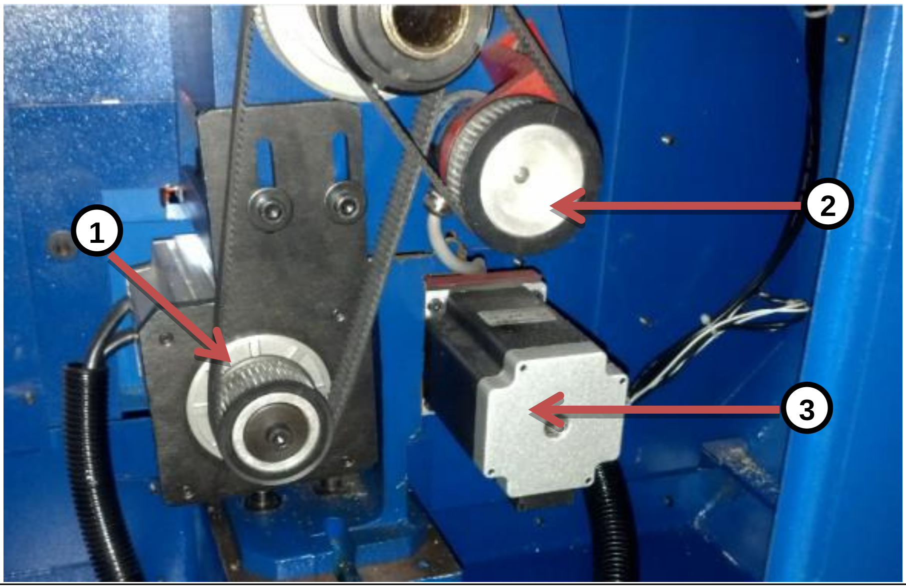
2.2.2. Left Side Panel

The graphic below shows the machine as viewed from the left side.



1. PC power supply
2. Coolant power supply
3. Jog pendant port
4. FANUC panel port
5. Ethernet port
6. Left access door

The graphic below shows the contents exposed by opening the left side access door.



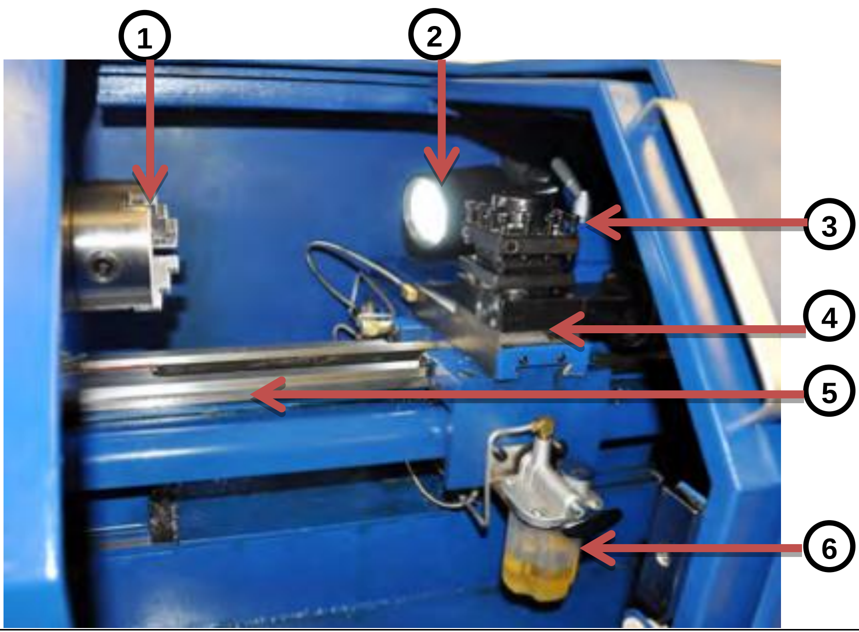
1. Spindle drive
2. Encoder
3. Z-axis drive motor

Note:

- The left door panel provides access to the spindle shaft and belts, the Z-axis stepper motor, and related components. Do not handle these components unless they require maintenance.

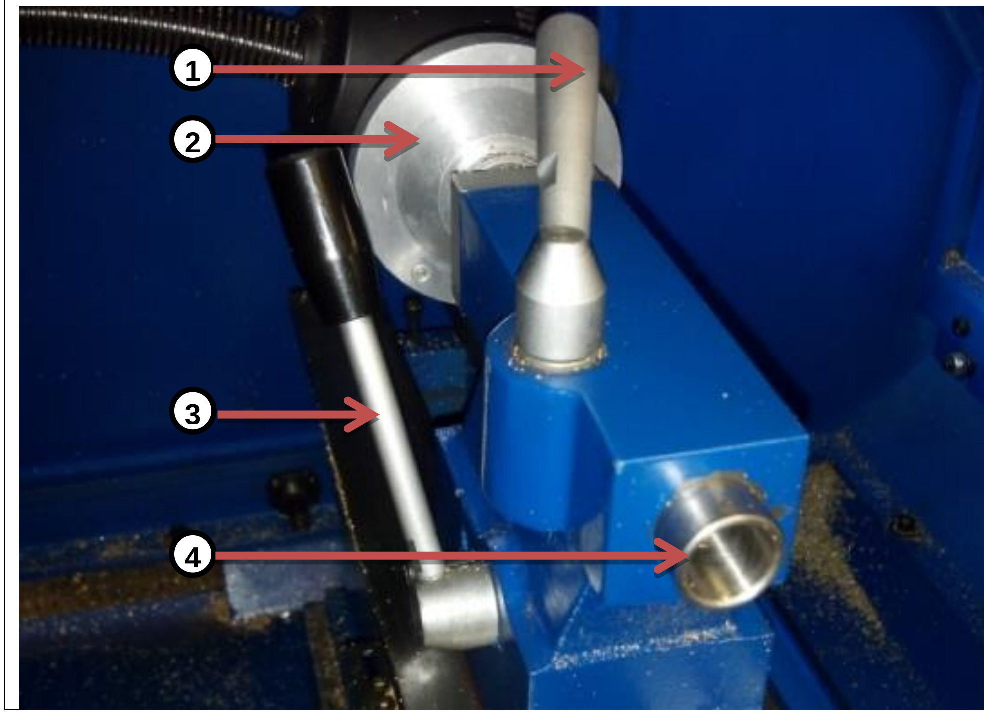
2.2.3. Enclosure (Front View)

The graphic below shows the contents exposed by opening the safety door.



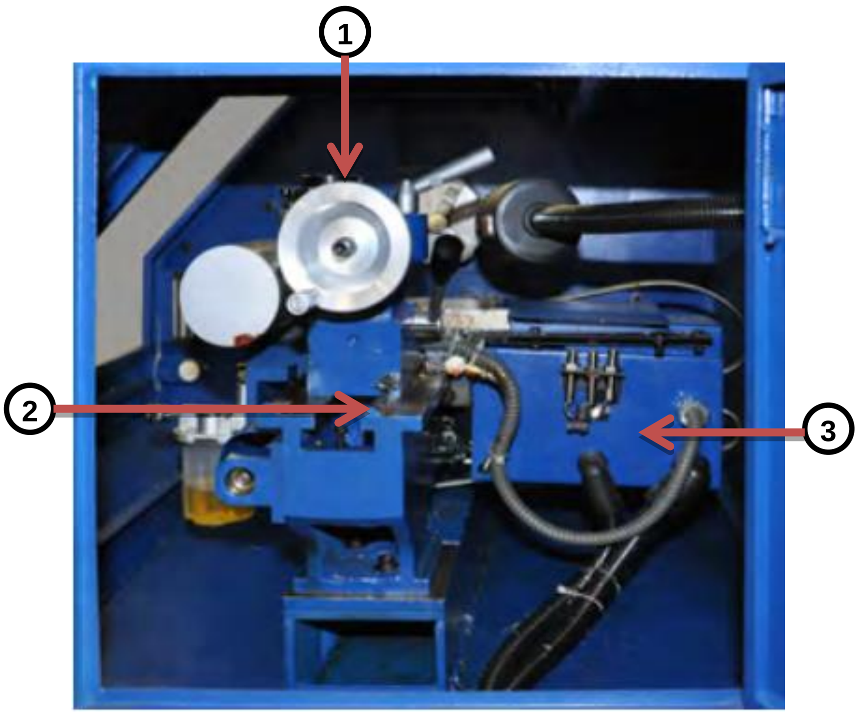
1. 3-Jaw chuck
2. Work light
3. Tool turret
4. Cross slide
5. Bed
6. Lubricant reservoir

The graphic below shows the tailstock, used to secure the end of long workpieces.

	1. Quill lock 2. Tailstock handwheel 3. Tailstock lock 4. Quill

2.2.4. Enclosure (Right Side View)

The graphic below shows the right side of the enclosure area. These components are accessed by opening the right side panel.



1. Tailstock handwheel
2. Serial number (on top of rear rail)
3. X-axis drive motor housing

2.3. OVERVIEW OF CNCBASE/MOTION CONTROL SOFTWARE

The heart of the BenchTurn 7000 turning center is the control software (CNCMotion or CNCBase) that runs on your computer. Using industry standard EIA RS- 274D NC codes, the control software provides for two-axis CNC programming and turning.

The control software is extremely easy to use with all necessary functions readily available to run a part program.

CNCBase and CNCMotion differ only in that CNCMotion provides 3D simulation of the turning process.

For more information on the control software, see 5 Using the Control Software, pg. 44.

2.4. STANDARD ACCESSORIES

The accessory kit supplied with the turning center contains all the tools and hardware necessary for installing and maintaining the turning center. Additional tool holding devices and tools are available as options.

The table below lists the standard accessories supplied with the BenchTurn 7000.

Info Table: Standard Accessories
Tailstock
4" 3-jaw chuck with key
4-station automatic tool turret
One shot lubrication system
Internal work light
Turning center accessory package:

Item	Qty	Description
1	2	Dead center (1 short, 1 long)
2	1	Allen wrench set (4 keys)
3	1	8-10mm Wrench
4	1	12-14mm wrench
5	1	CNC Base Control Software
6	2	Electrical panel keys
7	1	5x20 mm fuse - 10 Amp
8	1	Chuck key, T-handle
9	3	Chuck jaws
10	1	Ethernet Cable

2.5. OPTIONAL ACCESSORIES

Intelitek offers a variety of turning center accessories, CAM software, curriculum, and documentation.

For more information about these products call your Intelitek dealer, call Intelitek directly at (800)221-2763 or (603) 413-2600, or browse our web site www.intelitek.com.

3. Installing the Hardware and Software

This section presents instructions for installing the hardware and software components.

Procedure Outline: Installation			
No.	**Step**	**Section**	**Page**
1	Prepare your hardware for installation.	3.1	13
2	Install the hardware.	3.2	17
3	Install and configure the software.	3.3	20

This section also presents the following information.

Section Contents: Installing the Hardware and Software		
Name	**Section**	**Page**
Contacting Technical Support	3.4	34
Returning Defective Products	3.5	35

3.1. PREPARING FOR INSTALLATION

This section presents instructions for preparing the work space and machine for installation.

Procedure Outline: Preparing for Installation			
No.	**Step**	**Section**	**Page**
1	Verify that the computer to be used with the turning center meets minimum requirements.	3.1.1	14
2	Prepare a work space for the turning center.	3.1.2	14
3	Remove the crating.	3.1.3	14
4	Unpack and set up the turning center.	3.1.4.	15
5	Check your shipment to ensure that all items ordered are present and undamaged.	3.1.5	16
6	Register your turning center to activate the warranty.	3.1.6.	17

3.1.1. Verifying Computer Requirements

Use the checklist below to ensure that the computer that will be attached to the turning center meets minimum requirements.

Checklist: Verifying Computer Requirements

- Windows 7/Windows 8/Windows 10 - 32 or 64bit
- 512 MB RAM (1 GB Recommended)
- CD-ROM
- 100 MB of available hard drive space (300 MB Recommended)
- VGA graphics or better graphics display (minimum 256 colors)
- Available Ethernet port
- A mouse or other pointing device
- ATX Power Supply (Recommended)

Note: Your operating system might have additional hardware requirements.

3.1.2. Preparing the Work Space

Use the checklist below to ensure that the work space is ready for the installation of the machine.

Checklist: Preparing the Work Space

- A sturdy table on which you will place the turning center and your computer. Placing the table against a wall provides more stability.
- For customers in the U.S.A.: A 120VAC, 15 Amp outlet
- For international customers: A 220VAC, 8 Amp outlet

Product Care

We recommend the use of a voltage surge protector and line filter to protect your computer system. A voltage surge protector is not supplied with the BenchTurn 7000.

3.1.3. Removing the Crating

Follow the procedure below when removing the crating after delivery of the product.

Procedure: Removing the Crating

1. Inspect the crating for any visible signs of damage. If there is damage to the crating, contact the shipping company and Intelitek Customer Support.
2. Cut any banding on the outside of the crate.
3. Remove the top of the crate.
4. Remove the sides of the crate.

Take Note	Intelitek will not be responsible for any damage caused during shipping when components are not returned in the original packing materials. Store the packing materials at least until the installation is complete and proper operation has been verified.

3.1.4. Unpacking and Setting up the Turning Center

Follow the procedure below for unpacking and setting up the turning center.

Procedure: Unpacking and Setting up the Turning Center

1. Position the pallet near the table on which you'll set the turning center. The table should be located against a wall for maximum support.
2. Remove the staples that attach the bottom of the cardboard container to the pallet.
3. Cut the banding around the container.
4. Lift the cardboard cover off the top of the container.
5. Remove the sides of the container.
6. Inspect the turning center chassis for signs of visual damage such as a broken shield, a dent in the chassis, or damaged cables.
7. Call Intelitek Customer Support if any damage is noted.
8. Remove the four bolts that hold the turning center base to the pallet, using a 19mm wrench.
9. Store the bolts and other packaging materials, in case the product has to be returned or transported.
10. Lift the turning center off of the pallet and onto the table. If lifting the machine manually:
    a. Turn the four lift-handles out.
    b. With one person on each corner of the machine, carefully lift the machine by the lift handles onto the workbench.
    c. Return the four handles to their original positions underneath the enclosure.
11. Once the machine is on the table, position the turning center correctly for turning.
12. Remove the protective paper from the safety door.
13. Open the front door and remove the components from the enclosure.

3.1.5. Checking your Shipment

Follow the procedure below for checking your shipment once unpacked.

Procedure: Checking your Shipment
1. Locate the packing slip. This slip lists all of the items you should have received with your turning center. **2.** Check that all items on the packing slip are present. See the checklist below. **3.** Contact Intelitek Customer Support immediately if any item is missing.

Use the checklist below to ensure that all items listed on the packing slip are present in the delivery.

<table>
<tr><th colspan="2">Checklist: Checking your Shipment</th></tr>
<tr><th>No.</th><th>Item</th></tr>
<tr><td>1</td><td>BenchTurn 7000 Turning Center</td></tr>
<tr><td>2</td><td>Installation disk for CNCBase/Motion software</td></tr>
<tr><td>3</td><td>Documentation pack</td></tr>
<tr><td>4</td><td>Accessory kit
The accessory kit contents are shown below.

<table>
<tr><th>Item</th><th>Qty</th><th>Description</th></tr>
<tr><td>1</td><td>2</td><td>Dead center (1 short, 1 long)</td></tr>
<tr><td>2</td><td>1</td><td>Allen wrench set (4 keys)</td></tr>
<tr><td>3</td><td>1</td><td>8-10mm Wrench</td></tr>
<tr><td>4</td><td>1</td><td>12-14mm wrench</td></tr>
<tr><td>5</td><td>1</td><td>CNC Base Control Software</td></tr>
<tr><td>6</td><td>2</td><td>Electrical panel keys</td></tr>
<tr><td>7</td><td>1</td><td>5x20 mm fuse - 10 Amp</td></tr>
<tr><td>8</td><td>1</td><td>Chuck key, T-handle</td></tr>
<tr><td>9</td><td>3</td><td>Chuck jaws</td></tr>
<tr><td>10</td><td>1</td><td>Ethernet Cable</td></tr>
</table>
</td></tr>
<tr><td>5</td><td>Additional accessories ordered</td></tr>
</table>

3.1.6. Registering Your Turning Center

Follow the procedure below to register your turning center.

Procedure: Registering Your Turning Center

1. Locate the box that contains the documentation and installation disk.
2. Locate the registration card within that box.
3. Complete the card, printing all information clearly.
4. Return the card to Intelitek Customer Support at the address below,

    Intelitek Customer Support

    18 Tsienneto Road

    Derry, NH 03039

    USA

   or fax to 603-625-2137

3.2. INSTALLING THE HARDWARE

This section presents instructions for installing the BenchTurn 7000 hardware.

Procedure Outline: Hardware Installation

No.	Step	Section	Page
1	Connect the turning center to a computer.	3.2.1	17
2	Connect the turning center to a power source.	3.2.2	18
3	Configure and use the tailstock.	3.2.3	19
4	Install additional accessories purchased.	3.2.4	20

3.2.1. Connecting the Turning Center to a Computer

You will connect the turning center directly to a computer. Connection to the network (if required) is provisioned by the computer.

Safety

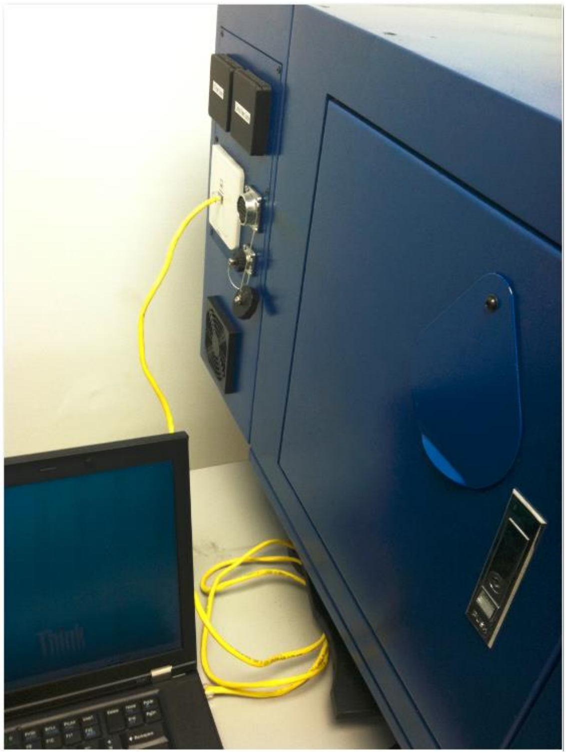
Do not connect power to the turning center or the computer until instructed to do so in the following procedures.

Follow the procedure below to connect the turning center to a computer.

Procedure: Connecting the Turning Center to a Computer

1. If not done previously, verify that the computer you are planning to use meets minimum requirements. See 3.1.1 Verifying Computer Requirements, pg. 14.

2. Use a cable with 8P8C (RJ-45) connectors at both ends to connect the turning center to the computer, as shown.



3.2.2. Connecting the Power

Safety

The turning center has an AC power cord terminated by a three-prong plug. The power cord should be plugged into a three-hole, grounded receptacle. If a grounding adapter is used to accommodate a two-prong receptacle, the adapter wire must be attached to a known ground. Never remove the third prong from the plug on the AC power cord.

Follow the procedure below to connect the turning center to a power supply.

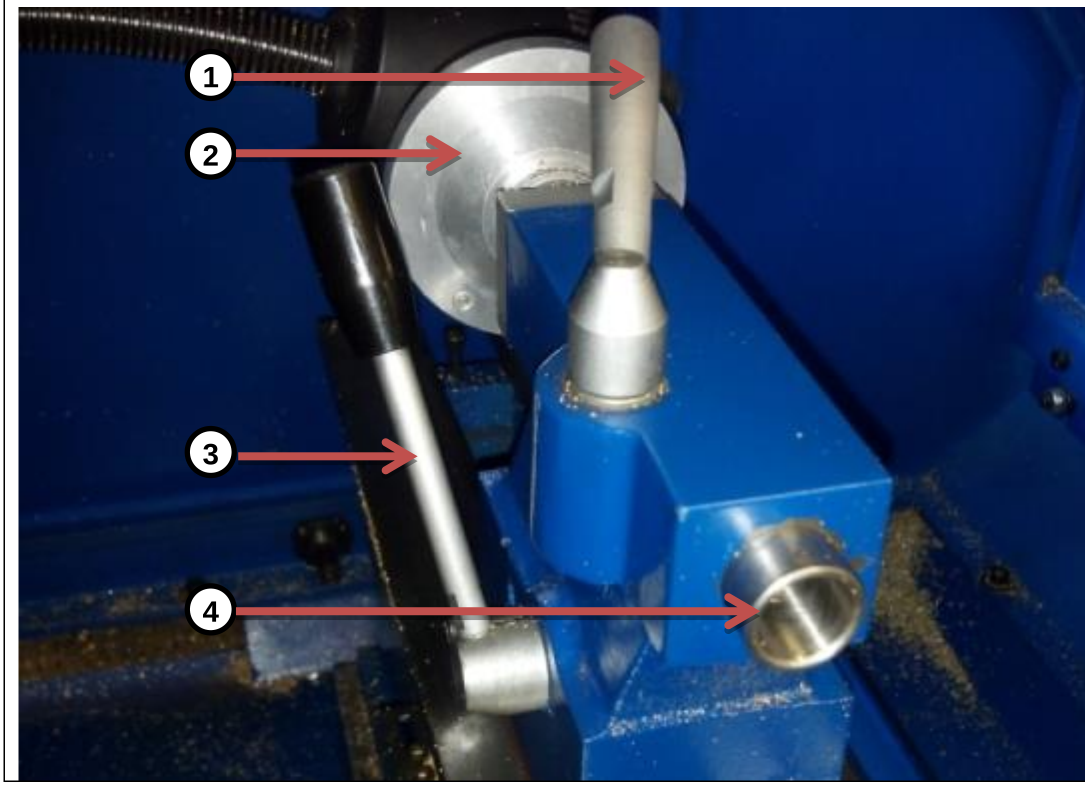
Procedure: Connecting the Power

1. Ensure that the turning center's power switch, located at its side, is set to the OFF position.

2. Connect the power cord from the turning center to the power source.

3.2.3. Configuring and Using the Tailstock

The BenchTurn 7000 is supplied with a tailstock already installed and aligned. Its components are shown below.



1. Quill lock
2. Tailstock handwheel
3. Tailstock lock
4. Quill

The use of the tailstock is summarized in the table below.

Tasks: Using the Tailstock	
Task	**Instruction**
Insert the live center into the quill	Push the live center into the quill firmly.
Extend and retract the quill	Turn the handwheel at the rear.
Lock or unlock the quill	Use the silver quill lock located above the tailstock.
Lock or unlock the tailstock	Use the tailstock lock lever located on the side of the tailstock facing the back of the machine enclosure.
Work without the tailstock	Either remove the tailstock from the bed, or lock it at the end of the bed with the quill fully retracted.

3.2.4. Installing Accessories

Each accessory kit is supplied with an installation guide.

Safety

To avoid stressing the turning center and creating a hazardous turning environment, use only those accessories designed for use with the BenchTurn turning center, available through Intelitek Corporation.

Product Care

Complete the hardware and software installation procedures (see 3.3 Installing the Software below), and test the functioning of the basic machine, before installing accessories.

3.3. INSTALLING THE SOFTWARE

This section presents instructions for installing the control software (CNCMotion or CNCBase) on the computer.

Procedure Outline: Software Installation			
No.	**Step**	**Section**	**Page**
1	Ensure that your computer meets the minimum requirements.	3.3.1	20
2	Check whether you have administrative privileges on the computer. Software installation requires administrative privileges.	3.3.2	21
3	Run the installation to install the software.	3.3.3	22
4	License your software.	3.3.4	27
5	Configure the software for your machine and accessories.	3.3.5	27
6	Configure the IP address of your computer.	3.3.6	31

This section also presents the following information:

Section Contents: Installing the Software		
Name	**Section**	**Page**
Uninstalling the Software	3.3.7	33

3.3.1. Verifying Computer Requirements

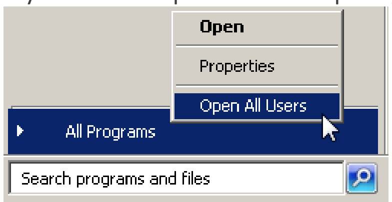
If not done previously, verify that the computer you are planning to use meets the minimum requirements. See 3.1.1 Verifying Computer Requirements, pg. 14.

If installing the software on a computer to be used only for writing and verifying NC programs, but not for interacting with the hardware itself, the requirement for LAN cards is not relevant.

3.3.2. Verifying Your Administrative Privileges

Administrative privileges are required to install the software.

If you are not sure whether you have administrative privileges on the computer, perform the following check:

Verifying Your Administrative Privileges
If the computer uses: • Windows 7, click the Windows Start button, then right-click the All Programs button. • An earlier version of Windows, right-click on the Windows Start button. If you see the Open All Users option, you do have administrative rights on the computer.  If you do not see the Open All Users option, you do not have administrative rights on the computer. Contact your system administrator for assistance.

3.3.3. Running the Installation

Install the software at this point.

During installation you will be presented with various installation options. Before you begin, verify your requirements:

Info Table: Software Requirements	
Requirement	**Options and Explanation**
CNCBase or CNCMotion	CNCBase and CNCMotion differ only in that CNCMotion includes a virtual machine simulation, CNCBase does not. CNCBase is the usual requirement for the computer connected to the turning center.
Fanuc Emulator	Select a Fanuc Emulator option if Fanuc emulation is required.
Machine	Verify which one of Intelitek's CNC machines will be connected to the computer.

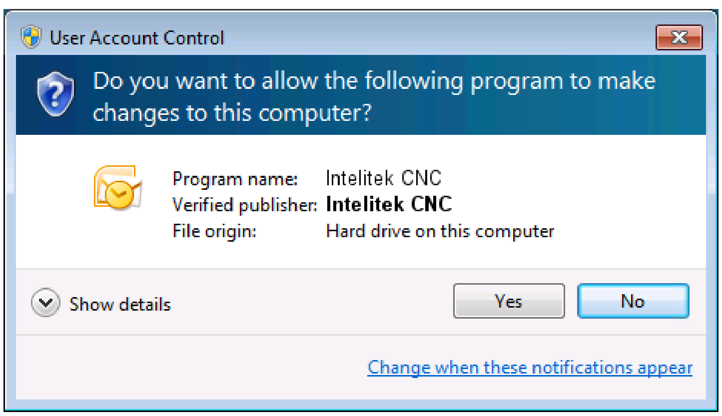
Follow this procedure to run the installation.

Procedure: Running the Installation

1. Insert the installation disk into the CD/DVD drive. The installation program should open automatically.

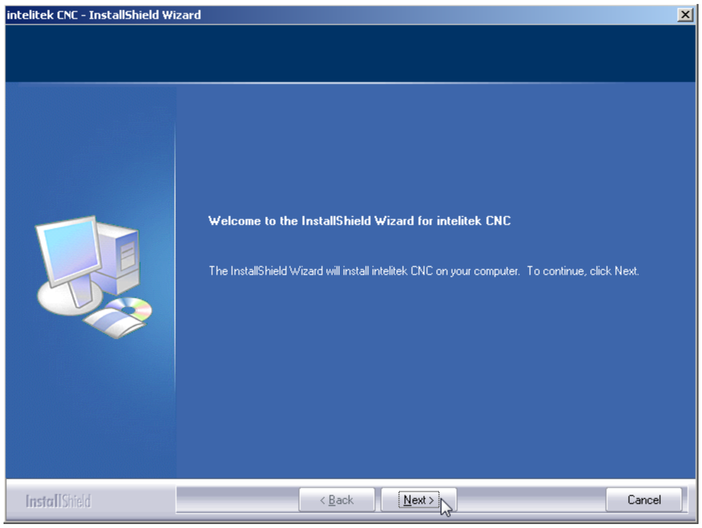
   If the installation does not open automatically, navigate to the *Install* folder and launch **iCNC.exe**.

2. If the User Account Control message displays, click **Yes**.

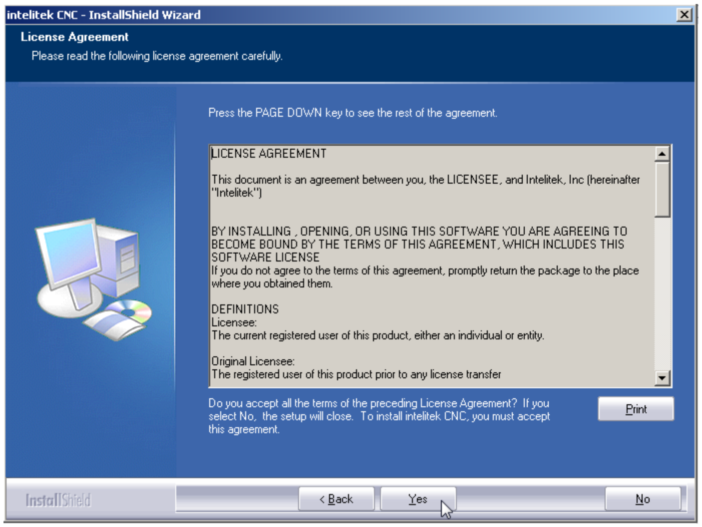
   The installation begins and the Welcome screen is displayed.

3. Click **Next**.



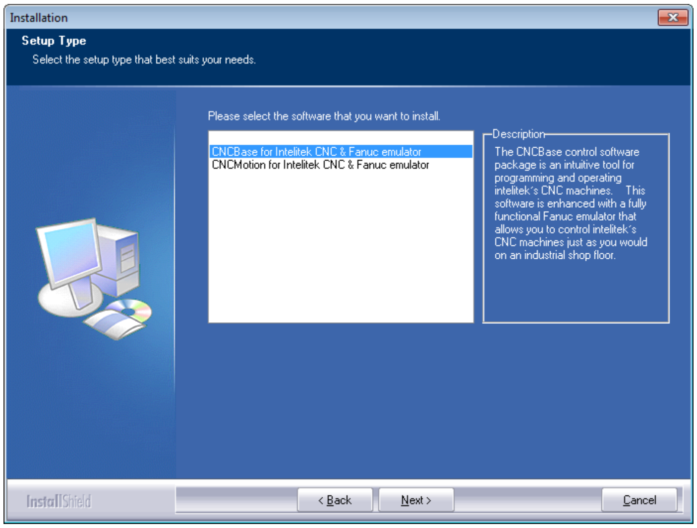
The License Agreement is displayed.

4. Click **Yes** to accept and continue.



The Software Selection screen is displayed.

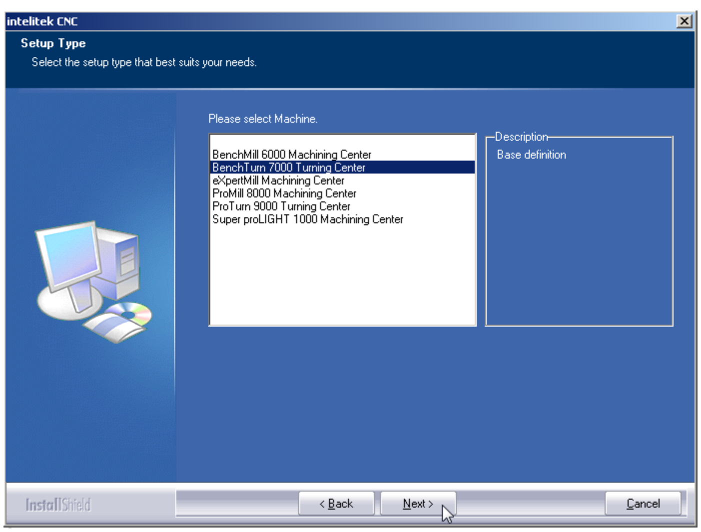
5. Select the software to install. It is important that the software you select matches the license you have purchased.



6. Click **Next**.

   The Machine Selection screen is displayed.

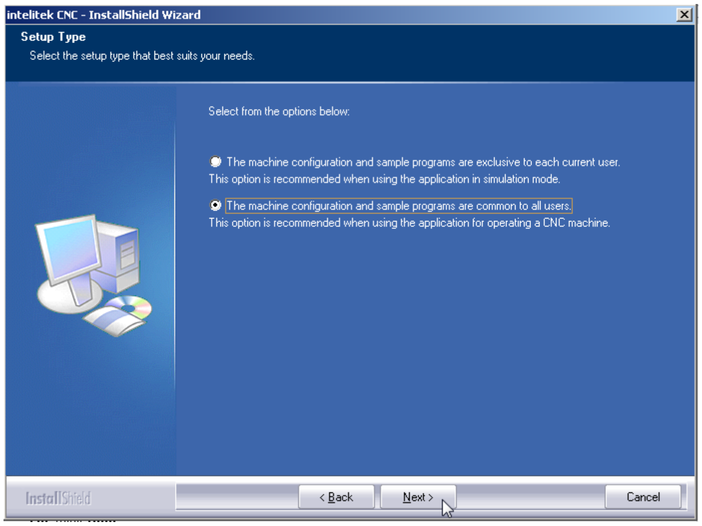
7. Select the machine you will be using. It is important that the machine selected matches the license you have purchased.



8. Click **Next**.

   The Configuration Options screen is displayed.

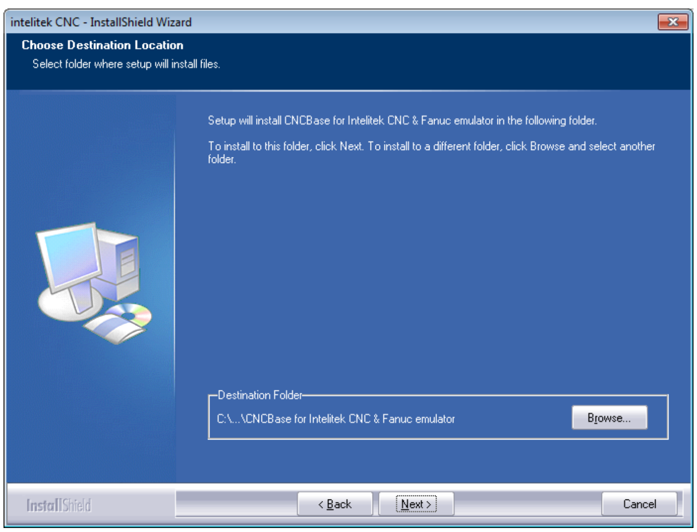
9. Select whether the configuration and sample programs are to be exclusive to each user (first option) or common to all users (second option). The first option is recommended when running software in simulation mode. The second option is highly recommended when running software with a physical CNC machine.



10. Click **Next**.

The Choose Destination Location screen is displayed.

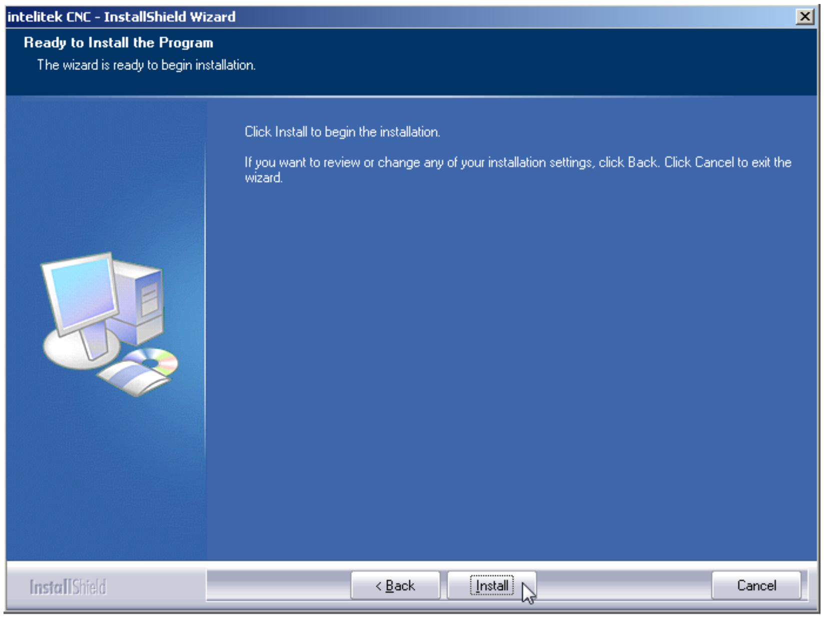
11. If necessary, click **Browse** to change the destination folder.



12. Click **Next**.

The Ready to Install screen is displayed.

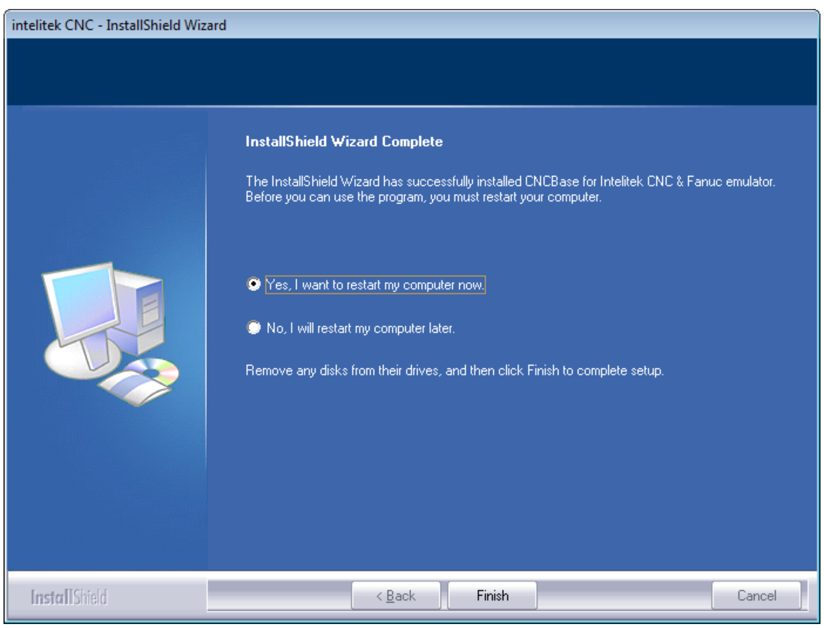
13. Click **Install**.



14. Wait while installation is performed.

The InstallShield Wizard Complete screen is displayed.

15. Select **Yes, I want to restart my computer now**.



16. Click **Finish**.

Your computer will restart and installation will be finalized.

3.3.4. Licensing the Software

For details on licensing your software and managing or transferring your license, refer to the Licensing Help document that can be found in the Books folder of the software installation disk. You can also access the Licensing Help document by clicking the Help button during software registration.

Note that CNCBase does not require registration. If you have purchased CNCMotion, that software does require registration.

3.3.5. Configuring the Software

The installation program automatically configures most software parameters based on the selections you make during installation.

The Configuration Program can be used to:

- Modify selections made during installation.
- Configure machine accessories installed.

This section presents instructions for configuring the control software (CNCMotion or CNCBase) on the computer.

Procedure Outline: Configuring the Software			
No.	**Step**	**Section**	**Page**
1	Run the configuration program.	3.3.5.1	20
2	Change configuration settings using the configuration program.	3.3.5.2	22
3	Add optional accessories to the machine.	3.3.5.3	30

3.3.5.1. Running the Configuration Program

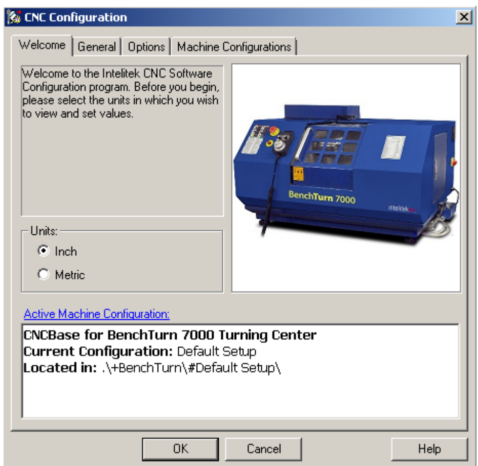
The Configuration Program is launched from your Windows Start menu.

If you try to launch the Configuration Program while the CNCBase/Motion software is open, you will be asked to close CNCBase/Motion first.

Procedure: Running the Configuration Program

1. Ensure that CNCBase/Motion is not currently running.
2. Click the Windows Start button. 
3. Click **All Programs**.
4. Locate and click the **CNCBase/Motion for Intelitek CNC & Fanuc emulator** folder.
5. Click **CNCBase/Motion Configuration**.

   The CNC Configuration window displays.

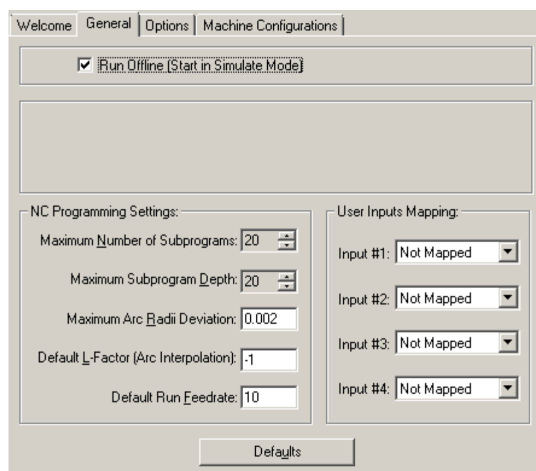


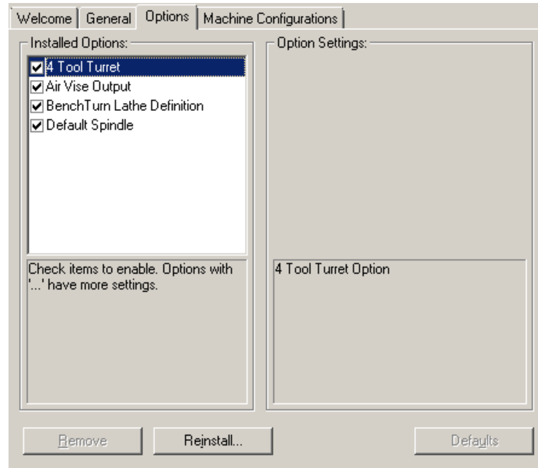
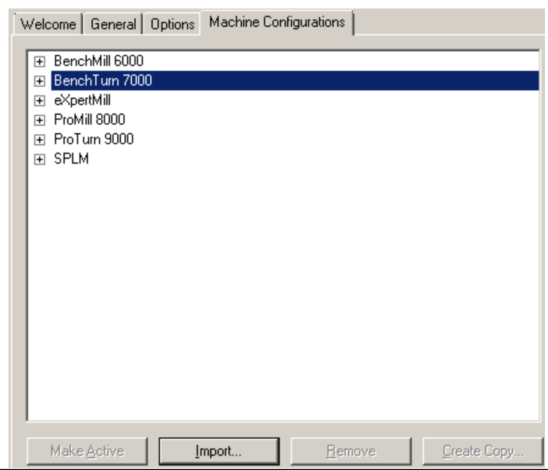
3.3.5.2. Using the Configuration Program

This table summarizes the use of the configuration program.

Tasks: Using the Configuration Program	
Task	**Instructions**
View all available settings	Click the tabs at the top of the window.
Access online help	Click the **Help** button.
Save changes made	Click **OK**. Clicking OK will close the configuration program. Make all required changes before clicking OK.

The table below summarizes the configuration options available on each of the four tabs of the CNC Configuration Program.

Info Table: CNC Configuration Program Tabs		
 		
Tab	**Main Options**	
Welcome	Units (Inch or Metric)	
General	Run in offline (simulation) or online mode. For more information on switching between offline and online modes, see 5.2 Selecting Online or Simulation Mode, pg. 46. NC programming settings User inputs mapping	

Options	Lists installed options and allows you to install addition options. Click **Reinstall** to install others. For more information, see 3.3.5.3 Adding Installed Optional Accessories, pg. 30.	
Machine Configuration	Allows you to select a different machine, and to save and load previously defined configurations.	

3.3.5.3. Adding Installed Optional Accessories

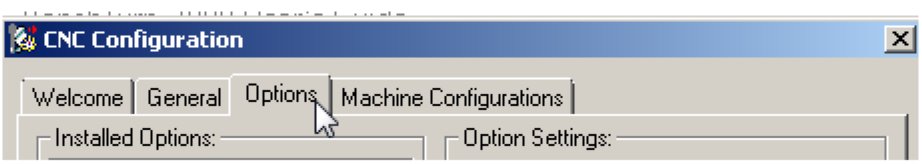
Optional accessories are available for the BenchTurn 7000 (see 2.5 Optional Accessories, pg. 12). After installing the accessory hardware, the control software must be reconfigured.

Detailed instructions are provided in the installation guide supplied with each accessory. General instructions are provided below.

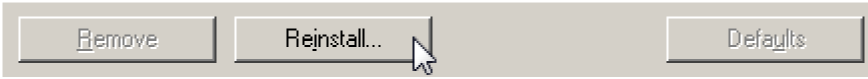
Follow this procedure to configure the control software for a new accessory.

Procedure: Configuring Control Software for New Accessory

1. Run the configuration program (see section 3.3.5.1, pg. 28).
2. Click the **Options** tab.

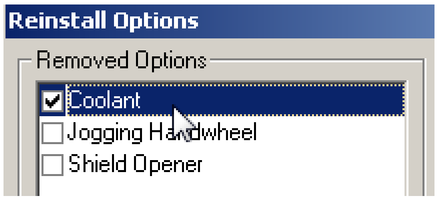


3. Click **Reinstall**.

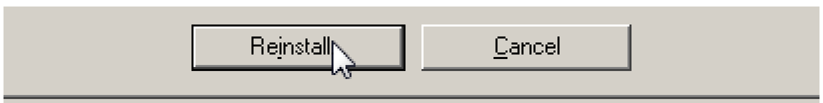


The Reinstall Options window is displayed.

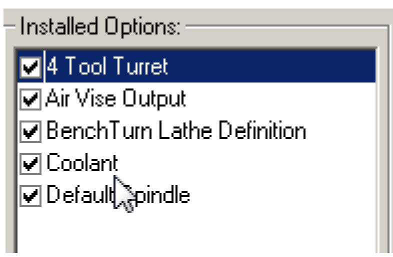
4. Select the option to be added.



5. Click **Reinstall**.



The Reinstall Options window closes. The selected option is now listed in the Installed Options list.



3.3.6. Configuring the IP Address

Before using the hardware, you must configure its IP address on the network.

This utility configures the IP address of the computer's LAN adapter.

Take Note

You must have administrator access to your computer to run the Machine IP Changer utility.

Take Note

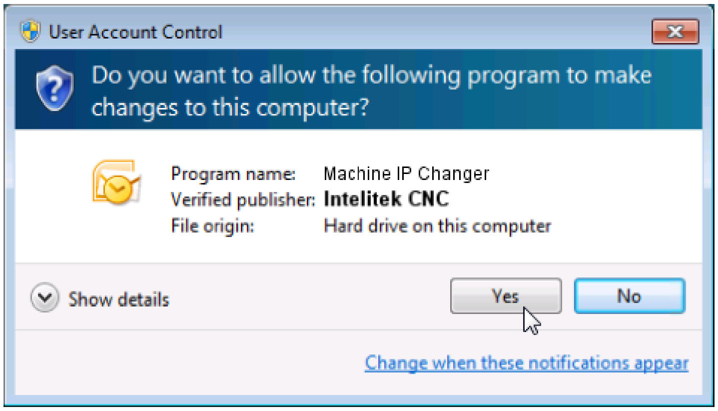
To reconnect to the network over the LAN, you will need to restore the settings of your LAN adapter.

Follow this procedure to configure the IP address.

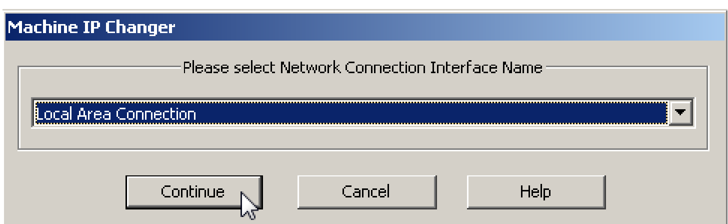
Procedure: Configuring the IP Address

1. Run the Machine IP Configuration utility. To do so, locate the CNCBase/Motion for Intelitek CNC & Fanuc emulator folder and click **Machine IP Configuration**.

2. Click **Yes** if asked for permission.

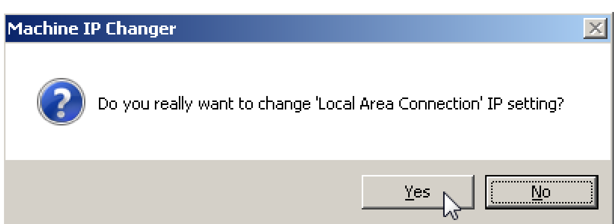


3. From the dropdown list, select the local area network or network card that you wish to use for the CNC machine.

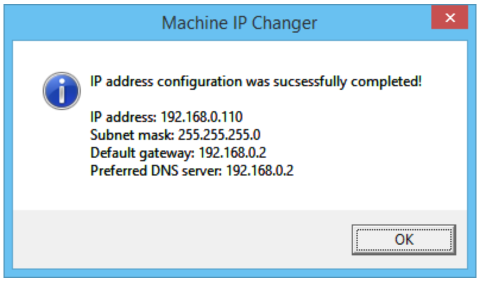


4. Click **Continue**.

5. Click **Yes** when asked for confirmation of your selection.



When the process is finished, Machine IP Changer displays the configuration for all active network connections.



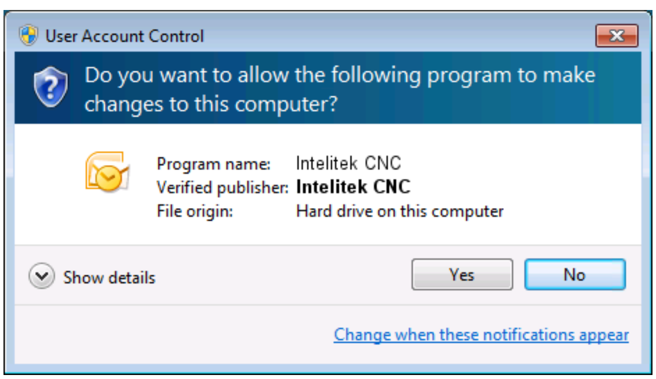
6. Click **OK** to close the program.

3.3.7. Uninstalling the Software

When necessary, follow this procedure to uninstall the software.

Procedure: Uninstalling the Software

1. Click the Windows Start button. 
2. Click **All Programs**.
3. Locate and click the **CNCBase/Motion for Intelitek CNC & Fanuc emulator** folder.
4. Click **Uninstall**.
5. Click **Yes** if the User Account Control message displays.

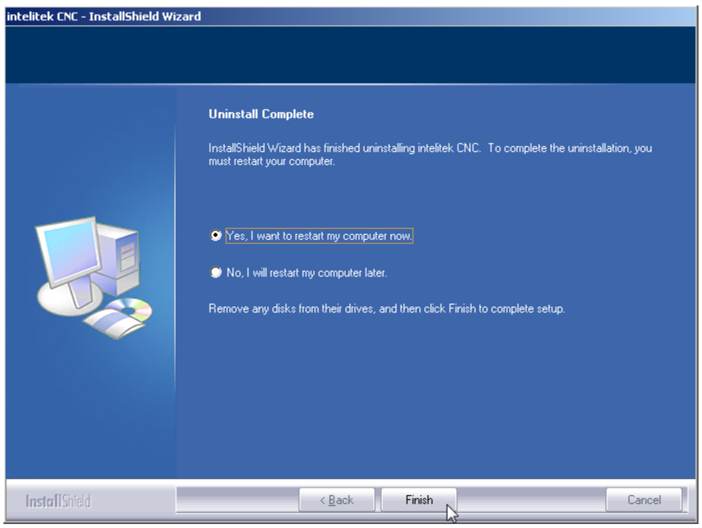


The Uninstall Wizard is displayed, asking for confirmation.

6. Click **Yes** to uninstall CNCBase/Motion.

7. Wait while the software is uninstalled.

   The Uninstall Complete window is displayed.



8. Select **Yes, I want to restart my computer now**.

9. Click **Finish**.

   Your computer will restart and uninstallation will be finalized.

3.4. CONTACTING TECHNICAL SUPPORT

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

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

Info Table: Requirements for Technical Support

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

Take Note	When you call, make sure you have access to both your turning center and your computer. This will allow our technical support representatives to walk through the problem with you.

Technical support contact details:

Info Table: Intelitek Technical Support Contact Details	
Toll-free (U.S. only)	(800) 221-2763
Direct Dial	(603) 413-2600
e-mail	support@intelitek.com
Web site	www.intelitek.com

3.5. RETURNING DEFECTIVE PRODUCTS

Intelitek 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.

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

Follow this procedure to return defective products.

Procedure: Returning Defective Products

1. Contact Intelitek Technical Support and describe the problem.
2. If the Technical Support representative decides that the product is defective and has to be returned, the Technical Support representative will issue a Return Materials Authorization number (RMA). Store this number safely.
3. Pack the product to be returned in its original packaging and crate, as was packed originally.
4. Write the RMA number 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.
5. Have the package returned to Intelitek's offices, as directed by the Technical Support representative.

4. Maintaining the BenchTurn 7000

Product Care	Preventative maintenance of the BenchTurn 7000 is essential for ensuring a long and trouble-free service life.

This section presents instructions for maintaining the turning center and computer.

Task: Maintaining the Turning Center		
Task	**Section**	**Page**
Keep the machine clean.	4.1	36
Maintaining individual lathe components.	4.2	36
Follow a maintenance schedule.	4.3	40
Adjust the heights of tools in the turret.	4.4	41
Maintaining a computer in a shop environment.	4.5	43

4.1. CLEANING THE TURNING CENTER

Keeping your machine clean is the easiest and most important maintenance practice.

Procedure: Cleaning the Turning Center

- Remove all chips from the machine after every use.
- Pay particular attention to chip build-up on the machine bed. Chip build-up can cause wear and damage to the linear bearings.

Product Care	If you clean a component of the turning center that requires lubrication, make sure to lubricate it after cleaning.

4.2. MAINTAINING INDIVIDUAL LATHE COMPONENTS

Each of the turning center's major components must undergo routine maintenance.

This section provides maintenance instructions for each major component.

Maintaining Individual Lathe Components		
Component	**Section**	**Page**
Lathe bed	4.2.1	37
Lathe bed bearings	4.2.2	37

4.2.1. Maintaining the Lathe Bed

The lathe bed, saddle, and ball screw all require constant lubrication to prevent wear and rust. The BenchTurn 7000 is supplied with a one-shot system that simplifies lubrication of these components.

Product Care

Use 15 weight way oil only.

Follow the guidelines below to ensure proper lubrication of the lathe bed.

Guidelines: Lubricating the Lathe Bed

- Operate the one-shot lubricating system before each use.
  To operate, pull on the handle of the one-shot lubricating system and release.
- Keep the reservoir filled with 15 weight way oil.
- Maintain a film of lubricant on the surface of the lathe bed to minimize friction and wear.
- Ensure that all non-painted surfaces on the lathe are coated with oil to prevent rust.

4.2.2. Maintaining the Lathe Bed Linear Bearings

Play in the saddle could indicate that the lathe bed bearings require adjustment. The bearings are factory-adjusted and should be checked at least every three months.

Contact your Intelitek customer service group for maintenance or service instructions.

4.2.3. Maintaining the Ball Screw

The BenchTurn 7000 Turning Center uses pre-loaded ball screws on both axes. The screws are lubricated at the factory with a special long-life, waterproof ball screw lubricant. Additionally, the ball screw is lubricated via the one-shot lubrication system.

One-shot lubrication should be performed before each use of the machine. See 4.2.1 Maintaining the Lathe Bed, pg. 37, for instructions.

4.2.4. Maintaining the Tailstock

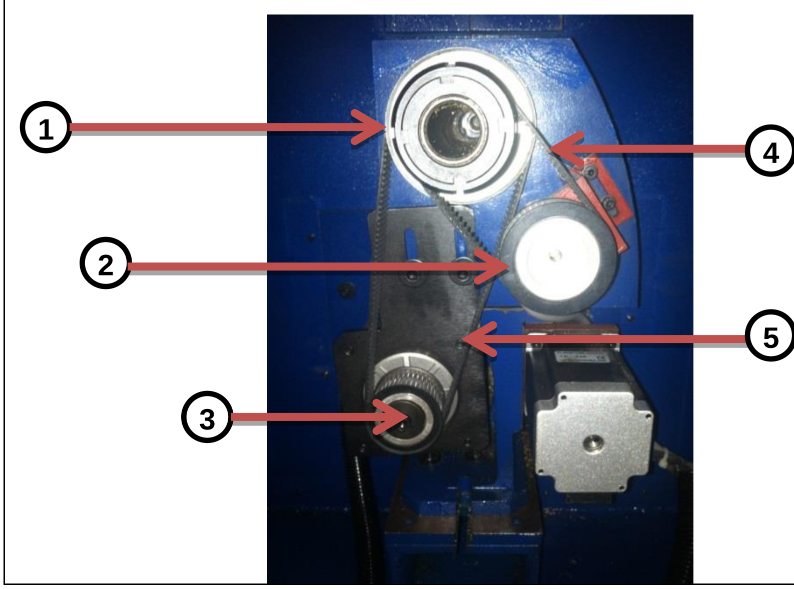
The turning center tailstock requires very little maintenance.

Ensure that the lathe bed is well lubricated at all times to prevent rust. Use 15 weight way oil only.

4.2.5. Maintaining the Spindle Motor and Encoder Belt

The spindle motor and encoder belts will wear out quickly if they become loose. If a belt squeals at slow speeds, it may be loose or worn.

The belts that drive the turning center can be found inside the left side door, as shown below.



1. Spindle
2. Encoder
3. Spindle motor
4. Encoder belt
5. Spindle belt

The spindle motor and an encoder are mounted to the back of the spindle. The spindle motor not only drives the spindle (by way of the spindle drive belt) but also rotates the encoder shaft (by way of the encoder drive belt). The encoder transmits the spindle speed information to the computer.

Product Care

1. Check the encoder shaft and spindle drive belts every two months, or whenever a squealing sound is heard.
2. Replace the encoder shaft and spindle drive belts only with belts supplied by Intelitek. Use of improper belts can cause machine damage and increase wear on components.

Follow this procedure to replace the encoder shaft and spindle drive belts.

Procedure: Replacing the Encoder Shaft and Spindle Drive Belts
1. Loosen the screws holding the red bracket shown in the photo above.
2. Remove the encoder shaft belt.
3. Loosen the two screws that hold the spindle motor mounting plate in place using an Allen wrench.
4. Adjust the spindle motor mounting plate so the spindle drive belt becomes loose.
5. Remove the belt from the spindle motor shaft pulley.
6. Replace the belts.
7. Tighten the spindle drive belt by returning the mounting plate back toward its original position.
8. Tighten the two screws while holding the panel so the belt stays taut.

4.2.6. Maintaining the Axis Drive Belts

The Z axis drive belt should not require frequent adjustment. Check the belt only if you notice a loss in machining accuracy.

4.3. MAINTENANCE SCHEDULE SUMMARY

Follow the maintenance schedule outlined in the table below.

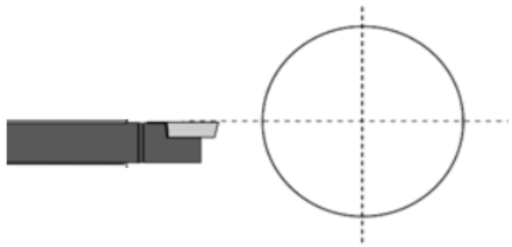
Guidelines: Maintenance Schedule					
	Continuously	**Before Every Use**	**After Every Use**	**Every 2 Months**	**Every 3 Months**
Clean chips from the turning center			X		
Coat exposed surfaces with light oil			X		
Activate the one-shot lubrication system		X			
Maintain the level of 15 weight way oil in the one-shot lubricating system	X				
Check and adjust the lathe bed linear bearings					X
Check spindle and encoder drive belts				X	

4.4. ADJUSTING TURRET TOOL HEIGHTS

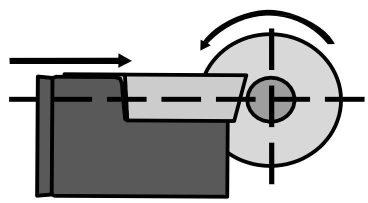
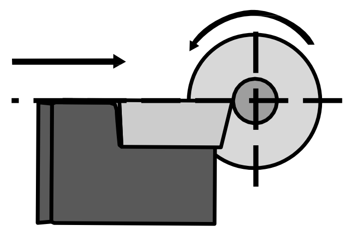
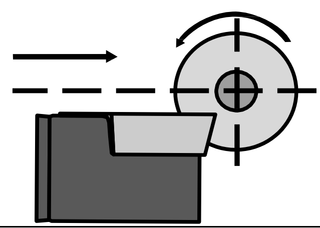
The BenchTurn 7000 is supplied with a 4-position automatic tool turret as standard. This allows your turning center to make automatic tool changes while running NC programs.

It is critical for part tolerances that the tool tip and spindle centerline lie on the same plane, as shown below.

The graphic below illustrates correct alignment between the tool and the spindle centerline.



The graphics below demonstrate the importance of proper alignment.

		
A cutting tool higher than the spindle centerline rubs against the workpiece as it approaches the center of the stock.	A cutting tool properly positioned along the spindle centerline cuts the stock cleanly.	A cutting tool lower than the spindle centerline leaves material on the stock when performing facing cuts.

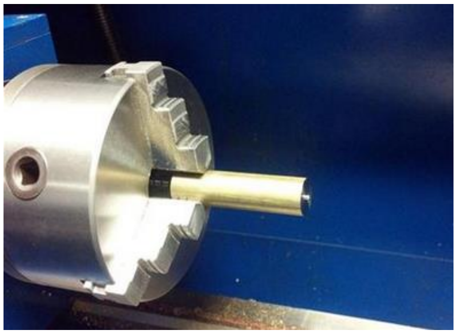
Because different tools may be set up differently, you may need to make some gross adjustments to the tool turret before you can make fine adjustments and start turning parts.

Follow this procedure to adjust the tool turret.

Procedure: Adjusting the Tools in the Turret

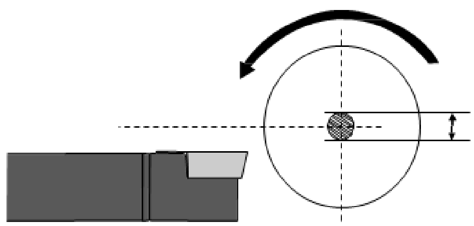
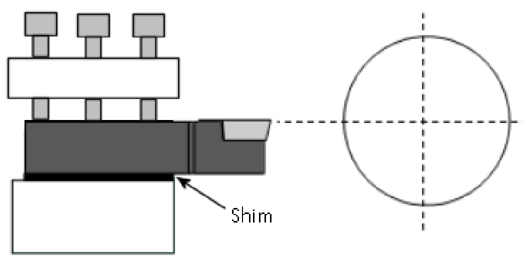
1. Mount all the tools that you intend to use in the tool turret.

2. Mount a workpiece in the chuck.



3. Perform a facing operation using the first tool.

   If material is left in the center of the stock, it is because the tool is below the center-line and must be shimmed up. The amount which must be shimmed is equal to half the diameter of the material left, as illustrated below.

Measure the diameter of the material left after following the instructions above.	
The thickness of the shim placed below the tool should be equal to half the diameter of the material measured.	

4. Once one tool is aligned, the rest of the tools can be aligned by measuring from the cross-slide to the tool tip using a pair of calipers.

4.5. MAINTAINING THE PC IN A SHOP ENVIRONMENT

Maintaining a personal computer and software in a shop environment requires extra precautionary measures. See your owner's manual for maintenance procedures specific to your computer.

Follow the guidelines listed in the table below.

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

5. Using the Control Software

The control software, CNCBase or CNCMotion, is used to control all aspects of machine function, to edit and run NC programs, and to verify those programs in simulation mode. CNCMotion additionally provides 3D simulation of the turning process.

For installation and configuration instructions, see 3.3 Installing the Software.

This section presents the following information:

Section Contents: Control Software

5.1. LAUNCHING THE CONTROL SOFTWARE

CNCBase/Motion can be used with or without the turning center attached to the computer. If you intend to use the turning center, follow the safety guidelines below before launching the software.

Safety

1. The safety door should be closed, and the Emergency Stop button released, before launching the software in on-line mode.
2. The turning center must be powered up and connected to the computer before launching the software in on-line mode.
3. Review the complete safety guidelines in Chapter 1, pg. 1.

Follow this procedure to launch the control software.

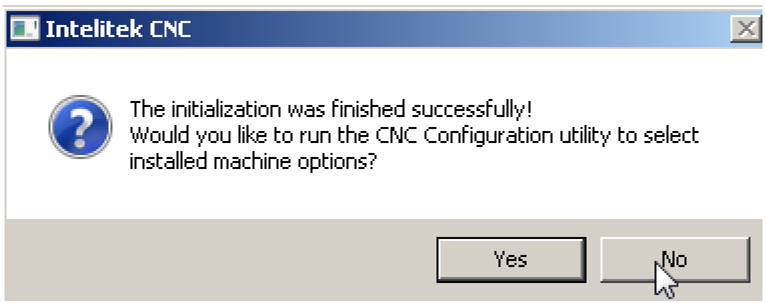
Procedure: Launching the Control Software

1. If you intend to use the turning center, follow the safety information above.
2. Click the Windows Start button .

3. Click **All Programs**.
4. Locate and click the **CNCBase/Motion for Intelitek CNC** folder.

5. Click **CNCBase/Motion for Intelitek CNC**

6. Click **No** if the message below displays**.** This message is only displayed the first time the software is run after installation.



The software opens.

5.2. SELECTING ONLINE OR SIMULATION MODE

Both CNCBase and CNCMotion can be run in two modes:

- **On-line mode**
  For use when controlling the BenchTurn 7000.

Safety

1. The safety door should be closed, and the Emergency Stop button released, before launching the software in on-line mode.
2. The turning center must be powered up and connected to the computer before launching the software in on-line mode.

- **Simulation mode**
  For use without the BenchTurn 7000 connected. In simulation mode, you can write, edit, and verify NC programs as in on-line mode, but you cannot control or send NC programs to the BenchTurn 7000.

Follow the procedure below to toggle between on-line and simulation mode.

Procedure: Selecting On-line or Simulation Mode

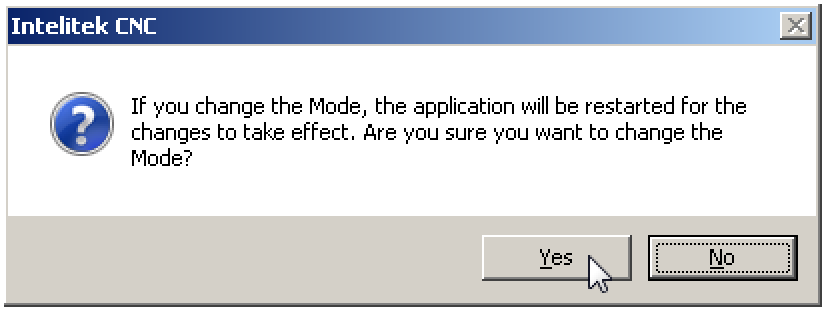
1. Launch CNCBase/Motion.
2. Click **Setup** in the main menu.

   The two modes are listed at the top of the Setup menu. The mode that is currently active is checked.

3. To change the mode, click the unchecked mode.

   A confirmation message is displayed.

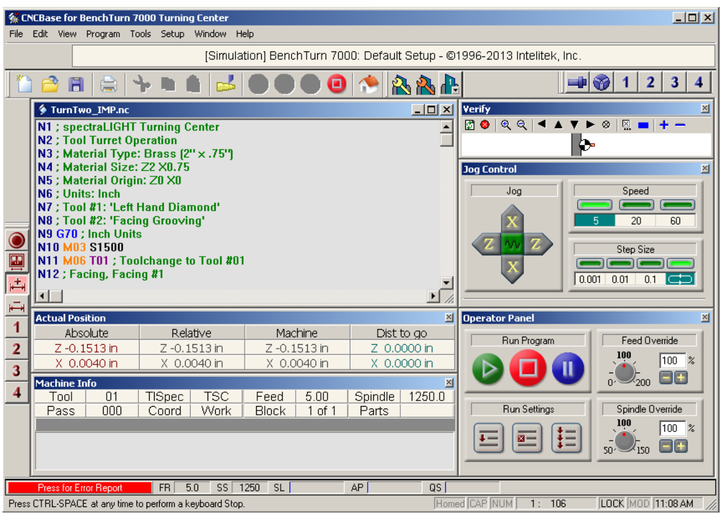
4. Click **Yes**.



The software restarts and opens in the selected mode.

5.3. SOFTWARE INTERFACE

You should become familiar with the main parts of the control software screen prior to use.



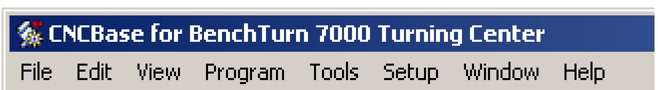
This section provides information on the following screen areas:

5.3.1. Toolbars

This section includes information on the following toolbars:

5.3.1.1. Main Menu

The Main Menu contains all of the menu commands. For an explanation of each menu and its relative commands, refer to the online help.



The table below summarizes all options listed in the Main Menu.

Info Table: Main Menu		
Menu	**Option**	**Function**
File	New	Opens a new, blank Program Editing window. See 5.3.3 Program Editing Window, pg. 60
	Open	Opens an NC program that was saved previously. See 5.5 Opening an NC File, pg. 66
	Close	Closes the currently active Program Editing window.
	Save	Saves the program in the currently active Program Editing window, using its current name.
	Save as	Saves the program in the currently active Program Editing window, under a new name that you specify.
	Print	Prints the NC program in the currently active Program Editing window.
	Print setup	Opens the Print Setup window in which you can set up a printer for printing NC programs.
	Choose machine	For selecting which NC machine configuration is in use. See 3.3.5 Configuring the Software, pg. 27
	Save a copy of current configuration	Saves the current machine configuration, so that you can reload it later. See 3.3.5 Configuring the Software, pg. 27

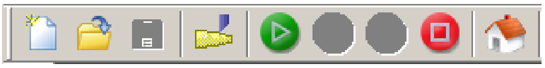
	Exit	Closes the software. If you have made any unsaved changes to an NC program, you will be asked for confirmation before closing.
Edit	Undo	Undo the most recent editing command.
	Redo	Redo the most recent Undo command.
	Cut	Cut selected text to the clipboard.
	Copy	Copy selected text to the clipboard.
	Paste	Paste text from the 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. Note: The Goto Line does not reference the "N" code in the NC file. The line number is counted starting at one and increments in steps of one, regardless of the numbering used in the NC code.
	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 in the Program Editing window.
View	Actual Position	Open or close the Actual Position Window. See 5.3.2.2 Actual Position Panel, pg. 58.
	Absolute Position	Open or close the Absolute Position Window. See 5.3.2.3 Absolute Position Panel, pg. 59.
	Machine Info	Open or close the Machine Info panel. See 5.3.2.4 Machine Info Panel, pg. 59.
	Jog Control	Open or close the Jog Control Panel. See 5.3.4.1 The Jog Control Panel, pg. 62.
	Operator Panel	Open or close the Operator Panel. See 5.3.4.2 The Operator Panel, pg. 64.
	Verify Window	Open or close the Verify Window. See 5.6 Verifying an NC Program, pg. 67.
	Toolbars	Open or close one of the toolbars.

Program	Run/Continue	Start or resume running the current NC program. See 5.7 Running an NC Program, pg. 73.	
	Verify	Verify the current NC program. See 5.6 Verifying an NC Program, pg. 67.	
	Estimate Runtime	Estimate the runtime of the current NC program.	
	Pause	Pause the NC program after the current line of NC code finishes executing. Spindle continues to turn.	
	Feedhold	Immediately pauses the NC program. Stops movement of all axes while spindle continues to turn.	
	Stop	Immediately halts the currently running NC program. Stops both axes movement and spindle.	
Tool	Setup Library	Define tools.	See 5.3.1.3 Tool Menu and Turret Control Toolbar, pg. 54.
	Select Tool	Select a tool for use.	
	Select Tool From	Select a tool to use from a menu.	
	Configure Turret	Assign a tool to each turret station.	
	Operate Turret	Select a turret position to move into cutting position.	
Setup	On-line	Change from simulation mode to on-line.	
	Simulation	Click to change from on-line mode to simulation mode.	
	Set Position	Establish the X and Z position of the tool. See 5.4 Homing, pg. 65.	
	Zero Position	Set the current tool position to X=0,Z=0. See 5.4 Homing, pg. 65.	
	Jog Settings	Establish speed and distance parameters for jogging the tool. See 5.3.4.1 The Jog Control Panel, pg. 62.	
	Run Settings	Establish options for running an NC part program. See 5.7 Running an NC Program, pg. 73.	
	Verify Settings	Establish options for verifying an NC part program. See 6.7 Verifying the Program, pg. 86.	
	Set/Check Home	Establish or check a fixed known position on the machine. See 5.4 Homing, pg. 65.	
	Goto Position	Automatically move the tool to a specific set of coordinates.	
	Units	Select Inch or Metric units of measure.	

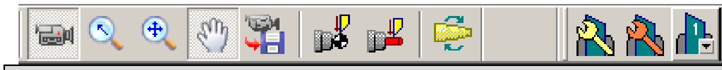
	Coordinate Systems	Define multiple coordinate systems.
	Offsets	Modify the table of Offset values used for certain NC codes.
	Spindle	Specify a spindle speed if you have not used an S code in your NC program.
	Backlash	Define the amount of play in the turning screws.
	Soft Limits	Establish and configure software limits for each axis.
	Preferences	Establish defaults for saving files and security features.
Window	Run and Edit Screen	Loads the preset display configuration for running NC programs: Operator panel, Verify window, Machine info.
	Verify Screen	Loads the preset display configuration for verifying NC programs: Verify window, Machine info.
	Program Screen	Select how multiple NC program windows display: tiled or cascading.
	Close all windows	Closes all software panels and windows, including NC programs.
Help	Help	Opens the built-in Help system.
	Tip of the day	Shows a specific tip to help you take more advantage of the software.
	About	Shows software version and copyright information.

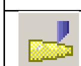
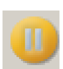
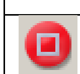
5.3.1.2. Standard Toolbar

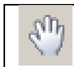
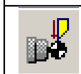
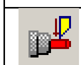
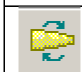
The Standard Toolbar provides easy access to the most often used commands available in the software. The Standard Toolbar includes the buttons below:



When using CNCMotion, the following additional buttons are present:

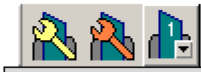


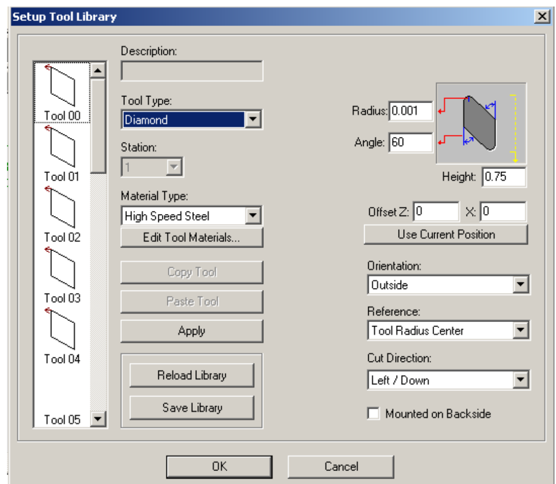
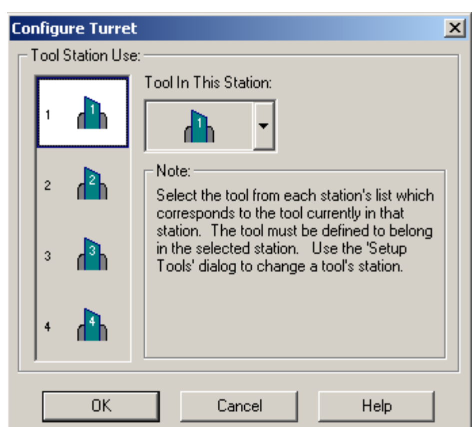
Info Table: Standard Toolbar		
Icon	**Name**	**Function**
	New	Opens a new NC part program file.
	Open	Opens an existing NC part program file.
	Save	Saves the current NC part program file to disk or drive.
	Verify	Verifies the program.
	Run	Runs the current NC part program, and recommences the program after a pause.
	Pause	Causes the currently running program to pause once the current block in the NC program is complete. The program will continue from the next line once the operator resumes operation.
	Feedhold	Pauses the currently running program immediately, even if the current block in the NC program has not been fully executed. The spindle continues spinning. The program will continue from the point at which it stopped once the operator resumes operation.
	Stop	Halts the currently running NC part program.
	Home	Opens the Machine Home window.
Available in CNCMotion Only		
	Show 3D Image	Toggles the 3D display on and off.
	Redirect Camera	Initiates camera redirection: after clicking this icon, click any point on the 3D image to center the camera on that point.
	Follow Me Camera	Initiates camera following mode: after clicking this icon, click any point on the 3D image to center the camera on that point. If that point moves during simulation, the camera will readjust to keep that point at the center of focus.

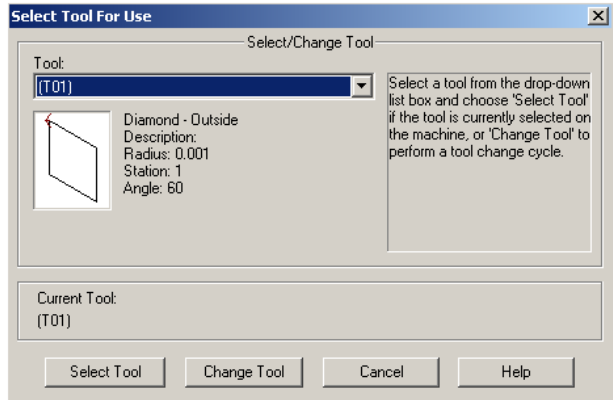
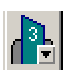
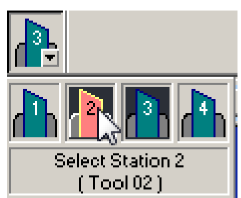
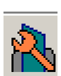
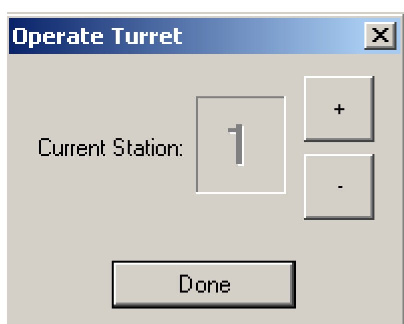
	Drag Image	Initiates image dragging mode: after clicking this icon, click and drag the 3D image to reposition it within the window.
	Save Camera Position	Saves the current viewing angle and position of the 3D window. The next time you open CNCBase/Motion, that saved view will be restored automatically.
	Send Tool to Origin	Moves the tool to the workpiece origin immediately. (Available in Simulation mode only.)
	Send Tool to Point	Displays a cursor: click any point on the workpiece, and the tool will move directly to that point. (Available in Simulation mode only.)
	Reset Workpiece	Returns the workpiece to its original uncut form in the 3D window.

5.3.1.3. Tool Menu and Turret Control Toolbar

The configuration of the tools and the default 4-position tool turret is performed in the windows listed below, all accessible from the Tools menu in the Main Menu. Some are also accessible from the Turret Control Toolbar.

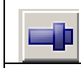


Info Table: Tool Menu and Turret Control Toolbar			
Menu Name	**Turret Control Toolbar Icon**	**Function**	**Window**
Tool Setup Library	-	Allows you to specify the details of up to twenty tools to be used. Specifications include tool type (shape), material, radius, angle, and more.	 Click any tool listed in the left panel, edit its specifications, and click **OK** to save.
Configure Turret		Opens the Configure Turret window. In this window you specify which tool is positioned in each of the tool turret locations. The tools must first be defined using the Setup Tool Library window.	 Select a tool station from the *Tool Station Use* drop-down list. Select the tool that is positioned in that station from the *Tool in This Station* drop-down list.

Select Tool for Use	-	Opens the Select Tool for Use window. This window allows you to select a different tool to be moved to turning position.	 The Tool drop-down list lists all tools defined in Configure turret window. If you select a tool that is: • Currently configured within the tool turret (in the Configure Turret window), then click **Change Tool**. The tool turret will reposition itself so that the selected tool will now be in the machining position. • Not configured within the tool turret, click **Select Tool**. The Configure Turret window is displayed.
Select Tool From		Indicates which tool station is currently in the turning position, and allows you to select a different station to be moved to turning position.	 The number displayed in the toolbar icon indicates which station is at turning position. You can select a different station to be moved to turning position.
Operate Turret		Opens the Operate Turret window. Use this window to command the turret to change position.	 Change which station is in the turning position using the + and – buttons.

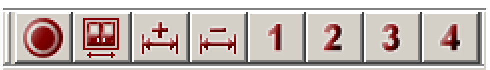
5.3.1.4. Outputs Toolbar

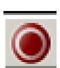
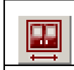
The Outputs toolbar is an active toolbar. It provides switches to supply power to the spindle, and to the Accessory outlets on the right side of the BenchTurn 7000. Switches for Robotic outputs 1 through 4 are also provided. Power is ON when the buttons are depressed.

Info Table: Outputs Toolbar		
Icon	**Name**	**Function**
	Spindle Output	Turns the spindle on/off.
	Output 1 to Output 4	Clicking a numbered Output button provides 24V power to the output of that number on the right side of the machine.

5.3.1.5. Inputs Toolbar

The Inputs Toolbar is an inactive toolbar. It provides information only on the state of the Emergency Stop, the safety door, and the limit switches. Indicators for robotic inputs 1, 2, 3, and 4 are also provided. An input is active (on) when the button is depressed.



Info Table: Inputs Toolbar		
Icon	**Name**	**Function**
	E-Stop	Indicates when the Emergency Stop is pressed.
	Safety Door	Indicates when the safety door is open.
	Negative Limit	Indicates when the negative X-axis proximity switch is on.
	Positive Limit	Indicates when the positive X-axis proximity switch is on.
	Robot Inputs 1 to 4	Indicates when an input is received at one of the four robot input ports.

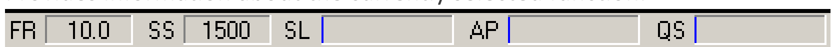
5.3.2. Information Areas

This section presents the following information:

Section Contents: Information Areas		
Item	**Section**	**Page**
Status Bar	5.3.2.1	57
Actual Position Panel	5.3.2.2	58
Absolute Position Panel	5.3.2.3	59

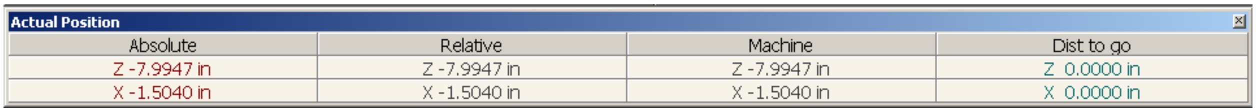
5.3.2.1. Status Bar

The Status bar provides status information on the NC program in progress, the software, and the computer.

Info Table: Status Bar	
Item	**Description**
Left Side	Provides information about the currently selected function. 
FR	Shows the current feed rate.
SS	Shows the current spindle speed.
SL	Shows spindle load.
AP	Shows relative air pressure.
QS	Shows queue status.
Right Side	Provides various status information.  When the indicator is dimmed, the function is in the off condition. For example:  The program has been modified  The program has not been modified.
Homed	The turning center is homed / not homed.
CAP	The Caps Lock key is activated / not activated.
NUM	The Num Lock key is activated / not activated.
(16: 106)	The current line and total number of lines in the program.
LOCK	The current NC part program is locked for editing / not locked for editing.
MOD	The current NC part program has been modified / has not been modified.
(5:07 PM)	The current time according to your computer.

5.3.2.2. Actual Position Panel

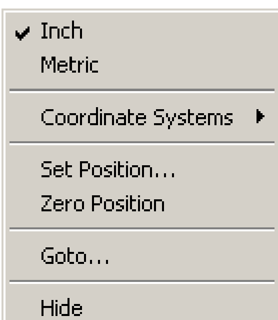
The Actual Position panel provides information on the current X and Z coordinates of the tool position. The units of measure in the Actual Position window are determined by the Units command under the Set-up menu.



The Actual Position Window displays the current position of the machining tool in four coordinate systems.

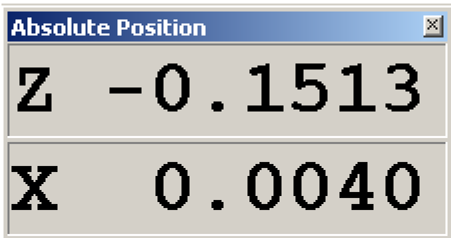
Info Table: Absolute Position Panel	
Column	**Displays**
Absolute	The position of the tool in the current coordinate system.
Relative	The position of the tool relative to the Work coordinates.
Machine	The position of the tool relative to the machine's home position.
Dist to go	The distance remaining until the end of the current line of code (if a program or NC code is currently running).

Right clicking the Actual Position window provides other options such as Set Position, Zero position, Goto and Hide.



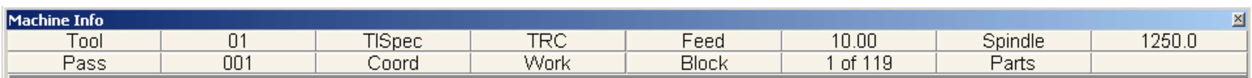
5.3.2.3. Absolute Position Panel

The Absolute Position Window displays the absolute position of the tool in the currently selected coordinate system. This information is identical to the information contained in the first column of the Actual Position Window. The Absolute Position is often the only coordinate information a user is interested in, and is therefore also available independently.

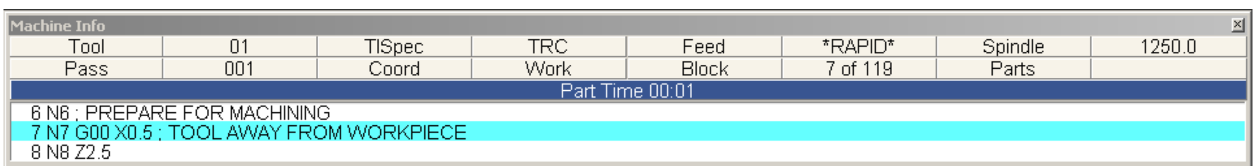


5.3.2.4. Machine Info Panel

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



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



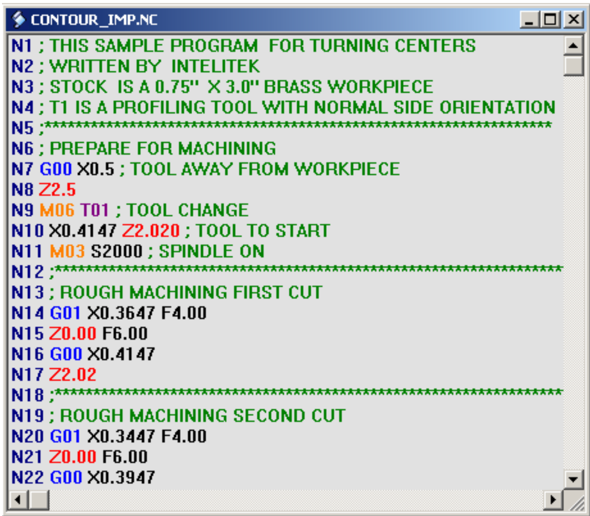
The Machine Info panel includes the following information.

Info Table: Machine Info Panel	
Item	**Description**
Tool 02	Shows the tool number currently in use.
TlSpec TRC	Shows whether the tool is referenced according to its Tool Radius Center (TRC), or according to its Theoretical Sharp Corner (TSC).
Feed 20.00	Shows the current feed rate, in inches/min or mm/min.
Spindle 1250.0	Shows the current spindle speed, in RPM.
Pass 001	Shows how many times the program has been run.
Coord Work	Current work coordinates.
Block 1 of 119	Shows the program line number currently being executed, and the total number of program lines.
Parts	Counts how many parts have been made.

5.3.3. Program Editing Window

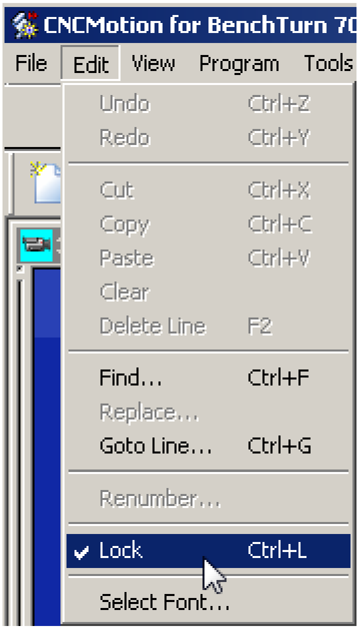
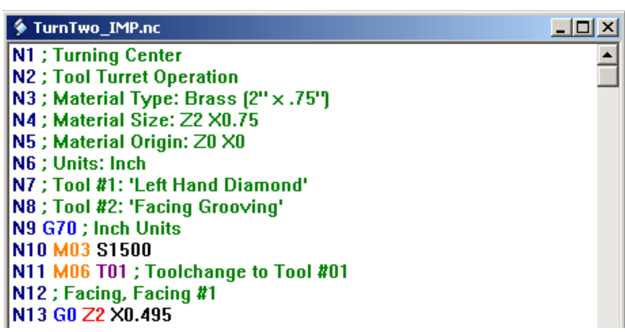
Whenever you open an NC part program file, it is displayed in its own Edit window. You can have multiple Edit windows open at a time.

A sample Edit window is shown below.



By default, the Edit window is locked, meaning that you cannot edit the program within it. A locked Edit window has a grey background, and an unlocked Edit window has a white background.

Follow this procedure to unlock an NC program for editing.

Procedure: Unlocking an NC Program for Editing
1. Click **Edit** | **Lock** in the Main Menu to remove the lock. Alternatively, press **Ctrl-L** on your keyboard.  The Edit window's background color changes to white. Editing is now enabled. 

5.3.4. Control Panels

The BenchTurn 7000 does not have any controls on the machine itself, other than the Emergency Stop and door release buttons. All control operations are performed from the control software.

There are two control panels:

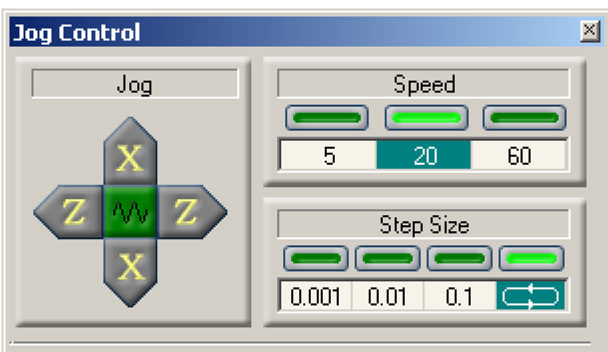
- The Jog Control panel allows you to move the tool in both the X and Z directions, and to control the speed and step size of that motion.
- The Operator panel allows you to run programs, control how programs run, and control the feed rate, and spindle speed overrides.

This section presents the following information:

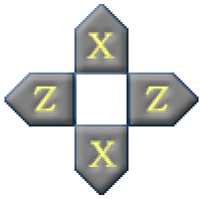
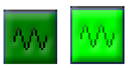
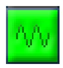
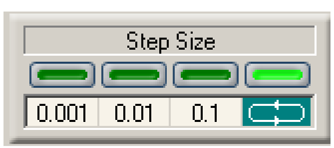
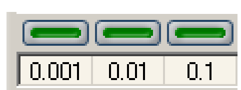
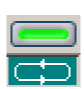
Section Contents: Control Panels		
Section	**Name**	**Page**
5.3.4.1	The Jog Control Panel	62
5.3.4.2	The Operator Panel	64

5.3.4.1. The Jog Control Panel

The Jog Control panel allows you to move the tool in both the X and Z directions, and to control the speed and step size of that motion.

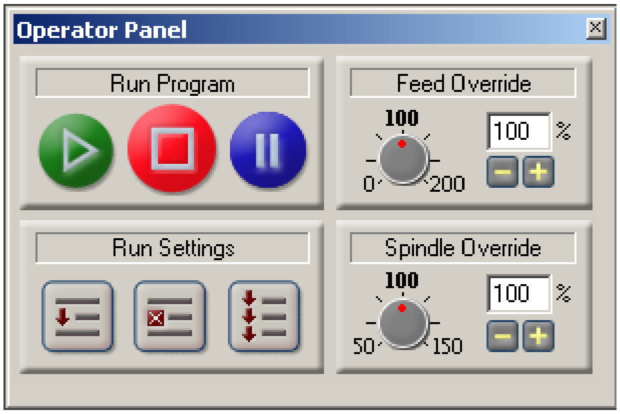


The controls on the Jog Panel are explained below.

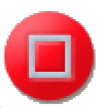
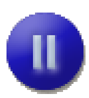
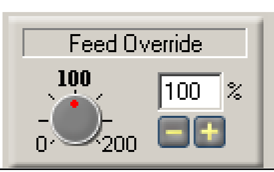
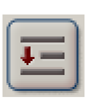
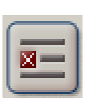
Info Table: Jog Control Panel	
	Pressing the X and Z buttons moves the tool in the X and Z directions, positively or negatively. You can also use the arrow buttons on your keyboard, when the  button is active. The X and Z controls are not displayed when the handwheel accessory is in use.
	Click  to allow the arrow keys on the keyboard to control jog motion. Unclick to prevent the arrow keys from controlling jog motion. This button is automatically activated after clicking one of the X and Z buttons. With a handwheel connected: Click on the green button to activate the handwheel. The arrow buttons will disappear. Click again to deactivate the handwheel and restore the arrow buttons. To deactivate the jog controls, click anywhere in the control software window other than the jog panel.
	The axis will move at the speed selected in the Speed area. Units are in inches/min, or mm/min. You can change these preset speeds in the Settings window, accessed by clicking **Setup** | **Jog Settings** in the Main Menu.
	 Sets the step size, in inches or mm. If one of these buttons is activated, each time you press an X or Z button the axis will make a single motion determined by the step size. You can change these preset step sizes in the Settings window, accessed by clicking **Setup** | **Jog Settings** in the Main Menu.  Sets the jog motion to continuous. If this button is activated, holding down an X or Z button will cause the axis to move continuously at the speed set in the Speed area.

5.3.4.2. The Operator Panel

The Operator panel allows you to run programs, control how programs run, and control the feed rate and spindle speed overrides.



The controls on the Operator Panel are explained below.

Info Table: Operator Panel	
	Runs the program, and recommences the program after a pause.
	Halts the currently running NC part program.
	Pauses the currently running program immediately, even if the current block in the NC program has not been fully executed. The spindle continues spinning. The program will continue from the point at which it stopped once the operator resumes operation.
	Sets the feed rate override. The actual feed rate while turning will be calculated as the feed rate specified in the NC program multiplied by the percentage specified here.
	Sets the spindle speed override. The actual spindle speed while turning will be calculated as the spindle speed specified in the NC program multiplied by the percentage specified here.
	Optional Skip Allows you to execute or ignore any optional skips (M00) you have embedded in the NC program.
	Optional Stop Allows you to execute or ignore any optional stops (M01) you have embedded in the NC program.

	Single Step Causes the NC program to pause after each block is executed. This allows you to check each step of the cutting operation.

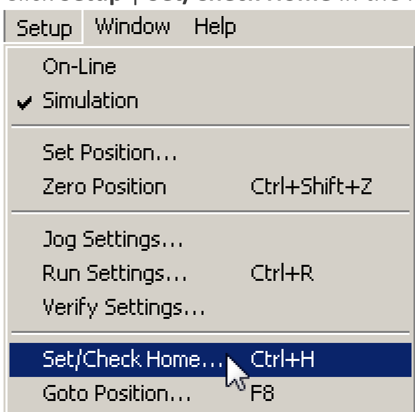
5.4. HOMING

The machine's Home position is a predefined position. The turning center uses this point as a reference for all machine coordinate movements. This allows you to use the Soft Limits and Coordinate Systems commands (under the Setup Menu) to move the turning center consistently to the same location.

Follow this procedure to home the machine.

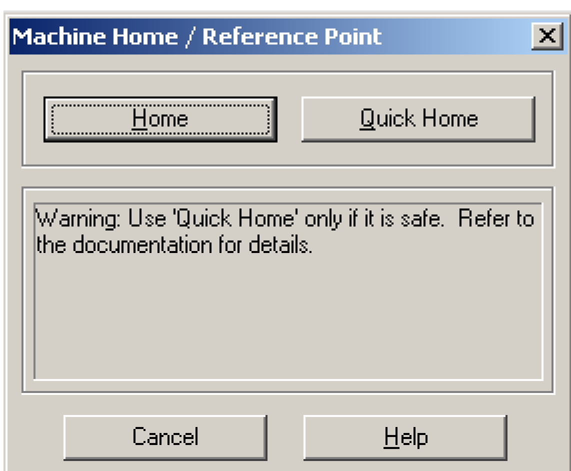
Procedure: Homing

1. Either:
   - Click **Setup** | **Set/Check Home** in the Main Menu, or



   - Press **Ctrl-H** on your keyboard, or
   - Click the Home button  in the Standard Toolbar.

   The Machine Home / Reference Point window is displayed.



2. Click:

   - **Home** to send the machine to the home position at regular speed. (Recommended)
   - **Quick Home** to send the tool to the home position at a rapid speed. Use this option only if you are sure that doing so is safe.

   The machine will move to its home position.

5.5. OPENING AN NC FILE

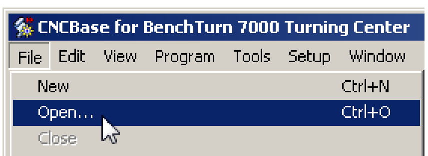
The control software allows NC files to be saved and then opened again at a later time. In addition, the control software is supplied with a number of sample NC files.

Follow this procedure to open a sample NC file.

Procedure: Opening a Sample NC File

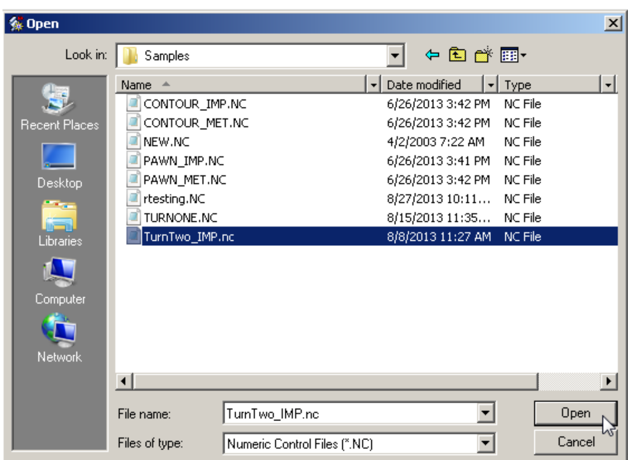
1. Launch CNCBase/Motion (see 5.1 Launching the Control Software, pg. 44).

2. Click **File | Open** in the Main Menu.

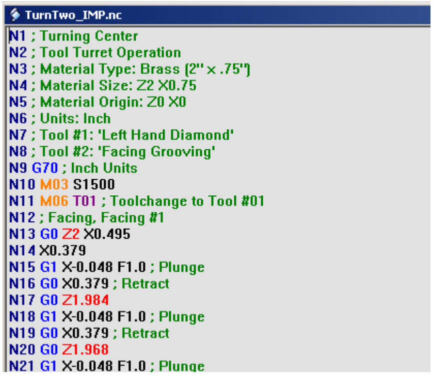


   The Open window is displayed.

3. Select the program and click **Open**.

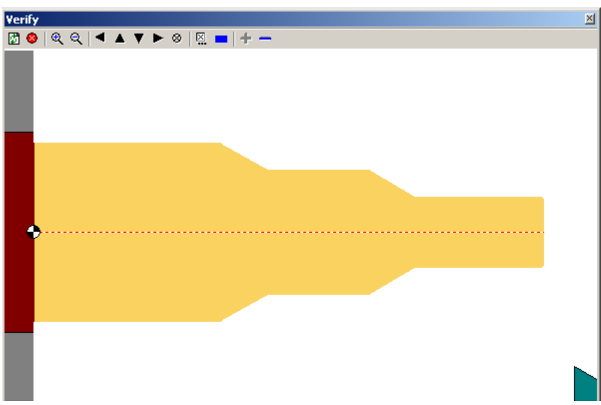


The NC program is displayed.



5.6. VERIFYING AN NC PROGRAM

Tool path verification allows you to check for programming errors before actually running the part program on the Turning Center. The Verify Window displays a 2D simulation of your part program.

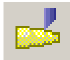


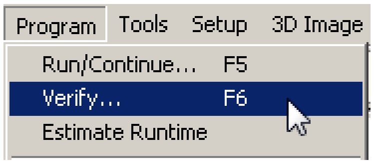
This section presents the following information:

5.6.1. Launching Verification

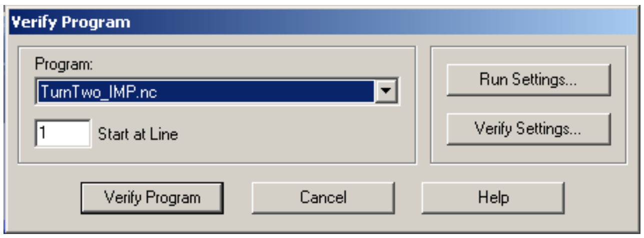
Follow the procedure below to launch verification.

Procedure: Launching Verification

1. Either click the Verify icon  on the Standard Toolbar, or click **Program** | **Verify** on the Main **Menu,** or press **F6** on your keyboard. .

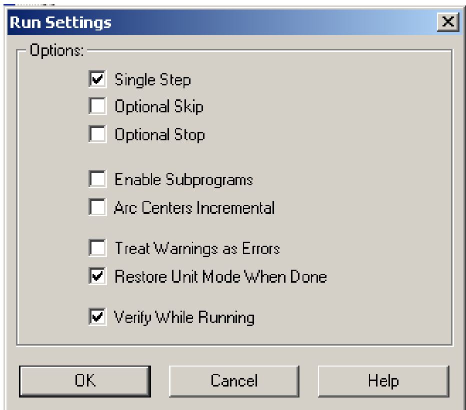


The Verify Program window is displayed.

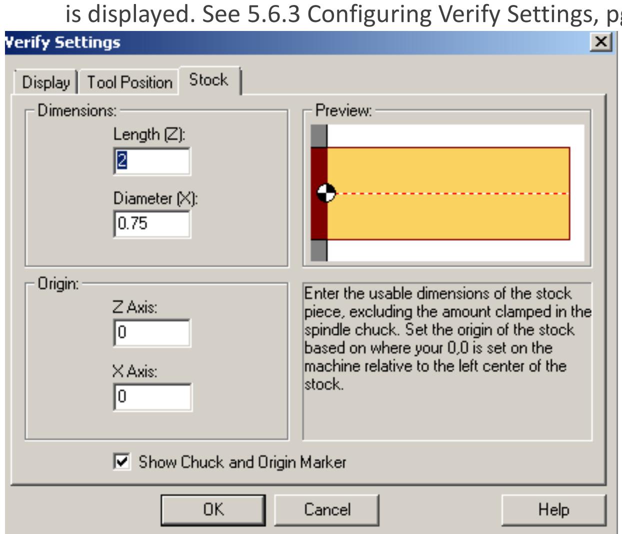


2. Click:

- **Verify Program** to commence verification in the Verify window.
- **Run Settings** to open the Run Settings window. The settings here specify how the program will run. See 5.6.2 Modifying Run Settings, pg. 69.



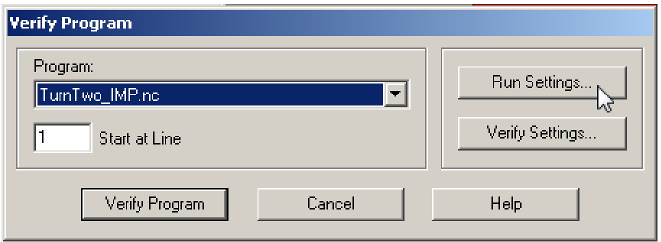
- **Verify Settings** to open the Verify Settings window. The settings here specify how the verification is displayed. See 5.6.3 Configuring Verify Settings, pg. 71.



5.6.2. Modifying Run Settings

The settings in the Run Settings window specify how the program will run.

The Run Settings window is accessed by clicking **Run Settings** on the Verify Program window.



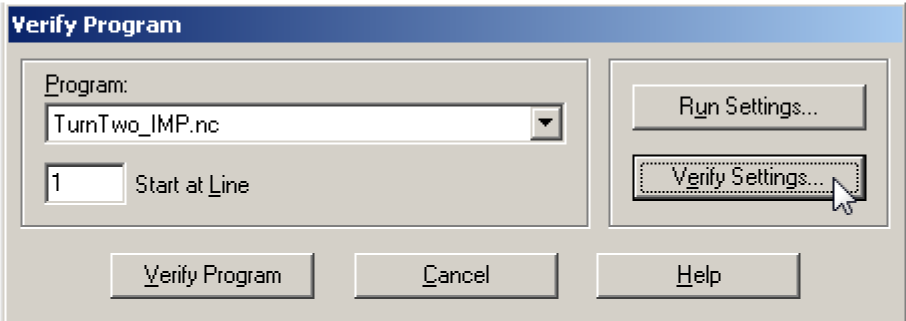
The settings available in this window are described below.

Info Table: Run Settings Window	
Single Step	Allows you to run the program one line at a time, pausing after each line is executed. (Click the Resume button  to continue program operation.)
Optional Skip	Allows you to execute or ignore any optional skips (M00) you have embedded in the NC program.
Optional Stop	Allows you to execute or ignore any optional stops (M01) you have embedded in the NC program.
Enable Subprograms	Must be checked if the program uses subprograms. If this option is disabled, M98 (Call to subprogram) commands generate an error.
Arc Centers Incremental	Specifies the Fanuc mode as the default mode for programming arc centers, in which arc centers are always incremental. When this box is unchecked, 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. To override the default, place the Incremental Arc Centers (%) or Absolute Arc Centers ($) code in the first line of the NC file.
Treat Warnings as Errors	When this item is checked, any warning will halt the program, resulting in a program stop. When motion is stopped, all outputs are turned off. This command is used for special applications, such as laser welding, where you do not want any unexpected pauses in the program execution.
Restore Unit Mode When Done	Restores the original unit mode (inches or metric) regardless of the units specified in the current NC program. Check this box 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.
Verify While Running	When this box is checked, the Verify window will display the program verification while the program is running.

5.6.3. Configuring Verify Settings

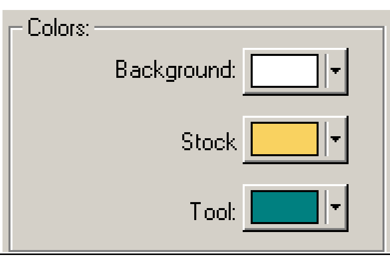
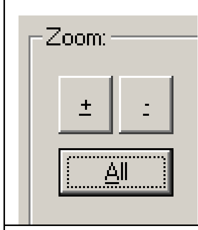
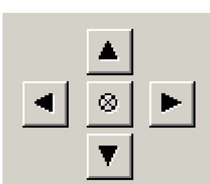
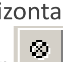
The settings in the Verify Settings window specify how the verification is displayed

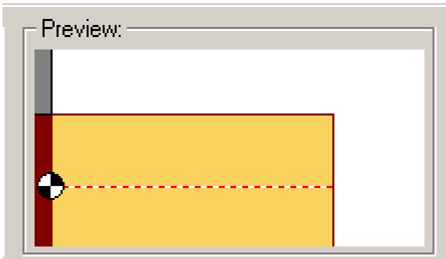
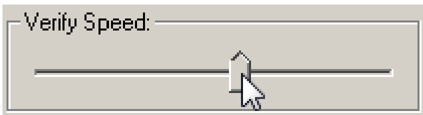
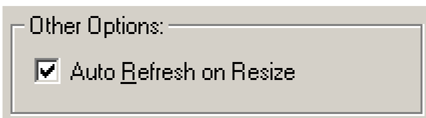
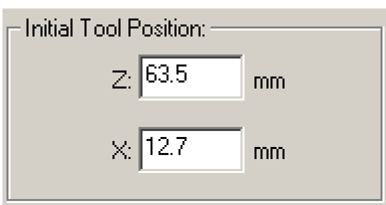
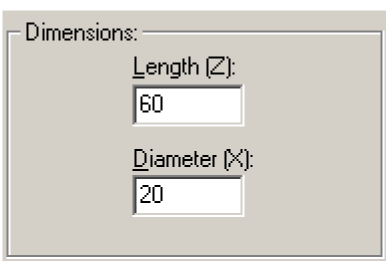
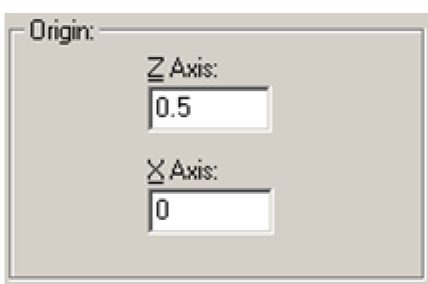
The Verify Settings window is accessed by clicking **Verify Settings** on the Verify Program window.



The Verify Settings window consists of three tabs.

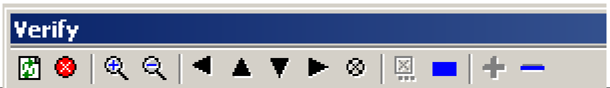
The settings available in this window are described below, arranged by tab.

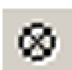
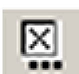
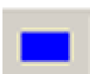
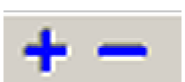
Info Table: Verify Settings Window	
Display Tab	
	Use the Colors controls to choose the colors of the background, stock, and tool in the Verify Window.
	Use the Zoom + and - controls to alter the displayed size of the workpiece in the Verify window.
	Use the repositioning controls to control the vertical and horizontal positioning of the workpiece in the Verify window. The center  button centers the tool in the window.

	All changes are reflected in the Preview window.
	Use the Verify Speed control to alter the speed at which the verify program will be executed in the Verify Window. If you would like to view the verify program in slow motion, move the slider to the left. If you would like to view the verify program at a faster speed, move the slider to the right.
	When Auto Refresh is checked, whenever you resize the Verify window the verification will be performed immediately, automatically.
Tool Position Tab	
	Specifies the X-axis and Z-axis location of the initial tool position, relative to the origin specified on the Stock tab.
Stock Tab	
	Specifies the length and diameter of the stock to be used. The length specified must exclude the length of the stock held within the chuck.
	Moves the origin of the stock in relation to the center of the chuck face.
	Check the Show Chuck and Origin Marker checkbox to make the chuck and origin symbol visible in the Verify simulation.

5.6.4. Using the Verify Window Controls

The buttons on the toolbar inside the Verify window provide quick access to the display settings.



Info Table: Verify Window Controls	
	Replays the verification process.
	Stops the verification process.
	Zooms in/out.
	Repositions the view.
	Centers the view.
	Opens the Verify Settings window.
	Resets the workpiece in the Verification window.
	Speeds up/slows down the verification process.

5.7. RUNNING AN NC PROGRAM

This section provides instructions on how to run an NC program.

Safety

Before running an NC program for the first time, you are advised to follow the tutorial presented in chapter 6 Tutorial: Turning a Sample Part, pg. 76.

Safety

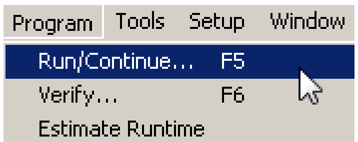
Before running the program:

1. Close the safety door.
2. Wear safety glasses.
3. Review all other safety precautions in 1 Safety Guidelines, pg. 1.
4. Be prepared to press the Emergency Stop button on the machine, if anything goes wrong.

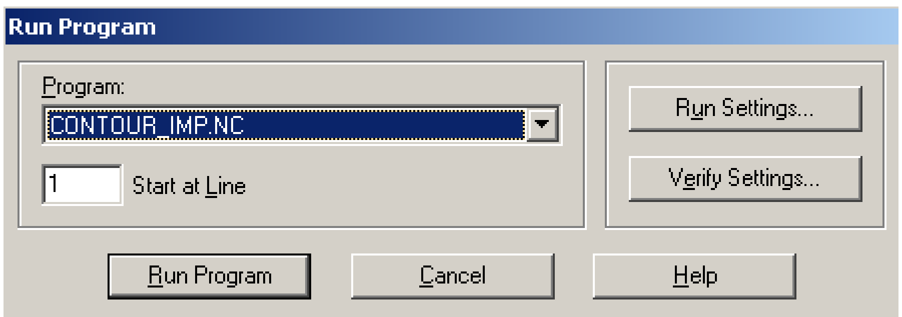
Follow this procedure to run the program:

Procedure: Running the Program

1. Follow the safety instructions presented above.
2. Click **Program** | **Run/Continue** in the Main Menu.



The Run Program window is displayed.

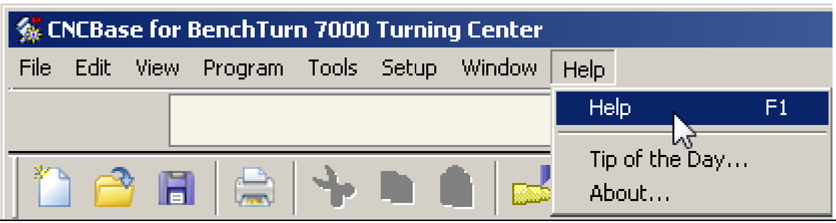


3. Ensure that *Start at Line* is set to line 1.
4. Click:
   - **Run Program** to begin running your program.
   - **Run Settings** to open the Run Settings window. The settings here specify how the program will run. See 5.6.2 Modifying Run Settings, pg. 69.
5. Once the program has ended, press the Emergency button, open the safety door, and remove the finished part.

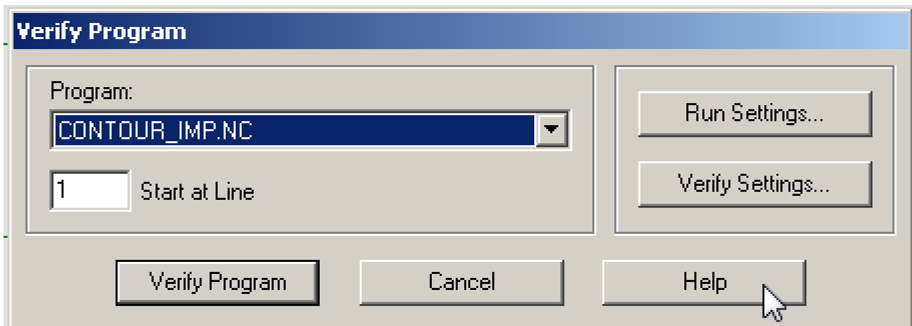
5.8. ACCESSING HELP

The comprehensive online help can be accessed in the following ways from within the software.

<table>
<tr><th>Info Table: Accessing Help</th></tr>
<tr><td>Press F1 on your keyboard.</td></tr>
<tr><td>Click Help in the main menu, and then select Help.



</td></tr>
<tr><td>Click the Help button located on many of the windows to open the relevant Help page.



</td></tr>
</table>

6. Tutorial: Turning a Sample Part

This section provides detailed instructions for turning a simple sample part, covering the entire process from NC program verification through turning a complete part on the BenchMill 7000. The tutorial will follow the procedure below.

Procedure Outline: Tutorial			
No.	**Step**	**Section**	**Page**
1	Review safety procedures.	6.1	76
2	Prepare tools and materials required.	6.2	76
3	Open the sample NC file.	6.3	77
4	Determine the stock size required to turn the part.	6.4	78
5	Adjust the verification simulation settings.	6.5	79
6	Define the tool to be used.	6.6	85
7	Verify the program.	6.7	86
8	Test the program without a workpiece in place.	6.8	87
9	Mount the workpiece.	6.9	93
10	Run the program.	6.10	96

6.1. REVIEWING SAFETY PROCEDURES

Like any other power tool, the BenchTurn Turning Center is a potentially dangerous machine if operated in a careless manner. The importance of safely operating the BenchTurn Turning Center, including the need for protection against personal injury and the prevention of damage to the equipment, cannot be stressed enough.

Safety

Ensure that you are familiar with all safety guidelines in 1 Safety Guidelines, pg. 1, before continuing.

6.2. PREPARING TOOLS AND MATERIALS

For this tutorial you will require the following:

Tools and Materials List: Tutorial

- One 3" (length) x 0.75" (diameter) cylindrical piece of machinable brass, aluminum, Delrin, or wax
- A left-hand profiling tool

6.3. OPENING THE SAMPLE NC FILE

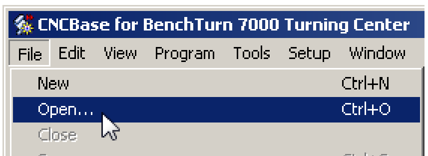
In this step you will launch CNCBase/Motion and will open a sample NC file.

Follow the procedure below to open a sample NC file.

Procedure: Opening a Sample NC File

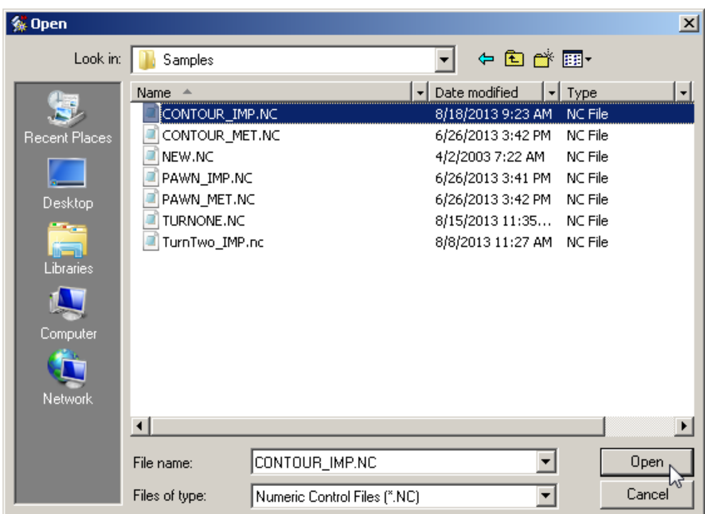
1. Launch CNCBase/Motion. See 5.1 Launching the Control Software, pg. 44.

2. Click **File | Open** in the Main Menu.

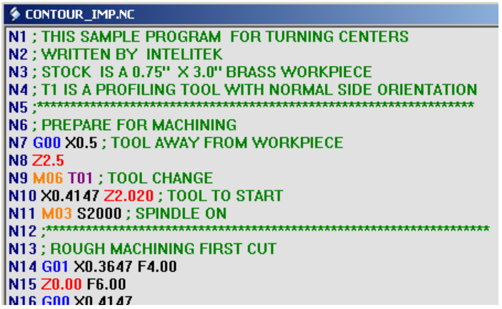


The Open window is displayed.

3. Select **Contour_IMP.NC** and click **Open**.



The NC program is displayed. The grey background indicates that the program is currently locked for editing.



```
CONTOUR_IMP.NC
N1 ; THIS SAMPLE PROGRAM  FOR TURNING CENTERS
N2 ; WRITTEN BY  INTELITEK
N3 ; STOCK  IS A 0.75" X 3.0" BRASS WORKPIECE
N4 ; T1 IS A PROFILING TOOL WITH NORMAL SIDE ORIENTATION
N5 ;**************************************************************
N6 ; PREPARE FOR MACHINING
N7 G00 X0.5 ; TOOL AWAY FROM WORKPIECE
N8 Z2.5
N9 M06 T01 ; TOOL CHANGE
N10 X0.4147 Z2.020 ; TOOL TO START
N11 M03 S2000 ; SPINDLE ON
N12 ;**************************************************************
N13 ; ROUGH MACHINING FIRST CUT
N14 G01 X0.3647 F4.00
N15 Z0.00 F6.00
N16 G00 X0.4147
```

6.4. DETERMINING THE STOCK SIZE

For the Verify window to accurately simulate the NC program, you will have to specify the stock size before running the verification.

Turning stock is defined by two variables:

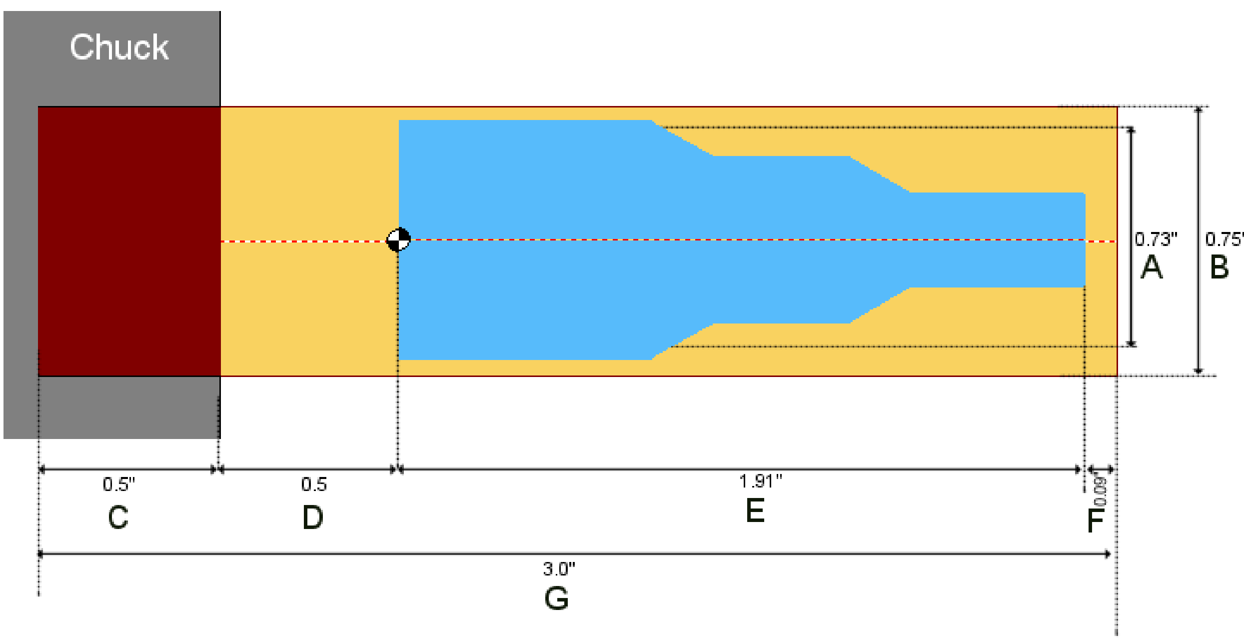
- Its diameter (which extends in the X, or vertical, direction)
- Its length (which extends in the Z, or horizontal direction)

In this step you will only calculate required dimensions; you will not enter them into the software.

Follow the procedure below to determine the stock size.

Procedure: Determining Stock Size

The procedure makes reference to the graphic below. The yellow outline shows the stock, and the blue shape shows the final part.



No.	Description	Letter Indicator	Example Value
1	Examine your part drawing (indicated in blue in the picture) and determine the diameter of the widest section of the part.	A	0.73”
2	Round that value up to the next available stock diameter. This will be the diameter (X value) of your stock. For example, if the maximum diameter of the part is 0.73 inch, you might select a workpiece of 0.75 inch diameter, if that is a standard available diameter.	B	0.75”
3	Examine your part drawing and determine the length of the part.	E	1.91”
4	Add an extra 0.5 inches to that length to provide extra length by which the chuck can hold the workpiece.	C	0.5”
5	Add a short additional length so that the tool will not have to cut right up to the point where the workpiece meets the chuck, risking collision. In this example a value of 0.5” is used.	D	0.5”
6	Round the total calculated value up to the next available stock length. For example, if the sample part will be 1.91 inches long, we add 0.5 inches for the chuck length, and add another 0.5 inches for a safe turning distance from the chuck, giving a required length of 2.91 inches. We then round that up to 3.0 inches, if that is the next available stock length.	G	3.0”

6.5. CONFIGURING THE VERIFY SETTINGS

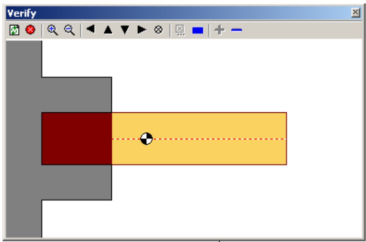
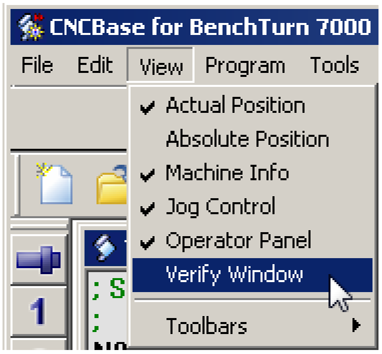
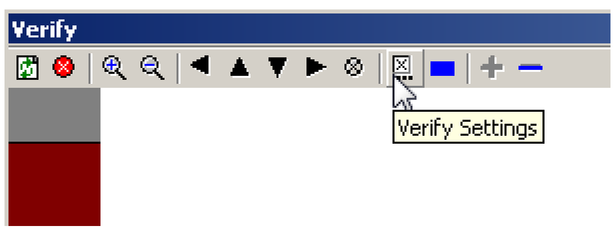
Before you run the verification simulation, you must adjust the verification settings so that the verification simulation will accurately simulate the tool-workpiece combination you will be using.

This section provides instructions for configuring the Verify Settings.

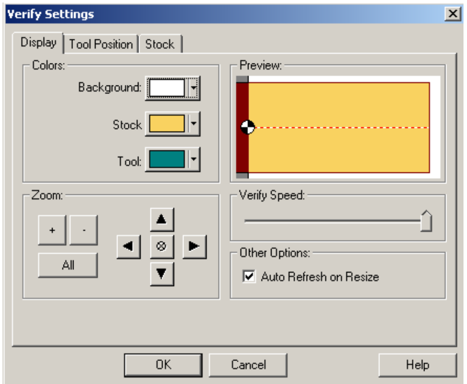
Procedure Outline: Configuring Verify Settings			
No.	**Step**	**Section**	**Page**
1	Open the Verify Settings Window.	6.5.1	80
2	Adjust the display settings.	6.5.2	81
3	Set the position from which the tool must start.	6.5.3	81
4	Specify the size of the stock and location of the origin from which measurements are taken.	6.5.4	83

6.5.1. Accessing the Verify Settings Window

Follow the procedure below to access the Verify Settings window.

Procedure: Accessing the Verify Settings Window
1. Check to see if the Verify window, shown below, is currently displayed on the CNCBase/Motion screen.  If the Verify window is not displayed, click **View**, and click **Verify Window** to open it.  **2.** Click the Verify Settings  button in the Verify window. 

The Verify Settings window is displayed.



6.5.2. Adjusting the Display Settings

Follow the procedure below to adjust the display settings.

Procedure: Adjusting the Display Settings

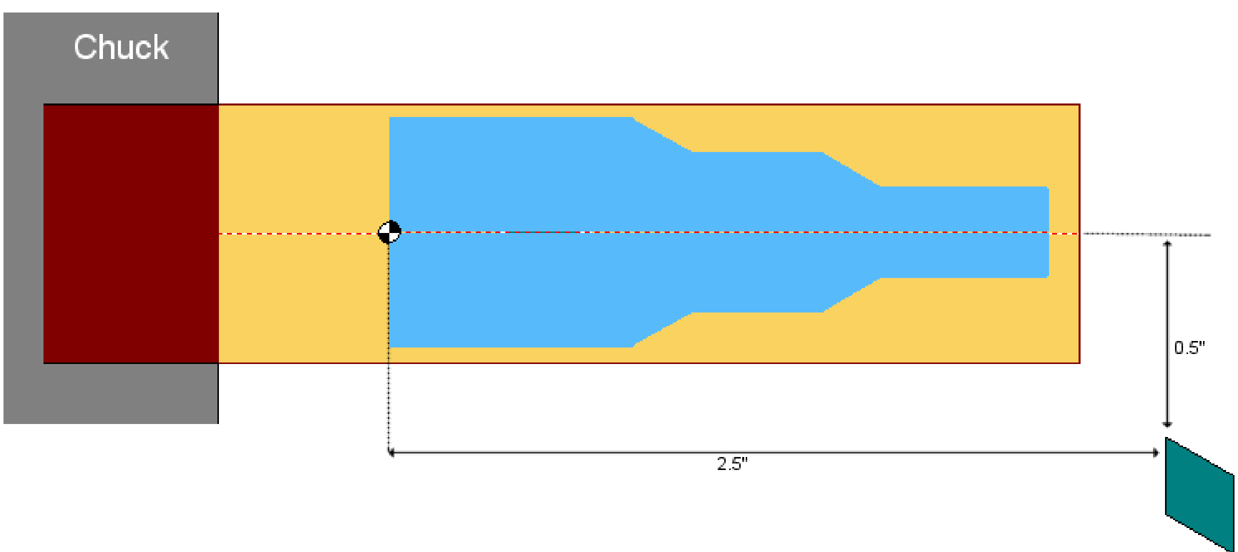
1. Click the **Display** tab. This tab is used to specify the appearance of the Verify animation.

2. Make all required settings. See 5.6.3 Configuring Verify Settings, pg. 71.

6.5.3. Setting the Initial Tool Position

In this step you will specify the tool's initial location in the Verify simulation.

In our example, we would like the tool to start at the position as shown below.

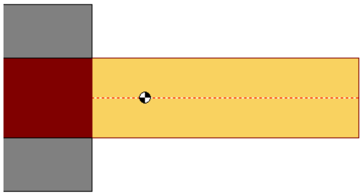
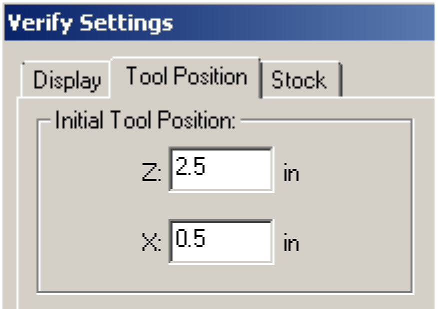


The ⊕ symbol indicates the program origin (X0,Z0).

The tool starting position is thus:

- Z = 2.5”
- X = 0.5”

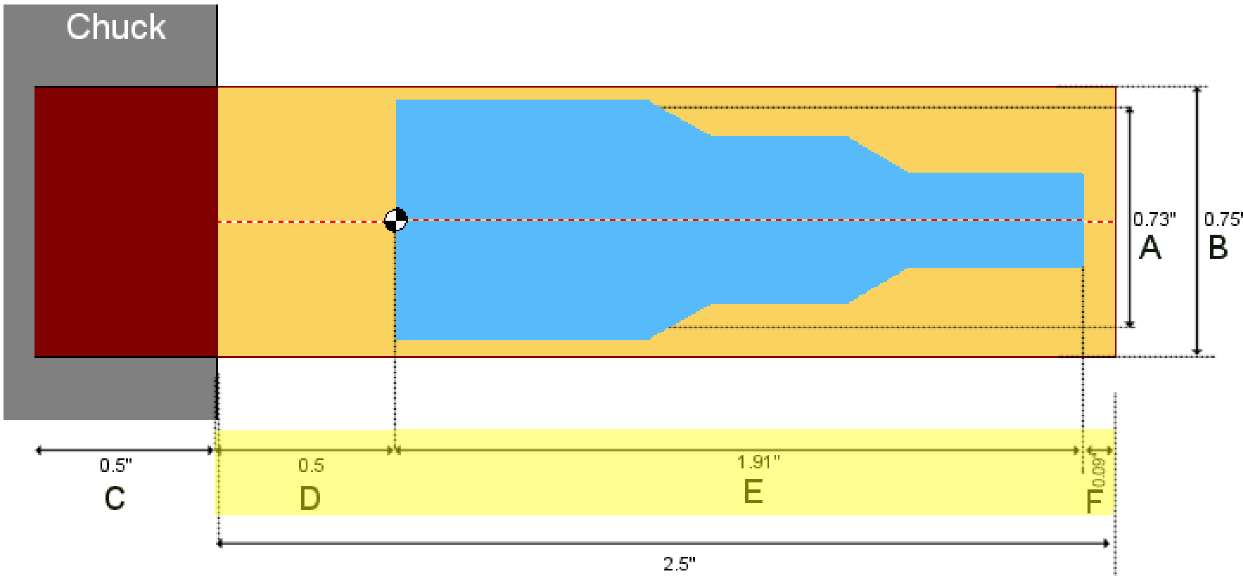
Follow the procedure below to set the tool’s initial position.

Procedure: Setting the Initial Tool Position
1. Click the **Tool Position** tab. This tab is used to specify the initial tool position relative to the origin, which we will later define as the point along the center of the workpiece, 0.5” from the chuck, as indicated by the ⯐ symbol below.  **2.** Set the Z value to 2.5. **3.** Set the X value to 0.5.  The initial tool position is now set.

6.5.4. Setting the Stock Dimensions and Origin

You will next specify the length and diameter of the stock, and then set the origin of the axes to be referenced.

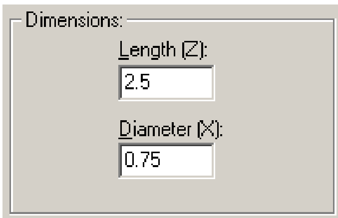
It is important to note that the stock length refers to the length of the stock outside the chuck. In this example, that length is 2.5 inches as shown in the highlighted zone below.



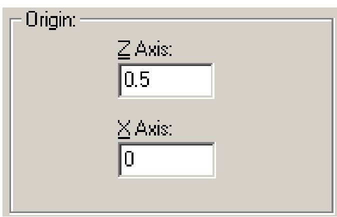
Follow the procedure below to set stock dimensions and origins.

Procedure: Setting Stock Dimensions and Origins

1. Select the **Stock** tab.
2. Enter the stock dimensions for the contour_imp.nc part program. The stock dimensions define the dimensions of the stock outside the chuck.
   - Z=2.5"
   - X=0.75"



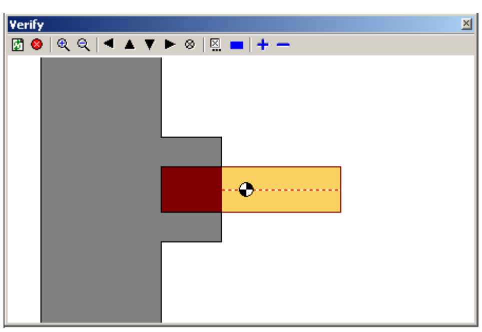
3. Set the Origin of Stock to
   - Z=0.5"
   - X=0"



4. Check the **Show Chuck and Origin Marker** checkbox to make the chuck and origin symbol visible in the Verify simulation.

5. Select **OK**.

   The window closes, and your changes are applied to the workpiece in the Verify Window.



6.6. DEFINING THE TOOL

You will use a diamond shaped profiling tool to turn this part. You will use the parameters for this particular tool for the tool path verification as well.

You will first define the tool and will then specify that tool as the tool to be used during verification.

Follow the procedure below to define the tool for verification.

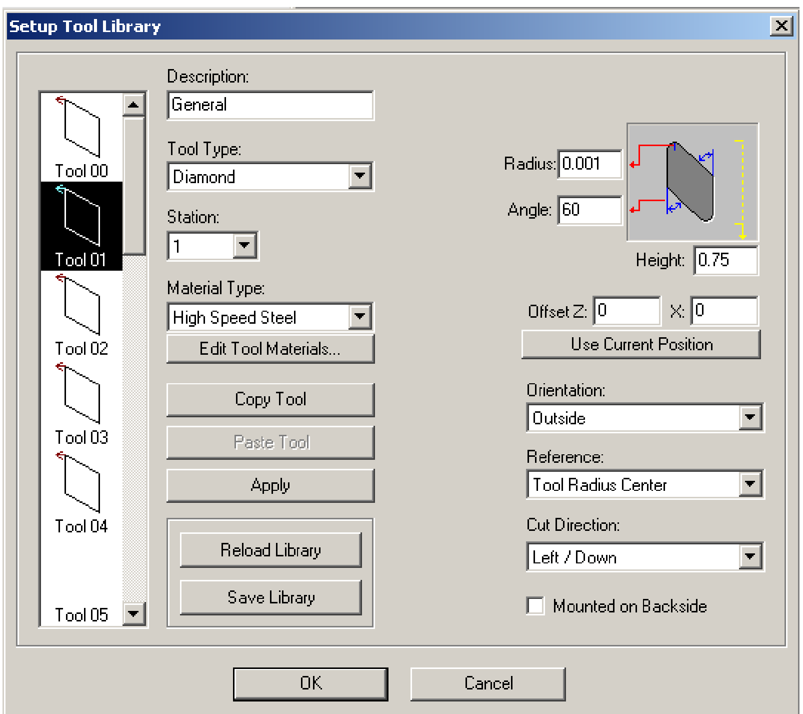
Procedure: Defining the Tool

1. Click **Tools** | **Select Setup Library** in the Main Menu.

   The Setup Tool Library window is displayed. There are already a number of tools defined.

2. Select **Tool 01**, a left hand diamond profiling tool with outside orientation.

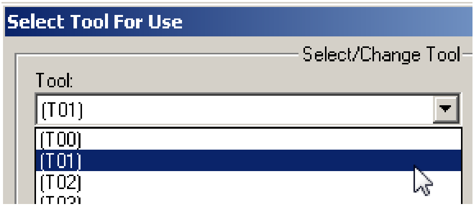
3. Check that the settings for Tool 01 are as shown below. If they are not, modify the settings to match the settings shown.



4. Click **OK** to close the Setup Tool Library.

5. Click **Tools** | **Select Tool** in the Main Menu. The Select Tool for Use window is displayed

6. Select the tool you have just defined, T01, from the *Tool* drop-down list.



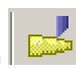
7. Click **Select Tool**.
8. Tool 01 is now selected.

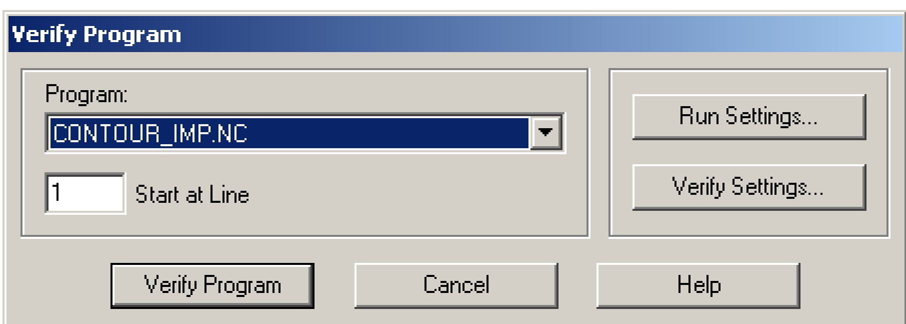
6.7. VERIFYING THE PROGRAM

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

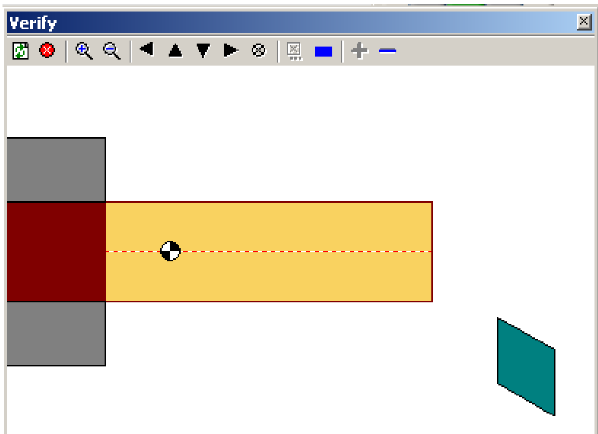
Follow the procedure below to verify the program.

Procedure: Verifying the Program

1. Click **Program** | **Verify** in the Main Menu or click the Verify icon  in the Standard Toolbar.
2. Ensure that Start at Line is set to **1**. This specifies that the program should be run from the first line onward.
3. Click **Verify Program**.



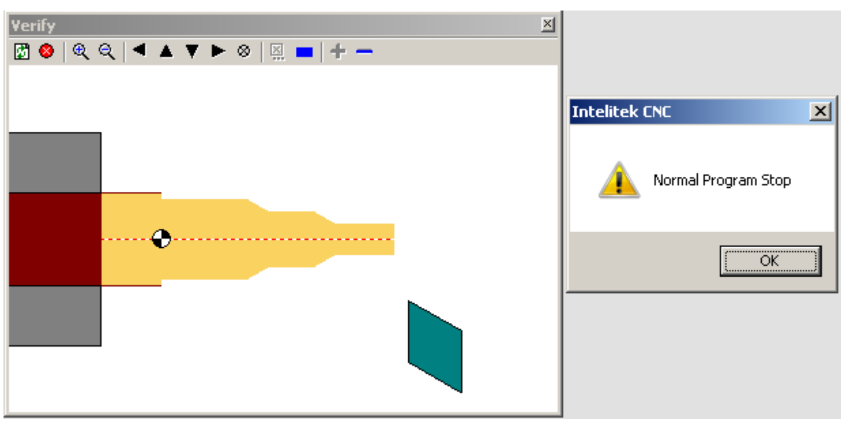
In the Verify window, the cutting tool you specified earlier is now displayed at the initial position you specified.



During verification, a Pause message may be displayed, either because of a programmed pause or for a tool change.

4. Press F5 or click  to continue verification.

   The program is verified. Upon completion the Normal Program Stop window is displayed.



5. Click **OK**.

6.8. PERFORMING A DRY RUN

The Verify process you have just completed verified that the path the tool is to follow will almost certainly not result in any collisions with the workpiece or chuck.

An additional and very important step is to run your NC program on the hardware, without the workpiece loaded. This is called a dry run. The dry run will ensure that all movements make sense and that the tool is in no danger of striking any fixtures of the turning center.

Although the dry run is performed without the workpiece in place, you will need to load the workpiece into the chuck initially so that you can set the point of origin correctly.

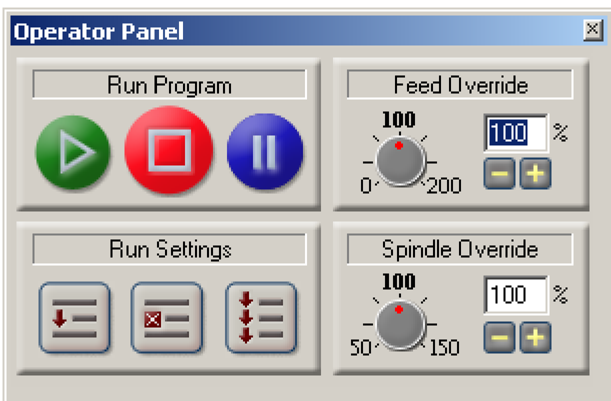
Safety

The Emergency button must be pressed in before starting this procedure.

Follow this procedure to perform a dry run.

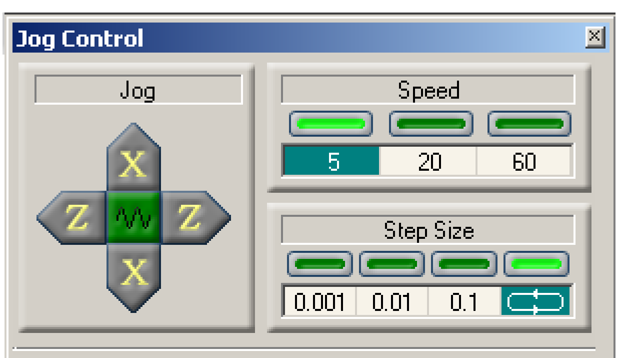
Procedure: Performing a Dry Run

1. Ensure that the Emergency button on the machine is pressed in.
2. Ensure that your diamond profiling tool is in position 1 of the tool turret.
3. Mount the workpiece in the chuck. Your workpiece should be a cylindrical piece, 3.0" in length with 0.75" diameter.
4. Locate the Operator panel on your screen.



If it is not displayed, click **View** | **Operator Panel** in the Main Menu.

5. Locate the Jog Control panel on your screen.



If it is not displayed, click **View** | **Jog Control** in the Main Menu.

6. Use the Jog Control panel to jog the tool to the front right corner of the workpiece. Note the following:

   a. Click the arrow buttons, or press the arrows keys on your keyboard, to move the tool.

   b. To make the tool move continuously while one of the X and Z buttons is pressed:

   i. Click the Continuous Jog button  in the Step Size area.

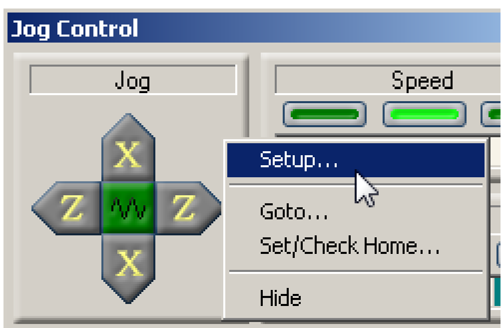
   ii. Select the speed of motion in the Speed area.

   c. To make the tool move a fixed distance each time a Z or X button is pressed, click one of the numbered step sizes in the Step Size area.

   d. To adjust the preset Speed and Step Size values:

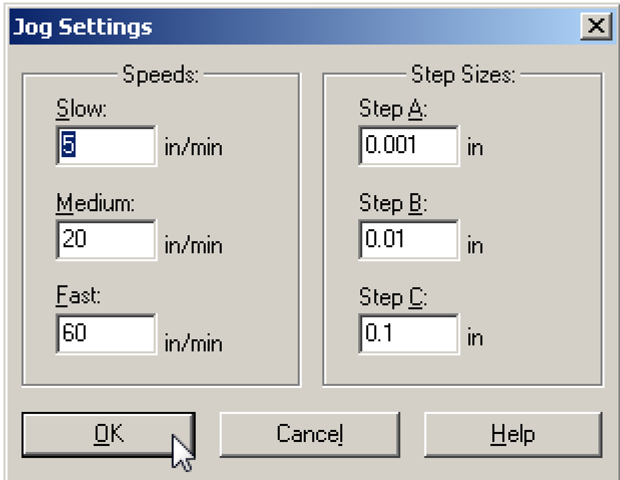
   i. Right-click anywhere in the Jog Control panel. A menu is displayed.

   ii. Click **Setup**.

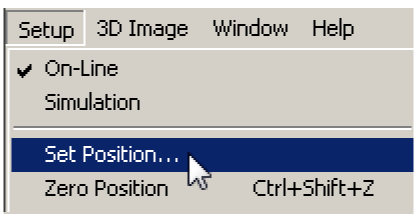


The Jog Settings window is displayed.

iii. Change the settings as required and click OK.

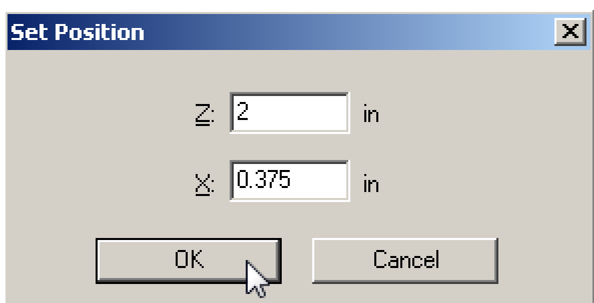


7. Once the tool has reached the workpiece corner as explained previously, click **Setup** | **Set Position** in the Main Menu.

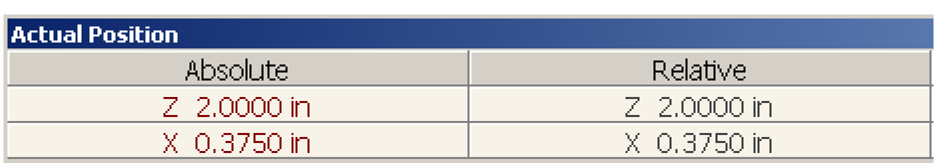


The Set Position window is displayed.

8. Set Z to 2 inches and X to 0.375 inches.



The position values in the Actual Position window will be set accordingly.



9. Jog the tool back and away from the workpiece.

10. Press the Emergency Stop button.

11. Open the safety door.

12. Remove the workpiece.

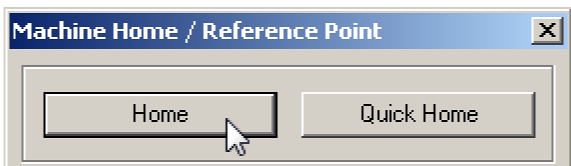
13. Close the safety door.

14. Release the Emergency Stop button.

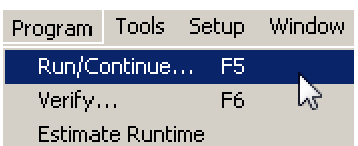
15. Click **Setup** | **Set/Check Home** in the main menu, or press Ctrl-H on your keyboard.

    The Machine Home / Reference Point window is displayed.

16. Click **Home**.

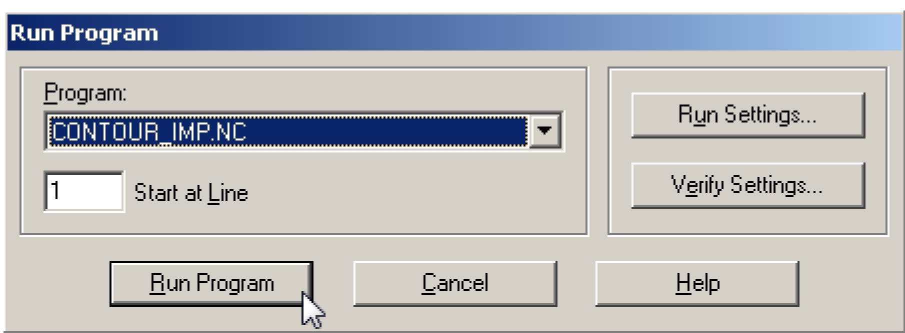


17. Click **Program** | **Run/Continue** in the Main Menu.



    The Run Program window is displayed.

18. Click the **Run Program** button. The machine begins running the program.



Safety	Be prepared to press the Emergency Stop button on the turning center if it looks like a collision may occur.

As the part program runs, observe the tool motion in relation to the chuck, other fixtures in the machine, and the future location of the workpiece. Look for signs of a possible tool crash and be prepared to press the Emergency Stop switch on the turning center. Edit the program as required. When you are satisfied that the tool motions are correct, you can proceed to the next step – mounting the workpiece and turning the part.

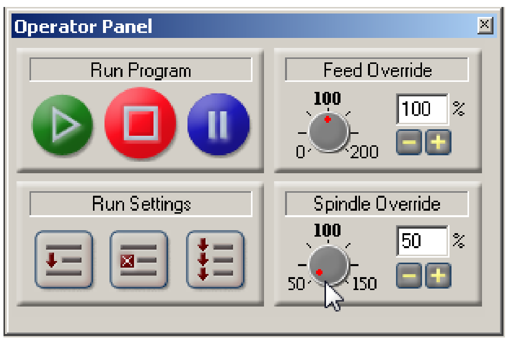
6.9. MOUNTING THE WORKPIECE

Once the NC program has been tested by performing a dry run, the actual part can be machined. First, the workpiece must be mounted.

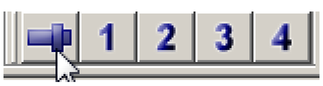
Follow this procedure to mount the workpiece.

Procedure: Mounting the Workpiece

1. Use the Jog Keypad to jog the tool far from chuck.
2. Push the Emergency Stop button.
3. Open the safety door.
4. Mount the 3.0" x 0.75" workpiece in the chuck. Take care to position the workpiece perpendicular to the tool turret.
5. Close the safety door.
6. Release the Emergency Stop button.
7. Set the Spindle Override to its minimum setting in the Operator Panel.



8. Click the Spindle button on the Outputs Toolbar to turn on the spindle.

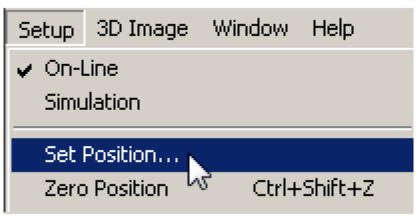


9. Jog the tool slowly until it just touches the workpiece anywhere along the side (length) of the workpiece. Follow the guidelines below:

a. Use Continuous mode (selected in the Step Zone area of the Jog Control panel) to move the tool towards the workpiece, but stop motion before the tool is in range of touching the workpiece.

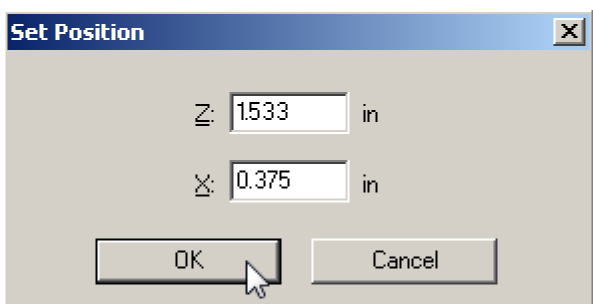
b. Once the tool is near the workpiece, switch to Step mode, using a small step size. Jog the tool, step by step, until it touches the workpiece.

10. Once the tool is just touching the workpiece along its side, click **Setup** | **Set Position** in the Main Menu.



The Set Position window is displayed.

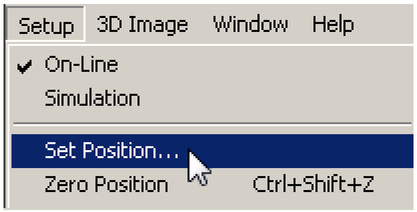
11. Set X to 0.375. Do not change the Z value at this stage.



12. Click **OK**.

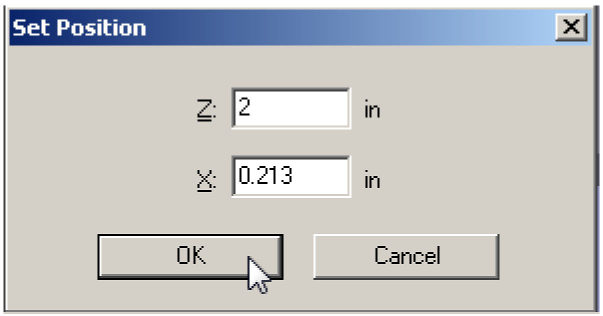
13. Jog the tool slowly until it just touches the workpiece anywhere along its face. Follow the guidelines provided in step 9 above.

14. Once the tool is just touching the workpiece face, click **Setup** | **Set Position** in the Main Menu.



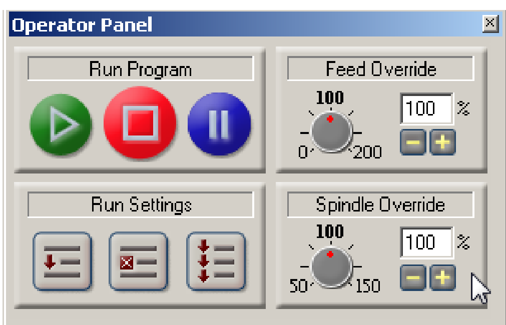
The Set Position window is displayed.

15. Set Z to 2. Do not change the X value at this stage.



16. Click **OK**.

17. Set the Spindle Override to 100% in the Operator Panel.



6.10. RUNNING THE PROGRAM

Now that the program has been checked in the Verify window and you have performed a dry run, the program can finally be run on an actual workpiece.

Safety

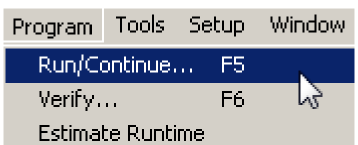
Before running the program:

1. Close the safety door.
2. Wear safety glasses.
3. Review all other safety precautions in 1 Safety Guidelines, pg. 1. Be prepared to press the Emergency Stop button on the machine if anything goes wrong.

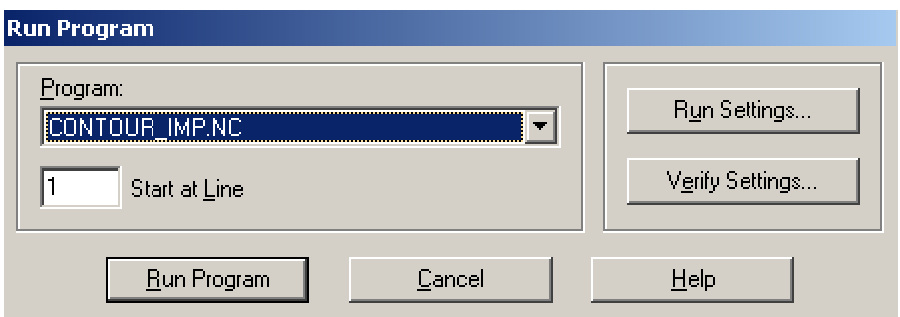
Follow this procedure to run the program:

Procedure: Running the Program

1. Follow the safety instructions presented above.
2. Click **Program** | **Run/Continue** in the Main Menu.



The Run Program window is displayed.



3. Ensure that Start at Line is set to line 1.
4. Click **Run Program** to begin running your program.
5. Once the program has ended, press the Emergency button, open the safety door, and remove the finished part.

7. Basic CNC Programming

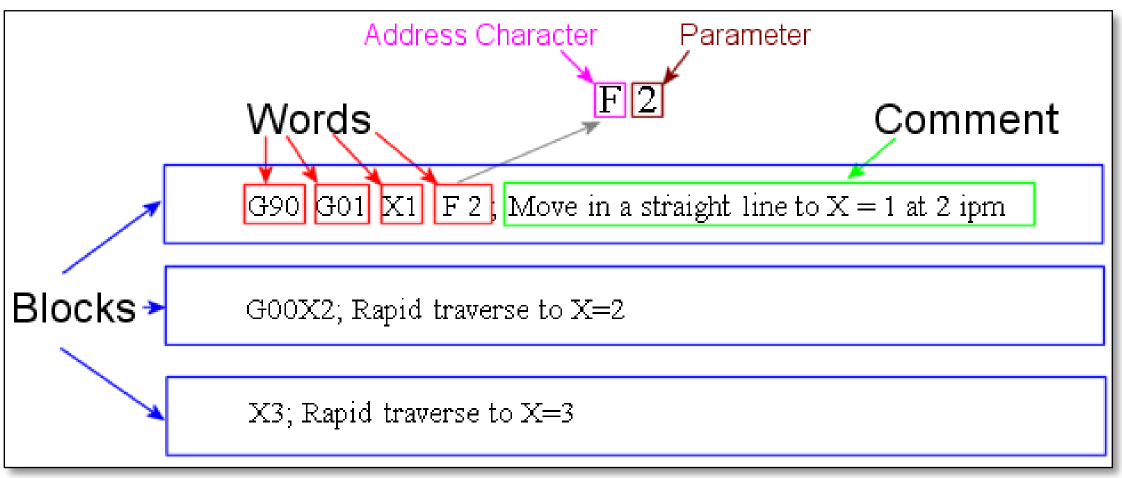
This section provides a basic reference for basic CNC programming.

This section presents the following information.

7.1. ELEMENTS OF AN NC PART PROGRAM

An NC program is composed of blocks (lines) of code. 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). Each line can end with a comment which is ignored while machining.

These concepts are illustrated in the graphic below.



There are many categories of address characters used in NC part programs for the BenchTurn Turning Center (see 7.4 NC Codes, pg. 100).

Each block of NC code specifies the movement of the cutting tool on the turning center and a variety of conditions that support it. For example, a block of NC code example below demonstrates the use:

N1G90G01X.5Z1.5F1

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

Example Code **N1G90G01X.5Z1.5F1**	
N1	This is the block sequence number for the program. Block 1 is the first block in the program.
G90	Indicates absolute coordinates are used to define tool position.
G01	Specifies linear interpolation.
X.5	Specifies the X axis destination position as 0.5".
Z1.5	Specifies the Z axis destination position as 1.5". The cutting tool will move to the absolute coordinate position (0.5,1.5).
F1	Specifies a feed rate of 1 inch per minute, the relative velocity at which the tool is advanced along the workpiece.

7.2. GENERAL PROGRAMMING SUGGESTIONS

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

Info Table: General Programming Suggestions		
No.	**Topic**	**Description**
1	Sequence of words in a block	The sequence of words (address characters and parameters) in an NC block must appear in the following order: %, , \, /, N (O), G, X (U),Z (W), I, K, R, Q, L, F, M, S, T, P, ; A different order may cause unpredictable results.
2	Repeating words in multiple lines	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 the following codes: N words, I and K, G04, G05, G25, G26, G92, F used for dwell, M02, M20, M25, M26, M30, M47, M98 or M99.
3	Multiple G words in a single block	You can use more than one G code in a block; however, you can use only one G code from any one group in a single block.
4	M codes in a single block	M codes should be placed on separate blocks to avoid confusion over whether an M code is activated during or after a motion command.
5	Use of N words	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.
5	Use of O words	An O code is required to mark the beginning of a subprogram and does not have to be in sequence with the N codes.

Info Table: General Programming Suggestions		
No.	**Topic**	**Description**
6	First steps in a part program	The first portion of a part program should turn on the spindle and establish the feed rate and spindle speed.
7	Referencing the zero point	Part programs should reference the zero point with Z0 at the point where the tool just touches the work piece. This convention allows for standardization of programming.
8	First movement command in program	The first instruction in a part program should move the tool to the starting position. This makes restarts much easier.
9	Last movement command in program	The last block of a program should move the tool back to the starting position. The tool will then be in position to start cutting another part.

7.3. REVIEWING AN NC PROGRAM

Once an NC program has been written, it must be checked carefully before the first part is machined. Errors in an NC program can cause machine damage and injury to the operator.

Follow the procedure below to check an NC program.

Procedure: Reviewing an NC Program

1. Double-check all program blocks against your coding sheet to locate and correct typographical errors.
2. Look for the typical coding error that places two X codes or two Z codes in the same block.
3. Be sure that all required coordinates have been written into appropriate blocks.
4. Verify the part program to discover any program errors.
5. Run the part program without mounting stock in the turning center to see if the tool movements are logical.

7.4. NC CODES

NC codes are divided according to category. Some categories consist of one code only (single category type), while other categories include a large number of different codes (multiple category type).

The table below lists all code categories. Information on all codes is presented in the sections referenced.

Info Table: G Code Categories				
Code	**Function**	**Type of Category**	**Section**	**Page**
%	Incremental Arc Centers (Fanuc)	Single	7.4.1	101
$	Absolute Arc Centers.	Single	7.4.2	101
\	Skip.	Single	7.4.3	101
/	Optional skip.	Single	7.4.4	102
F	Feed rate in inches per minute, with G04, the number of seconds to dwell.	Single	7.4.5	102
G	Preparatory codes.	Multiple	7.4.6	102
H	Input and output selection number	Single	7.4.7	115
I	Arc center, X axis dimension (circular interpolation).	Single	7.4.8	116
K	Arc center, Z axis dimension (circular interpolation).	Single	7.4.9	116
L	Loop counter Program cycle (repeat) counter for blocks and sub- programs, angle of arc resolution.	Single	7.4.10	117
M	Miscellaneous codes.	Multiple	7.4.11	119
N	Block number (user reference only).	Single	7.4.12	124
O	Subprogram starting block number.	Single	7.4.13	124
P	Subprogram reference number (with M98 or M99).	Single	7.4.14	125
Q	Depth of cut. Peck depth for pecking canned cycle.	Single	7.4.15	125

R	Configures the shape of a taper (with G77).	*Single*	7.4.16	125
S	Spindle speed.	Single	7.4.17	126
T	Tool selection.	Single	7.4.18	126
U	Incremental X motion dimension.	Single	7.4.19	126
W	Incremental Z motion dimension.	Single	7.4.20	126
X	X axis motion coordinate.	Single	7.4.19	126
Z	Z axis motion coordinate.	Single	7.4.20	126
;	Comment	Single	7.4.21	126

7.4.1. % Code: Incremental Arc Center

The incremental arc center code selects the Fanuc mode of 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 based on incremental coordinates, regardless of whether the system is in G90 (absolute) or G91 (incremental) coordinate mode. In contrast, arc center specifications in the EIA-274 programming mode (specified by the $ code) follow the selected coordinate mode (absolute or incremental).

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

The default arc center mode is defined in the Run Settings window, accessed by clicking Setup | Run Settings, in the Main Menu.

7.4.2. $ Code: 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, arc centers are based on the selected coordinate mode: absolute (G90) or incremental (G91). In contrast, arc center specifications in Fanuc mode (specified by the % code) are always incremental, regardless of whether the system is set to absolute or incremental coordinate mode.

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

The default arc center mode is defined in the Run Settings window, accessed by clicking Setup | Run Settings, in the Main Menu.

7.4.3. \ Code: Skip

The Skip code causes the program line to be skipped when the program is run.

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.

You can also use the Skip code (\) with a parameter to instruct the control software 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.

7.4.4. / Code: Optional Skip

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

Lines marked with the optional skip (/) code are skipped only when the Optional Skip parameter from the Run Settings window is checked (see 5.6.2 Modifying Run Settings, pg. 69).

To use the Optional Skip code (/), place the code at the beginning of the line you wish to skip. Select the Optional Skip option from the Run Settings window or the Operator Panel.

You can use the optional skip code with a parameter to instruct the control software to execute the line of code every nth pass. Place the code at the beginning of the line you wish to optionally skip. The syntax is: /n, where n is the number of passes between executions.

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

7.4.5. F Code: Feed Rate

The F Code is used to specify the velocity at which the tool moves (the feed rate). The feed rate is specified in inches or millimeters per minute, depending on the selected unit of measure. For example, F3 specifies a feed rate of 3 inch/min when using inch units or 3 mm/min when using metric units.

The feed rate should be set to a low value (up to 8 inch/min) for cutting operations, though higher feed rates can be achieved in soft metals such as brass and aluminum.

The control software limits the programmed feed rate so that it doesn’t exceed the maximum allowed by the turning center.

The F Code is also used to specify the number of seconds to dwell when used with the G04 code. For example, G04F5 causes the machine to dwell for five seconds. See 7.4.6.3 The Wait Group, pg. 104.

7.4.6. G Codes: Preparatory Codes

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

There is a large number of G codes, each differentiated by the number following the G. For example, G01, G90, and G71 are all different G codes.

The various G codes are divided into different groups. Multiple G codes from different groups can appear in each NC block. However, you may not place more than one G code from a group in one block.

The BenchTurn 7000 supports the following G code groups.

Info Table: G Code Groups			
Group	**Includes Codes**	**Section**	**Page**
The Interpolation Group	G00, G01, G02, G03	7.4.6.1	103
The Units Group	G70, G71, G20, G21	7.4.6.2	103
The Wait Group	G04, G05, G25, G26	7.4.6.3	104
The Canned Cycle Group	G32, G72, G73, G77, G79, G80, G81, G83	7.4.6.4	105
The Programming Mode Group	G90, G91	7.4.6.5	105
The Preset Position Group	G28, G29, G92, G98, G99	7.4.6.6	106
The Coordinate Systems Group	G54, G55, G56, G57, G59	7.4.6.7	107
The Compensation Group	G39, G40, G41, G42	7.4.6.8	108
The Scaling Group	G50, G51, P	7.4.6.9	114
The Rotation Group	G68, G69	7.4.6.10	115
The Polar Programming Group	G15, G16	(Not described in this Guide)	

7.4.6.1. 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:

Info Table: Interpolation Group			
Code	**Function**	**Section**	**Page**
G00	Rapid Traverse	8.3	132
G01	Linear interpolation	8.1	128
G02	Circular interpolation (clockwise)	8.2	129
G03	Circular interpolation (counterclockwise)		

7.4.6.2. 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 setting 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. If placed later in the program, only the parts of the program following the code are affected.

These codes can be used to switch between inch and metric modes throughout the program.

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

Info Table: Units Group	
Code	**Function**
G70	Inch Units
G71	Metric Units
G20	Fanuc inch units
G21	Fanuc metric units

7.4.6.3. 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:

Info Table: Wait Group	
G04	Dwell (wait) Pause between motions on all axes for the number of seconds specified by the F code, then continue the program. This is used primarily for robot integration. Because the F code is used to specify the number of seconds, you cannot also specify a new feed rate in the same block. Example: G04F10; Wait for 10 seconds **Safety** Do not use G04 to create a pause for a tool change during a program. Use G05 instead.
G05	Pause Used for operator intervention. Stop motion on all axes until the operator manually resumes program execution using the Run/Continue command or the Run button .
G25	Wait until input #1 (Robot 1 or user input 5) goes high before executing the operations in this block. Used for robot synchronization (see 11 Automation Integration, pg. 174). Use the H code to specify an input other than the default, H5 (see 7.4.7 H Code: Input Selection Number, pg. 115.) Example: G25H13; Wait until user input 3 goes high.

G26	Wait until input #1 (Robot 1 or user input 5) goes low before executing the operations on this block. Used for robot synchronization (see 11 Automation Integration, pg. 174). Use the H code to specify an input other than the default, H5. Example: G26H12; Wait until user input 2 goes low.
G31	Move linearly to a specified coordinate but monitor the state of a specified input during motion. If the state of the input changes from High to Low or from Low to High (as specified), then stop the motion. Jump to a specified line in the NC program to continue. The H word specifies the input to watch (see 7.4.7 H Code: Input Selection Number, pg. 115). If a negative sign is used before the input number, the motion will stop if the input state changes from High to Low. Otherwise, the motion will stop if the input state changes from Low to High. The P word specifies the line number to jump to (see 7.4.14 P Code: Subprogram Reference Number, pg. 125). Example 1: G31 X5 Z5 H11 P500; Move linearly to X5Z5. If input 1 changes from Low to High, stop the motion. Continue at line 500 in the program. Example 2: G31 X3 Z2 H-12 P30; Move linearly to X3Z2. If input 2 changes from High to Low, stop the motion. Continue at line 30 in the program.

7.4.6.4. 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.

Detailed information on many canned cycles is presented in chapter 8, as detailed below.

The supported Canned Cycle codes are listed below.

Info Table: Canned Cycle Group			
Code	**Function**	**Section**	**Page**
G32	Canned cycle thread cutting	8.4.1	133
G72	Canned cycle arc turning, clockwise	8.4.2	139
G73	Canned cycle arc turning, counter clockwise		
G77	Canned cycle side turning	8.4.3	141
G79	Canned cycle end turning	8.4.4	146
G80	Canned cycle cancel	8.4.5	147
G81	Canned cycle drilling	8.4.6	147
G83	Canned cycle peck drilling		

7.4.6.5. 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.

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

The supported Programming Mode codes are listed below.

Info Table: Programming Mode Group	
Code	**Function**
G90	Absolute programming mode: All X and Z axes coordinates are specified relative to a (0,0) location on the turning center.
G91	Incremental programming mode: Each motion to a new coordinate is specified relative to the previous coordinate.

7.4.6.6. The Preset Position Group

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

This table lists the supported Preset Position codes.

<table>
<tr><th colspan="4">Info Table: Preset Position Group</th></tr>
<tr><th>Code</th><th>Function</th><th>Section</th><th>Page</th></tr>
<tr><td>G28</td><td>Homes the machine. Can specify a point to pass through on the way to the home position.</td><td rowspan="2">7.4.6.6.1</td><td rowspan="2">106</td></tr>
<tr><td>G29</td><td>Return to reference point: Moves the tool to a coordinate specified by XZ. Typically used after a G28 code.</td></tr>
<tr><td>G92</td><td>Preset position: This code works like the Set Position command under the Setup Menu. The X and Z coordinates following a G92 code define the new current position of the tool.</td><td>7.4.6.6.2</td><td>107</td></tr>
<tr><td>G98</td><td>Rapid move to initial tool position after canned cycle complete.</td><td rowspan="2">8.4</td><td rowspan="2">132</td></tr>
<tr><td>G99</td><td>Rapid move to point R (surface of material or other reference point) after canned cycle complete.</td></tr>
</table>

7.4.6.6.1. G28 and G29: Homing Commands

The Homing feature in the control software sends the machine to the predefined Home position (0,0). This is used as a reference for other motion.

The homing commands (G28, G29) allow you to return to and check this established position. The turning center uses this point as a reference for all machine coordinate movements. This allows you to use the Soft Limits and Coordinate Systems commands (under the Setup Menu) to move the turning center consistently to the same location.

Before you can use any homing commands, or the Soft Limits and Coordinate Systems commands, you must use the Set/Check Home command to establish an initial reference point. See 5.4 Homing, pg. 65, for information on using the Set/Check Home command.

Using G28 Code

G28 homes the machine, zeroing the machine coordinates. Optionally, the machine can be instructed to pass through specified coordinates on its way to the home position. For example, G28 X1 Z2 commands the machine to pass through X1 Z2 and then move to home position.

Using G29 Code

The G29 code moves the tool at a rapid traverse rate to a coordinate specified by XZ. If you have set an intermediate point on one or more axes, the machine first rapids from the current position to the intermediate point then continues to the specified destination. If you command a G29 code in incremental mode, your specified XZ point is relative to the intermediate point. If you have not specified an intermediate point, your specified XZ point is relative to the current position. Use the G29 code after a G28 code to return the tool to a position closer to the part. The example below shows the use of a G28 code and a G29 code.

N1G28X2Z-1; INTERMEDIATE POINT THEN HOME N2G29X4Z1; GO TO G29 POINT

7.4.6.6.2. G92: Preset Position

The G92 code is used to initialize the current tool position. In other words, the G92 code can be used to redefine the X and Z values of the tool's current position. The X and Z coordinates following a G92 code define the new current position of the tool.

The tool position can also be redefined through the control software, by clicking Setup | Set Position in the Main Menu.

7.4.6.7. The Coordinate System Group

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

For instance, you can run a part program using a typical coordinate system (with the point of origin on the left side of the workpiece next to the chuck, along the centerline of the workpiece), and then select another coordinate system which has its origin at a different point on the surface of the workpiece.

Multiple coordinate systems can be useful for different size workpieces, or for special set-up conditions.

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

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

Info Table: The Coordinate System Group						
G 53	For rapid traverse motion to specified machine coordinates.					
G Code	G54	G55	G56	G57	G58	G59
Coordinate System Activated	1	2	3	4	5	6

7.4.6.8. The Compensation Functions Group

Cutter compensation automatically adjusts to compensate for variations in a cutting tool's radius. It uses tool radius values from the Setup Offsets table to determine the compensation offset value.

The supported compensation codes are listed below.

Info Table: Compensation Functions Group			
Code	**Function**	**Section**	**Page**
G39	Corner offset circular interpolation	7.4.6.8.1	108
G40	Cancel cutter compensation	7.4.6.8.2	110
G41	Left cutter compensation	7.4.6.8.3	112
G42	Right cutter compensation		
D	Specifies offset number from offset table.	7.4.6.8.4	113

7.4.6.8.1. G39: Corner Offset Circular Interpolation

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:

NC Code Example: G91 G41D1… … Y.25 X.25 G39I0J-1; CORNER OFFSET	
Code	**Explanation**
G91	Incremental mode
G41D1…	Start left cutter compensation, using entry #1 from the offset table.
… Y.25 X.25	Specifies the cutting path.
G39I0J-1; CORNER OFFSET	Complete the current segment and move to the default endpoint (I10J-1), creating an arc with a radius of the offset distance.

7.4.6.8.2. G40: Cancel Cutter Compensation

Use the G40 Cancel cutter compensation code to cancel cutter compensation. G40 is modal.

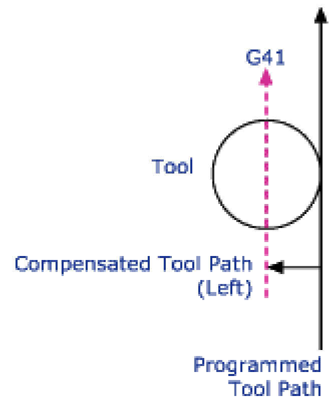
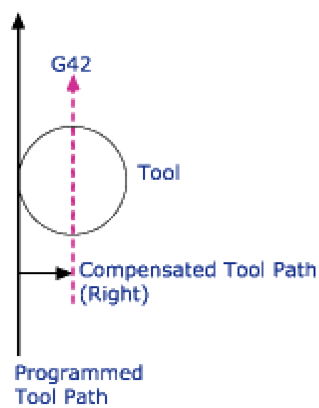
There are six methods for cancelling cutter compensation, described in the table below.

Info Table: Cancelling Cutter Compensation		
Method	**Explanation**	**Example**
G40	The G40 code cancels cutter compensation. The cutter moves from the offset path to the programmed end point.	G91G41D1 … X.25 Y-.25 Z.2; RETRACT **G40** X-.5Y-.25 M2
G40XYZ	The G40 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
G40XYZIJK	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
D0	Setting the offset number to zero cancels cutter compensation. The cutter moves from the offset path to the programmed end point. Setting the offset number to zero has the same effect as cancelling cutter compensation (as for G40 listed above). However, cutter compensation is still active.	G91G41D1 … X.25 Y-.25 Z.2; RETRACT **D0** X-.5Y-.25 M2

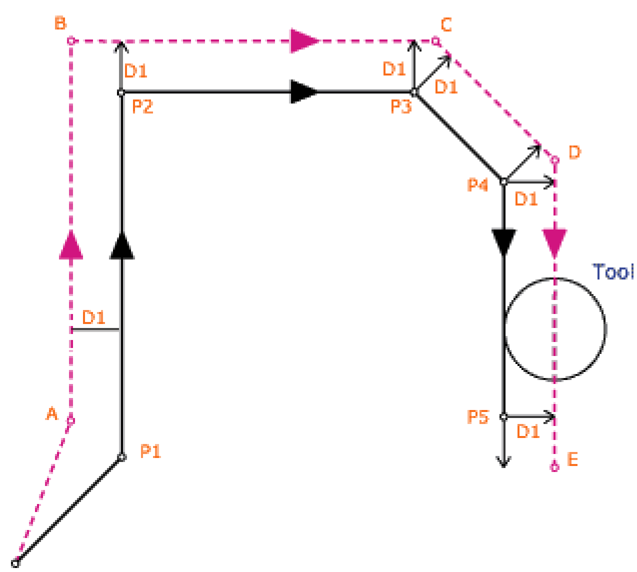
D0XYZ	The D0 cancels the cutter compensation, but a subsequent motion (X-.5Y-.25) is included in the program. The tool moves towards the programmed path in the direction of X-.5Y-.25. Setting the offset number to zero has the same effect as cancelling cutter compensation. However, cutter compensation is still active.	G91G41D1 … X.25 Y-.25 Z.2;RETRACT D0 X-.5Y-.25 M2
G41/42D0 XYZIJK	An IJK vector specifies the direction of movement after cutter compensation is cancelled. Setting the offset number to zero has the same effect as cancelling cutter compensation. However, cutter compensation is still active.	G91G41D1 … X.25 Y-.25 Z.2; RETRACT D0 X-.5Y-.25I-.5J-.25 M2

7.4.6.8.3. G41 and G42: Left and Right Cutter Compensation

The G41 and G42 codes command left and right cutter compensation as illustrated below.

Info Table: Left and Right Cutter Compensation		
Code	**Explanation**	**Explanation**
G41	Left compensation when you need to move the tool to the left of the programmed tool path.	
G42	Right compensation when you need to move the tool to the right of the programmed tool path.	

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

NC Code Example: 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	
Explanation	**Motion**
The black lines show the specified tool path. The purple dashed lines show the compensated tool path that the tool will follow. Note that the distance between the two paths is always as specified by D1.	

7.4.6.8.4. D: Specify Cutter Compensation Value

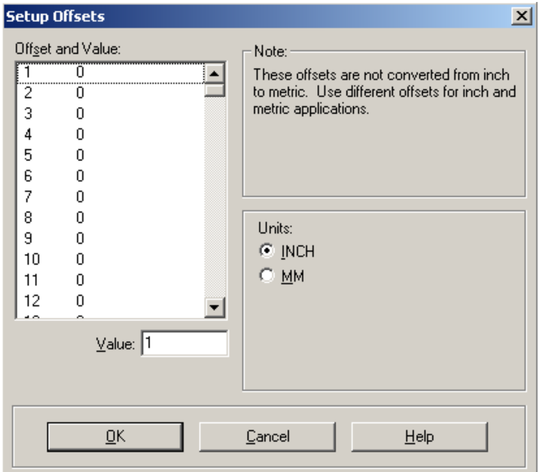
The D code selects a value for cutter compensation or tool offset adjustments by referencing values in the control program offset table.

For example, D1 selects entry #1 from the offset table.

D0 cancels compensation.

The offsets specified by D1 to D199 are set in the Setup Offsets table, accessed by clicking **Setup** | **Offsets** in the Main Menu.

Note that an NC program will use the offset values as listed in this window, without converting for metric or imperial systems. The Units selection area in the window converts the values in this window.



7.4.6.9. The Scaling Group

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 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 software 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 code I is used to specify the X axis center of scaling (see 7.4.8. I Code: X Axis Coordinate of Center Point, pg. 116). The code K is used to specify the Z axis center of scaling (see 7.4.9 K Code: Z Axis Coordinate of Center Point, pg. 116).

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

Take Note

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

Use the following codes for scaling:

Info Table: Scaling Group	
Code	**Function**
G50	Cancels scaling
G51	Invokes scaling
P	Uniform scale multiplier

The following example demonstrates the use of scaling:

<table>
<tr><th colspan="2">Example Code: Scaling</th></tr>
<tr><th>Code</th><th>Explanation</th></tr>
<tr><td>G51X0Z0I2.J2</td><td><table><tr><td>G51</td><td>Initiates scaling.</td></tr><tr><td>X0Z0</td><td>Sets the origin of scaling to X0Z0.</td></tr><tr><td>I2J2</td><td>Scales both axes by 2.</td></tr></table></td></tr>
</table>

7.4.6.10. The Rotation Group

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.

The supported Rotation codes are listed in this table.

Info Table: Rotation Group	
Code	**Function**
G68	Invokes rotation.
G69	Cancels rotation.
R	When used with G68, specifies the angle of rotation, in degrees.

The following example demonstrates the use of rotation:

<table>
<tr><th colspan="2">Example Code: Rotation</th></tr>
<tr><th>Code</th><th>Explanation</th></tr>
<tr><td>G68X0Z0R90</td><td><table><tr><td>G68</td><td>Initiates rotation.</td></tr><tr><td>X0Z0</td><td>Sets the origin of rotation to X0Z0.</td></tr><tr><td>R90</td><td>Sets rotation to 90 degrees.</td></tr></table></td></tr>
</table>

7.4.7. H Code: Input Selection Number

The H code is used to specify inputs and outputs in robot integration (see 11 Automation Integration, pg. 174).

Use the H code in conjunction with:

- The wait codes G25 and G26 to specify the input number.
  The H code and input must be specified.
- The transmit codes M25 and M26 for interfacing with robots or other external devices to specify the output number.

H codes specify inputs and outputs as defined in the table below.

Info Table: H Code				
H Code	**H11**	**H12**	**H13**	**H14**
Input specified for Wait Codes G25 and G26	1	2	3	4
Output specified for Transmit codes M25 and M26	1	2	3	4

7.4.8. I Code: X Axis Coordinate of Center Point

The I code specifies the X coordinate of the center of an arc or circle. If no I code is specified when specifying an arc or circle, the system uses the current X axis location as the X axis center of the arc.

The I code is used in both absolute and incremental programming modes. In Fanuc mode, all arc centers are specified in incremental mode.

The value following the I code is interpreted differently in absolute and incremental programming modes, as follows:

Info Table: I Code in Absolute and Incremental Modes		
Mode	**Activated by**	**I value specifies:**
Absolute	G90	The X coordinate of the arc/circle center, measured from the origin. Note: In Fanuc mode, all arc centers are specified in incremental mode.
Incremental	G91	The distance along the X direction from the starting point of motion to the arc/circle center.

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. See 7.4.6.9 The Scaling Group, pg. 114.

7.4.9. K Code: Z Axis Coordinate of Center Point

The K code specifies the Z coordinate of the center of an arc or circle. If no K code is specified when specifying an arc or circle, the system uses the current Z axis location as the Z axis center of the arc.

The K code is used in both absolute and incremental programming modes. In Fanuc mode, all arc centers are specified in incremental mode.

The value following the K code is interpreted differently in absolute and incremental programming modes, as follows:

Info Table: K Code in Absolute and Incremental Modes		
Mode	**Activated by**	**K value specifies:**
Absolute	G90	The Z coordinate of the arc/circle center, measured from the origin. Note: In Fanuc mode, all arc centers are specified in incremental mode.
Incremental	G91	The distance along the Z direction from the starting point of motion to the arc/circle center.

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. See 7.4.6.9 The Scaling Group, pg. 114.

7.4.10. L Code: Angle of Arc Resolution, Loop Counter

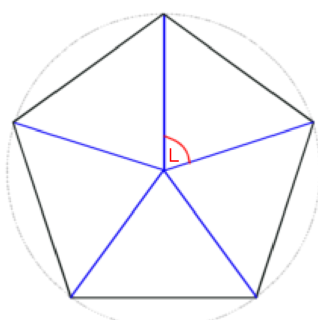
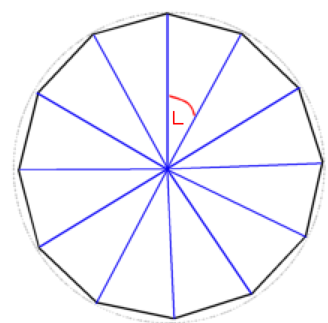
The L code is used in three unrelated ways, as listed in this table.

By default, an L code that is not specifically used as a counter is treated as specifying the arc/circle resolution factor.

7.4.10.1. L Code: Angle of Arc Resolution

When specifying an arc or circle, the L code specifies the angle of arc resolution in circular interpolation programming. When the system executes a circular motion, it splits the arc into a series of line segments to approximate the circle. The L code specifies the resolution in terms of the angle in degrees across which a line segment approximates a portion of the arc. Setting a low value of L increases the resolution of the arc or circle, and thus creates a smoother cut. However, very fine resolutions may cause the turning center to hesitate while cutting.

The table below illustrates the effect of the L value.

Info Table: L Code Specifying Angle of Arc Resolution		
	High Value of L	**Low Value of L**
Example		
Resolution	Resolution is low; the approximated circle is made up of few lines.	Resolution is higher; the approximated circle is made up many lines.

Default Settings

The default setting for the turning center is 2 degrees, and typically this will work quite well. The L code can be a fraction of a degree (such as L0.5), but it must be large enough so that the turning center will move at least the minimum motion distance (0.0005") on each of the straight line motions.

Calculating L

If you know the chord length you would like used when approximating an arc or circle, calculate the angle L as follows:

Calculation: Calculating the Value of L	
Calculation	**Where:**
$L = \frac{360 \cdot c}{2\pi \cdot R}$	c is the desired chord length.
	R is the arc or circle radius.
Example	To calculate the L value for a chord length of 0.001" and a radius of 0.5", you would write the equation as: $L = \frac{360 \cdot c}{2\pi \cdot R} = \frac{360 \cdot (.001)}{2\pi \cdot .5} = 0.1146°$

Negative L Values

A negative value for L will generate a normalized L factor, calculated as:

Angle {degrees} = L / Radius { inches}

For example, with the default L factor of -1:

Calculation: Calculating the Angle of Arc when Using Normalized L of -1			
Radius	**L Value**	**Calculation**	**Result**
0.5 inch	-1	Angle = L/0.5 = 1/1	The arc will have line segments approximating every 2 degree of the arc.
1 inch	-1	Angle = L/R = 1/1	The arc will have line segments approximating every 1 degree of the arc.
2 inches	-1	Angle = L/2 = 1/2	The arc will have line segments approximating every 0.5 degrees of the arc.

7.4.10.2. L Code: Loop and Program Counter

The L code functions as a counter when used with codes M98 and M47:

Info Table: L Code Used as a Counter			
When Used With	**L Specifies**	**Section**	**Page**
M47	The number of times a program must be repeated.	7.4.11 M Codes: Miscellaneous Codes	119
M98	The number of times a subprogram must be repeated.	8.5 Subprogram Programming	150

7.4.11. M Codes: Miscellaneous Codes

M codes control a variety of turning center functions while a part program is running.

The following general rules must be followed when using M codes:

Info Table: Rules for Use of M Codes		
1	Only one M code may be used in a program block.	
2	M codes should be specified in their own blocks. Using M codes together with other codes in a block can cause confusion about which code is executed first, however:	
	i	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.
	ii	All M codes used to turn off a device execute after the tool motion for that block is completed.

The supported M codes are listed below.

Info Table: Miscellaneous Mode Group		
Subgroup	**Code**	**Function**
Program Stop/End Group	M00	Pause Allows you to place a pause in your code. Acts like a G05 pause.
	M01	Optional Stop Allows you to place an optional 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 window (see 5.6.2 Modifying Run Settings, pg. 69) and on the Operator Panel (see 5.3.4.2 The Operator Panel, pg. 64). With Optional Stop on, the M01 works like a G05 pause. With Optional Stop off, the M01 code is ignored, and the other codes on the block are executed as usual.
	M02	End of Program Takes effect after all motion has stopped; turns off drive motors, and all outputs, including the spindle and the accessory outlets.
	M30	End of program: Same as M02.
Spindle and Axis Motor Group	M03	Spindle Motor On Activated concurrently with motion specified in the program block; remains in effect until superseded by M05.
	M05	Spindle Motor Off Activated after the motion specified in the program block; remains in effect until superseded by M03.
	M38	Drive Motors Standby Typically used to turn the drive motors to standby (low power) before a pause (G05); activated after the motion specified in the program block. M38 is useful when the machine is left unattended for a long time, such as the interval before reloading a part.

Tool Change Group	M06	Tool Change (see 9.3 Writing an NC Program for Multiple Tools, pg. 154)
I/O Group	M25	Set Output #H On Used for robot synchronization. Use the H code to specify an output (see 7.4.7 H Code: Input Selection Number, pg. 115). See 11 Automation Integration, pg. 174
	M26	Set Output #H Off Used for robot synchronization. Use the H code to specify an output (see 7.4.7 H Code: Input Selection Number, pg. 115). See 11 Automation Integration, pg. 174
Program Management Group	M20	Chain to Next Program This code is used to chain several NC files together. It is placed at the end of a part program and is followed on the next line by the file name of another program to be executed when all motion stops. Below is an example of a part program chain to another program: N37 Z.2 N38 M20 PROGRAM2.NC; Chain to PROGRAM TWO If the two programs you are chaining are not in the same directory on your computer, you must specify the full path name for the next program file. If the software cannot locate the specified file, you will be prompted to find it.
	M22	Output current position or other information to file. See 7.4.11.1 M22 Code: Output Current Position to File, pg. 121.
	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. See 8.5 Subprogram Programming, pg. 150.
	M99	Return from Subprogram Goto See 7.4.11.2 M99 Code: Return from SubProgram, Goto, pg. 123, and 8.5 Subprogram Programming, pg. 150.

	M105	Operator Message A nonstandard Intelitek code used to display messages. See 7.4.11.3 M105 Code: Operator Message, pg. 123.
Homing Group	M111	Home the X axis.
	M112	Home the Z axis.

7.4.11.1. M22 Code: 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 and Z machine coordinates to a file. The proper format for using this code is:

M22([filename.ext [,A]]) [text and macros]

Items in brackets [ ] are optional, except that a filename is required for the first M22 code used.

This table details the use of the M22 code.

Info Table: Using the M22 Code **M22***([filename.ext [,A]]) [text and special codes]*	
Parameter	**Notes**
Filename	• Must be enclosed in parentheses. • Must be specified the first time the M22 code is used in the program. • If not specified with subsequent M22 codes, the first specified file name will be used. However, empty parentheses must still be used. • If the file does not exist, it will be created
A	• If the file name is followed by *,A:* ▪ The new data is added to the end of the existing file; the existing data in the file is not deleted. ▪ The new data is automatically added on a new line in the file. • If the file name is not followed by *,A*, existing data in the file is deleted, and the new data is added.
text	• Enter standard text to be written in each line. Text can be written in front of, between, or after macros. • If no text or macros are specified, the actual machine position data will be written.
special codes	• Codes such as @TD (time of day) and @ (cycle number) can be specified to include a range of data. See below in this section for a table of all available special codes.

The following pseudo-code illustrates the use of the M22 code

Example Code: Use of M22

```
. . .       ; code to move to position
; Open my1.xz, discard contents, write coordinates M22(my1.xz)
. . .       ; code to move to next position
; Append to currently open data file M22( )
. . .       ; code to move to next position
;Open my2.xz and append coordinates M22(my2.xz,A)
```

The following special codes can be used with M22 to generate run-time reports.

Info Table: Special Codes for Use With M22

Code	Description
@X	Current X position (in current coordinate system)
@Z	Current Z position (in current coordinate system)
~ (tilde)	New line (starts a new line in the file)
@TD	Time of day (12hour): “11:59:59AM”
@TC	Time (elapsed) for cycle: “99:11:59” (0’s trimmed from left)
@TT	Time total (of program run): “99:11:59”
@TA	Time Average (per cycle): “99:11:59” (“??:??” if first part)
@TL	Current Tool number: “5”
@C	Cycle number (current pass): “3”
@D	Date: “12/31/97”
@FN	Current file (without path): “PART.NC” (“UNTITLED.NC” if untitled)
\t	Tab
\\	Outputs a single backslash (‘\’) character to the file

Below is another annotated example of the use of M22 in an NC program. All text on the same block, after the closing parenthesis, is output to the file, with all valid special codes being replaced as it is written.

Example Code: Use of M22 with Special Codes

```
; 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.
```

7.4.11.2. M99 Code: 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.

7.4.11.2.1. Using M99 with Subprograms

When used in a subprogram, this code returns the program flow to the block following the last M98 (Call to Subprogram) command.

If the M98 codes is used together with an L code to specify multiple calls to the subprogram, the M99 code will return to the block containing the M98 code until all the specified number of subprogram calls have been made; it will then proceed to the block following the M98.

You can use the P code with a block number to specify a different block to jump 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.

7.4.11.2.2. Using M99 as a Goto Command

The M99 code can be used in the main NC program as a Goto command to jump to any line before the first subprogram (as denoted by an O code).

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

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, for example:

M99P50; Jump to line 50 in the current program.

7.4.11.3. M105 Code: Operator Message

This command is used to display messages in the control software. It provides a way to display messages to the operator on the message bar while an NC program is running. To pause the program with a custom message, place a ! character at the start of the message.

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

The M105 code is used in the format: *M105 (the message); comment*

This is illustrated in the example below:

Example Code: Use of M105
M105 (End of Roughing Segment); Normal Message, doesn't pause

Message functionality can be altered using the special codes listed below.

Info Table: Special Codes for Use With M105	
Code	**Function**
!	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 M105 code is used with special codes as in the format below:

M105 (alternate character plus the message) ;comment

For example:

Example Code: Use of M105 with Special Codes
M105 (~WARNING); Warning Message, doesn't pause M105 ( ) ; Clears current message
M105 (!Please stop and read this!) ; Normal Message, pauses
M105(~!I Proceed with Caution!) ; Warning Message, pauses
M105 (\b\b\b) ; Clears current message, beeps 3 times, and doesn't pause

7.4.12. N Code: Block Number

N codes have two uses:

- To provide destination references for Goto codes (M99) elsewhere in the program.
- To clearly show the organization of the code and improve readability.

Using the N code is optional; however, when you do use the N code, it must be the first character in the block.

Apart from the uses listed above, N codes are ignored by the control software. 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.

You can automatically number, renumber, or remove numbering from the program using the control software. Click **Edit** | **Renumber** in the Main Menu.

7.4.13. O Code: Subprogram Block Number

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 using the P code to specify which subprogram to execute. To return from the subprogram, use the M99 code. See 8.5 Subprogram Programming, pg. 150.

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:

Example Code: Use of O Code

```
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
```

7.4.14. P Code: Subprogram Reference Number

P codes are used with the following codes:

Info Table: Use of P Code

No.	Used with Code	To	Section	Page
1	G31	Reference a GOTO target block.	7.4.6.3	104
2	M98	Reference a subprogram using the subprogram block number.	8.5	150
3	M99	Specify a return block number as a GOTO target.	7.4.11.2.2	123

7.4.15. Q Code: Depth of Cut and Peck Depth

The Q code is used in canned cycles to define the depth of cut. In drilling cycles Q specifies the incremental depth of each peck.

See 8.4 Canned Cycle Programming, pg. 132.

7.4.16. R Code: Drilling Start Location

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. See 8.4.6 G81 & G83: Straight and Peck Drilling, pg. 147.

The R code is also used to specify the rotation angle, in degrees, with the G68 code. See 7.4.6.10 The Rotation Group, pg. 115.

7.4.17. S Code: Spindle Speed

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

If the spindle is off when the S code is used, the spindle speed is stored and used when the spindle is turned on again within the program by the M03 command.

7.4.18. T Code: Tool Selection

A T code is used to specify the tool (by number) from the tool turret to be used for an operation. 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.

Product Care	Using multiple tools is an advanced operation and should not be attempted by persons unfamiliar with using the BenchTurn 7000 turning center.

See 9 Multiple Tool Programming, pg. 152.

7.4.19. X and U Codes: X Axis Coordinate

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.

7.4.20. Z and W Codes: Z Axis Coordinate

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

7.4.21. Comment Codes

The control software allows you to add comments (notes) to your NC code lines. The control software recognizes two comment codes: a semicolon ; and an open parenthesis (. These two comment codes are equivalent. The use of either of these codes within an NC block indicates that a comment follows.

Comments must follow all other NC codes in the block. Comments are ignored when the part program is executed. Comments can be placed on a block without any NC codes.

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

Example Code: Comment	
Code	**Explanation**
X0Z0; MOVE TO ZERO POINT	The comment in the example tells us that the X and Z codes in this block command the cutting tool to move to the zero point (coordinate 0,0).

Comments can be combined with the G05 pause and the M06 tool change codes to display messages to the operator during program execution. The M105 code, however, provides a more versatile and powerful message facility (see 7.4.11.3 M105 Code: Operator Message, pg. 123).

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

Example Code: Comment	
Code	**Explanation**
G105(!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.

Take Note

The control software can strip comments from a program with a single command. The comments cannot be subsequently replaced. To do so, right-click in the Program Editing window and select **Renumber**. Check the **Remove Comments** checkbox and click **Do it!**

8. NC Programming Routines

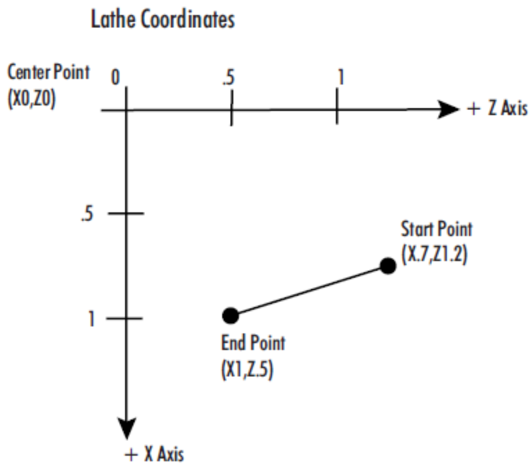
This chapter describes the use of the following NC programming routines:

8.1. LINEAR INTERPOLATION PROGRAMMING

Linear interpolation is the movement of the tool in a straight line from its current position to a coordinate location specified by an NC code.

Consider the following example code: N5G90G01X1Z.5F2

NC Code Example: N5G90G01X1Z.5F2	
Code	**Explanation**
N5	The line sequence number is 5.
G90	Coordinates are given using absolute dimensioning.
G01	Linear interpolation is specified. The G01 code is required when switching from circular interpolation or rapid traverse positioning back to linear interpolation.
X1	X axis coordinate of end point = 1
Z.5	Z axis coordinate of end point = .5

Example Motion	If we assume that the current position of the tool is X.7, Z1.2, the tool movement generated by the example code will be as follows. 
Equivalent Code	An equivalent movement is achieved with incremental dimensioning (G91): N5G91G01X.3Z-.7F2

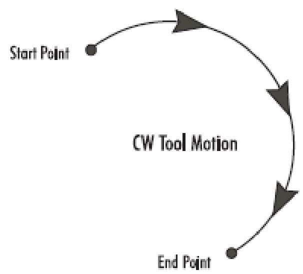
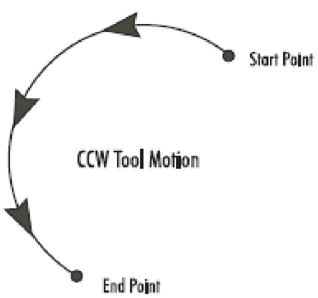
8.2. CIRCULAR INTERPOLATION PROGRAMMING

Circular interpolation moves the cutting tool along an arc from the starting point specified in one line to an end point specified in the next line.

The curvature of motion is determined by the location of the center point (I and K), which must also be specified in the second NC line. Whether I and K are interpreted in incremental or absolute form is specified in the Run Settings window, using the Arc Centers Incremental setting (see 5.6.2 Modifying Run Settings, pg. 69).

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.

The codes used in circular interpolation are listed below.

Info Table: Circular Interpolation Codes	
Code	**Explanation**
G02	Moves tool along circular path in clockwise direction. 
G03	Moves tool along circular path in counterclockwise direction. 
I	Specifies X-axis coordinate of center point
K	Specifies Z-axis coordinate of center point

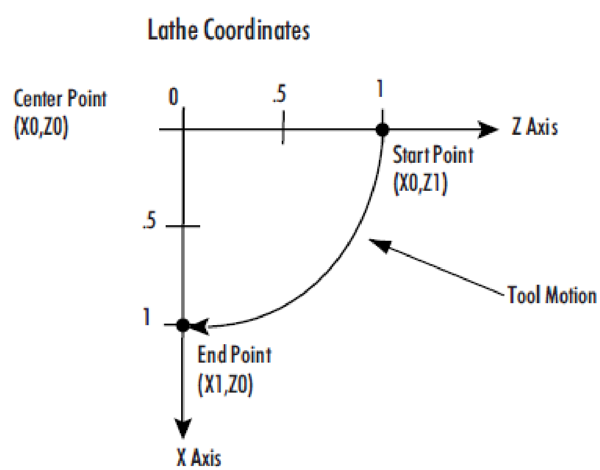
An example of circular interpolation is shown in the following table.

NC Code Example:

N8$

N9G90X0Z1;SET START POINT
N10G02X1Z0I0K0F2;CLOCKWISE TO X1,Z0

Code	Explanation
N8$	Sets absolute arc center mode.
N9G90X0Z1;SET START POINT	First line defines the starting point.

<table>
<tr>
<td>N10G02X1Z0I0K0F2;CLOCKWISE TO X1,Z0</td>
<td>Second line defines the end point and the center point.
<table>
<tr><td>N10</td><td>The line sequence number is 10</td></tr>
<tr><td>G02</td><td>The tool will proceed in a clockwise direction from the starting point to specified (X, Z) coordinates; center point of arc is specified by (I,K) coordinates</td></tr>
<tr><td>X1</td><td>X axis coordinate of end point = 1</td></tr>
<tr><td>Z0</td><td>Z axis coordinate of end point = 0</td></tr>
<tr><td>I0</td><td>I coordinate of center point of arc = 0</td></tr>
<tr><td>K0</td><td>K coordinate of center point of arc = 0</td></tr>
<tr><td>F2</td><td>Feed rate is 2 inches per minute</td></tr>
</table>
</td>
</tr>
<tr>
<td>Example Motion</td>
<td>Assuming the start point is X0,Z1, the tool path generated by the preceding lines would be as below.


</td>
</tr>
<tr>
<td>Equivalent Code</td>
<td>An equivalent movement is achieved with incremental dimensioning (G91):
N8%
N9G90X0Z1;SET START POINT
N10G91;SWITCH TO INCREMENTAL PROGRAMMING
N11G02X1Z-1I0K-1F2;CLOCKWISE TO X1,Z0

In the second line, the X and Z values are the distance the tool is to move from its current position. In both cases, the I and K values are equal to the X and Z distance from the start point to the center point.</td>
</tr>
</table>

8.3. RAPID TRAVERSE PROGRAMMING

On the BenchTurn turning center, the rapid traverse code (G00) moves the tool at the maximum available feed rate (80 ipm) to the specified coordinates. Rapid traverse is often used to reposition the tool before ending a program, or in preparation for the next cut.

Safety	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 a new feed rate is specified.

Rapid traverse is not affected by the feed rate scale factor. If all the positioning motions are performed in rapid traverse mode (G00) and all the cutting motions are performed in linear (G01) or circular interpolation (G02, G03) modes, cutting rates can be adjusted up or down using the feed rate scale factor without affecting rapid traverse motions.

An example of the use of rapid traverse is shown below.

NC Code Example:

```
G90G01X1F2; MOVE IN A STRAIGHT LINE TO X = 1 AT 2 IPM
G00X2; RAPID TRAVERSE TO X=2
X3; RAPID TRAVERSE TO X=3
G01X4; MOVE IN A STRAIGHT LINE TO X=4 AT 2 IPM
```

Code	Explanation
G90G01X1F2; MOVE IN A STRAIGHT LINE TO X = 1 AT 2 IPM	G90 – Sets absolute coordinates G0X1 – Moves the tool to position X = 1, using linear interpolation. F2 – Sets the feed rate to 2 inches per minute.
G00X2; RAPID TRAVERSE TO X=2	Specifies rapid traverse (G00) to position X = 2.
X3; RAPID TRAVERSE TO X=3	Moves tool to position X = 3. Rapid traverse mode is still active.
G01X4; MOVE IN A STRAIGHT LINE TO X=4 AT 2 IPM	G01 – Turns off rapid traverse mode and engages linear interpolation. X4 – Moves the tool to position X=4 at 2 inches per minute.

8.4. CANNED CYCLE PROGRAMMING

Canned cycle commands allow you to perform many operations by specifying a small number of 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 table below lists all canned cycles supported by the BenchTurn 7000 and its control software.

Info Table: Supported Canned Cycles			
Code	**Explanation**	**Section**	**Page**
G32	Canned cycle thread cutting	8.4.1	133
G72	Canned cycle arc turning (clockwise)	8.4.2	139
G73	Canned cycle arc turning (counterclockwise)		
G77	Canned cycle side turning (Fanuc G90)	8.4.3	141
G79	Canned cycle end turning (Fanuc G92)	8.4.4	146
G80	Canned cycle cancel	8.4.5	146
G81	Straight drilling	8.4.6	147
G83	Peck drilling		

The following codes are used within canned cycle codes.

Info Table: Codes Used in Conjunction with Canned Cycles	
Code	**Explanation**
G98	Rapid to initial position after canned cycle complete; this is the system default.
G99	Rapid to point R after canned cycle complete.
K	Specifies the number of repeats. The default is 1. When K=0, drilling data is stored.
P	Specifies the length of dwell time in seconds.
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. In side and end turning cycles the Q value is used to specify the depth of each roughing cut.
R	Used for specifying a starting reference point for peck drilling or for specifying tapers for side and end turning cycles. The point can be at the material surface or at another reference point.

8.4.1. G32: Threading

The BenchTurn 7000 and its control software support threading through the use of the G32 canned cycle.

Threading with a single point tool is accomplished by taking a series of cuts along the same helix of the thread (also known as chasing a thread).

This section presents the following information:

8.4.1.1. Threading in NC Code

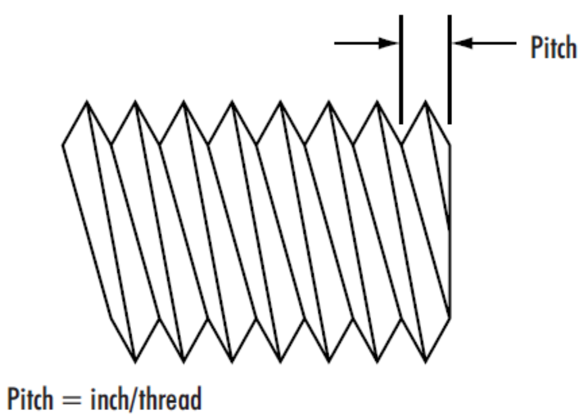
A threading operation requires that a number of parameters be calculated beforehand. In addition, NC code for threading must be preceded by some preparatory code.

This section presents instructions for planning and coding an NC program for threading.

Procedure Outline: Threading in NC Code			
No.	**Description**	**Section**	**Page**
1	Calculate and specify the pitch of the thread.	8.4.1.1.1	134
2	Calculate and specify the spindle speed and feed rate.	8.4.1.1.2	135
3	Prepare NC code to be performed before threading begins.	8.4.1.1.3	136
4	Write the NC code for a threading operation.	8.4.1.1.4	137

8.4.1.1.1. Specifying the Pitch

The pitch of the thread is the distance that the tool moves along the Z axis for each revolution of the spindle. The graphic below illustrates the concept of pitch.



The pitch is specified by the F code in the NC program as the pitch distance, measured in inches per thread.

Generally, machine screws used in the United States conform to the American National, or Unified Thread System. Threads are specified by the outside diameter of the thread and by the threads per inch, such as 1/4-20. The thread 1/4-20 has a pitch of 20 threads per inch, or a thread distance of 1/20 inches per thread.

To turn a thread of specification 1/4-20, F would be set to 1/20, or 0.05.

8.4.1.1.2. Specifying the Spindle Speed and Feed Rate

A number of considerations must be taken into account when specifying the spindle speed and feed rate. These are listed below and explained in the sections that follow.

Guidelines: Specifying Spindle Speed and Feed Rate			
No.	**Description**	**Section**	**Page**
1	The spindle speed and feed rate must be accurately set as their ratio affects the pitch distance produced. When using a canned cycle for threading, the NC program must specify the pitch directly, and the required feed rate is calculated automatically.	Feed Rate Calculation, below	135
2	The spindle speed and feed rate selected must not exceed the capabilities of the turning center.	Turning Center Maximum Cutting Feed Rate, below	136

Feed Rate Calculation

> When using a canned cycle for threading, the NC program must specify the pitch directly, and the required feed rate is calculated automatically. This section explains how to calculate the feed rate when threading without using a canned cycle.

The spindle speed and feed rate for a threading operation must be calculated accurately based on the desired pitch of the thread as there is a direct ratio between the spindle speed, the feed rate, and the pitch of the thread:

Pitch distance (inches/rev) X Spindle Speed (RPM) = Feed rate (inches/min)

In this example, the required feed rate to turn a 1/4-20 thread at a spindle speed of 200 RPM is calculated.

Example: Calculating Threading Feed Rate		
No.	**Step**	**Demonstration**
1	Calculate the pitch distance	The thread is specified as 1/4-20. The pitch in threads per inch is thus 20 inches per thread. The pitch in inches per thread is thus 1/20 inches or 0.05 inches.

2	Calculate the feed rate	Use the formula below: *Pitch distance (inches/rev) X Spindle Speed (RPM) = Feed rate (inches/min)* 0.05 inches/rev X 200 RPM = Feed Rate Feed rate = 10 inches per minute

Turning Center Maximum Cutting Feed Rate

Although the calculation presented in the previous section produces theoretically appropriate spindle speed and feed rates, the physical specifications of the turning center must be taken into account, particularly the maximum cutting feed rate.

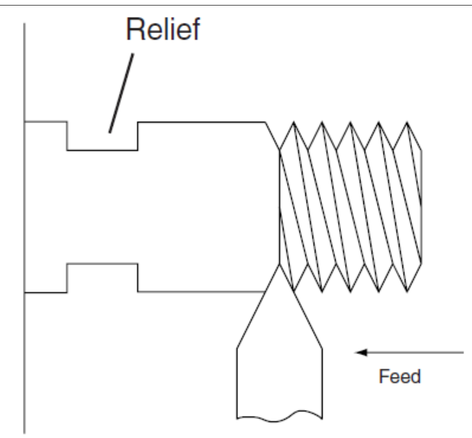
The turning center's maximum cutting feed rate while threading is 20 inches per minute. The spindle speed selected must be such that the calculated cutting feed rate will not exceed this value. As the ratio is given by

Pitch distance (inches/rev) X Spindle Speed (RPM) = Feed rate (inches/min),

choosing a high spindle speed will result in a high calculated value of feed rate. If your calculated feed rate exceeds the turning center's specifications, select a lower spindle speed.

8.4.1.1.3. Pre-threading Operations

There are three operations that must be performed by the NC code before threading is started.

Info Table: Pre-threading Operations	
No.	**Operation**
1	Before the threading cycle is performed, a groove or relief must be cut behind the eventual end point of the thread. If such a groove is not cut, the tool will be removed abruptly from the workpiece, making the end of the helix straight. 
2	The start point for the threading operation must not be along the workpiece surface. Instead, the tool should start the operation some point away from the workpiece along the Z axis. This will ensure that the tool and workpiece are in motion before cutting begins.
3	After commanding the spindle speed to slow down for threading, a dwell must be implemented to allow the spindle time to slow to the commanded speed before threading begins.

8.4.1.1.4. Programming a Threading Operation Using a Canned Cycle

The G32 code commands a threading operation.

The following codes are used together with G32.

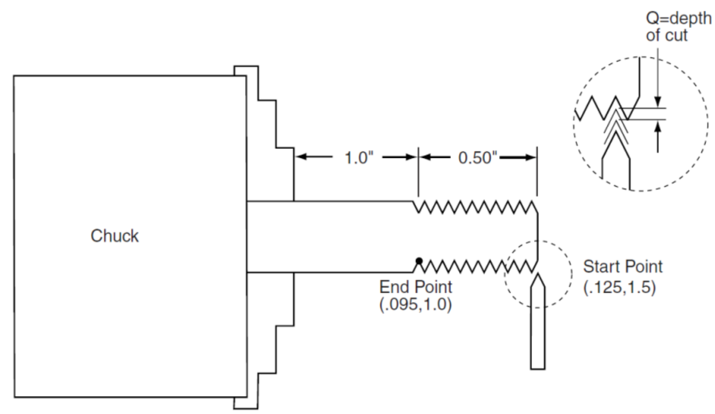
Info Table: Codes Used with G32 Threading Operation	
Code	**Description**
X, Z	Specify the coordinates of the end point of the thread.
Q	Specifies the depth of cut. The recommended Q depth for threading is very small; from 0.001” to 0.003”.
F	Specifies the pitch, in units of inches per thread.

The code below demonstrates the use of a threading cycle. The program will cut a 20-pitch thread on a 0.25” diameter rod.

NC Code Example: Threading Cycle

```
G0X.125Z1.5;TO SET THE START POINT
S200M03;SPINDLE SPEED 200RPM
G04F5;DWELL FOR 5 SECONDS
G32X.095Z1Q.002F.05;X & Z ENDPOINTS, Q=DEPTH OF CUT, F=PITCH
G80;END CANNED CYCLE
```

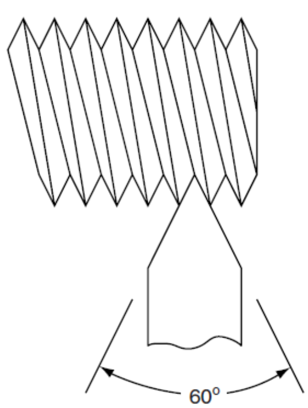
Code	Explanation
G0X.125Z1.5	Rapids to X.125Z1.5. This point is some way from the workpiece to ensure that when threading starts, both the tool and spindle will already be in motion.
S200M03;SPINDLE SPEED 200RPM	Sets the spindle speed to 200 RPM and turns on the spindle.
G04F5;DWELL FOR 5 SECONDS	Dwells for 5 seconds to allow the spindle time to slow down to its new speed of 200 RPM.

<table>
<tr>
<td>G32X.095Z1Q.002F.05;X & Z ENDPOINTS, Q=DEPTH, F=PITCH</td>
<td>
<table>
<tr><td>G32</td><td>Threading canned cycle</td></tr>
<tr><td>X.095Z1</td><td>Sets the end point of the thread to X.095Z1.</td></tr>
<tr><td>Q</td><td>Specifies a depth of cut of 0.002 inches.</td></tr>
<tr><td>F.05</td><td>Specifies a pitch distance of 0.05 inches. This is equivalent to a pitch of 20 threads per inch.</td></tr>
<tr><td>G80</td><td>End canned drilling cycle.</td></tr>
</table>
</td>
</tr>
<tr>
<td>G80;END CANNED CYCLE</td>
<td>Ends the threading canned cycle</td>
</tr>
<tr>
<td>Example Motion</td>
<td>The program presented here will create a 1/2"-long 1/4-20 thread as shown below.


</td>
</tr>
</table>

8.4.1.2. Thread Tooling

Most threads are cut with a 60° threading tool, as shown below.



For fine threads, the tool can be ground to a sharp point. For coarse threads, you may wish to radius the point.

An external threading tool is used for external threading operations. An inside threading tool is used for internal threading, such as on a nut.

8.4.1.3. Setting Up for Threading

Follow the guidelines below for setting up the turning center.

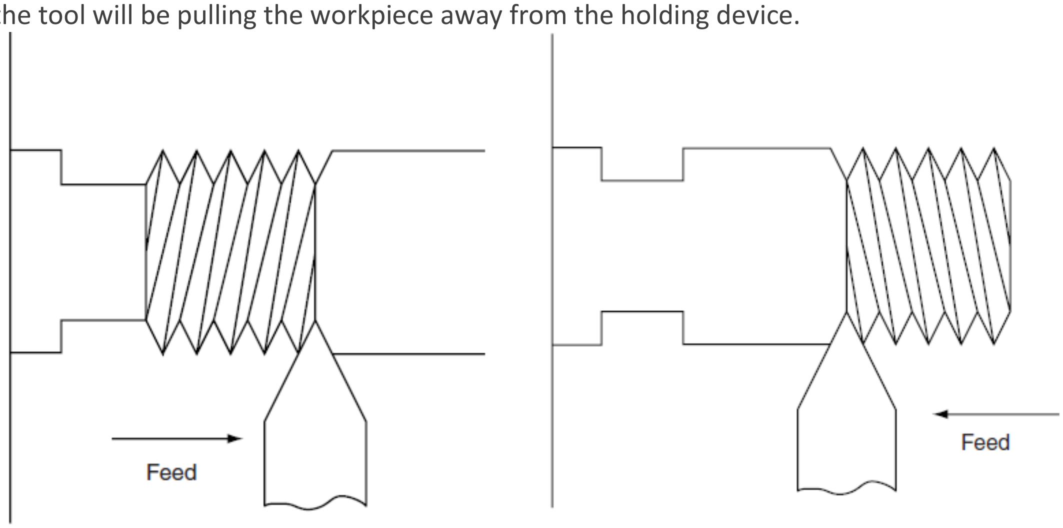
Setup Guidelines for Threading
• The workpiece must be mounted in a chuck or collet. • The tool height must be set exactly on the centerline of the workpiece for a correct thread angle. • Locate the zero point of the workpiece just as you would for other turning operations.

8.4.1.4. Cutting Left-Hand Threads

Most threaded parts are made with right-hand threads. The turning center can turn both right-hand and left-hand threads. The same tool is used for both left and right-handed threading; the direction of the cut is altered.

The procedure for cutting left-hand threads on the turning center is the same as for right-hand threads, except the feed is reversed so the cut is made from the left to the right.

When turning a left-hand thread, ensure that the workpiece is tightly secured in the chuck or collet as the tool will be pulling the workpiece away from the holding device.



Left-Hand Threading Right-Hand Threading

8.4.1.5. Internal Threading

Most of the same rules that apply to external threading also apply to internal threading. The final X value when turning an internal thread will be larger than the initial value.

To perform single point internal threading, a hole is first drilled to 1/16 inch diameter less than the minor diameter. A boring bar is then used to bore the hole to the minor diameter of the thread. Holes that are smaller than the minimum diameter of the boring bar can turn cannot be tapped.

8.4.2. G72 & G73: Arc Turning Canned Cycles

G72 can be used to cut an arc in one quadrant in a clockwise direction. These cuts are made using a facing orientation and a profiling tool.

Take Note

A canned radius turning cycle can only be performed in one quadrant; it cannot be performed across quadrants.

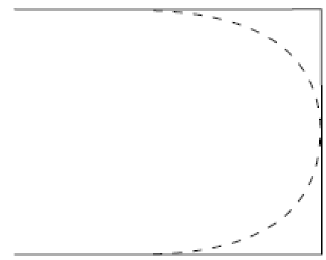
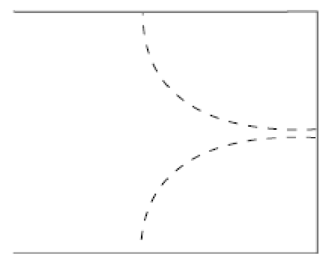
See the example below for the use of G72. G73 works the same way but specifies a counterclockwise arc.

NC Code Example:

```
G0X0.001Z1;START POINT
G72X.5Z.5I0K.5Q.04F3
G80
```

Note: The program uses absolute arc centers. See 7.4.2 $ Code: Absolute Arc Centers, pg. 101.

<table>
<tr><th>Code</th><th>Explanation</th></tr>
<tr><td>G0X0.001Z1;START POINT</td><td>Rapid traverse to X0.001, Z1.
This is the arc starting point.</td></tr>
<tr><td>G72X.5Z.5I0K.5Q.04F3</td><td><table>
<tr><td>G72</td><td>Specifies a clockwise arc.</td></tr>
<tr><td>X.5Z.5</td><td>Sets the end point of the arc.</td></tr>
<tr><td>I0K.5</td><td>Sets the center point of the arc.</td></tr>
<tr><td>Q0.4</td><td>Sets the depth of cut to 0.04 inches.</td></tr>
<tr><td>F3</td><td>Sets feed rate to 3 inch/min.</td></tr>
</table></td></tr>
<tr><td>G80</td><td>Cancels the canned cycle.</td></tr>
</table>

<table>
<tr>
<td>Example Motion</td>
<td>Example programs are shown below. The dotted lines represent the resulting tool paths.
<table>
<tr><th>Canned Cycle Command</th><th>NC Code</th><th>Motion</th></tr>
<tr><td>G72 example (as discussed above)</td><td>G0X0.001Z1;START POINT
G72X.5Z.5I0K.5Q.04F3
G80</td><td></td></tr>
<tr><td>G73 example (G73 replaces G72 in previous example, and the I,K coordinates are adjusted to reposition the arc center.)</td><td>G0X0.001Z1;START POINT
G73X.5Z.5I.5K1Q.04F3
G80</td><td></td></tr>
</table>
</td>
</tr>
</table>

8.4.3. G77: Side Turning Canned Cycles

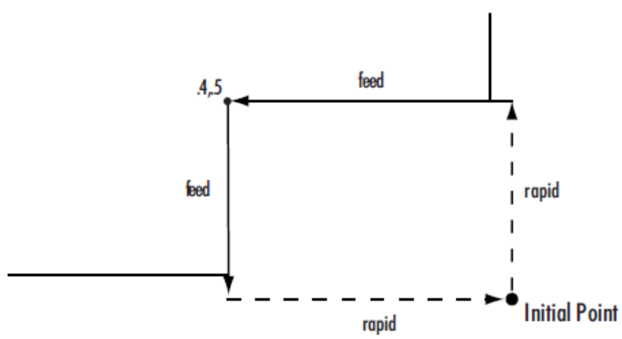
The G77 code can be used to perform four functions as listed below.

Section Contents: G77 Side Turning Canned Cycles

8.4.3.1. Straight Side Turning

Straight side turning is greatly simplified by the use of the G77 code. The tool makes a rapid move to the X depth, feeds to the Z coordinate, then backs out and returns to the start point.

The following is an example of code using the G77 code for straight side cuts.

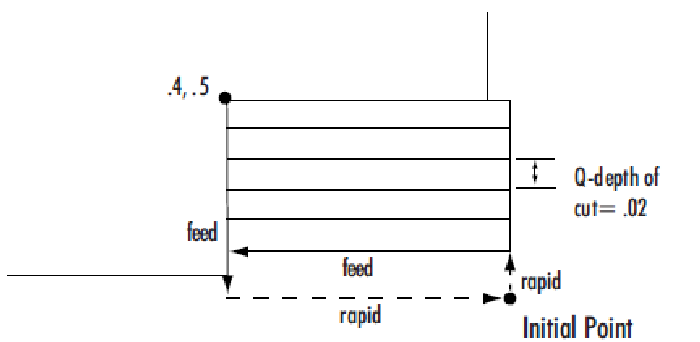
<table>
<tr><td colspan="2">NC Code Example:
G00X.6Z.8
G77X.4Z.5F10
G80</td></tr>
<tr><td>Code</td><td>Explanation</td></tr>
<tr><td>G00X.6Z.8</td><td>Rapid traverse to the start point, X.6Z.8.</td></tr>
<tr><td>G77X.4Z.5F10</td><td><table><tr><td>G77</td><td>Starts the canned cycle.</td></tr><tr><td>X.4Z.5</td><td>Sets the end point of the side cut.</td></tr><tr><td>F</td><td>Sets the feed rate to 10 inch/min.</td></tr></table></td></tr>
<tr><td>G80</td><td>Cancels the G77 canned cycle.</td></tr>
<tr><td>Example Motion</td><td>The example code will result in the motion shown below.
</td></tr>
</table>

8.4.3.2. Roughing Cuts

Performing roughing cuts with the G77 code is similar to straight side turning, except that a Q code is placed in the block to specify the depth of each cut.

Without the Q code, the tool will make the side cut to the specified depth in a single cut. With the Q code the tool will make multiple shallower cuts of the specified depth of cut, until the final specified depth is reached.

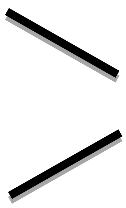
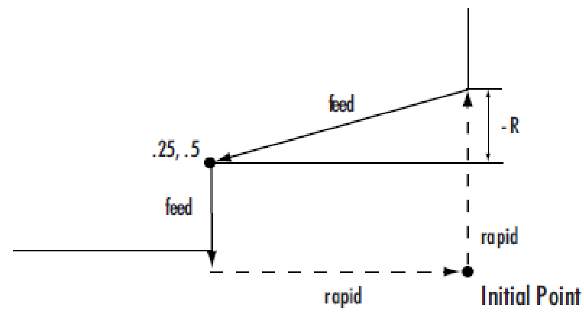
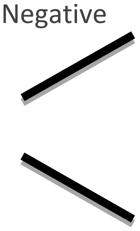
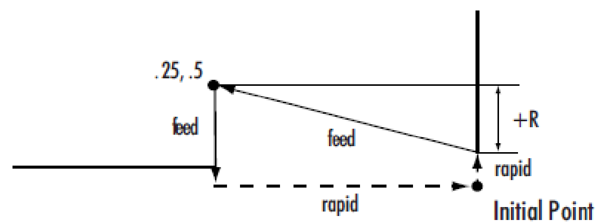
The following is an example of code using the G77 code for roughing cuts.

<table>
<tr><td colspan="2">NC Code Example:
G00X.6Z.8
G77X.25Z.5Q.02F10
G80</td></tr>
<tr><td>Code</td><td>Explanation</td></tr>
<tr><td>G00X.6Z.8</td><td>Rapid traverse to the start point, X.6Z.8.</td></tr>
<tr><td>G77X.4Z.5Q.02F10</td><td><table>
<tr><td>G77</td><td>Starts the canned cycle.</td></tr>
<tr><td>X.4Z.5</td><td>Sets the end point of the side cut.</td></tr>
<tr><td>Q.02</td><td>Sets depth of cut to 0.02 inch per cut</td></tr>
<tr><td>F</td><td>Sets the feed rate to 10 inch/min.</td></tr>
</table></td></tr>
<tr><td>G80</td><td>Cancels the G77 canned cycle.</td></tr>
<tr><td>Example Motion</td><td>The example code will result in the motion shown below.

</td></tr>
</table>

8.4.3.3. Tapering

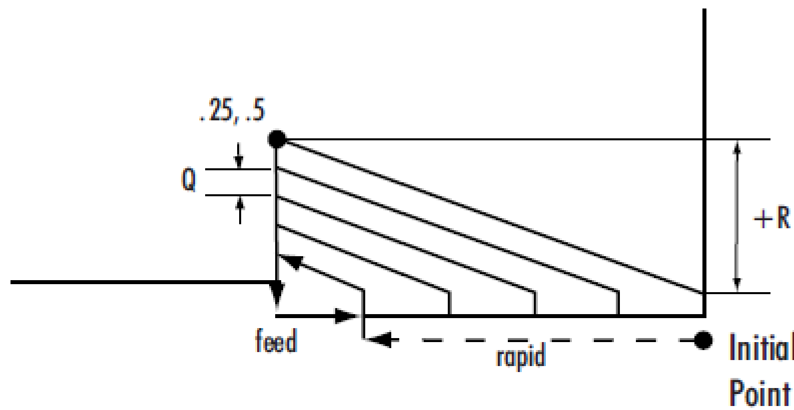
Tapers are created by using an R code in conjunction with G77.

The R value specifies the difference in depth between the start of the taper and the end of the taper (specified by the X code). The sign of the slope produced (positive or negative) depends on whether R is set to a positive or negative value, as in the examples below.

NC Code Example: Tapering			
Slope Required	**Sign of R**	**Example Code**	**Example Motion**
Positive 	Negative	G77X.25Z.5R-.1F10	
Negative 	Positive	G77X.25Z.5R.1F10	

The example below illustrates tapering with roughing cuts, using R and Q codes.

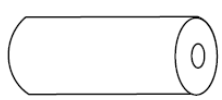
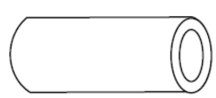
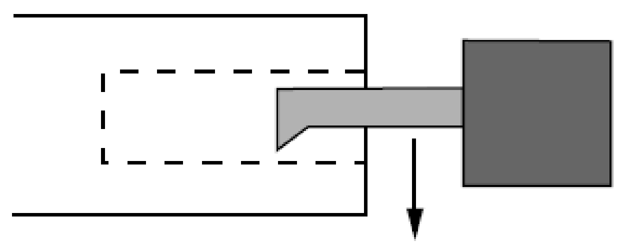
NC Code Example: G77X.25Z.5Q.02R.1F10	
Code	**Explanation**
G77X.25Z.5	Starts the G77 canned cycle, specifying the position of the end of cut.
Q.02	Specifies a depth of cut of 0.02 inches
R.1	Specifies a negative slope, with the taper starting 0.1 inches further from the axis than the final cut depth.

Example Motion	The code G77X.25Z.5Q.02R.1F10 will produce a negative taper, ending at X.25Z.5, starting at a depth 0.1 inches below that position. The taper will be cut with roughing cuts of depth 0.02 inches. 

8.4.3.4. Boring

Boring (an inside turning cycle) is achieved by specifying a smaller X value at the beginning of the operation than at the end. The tool moves into the workpiece, nears the center, and then moves outward with each cut, as shown below.

Note that boring requires that a hole be pre-drilled to allow the boring tool into the workpiece.

Before Boring	After Boring	Boring Operation
		

An R value can be added to turn an inside taper.

The example below illustrates a boring cycle.

NC Code Example:

```
G00X.25Z1.1
G77X.375Z.5Q.03F10
G80
```

Code	Explanation
G00X.25Z1.1	Moves the tool to the outside diameter of the hole at the face of the stock.
G77X.375Z.5	Starts a boring cycle that will create an internal cavity with the coordinate X.375Z.5 specifying the deepest internal corner of the bore.
Q0.03	Specifies a depth of cut of 0.03 inches. Multiple cuts of this depth will be made until the depth X.375 is reached.

F10	Specifies the feed rate as 10 inches/minute.
G80	Cancels the canned cycle.

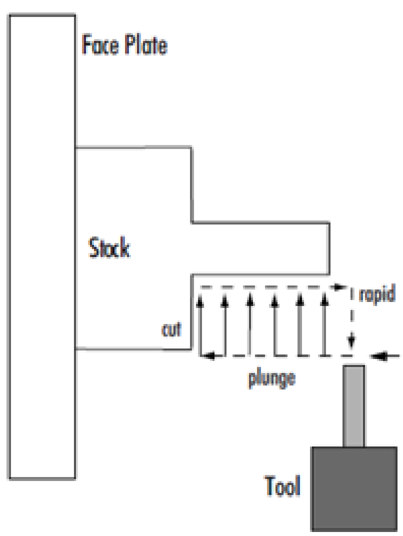
8.4.4. G79: End Turning Cycles

The G79 code can be used to perform an end turning cycle. The cuts are made on the face of the stock. As for G77, G79 can be used with Q codes (for setting depth of cut) and R codes (for specifying a taper).

A typical use of the G79 code is shown in the example below.

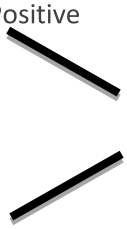
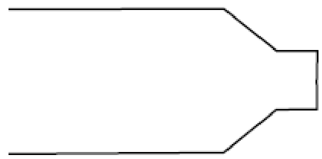
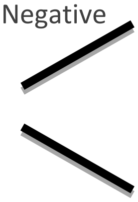
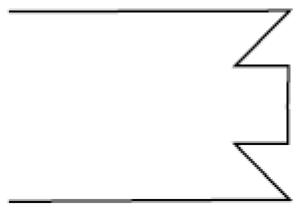
NC Code Example:

```
G0X.375Z1; START POINT
G79X.2Z.8Q.03F10
G80
```

<table>
<tr><th>Code</th><th>Explanation</th></tr>
<tr><td>G0</td><td>Moves the tool to the start point above the face of the part.</td></tr>
<tr><td>G79X.2Z.8Q.03F10</td><td>
<table>
<tr><td>G79</td><td>Starts the canned cycle.</td></tr>
<tr><td>X.2Z.8</td><td>Specifies the final depth of the cut as X.2 and the length of the cut in the Z direction as Z.8.</td></tr>
<tr><td>Q.03</td><td>Sets depth of cut to 0.03 inch per cut.</td></tr>
<tr><td>F10</td><td>Sets the feed rate to 10 inch/min.</td></tr>
</table>
</td></tr>
<tr><td>Example Motion</td><td>The example code will result in the motion shown below.


</td></tr>
</table>

The G79 code can be used with R codes to combine end turning and tapering, as illustrated below.

NC Code Example: End Turning with Tapering		
Slope Required	**Sign of R**	**Example Part**
Positive 	Negative	
Negative 	Positive	

8.4.5. G80: Cancelling a Canned Cycle

Use the G80 code to cancel a canned cycle. This code cancels the currently running canned cycle and resumes normal operation. All other turning data is canceled as well.

You can also cancel canned cycles by using a G00 or G01 code, as a G80 code is automatically performed as part of G00 and G01.

8.4.6. G81 & G83: Straight and Peck Drilling

The G81 code performs straight drilling operations. G83 is used for peck drilling.

The R code is used to specify a Z axis reference point for peck drilling. The point can be at the material surface or at another reference point. By specifying an R value of zero, the tool will return to the initial point after drilling to point Z.

A typical use of the G81 and G83 codes is shown in the following example.

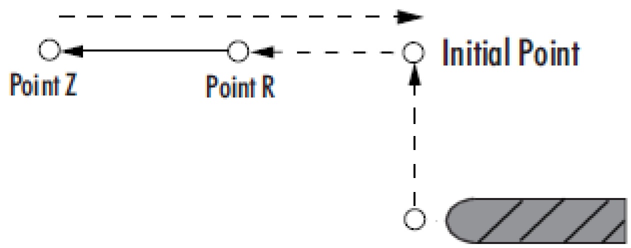
NC Code Example:

```
G0X0Z1;RAPID TO 0,1
G81Z.9R1F2;CENTER DRILL TO DEPTH OF Z.9 FROM Z1 FEED 2, RAPID TO INITIAL POINT
G83Z.5R1Q.1F3;PECK DRILL TO Z.5 FROM Z1 EACH PECK .1, RAPID TO POINT R
G80;CANCEL CANNED CYCLE
M2;END PROGRAM
```

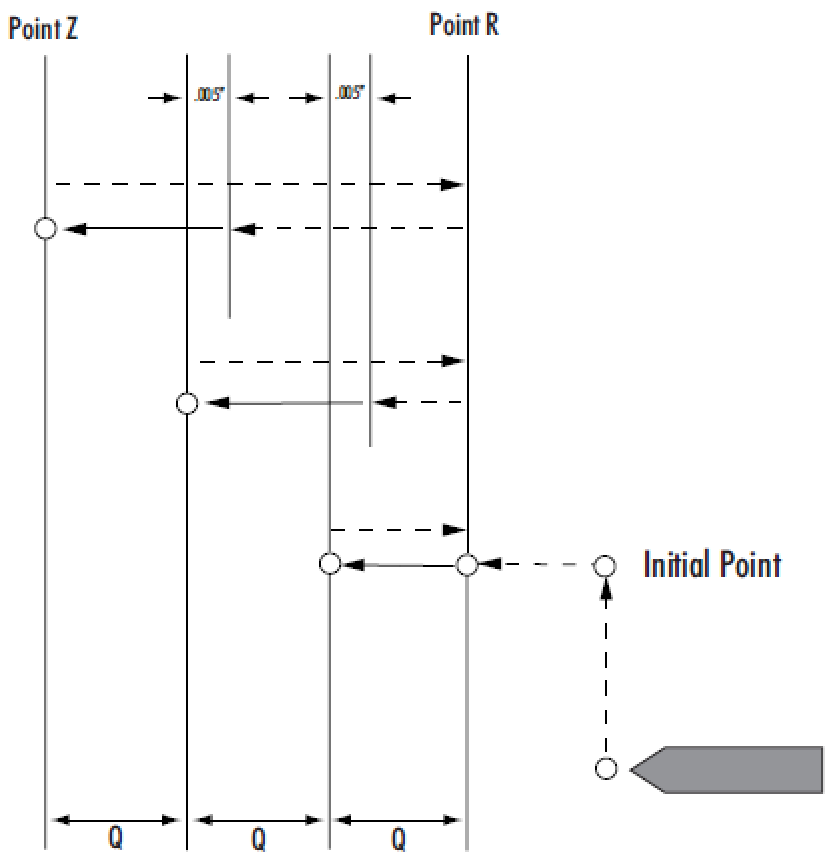
<table>
<tr><th>Code</th><th>Explanation</th></tr>
<tr><td>G0X0Z1;RAPID TO 0,1</td><td>Rapid motion to X0 Z1.</td></tr>
<tr><td>G81Z.9R1F2;CENTER DRILL TO DEPTH OF Z.9 FROM Z1 FEED 2, RAPID TO INITIAL POINT</td><td><table><tr><td>G81Z.9</td><td>Straight drill to Z0.9</td></tr><tr><td>R1</td><td>Retract to Z1</td></tr><tr><td>F2</td><td>Feed rate = 2 inches/min</td></tr></table></td></tr>
<tr><td>G83Z.5R1Q.1F3;PECK DRILL TO Z.5 FROM Z1 EACH PECK .1, RAPID TO POINT R</td><td><table><tr><td>G83Z.5</td><td>Peck drill to Z.5</td></tr><tr><td>R1</td><td>Retract to Z1</td></tr><tr><td>Q.1</td><td>Drill a maximum of 0.1 inch per peck.</td></tr><tr><td>F3</td><td>Feed rate = 3 inches/min</td></tr></table></td></tr>
<tr><td>G80</td><td>Cancel canned cycle.</td></tr>
<tr><td>M2</td><td>End program.</td></tr>
</table>

Example Motion

The straight drill cycle will result in motion as shown below.



The peck drill cycle will result in motion as shown below.



8.5. SUBPROGRAM PROGRAMMING

Subprograms are used to execute repetitive routines in an NC program. The subprogram is entered in the NC code only once, but can be called and run any number of times. This is especially useful if the turning operation you wish to repeat is lengthy or complex.

The NC codes used for sub-programming on the BenchTurn turning center are listed in the following table.

Info Table: Sub-programming NC Codes	
Code	**Description**
M98	Call to subprogram.
M99	Return from subprogram.
P	The P code is used to reference the first line of the subprogram (which begins with an O code). The P code immediately follows an M98. The P code can also be used with an M99 code to specify the line number in the main program to be run next.
L	The L code is used as a loop counter when used in sub-programming. The computer executes the subprogram the number of times set by the L code. For instance, if the code is L5, the subprogram is executed five times. After the fifth cycle, the program will not restart the subprogram but will instead continue to the next step in the program. (Optional)
O	The O code replaces the N code on the first line of a subprogram.

The general program flow when using a subprogram is illustrated in the following table.

Info Table: Subprogram Flow	
No.	**Description**
1	An M98 and P code located in the main program call the subprogram. The P code specifies which subprogram to run. Each subprogram is labeled with an O code instead of an N code.
2	The main program is interrupted, and the subprogram begins to run.
3	An M99 code in the subprogram ends the subprogram. The main program recommences from the point at which it was interrupted.

Subprogram calls can also be nested within other subprogram calls. This means that while a subprogram is being executed, it can call another subprogram. The default number of levels that subprograms can be nested is 20 levels deep.

The sample subprogram below makes a series of light chamfered grooves. It is set up for stock with a 0.75" diameter and 2" length so you should use a piece slightly longer than that, say 0.75" X 2.5", if you plan to test it.

Note that the file uses absolute programming in the main program and incremental programming in the subprogram. This makes it necessary to use the G90 code after the subprogram is executed to allow motion back to the start point.

NC Code Example:

```
;THIS FILE FOR 2.5 INCH BY 0.75 INCH STOCK MOUNTED IN CHUCK
;USE WITH A PROFILING TOOL NORMAL SIDE ORIENTATION
;SET THE START POINT AT Z2 X0.375
G0G90M03;ABSOLUTE PROGRAMMING
G0X0.380Z2
M98P1000L4;CALLS SUBPROGRAM 1000 AND EXECUTES IT 4 TIMES
G90;ABSOLUTE PROGRAMMING
G0X0.38
G0Z2
M02;END OF PROGRAM
O1000;START OF SUBPROGRAM
G91;INCREMENTAL PROGRAMMING SELECTED
G1X-0.040Z-0.040F3
G1Z-0.125 G1X0.040Z-0.040 G0Z-0.20
M99;END OF SUBPROGRAM
```

Note: Only selected lines are explained below.

Code	Explanation
M98P1000L4;CALLS SUBPROGRAM 1000 AND EXECUTES IT 4 TIMES	M98P1000: Call the subprogram whose O code is O1000. L4: Run the subprogram 4 times.
O1000;START OF SUBPROGRAM	Indicates the start of subprogram 1000.
M99;END OF SUBPROGRAM	Indicates the end of the subprogram.

9. Multiple Tool Programming

The BenchTurn 7000 turning center is equipped with a 4 position tool-turret as standard. Using multiple tool programs provides the advanced user with the ability to create more complex parts on the turning center.

This section presents instructions for writing NC code using multiple tools.

There are six basic steps in setting up the turning center for multiple tool operation:

Procedure Outline: Multiple Tool Programming			
No.	**Description**	**Section**	**Page**
1	In the control software, specify the tools being used.	9.1	153
2	In the control software, specify how the tools are configured in the tool turret.	9.2	153
3	Write the NC program.	9.3	154
4	Define and configure one tool as a reference tool.	9.4	155
5	Set the offsets for the other tools, relative to the reference tool.	9.5	156
6	Test the NC program.	9.6	158

This section also presents a tutorial on multiple tool programming.

Section Contents: Multiple Tool Programming		
Description	**Section**	**Page**
Tutorial: Running a Multi-tool Program	9.7	159

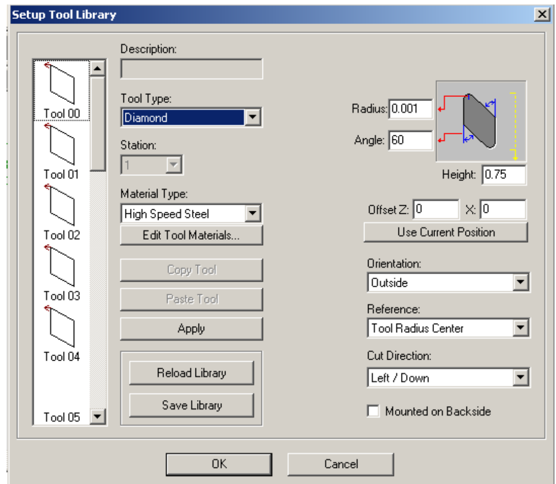
9.1. SPECIFYING THE TOOLS

For the control software to successfully run a multi-tool program, you must specify the tools being used.

Follow the procedure below to specify the tools.

Procedure: Specifying the Tools

1. Click **Tools** | **Setup Library** to access the Setup Tool Library.
2. Click a tool in the left panel to select it, and enter the relevant settings in the fields on the right.
3. Click **OK** to save changes made and close the window.



9.2. CONFIGURING THE TURRET

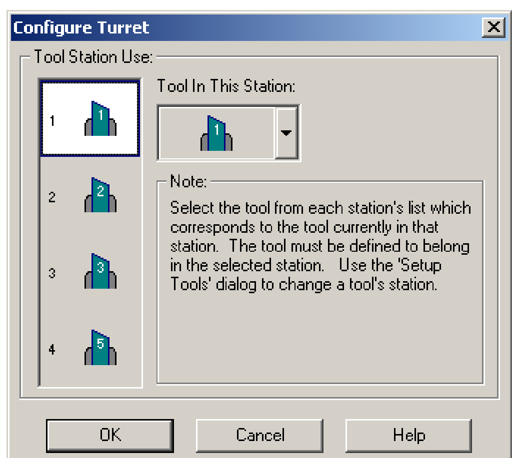
After having specified the tools used, you have to specify how they are arranged in the tool turret.

Follow the procedure below to configure the turret.

Procedure: Configuring the Turret

1. Click **Tools** | **Configure Turret** to access the Configure Turret window.

   The Tool Station Use area shows which tool is currently present in each of the four turret positions.
2. To specify a different tool at a position, click the turret position in the Tool Station Use area, and then select the tool at that position from the Tool in This Station menu.
3. Click **OK** to save changes made and close the window.



9.3. WRITING AN NC PROGRAM FOR MULTIPLE TOOLS

The T code is used in the NC program to offset the cutter so that the NC program becomes independent of the cutter length. The offset for each tool is specified in the Setup Tool Library window. This means you can replace a worn tool with a tool of a different length without changing the NC program; you need only enter a new offset.

The T code can be located anywhere within the block of NC code, but is normally placed after any of the G codes.

Use the M06 code with a T code for multiple tool operations. If you do not use an M06 code, the machine will use the tool currently selected in the turret and interpret the T code as an offset reference only.

When used with an M06, the T code indicates, by number, which tool and which offset to use. For example, M06T2 tells the turret to rotate to Tool #02 and use the offset specified for Tool #02.

Safety	Before any tool change, the turret must be distanced from the workpiece to prevent collisions.

The tool that is shown in the Machine Info window is the default tool for the start of a program. The control software assumes that tool is in use unless another tool is specified with a T code. Once a tool is specified, any X and Z coordinates are applied to that tool until another T code is encountered in the NC program.

In the example below, the Machine Info window shows that Tool #01 is in use. Unless the program includes a T code, the offset specified for Tool #01 in the Setup Tool Library will be used throughout.

Machine Info

Tool	01	TlSpec	TSC
Pass	001	Coord	Work

The code below demonstrates the use of T and M06 codes for changing tools.

Example Code: Tool Change

```
N7 ; Tool #1: 'Left Hand Diamond'
N8 ; Tool #2: 'Facing Grooving'
N9 G70 ; Inch Units
N10 M03 S1500
N11 M06 T01 ; Toolchange to Tool #01
```

Code	Explanation
N7 ; Tool #1: 'Left Hand Diamond' N8 ; Tool #2: 'Facing Grooving'	Comment lines describing the two tools in use.
N9 G70 ; Inch Units	Set units to inches.
N10 M03 S1500	Turn on the spindle, with speed of 1500RPM.

N11 M06 T01 ; Toolchange to Tool #01	Rotate the turret to Tool #01 and use the offset for Tool #01.

9.4. ESTABLISHING THE REFERENCE TOOL

When using multiple tools, a reference tool, normally Tool #01, is set to zero for the X and Z axes. This establishes a reference tool position which is used as a reference point for additional tools.

In the procedure below, we will use Tool #01 as the reference tool and Tool #02 as the additional tool.

Procedure: Establishing the Reference Tool

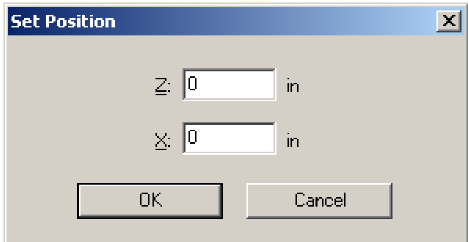
1. Decide on a reference point. A reference point is a point on the workpiece, or on a gauge, to which you will jog the tip of each tool.
2. Insert the workpiece or gauge to be used for the reference point.
3. Ensure that Tool #01 is positioned in the spindle.
4. Close the safety door.
5. Open the Jog Control Panel if not already open. To do so, click **View** | **Jog Control** in the Main Menu.
6. If using a workpiece for the reference point, turn the spindle on, setting it to its minimum speed. If using a gauge, ensure that the spindle is off.
7. Jog the tool slowly until it just touches the reference point. Follow the guidelines below.
   a. Use Continuous mode (selected in the Step Zone area of the Jog Control panel) to move the tool towards the reference point, but stop motion before the tool is in range of touching the reference point.

   b. Once the tool is near the reference point, switch to Step mode, using a small step size. Jog the tool, step by step, until it touches the reference point.

8. Click **Setup** | **Set Position** the Main Menu. The Set Position window opens.



9. Set both Z and X to 0.

10. Click **OK**.

Tool #1 is now established as the reference tool.

9.5. SETTING TOOL OFFSETS

Once the reference tool is established, you can define offsets for the other tools to be used. You will first determine what the offset settings should be. You will then enter those offsets into the control software.

Product Care

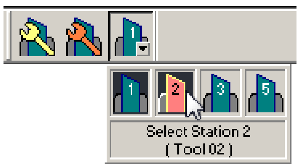
Do not remove the tools from the turret after establishing the reference tool and setting the offsets of other tools.

The entire procedure must be repeated each time tools are installed in the turret.

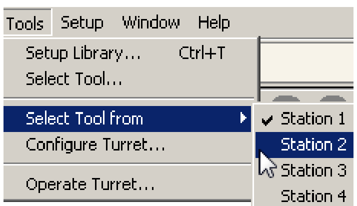
Follow this procedure to set tool offsets.

Procedure: Setting Tool Offsets

1. Establish the reference tool as described in the previous section, 9.4 Establishing the Reference Tool, pg. 155.
2. Jog the tool turret to a safe distance from the workpiece or gauge to prevent a collision when changing the tool.
3. Select Tool #02 from the Turret Control toolbar as shown below,

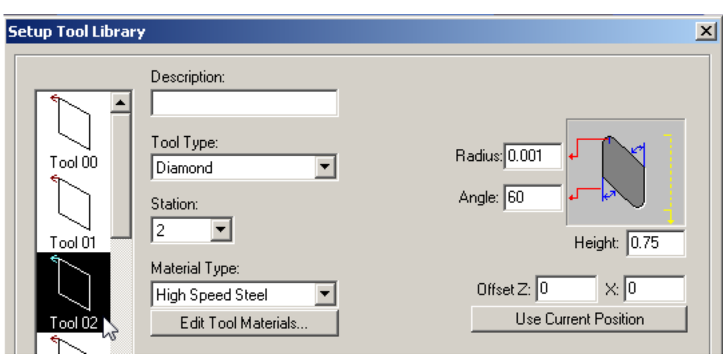


or click **Tools** | **Select Tool From**, and choose Tool #02.

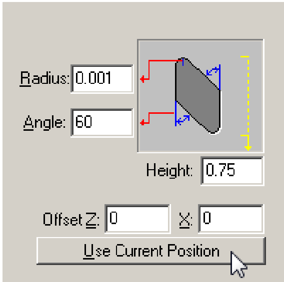


4. Jog the tool slowly until it just touches the reference point. As in the previous section, use Continuous mode to approach the reference point, and Step mode to jog the tool until it touches the reference point.
5. Take note of the Z and X coordinates displayed in the Position Window. These are the offset values for the selected tool.

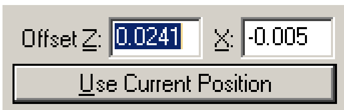
6. Click **Tools** | **Setup Library** in the Main Menu.
7. Select Tool #02 from the list at the left.



8. **Click Use Current Position.**



The offset values are automatically filled with the position values found in step 5.



9. Click **OK** to save the changes and close the window.

   The offset for Tool #02 is now defined.

10. Repeat this procedure for Tools #03 and #04, if they will be used.

9.6. TESTING THE MULTI-TOOL PROGRAM

As for any NC program, a multi-tool NC program must be tested by performing a dry run (running the program without a workpiece in place), before running the program with a workpiece in place.

Safety

Always complete the Safety Checklist (see 1.2 Safety Checklist, pg. 5) before running a program on the turning center.

Throughout the test, be prepared to press the emergency stop button on the turning center in case of a tool crash.

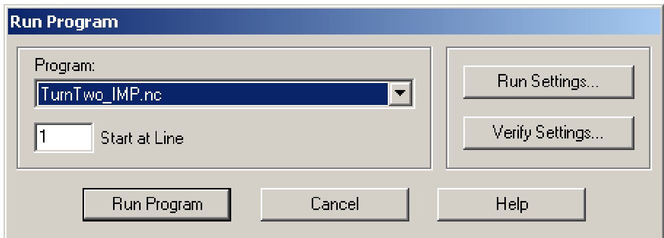
Follow this procedure to perform a dry run on a multi-tool program.

Procedure: Testing a multi-tool program

1. With all tools in their respective positions in the tool turret, close the safety door, put on your safety goggles, and complete the Safety Checklist (see 1.2 Safety Checklist, pg. 5).

2. Click **Program** | **Run/Continue** in the Main Menu, or click the Run button in the Standard Toolbar.

   The Run Program window is displayed.



3. Ensure that *Start at Line* is set to **1**.

4. Click **Run Program**.

5. Throughout the test, be prepared to press the emergency stop button on the turning center in case of a tool crash.

6. Observe the turning process, noting any corrections to be made in the NC program.

   After successfully testing the program in a dry run, you can then run the program with a workpiece in place.

9.7. TUTORIAL: RUNNING A MULTI-TOOL PROGRAM

This section provides detailed instructions for turning a sample part using multiple tools, covering the entire process from NC program verification through to turning a complete part on the BenchMill 7000. The tutorial will follow the following procedure.

Procedure Outline: Tutorial			
No.	**Description**	**Section**	**Page**
1	Review safety procedures.	9.7.1	160
2	Prepare required tools and materials.	9.7.2	160
3	Open the sample NC file.	9.7.3	160
4	Define the tools to be used.	9.7.4	160
5	Configure the tool turret.	9.7.5	161
6	Configure the Verify settings.	9.7.6	161
7	Verify the program.	9.7.7	162
8	Establish the reference tool.	9.7.8	162
9	Set the offsets for the other tools.	9.7.9	163
10	Test the program without a workpiece in place.	9.7.10	163
11	Mount the workpiece.	9.7.11	163
12	Run the program.	9.7.12	163

9.7.1. Reviewing Safety Procedures

Like any other power tool, the BenchTurn Turning Center is a potentially dangerous machine if operated in a careless manner. The importance of safely operating the BenchTurn Turning Center, including the need for protection against personal injury and the prevention of damage to the equipment, cannot be stressed enough.

Safety	Ensure that you are familiar with all safety guidelines in 1 Safety Guidelines, pg. 1, before continuing.

9.7.2. Preparing Tools and Materials

For this tutorial you will require the following:

Tools and Materials List: Tutorial
One 3" (length) x 0.75" (diameter) cylindrical piece of machinable brass, aluminum, Delrin, or wax

9.7.3. Opening the Sample NC File

Open the sample file *TurnTwo_IMP.nc*.

For instructions on opening NC files, see 5.5 Opening an NC File, pg. 66.

9.7.4. Defining the Tools

The sample NC program uses three tools to turn the part. These tools must be defined in the **Setup Tool Library**.

For instructions on defining tools, see 9.1 Specifying the Tools, pg. 153.

Specify the three tools as detailed in the following table.

Tool Specifications		
	Tool 1	**Tool 2**
Description	Left Hand Diamond	Facing Grooving
Tool Type	Diamond	Grooving
Station	1	2
Material Type	High Speed Steel	High Speed Steel
Radius	0.0001	0.0001
Angle	30	0.01
Orientation	Outside	Facing
Reference	Tool Radius Center	Tool Radius Center
Cut Direction	Left/Down	Right/Up

9.7.5. Configuring the Tool Turret

Once the tools are defined, you must configure the tool turret, specifying which tool is present in each of the turret positions.

For instructions on configuring the turret, see 9.2 Configuring the Turret, pg. 153.

Configure the tool turret as detailed in this table.

Turret Configuration		
	Tool Station 1	**Tool Station 2**
Tool	Tool 01#	Tool #02

9.7.6. Configuring the Verify Settings

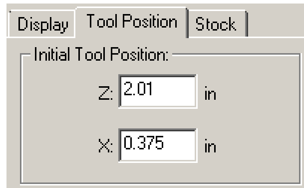
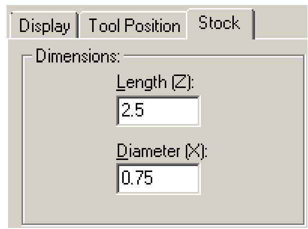
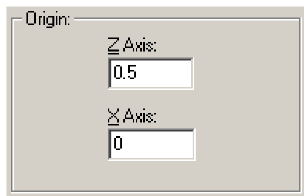
Before running the NC program on the turning center, it must be tested in the Verify window of the control software.

The Verify settings must be configured correctly so that the verification process will test the NC program accurately.

For instructions on configuring the Verify settings, see 5.6.3 Configuring Verify Settings, pg. 71.

For an example on configuring the Verify settings, see 6.5 Configuring the Verify Settings, pg. 79.

Make the following settings in the Verify Program window.

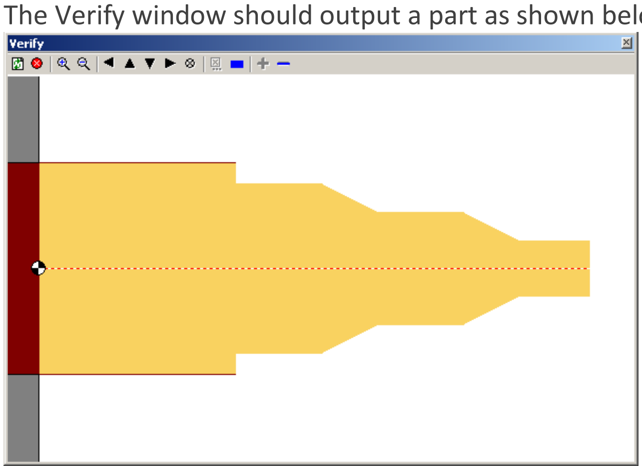
Verify Settings		
	Setting	**Tool 2**
Initial Tool Position	Z = 2.01 inch X = 0.375 inch	
Stock Dimensions	Z = 2.5 inch X = 0.75 inch	
Origin	Z = 0.5 X = 0	

9.7.7. Verifying the Program

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

For an example of verifying a program, see 6.7 Verifying the Program, pg. 86.

The Verify window should output a part as shown below.

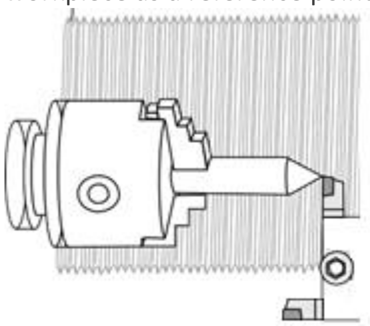


9.7.8. Establishing the Reference Tool

When using multiple tools, a reference tool, normally Tool #01, 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 instructions on establishing a reference tool, see 9.4 Establishing the Reference Tool, pg. 155.

Set the reference point at the bottom of the front right corner of the workpiece (the end of the stock furthest from the chuck and at the edge of the stock). You can also select another point on the workpiece as a reference point, as shown in the illustration below.



9.7.9. Setting Tool Offsets

The first tool for this program is a Left-Hand tool used for performing the facing cuts. The second tool is a cutoff tool. In this situation the facing tool is the reference tool. Only the offset of the cutting tool must be set.

For instruction on setting tool offsets, see 9.5 Setting Tool Offsets, pg. 156.

9.7.10. Testing the Program

As for any NC program, a multi-tool NC program must be tested by performing a dry run (running the program without a workpiece in place), before running the program with a workpiece in place.

For instructions on performing a dry run, see 9.6 Testing the Multi-tool Program, pg. 158.

9.7.11. Mount the Workpiece

Once you have performed a successful dry run, prepare for actual turning by mounting the workpiece.

For instructions on mounting the workpiece, see 6.9 Mounting the Workpiece, pg. 93.

9.7.12. Running the Program

Now that the program has been checked in the Verify window and by performing a dry run, the program can finally be run on an actual workpiece.

For instructions on running the program, see 6.10 Running the Program, pg. 96.

10. An Introduction to CNC Turning

This section provides a basic introduction to CNC turning. The following topics are covered.

10.1. UNDERSTANDING COORDINATE SYSTEMS

To understand how the coordinate system works in turning, the following concepts must be explained:

10.1.1. X and Z Axes

In turning, the Z axis is always the spindle axis. On the BenchTurn 7000 turning center, the Z axis is horizontal. The X axis is also horizontal, but perpendicular to the Z axis and parallel to the cross slide.

In NC programming, the programs are written as though the workpiece is stationary and the tool is moving. The motion of the tool from right to left or left to right is along the Z axis. The motion of the tool from front to back is along the X axis.

The location and direction of the X and Z axes are shown in the graphic below.

10.1.2. Machine Home Position

The machine home is a specific and factory-set location to which the tool can be sent through the control software.

The machine home location is usually located on the Z axis centerline on the plane along the front of the chuck. The machine home position is usually some distance away from the chuck (in the Z+ direction), so that sending the tool to the home position will not pose an impact hazard.

The machine uses the home position as a reference point for all operations. If the machine is not homed (sent to the machine home position), it cannot accurately locate the workpiece on the cross slide.

The machine is homed by clicking **Setup** | **Set/Check Home** in the Main Menu.

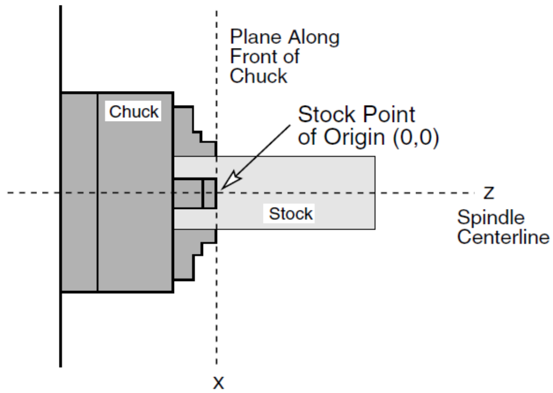
It is always advisable to home the machine before running a program on it.

10.1.3. Work Coordinates

The location of the tool at any time can be described by its position along the X and Z axes. However, the origin point (0,0) is not factory-set and can be defined as any point within the work area.

The user defines the point of origin, and the machine will measure X and Z coordinates from that point. (The machine home position is almost never used as the point of origin, so the coordinates of the home position are seldom (0,0).)

The point of origin can be located anywhere on the workpiece, but is often set to the center of the workpiece where it exits the chuck, as shown below.



To define the origin as illustrated above, jog the tool to a specific location (such as a corner of the workpiece) and define the coordinates of that point based on knowledge of the workpiece dimensions. In that way, the coordinates of the stock point of origin are indirectly defined as being (0,0).

For example, for a workpiece that is 3” long with a 0.75” diameter: send the tool to the position at the top edge of the front of the workpiece, and define that point as X= 0.375, Z= 2.5. The origin is now set to X = 0, Z = 0.

10.1.4. Multiple Coordinate Systems

For more advanced operations, such as turning multiple parts on the same piece of stock, set up multiple coordinate systems.

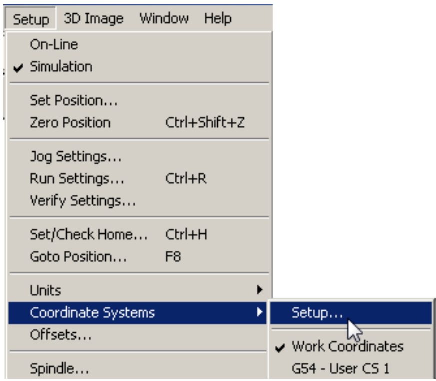
For information on how coordinate systems are activated in an NC program, see 7.4.6.7 The Coordinate System Group, pg. 107.

For example, if you have an NC program that machines a complex shape and you want to machine that shape in multiple places on the same part, use the following procedure:

Procedure: Using Multiple Coordinate Systems

1. Move the tool tip to the work coordinate point of origin (0,0) using the **Set Position** command from the **Setup** menu.

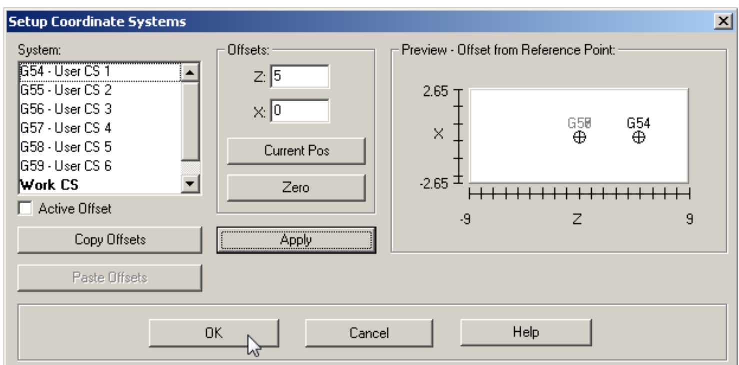
2. Select the **Coordinate Systems** command from the **Setup** Menu.



The Setup Coordinate Systems window is displayed.

3. Select one of the **G54** to **G59** codes

4. Enter the coordinates for the first workpiece, and click on **Apply**.



5. Repeat this procedure for as many coordinate systems as necessary (up to 6) by setting up a coordinate system for each point on the part that corresponds to the zero point of the shape you are turning.

6. In your program, use the Coordinate Systems group G codes to switch between coordinate system and run the subprogram that contains the NC code for the complex shape.

10.2. SETTING SPINDLE SPEEDS

Spindle speed refers to the rotational speed at which the spindle rotates the workpiece around its Z axis. Spindle speed is usually defined in units of rotations per minute (RPM).

When selecting a spindle speed, the following factors must be taken into account.

Info Table: Spindle Speed Factors	
Factor	**Description**
Workpiece diameter	Turning speed is inversely proportional to the diameter of the workpiece; the larger the workpiece, the slower the recommended turning speed.
Relative material hardness	Turning speed is inversely proportional to the relative hardness of the material; the harder the material, the slower the recommended turning speed.
Heat production	High spindle speeds may produce excess heat which can cause the workpiece to expand. If the workpiece expands, the cutting tool will rub rather than cut the material, resulting in a poor surface finish.
Material type	Some materials require higher spindle speeds to ensure a good finish.
Load on spindle motor	Heavy cuts at low speeds will make the motor run hotter than lighter cuts at higher speeds. The selected feed rate and depth of cut should not cause the spindle motor to greatly lose speed or cause the tool to chatter against the workpiece.

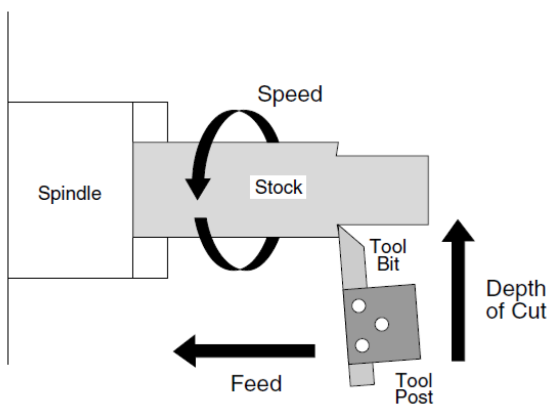
10.3. SETTING FEED RATE AND DEPTH OF CUT

Feed rate (also known as feed), and depth of cut (also known as cut) are central terms in NC turning.

Normal turning on a turning center involves reducing the diameter of a workpiece. This is accomplished by advancing the cutting tool into the workpiece by an appropriate amount (depth of cut).

The rate of tool travel is called the feed rate..

These concepts are illustrated below.



When selecting a feed rate and depth of cut, the factors listed in the table below must be taken into account. You should consult your machinist's handbook for selecting a feed rate based on spindle speed and material type. Experience and experimentation will enable you to establish feed rates best suited to particular applications.

Info Table: Feed Rate and Depth of Cut Factors	
Factor	**Description**

Spindle speed	The feed rate and depth of cut must be suitable for the selected spindle speed.
Material used	Material properties may dictate maximum and minimum feed rates and depths of cut. Surface finishes may suffer if machining parameters are not set appropriately for the material.
Lubrication type	Some lubricants can allow for higher feed rates and deeper depths of cut than others.
Cutting tool type	Some cutting tools are capable of making deeper cuts and maintaining faster feed rates than others.
Machine capabilities	Excessive depth of cut and high feed rates place greater strain on the spindle than allowable.

10.4. SELECTING LUBRICANTS AND COOLANTS

Lubricants remove heat from the tool and workpiece and are often used when high production rates are required or when cutting very hard materials, such as stainless steel. A mixture of one part soluble oil to six parts water may be used on steel to assist in producing a smoother finish and to reduce tool chatter. Aluminum and aluminum alloys may require the use of paraffin, oil or kerosene to prevent chips from welding to the tool's cutting edge. Brass and cast iron are always turned dry.

When lubrication is necessary, small amounts of water-soluble cutting fluids are recommended for use on the BenchTurn turning center. Lubricants should be wiped from the machine after use, as some petroleum-based fluids may damage the electrical wiring insulation and other components.

Short runs of parts on Delrin or aluminum, such as would be performed in a school or college laboratory, do not require the use of coolant.

Take Note	The BenchTurn 7000 turning center is designed for flood cooling. A cooling accessory is available. Contact your dealer or Intelitek.

10.5. TOOL TYPES

Cutting tools are usually made from hardened steel and are ground to various shapes. The clearances ground behind cutting edges are adjusted for the type of material the tool will cut and the direction the tool will be fed along the workpiece. Tools are often ground to shape by the operator to suit a particular cutting requirement.

Insufficient clearance behind the cutting edge will cause the tool to rub against the workpiece. Excessive clearance will produce a ridged or wavy finish due to the small length of tool edge in contact with the workpiece. Standard tool types are: turning tools, side tools, parting tools and boring tools. Carbide tooling has become popular in recent years. Carbide is more brittle than steel, but has a longer tool life.

The following common tool types are described in the sections that follow:

10.5.1. Side Tools

Side tools are used to face-off the ends of shoulders or to make facing cuts in the surface of a workpiece held in a chuck. They may also be used as turning tools.

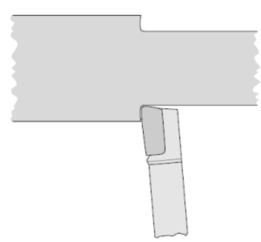
Side tools are used to reduce the workpiece to a desired diameter.

The shape of the cutting edge and the clearance (behind the point between the end of the tool and the workpiece) determine the surface finish of the workpiece. Rough cuts are made in small increments until the tool is within approximately 0.010 inch (0.25 mm) of the desired diameter. Final cuts are made at slow feed rates with a very shallow depth of cut.

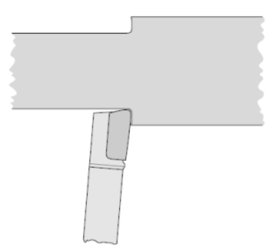
Side tools cut very flat surfaces and can be used to produce a part with an exact thickness.

Side tools are available for right side and left side turning:

Right side (or right-hand) tools feed from right to left.

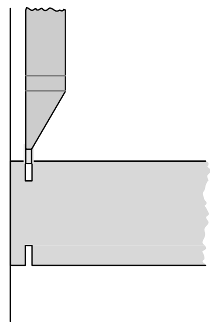


Left side (or left-hand) tools feed from left to right.



10.5.2. Parting Tools

A parting tool has a dual-edged, dovetail-shaped cutting end which is used to cut workpieces. The parting tool is plunged into the workpiece and the cross slide is moved across the lathe bed until the workpiece is severed, as shown here.

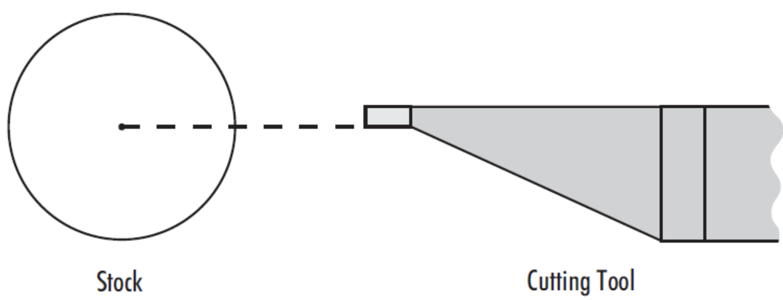


Parting tools are clamped to a special tool post with a minimum of over-hang for maximum rigidity and chatter-free operation.

The cutoff point should be located as close to the spindle center as possible. The cutting end of the parting tool should be perpendicular to the workpiece to avoid sideward drift. A small square is useful in aligning the tool perpendicular to the workpiece. Additionally, the height of the tip of the tool should be positioned so it is vertically aligned with the center of the stock.

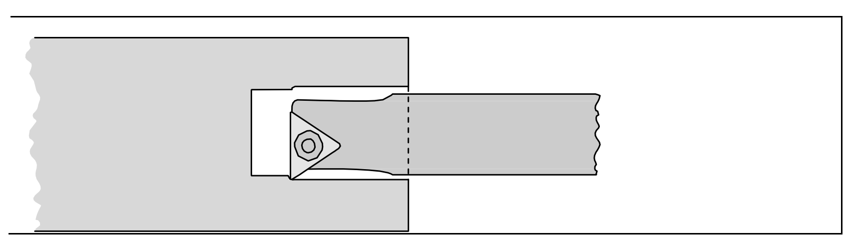
Cutoff operations are performed at a slow turning speed because the parting tool has a large amount of cutting edge in contact with the workpiece. If the tool chatters or produces noise, the turning speed and feed rate should be reduced.

Make sure to align the tip of the cutoff tool with the center of the stock, as shown below.



10.5.3. Boring Tools

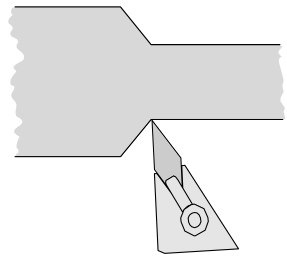
Boring tools are used to enlarge or modify a drilled or cored hole in a workpiece. Clearance must be maintained behind the cutting point of the tool



A slow feed rate and frequent tool withdrawals are required when boring because chips cannot freely escape from the hole. Depth of cut and feed rates must be reduced to avoid chatter. The tool should not be driven deeply into a hole. When boring a hole where a flat bottom is required, stop the feed at least 0.002 inch from the desired depth of the smaller hole being bored out.

10.5.4. Profiling Tools

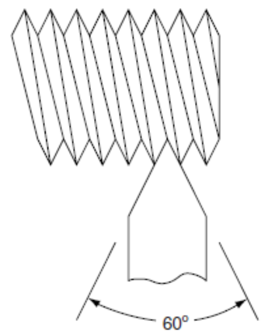
Profiling tools are quite popular in CNC applications because they can cut on both sides and in both directions. A profiling tool cuts in the same way as a side tool.



10.5.5. Threading Tools

Most threads are cut with a 60̂ threading tool. For fine threads, the tool can be ground to a sharp point. For coarse threads, you may wish to radius the point.

A threading tool is used for external threading operations. An inside threading tool is used for internal threading, such as on a nut.



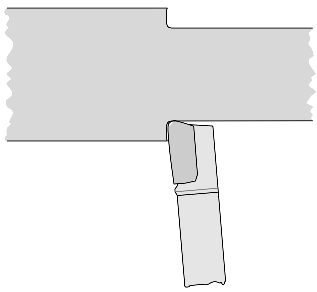
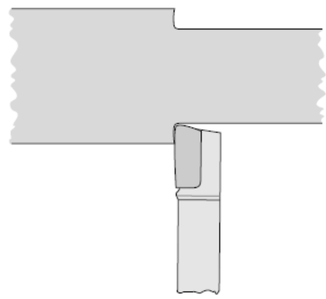
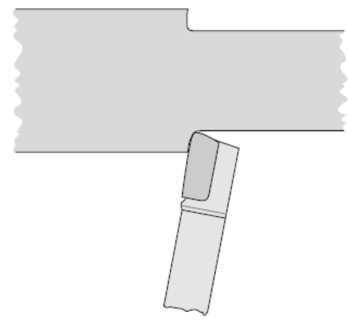
10.6. MOUNTING THE CUTTING TOOL

Each cutting tool used in the turning operation must be sharp and tightly inserted in the tool turret. The cutting edge of the tool must be on the centerline or just below the centerline (0.004 inch or 0.1mm maximum) of the axis of rotation of the lathe. For more information on aligning tools by adjusting their heights, see 4.4 Adjusting Turret Tool Heights, pg. 41.

The cutting tool is mounted by loosening the mounting screws at the top of the tool turret and positioning the tool in the slot beneath the mounting screws. To assure a rigid mounting and to avoid chatter, the cutting edge of the tool should not protrude more than necessary from the tool post. The further the tool extends from the tool post, the more chatter will occur.

Make sure that only the very tip of the tool is doing the cutting. All tools must be positioned in a manner that allows them to access all areas they are to cut. The exception to this is the cutoff tool, which should always be perpendicular to the stock.

The table below illustrates correct and incorrect tool angles for a right-handed turning tool.

Info Table: Correct and incorrect tool angles		
		
Correct	Incorrect	Incorrect

10.7. SHARPENING THE TOOLS

A cutting tool must be sharpened regularly to preserve its original cutting angle and shape. Longer tool life will be obtained from cutting edges if they are finished with a small oilstone. Only the cutting end and sides of the tool should be ground as required. Never grind the top face of the tool.
Alternatively, replace tools as required.

11. Automation Integration

This section provides information and instructions required to integrate the BenchTurn 7000 within a flexible manufacturing system (FMS).

Section Contents: Automation Integration

11.1. INTEGRATION INSTRUCTIONS

In order to be integrated into an FMS the BenchTurn 7000 must be able to work with machine related automation functions like an automated shield and an automated clamping device. It must also be able to communicate with a robot for the loading and unloading sequence and be able to run a G Code by command from an external control device, such as a robot program, device driver or other control entity.

This section provides instructions on how to integrate the BenchTurn 7000 with various other components.

Section Contents: Integration Instructions

11.1.1. Integrating with an Automated Shield (Pneumatic)

Take Note	Detailed installation instructions are provided with each optional accessory purchased.

The automated shield must be closed during machine operation to protect the operator, as shown in the photograph below (left). The shield must be opened to allow access for an automated loading device.

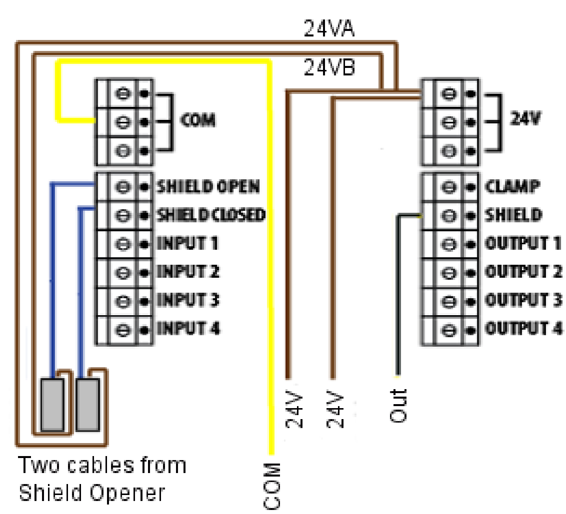
Sensors on the pneumatic piston send signals indicating its current open/closed status to the machine through an input port. The machine sends open/close commands to the shield through an output port.

The I/O ports are located at the right side of the machine, as shown in the photograph below (right).

The automatic shield opener, in closed position

The I/ ports located at the right side of the machine

Wiring guidelines are presented in the diagram and table below.



From Shield Opener Assembly	To Machine I/O Panel
24V A	24 V
24V B	24 V
OPEN	OUT
CLOSE	COM

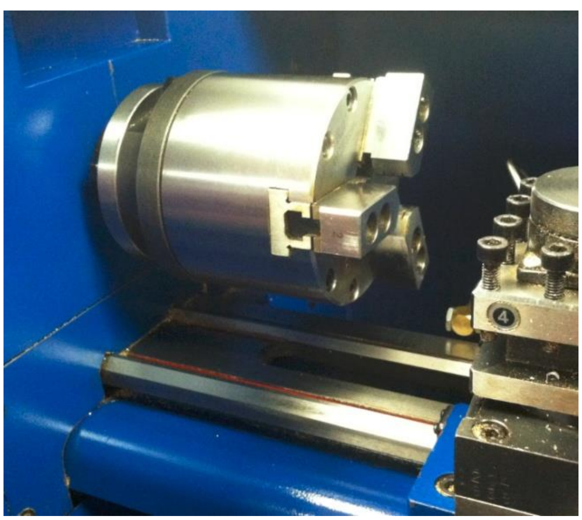
11.1.2. Integrating with an Automated Clamping Device (Pneumatic)

Take Note

Detailed installation instructions are provided with each optional accessory purchased.

The automated clamping device, shown in the photograph below (left), closes and holds the part securely during the machining process. The device opens to allow an automated loading device to load a new workpiece or remove a finished part.

The pneumatic pressure regulator system, shown in the photograph below (right), is mounted on the spindle motor access door on the left side of the machine as shown. This system regulates the air sent to the pneumatic chuck forcing it to clamp onto the work piece.



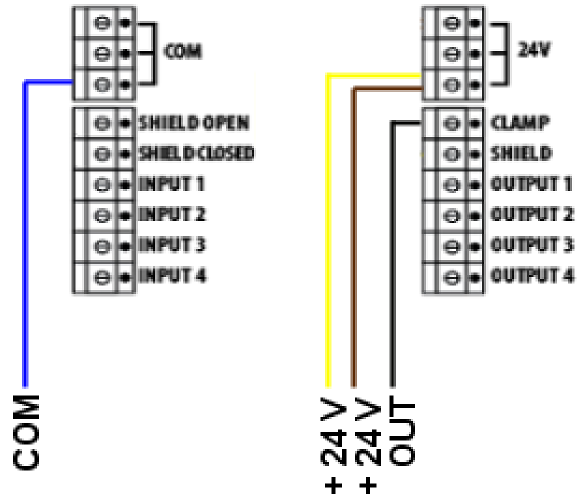
The automatic chuck

The pneumatic pressure regulator

The machine sends open/close commands to the automated clamping device through an output port.

The I/O ports are located at the right side of the machine.

Wiring guidelines are presented in the diagram and table below.



From Air Chuck Kit	To Machine I/O Panel
COM (Blue)	COM
+ 24 V (Yellow/Green)	24 V
+ 24 V (Brown)	24 V
OUT (Black)	CLAMP

11.1.3. Interfacing with a Robot or other FMS Entity

The BenchTurn 7000 CNC machine has a simple interface for integration in an FMS cell. Such integration facilitates, for example, automated part loading and unloading between turning operations.

Most vertical articulated robots on the market can be integrated with Intelitek CNC machines.

Specific G Codes, called CHAIN TO FILE and CHAIN to COM G Codes, allow the BenchTurn 7000 to receive string information. An external control device, such as a robot or device driver, can run any CNC program via this interface. The BenchTurn 7000 in turn notifies the external control device that the requested CNC program has been completed, via a machine output. Unlike for the machine automation devices, such as an automated shield or clamping device, the “handshake” output from the CNC machine to an external control device is wired via a relay to separate the electrical circuits of the CNC machine and the external controller.

intelitek Dynamic Solutions, Inspired Classrooms

Follow the procedure below to set up the wiring for output activation.

Procedure: Wiring for Output Activation
1. Wire the 24V outlet on the CNC machine's I/O interface to the A1 leg of the relay, including a flyback diode for protection against spikes. **2.** Wire the A2 leg to the desired output on the CNC machine. **3.** When the selected output changes to ON, it will connect the 24V output to COM, thus energizing the relay.

11.2. CNC PROGRAMMING FOR ROBOTIC COMMUNICATION

This section provides information on writing CNC programs for use in an FMS.

Section Contents: CNC Programming for Robotic Communication

11.2.1. NC Codes for Robotic Communication

The following table lists the NC codes used for robotic communication.

Info Table: NC Codes for Robotic Communication

Code	Function	
G25	Wait for High signal	See 11.2.2 G Code Programming for Input Signals, pg. 179.
G26	Wait for Low signal	
M25	Transmit High signal	
M26	Transmit Low signal	
H#	Specifies the input or output number. The default is H1. The H code is used in conjunction with the Wait and Transmit codes.	
Example - Input	G25H3 This code tells the CNC machine to wait until the state at input #3 goes High. Assuming the robot's initial output state is Low, if this line of code is placed at the beginning of the program, the CNC machine waits until input #3 goes High, then executes	

	the next line of code.
Example - Output	M25H1 This code tells the CNC machine to output a High signal through output #1.

11.2.2. G Code Programming for Input Signals

Both the G25 and G26 codes pause program operation until a signal change in the specified direction is registered at the specified input. This is summarized in the following table.

Info Table: G Code Programming for Input Signals

Code	**Function**	**If the signal is initially**	**Program will continue after signal**
G25	Wait for High signal	Low	changes to High
		High	changes to Low and then back to High
G26	Wait for Low signal	Low	changes to High and then back to Low
		High	changes to Low

Product Care

If the CNC machine does not respond to the robot as you have programmed it to, check that you have wired the robot to the interface correctly and that the robot's initial output state was not changed to High while connecting the robot.

Take Note

In most cases the G commands can be omitted by using a VB script to send commands from the robot to the CNC machine (for example, to open the door, open the clamping device, etc.)

M commands used to send signals from the CNC machine to the robot cannot be replaced by VB script.

11.3. SAMPLE ROBOT - CNC COMMUNICATION SEQUENCE

This section presents a sample communication sequence between a robot and a BenchTurn 7000 CNC machine, and includes sample programs. Note that you may need to customize the samples for your specific CNC machine.

This sample is based on a configuration that uses one CNC output Chain to File option. One could alternatively create a working interface based on one CNC output and one CNC input, using the G and M codes presented previously and corresponding sequences in the robot program.

This sample shows a typical sequence with a robot run by the SCORBASE programming language from Intelitek, with the robot defined as the master and the CNC as the slave. The machine is waiting in standby by running a program that monitors the communication channel (either a RS232 port or a file).

The robot uses program code and script to send commands to the machine and monitors the machine's status via an input.

Step 1
The CNC program (START.NC) sets the CNC output so that the robot's input will be ON when START.NC is running.

START.NC (Sample)

```
;-------------------------------------------
;   First program to run
;-------------------------------------------
M25 H11 ;USER OUT#1 ON
M20;CHAIN TO PROGRAM
CHAIN_FILE O:\project_name\WS3\TURN\CHAIN_FILE.TXT
```

Step 2
The robot program uses a script file similar to the script below to send commands to the CNC machine.

CHAINL.VBS (Sample)

```
' File: CHAIN.VBS      Date: 03-10-2014
Set objArgs = WScript.Arguments
NameofFile = objArgs(0)
'WScript.Echo NameofFile
writeFile NameOfFile
Sub WriteFile(NcProgram)
   Const FileDirectory = "O:\JERANTUT\WS3\TURN\"
   Const ForReading = 1, ForWriting = 2, ForAppending = 3
   Const TristateUseDefault = -2, TristateTrue = -1, TristateFalse = 0
   Dim fs, f, ts, s,TempfileName
   Set fs = CreateObject("Scripting.FileSystemObject")
   TempFileName = FileDirectory + "chain_file.$$$"
   FileName = FileDirectory + "chain_file.txt"
   fs.CreateTextFile TempFileName          'Create a file
   Set f = fs.GetFile(TempFileName)
   Set ts = f.OpenAsTextStream(ForWriting, TristateUseDefault)
   ts.Write FileDirectory             'write into the file
   ts.Write NcProgram                      'write into the file
   ts.Close  ' close the file
   fs.Copyfile TempFileName,FileName
   fs.deletefile TempFileName
End Sub

Sub CLEAR()
```

```
   WriteFile("CLEAR.NC")
End Sub

Sub OCHUCK()
   WriteFile("OCHUCK.NC")
End Sub

Sub CCHUCK()
   WriteFile("CCHUCK.NC")
End Sub

Sub ODOOR()
   WriteFile("ODOOR.NC")
End Sub

Sub CDOOR()
   WriteFile("CDOOR.NC")
End Sub
Sub SendFile(CNCProgNumber)
   WriteFile(CNCProgNumber & ".nc")
End Sub
```

Step 3
The robot checks the input signal received from the CNC machine. If the machine signals that it is idle, the robot sends a command to the CNC machine to clear the loading area, and will start the loading procedure. By this time the robot will already have picked one part from a local storage device.

```
If Input 1 Off Call Subroutine BT7000 NOT READY
Call Subroutine PLACE TOOL TURRET IN LOADING POSITION
Remark: ****************************************************************************************************
Set Subroutine PLACE TOOL TURRET IN LOADING POSITION
Call Subroutine SCRIPT.CLEAR
Call Subroutine SYNCHRONIZE_WITH_BT7000
Return from Subroutine
Remark: ****************************************************************************************************
```

Step 4
The program name is transferred to the CNC control via the script file and the machine executes the task.

CLEAR.NC (Sample)

M26 H11 ;USER OUT#1 OFF

G0X55Z185

G04F1;MAKE SURE OUTPUT IS SEEN

M20;CHAIN TO PROGRAM

START.NC

Step 5
The robot monitors the CNC's input signal. The robot opens the door if the machine signals that it is idle.

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine OPEN DOOR

Remark: **

Set Subroutine OPEN DOOR

Call Subroutine SCRIPT.ODOOR

Call Subroutine SYNCHRONIZE_WITH_BT7000

Return from Subroutine

Remark: **

Step 6
The program name is transferred to the CNC control via the script file and the machine executes the task.

ODOOR.NC (Sample)

M26 H11 ;USER OUT#1 OFF

M25 H102 ;OPEN DOOR

G04F2;MAKE SURE OUTPUT IS SEEN

G25 H132; Wait door open

M20;CHAIN TO PROGRAM

START.NC

Step 7
The robot monitors the CNC's input signal. The robot opens the clamping device if the machine signals that it is idle.

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine OPEN CHUCK

Remark: **

Set Subroutine OPEN CHUCK

Call Subroutine SCRIPT. CHUCK

Call Subroutine SYNCHRONIZE_WITH_BT7000

Return from Subroutine

Remark: **

Step 8
The program name is transferred to the CNC control via the script file and the machine executes the task.

OCHUCK.NC (Sample)

M26 H11 ;USER OUT#1 OFF

M26 H4;CLOSE CHUCK

G04F1;MAKE SURE OUTPUT IS SEEN

M20;CHAIN TO PROGRAM

START.NC

Step 9
The robot monitors the CNC's input signal. If the machine signals that it is idle, the robot inserts the part into the clamping device and then closes it.

Go to Position SCRIPT.P3 Speed 50 (%)

Go to Position SCRIPT.P2 Speed 50 (%)

Go Linear to Position SCRIPT.P1 Speed 30 (%)

Go to Position SCRIPT.PB1 Speed 5 (%)

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine CLOSE CHUCK

Remark: ***

Set Subroutine CLOSE CHUCK

Call Subroutine SCRIPT. CCHUCK

Call Subroutine SYNCHRONIZE_WITH_BT7000

Return from Subroutine

Remark: ***

Step 10
The program name is transferred to the CNC control via the script file and the machine executes the task.

CCHUCK.NC (Sample)

M26 H11 ;USER OUT#1 OFF

M25 H4;CLOSE CHUCK

G04F1;MAKE SURE OUTPUT IS SEEN

M20;CHAIN TO PROGRAM

START.NC

Step 11
The robot monitors the CNC's input signal. If the machine signals that it is idle, the robot exits the machine and closes the door.

Open Gripper

Go Linear to Position SCRIPT.P2 Speed 30 (%)

If Input 1 Off Call Subroutine BT7000 NOT READY

Go to Position SCRIPT.P3 Speed 50 (%)

Go to Position SCRIPT.P4 Speed 50 (%)

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine CLOSE DOOR

Step 12
The program name is transferred to the CNC control via the script file and the machine executes the task.

CDOOR.NC (Sample)

M26 H11 ;USER OUT#1 OFF

M26 H102 ;CLOSE DOOR

G04F2;MAKE SURE OUTPUT IS SEEN

G25 H131; Wait door closed

M20;CHAIN TO PROGRAM

START.NC

Step 13
Depending on the system environment, the manufacturing CNC code can be activated via an external control program (i.e. device driver) or directly from the robot program. In both cases the robot controller will monitor the CNC machine status and will wait until the CNC program is finished. The monitoring is done via an input interrupt to allow the robot to perform other tasks while waiting. The interrupt is defined at the start of the program and is enabled or disabled as required in the program.

Remark: **

Set Subroutine INITC

Load script file: PCPLC3.VBS

Disable Input Interrupt 1

On Input Interrupt 1 On Run Subroutine BT7000_CYCLE_FINISHED

Return from Subroutine

Remark: **

BRASS_STEP.NC (Sample)

M26 H11 ;USER OUT#1 OFF

M3 S1500

N10 g00 x10 z54

;**

Between these two lines the actual manufacturing code is written

;*************************************

G0 X20 ; Retract

M5

G00 X20Z150

M20; CHAIN TO PROGRAM

START.NC

Step 14
If the input interrupt that monitors the CNC machine is activated, the robot will start the unloading sequence as soon as the robot is available. The sequence starts with the robot checking the input signal coming from the CNC machine. If the machine signals that it is idle the robot will start the unloading procedure.

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine PLACE TOOL TURRET IN LOADING POSITION

Remark: ***

Set Subroutine PLACE TOOL TURRET IN LOADING POSITION

Call Subroutine SCRIPT.CLEAR

Call Subroutine SYNCHRONIZE_WITH_BT7000

Return from Subroutine

Remark: ***

Step 15
The program name is transferred to the CNC control via the script file and the machine executes the task.

CLEAR.NC (Sample)

M26 H11 ;USER OUT#1 OFF

G0X55Z185

G04F1;MAKE SURE OUTPUT IS SEEN

M20;CHAIN TO PROGRAM

START.NC

Step 16
The robot monitors the CNC's input signal. The robot opens the door if the machine signals that it is idle.

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine OPEN DOOR

Remark: ***

Set Subroutine OPEN DOOR

Call Subroutine SCRIPT.ODOOR

Call Subroutine SYNCHRONIZE_WITH_BT7000

Return from Subroutine

Remark: ***

Step 17
The program name is transferred to the CNC control via the script file and the machine executes the task.

ODOOR.NC (Sample)

M26 H11 ;USER OUT#1 OFF

M25 H102 ;OPEN DOOR

G04F2;MAKE SURE OUTPUT IS SEEN

G25 H132; Wait door open

M20;CHAIN TO PROGRAM

START.NC

Step 18
The robot enters the machine, reaches the part and closes its gripper. It monitors the CNC's input signal, and opens the clamping device if the machine signals that it is idle.

Go to Position SCRIPT.P3 Speed 50 (%)

Go to Position SCRIPT.P2 Speed 50 (%)

Open Gripper

Go Linear to Position SCRIPT.P1 Speed 30 (%)

Go to Position SCRIPT.PB1 Speed 5 (%)

Close Gripper

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine OPEN CHUCK

Remark: **

Set Subroutine OPEN CHUCK

Call Subroutine SCRIPT. OCHUCK

Call Subroutine SYNCHRONIZE_WITH_BT7000

Return from Subroutine

Remark: **

Step 19
The program name is transferred to the CNC control via the script file and the machine executes the task.

OCHUCK.NC (Sample)

M26 H11 ;USER OUT#1 OFF

M26 H4;CLOSE CHUCK

G04F1;MAKE SURE OUTPUT IS SEEN

M20;CHAIN TO PROGRAM

START.NC

intelitek
Dynamic Solutions, Inspired Classrooms

Step 20
The robot monitors the CNC's input signal. If the machine signals that it is idle the robot extracts the part from the clamping device and closes it.

Go to Position SCRIPT.PB1 Speed 5 (%)

Go to Position SCRIPT.P2 Speed 50 (%)

Go to Position SCRIPT.P3 Speed 50 (%)

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine CLOSE CHUCK

Remark: **

Set Subroutine CLOSE CHUCK

Call Subroutine SCRIPT. CCHUCK

Call Subroutine SYNCHRONIZE_WITH_BT7000

Return from Subroutine

Remark: **

Step 21
The program name is transferred to the CNC control via the script file and the machine executes the task.

CCHUCK.NC (Sample)

M26 H11 ;USER OUT#1 OFF

M25 H4;CLOSE CHUCK

G04F1;MAKE SURE OUTPUT IS SEEN

M20;CHAIN TO PROGRAM

START.NC

Step 22
The robot monitors the CNC's input signal. If the machine signals that it is idle, the robot exits the machine and then closes the door.

Open Gripper

Go Linear to Position SCRIPT.P2 Speed 30 (%)

If Input 1 Off Call Subroutine BT7000 NOT READY

Go to Position SCRIPT.P3 Speed 50 (%)

Go to Position SCRIPT.P4 Speed 50 (%)

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine CLOSE DOOR

Step 23
The program name is transferred to the CNC control via the script file and the machine executes the task.

CDOOR.NC (Sample)

M26 H11 ;USER OUT#1 OFF

M26 H102 ;CLOSE DOOR

G04F2;MAKE SURE OUTPUT IS SEEN

G25 H131; Wait door closed

M20;CHAIN TO PROGRAM

START.NC

Step 24
The robot transfers the part to the next process or to its target storage.

11.4. SAMPLE ROBOTIC - CNC LNTEGRATION PROGRAMS

This section presents a selection of sample programs used for integrating a CNC machine in an FMS.

Section Contents: Sample Robotic – CNC Integration Programs

11.4.1. Sample NC Programs

START.NC

;---

; First program to run

;---

M25 H11 ;USER OUT#1 ON

M20;CHAIN TO PROGRAM

CHAIN_FILE O:\project_name\WS3\TURN\CHAIN_FILE.TXT

CLEAR.NC

M26 H11 ;USER OUT#1 OFF

G0X55Z185

G04F1;MAKE SURE OUTPUT IS SEEN

M20;CHAIN TO PROGRAM

START.NC

ODOOR.NC

M26 H11 ;USER OUT#1 OFF

M25 H102 ;OPEN DOOR

G04F2;MAKE SURE OUTPUT IS SEEN

G25 H132; Wait door open

M20;CHAIN TO PROGRAM

START.NC

CDOOR.NC

M26 H11 ;USER OUT#1 OFF

M26 H102 ;CLOSE DOOR

G04F2;MAKE SURE OUTPUT IS SEEN

G25 H131; Wait door closed

M20;CHAIN TO PROGRAM

START.NC

OCHUCK.NC

M26 H11 ;USER OUT#1 OFF

M26 H4;CLOSE CHUCK

G04F1;MAKE SURE OUTPUT IS SEEN

M20;CHAIN TO PROGRAM

START.NC

CCHUCK.NC

M26 H11 ;USER OUT#1 OFF

M25 H4;CLOSE CHUCK

G04F1;MAKE SURE OUTPUT IS SEEN

M20;CHAIN TO PROGRAM

START.NC

BRASS_STEP.NC

M26 H11 ;USER OUT#1 OFF

M3 S1500

;G0 X30 Z70

;*************************************

N10 g00 x10 z54

N20 ;******************first step*************************

N30 g01 x9.0 z53.8 f300 ;18.0mm diameter

N40 m03 s2000

```
N50 g01 z13.5 f300
N60 g01 x10
N70 g00 z54
N80 g01 x8.5 z53.8 f300 ;17.0mm diameter
N90 g01 z13.5 f300 s2050
N100 g01 x10
N110 g00 z54
N120 g01 x8.0 z53.8 f300 ;16.0mm diameter
N130 g01 z13.5 f300 s2100
N140 g01 x10
N150 g00 z54
N160 g01 x7.6 z53.8 f300 ;15.2mm diameter
N170 g01 z13.5 f300 s2150
N180 g01 x10
N190 g00 z54
N200 g01 x7.5 z53.8 f300 ;15.0mm diameter
N210 g01 z13.5 f50 s2500
N220 g01 x10 f300
N230 g00 z54
N240 ;*******************second step**************************
N250 g01 x7.0 z53.8 f300 ;14.0mm diameter
N260 g01 z33.5 f300 s2200
N270 g01 x10
N280 g00 z54
N290 g01 x6.5 z53.8 f300 ;13.0mm diameter
N300 g01 z33.5 f300 s2250
N310 g01 x10
N320 g00 z54
N330 g01 x6.0 z53.8 f300 ;12.0mm diameter
N340 g01 z33.5 f300 s2300
N350 g01 x10
N360 g00 z54
N370 g01 x5.5 z53.8 f300 ;11.0mm diameter
N380 g01 z33.5 f300 s2350
N390 g01 x10
N400 g00 z54
N410 g01 x5.1 z53.8 f300 ;10.2mm diameter
N420 g01 z33.5 f300 s2280
N430 g01 x10
N440 g00 z54
N450 g01 x5.0 z53.8 f200 ;10.0mm diameter
```

```
N460 g01 z33.5 f50 s2500
N470 g01 x10 f300
N480 g00 z54
N490 ;*******************arc 1***************************
N500 g01 x3 z53.5 f300 ;start point for arc
N510 g02 x8 z48.5 i3 k48.5 f50
N520 g01 x10 f300
N530 g00 z54
N540 g01 x2 z53.5 f300 ;start point for arc
N550 g02 x7 z48.5 i2 k48.5 f50
N560 g01 x10 f300
N570 g00 z54
N580 g01 x1 z53.5 f300 ;start point for arc
N590 g02 x6 z48.5 i1 k48.5 f50
N600 g01 x10 f300
N610 g00 z54
N620 g01 x0 z53.5 f300;start point for arc
N630 g02 x5 z48.5 i0 k48.5 f50
N640 g01 x10 f300
N650 ;*******************arc 2***************************
N660 g01 x6.25 z33.5 f300 ;start point for arc
N670 g02 x7.5 z32.25 i6.25 k32.25 f50
N680 g01 x10 f300
N690 ;*******************arc 3***************************
N700 g01 x8.25 z13.5 f300 ;start point for arc
N710 g02 x9.5 z12.25 i8.25 k12.25 f50
N720 g01 x10 f300
N730 g00 z54
N740 M05
;**************************************
;G0 X20 ; Retract
;x20z150
M20;CHAIN TO PROGRAM
START.NC
```

11.4.2. Sample Device Driver Script File

Shown below is a sample device driver script file for use in OpenCIM or FMS.

REQUEST	ACTION	RETURN
OPERATE0	DRAW(---BT7000 OPERATE ---)	
	sendmsg(2581)	

REQUEST	ACTION	RETURN
	DRAW(-- OPERATING ---)	
	MSWINDOWS(cscript O:\project_name\WS3\TURN\CHAIN.VBS P1)	
	SENDSTR(V1,RUN WAIT_CYCLE_END_BT7000)	
	WAITSTR(V3,18000000)	
	SENDMSG(2582)	
	SENDMSG(2580)	
	DRAW(---END---)	
END		
ABORT	ABORT()	
END		
INITC		
END		
OPEN DOOR	DRAW(---OPEN BT7000 DOOR ---)	
	MSWINDOWS(cscript O:\ project_name \WS3\TURN\CHAIN.VBS ODOOR)	
END		
CLOSE DOOR	DRAW(---CLOSE BT7000 DOOR ---)	
	MSWINDOWS(cscript O:\ project_name \WS3\TURN\CHAIN.VBS CDOOR)	
END		
OPEN CHUCK	DRAW(---OPEN BT7000 CHUCK ---)	
	MSWINDOWS(cscript O:\ project_name \WS3\TURN\CHAIN.VBS OCHUCK)	
END		
CLOSE CHUCK	DRAW(---CLOSE BT7000 CHUCK ---)	
	MSWINDOWS(cscript O:\ project_name \WS3\TURN\CHAIN.VBS CCHUCK)	
END		
CLEAR	DRAW(---CLEAR BT7000 TURRET---)	
	MSWINDOWS(cscript O:\ project_name \WS3\TURN\CHAIN.VBS CLEAR)	
END		

11.4.3. Sample SCORBASE Programs

Shown below is a sample SCORBASE program for use in a typical FMS station in an OpenCIM/FMS environment.

Remark: $ Beginning of automatically generated code

Call Subroutine $PICK_AND_PLACE_0,31,1,1,3,101

Set Subroutine $PICK_AND_PLACE_0,31,1,1,3,101

Set Variable TASK_ID = 300014

Set Variable PART_ID = 0

Set Variable SOURCE_DEVICE_ID = 31

Set Variable SOURCE_DEVICE_INDEX = 1

Set Variable TARGET_DEVICE_ID = 1

Set Variable TARGET_DEVICE_INDEX = 3

Set Variable PICK_AND_PLACE_NOTE = 101

Call Subroutine AUTOEXEC

Call Subroutine GET031

Call Subroutine PUT001

Return from Subroutine

Remark: $ End of automatically generated code

Remark: **

Set Subroutine INITC

Load script file: PCPLC3.VBS

Disable Input Interrupt 1

On Input Interrupt 1 On Run Subroutine BT7000_CYCLE_FINISHED

Return from Subroutine

Remark: **

Set Subroutine AUTOEXEC

Set Variable SCRIPT.PART_ID = PART_ID

Set Variable SCRIPT.SOURCE_DEVICE_ID = SOURCE_DEVICE_ID

Set Variable SCRIPT.SOURCE_DEVICE_INDEX = SOURCE_DEVICE_INDEX

Set Variable SCRIPT.TARGET_DEVICE_ID = TARGET_DEVICE_ID

Set Variable SCRIPT.TARGET_DEVICE_INDEX = TARGET_DEVICE_INDEX

Set Variable SCRIPT.PICK_AND_PLACE_NOTE = PICK_AND_PLACE_NOTE

Return from Subroutine

Remark: **

Set Subroutine GET001

Print to Screen: GET TEMPLATE FROM CONVEYOR (CNV1)

Call Subroutine SCRIPT.GET_FROM_CNV1

Print to Screen: P1,P2,P3,PB1: 'SCRIPT.P1', 'SCRIPT.P2','SCRIPT.P3','SCRIPT.PB1'

Go to Position SCRIPT.P3 Fast

Go to Position SCRIPT.PB1 Fast

Open Gripper

Go to Position SCRIPT.P2 Fast

Go Linear to Position SCRIPT.P1 Speed 30 (%)

Close Gripper

Go to Position SCRIPT.P2 Fast

Send Message $Start to MANAGER ID=TASK_ID

Return from Subroutine

Remark: **

Set Subroutine PUT001

Print to Screen: PUT TEMPLATE ON CONVEYOR (CNV1)

Call Subroutine SCRIPT.PUT_TO_CNV1

Print to Screen: P1,P2,P3,PB1: 'SCRIPT.P1', 'SCRIPT.P2','SCRIPT.P3','SCRIPT.PB1'

Go to Position SCRIPT.P3 Fast

Go to Position SCRIPT.PB1 Fast

Go to Position SCRIPT.P2 Fast

Go Linear to Position SCRIPT.P1 Speed 50 (%)

Open Gripper

Go Linear to Position SCRIPT.P2 Speed 30 (%)

Go to Position SCRIPT.P3 Fast

Send Message $Finish to MANAGER ID=TASK_ID

Send Message $End to MANAGER ID=TASK_ID

Return from Subroutine

Remark: **

Set Subroutine GET031

Print to Screen: GET FROM BUFFER3

Call Subroutine SCRIPT.GET_FROM_BUFFER3

Print to Screen: P1,P2,P3,PB1: 'SCRIPT.P1', 'SCRIPT.P2','SCRIPT.P3','SCRIPT.PB1'

Go to Position SCRIPT.P3 Fast

Go to Position SCRIPT.PB1 Fast

Open Gripper

Go to Position SCRIPT.P2 Fast

Go Linear to Position SCRIPT.P1 Speed 30 (%)

Close Gripper

Go Linear to Position SCRIPT.P2 Fast

Send Message $Start to MANAGER ID=TASK_ID

Return from Subroutine

Remark: **

Set Subroutine PUT031

Print to Screen: PUT TO BUFFER3

Call Subroutine SCRIPT.PUT_TO_BUFFER3

Print to Screen: P1,P2,P3,PB1: 'SCRIPT.P1', 'SCRIPT.P2','SCRIPT.P3','SCRIPT.PB1'

Go to Position SCRIPT.P3 Fast

Go to Position SCRIPT.PB1 Fast

Go to Position SCRIPT.P2 Fast

Go Linear to Position SCRIPT.P1 Speed 30 (%)

Open Gripper

Go Linear to Position SCRIPT.P2 Fast

Send Message $Finish to MANAGER ID=TASK_ID

Send Message $End to MANAGER ID=TASK_ID

Return from Subroutine

Remark: **

Set Subroutine GET032

Print to Screen: GET FROM LATHE (BT7000)

Call Subroutine SCRIPT.GET_FROM_LATHE1

Print to Screen: P1,P2,P3,P4,PB1,PB2: 'SCRIPT.P1', 'SCRIPT.P2','SCRIPT.P3','SCRIPT.P4','SCRIPT.PB1','SCRIPT.PB2'

Go to Position SCRIPT.P4 Speed 50 (%)

Go to Position SCRIPT.PB1 Fast

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine PLACE TOOL TURRET IN LOADING POSITION

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine OPEN DOOR

Go to Position SCRIPT.P3 Speed 50 (%)

Open Gripper

Go to Position SCRIPT.P2 Speed 50 (%)

Go Linear to Position SCRIPT.P1 Speed 30 (%)

Go to Position SCRIPT.PB1 Speed 30 (%)

Close Gripper

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine OPEN CHUCK

Go to Position SCRIPT.PB2 Speed 30 (%)

Go Linear to Position SCRIPT.P2 Speed 30 (%)

Go to Position SCRIPT.P3 Speed 50 (%)

Go to Position SCRIPT.P4 Speed 50 (%)

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine CLOSE DOOR

Send Message $Start to MANAGER ID=TASK_ID

Return from Subroutine

Remark: **

Set Subroutine PUT032

Print to Screen: PUT_TO_LATHE (BT7000)

Call Subroutine SCRIPT.PUT_TO_LATHE1

Print to Screen: P1,P2,P3,P4,PB1,PB2: 'SCRIPT.P1', 'SCRIPT.P2','SCRIPT.P3','SCRIPT.P4','SCRIPT.PB1','SCRIPT.PB2'

Go to Position SCRIPT.P4 Speed 50 (%)

Go to Position SCRIPT.PB2 Fast

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine PLACE TOOL TURRET IN LOADING POSITION

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine OPEN DOOR

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine OPEN CHUCK

Go to Position SCRIPT.P3 Speed 50 (%)

Go to Position SCRIPT.P2 Speed 50 (%)

Go Linear to Position SCRIPT.P1 Speed 30 (%)

Go to Position SCRIPT.PB1 Speed 5 (%)

Call Subroutine CLOSE CHUCK

Open Gripper

Go Linear to Position SCRIPT.P2 Speed 30 (%)

If Input 1 Off Call Subroutine BT7000 NOT READY

Go to Position SCRIPT.P3 Speed 50 (%)

Go to Position SCRIPT.P4 Speed 50 (%)

If Input 1 Off Call Subroutine BT7000 NOT READY

Call Subroutine CLOSE DOOR

Send Message $Finish to MANAGER ID=TASK_ID

Send Message $End to MANAGER ID=TASK_ID

Return from Subroutine

Remark: **

Set Subroutine GET033

Print to Screen: GET PART FROM GRAVITY FEEDER (GFDR1)

Call Subroutine SCRIPT.GET_FROM_GFDR1

Print to Screen: P1,P2,P3,PB1: 'SCRIPT.P1', 'SCRIPT.P2','SCRIPT.P3','SCRIPT.PB1'

Go to Position SCRIPT.PB1 Fast

Go to Position SCRIPT.P4 Speed 50 (%)

Wait Until Digital Input 2 is ON

Go to Position SCRIPT.P2 Speed 50 (%)

Open Gripper

Go Linear to Position SCRIPT.P1 Speed 30 (%)

Close Gripper

Go Linear to Position SCRIPT.P3 Speed 50 (%)

Go to Position SCRIPT.P4 Speed 50 (%)

Send Message $Start to MANAGER ID=TASK_ID

Return from Subroutine

Remark: **

Set Subroutine PUT033

Print to Screen: PUT TO GRAVITY FEEDER

Print to Screen: SHOULD NEVER HAPPEN

Return from Subroutine

Remark: **

Set Subroutine GET034

Print to Screen: GET FROM RACK2

Call Subroutine SCRIPT.GET_FROM_RACK2

Print to Screen: P1,P2,P3,PB1: 'SCRIPT.P1', 'SCRIPT.P2','SCRIPT.P3','SCRIPT.PB1'

Go to Position SCRIPT.P3 Speed 50 (%)

Go to Position SCRIPT.PB1 Fast

Go to Position SCRIPT.P2 Speed 50 (%)

Open Gripper

Go Linear to Position SCRIPT.P1 Speed 30 (%)

Close Gripper

Go Linear to Position SCRIPT.P2 Speed 50 (%)

Go to Position SCRIPT.P3 Speed 50 (%)

Send Message $Finish to MANAGER ID=TASK_ID

Send Message $End to MANAGER ID=TASK_ID

Return from Subroutine

Remark: **

Set Subroutine PUT034

Print to Screen: PUT TO RACK2

Call Subroutine SCRIPT.PUT_TO_RACK2

Print to Screen: P1,P2,P3,PB1: 'SCRIPT.P1', 'SCRIPT.P2','SCRIPT.P3','SCRIPT.PB1'

Go to Position SCRIPT.P3 Speed 50 (%)

Go to Position SCRIPT.PB1 Fast

Go to Position SCRIPT.P2 Speed 50 (%)

Go Linear to Position SCRIPT.P1 Speed 30 (%)

Open Gripper

Go Linear to Position SCRIPT.P2 Speed 50 (%)

Go to Position SCRIPT.P3 Speed 50 (%)

Send Message $Finish to MANAGER ID=TASK_ID

Send Message $End to MANAGER ID=TASK_ID

Return from Subroutine

Remark: ***

Set Subroutine SYNCHRONIZE_WITH_BT7000

Print to Screen: Synchronizing with BT7000 for Loading/Unloading

BT7000_IDLE_OFF:

Wait Until Digital Input 1 is OFF

Wait 1 (10ths of seconds)

BT7000_IDLE_ON:

Wait Until Digital Input 1 is ON

Wait 1 (10ths of seconds)

BT7000_SIGNAL_ON:

Return from Subroutine

Remark: ***

Set Subroutine WAIT_CYCLE_END_BT7000

Wait 10 (10ths of seconds)

Print to Screen: WAIT_CYCLE_END_BT7000

Enable Input Interrupt 1

Return from Subroutine

Remark: ***

Set Subroutine BT7000_CYCLE_FINISHED

Print to Screen: BT7000 is ready

Disable Input Interrupt 1

Print to Screen: Send message ENDTURN to Dev. 32

Send Message ENDTURN to Device Driver ID=32

Return from Subroutine

Remark: **

Set Subroutine OPEN DOOR

Call Subroutine SCRIPT.ODOOR

Call Subroutine SYNCHRONIZE_WITH_BT7000

Return from Subroutine

Remark: **

Set Subroutine CLOSE DOOR

Call Subroutine SCRIPT.CDOOR

Call Subroutine SYNCHRONIZE_WITH_BT7000

Return from Subroutine

Remark: **

Set Subroutine OPEN CHUCK

Call Subroutine SCRIPT.OCHUCK

Call Subroutine SYNCHRONIZE_WITH_BT7000

Return from Subroutine

Remark: **

Set Subroutine CLOSE CHUCK

Call Subroutine SCRIPT.CCHUCK

Call Subroutine SYNCHRONIZE_WITH_BT7000

Return from Subroutine

Remark: **

Set Subroutine PLACE TOOL TURRET IN LOADING POSITION

Call Subroutine SCRIPT.CLEAR

Call Subroutine SYNCHRONIZE_WITH_BT7000

Return from Subroutine

Remark: **

Set Subroutine BT7000 NOT READY

Print to Screen & Log: BT7000 NOT READY!!! CHECK AND RESTART PRODUCTION!!!

Print to Screen & Log: OR CONTINUE PRODUCTION FROM CURRENT LOCATION.

Return from Subroutine

Remark: **

Set Subroutine SHUTDOWN

Print to Screen: MOVING TO SHUTDOWN POSITION (Robot&LSB)

Go to Position 499 Speed 50 (%)

Close Gripper

Return from Subroutine

Remark: **

11.4.4. Sample VB Script File

CHAINL.VBS

```
' File: CHAIN.VBS      Date: 03-10-2014
Set objArgs = WScript.Arguments
NameofFile = objArgs(0)
'WScript.Echo NameofFile
writeFile NameOfFile
Sub WriteFile(NcProgram)
   Const FileDirectory = "O:\JERANTUT\WS3\TURN\"
   Const ForReading = 1, ForWriting = 2, ForAppending = 3
   Const TristateUseDefault = -2, TristateTrue = -1, TristateFalse = 0
   Dim fs, f, ts, s,TempfileName
   Set fs = CreateObject("Scripting.FileSystemObject")
   TempFileName = FileDirectory + "chain_file.$$$"
   FileName = FileDirectory + "chain_file.txt"
   fs.CreateTextFile TempFileName          'Create a file
   Set f = fs.GetFile(TempFileName)
   Set ts = f.OpenAsTextStream(ForWriting, TristateUseDefault)
   ts.Write FileDirectory            'write into the file
   ts.Write NcProgram                'write into the file
   ts.Close  ' close the file
   fs.Copyfile TempFileName,FileName
   fs.deletefile TempFileName
End Sub

Sub CLEAR()
   WriteFile("CLEAR.NC")
End Sub

Sub OCHUCK()
   WriteFile("OCHUCK.NC")
End Sub

Sub CCHUCK()
   WriteFile("CCHUCK.NC")
End Sub

Sub ODOOR()
   WriteFile("ODOOR.NC")
End Sub
```

```
Sub CDOOR()
   WriteFile("CDOOR.NC")
End Sub

Sub SendFile(CNCProgNumber)
   WriteFile(CNCProgNumber & ".nc")
End Sub
```