

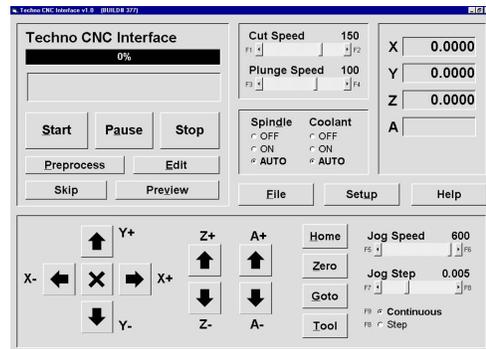
Techno Inc. SINCE 1986
CNC ROUTER SYSTEMS

TECHNO CNC SERVO GCODE INTERFACE

From Build #377



BUILD #377



Techno, Inc.
 2101 Jericho Turnpike
 New Hyde Park, NY 11042-5416
 Phone: (516) 328-3970
 Fax: (516) 358-2576
www.technocnc.com

HTM03250510

Software License Agreement

If you agree to these terms and conditions, Techno, Inc. ("TECHNO") grants you a nonexclusive license to use the accompanying software (the "Software") and documentation. The Software and documentation are referred to in this Agreement as "Licensed Materials". If you do not agree with these terms and conditions, then return the Techno Software CD-ROM to your dealer.

1. License of Ownership

Title to the Licensed Materials is not transferred to the customer, but is the sole and exclusive property of TECHNO and/or its licensed dealers. The customer is entitled solely to use the Licensed Materials according to the terms and conditions of this Agreement.

2. Grant of License

The terms of this Agreement, set forth by Techno and contingent upon the customer's legally bound acceptance, grants the nonexclusive right to use one copy of the Software on a single computer. Physically transferring the Software from one computer to another is allowed only if the Software is used on one computer at a time. Each computer on a network must have its own separately licensed software.

3. Non Transferable

You may not transfer the program to another party without prior written consent of TECHNO. You may not sublicense, lease, rent, share or otherwise transfer your right to use the Licensed Materials, nor any other rights granted to you under this Agreement, except as stated in this Agreement.

4. Copyright Restrictions

This software is owned by TECHNO and is protected by the United States copyright laws. You may make one copy of the Software for backup purposes and/or to transfer the Software to a single hard disk. You agree not to remove any existing copyright notice from many of the Licensed Materials. You also agree not modify or translate the program or the related documentation without prior written consent of TECHNO.

5. Agreement of Restrictions

You agree to communicate, within reason, the restricted conditions of this Agreement to those persons directly employed, or under the customer's direction and control that come into contact with the Licensed Materials. You agree to take reasonable steps to ensure that the terms of this Agreement are followed and that no unauthorized copy, publication or distribution of any of the Licensed Materials is made.

6. Compliance

The customer shall notify TECHNO in writing of any unauthorized use in the event that such unauthorized use does not cease.

7. Limitation of Warranties and Liability

All TECHNO software and documentation are produced by qualified people according to rigorous guidelines. HOWEVER, TECHNO MAKES NO WARRANTY, EXPRESSED OR IMPLIED, THAT THE LICENSED MATERIALS ARE FREE OF ERRORS OR THAT PRODUCTS DESCRIBED HEREIN ARE SUITABLE FOR ANY SPECIFIC APPLICATION. TECHNO ASSUMES NO RESPONSIBILITY FOR LOSS OR DAMAGES RESULTING FROM THE LICENSED MATERIALS OR FROM USE OF THE PRODUCTS HEREIN. TECHNO reserves the right to alter its hardware, software, and any applicable documentation without notice.

TECHNO MAKES NO WARRANTY THAT ITS PRODUCTS ARE FIT FOR ANY USE OR PURPOSE TO WHICH THEY MAY BE PUT BY THE CUSTOMER, WHETHER OR NOT SUCH USE OR PURPOSE HAS BEEN DISCLOSED TO TECHNO IN SPECIFICATIONS OR DRAWINGS PREVIOUSLY OR SUBSEQUENTLY PROVIDED, AND WHETHER OR NOT THE LICENSED MATERIALS ARE SPECIFICALLY DESIGNED AND/OR MANUFACTURED FOR SUCH USE.

THIS WARRANTY IS IN LIEU OF ALL OTHER WARRANTIES, EXPRESSED OR IMPLIED. ALL OTHER WARRANTIES ARE HEREBY DISCLAIMED.

SAFE OPERATION OF YOUR MACHINE

Read these instructions thoroughly BEFORE operating machine

WARNING: IMPROPER OR UNSAFE OPERATION OF THE MACHINE WILL RESULT IN PERSONAL INJURY AND/OR DAMAGE TO THE EQUIPMENT.

1. Keep fingers, hands, and all other objects away from machine while power is on.
2. Disconnect power to all system components when not in use, when changing accessories, and before servicing.
3. Do not loosen, remove, or adjust machine parts or cables while power is on.
4. Exercise care with machine controls and around keyboard to avoid unintentional starting.
5. Make sure voltage supplied is appropriate to specifications of components.
6. Machines must be plugged into three-pronged grounded outlets. Do not remove the grounding plug or connect into an ungrounded extension cord.
7. Keep cables and cords away from heat, oil, and sharp edges. Do not overstretch or run them under other objects or over work surfaces.
8. Use proper fixtures and clamps to secure work. Never use hands to secure work.
9. Do not attempt to exceed limits of machine.
10. Do not attempt to use machine for purposes other than what is intended.
11. Use machine only in clean, well-lit areas free from flammable liquids and excessive moisture.
12. Stay alert at all times when operating the machine.
13. Always wear safety goggles.
14. Do not wear loose-fitting clothing when operating machine. Long hair should be protected.
15. Always maintain proper balance and footing when working around the machine.
16. Maintain equipment with care. Keep cutting tools clean and sharp. Lubricate and change accessories when necessary. Cables and cords should be inspected regularly. Keep controls clean and dry.
17. Before using, check for damaged parts. An authorized service center should perform all repairs. Only identical or authorized replacement parts should be used.
18. Remove any adjusting keys and wrenches before turning machine on.

PREVENT FIRE HAZARDS by using the proper feeds, speeds, and tooling while operating your Techno machine. For example, setting feeds and speeds too low and/or using dull tool bits creates friction at the material. The friction generates heat which can result in a fire that can be drawn through the vacuum table without you knowing it. Be very careful when cutting composite material, especially wood composites like MDF and Particleboard.

DO NOT OPERATE MACHINE IF YOU ARE UNFAMILIAR WITH THESE SAFE OPERATING INSTRUCTIONS. DO NOT OPERATE MACHINE WITHOUT KNOWING WHERE THE EMERGENCY STOP SWITCH IS LOCATED.

TECHNO LIMITED WARRANTY

Terms and Conditions for Limited Warranty and Repairs

Warranty

All Techno mechanical components are warranted against manufacturer's defects in material and workmanship for a period of one (1) year from the time of shipment from Techno facilities. All Techno electrical components are similarly warranted for a period of one (1) year from the time of shipment from Techno facilities. Techno's sole obligation under this warranty is limited to repairing the product or, at its option, replacing the product without additional charge, provided the item is properly returned to Techno for repair as described below. The provisions of this warranty shall not apply to any product that has been subjected to tampering, abuse, improper setup or operating conditions, misuse, lack of proper maintenance, or unauthorized user adjustment. Techno makes no warranty that its products are fit for any use or purpose to which they may be put by the customer, whether or not such use or purpose has been disclosed to Techno in specifications or drawings previously or subsequently provided, and whether or not Techno's products are specifically designed and/or manufactured for such a purpose. NOTE: Drive motors (servo or stepper) are considered "mechanical components".

THIS WARRANTY IS IN LIEU OF ALL OTHER WARRANTIES EXPRESSED OR IMPLIED. ALL OTHER WARRANTIES, INCLUDING, BUT NOT LIMITED TO, ANY WARRANTY OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE, WHETHER EXPRESSED, IMPLIED, OR ARISING BY OPERATION OF LAW, TRADE USAGE, OR COURSE OF DEALING, ARE HEREBY DISCLAIMED. THERE ARE NO WARRANTIES THAT EXTEND BEYOND THE DESCRIPTION ON THE FACE HEREOF.

Limitation of Remedy

In no event shall Techno be liable for any incidental, consequential, or special damages of any kind or nature whatsoever. Techno is in no way liable for any lost profits arising from or connected to this agreement or items sold under this agreement, whether alleged to arise from breach of contract, expressed or implied warranty, or in tort, including, without limitation, negligence, failure to warn, or strict liability.

Return Procedure

Before returning any equipment in or out of warranty, the customer must first obtain a return authorization number and packing instructions from Techno. No claim will be allowed nor credit given for products returned without such authorization. Proper packaging and insurance for transportation is solely the customer's responsibility. After approval from Techno, the product should be returned with a statement of the problem and **transportation prepaid**. If, upon examination, warranted defects exist, the product will be repaired or replaced at no charge, and shipped prepaid back to the customer. Return shipment will be by common carrier (i.e., UPS). If rapid delivery is requested by customer, then such transport is at the customer's expense. If an out-of-warranty situation exists, the customer will be notified of the repair costs immediately. At such time, the customer must issue a purchase order to cover the cost of the repair or authorize the product to be shipped back as is, at the customer's expense. In any case, a restocking charge of 20% will be charged on all items returned to stock.

Field Service

Repairs are ordinarily done at Techno's New Hyde Park, New York facility, where all necessary instrumentation is available. This instrumentation is difficult to transport, so field service is severely limited, and will only be supplied at Techno's discretion. If field service is required and is performed at Techno's sole discretion, all relevant expenses, including transportation, travel time, subsistence costs, and the prevailing cost per hour (eight hour minimum) are the responsibility of the customer.

Unforeseen Circumstances

Techno is not liable for delay or failure to perform any obligations hereunder by reason of circumstances beyond its reasonable control. These circumstances include, but are not limited to, accidents, acts of God, strikes or labor disputes, laws, rules, or regulations of any government or government agency, fires, floods, delays or failures in delivery of carriers or suppliers, shortages of materials, and any other event beyond Techno's control.

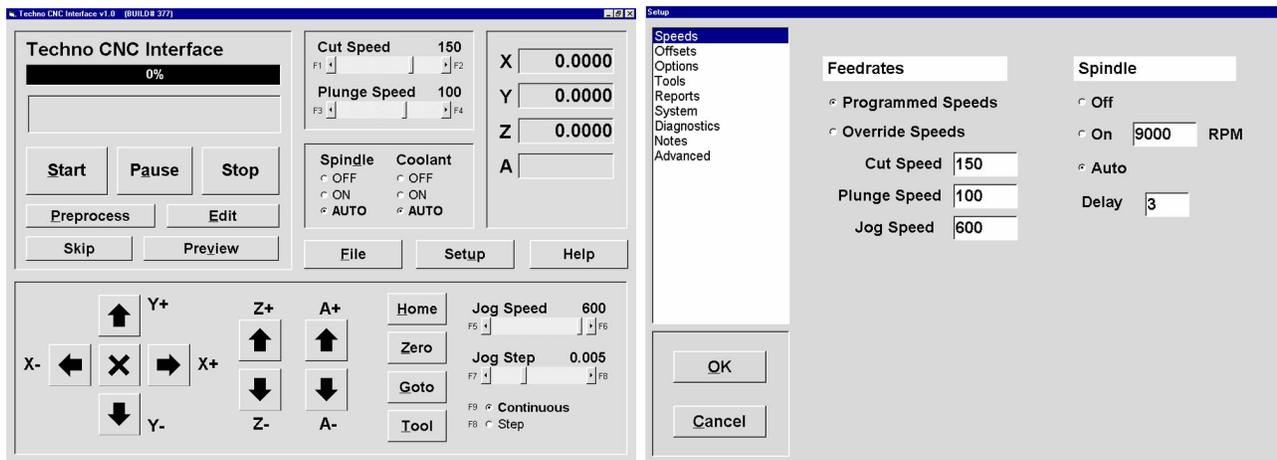
Entire Agreement/Governing Law

The terms and conditions contained herein shall constitute the entire agreement concerning the terms and conditions for the limited warranty described hereunder. No oral or other representations are in effect. This Agreement shall be governed in all respects by the laws of New York State. No legal action may be taken by any party more than one (1) year after the date of purchase.

TECHNO RESERVES THE RIGHT TO CHANGE DESIGNS, SPECIFICATIONS, PRICES, AND ANY APPLICABLE DOCUMENTATION WITHOUT PRIOR NOTICE.

Table of Contents

	<u>Page</u>
Software License Agreement i
Safety Instructions ii
Limited Warranty iii
I. The Techno CNC Interface 5
System Requirements 5
Install PCI Servo Card & Software 6
Techno CNC Interface Main Menu 6
Jogging Attributes 7
Execution Attributes 8
II. Basic Machine Operations Tutorial 9
III. Tool Stand Locations Tutorial 14
IV. Tool Length Offsets Tutorial 20
V. Automatic Toolchange File Tutorial 26
VI. Manual Tool Quick-Change Tutorial 31
VII. Manual Tool Change Tutorial 37
VIII. Explanation of Functions & Terms 45
IX. G-CODE Commands 56
X. Appendices 80



I. The Techno CNC Interface

NOTE: For *Standard LC Machines* and *Tabletop Routers* the 5.75" long **PCI Interface Card** is installed. For the *LC Plus* and *RG Series Machines* the 9.25" **PCI Controller Card** and Riser Card are installed. For *Premium Class Machines* the PCI Controller Card is factory installed.

Loading Card(s) & Software

1. Turn off your computer, unplug the power cord and remove the cover.

WARNING: Ground yourself during installation.

2. Remove PCI card from its protective packaging and locate an available PCI slot.

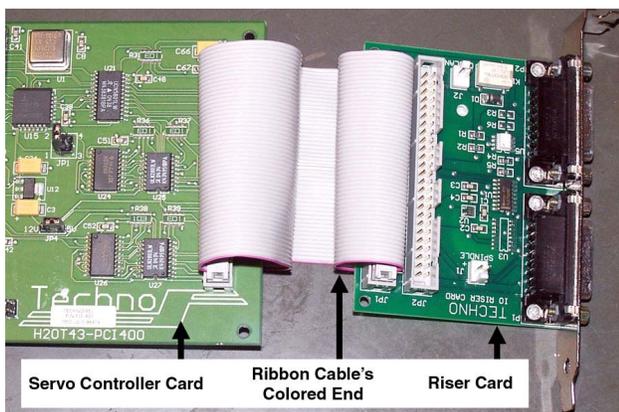
NOTE: The PCI card's connectors mate with the PC Mother Board in only one way. Do not force installation.

3. Gently but firmly insert the PCI card into the vacant PCI slot. Secure with screw.

For Standard LC Machines and Tabletop Routers, Skip to Step 4.

For LC Plus and RG Series Machines continue to Step 3a.

- 3a. Locate another available PCI slot and secure Riser Card (no PCI connections required).
- 3b. Connect the Riser Card to the PCI card using the ribbon cable provided.



System Requirements

To use the Techno CNC Interface, you must have a PC with these minimum requirements:

- **Windows 98, WIN2000, WINME, WINXP**
- **Pentium III or Celeron 600 MHz processor**
- **One available PCI slot**

WARNING: The colored end of the ribbon cable **MUST BE** attached to each card's PIN 1 connector. On PCI card, PIN1 is side closest to Techno logo. On Riser card, it is the side where "TECHNO IO RISER CARD" is printed.

- 3c. Connect Start Stop Box connector to the P2 Riser Card Connector (opposite side of the colored ribbon).

NOTE: If you are operating a four axes machine, then you will have to connect the touchpad to the Riser Card's P1 connector.

4. Close the computer.
5. Connect the cable from the Servo Control Box to the PCI card. Power on.

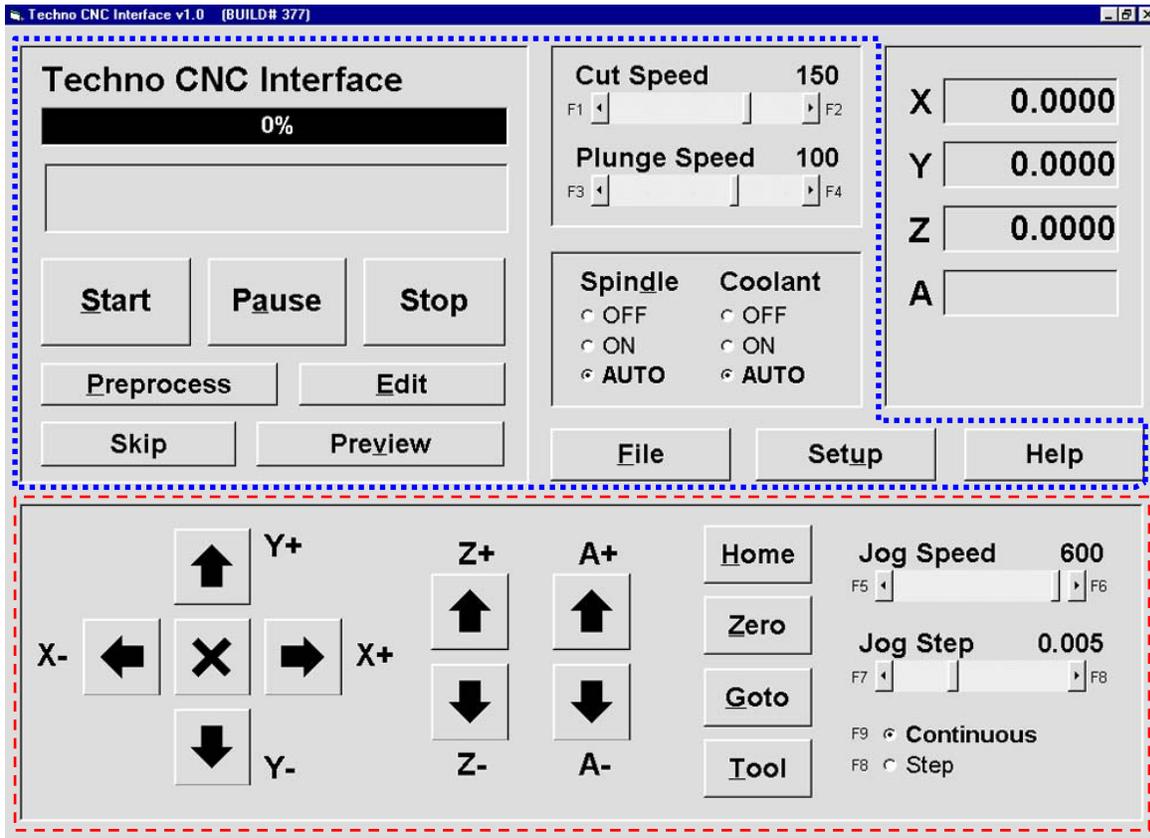
WARNING: Make sure all other applications are closed before attempting to install the Interface, including virus protection software and disable any network card connections.

6. When Windows starts it will detect the PCI card as "new hardware." When prompted to search for a suitable driver insert the Techno CD.
7. When prompted for "optional search locations" choose computer's CD-drive.
8. Select the **Setup Techno CNC Interface** button on the Interface prompt and follow the onscreen instructions.

NOTE: Keep the Techno CD in a safe place. It contains addition documentation (PDF files).

9. Start the Interface by selecting **TECHNO CNC INTERFACE** in your **START/PROGRAMS** menu.

Main Menu



= File Execution Functions



= Jogging Functions

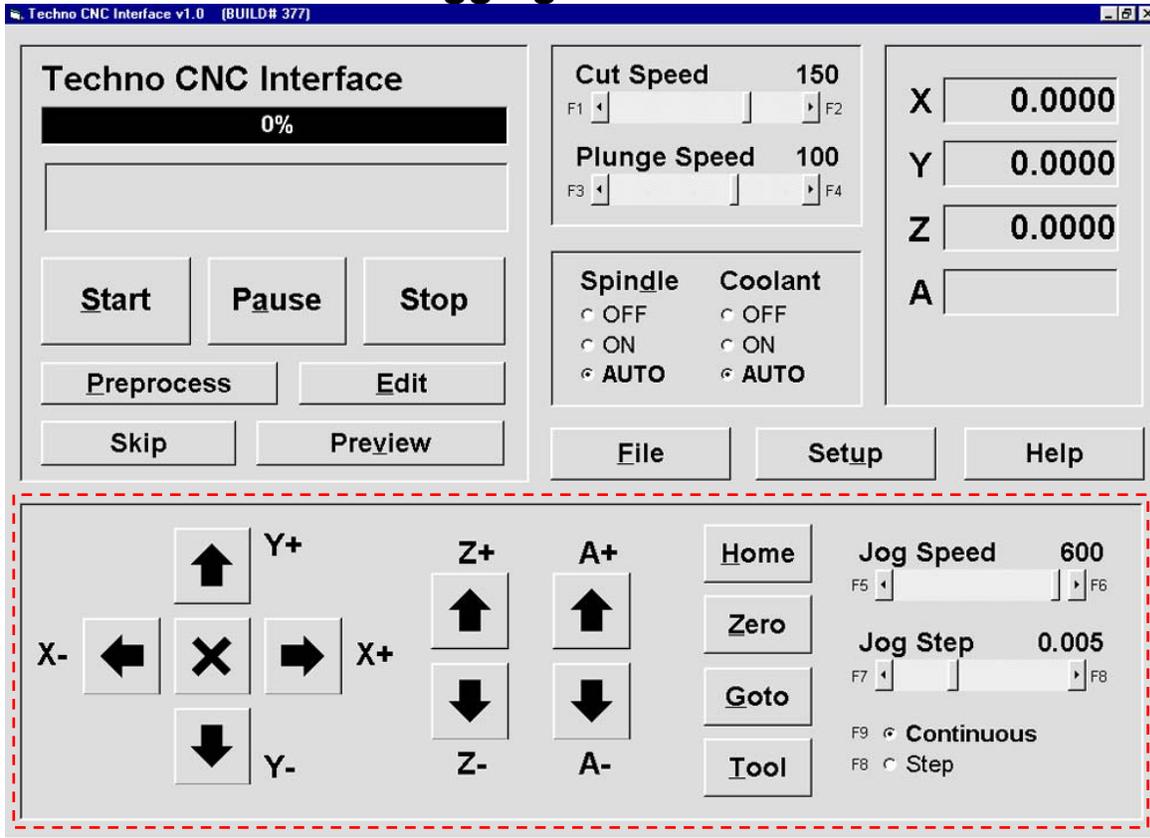
The Techno CNC Interface Main Menu is extremely user-friendly. There are two primary functions through which all the controls are used: **JOGGING** and **FILE EXECUTION**.

Jogging the machine is synonymous with *moving the machine's axes*. All of the Main Menu's jogging functions either actively jog the machine or set up parameters and controls to jog the machine. In the lower portion of the Main Menu (shown above) are the jogging controls. Also, see the Position Counters in the top right quadrant of the Main Menu. These counters give a constant numerical position of where you jog any axis. The following page will provide a brief overview of the jogging functions.

File execution refers to the *retrieval, manipulation and actual running of a GCODE file*. In this Main Menu you can: view the file's toolpath, edit code, set speeds and parameters and start-pause and-resume a program run. The upper portion of the Main Menu allows you do perform all these tasks and more. See page 7 for a brief overview of the various file execution functions.

The Main Menu's upper right quadrant displays the Interface's *Positional Display*. This function is useful in both Jogging and File Execution operations. While jogging, you can track the distance of travel for your axes, and while running a file, the Positional Display shows the actual travel of the axes in motion.

Jogging Functions



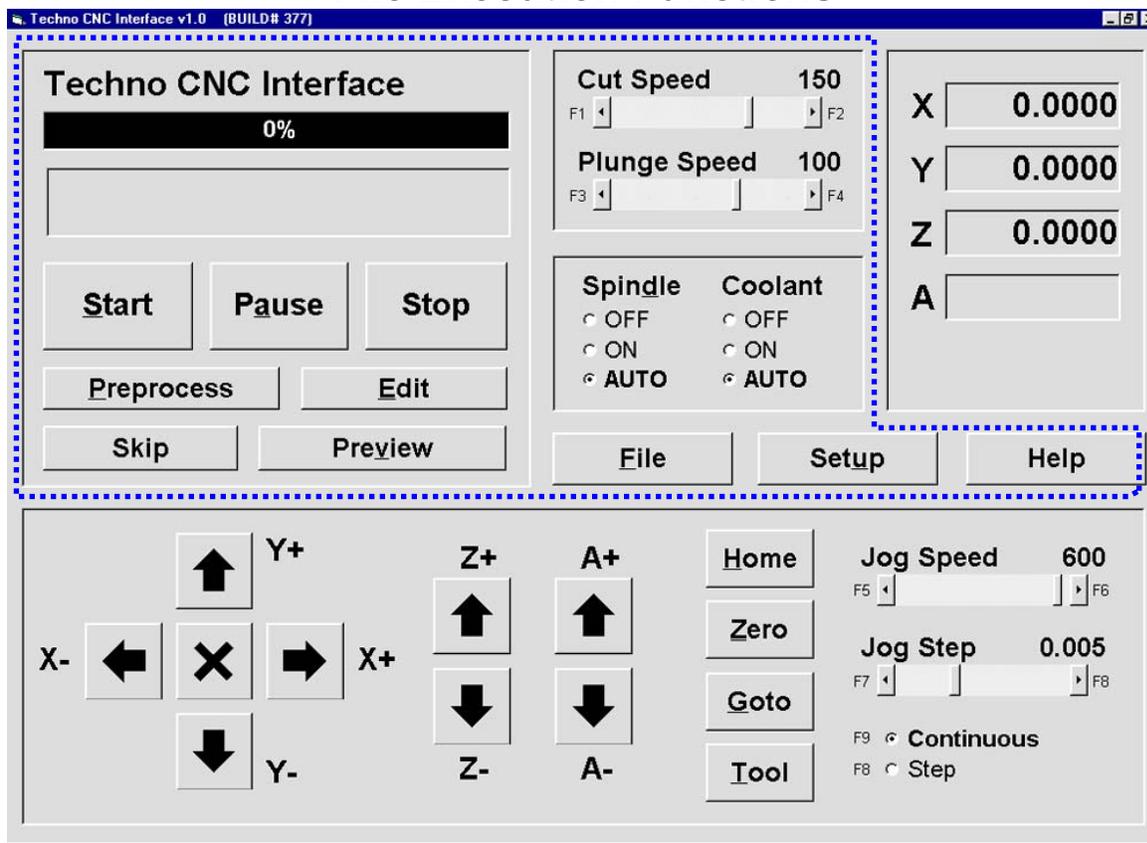
Notice the square buttons with the black arrows. Each of those buttons will move an axis in the direction designated by the letter and value adjacent to it (i.e. Y+). If the **Continuous** radio button is selected, you can hold down a jog button and the axis will move 'continually'. If you have **Step** selected, the axis will only move one step-size at a time. The **Jog Step** scroll bar sets the incremental distance. The **Jog Speed** scroll bar sets the speed at which the axes will travel while jogging.

Notice the Positional Display Counters during jogging. The travel values will go into negative numbers as well as positive numbers. The A-axis represents the availability for a fourth axis.

The column of four buttons (**Home**, **Zero**, **Goto**, and **Tool**) all allow you to perform shortcut operations. Submenus scroll-out on top of the Main Menu when these buttons are selected.

-  The **Home** button moves an axis or all axes to a specified distance away from the axis' limit switch (also known as Machine Home).
-  The **Zero** button allows you to set an axis or all axes' position counters to zero.
-  The **Goto** button allows you to move an axis or all axes to a specified location or move them all to the Machine Origin (by pressing Origin).
-  The **Tool** button brings up a Toolchange Operations window that allows you to manipulate the different aspects of a toolchange application (i.e. touch off Z-axis zero position).

File Execution Functions



In any File Execution, the first step is to retrieve a file. The **File** button opens a window that allows you to search your PC to retrieve and open a file.

Once you have your file, if you want to edit the code, click the **Edit** button. If you want to Preview the file's toolpath, use the **Preview** button. And before you can even use the **Start, Pause or Stop** functions, the file must always be **Preprocess**-ed. Notice how the Progress Bar displays the status of the preprocess function (in a completion percentage) as well as the percentage and current GCODE commands of a file being run.

The **Cut & Plunge Speed** scroll bars are file execution parameters. Use them to adjust the speed at which the axes (X, Y = **Cut**; Z = **Plunge**) travel during a cutting command. The **Spindle & Coolant** radio buttons are another set of controls for file execution. Use these controls according to your program requirements.

The **Setup** button brings up the Setup Window with its bookmarked submenu display screen. From here you can set file execution parameters ranging from Feedrates to Scale Factors.

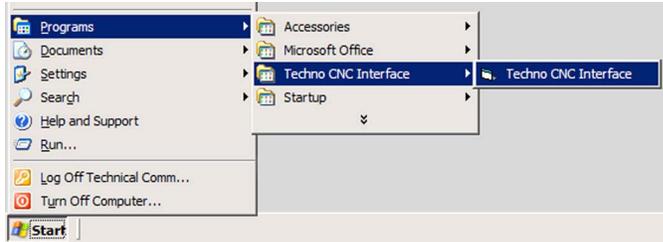
II. Basic Machine Operations Tutorial

This simple tutorial employs and explains the main functions of the Techno CNC GCODE Interface by running the sample program, **2box.nc**. If you have any difficulty with the tutorial or are just learning to use your machine, we recommend reading the INTERFACE FUNCTIONS before proceeding.

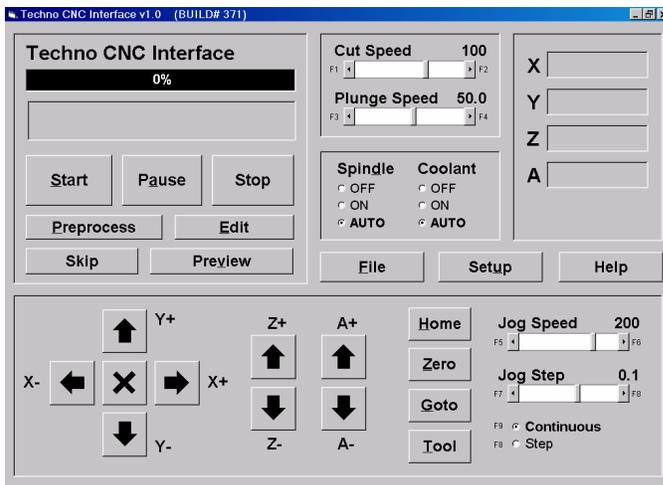
Note: This Tutorial was written assuming you have already setup the touchpad and have a tool bit loaded.

1. Start the Interface

- A. From your computer's Start menu, choose Programs/Techno CNC Interface.



The **Main Menu** appears:



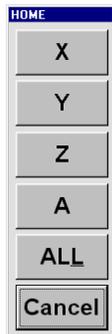
Note: This tutorial is written assuming your **X/Y Machine Orientation** is set to the Interface default setting of -1. If this has somehow been altered, go into **SETUP/SYSTEM** and make sure the X/Y Orientation is set to -1.

2. Home the Machine

- A. Press **Home**

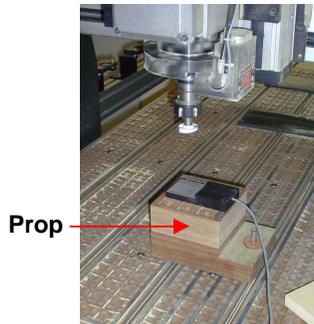
The **Home** Menu appears:

- B. Press **ALL**



3. Zero the Z-axis

Note: Prop the touchpad on top of something high enough so that your Z-axis zero position is well above the table surface (see picture below). Do this to ensure that the tool bit does not travel into the table. The prop height is supposed to represent the height of the material to be cut. After you zero the Z-axis, remember to remove the prop.



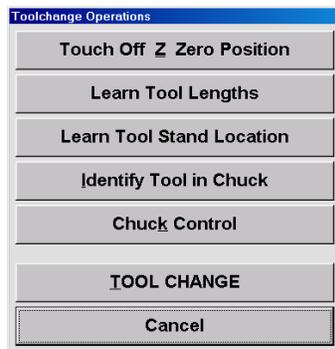
A. Jog your tool over the touchpad

Note: Prior to zeroing the Z-axis, we will test the touchpad and see if it is working properly. Follow the Steps and Warning below.

Warning: If at any time you need to stop the touchpad test, hit the ESC button on your keyboard or the red E-stop button.

B. Press

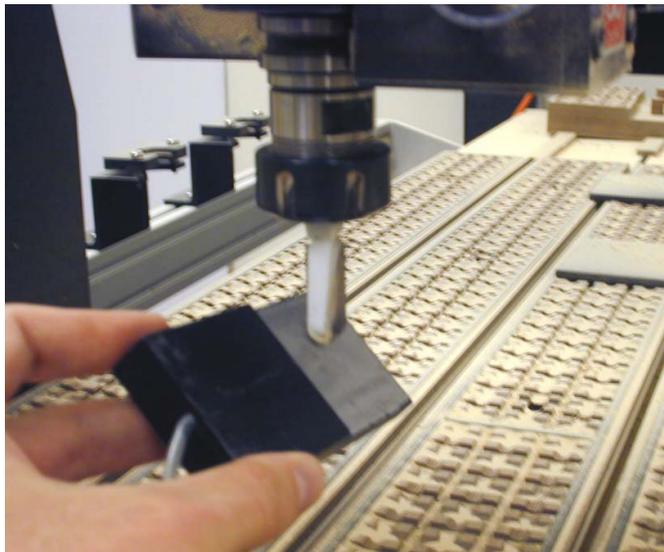
The **Tool** Menu appears:



The Test: Here we will start the *Touch Off Process*. The Z-Axis will automatically travel down to the touchpad. If the touchpad is working properly, once the tool bit (or spindle head) comes into contact with the pad, the Z-axis will retract immediately. This test involves the user manually raising the touchpad up to the spindle during the *Touch Off Process* as the Z-axis is descending (see picture below). When you lift the touchpad up to the tool bit, the Z-axis retracts, then the touchpad is operational. If the Z-axis DOES NOT retract, then immediately press the ESC keyboard button (or E-Stop button) to stop the *Touch Off Process*. Then, check your touchpad setup before returning to this tutorial.

C. Press

The Z-axis will automatically descend.



D. Manually lift the touchpad to the tool bit and then gently touch the pad to the tool.

Note: If the Z-axis retracts upon contact, continue to Step 3-E. If it does not retract, check your touchpad setup before returning to this tutorial.

E. Using the arrow buttons on the Interface's Main Menu jog your tool over the touchpad, which should be placed on top of a prop.

F. Press

G. Press

The Z-axis will automatically lower the Z-axis to the touchpad and retract when it makes contact. This *Touch Off Z Zero Position* process sets your Z-axis zero position automatically.

Note: You can now remove the touchpad and the prop from the table surface.

4. Zero the X and Y Axes

A. Jog the X/Y axes to a central position on your gantry table.

Note: Zero positions are frequently set at the front left corner of your workpiece.

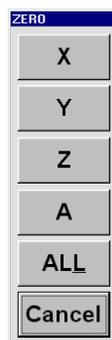
B. Press

The **Zero** Menu appears:

C. Press

This will Zero the X-axis.

D. Repeat process for Y-axis.



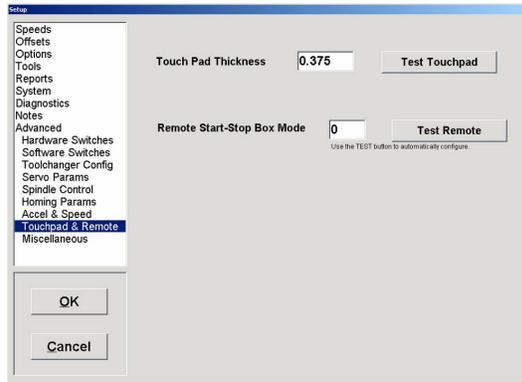
Note: Previewing the GCODE commands in Step 5-D is not required for running a file, but it is a useful feature to know.

F. Press 

7. Check Speeds and Parameters

A. Press 

The **Setup** Window appears:



B. Navigate through the different bookmarked submenus by clicking on the names in the left-side display screen.

C. Press  to exit Setup without making any changes.

8. Preprocess the File

A. Press 

Watch the **Progress** bar move from left to right.

Warning: Before you begin to run a file, remember the different methods you have to stop the program run. For emergencies always use the **E-Stop Button**, located on your controller or start stop box. For non emergencies or to pause and resume a program run, use the **Pause Button** on your Start Stop Box.

Check that your **Z-Axis** zero position provides enough space between the tool and the table so that when you run the file the tool does not plunge into the table. In addition to checking the **Z-Axis**, jog both the **X** and **Y Axes** to the center of the table because the file will not run if it is at the end of any axes.

9. Run the File

A. Press 

III. Tool Stand Locations Tutorial

For Automatic Tool Change Spindles Only!

In this Tutorial you learn the important task of setting your tool stand locations. You will learn the process of setting the first two tool stand locations; after that follow the same directions for any additional stands. You must complete this (and the next) Tutorial before starting the Automatic Tool Change Tutorial (Chapter V).

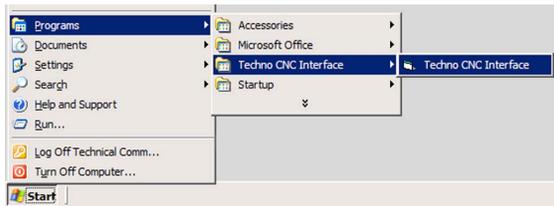
Note: This Tutorial was written assuming that:

1. You have already setup the touchpad.
2. Your tool stands are located at the side of the machine the X-axis will home to (left if you are standing in front of the machine).

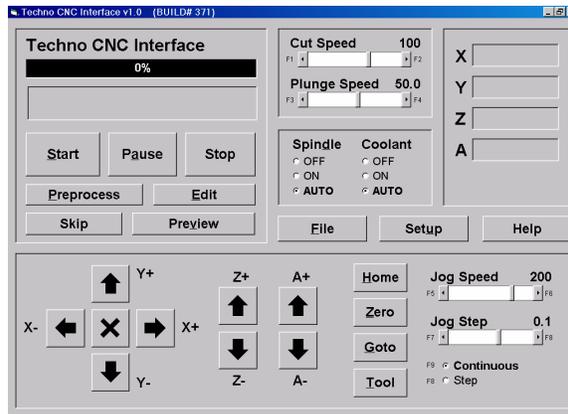
If you have a ROTARY CAROUSEL TOOLCHANGER, the settings are programmed at the factory. Call Techno for any additional information or troubleshooting

1. START THE INTERFACE

- A. From your computer's Start menu, choose Programs/Techno CNC Interface.



The Techno CNC Interface Main Menu appears:



2. HOME THE MACHINE

- A. Press **Home**

The **Home** Menu appears:

- B. Press **ALL**



A Word on Chuck Control

The process of teaching a tool stand location entails opening and closing the spindle's chuck, while selecting a specified time delay to either load the tool holder into the chuck or catch a tool holder as the chuck opens and releases it. Make sure you are not distracted when performing OPEN/CLOSE chuck operations.

3. EMPTY CHUCK AND TOOL STANDS 1 & 2, THEN PLACE THE TOOL HOLDERS OFF TO THE SIDE.

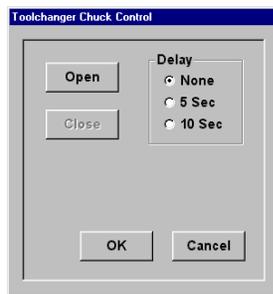
If a tool holder is currently in the chuck, follow these directions to remove it.

To remove a tool holder from the chuck, be ready to move yourself over to the chuck within a 10 seconds. You need to be able to do this in order retrieve the tool holder before it is released. Once the chuck is opened, it releases the tool holder causing it to drop from the spindle. Retrieve the tool holder in this manner: firmly cradle and grip the sides of the tool holder prior to the chuck opening. When the chuck opens it will release the tool holder into your grip.

Warning: Do not place your hand, palm-up, under the tool to catch the tool holder. A sharp tool may cut you when the tool is released.

- A. Press **Tool**, then **Chuck Control**

The **Chuck Control** Window appears:



- B. Select '10 Sec' Delay and press **Open**.
- C. Go directly to the chuck and place your hand firmly around the tool holder, above the tool bit. When the chuck opens, it will release the tool holder into your hands. Place the tool holder off to the side.

Note: These directions are for one person to complete these tasks. If you have a co-worker that can help, instruct him to cradle and catch the tool holder while you operate the chuck control in the Interface.

4. LOAD TOOL HOLDER 1.

Identify the tool holder with the longest bit, this should be designated tool holder 1. Prepare to load tool holder 1 and CLOSE the chuck around it. Make sure to place the top of the tool holder cone all the way up into the spindle chuck. When the chuck closes, it will grasp tool holder 1 and lift it into the chuck a fraction of an inch. Allow for this lift.

- A. Press **Tool**, then **Chuck Control**.
- B. Select '10 Sec' Delay and then **Close**.
- C. Place the top of the tool holder 1 cone all the way up into the spindle chuck (see picture below) and wait the 10 seconds for the chuck to close, lifting tool holder 1 into the chuck.

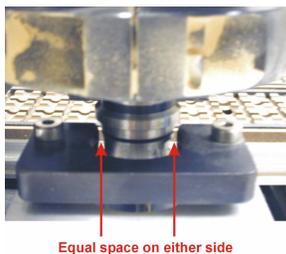


- D. Press **Cancel** to return to Main Menu.

5. JOG TOOL HOLDER 1 TO TOOL STAND 1

The process of setting tool stand location requires highly accurate positioning and therefore, this process should not be rushed.

- A. Jog tool holder 1 just outside the opening of tool stand 1 (see picture below). If you are a 'one-person' operation, you will need to move the keyboard as close to the tool stands as possible. A closer position will better enable you to set the right location. If you have a co-worker available to help you, have him stand by the tool stand and direct you while you jog the machine.

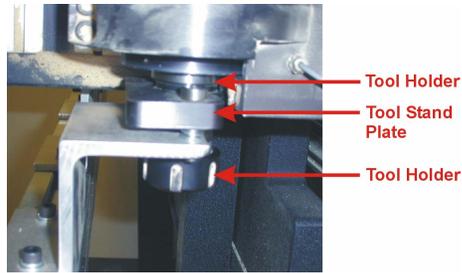


Note: At this point the positioning and margin for error gets pretty tight. We compensate for this by switching from 'Continuous' to 'Step'.

- B. Change the radio button in the lower right hand corner of the Main Menu from **Continuous** to **Step**. Then change the Step Jog value to 0.01. You can change the Jog Step value exactly by double-clicking on the words above the scroll bar. A sub window appears. Enter the value 0.01 and press OK. Now, when you jog, one click will move the axis 0.01 inch per click.

6. POSITION TOOL HOLDER 1 IN TOOL STAND 1

- A. Carefully jog tool holder 1 into the middle of tool stand 1. Make sure you can freely spin tool holder 1 (with your hand).
- B. Jog the Z-axis down (Z-) one click at a time until tool holder 1 comes into contact with the tool stand plate (see picture below). Then Jog the Z-axis up (Z+) 4 clicks.



Note: From this position we begin the finishing touches for setting the tool stand location. We are going to do the following:

- 👉 Find the X-axis tool stand middle position (Step 7)
- 👉 Find the Y-axis tool stand middle position (Step 7)
- 👉 Find the appropriate Z-axis tool stand position height (Step 8)
- 👉 Set the tool stand location (Step 9)

7. FIND THE X AND Y-AXES TOOL STAND MIDDLE POSITIONS

- A. Jog tool holder 1 to the middle of the tool stand 1 along the X-axis. At the desired middle position you should be able to spin the tool holder, using your hand, without obstruction.

To determine a highly accurate X-axis middle position, jog the tool holder in a positive direction (X+) until tool holder 1 cannot spin freely and easily.

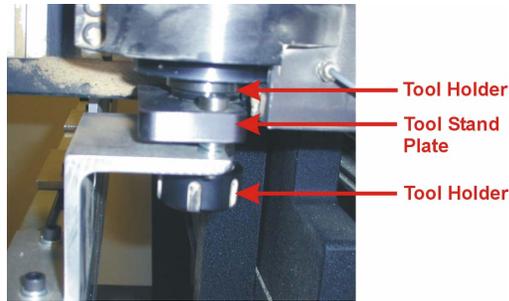
Warning: Do not jog beyond the point where the tool cannot spin at all.

Then, counting the number of jog-clicks, jog the X-axis in a negative direction (X-) until tool holder 1 reaches the other side of the tool holder. Take the number of jog-clicks it took to get from one side (X+) of the tool stand to the other (X-), divide by 2, and jog back that number of clicks to a very exact middle position. **Example:** if the total number from one side to the other was 20 ($20/2=10$), then jogging back 10 jog-clicks (X+) will be an accurate center position for the X-axis.

- B. Repeat process for the Y-axis.

8. SET THE Z-AXIS POSITION

- A. Jog the Z-axis down (Z-) until tool holder 1 initially contacts the tool stand plate (see picture below). Jog down 2 more jog-clicks (Z-).



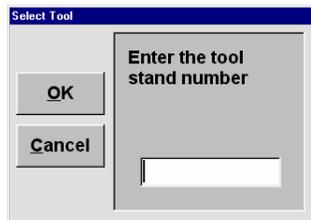
Warning: DO NOT go more than 3 clicks down on the Tool stand Plate (using a 0.01 Jog Step).

9. TEACH THE TOOL STAND LOCATION

We are now ready to set the tool stand location.

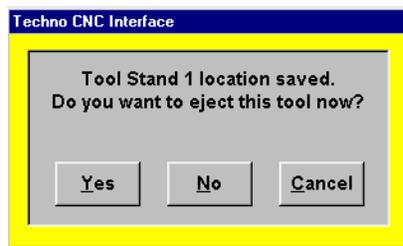
- A. Press , then .

The **Learn Tool Stand Location** pop-up window appears:



- B. Type 1, and hit .

A window appears confirming your saved tool stand location, and asking you whether or not you want to eject the tool now.



- C. Press .

The Machine will now drop off tool holder 1 in tool stand 1.

Note: If there is a problem setting the tool stand location, start over beginning at Step 3. When programming tool changes, whatever tool is in this holder will now be referred to as Tool 1 (or in GCODE terms T1). For instance, if a 1/4" tool is placed in the holder and the program calls for T1 to be 1/8", the machine will not be able to identify this mistake. ***It is the programmer/operators responsibility to make sure the proper tools are placed in the designated holders.*** Now, perform the tests in steps 10 and 11.

10. Test A: Pick-up

Here we want to make sure that the position just programmed is suitable for picking up tool holder 1 from tool stand 1.

- A. Press , then .
- B. From the pop-up window, type 1 and press .

The machine should now travel back to tool stand 1 and pick tool holder 1. If this works, continue on to Test B in Step 11.

11. Test B: Drop-off

Here we want to make sure that the position just programmed is suitable for dropping-off tool holder 1 into tool stand 1.

- A. Press , then .
- B. From the pop-up window, type 0 and press .

The machine should now travel back to tool stand 1 and drop-off tool holder 1. If this works, tool stand 1 has been properly set.

Note: Before starting the setup for tool holder 2, switch back to **Continuous**, while jogging to tool stand 2. Once you get tool holder 2 at the opening of tool stand 2, then switch back to **Step** at 0.01 inch.

10. Repeat this process for tool holder 2, and tool stand 2.

Note: Substitute the #2 for any instance the Tutorial mentions #1 (Tool holder 1 or Tool stand 1). For example, in Step 9-B, type 2 (not 1) before hitting the OK button.

Make sure you set a tool stand location for tool holder 2. Chapter VI, **Automatic Tool Change Tutorial** will require two tools. If you wish to set any additional tool stand Locations (i.e. 3, 4...), that's fine, just know that for now, we require at least 1 & 2.

IV. Tool Lengths Tutorial

For Automatic & Manual Quick-Change Tool Change Spindles Only!

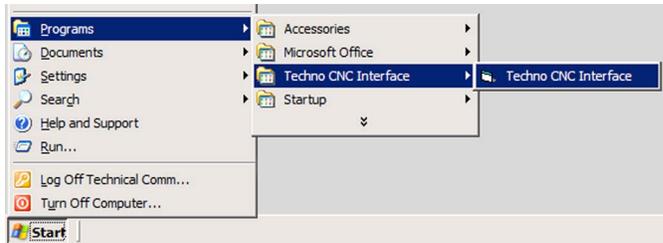
Tool Change operations often involve tool bits of varying lengths. The touchpad is used to record each tool length for one reason: to compensate and thus maintain a consistent Z-axis zero position. In this Tutorial, you learn the important task of calibrating the length of each tool bit for each tool holder. You will learn this process for the first two tool holders; after that follow the same directions for any additional tool holders. You must complete this (and the previous) Tutorial before starting the Automatic Tool Change Tutorial.

Note: This Tutorial was written assuming that:

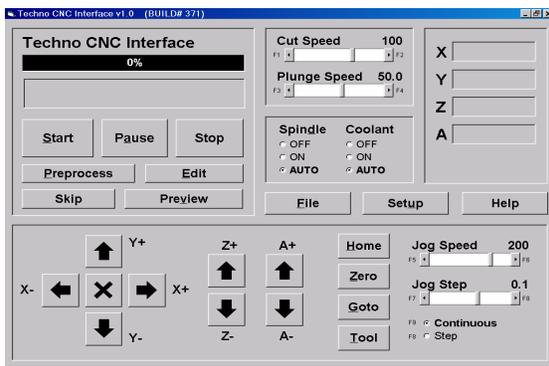
1. You have already setup the touchpad.
2. Your tool stands are located at the side of the machine the X-axis will home to (left if you are standing in front of the machine).
3. You have already completed *III. Tool Stand Locations Tutorial*.
4. You have loaded two tool bits of varying lengths into two tool holders.

1. START THE INTERFACE

A. From your computer's Start menu, choose Programs/Techno CNC Interface.



The Techno CNC Interface Main Menu appears:



2. HOME THE MACHINE

A. Press **Home**

The **Home** Menu appears:

B. Press **ALL**



A Word on Chuck Control for Manual Quick-Change Tool Changes

The process of Learning/Calibrating Tool Lengths entails opening and closing the chuck, while selecting a specified time delay to either load the tool holder into the chuck or retrieve a tool holder as the chuck opens and releases it. Make sure you are not distracted when performing OPEN/CLOSE chuck operations.

A Word on the Touchpad

The touchpad should always be placed in the same place for all the tools being calibrated. The Interface already knows the thickness of the touch pad (0.375). So, when the tool bit comes in contact with the touchpad and retracts the Z-axis, the Interface records the position, less the touchpad thickness.

3. PLACE THE TOUCHPAD ON THE TABLE SURFACE

Note: For Automatic Tool Changers: We begin this section assuming you have already set tool stand locations 1 & 2. If you have not done so, complete the previous tutorial (III. Tool Stand Location).

Automatic or Manual Quick-Change, that is the question...

The previous tutorial, Learn Tool Stand Locations, applied to Automatic Tool Change operations, only. The subject of this tutorial, however, applies to both Automatic and Manual Quick-Change spindles. Thus, when relevant, we will explain the appropriate procedure for both.

4. LOAD TOOL 1

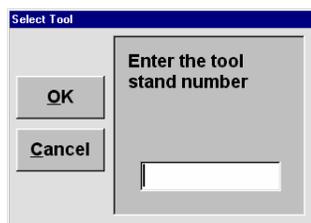
Tool 1 should be the tool holder with the longest tool bit.

For Automatic Tool Changer

If tool holder 1 is in the chuck, skip to Step 5. If tool holder 1 is not in the chuck, continue to Step 4-A.

A. Press , then .

The **Select Tool** pop-up window appears:



B. Type 1 and press .

The machine will now retrieve tool holder 1 from tool stand 1.

For automatic tool changers, skip to Step 5.

For Manual Quick-Change Tool Changer

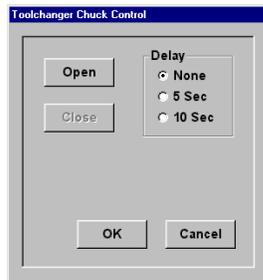
If tool holder 1 is in the chuck, skip to Step 5; if tool holder 1 is not in the chuck, continue to Step 4-C.

Note: If there is not a tool holder present in the chuck, then continue to Step 4-C, BUT skip over Step 4-E. (so, in this case your progress would be as follows: Step 4-C, Step 4-D, Step 4-F, Step 4-G, etc.)

Warning: Manually retrieve a tool holder in this manner: firmly cradle and grip the sides of the tool holder prior to the chuck opening. When the chuck opens it will release the tool holder into your grip. Do not place your hand, palm-up, under the tool to catch the tool holder. A sharp tool may cut you when the tool is released.

C. Press **Tool**, then **Chuck Control**.

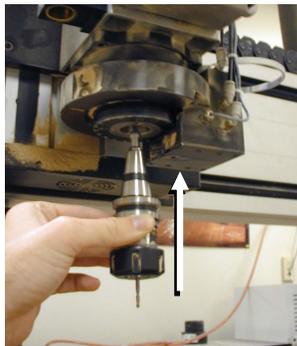
The **Chuck Control** window appears:



D. Select '10 Sec' Delay and press **Open**.

E. Go directly to the chuck and place your hand firmly around the tool holder, above the tool bit. When the chuck opens, it will release the tool holder into your hands. Place the tool holder off to the side.

F. Place tool holder 1 in chuck.



Note: Prepare to load tool holder 1 and close the chuck around it. Place the top of the tool holder cone all the way up into the spindle chuck. When the chuck closes, it will grasp the tool holder and lift it into the chuck a fraction of an inch. Allow for this lift.

G. Select '10 second' delay, and press **Close**.

H. Press **Cancel** to return to Main Menu.

5. JOG TOOL HOLDER 1 INTO POSITION AND TEST TOUCHPAD

Calibrating tool lengths is a pre-programmed process of the Interface. Once you activate this process the machine runs on its own. However, we will perform a test prior to going all the way through this process to make sure the touchpad is activated and working properly.

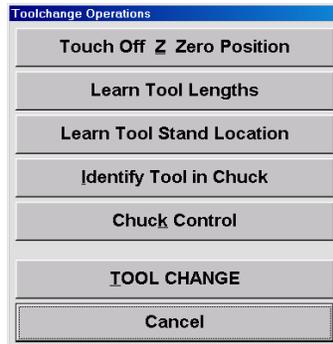
Warning: Remember to use the **E-Stop Button**, located on your controller or start stop box, to immediately stop the machine.

A. Jog tool holder 1 over the touchpad, about 4 to 5" above.

Note: Although in this Tutorial we are learning how to set tool lengths, for the purpose of *testing* the touchpad, it's better to use **TOUCH OFF Z ZERO POSITION**, rather than the **LEARN TOOL LENGTHS** button. After the test is complete, then we will use the **LEARN TOOL LENGTHS** button.

B. Press **Tool**.

The **Tool** Menu appears:



In Step 5-C the Z-axis will start to descend. Normally, you would wait until the Z-axis brings the tool bit to the touchpad and then, if working properly, the Z-axis would retract. For our test, however, we want you to pick up the touchpad and bring it to the tool holder right after the Z-axis begins its descent (see picture below). If the touchpad is working properly, the Z-axis will retract as soon as it makes contact with the tool.



C. Press **Touch Off Z Zero Position**.

Warning: If the Z-axis DOES NOT retract (after Step 5-D), press the **E-Stop Button**, located on your controller or start stop box, to immediately stop the machine.

D. Once the Z-axis begins its descent, pick up the touchpad and gently make contact with the tool holder or bit.

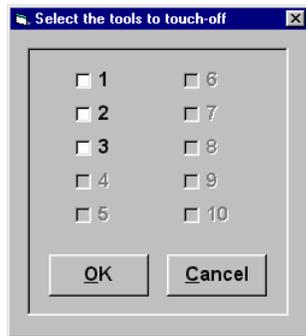
Note: If the Z-axis retracts upon contact, continue to Step 6-A. If it does not retract, press the **E-Stop Button** to stop the Z-axis and check your touchpad setup before returning to this Tutorial.

6. LEARN TOOL LENGTH FOR TOOL 1

A. Re-position tool holder 1 over the touchpad, about 4 to 5" above.

B. Press **Tool**, then **Learn Tool Lengths**.

The **Learn Tool Lengths** window appears:



C. Click inside the box next to #1 (so that a check appears) and press .

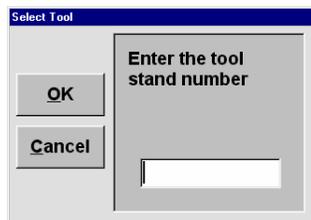
Note: After Step 6-C, the Z-axis will begin its descent toward the touchpad. When the tool bit contacts the touchpad, the Z-axis will travel back up and the Interface will record the tool length for tool holder 1.

7. RETURN TOOL HOLDER 1 AND LOAD TOOL HOLDER 2

For Automatic Tool Changer

A. Press , then .

The **Select Tool** pop-up window appears:



B. Type 2 and press .

Machine will return tool holder 1 to its tool stand, then retrieve tool holder 2 from its tool stand. For an ATC spindle, continue to Step 8.

For Manual Quick-Change Tool Changer

C. Press , then .

D. In the **Chuck Control** window select '10 second' delay and press .

E. Go directly to the chuck and place your hand firmly around the tool holder 1, above the tool bit. When the chuck opens, it will release tool holder 1 into your hands. Place it off to the side.

F. Place tool holder 2 in chuck.

Note: Prepare to load tool 2 and close the chuck around it. Place the top of the tool holder cone all the way up into the spindle chuck. When the chuck closes, it will grasp the tool holder and lift it into the chuck a fraction of an inch. Allow for this lift.

- G. Select '10 second' delay, and press **Close**.
- H. Press **Cancel** to return to Main Menu.

8. REPEAT STEP 6 FOR TOOL HOLDER 2

Substitute the #2 for any instance the Tutorial mentions #1 (or tool holder 1).

9. REMOVE TOOL HOLDER 2 FROM CHUCK

For Automatic Tool Changer

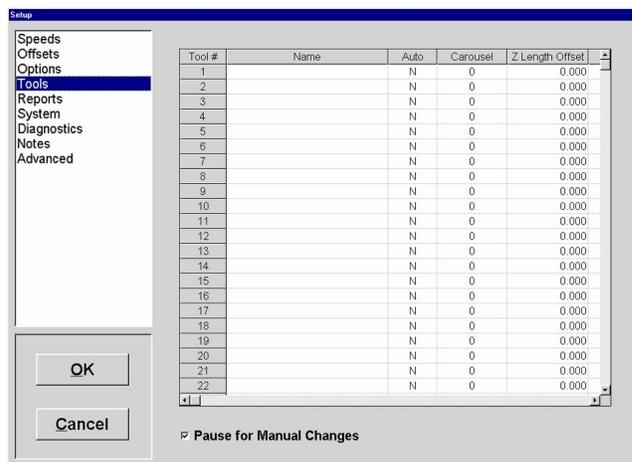
- A. Press **Tool**, then **TOOL CHANGE**.
- B. Type 0, then press **OK**.

Machine will return tool holder 2 to its tool stand, without picking-up any other tool holder.

For Manual Quick-Change Tool Changer

- C. Press **Tool**, then **Chuck Control**.
- D. From the **Chuck Control** window select a '10 Sec' Delay and press **Open**.
- E. Go directly to the chuck and place your hand firmly around the tool holder, above the tool bit. When the chuck opens, it will release the tool holder into your hands. Place the tool holder off to the side.

Now tool 1 and tool 2 should have the same zero position, even if they have radically different tool lengths. The Interface will remember and keep these tool length offsets until you go through the process re-teaching them again. To view the tool lengths in the Interface go to **Setup/Tools**.



V. Automatic Tool Change Tutorial

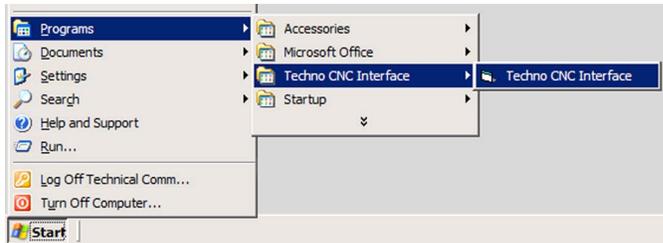
This tutorial explains and explains the main functions of the Techno CNC GCODE Interface as they pertain to an Automatic Tool Change (ATC) operation. Here you will learn the initial steps and functions by running the sample program, **2tools.nc**. If you have any difficulty with the tutorial or are just learning to use your machine, we recommend reading the INTERFACE FUNCTIONS before proceeding.

Note: This Tutorial was written assuming that:

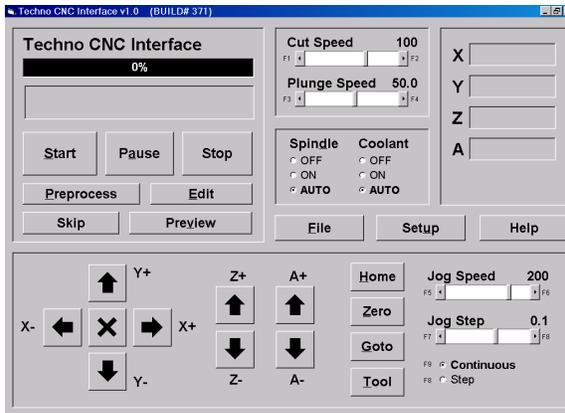
1. You have already setup the touchpad.
2. You have already set the tool stand locations and tool length offsets for at least the first two tools (as instructed in Chapters III and IV).

1. START THE INTERFACE

A. From your computer's Start menu, choose Programs/Techno CNC Interface



The Techno CNC Interface Main Menu appears:

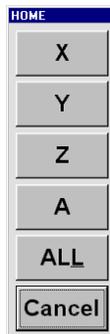


2. HOME THE MACHINE

A. Press **Home**

The **Home** Menu appears:

B. Press **ALL**

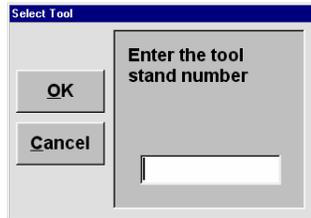


3. LOAD TOOL HOLDER 1.

If tool holder 1 is in the chuck, skip to Step 4. If tool holder 1 is not in the chuck, continue to Step 3-A.

A. Press , then .

The **Select Tool** pop-up window appears:

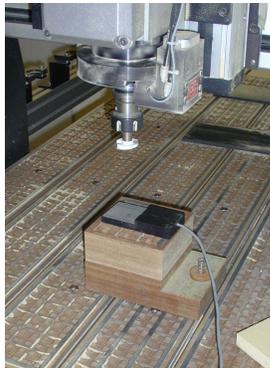


B. Type **1** and press .

The machine will now retrieve tool holder 1 from tool stand 1.

4. ZERO THE Z-AXIS

Note: Prop the touchpad on something high enough so that the Z-axis zero position will be set well above the table surface (see picture below). Do this to ensure that the tool bit does not travel into the table or material during the program run. Make sure remember to remove the prop after you zero the Z-axis.



A. Jog your tool over the touchpad.

Prior to Zeroing the Z-axis, we will test the touchpad and if it is working properly, then we will Zero the Z-axis. Follow the Steps and Warning below.

Warning: Remember to use the **E-Stop Button**, located on your controller or start stop box, if you need to immediately stop the machine.

B. Press .

The Test: Here we will start the *Touch Off Process*, wherein the Z-Axis will automatically travel down to the touchpad. If the touchpad is working properly, once the tool comes into contact with the pad, the Z-axis will retract immediately. This test involves the user manually lifting the touchpad up to the tool, as in Step 3-D (see picture below). When you lift the touchpad up to the tool, if the Z-axis retracts, then the pad is operational. If the Z-axis DOES NOT re-tract, immediately press red E-stop button to stop the Z-axis. Then, check your touchpad setup before returning to this Tutorial.

C. From the **Tool** Menu press .

The machine will automatically lower the Z-axis.

D. Manually lift the touchpad to the tool and then gently touch the pad to the tool (see picture below).



If the Z-axis retracts upon contact, continue to Step 3-E. If it does not retract, check your touchpad setup before returning to this Tutorial.

E. Jog your tool over the touchpad.

F. Press .

G. Press .

The Z-axis will automatically lower the Z-axis to the touchpad and retract when it makes contact.

H. Clear the touchpad and the prop from the table surface.

5. ZERO THE X AND Y AXES

A. Jog the X/Y axes to a central position on your gantry table.

Note: Zero positions are frequently set at the front left corner of your workpiece.

B. Press .

The **Zero** Window appears:

C. Press .

This will Zero the X-axis.

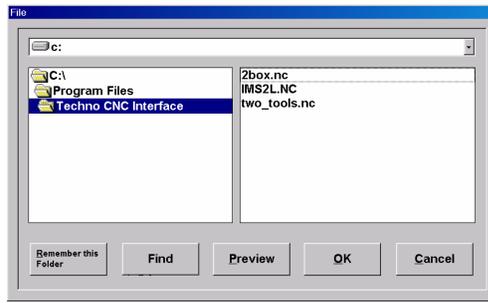
D. Repeat process for Y-axis.



6. OPEN TOOL CHANGE FILE

A. Press .

The **Open File** Window appears:

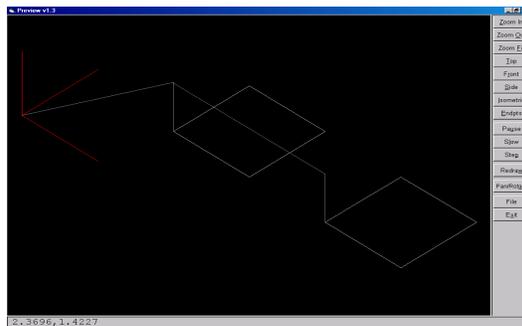


B. Locate your Techno CNC Interface folder and highlight **2tools.nc**, press .

7. PREVIEW THE FILE

A. Press .

The **Preview** Window appears

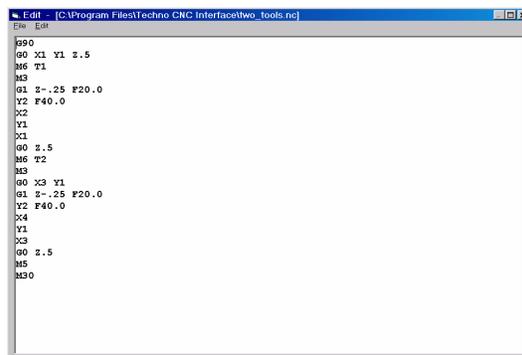


B. Manipulate the toolpath using the buttons along the right side.

C. Press to close window.

D. Press .

The **Edit** Window appears:



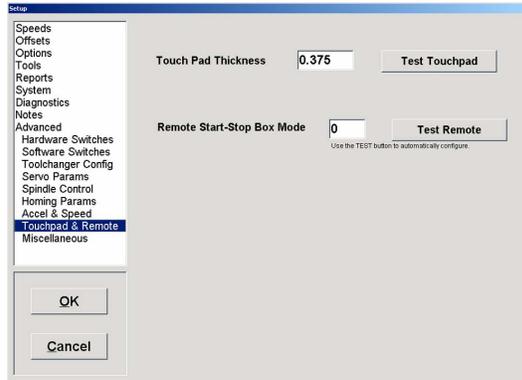
Note: Step 5-D (Edit/Step) is not a mandatory function to run a file; it is, however, a useful feature to know.

D. Press .

8. CHECK SPEEDS AND PARAMETERS

A. Press **Setup**.

The **Setup** Window appears:



B. Navigate through the different bookmarked submenus by clicking on the names in the left-side display screen.

Warning: Do not change any parameters in Setup. We will run **2tools.nc** using the Interface's default settings.

C. Press **Cancel** to exit Setup.

9. PREPROCESS THE FILE

A. Press **Preprocess**.

Watch the Progress bar move from left to right.

Warning: Before you begin to run a file, remember the different methods you have to stop the program run. For emergencies always use the **E-Stop Button**, located on your controller or start stop box. For non emergencies or to pause and resume a program run, use the **Pause Button** on your Start Stop Box.

Check that your **Z-Axis** zero position provides enough space between the tool and the table so that when you run the file the tool does not plunge into the table. In addition to checking the **Z-Axis**, jog both the **X** and **Y Axes** to the center of the table because the file will not run if the machine is at the end of any axes.

10. RUN THE FILE

Warning: Remember to use the **E-Stop Button**, located on your controller or start stop box, if you need to immediately stop the machine.

A. Press **Start**.

Watch how the machine runs **2tools.nc**. Notice, how the machine stops, returns Tool 1 and retrieves Tool 2 to finish the file.

VI. Manual Quick-Change Tool Change Tutorial

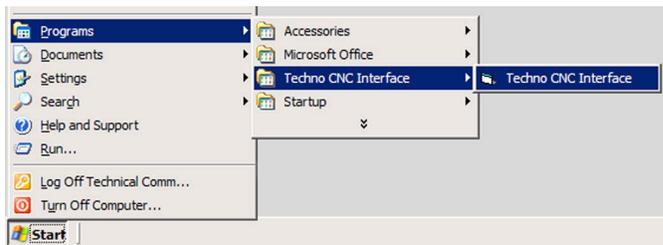
This tutorial employs and explains the main functions of the Techno CNC GCODE Interface as they pertain to a Manual Quick-Change Tool Change operation. You will learn the initial steps and functions by running the sample program, **2tools.nc**. If you have any difficulty with the tutorial or are just learning to use your machine, we recommend reading the INTERFACE FUNCTIONS before proceeding.

Note: This Tutorial was written assuming that:

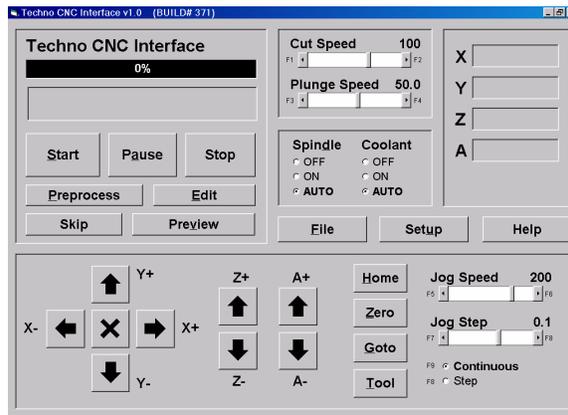
1. You have already setup the touchpad.
2. You have already set tool length offsets for at least the first two tools (as instructed in Chapters IV).

1. START THE INTERFACE

A. From your computer's Start menu, choose Programs/Techno CNC Interface.



The Techno CNC Interface Main Menu appears:



2. HOME THE MACHINE

A. Press **Home** .

The **Home** Window appears:

B. Press **ALL** .



3. LOAD TOOL HOLDER 1.

If tool holder 1 is in the chuck, skip to Step 4. If tool holder 1 is not in the chuck, continue to Step 3-A.

If a Tool Holder is currently in the Chuck, follow these directions to remove it.

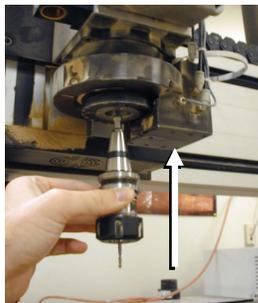
Retrieve the tool holder in this manner: firmly cradle and grip the sides of the tool holder prior to the chuck opening. The chuck opens when you press the Toggle Switch on the Manual Quick-Change box, which is adjacent to the spindle. Do not place your hand, palm-up, under the tool to catch the tool holder. A sharp tool may cut you when the tool is released.

Note: If the chuck is already empty, skip step Step 3-A and go directly to Step 3-B

- A. Open the chuck by pressing the Toggle Switch on the Manual Quick-change box (see picture below), retrieve the existing tool holder and put it of to the side.



- B. Insert tool holder 1 into the spindle (see picture below).



Note: When the chuck closes, it will grasp the tool holder and lift it into the chuck a fraction of an inch. Allow for this lift.

- C. Press the chuck switch to close the chuck.

4. ZERO THE Z-AXIS

Note: Prop the touchpad on something high enough so that your Z-axis zero position will be well above the table surface (see picture below). Do this to ensure that the tool does not travel into the table or material during the program run. Then, after you zero the Z-axis, remember to remove the prop.



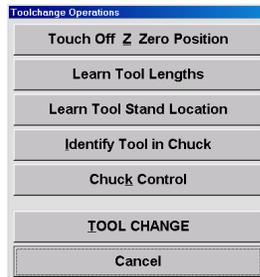
A. Jog your tool over the touchpad.

Note: Prior to Zeroing the Z-axis, we will test the touchpad and if it is working properly, then we will Zero the Z-axis. Follow the Steps and Warning below.

Warning: Remember to use the **E-Stop Button**, located on your controller or start stop box, if you need to immediately stop the machine.

B. Press .

The **Tool** Window appears:



The Test: Here we will start the *Touch Off Process*, wherein the Z-Axis will automatically travel down to the touchpad. If the touchpad is working properly, once the tool comes into contact with the pad, the Z-axis will retract immediately. Our test involves the user manually lifting the touchpad up to the tool, as in Step 3-D (see picture below). When you lift the touchpad up to the tool, if the Z-axis retracts, then the pad is operational. If the Z-axis DOES NOT re-tract, press the red E-stop button to stop the Z-axis. Then, check your touchpad setup before returning to this Tutorial.

C. Press .

The machine will automatically lower the Z-axis.



D. Manually lift the touchpad to the tool and then gently touch the pad to the tool (see picture above).

Note: If the Z-axis retracts upon contact, continue to Step 3-E. If it does not retract, check your touchpad setup and return to the beginning of this section of the tutorial after you have the correct setup.

E. Jog your tool over the touchpad.

F. Press .

G. Press .

The machine will automatically lower the Z-axis to the touchpad and retract when it makes contact.

Note: You can now clear the touchpad AND the prop from the gantry table.

5. ZERO THE X AND Y AXES

A. Jog the X/Y axes to a central position on your gantry table.

Note: Zero positions are frequently set at the front left corner of your workpiece.

B. Press .

The **Zero** Window appears:

C. Press .

This will Zero the X-axis.

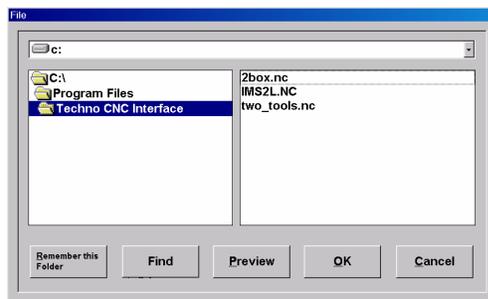
D. Repeat process for Y-axis.



6. OPEN MANUAL QUICK-CHANGE TOOL CHANGE FILE

A. Press .

The **Open File** Window appears:

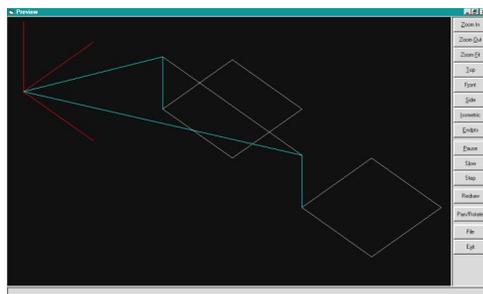


B. Highlight **2tools.nc** and press .

7. PREVIEW THE FILE

A. Press .

The **Preview** Window appears:

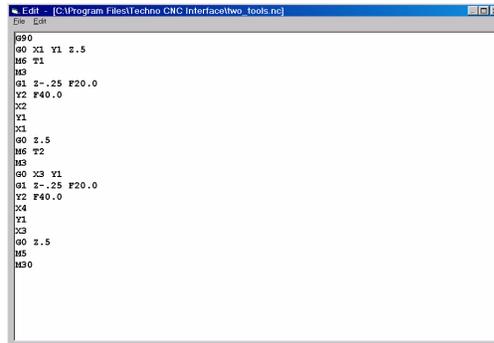


B. Manipulate the toolpath using the buttons along the right side.

C. Press **Exit** to close window.

D. Press **Edit**.

The **Edit** Window appears:



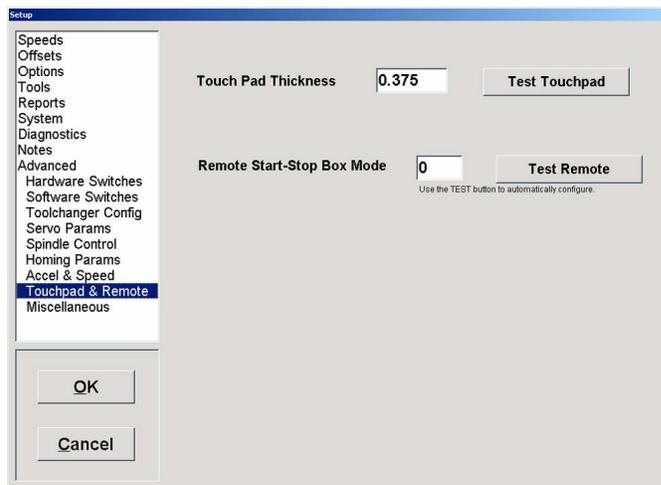
Note: Step 5-D (Edit/Step) is not a mandatory function to run a file; it is, however, a useful feature to know.

F. Press **Close**.

8. CHECK SPEEDS AND PARAMETERS

A. Press **Setup**.

The **Setup** Window appears:



B. Navigate through the different bookmarked submenus by clicking on the names in the left-side display screen.

Warning: Do not change any parameters in Setup. We will run **2tools.nc** using the Interface's default settings.

C. Press **Cancel** to exit Setup.

9. PREPROCESS THE FILE

- A. Press .

Watch the Progress bar move from left to right.

10. START THE FILE

Warning: Before you begin to run a file, remember the different methods you have to stop the program run. For emergencies always use the **E-Stop Button**, located on your controller or start stop box. For non emergencies or to pause and resume a program run, use the **Pause Button** on your Start Stop Box.

Check that your **Z-Axis** zero position provides enough space between the tool and the table so that when you run the file the tool does not plunge into the table. In addition to checking the **Z-Axis**, jog both the **X** and **Y Axes** to the center of the table because the file will not run if it is at the end of any axes.

- A. Press .

Approximately half way through the file, you will be instructed to load tool 2. Follow the onscreen prompts.

10. CHANGE TO TOOL 2

- C. Remove tool 1 using the chuck toggle switch.
- D. Insert tool 2 and close chuck (also using Toggle Switch).

The machine will now complete the file using tool 2.

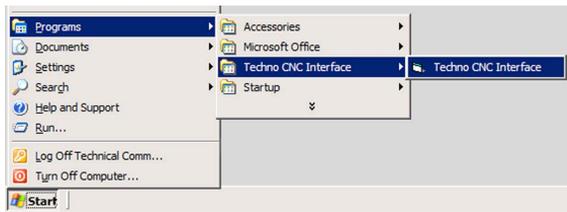
VII. Manual Tool Change Tutorial

This tutorial explains and details the main functions of the Techno CNC GCODE Interface as they pertain to a Manual Tool Change operation. You will learn the initial steps and functions by running the sample program, **2tools.nc**. If you have any difficulty with the tutorial or are just learning to use your machine, we recommend reading the INTERFACE FUNCTIONS before proceeding.

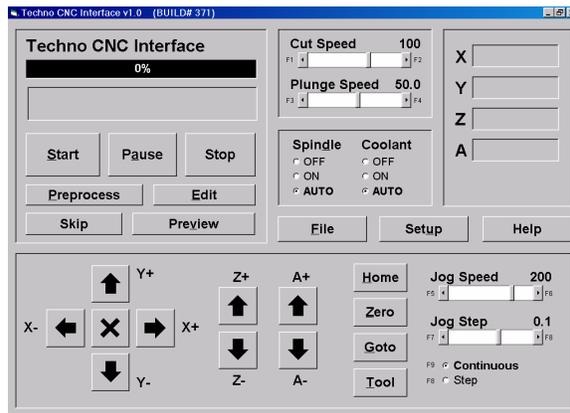
You will need some cutting material, at least 1.5 inches in height, that will be used only to set the Z-axis Zero Position at a high enough level that the cutter will not come close to contacting the gantry table. The sole purpose of this Tutorial is to familiarize the user with the process of changing cutters/tools during a tool change program. You will use the cutting material to set the Z-axis (and X/Y Zero) Part-Zero Position, then you will remove the material from the gantry.

1. START THE INTERFACE

- A. From your computer's Start menu, choose Programs/Techno CNC Interface.



The Techno CNC Interface Main Menu appears:



2. HOME THE MACHINE

- A. Press **Home**.

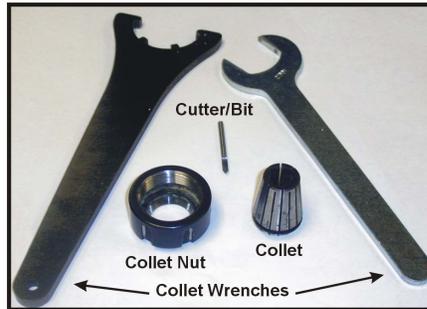
The **Home** Window appears:

- B. Press **ALL**.

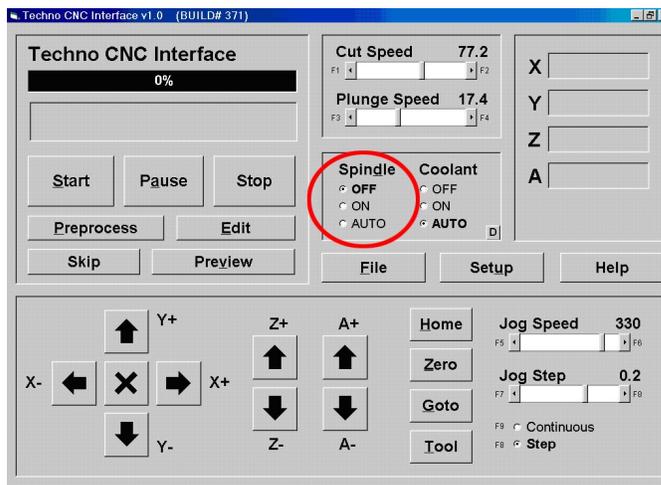


3. INSERT CUTTER/TOOL 1 INTO SPINDLE.

If tool 1 is already inserted, skip to Step 4.



Warning: Turn **OFF** power to the spindle before removing or replacing the collet!



- A. Remove the collet and collet nut from spindle, using the collet wrenches provided. Separate collet from collet nut.

It is important to know exactly how the collet fits into the collet nut. If not done properly, this could damage your spindle.

- B. Insert collet into collet nut and then turn upside down. See pictures below.



- C. Press down on top of the collet nut to snap the two together for proper fit.

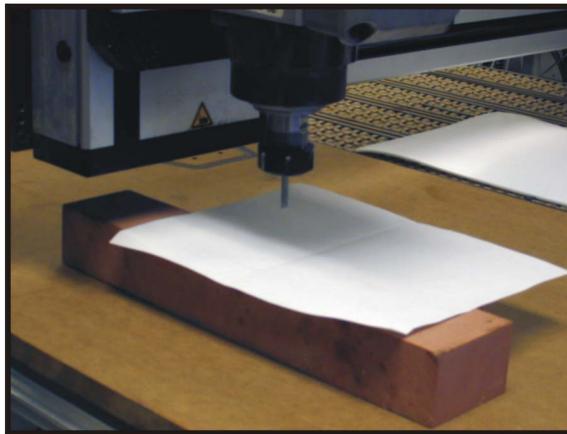
- D. Screw the collet nut (with collet properly inserted) onto the spindle. Continue screwing until either you cannot go any further by hand or the collet hole is too small to fit the cutter / tool bit.
- E. Get a coworker to help you secure the cutter in the collet. Have your coworker hold the cutter in the collet while you secure it with the collet wrenches.



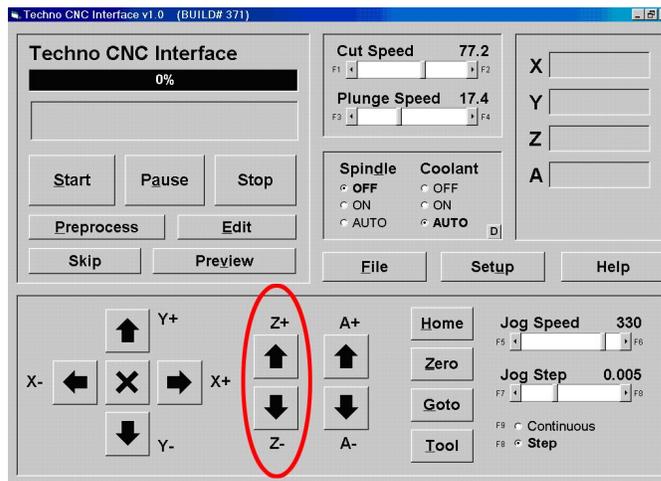
4. ZERO THE Z-AXIS

Note: Take your time while zeroing the Z-Axis. Perform each step slowly and deliberately.

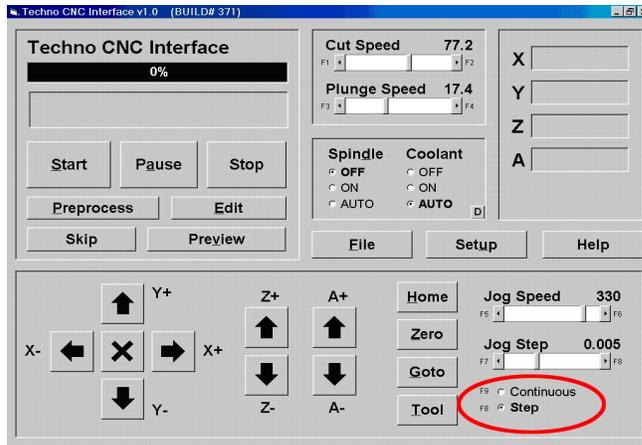
- A. Find some cutting material at least 1.5 inches in height to use as a decoy. Place the cutting material on the gantry table in a central, but easy to reach location.
- B. Place a regular piece of paper on top of the cutting material.



- C. Using the Z Jog Buttons to jog the Z-axis down about 1/4 to 1/2-inch above the paper.

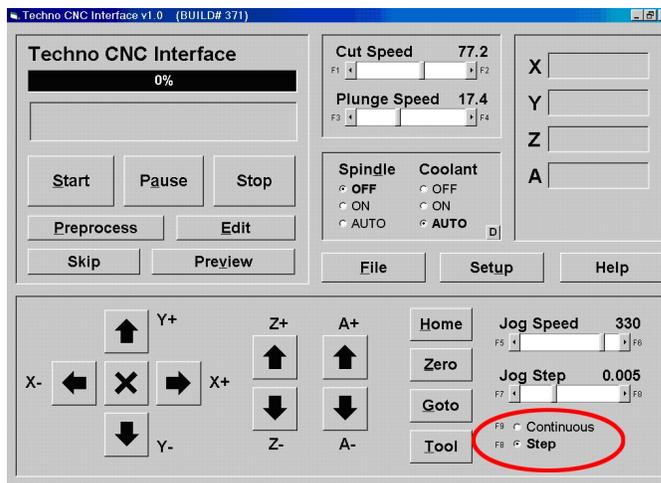


- D. In the Interface, switch from **Continuous** to **Step** (just click on the **Step** radio button or pressing F8 on your keyboard, see picture).



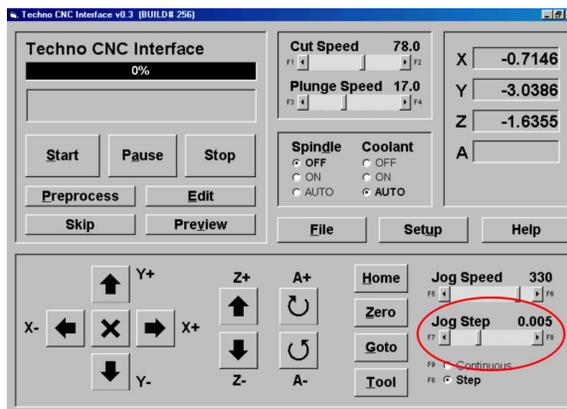
E. Now adjust the **Jog Step** controls to read 0.02 (or thereabouts).

This change from **Continuous** to **Step** allows you to set specific distances per one Jog-click. For example, after setting the Jog Step control to 0.02, every time you **single-click** the Z jog button, the Z-Axis will move **just** 0.02 in distance.



F: Slowly, Single-click the Z-axis down toward the paper until it's *almost*, but definitely not touching.

G. Now adjust the **Jog Step** controls again to read 0.005



- H. Now single-click the Z-axis toward the paper. Stop. Try to move the paper. If it moves freely, single-click again. Stop. Try to move paper.

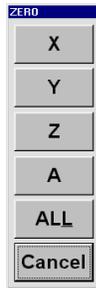
Continue this process until the cutter pinches the paper. Once the paper no longer moves, you are ready to zero the Z-axis.

- J. Press **Zero**.

The **Zero** Window appears:

- C. Press **Z**.

This will zero the Z-axis.



5. ZERO THE X AND Y AXES

- A. Jog the X/Y axes to a central position on your gantry table.

Note: Zero positions are frequently set at the front left corner of your workpiece.

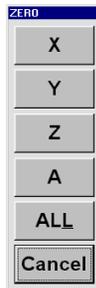
- B. Press **Zero**.

The **Zero** Window appears:

- C. Press **X**.

This will Zero the X-axis.

- D. Repeat process for Y-axis.

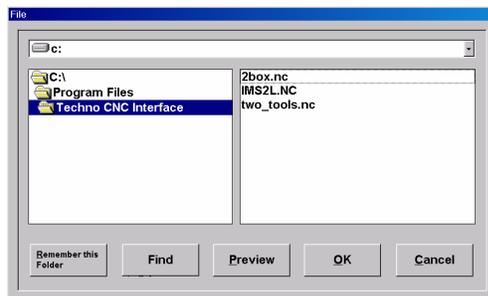


IMPORTANT! DO NOT PROCEED UNTIL YOU REMOVE the cutting material from the gantry and Jog the Z-axis up a safe distance from the gantry table. This is just a Tutorial to familiarize the user with the tool change process and the Interface in general, NOT an actual cutting application! Remove the cutting material to set the Z part-zero position and jog the Z-axis up.

6. OPEN MANUAL TOOL CHANGE FILE

- A. Press **File**.

The **Open File** Window appears:

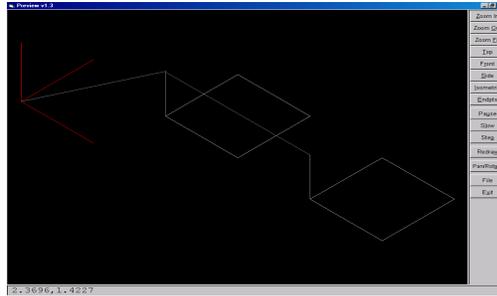


- B. Highlight **2tools.nc** and press **OK**.

7. PREVIEW THE FILE

A. Press **Preview**.

The **Preview** Window appears:

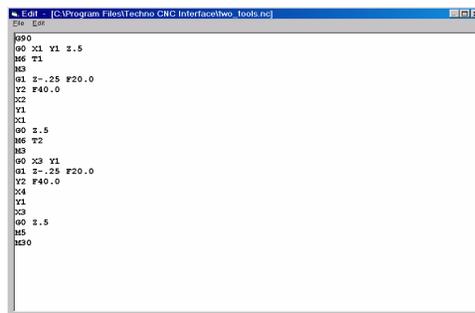


B. Manipulate the toolpath using the buttons along the right side.

C. Press **Exit** to close window.

D. Press **Edit**.

The **Edit** Window appears:



Note: Step 5-D (Edit/Step) is not a mandatory function to run a file; it is, however, a useful feature to know.

D. Press **Close**.

8. SET THE PAUSE FOR MANUAL CHANGES FUNCTION IN SETUP

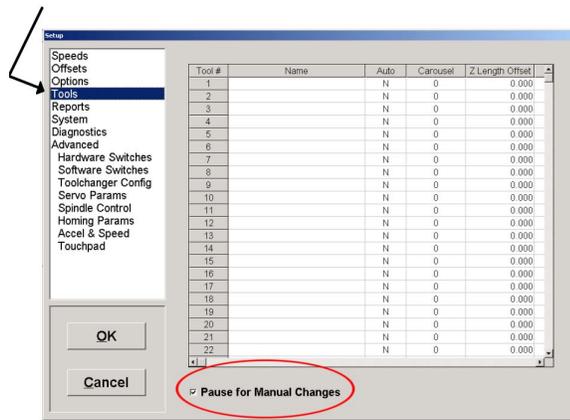
Warning: Do not change any parameters in Setup except those specified in this Tutorial. We will run **2tools.nc** using the Interface's default settings. Read the INTERFACE FUNCTIONS manual, before changing any parameters.

A. Press **Setup**.

The **Setup** Window appears.

Note: Navigate through the different bookmarked submenus by clicking on the names in the left-side display screen.

B. Click on **Tools**



- C. Select the **Pause for Manual Changes** function at the bottom of the **Setup/Tools** Window (see picture above). This function is activated when a check is in the box.

Note: This Tutorial also assumes your speeds/feeds are set correctly (i.e. in the neighborhood of 25 inches/per/minute).

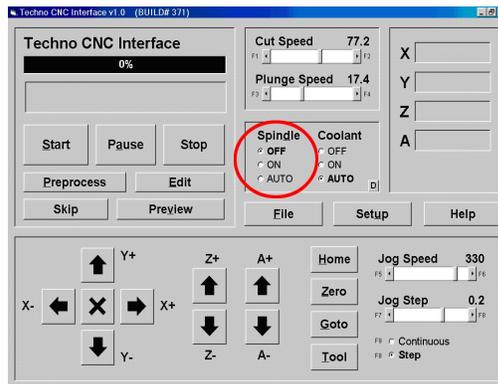
- D. Press  to exit **Setup** and to save any changes made.

9. PREPROCESS THE FILE

- A. Press .

Watch the Progress bar move from left to right.

10. TURN SPINDLE POWER OFF



Since this is not a live cutting Tutorial, the Spindle does not need to be activated.

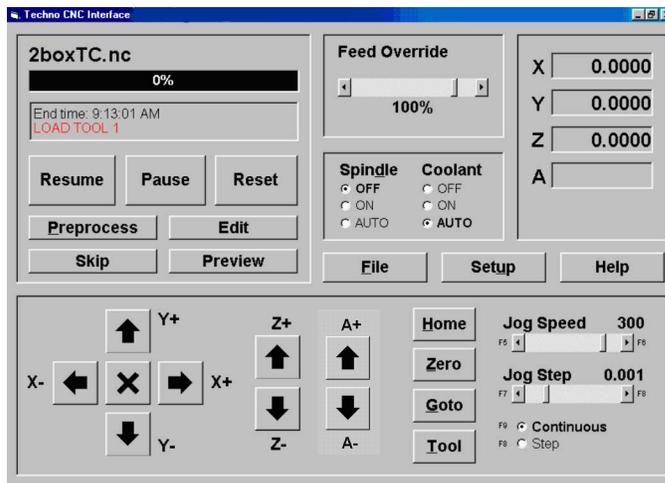
11. START THE FILE

Warning: Before you begin to run a file, remember the different methods you have to stop the program run. For emergencies always use the **E-Stop Button**, located on your controller or start stop box. For non emergencies or to pause and resume a program run, use the **Pause Button** on your Start Stop Box.

Check that your **Z-Axis** zero position provides enough space between the tool and the table so that when you run the file the tool does not plunge into the table. In addition to checking the **Z-Axis**, jog both the **X** and **Y Axes** to the center of the table because the file will not run if it is at the end of any axes.

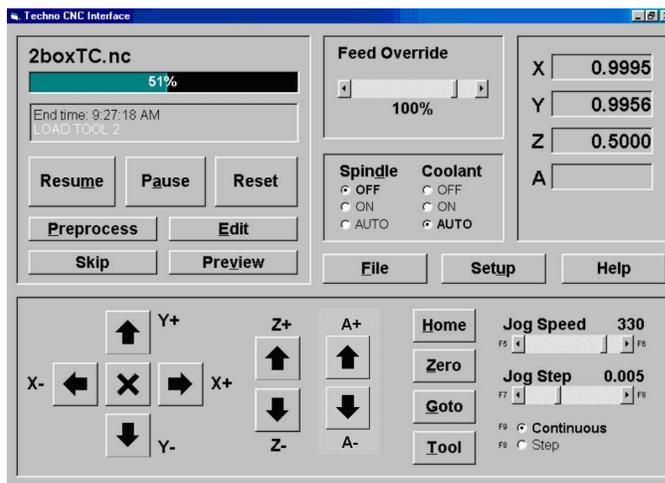
A. Press **Start**.

You will be prompted to load tool 1:



B. Since we already did that in Step 3, simply press **Resume** to start the program.

Approximately half way through the file, you will be instructed to load Tool 2.



Note: When the file stops for the tool change, if the Z-axis is too low for easy-access, jog the Z-axis up and/or over to perform the tool change.

Load tool 2 by following ALL the directions in Step 3, including zeroing the Z-axis (reset the cutting material so you can zero the new cutter using the same reference point(s) as the first). Simply, substitute tool 2 wherever it references tool 1.

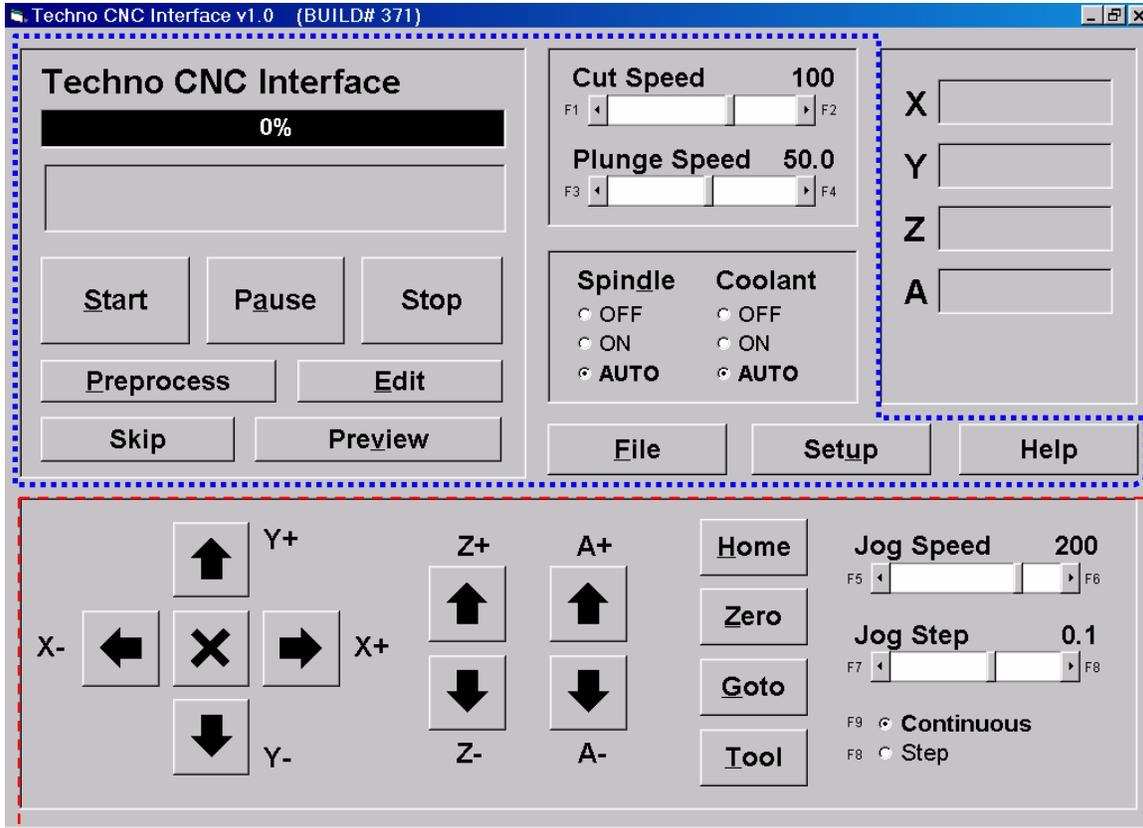
C. Load cutter/tool 2.

D. Press **Resume**.

The Interface will now finish the program run using tool 2.

VIII. Interface Functions

Techno CNC Interface Main Menu



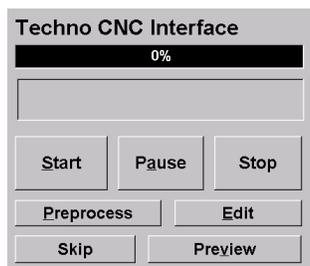
 = File Execution Functions

 = Jogging Functions

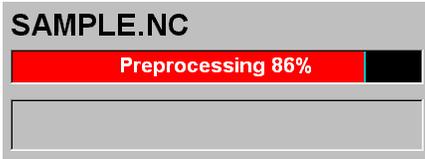
The Techno CNC Interface Main Menu is extremely user-friendly. It is the menu you will see and use the most. The Main Menu allows you to Select, Start and Pause a file; Home and Zero the machine; adjust Speeds and Spindle controls and much-much more.

Because the Main Menu has so many elements and functions, to better explain them to you, we divided it into its two functional partitions: *File Execution* and *Jogging*. Here we will discuss the different File Execution Functions first, then we will move on to Jogging Functions.

File Execution



If any section of the Main Menu could be called the **CONTROL CENTER**, this portion would; simply because from here you can control the operation of the file: Start, Pause and Stop a program run, Edit code, view the file's toolpath and progress, etc. Next to the buttons below, we will explain and show the relevance of each of these **FILE EXECUTION CONTROLS**, including their respective windows.



The **PROGRESS BAR** displays the percentage of your program's progress as well as progress during the preprocessing of your file (shown at left). When the percentage reaches 100, the bar turns from red to green.



Press the **START** button when you are prepared to run your file. If you pause the file, notice that the button becomes the Reset button.



Press the **PAUSE** button to temporarily stop a program run or preprocessing.



Press the **STOP** button to end a program run or preprocessing. If you pause a file-run, notice that the **STOP** button becomes the **RESET** button.

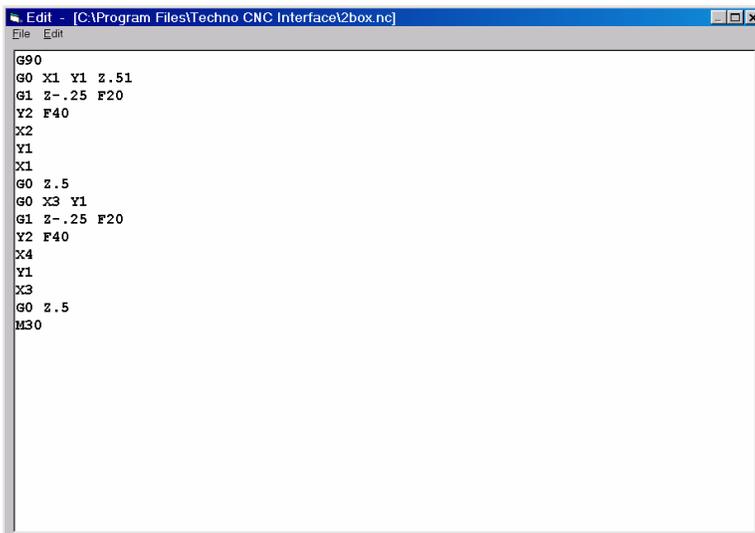


PREPROCESSING calculates all the programs motions prior to executing the file. This means calculating all the accelerations and decelerations based on the speeds and feeds. You will notice that the Preprocess function must be performed on any file to be run for the first time. If you try to run a new file without preprocessing, the Start button will appear 'greyed-out' and thus, inactive.



EDIT button brings up the Edit/Step window (as shown below). This function allows you to view and edit the file's text (or code).

The Edit Window



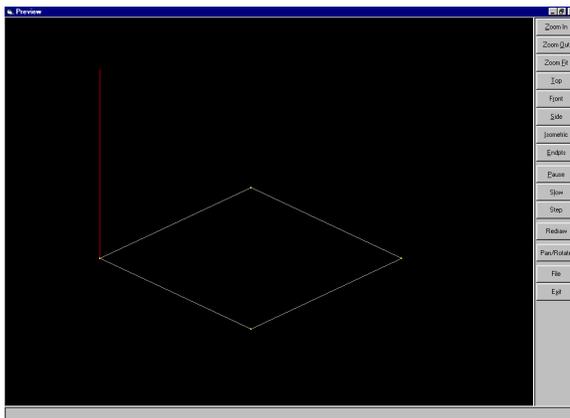
Skip

The **SKIP** button is designed to scan through a GCODE file to find the closest location to where the program was interrupted. For instance, if a one-hour file has been cutting for 45 minutes and the cutter broke while nobody was there, then how could the operator pickup from where the cutter broke? The Skip button allows the operator to stop the machine (giving you the opportunity to replace the cutter) and recalibrate the cutter length. Then, the operator jogs back to the approximate position from where the cutter broke and presses the **SKIP** button. The GCODE file (or program) is scanned; comparing the position you jogged the machine to. Once it matches the position of the machine relative to the command sequence of the file, it will display that location, allowing the operator to resume from that rediscovered position.

Preview

The **PREVIEW** button brings up the Preview window (as shown below). This function allows you to view the toolpath prior to the execution of your file. The buttons on the right side of the Preview window allow you to manipulate the display. You have various Zoom options, you can look at the toolpath from the Top, Front, Side and in an Isometric view, you can redraw the toolpath and you can toggle between the Pan and Rotate functions. If you are using a mouse, the left mouse button will allow you to pan, the right to rotate. If you are using a touchscreen, the function on the left of the slash (/) is activated. Toggle between the two by simply pressing the button.

The Preview Window

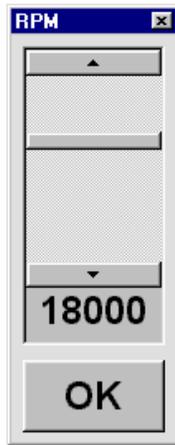


FILE EXECUTION PARAMETERS

Spindle	Coolant
<input checked="" type="radio"/> OFF	<input type="radio"/> OFF
<input type="radio"/> ON	<input type="radio"/> ON
<input type="radio"/> AUTO	<input checked="" type="radio"/> AUTO

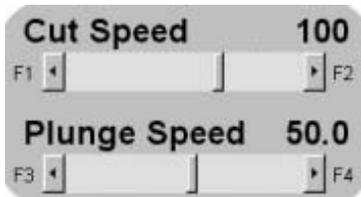
In order to utilize the optional **SPINDLE** and **COOLANT** controls in the Techno CNC Interface, the Spindle must be connected to the power box, and the Coolant must be connected to the controller card.

If your Spindle is connected, power is controlled by clicking the radio button corresponding to On, Off or Auto under Spindle. If you set the Spindle to Auto, power to the Spindle will go on and off as specified in your program (or GCODE file). Whenever the spindle is turned On, a delay window will pop up notifying you that the program has stopped running for a preset delayed amount of time (see the Speeds tab in the Setup window) allowing the spindle motor to ramp up to speed.

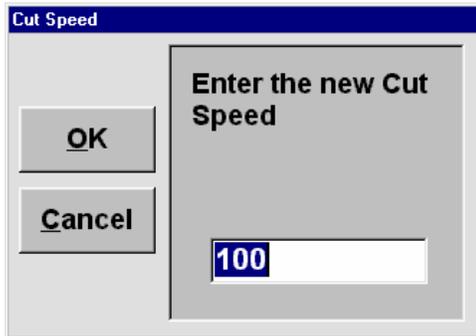


If you need to adjust the Spindle RPM rate, you can do this in the Speeds tab in Setup, OR you can perform a shortcut to changing the Spindle RPM by holding down for an extended mouse-click on the ON radio button (the program will pause during this operation). The Spindle RPM mini-window will appear (as shown at left).

If your Coolant is connected, its release is controlled by clicking the radio button corresponding to On, Off or Auto under Coolant. If you set it to Auto, Coolant will be released as specified in your program.



The Speed Scroll Bars set the speed, in units per minute, for their respective controls. **Cut Speed** is rate the M1 and M2 motors move their axes while the machine is in the process of cutting. **Plunge Speed** is the rate the M3 motor moves its axis while the machine is in the process of cutting.



If you would rather not use the scroll bar to adjust the Cut Speed, you can double click the area above the Cut Speed Scroll bar and a shortcut menu (shown at left) will appear.



If you would rather not use the scroll bar to adjust the Plunge Speed, you can double click the area above the Plunge Speed Scroll bar and a shortcut menu (shown at left) will appear.

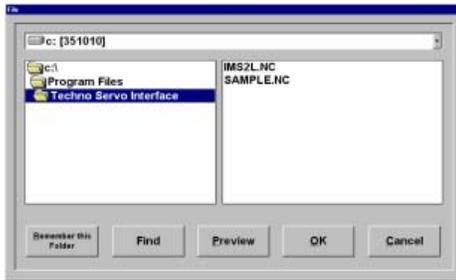
FILE EXECUTION SELECTION & SETUP



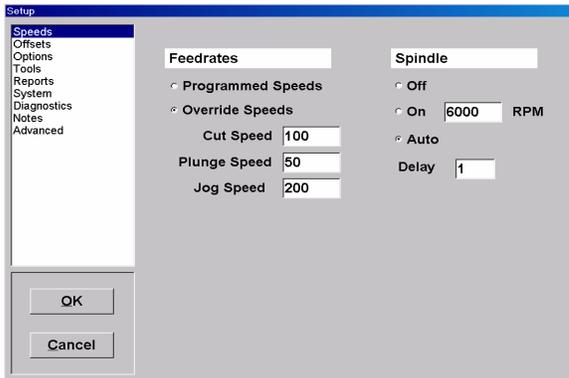
The horizontal set of **FILE**, **SETUP** and **HELP** buttons each bring up a different window or different sets of windows.



The **File** button brings up the File window (shown below). This function allows you to browse, choose and preview the file you want to run.



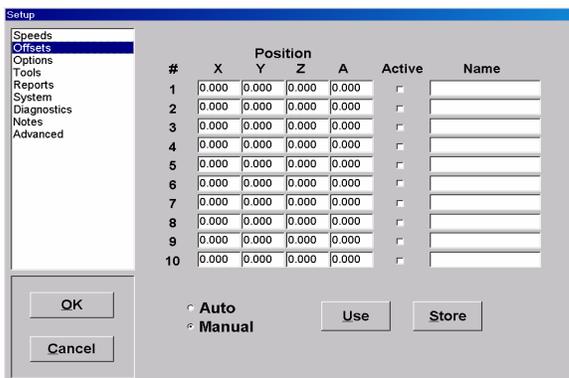
The **SETUP** button brings up the Setup window (as shown below) and its various tabbed panes within it. To access any of Setups different tabs, simply click on the name.



The **SPEEDS** tab allows you to adjust the Spindle's Speeds and feedrates. You have the option of using the CAD/CAM programmed speeds from your file or overriding the set federates and entering in new values in the Techno Interface. To choose the programmed speeds click on the Programmed Speeds radio button. This will grey-out the Cut, Plunge and Jog speeds. To ignore the programmed speeds and set them

yourself, click on the Override Speeds radio button and then fill in the desired speeds for Cut, Plunge and Jog in the adjacent input boxes.

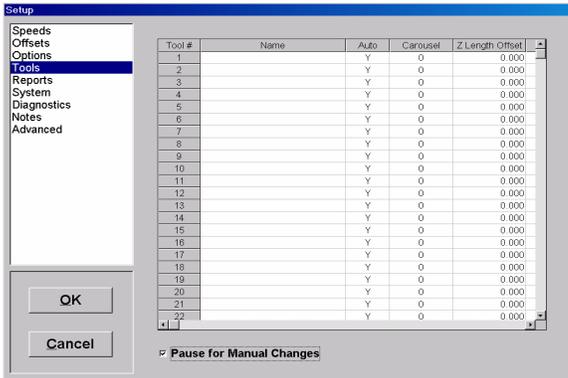
Set the Spindle Delay in seconds by inputting the value in the Delay input box (shown above with a two second delay).



The 'offset' in **OFFSETS** tab refers to the work-piece or work-area offsets. Note that there is only one Active box checked; this is because you can only activate one set of offsets at a time. You can have several offset values inputted into the many rows available, but only one row can be active. Choose between the Auto and Manual

radio buttons by clicking either radio button. Auto, automatically applies the selected offsets when the machine is homed.

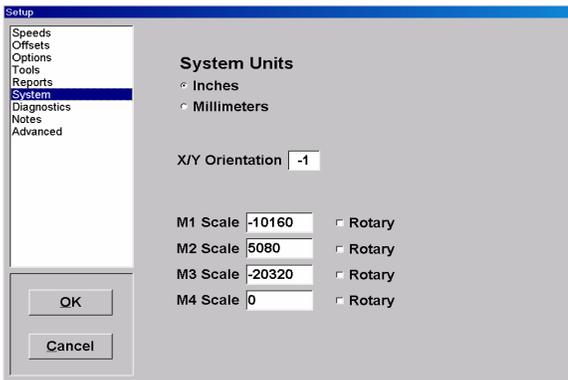
The **TOOLS** tab is the ‘Information Center’ for all your Tool data. The **Tool/Stand Location** and **Length Offset** columns in the scrollable area get programmed automatically when you teach the Tool Stand Locations and Tool Lengths, respectively.



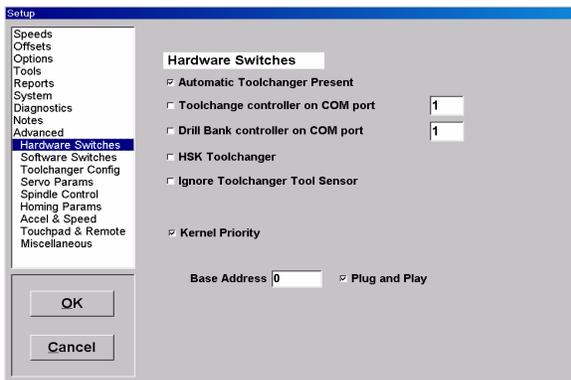
The Auto checkbox column to the right of the M3 column denotes whether you want an automatic or manual toolchange; a check means yes.

The **X Offset Y** column refers to the distance between multiple spindle head or tooling options on the Z-axis.

The Pause for Manual Changes check box controls whether the machine will pause, alerting you to a request for a manual toolchange.

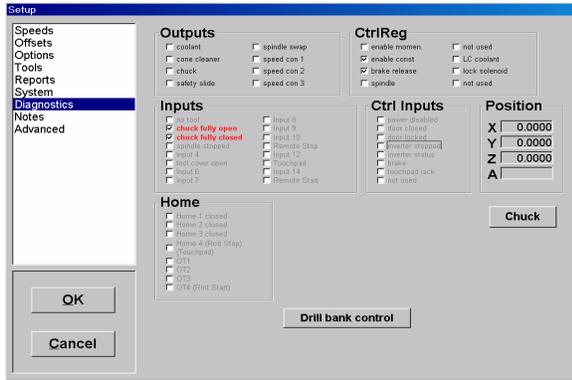


The **SYSTEM** tab refers to options for: units of measurement, machine orientation and motor scale factors. The X/Y Orientation is either +1 or -1. You can choose which one you want by clicking either button. Each button has a pictorial representing the positive directions of the X and Y-axes, from the reference point of you standing in front and facing the machine. The typical factory setting is set at -1.



The **ADVANCED** tab represents the Factory Default values set for Speed rates, Acceleration rates, etc. The values represent maximum limits for each function. As the tab implies, these settings are not to be changed by the novice user.

Note: A password can be entered to prevent operators from accidentally entering the setup and changing any settings with the nested tabs.



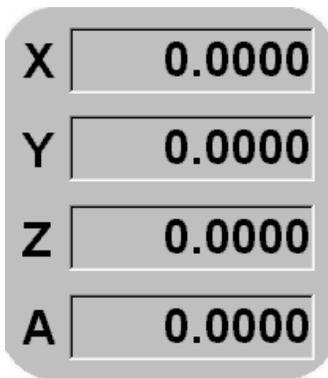
The **DIAGNOSTIC** tab is as a self-evaluation sheet for your machine and software, used in trouble shooting applications. If, for instance, you call for Tech Support, the Help Desk will more than likely direct you to the Diagnostic tab.

Help

The Help button brings up the pre-installed help system menu. Follow the onscreen instructions.

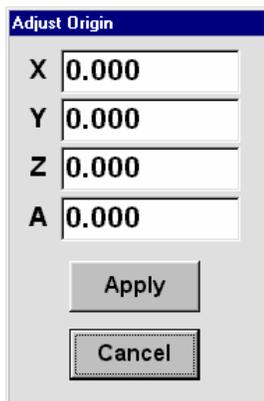
JOG FUNCTIONS

JOG POSITION



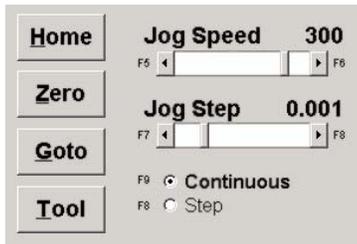
The **POSITION DISPLAY** reflects the present location of each axis in relation to the existing software home, either while jogging or during a program run. The position number tells you how many units away from the software home (in either direction) the axes are.

As you jog to the position where you want to set your new software home, the position counters (X, Y, Z [and A if applicable]) will change to reflect the new locations of the axes. To establish a location as your new software home position, the position counters in the Run Window must be reset to a Zero All position before running your program.



A shortcut window to manipulate the axes position will pop up when you double-click in the Position Display area. The purpose of this window (shown at left) is to be able to tweak the position counters while the program is in Pause-mode. After you make the changes you want, click the Apply button and the changes will be made when after the window closes.

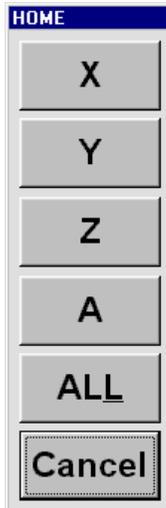
JOG PARAMETER CONTROLS



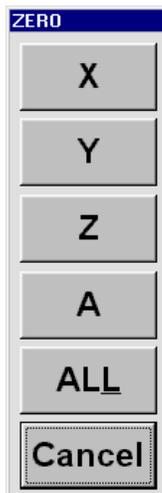
Set or change the **JOG SPEED** rate and **JOG STEP** size in any one of four ways. One, move the scroll bar. Two, use the arrows at either end of the scroll bar. Three, change the value by pressing the keyboard shortcuts, F5 through F8. Four, double click the area above the scroll bars and a shortcut window will appear. Simply, enter the desired value in the input box and press OK.

F9 will toggle between **CONTINUOUS** Motion or **STEP** Jogging. The Continuous/Step radio buttons give you the option of jogging 'continually' or one 'step' at a time. The difference in 'feel' should be obvious. When in Step-mode, the machine will jog in the set increment in Jog Step. When in Continuous-Mode, the axes jog in a smooth, even and continuous fashion.

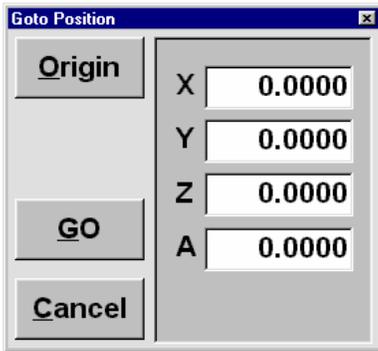
The vertical set of **HOME**, **ZERO**, **GOTO** and **TOOL** buttons each bring up a different window or different sets of windows. What they all have in common is that they all relate to machine positions. **HOME** moves an axis or axes to a specified distance away from an axis' limit switch (or end of travel) also known as machine home. The **ZERO** button allows you to set an axis, or the position counter of multiple axes, at zero. The **GOTO** button allows you to move the axes to its Origin or to a specified location by inputting values into the axes input boxes. The **TOOL** button brings up a Toolchange Operations window that allows you, through a series of sub menus, to manipulate any aspect of a toolchange application. Below are the windows that appear when you click on each button.



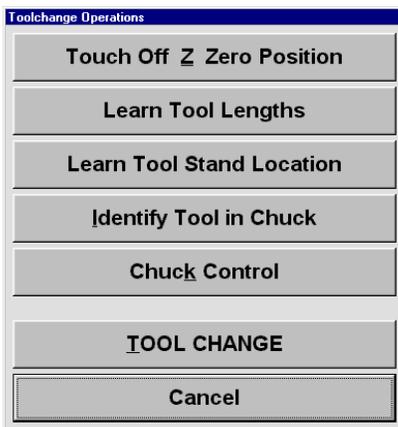
The **HOME** button window allows you to home any axis by pressing its letter or all of them by pressing ALL.



The **ZERO** button window allows you to zero any axis by pressing its letter or all of them by pressing ALL.



The **GOTO** button window allows you to move any axis to a specified location or move all of them to the machine Origin by pressing Origin.



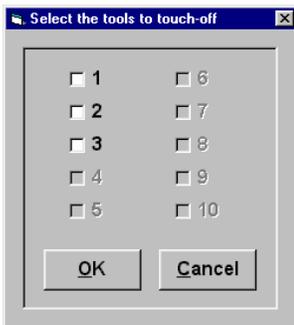
The **TOOLCHANGE OPERATION** window appears when you press Tool. This window will allow you to perform all the functions on the buttons contained within the window. The buttons and windows below show and explain these functions. Note: If you have not homed the machine before pressing the Tool button, this warning window (shown here) will appear.



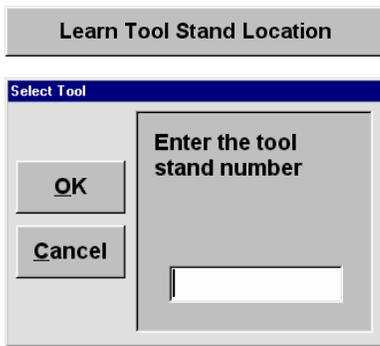
Before pressing the **TOUCH OFF Z ZERO POSITION** button, make sure the touch pad is directly under the cutter. Once you press this button, the Z-axis begins the touch-off process. The ESC button will abort the run.



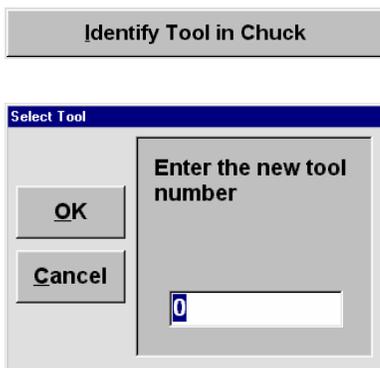
When you press the **LEARN TOOL LENGTHS** button, the Select the tool to touch off window appears (shown here).



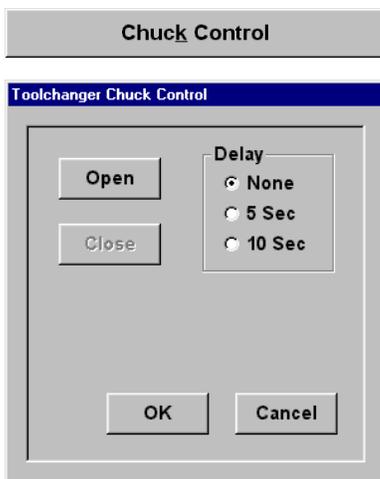
At this point you must click each tool # you want to calibrate the tool length offsets. Once selected, press OK and the machine starts to pick up and touch-off the tools. Again, this is done assuming you have the toolpad in position for the touch-off process.



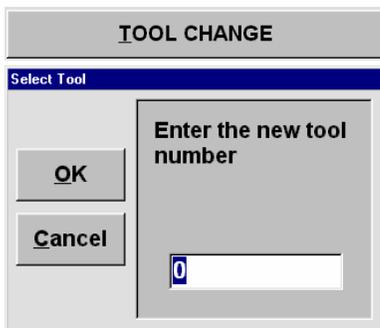
Press the **LEARN TOOL STAND LOCATION** button when you have already jogged the tool into the tool stand and you want to record this position relative to a specified tool stand number (i.e. 1, 2, 3 etc.). After pressing the button, the Select Tool window (shown at left) will appear. Enter the tool stand number into the input box, then press OK and you will have set the tool stand location for one of your tool stands.



Press **THE IDENTIFY TOOL IN CHUCK** button when you need to identify the tool in the chuck. For instance, if after a software crash, the Interface does not remember what number tool is in the chuck, the interface, however, does remember the tool stand locations. So, if you input the tool number into the input box (shown at right) you will be able to return the tool to the proper tool stand.

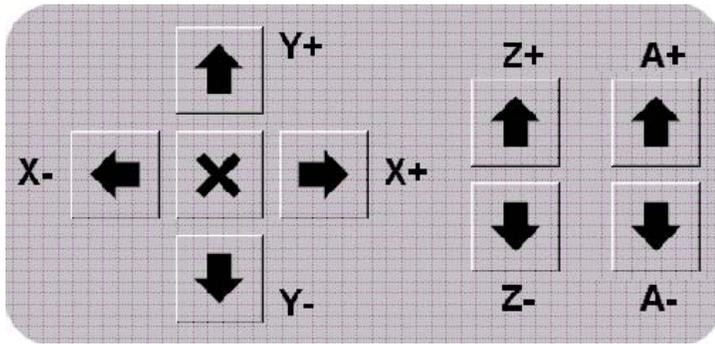


Press the **CHUCK CONTROL** button when you want to set the amount of time-delay before the chuck opens and releases the tool. Your options are None, 5 seconds and 10 seconds. The purpose here is to give you enough time to move from wherever your computer controls are to the spindle in order to catch the tool being released from the chuck.



Press the **TOOL CHANGE** button when you want to retrieve a specific tool. Simply enter the number of the tool you want into the Select Tool window's input box and press OK. The machine will then drop off a tool, if one is present in the chuck, and proceed to retrieve the tool number you specified.

JOG CONTROLS



Each axis has specific arrow buttons that will move (or jog) the axis in either a positive or negative direction. The value of the letter (= or -) designates in which direction that arrow will jog the axis. There are three sets of jog buttons: the X/Y-axes buttons, the Z-axis buttons and the A-axis buttons.

Summary Of GCODE Commands By Category

SET UP COMMANDS

CODE	COMMAND	FORMAT	PURPOSE	PAGE #
F	Feed Speed	Fn	Designates feed rate, or rate of movement, of the axes.	23
G4	Dwell Time	G4/d	Specifies a programmed delay during a drill cycle.	27
G90	Absolute Coordinates	G90	Indicates that <i>absolute coordinates</i> will be used to perform subsequent motion commands.	34
G91	Incremental Coordinates	G91	Indicates that <i>incremental coordinates</i> will be used to perform subsequent motion commands.	35
G92	Set Absolute Position	G92XxYyZzAa	Sets new absolute coordinates for the current position.	35
M0	Program Pause	M0	Pauses the .NC file run.	37
M30	End Of Data	M30	Designates the end of a block of commands in a file.	40
M3 M4	Spindle On	M3 <i>or</i> M4	Starts the spindle.	37
M5	Spindle Off	M5	Turns the spindle off.	38
M6	Tool Change	M6T#	Allows the tool to be changed by turning off the spindle and coolant and pausing the program.	39
M7 M8	Coolant On	M7 <i>or</i> M8	Turns the coolant on.	39
M9	Coolant Off	M9	Turns the coolant off.	40
M57	Digital I/O CNTRL	M57	Turns Output ON/OFF, Checks for Inputs	41
M90	Output Off	M90 OUT#	Turns off the designated output.	42
M91	Output On	M91 OUT#	Turns on the designated output.	42

ROUTING COMMANDS

CODE	COMMAND	FORMAT	PURPOSE	PAGE#
G0	Rapid Move	G0XxYyZzAa	Moves one or more of the axes, at the rapid speed, to a specified location.	23
G1	Linear Cutting Move	G1XxYyZzAa	Moves one or more of the axes along a straight line, at the cutting speed, to a specified location.	24
G2	Clockwise Arc	G2XxYyIiJi <i>or</i> G2XxZzIiKi <i>or</i> G2YyZzJiKi	Moves two of the axes, at the cutting speed, along an arc in a clockwise direction to a specified location.	25
G3	Counterclockwise Arc	G3XxYyIiJi <i>or</i> G3XxZzIiKi <i>or</i> G3YyZzJiKi	Moves two of the axes, at the cutting speed, along an arc in a counterclockwise direction to a specified location.	26

DRILL CYCLE COMMANDS

CODE	COMMAND	FORMAT	PURPOSE	PAGE #
G80	Drilling Cycle Off	G80	Turns a drill cycle off.	27
G81	Standard Drilling Cycle Without Dwell	G81XxYyZzRrPpFf	Provides a feed-in, rapid-out sequence used for standard drilling without a dwell time.	28
G82	Standard Drilling Cycle With Dwell	G82XxYyZzRrPpFf	Provides a feed-in, rapid-out sequence used for standard drilling with a specified dwell time.	29
G83	Peck Drilling Cycle	G83XxYyZzRrQqVvPpFf	Provides a series of feed-in, peck drilling motions with full retract used for drilling holes.	30
G87	Chip Break Drilling Cycle	G87XxYyZzRrQqVvWwPpFf	Provides a series of feed-in, rapid-out motions that is similar to peck drilling, but the retract is a specified distance.	32

GENERAL DRILL CYCLE COMMAND FORMAT

G8n Xx Yy Zz Rr Qq Vv Ww Pp Ff

- X** the x coordinate of the hole
- Y** the y coordinate of the hole
- Z** the z coordinate of the hole
- R** the reference height for start of drill plunge
- Q** the initial z peck increment
- V** the subsequent z peck increment.
- W** the peck clearance, z retract between pecks
- P** the dwell time (seconds)
- F** the drilling federate, plunge speed
- n** an integer in the range of zero to nine

G-Code And The Interface Program

The Electronic Industries Association developed a standard for a code it defines as an "interchangeable variable block data format for positioning, contouring, and contouring/positioning numerically controlled machines". The standard for this general machine code, known as G/M-Code or, more commonly, G-Code, is EIA-274-D.

The Techno G-Code CNC Interface is designed to recognize some of the standard G/M codes. The codes it supports can be used to revise an existing G-Code (.NC) file or to create an original file that can be run by the Techno G-Code Interface. The codes that are recognized by the Techno Interface are listed and described in detail in this chapter. And at the end, a sample G-Code file is written out for you to look over. For more comprehensive information about General Machine Code, or for a complete listing of all G/M codes, refer to a G-Code language manual.

 *In standard G-Code command definitions, the word "program" refers to the G-Code program, also known as the .NC file. Commands such as Program Stop or End Of Program refer to stopping or ending the .NC file, not the interface program.*

Each G/M code recognized by the interface program is described in this chapter using the format shown below.

Code Number	Command Name
Format	Presents the syntax of the command for a specific code. The <i>format</i> illustrates the way the command is written in the program. In many cases, the command will involve variables, which will be indicated by symbols that are defined in the text.
Purpose	Explains how the command is used and what it is used for.
Example	A sample set of commands for the code is presented and explained.

Commands Recognized By The Techno Interface

F Feed Speed

Format Ff

f = rate in units/minute

Purpose Designates the rate of movement of the axes. This command can be used to set the feed rate for cutting moves, the plunge rate for drilling cycles, and the rapid speed for rapid moves. It is ignored if **Override Program Speed** is selected in the Techno CNC G-Code Interface.

Example G1X5.00Y4.00Z-2.00F2.00

The linear cutting motion indicated above will be performed at a rate of two units per minute when **Override Program Speed** has not been selected in the interface program. If the unit selected in the interface program is inches, then the cutting motion will be performed at a rate of two inches per minute.

G0 Rapid Move

Format G0XxYyZzAa

X,Y,Z,A = axis identifiers

x, y, z, a = distance in incremental units, or absolute coordinates, to which the axes specified by the preceding upper case letter is designated to move

Purpose Moves one or more of the axes, at the rapid speed, to a specified location or a specified distance away from the current position. Location is designated by absolute coordinates when this command is preceded by the *Absolute Coordinate* command. Distance is specified by incremental units when this command is preceded by the *Incremental Coordinate* command.

A rapid move consists of a move of the X and/or Y axis along the XY plane and a move of the Z axis up or down. If the Z-axis move is Directed from a lower point to a higher point, it will be executed before the XY move; if it is directed from a higher point to a lower point, it will be executed after the XY move.

Rapid speed is indicated in the .NC file by the *Feed Speed* command. F commands are ignored by the interface program when **Override Program Speed** is selected. When this selection is made, the rapid speed specified in Set-Up will be used instead.

Example **G90**
 G0X2.00Y2.00Z2.00

The first command in this set indicates absolute motion, so the rapid move that follows (line 2) is made to the absolute coordinates (2,2,2). The axes move to a position that is two units away on each axis from the software home.

G1 **Linear Cutting Move**

Format **G1XxYyZzAa**

X,Y,Z,A = axis identifiers

x, y, z, a = distance in incremental units, or absolute coordinates, to which the axes specified by the preceding upper case letter is designated to move

Purpose Moves one or more of the axes, at the cutting or feed speed, along a straight line to a specified position or a specified distance away from the current position. Location is designated by absolute coordinates when this command is preceded by the *Absolute Coordinate* command. Distance is specified by incremental units when this command is preceded by the *Incremental Coordinate* command.

Example **G91**
 G1X-3.50Y-4.00Z1.00
 M30

The first command in this set indicates that the cutting command in line 2 will be performed in incremental motion. In line 3, the axes will move at the cutting speed as follows: the X axis will move 3.5 units away in a negative direction from its previous position, the Y axis will move 4 units away in a positive direction from its previous position, and the Z axis will move 1 unit away in a positive direction from its previous position.

G2

Clockwise Arc

Format

G2XxYyIiJi
or
G2XxZzIiKi
or
G2YyZzJiKi

X,Y,Z = axes on a three axis machine

x, y, z = distance in incremental units, or absolute coordinates, to which the axes specified by the preceding upper case letter is designated to move

I = X axis center point position

J = Y axis center point position

K = Z axis center point position

i = an incremental value representing the distance in units to the center position on the axis specified by the preceding letter

Purpose

Moves two of the axes, at the cutting speed, along an arc in a clockwise direction, to a specified location or a specified distance away from the current position. The arc is defined by the movement of the two axes around a designated center point, from the current position to a specified end point. The two axes used will determine the plane in which the axes will move along the arc.

If the Clockwise Arc command is preceded by the *Absolute Coordinate* command, the X,Y,Z values will be absolute. If it is preceded by the *Incremental Coordinate* command, these values will be incremental. Whether absolute or incremental, the X,Y,Z values represent axis positions marking the end point of the arc. Center point values are always incremental.

Example

G92X3.00Y3.00Z3.00
G91
G2X5.00Y-1.00I3.00J2.00
M30

The first command in this set establishes the current position as 3 units away on each axis in a positive direction from the software home. The line 2 command specifies that the axes move in incremental movement.

In line 3, the axes will move along an arc in a clockwise direction in the XY plane. The X and Y axes will move along the arc from the current position (3,3) to the end point, which is designated to be 5 units away in a positive direction along the X axis and 1 unit away in a negative direction along the Y axis from the current position. The end point is position (8,2). The X and Y axes will move clockwise around a center point that is 3 units away in a positive direction along the X axis and 2 units away in a positive direction along the Y axis from the current position. The center point position is (6,5).

G3

Counterclockwise Arc

Format

G3XxYyIiJi
or
G3XxZzIiKi
or
G3YyZzJiKi

X,Y,Z = axes on a three axis machine

x, y, z = distance in incremental units, or absolute coordinates, to which the axes specified by the preceding upper case letter is designated to move

I = X axis center point position

J = Y axis center point position

K = Z axis center point position

i = an incremental value representing the distance in units to the center position on the axis specified by the preceding letter

Purpose

Moves two of the axes, at the cutting speed, along an arc in a counterclockwise direction, to a specified location or a specified distance away from the current position. The arc is defined by the movement of the two axes around a designated center point, from the current position to a specified end point. The two axes used will determine the plane in which the axes will move along the arc.

If the Counterclockwise Arc command is preceded by the *Absolute Coordinate* command, the X,Y,Z values will be absolute. If it is preceded by the *Incremental Coordinate* command, these values will be incremental. Whether absolute or incremental, the X,Y,Z values represent axis positions marking the end point of the arc. Center point values are always incremental.

Example

G92X3.00Y3.00Z3.00
G90
G3Y9.00Z15.00J3.00K6.00
M30

The first command in this set establishes the current position as 3 units away on each axis in a positive direction from the software home. The line 2 command specifies that the axes move in absolute movement. In line 3, the axes move along an arc in a counterclockwise direction in the YZ plane. The Y and Z axes move along an arc from the current position (3,3,3) to the end point. The end point is designated to be 9 units away in a positive direction along the Y axis and 15 units away in a positive direction along the Z axis from the software home (0,0,0). The end point is position (3,9,15). Since center point values are always incremental, the Y and Z axes will move counterclockwise around a center point that is 3 units away in a positive direction along the Y axis and 6 units away in a positive direction along the Z axis from the current position. The center point position is (3,6,9).

G4 **Dwell Time**

Format **G4/d**
 or
 G4 d

d = duration of the timed delay in seconds

Purpose Specifies a programmed delay, indicated in seconds. Note that a forward slash (/) or blank space must follow G4 in this command.

Example **G0X2Y2Z2**
 G4/10
 G0X8.00 Y6.00 Z5.00

In this example, the two rapid moves will be separated by a 10 second delay.

G80 **Drilling Cycle Off**

Format **G80**

Purpose Turns off a drill cycle. This command should follow immediately after the last drilling cycle command.

Example **G90**
 G0X0Y0Z0
 G81X1.00Y2.00Z-1
 X4.00Y5.00
 X7.00Y8.00
 G80

In the command set above, the first command designates subsequent motion commands to be in absolute motion. The standard drill commands in lines 3, 4 and 5 create a series of three holes. (Notice that the drill command code is written only once, in line 3, although it applies to lines 4 and 5 as well.) After the third hole is drilled, the drill cycle is cancelled.

G81

Standard Drilling Cycle Without Dwell

Format

G81XxYyZz

X,Y,Z = axes on a three axis machine

x, y, z = distance in incremental units, or absolute coordinates, to which the axes specified by the preceding upper case letter is designated to move

Purpose

Provides a feed-in, rapid-out sequence used for drilling a series of holes. Standard drilling involves a single plunge of the tool at the plunge speed, and a single retraction of the tool to the initial Z position at the rapid speed. The initial Z position is the position of the Z axis before the drilling cycle starts.

The speed at which the Z axis descends into the material will either be the feed rate specified in the .NC file by the *Feed Speed* command or the plunge speed specified in Set-Up. The interface program will only recognize the *Feed Speed* command if **Override Program Speed** is not selected in Set-Up.

In drilling, the command code usually appears on the first line of the cycle only, although there may be several holes drilled in the cycle. Also, the Z axis value is always unsigned (i.e., there is no positive or negative sign in front of it).

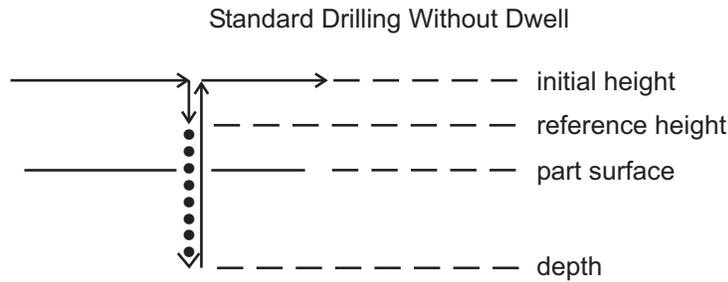
Example

```
G90  
G81X4.00Y6.00Z-1  
X6.00  
X8.00  
G80  
G0X10.00Y10.00Z0.00
```

Line 1 indicates that subsequent motion commands will be in absolute motion. In the standard drilling cycle initiated in line 2, the X axis moves 4 units in a positive direction away from the software home and the Y axis moves 6 units in a positive direction away from the software home. The X and Y axes move at the cutting speed set in the interface program. A hole is drilled at the new location (4,6). To drill the hole, the Z axis moves 1 unit in a negative direction away from its current (initial) position and then retracts to its initial position. The Z axis moves at the plunge speed set in the interface program.

Two other holes of the same depth are drilled, one at the (6,6) position and one at the (8,6) position. The drill cycle is ended in line 5, and a rapid move is executed in line 6. This command is executed in absolute motion, according to the *Absolute Coordinate* command in line 1.

Illustration



Key

.....	drill motion at plunge speed
—————	rapid motion at rapid speed
initial height	height at which rapid motions between drill holes takes place
reference height	height at which drill is moved at rapid speed before drilling takes place

G82 Standard Drilling Cycle With Dwell

Format

G4/d
G82XxYyZz

X,Y,Z = axes on a three axis machine

x, y, z = distance in incremental units, or absolute coordinates, to which the axes specified by the preceding upper case letter is designated to move

Purpose

Provides a feed-in, dwell, rapid-out sequence used for standard drilling and counterboring operations. G82 Standard Drilling With Dwell involves a single plunge of the tool at the plunge speed, a timed delay when the Z axis reaches its lowest point, and a single retraction of the tool to the initial Z position at the rapid speed. The initial Z position is the position of the Z axis before the drilling cycle starts. The duration of the delay is specified by the Dwell command preceding the drill command.

The speed at which the Z axis descends into the material will either be the feed rate specified in the .NC file by the *Feed Speed* command or the plunge speed specified in the Set-Up window of the interface program. The interface program will only recognize the *Feed Speed* command if **Override Program Speed** is not selected in Set-Up.

In drilling, the command code usually appears on the first line of the cycle only, although there may be several holes drilled in the cycle. Also, the Z axis value is always unsigned (i.e., there is no positive or negative sign associated with it).

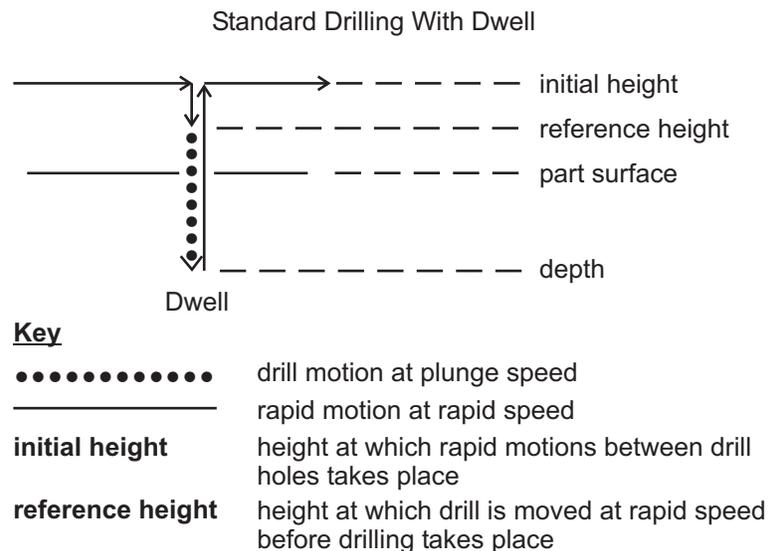
Example

```
G90
G82X10.00Y8.50Z-0.5P10
G80
G0X1.00Y1.00Z9.00
M30
```

The X and Y axes move at the cutting speed set in the interface program. A hole is drilled at the new location, (10,8.5). To drill the hole, the Z axis moves 0.5 units in a negative direction away from its current (initial) position at the plunge speed, dwells at the bottom of the hole for 10 seconds (P10), and then retracts to its initial position at the rapid speed.

The drill cycle is ended in line 3, and a rapid move is executed in line 4. This rapid move is performed in incremental motion, as specified by the *Absolute Coordinates* command in line 1.

Illustration



G83

Peck Drilling Cycle

Format

```
G83XxYyZtZfZs
or
G83XxYyZtZf
```

X,Y,Z = axes on a three axis machine

x, y, z = distance in incremental units, or absolute coordinates, to which the axes specified by the preceding upper case letter is designated to move

t = total Z axis depth

f = first peck increment

s = subsequent peck increments

Purpose

Provides a series of feed-in, rapid-out motions used for deep hole drilling. In peck drilling, the tool performs a series of plunges, or pecks, and retractions at the same XY position.

Each peck is done at the plunge speed, and each retraction is done at the rapid speed. The speed at which the Z axis descends into the material will either be the plunge rate specified in the .NC file by the *Feed Speed* command or the plunge speed specified in Set-Up. The interface program will only recognize the *Feed Speed* command if **Override Program Speed** is not selected in Set-Up.

The total depth for the Z axis is specified in the *Peck Drilling Cycle* command. The depth of the first peck and of all other (subsequent) pecks are also specified. The total number of pecks is determined by the total Z axis depth and the depths of the peck increments designated in the G-Code command. Assuming that the total Z axis depth is 10, the first peck increment is 4, and subsequent peck increments are 2, there will be a total of 4 pecks. This figure is arrived at by subtracting the depth of the first peck from the total Z axis depth, then dividing the remainder by the value for subsequent peck increments. If the value of subsequent peck increments is omitted from the G-Code command, all peck increments will have the same value as the first peck.

The Z axis retracts to its initial position after each peck increment. The initial position of the Z axis is its position before the drilling cycle starts. Also, the Z axis value is always specified as unsigned (i.e., there is no positive or negative sign associated with it).

In drilling, the command code usually appears on the first line of the cycle only, although there may be several holes drilled in the cycle.

Example

```
G90  
G83X-5.00Y-2.50Z12.00Z4.00Z2.00  
G80  
G0X10.00Y10.00Z0.00
```

In line 1 of the command set above, the *Absolute Position* command sets the current position as the new software home. Line 2 indicates that subsequent motion commands will be in absolute motion. The line 3 command specifies a peck drilling cycle to take place at the (-5,-2.5) position. The Z axis is specified to plunge a total of 12 units deep.

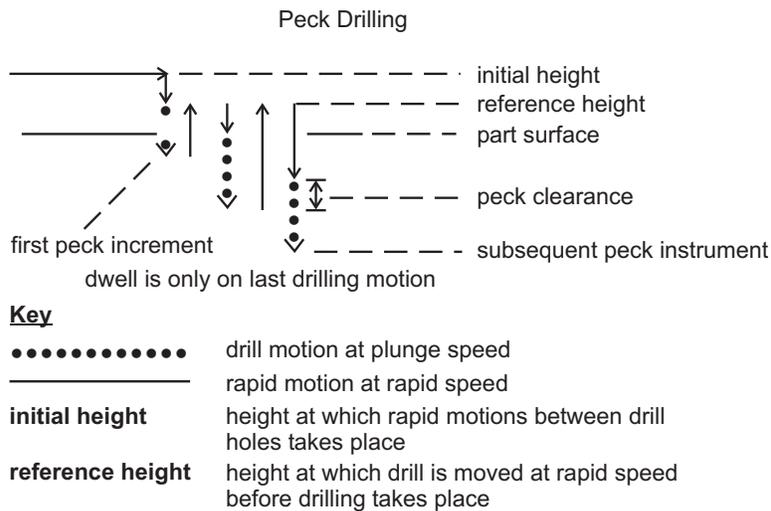
The first peck will be made at a depth that is 4 units away from the current Z axis position (0). The tool will then retract to the initial

height, the Z=0 point. There will be four additional pecks, making five pecks in total.

The second peck will be made at a position that is 2 units away from the bottom of the first peck. The bottom of the second peck will be -6 units along the Z axis from the initial position, which the tool will retract to after the peck is made. Each of the three remaining pecks in the series will be made at a position that is 2 units away in a negative direction from the bottom of the previous peck. When the tool retracts after each peck, it will return to the initial position, which in this case, is the Z=0 point.

The drill cycle is ended in line 4, and a rapid move is performed in line 5. The rapid move is executed in absolute motion, as specified by the *Absolute Coordinates* command in line 2.

Illustration



G87 Chip Break Drilling Cycle

Format **G87XxYyZtZfZs**
 or
 G87XxYyZtZf

X,Y,Z = axes on a three axis machine,
x, y = distance in incremental units, or absolute coordinates, to which the axes specified by the preceding upper case letter is designated to move
t = total Z axis depth
f = first peck increment
s = subsequent peck increments

Purpose

Provides a series of feed-in, rapid-out motions used for deep hole drilling. This drilling cycle is similar to peck drilling, except that the retract amount is a specified distance. The tool performs a series of plunges, or pecks, and retractions at the same XY position. Chip break drilling focuses on breaking the surface of the material rather than on withdrawing the tool completely from the piece.

The total depth for the Z axis is specified in the Peck Drill command. The depth of the first peck and of all other (subsequent) pecks are also specified. The total number of pecks is determined by the total Z axis depth and the depths of the peck increments specified in the G-Code command. Assuming that the total Z axis depth is 10, the first peck increment is 4, and subsequent peck increments are 2, there will be a total of 4 pecks. This figure is arrived at by subtracting the depth of the first peck from the total Z axis depth, then dividing the remainder by the value for subsequent peck increments. If the value of subsequent peck increments is omitted from the G-Code command, all peck increments will have the same value as the first peck.

Each peck is done at the plunge speed, and each retraction is done at the rapid speed. The speed at which the Z axis descends into the material will either be the feed rate specified in the .NC file by the *Feed Speed* command or the plunge speed specified in Set-Up. The interface program will only recognize the *Feed Speed* command if **Override Program Speed** is not selected in Set-Up.

In drilling, the command code usually appears on the first line of the cycle only, although there may be several holes drilled in the cycle. Also, the Z axis value is always unsigned (i.e., there is no positive or negative sign in front of it).

Example

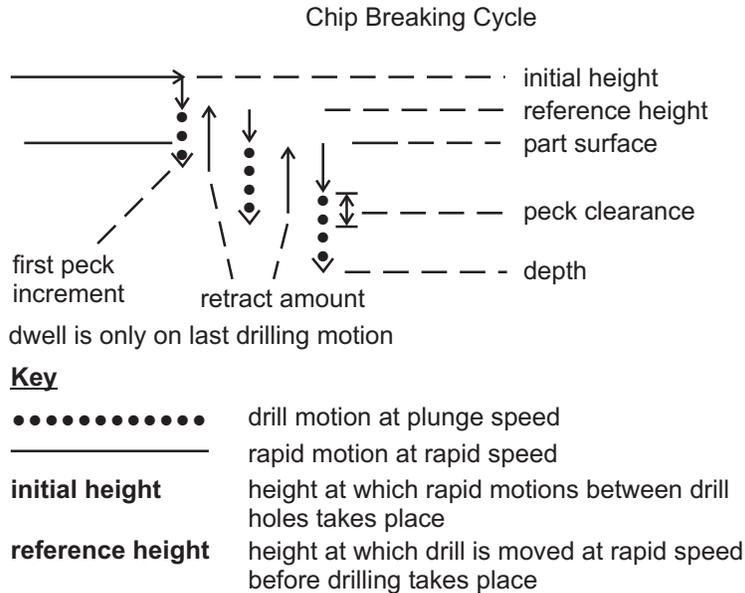
```
G92X0.00Y0.00Z0.00  
G90  
G87X-3.50Y-6.00Z6.00Z2.00  
G80  
G0X7.50Y8.00Z2.00
```

In line 1 of the command set above, the *Absolute Position* command sets the current position as the new software home. Line 2 indicates that subsequent motion commands will be in absolute motion. The line 3 command specifies a chip break drilling cycle to take place at the (-3.5,) position. The Z axis is specified to plunge a total of 6 units deep.

The first peck increment will be made at a depth that is 2 units away from the current Z axis position (0). The tool will then retract .01 inch. From that position, it will plunge to a depth that is 2 units away from the bottom of the first peck. There will be three pecks in all, and after each peck, the tool will retract .01 inch. Since the value of subsequent pecks is not specified, the value of the first peck is used for all pecks.

The drill cycle is ended in line 4, and a rapid move is executed in line 5. The rapid move is performed in absolute motion, as specified by the *Absolute Coordinates* command in line 2.

Illustration



G90 Absolute Coordinates

Format

G90

Purpose

Indicates absolute motion (i.e., movement of the axis to an absolute coordinate) for the motion commands that follow. The values specified for the axes in the motion commands are absolute with respect to the software home position. Absolute motion is the interface program default. All commands are performed in absolute motion unless incremental motion (G91) is specified in the .NC file.

Example

G90
G1X2.00 Y4.00 Z2.00
G1X-1.00 Y5.00 Z0.00

The first line of this command set specifies that subsequent motion commands will be performed in absolute motion.

The motion command in line two sends the X axis to a point two units away in a positive direction (+2) from the software home, the Y axis to a point four units away in a positive direction (+4) from the software home, and the Z axis to a point two units away in a positive direction (+2) from the software home. The coordinates the axes will move to are (2,4,2).

The motion command in line three sends the X axis to the -1 position in relation to the software home (-3 units away from its previous position), the Y axis to the +5 position (+1 unit away from its previous position), and the Z axis to the 0 position (the software home location, -2 units away from its previous position). The coordinates the axes will move to are (-1,5,0).

G91 Incremental Coordinates

Format

G91

Purpose Indicates incremental motion (i.e., movement of the axis to a position a specified distance away from its previous position) for the motion commands that follow. Once specified, commands continue to be performed in incremental motion until absolute motion is indicated in the .NC file.

Example

G91
G1X2.00 Y4.00 Z2.00
G1X-1.00 Y5.00 Z0.00

The first command specifies that subsequent motion commands will be performed in incremental motion.

The motion command in line two sends the X axis to a point two units away in a positive direction (+2) from its current position, the Y axis to a point four units away in a positive direction (+4) from its current position, and the Z axis to a point two units away in a positive direction (+2) from its current position. Assuming the current position of the machine is (1,1,1), the machine will then move to position (3,5,3).

The motion command in line three sends the X axis -1 unit away from its current position of +3, the Y axis +5 units away from its current position of +5, and the Z axis 0 units away from its current position of +3. The machine will move from (3,5,3) to (2,10,3).

G 92 Set Absolute Position

Format

G92 XxYyZz

X,Y,Z = axes whose coordinates are being specified
x, y, z = absolute coordinate being assigned to the axis specified by the preceding upper case letter

Purpose

Sets the new origin, or software home, by changing the absolute position of one or more axes. New absolute coordinates are assigned to the axes in their current positions, thereby establishing a new software home. Any absolute coordinate specified in a subsequent command is referenced to the new software home position. The current position may be designated as the new software home by assigning the absolute coordinate of 0 to each axis.

Example 1

```
G90  
G1X2.00 Y4.00 Z2.00  
G92X0Y0Z0  
G1X-1.00 Y5.00 Z0.00
```

The command on the first line indicates that subsequent motion commands will be performed in absolute motion.

In line two, the X axis will be sent to a point 2 units away in a positive direction from the existing software home, the Y axis will be sent to a point 4 units away in a positive direction from the existing software home, and the Z axis will be sent to a point 2 units away in a positive direction from the existing software home. The coordinates the axes will move to are (2,4,2).

The line three command establishes a new origin, or software home, using Absolute Position. Each axis is assigned a new absolute coordinate - 0 (zero) - for its current position, changing the coordinates of the current position from (2,4,2) to (0,0,0). The current position has been designated as the new origin, or software home. Subsequent motion commands will be referenced to this new software home.

In line four, the axes make a linear cutting move to absolute coordinates. These coordinates are relative to the new software home established in line three. The X axis is specified to move to a position -1 unit away from the new origin (1 unit in a negative direction away from its current position), the Y axis is specified to move to a position +5 units away from the new origin (5 units in a positive direction away from its current position), and the Z axis is specified to move to the new origin (which is its current position, so it does not move). The final coordinates of the axes will be (-1,5,0).

Example 2

```
G90  
G1X2.00 Y4.00 Z2.00  
G92X4Y3Z6  
G1X-1.00 Y5.00 Z0.00
```

The command on the first line indicates that subsequent motion commands will be performed in absolute motion. In line two, the axes are sent to the same positions designated by the line two command in Example 1. (See Example 1 for a description of axes movement.) The coordinates the axes will move to are 2,4,2.

The line three command establishes a new origin, or software home, using Absolute Position. Each axis is assigned a new absolute coordinate for its current position, changing the coordinates of the current position from (2,4,2) to (4,3,6). The current position is referenced to the new origin, or software

home, which is 4 units away in a negative direction along the X axis, 3 units away in a negative direction along the Y axis, and 6 units away in a negative direction on the Z axis. Subsequent motion commands will be referenced to this new origin as well.

In line four, the axes make a linear cutting move to absolute coordinates. These coordinates are relative to the new software home established in line three. The X axis is specified to move to a position -1 unit away from the new origin (5 units away in a negative direction from its current position), the Y axis is specified to move to a position +5 units away from the new origin (2 units away in a positive direction from its current position), and the Z axis is specified to move to the new origin (6 units away in a negative direction from its current position). The final coordinates of the axes will be (-1,5,0).

M0 **Program Pause**

Format **M0**

Purpose Stops the .NC file from running. The file will continue to run from the point where it stopped when you click on the Resume button in the interface..

Example **G91**
G1X4.00Y-2.00Z1.00
M0
G1X5.00Y0.00Z3.00

The first line of this command set specifies that subsequent motion commands will be performed in incremental motion.

Following the first linear cutting command (line 2), the file is stopped using the Program Stop command. After the Resume button is clicked, the file resumes with another linear cutting move.

M3 or M4 **Spindle On**

Format **M3**
or
M4

Purpose Starts the spindle. After the spindle is turned on, the file is paused for the amount of time specified next to Spindle Delay in Set-Up.

Spindle On commands will only be recognized by the Techno CNC G-Code Interface if Auto is selected in the interface.

Example 1

```
M3  
G90  
G1X2.50 Y4.00 Z-2.00  
G1X-1.00 Y5.50 Z-1.00  
M30
```

In the command set above, the Spindle On command precedes all motion commands intended to perform cutting (lines 3 and 4), since the spindle has to be turned on in order for cutting to take place. The *End Of Data* command, M30, turns the spindle off as it ends the block of commands.

Example 2

```
M4  
G91  
G1X8.50Y6.00Z-4.50  
G1X12.00Y2.50Z-3.50  
M30
```

The first command in this set turns the spindle on. Cutting moves are then performed in incremental motion. The *End Of Data* command turns the spindle off at the end of the command set.

M5 Spindle Off

Format

M5

Purpose

Turns the spindle off. This command will stop the spindle only if Auto has been selected in the interface.

Example

```
M3  
G90  
G1X2.00 Y4.00 Z-2.00  
M5  
M0  
M3  
G1X-1.00 Y5.00 Z-3.00  
M30
```

In the command set above, the spindle is designated to be turned on at the beginning, in line 1, and then turned off in line 4, following the linear cutting command. The program is stopped in line 5 to allow for adjustments of the machine, tool, or workpiece before the machine is designated to cut again. After the desired adjustments are made, the file will continue to run from the point where it stopped when the Resume button in the interface program's Run

window is clicked. The *End Of Data* command (M30), which marks the end of the file, turns the spindle off.

M6 Tool Change

Format

M6T#

T indicates "tool"
= tool number

Purpose

Allows the tool to be changed by turning the spindle and coolant off and by stopping the .NC file from running. After the tool change, if the spindle and coolant were on before, they will automatically be turned back on. The tool will be returned to its position prior to the interruption of the .NC file.

After the .NC file stops running as a result of a Tool Change command, the Tool Change window will pop up and show the number of the tool needed for the tool change. The machine can be jogged to a convenient position for changing the tool. The file will continue to run with the new tool in place when the Resume button is clicked.

Example

```
M3  
G90  
G1X2.00Y4.00Z-2.00  
M6T2  
G1X-1.00Y5.00Z-3.00  
M30
```

In the command set above, the spindle is turned on in line 1, absolute motion for subsequent motion commands is specified in line 2, and a linear cutting move is indicated in line 3. A change from one tool to another is indicated by the Tool Change command in line 4. After the change to tool #2 is completed, the spindle is automatically turned back on. The tool is returned to position (2,4,-2). After the Resume button in the interface program's Run window is clicked, the file continues to run, and a linear cutting move in absolute motion is performed. Finally, the *End Of Data* command designates the end of this block of commands and turns the spindle off.

M7 or M8 Coolant On

Format

M7
or
M8

Purpose

Turns the coolant on. This command will only be recognized by the Techno

CNC G-Code Interface if Auto is selected under Coolant in the interface.

Example

```
M7  
G91  
G1X3.00Y5.00Z-3.00  
G1X4.00Y4.00Z-4.00  
M30
```

In the command set above, the coolant is turned on in line 1, before any cutting is performed. Incremental motion for subsequent motion commands is specified in line 2, and linear cutting moves are indicated in lines 3 and 4. The *End Of Data* command designates the end of this block of commands and turns the coolant off.

M9 Coolant Off

Format

M9

Purpose

Turns the coolant off. This command will only be recognized by the Techno CNC G-Code Interface if Auto is selected under Coolant in the interface.

Example

```
M7  
G90  
G1X2.00Y4.00Z-2.00  
G1X-1.00Y5.00Z-3.00  
M9  
G1X-2.00Y6.00Z-4.00  
M7  
G1X-3.00Y8.00Z-2.00  
M30
```

The coolant is turned on, using the Coolant On command, at the beginning of this command set. After the first two cutting commands (lines 3 and 4), the coolant is turned off with the Coolant Off command (line 5). It is turned on again in line 7, after the next cutting command. The subsequent cut (line 8) is made with the coolant on. Then, the program is ended with the *End Of Data* command, which also turns the coolant off.

M30 End Of Data

Format

M30

Purpose

Designates the end of a block of commands in a file. This command also turns off the spindle and coolant if Auto has been selected in the interface. The end

of a G-Code program is normally marked as such using the End Of Data (M30). However, it is not necessary to include this command in an .NC file that will be run in the Techno CNC G-Code Interface. After the interface program executes the last command in the file, it will stop running the file, and will turn off the spindle and coolant.

Example

M48
G90
G1X2.00 Y4.00 Z2.00
G1X-1.00 Y5.00 Z0.00
M30

The block of commands above consists of two linear cutting moves performed in incremental motion. The block is ended by the *End Of Data* command, which also turns off the spindle and coolant.

M57 DIGITAL I/O CONTROL

Format

M57

Purpose

Turns Outputs on-and-off,; checks for Inputs

Example

M57,800 ~ M57,899

Wait for serial IO input # 0 ~ 99 to be ON

M57,900 ~ M57,999

Wait for serial IO input # 0 ~ 99 to be OFF

M57,1000

Set Input wait timeout to INFINITE

M57,1001 ~ M57,1999

Set Input wait timeout to 1 to 999 TENTHS of a second.

For example, **M57,1050** sets timeout to 5.0 seconds.

For example, **M57,1101** sets timeout to 10.1 seconds.

For example, **M57,1555** sets timeout to 55.5 seconds.

M57,600 ~ M57,699

Turns ON serial IO output # 0 ~ 99

For example, **M57,603** turns on output 3.

M57,700 ~ M57,799

Turns OFF serial IO output # 0 ~ 99

For example, **M57,721** turns off output 21.

M57,400 ~ M57,499

Wait for parallel IO input # 0 ~ 99 to be ON

M57,500 ~ M57,599

Wait for parallel IO input # 0 ~ 99 to be OFF

M90 **Output Off**

Format M90 OUT#

Purpose Turns the designated output off. This command is used in conjunction with the M91 command (see description below), which turns the output on. An output number from 0 to 15 must be specified for this command. Only one output can be designated to be turned off by a single command.

Example **M91 OUT1**
G90
G1X-5.00 Y2.00
G1X0.00 Y1.00
M90 OUT1

The first command in this example turns on output 1. The command in the second line indicates that the following commands will be performed in absolute motion. A linear cutting move is made: the X axis moves to the -5 position and the Y axis moves to the +2 position. In the next cutting move, the X axis moves to the 0 position and the Y axis moves to the +1 position. Then, output 1 is turned off.

M91 **Output On**

Format M91 OUT#

Purpose Turns the designated output on. This command is used in conjunction with the M90 command (see description above), which turns power to the output off. An output number from 0 to 15 must be specified for this command. Only one output can be designated to be turned on by a single command.

Example **M91 OUT1**
G90
G1X2.00Y4.00
G1X3.00Y3.00
M90 OUT1

In this example, output 1 is turned on. Then, absolute motion is specified for the commands that follow. A linear cutting move is made next: the X axis moves to the +2 position and the Y axis moves to the +4 position. In the next cutting move, the X axis moves to the +3 position and the Y axis moves to the +3 position. Then, output 1 is turned off.

LC Series and Lathe Control Board Callouts

