

OmniTurn Safety

Read and **heed** all “DANGER “ and “CAUTION” labels on the machine.

MOVING PARTS CAN CAUSE INJURIES! Keep hands and clothing clear of spindle and tooling plate at all times.

DO NOT OPERATE WITH DOOR or SPLASH SHIELD OPEN!

DO NOT OPEN DOOR DURING OPERATION!

DO NOT operate machine without guards, doors and covers in place.

DO NOT operated while under the influence of alcohol or drugs, including prescription drugs.

DO NOT wear loose clothing while operating machine.

DO NOT wear jewelry (necklaces, bracelets, watches, rings) while operating machine.

DO NOT allow unsupported bar-stock to extend past drawtube end.

Keep long hair tied up and out of the way.

Wear eye and face protection.

Report any malfunction to your supervisor.

READ BEFORE OPERATING THIS MACHINE:

- Only authorized personnel should work on this machine. Untrained personnel present a hazard to themselves and the machine, and improper operation will void the warranty.
- Check for damaged parts and tools before operating the machine. Any part or tool that is damaged should be properly repaired or replaced by authorized personnel. Do not operate the machine if any component does not appear to be functioning correctly. Contact your shop supervisor.
- Use appropriate eye and ear protection while operating the machine. ANSI approved impact safety goggles and OSHA approved ear protection are recommended to reduce the risks of sight damage and hearing loss.
- Do not operate the machine unless the doors are closed and the door interlocks are functioning properly. Rotating cutting tools can cause severe injury. When a program is running, the tooling table can move rapidly at any time in any direction.
- The Emergency Stop button is the large, circular red switch located on the Control Panel. Pressing the Emergency Stop button will stop all motion of the machine and the coolant pump. Use the Emergency Stop button only in emergencies to avoid crashing the machine.
- The spindle drive cabinet should be closed and the latches on the CNC control should be secured at all times except during installation and service. At those times, only qualified electricians should have access to the panels. When the main circuit breaker is on, there is high voltage throughout the spindle drive cabinet and CNC control, therefore, extreme caution is required.
- Consult your local safety codes and regulations before operating the machine. Contact the factory whenever safety issues need to be addressed.
- DO NOT modify or alter this equipment in any way. If modifications are necessary, all such requests must be handled by NC Electronics, Inc. Any modification or alteration of any OmniTurn Lathe could lead to personal injury and/or mechanical damage and will void your warranty.
- It is the shop owner's responsibility to make sure that everyone who is involved in installing and operating the machine is thoroughly acquainted with the installation, operation, and safety instructions provided with the machine BEFORE they perform any actual work. The ultimate responsibility for safety rests with the shop owner and the individuals who work with the machine.

OmniTurn Safety

- **Do not attempt to operate the machine before all of the installation instructions have been completed.**
- **Do not operate without proper training.**
- **Do not operate with the door open.**
- **Always wear safety goggles.**
- **The machine is automatically controlled and may start at any time.**
- **When using a chuck:**
 - **Do not exceed rated chuck rpm.**
 - **The chuck must be greased weekly and regularly serviced.**
 - **Chuck jaws must not protrude beyond the diameter of the chuck.**
 - **Do not machine parts larger than the chuck.**
 - **Follow all of the warnings of the chuck manufacturer regarding the chuck and work holding procedures.**
 - **Improperly or inadequately clamped parts may be ejected with deadly force.**
 - **Higher rpm reduces chuck clamping force.**
 - **Improperly clamped parts at high velocity may puncture the safety door. Reduced rpm is required to protect the operator when performing dangerous operations (e.g. turning oversized or marginally clamped parts). Turning oversized or marginally clamped parts is not safe.**
- **Unsupported bar stock must not extend past draw tube end.**
- **Never service the machine with the power connected.**
- **Windows must be replaced if damaged or severely scratched -Replace damaged windows immediately.**
- **Do not process toxic or flammable material. Deadly fumes can be present. Consult material manufacturer for safe handling of material by-products before processing.**

OmniTurn Safety

Follow these guidelines while performing jobs on the machine:

Normal operation - Keep the door closed and guards in place, while machine is operating.

Part loading and unloading – An operator opens the door or lifts splash-guard, completes task, closes door or guard before pressing cycle start (starting automatic motion).

Machining job set-up – Press emergency stop before adding or removing tooling.

Maintenance / Machine Cleaning – Press emergency stop or power off the machine before leaning into cutting area.

DO NOT PUT HANDS INTO THE CUTTING AREA ANYTIME THE MACHINE IS IN MOTION;
SEVERE INJURY MAY RESULT.

Unattended Operation:

Fully enclosed OmniTurn CNC lathes are designed to operate unattended; however, your machining process may not be safe to operate unmonitored.

As it is the shop owner's responsibility to set up the machine safely and use best practice machining techniques, it is also their responsibility to manage the progress of these methods. The machining process must be monitored to prevent damage if a hazardous condition occurs.

For example, if there is the risk of fire due to the material machined, then an appropriate fire suppression system must be installed to reduce the risk of harm to personnel, equipment and the building. A suitable specialist must be contacted to install monitoring tools before machines are allowed to run unattended.

It is especially important to select monitoring equipment that can immediately perform an appropriate action without human intervention to prevent an accident, should a problem be detected.

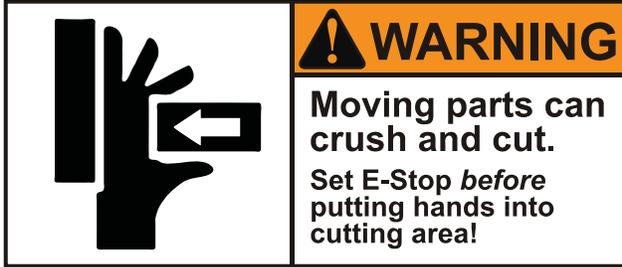
Additional guidelines for proper machine operation:

All turning machines contain hazards from rotating cutting tools, belts and pulleys, high voltage electricity, noise, and compressed air. When using turning machines and their components, basic safety precautions should always be followed to reduce the risk of personal injury and mechanical damage. READ ALL APPROPRIATE WARNINGS, CAUTIONS, AND INSTRUCTIONS BEFORE OPERATING THIS MACHINE.

Modifications to the machine:

DO NOT modify or alter this equipment in any way. If modifications are necessary, all such requests must be handled by the OmniTurn factory. Any modification or alteration of any OmniTurn lathe could lead to personal injury and/or mechanical damage and will void your warranty.

Safety Labels on OmniTurn Machines



This label is found on front of machine by the sliding door or splash guard.

Do not put your hands into the cutting area without setting CNC E-Stop first.



This label is found on front of machine by removable hatch, and on the left side removable panel.

Do not operate the machine with any hatch or panel open.



This label is found on front of machine at far left, and on left side removable panel.

Do not allow barstock to stick out past the end of the drawtube.

READ BEFORE OPERATING THIS MACHINE

All turning machines contain hazards from rotating cutting tools, belts and pulleys, high voltage electricity, noise, and compressed air.

When using turning machines, basic safety precautions should always be followed to reduce the risk of personal injury.

Only authorized personnel should work on this machine. Untrained personnel present a hazard to themselves and to the machine.

READ THE SAFETY SECTION OF THE USER'S MANUAL BEFORE OPERATING THIS MACHINE

This label is found on front of machine.

Only authorized operators who have been trained in basic shop-safety procedures and are aware of the potential hazards should operate any turning machine.

OmniTurn Safety

OmniTurn Factory

42820 Port Orford Loop
Port Orford, Oregon 97465

Phone: 541.332.7004

Fax: 541.332.1018

Email: sales@omniturn.com

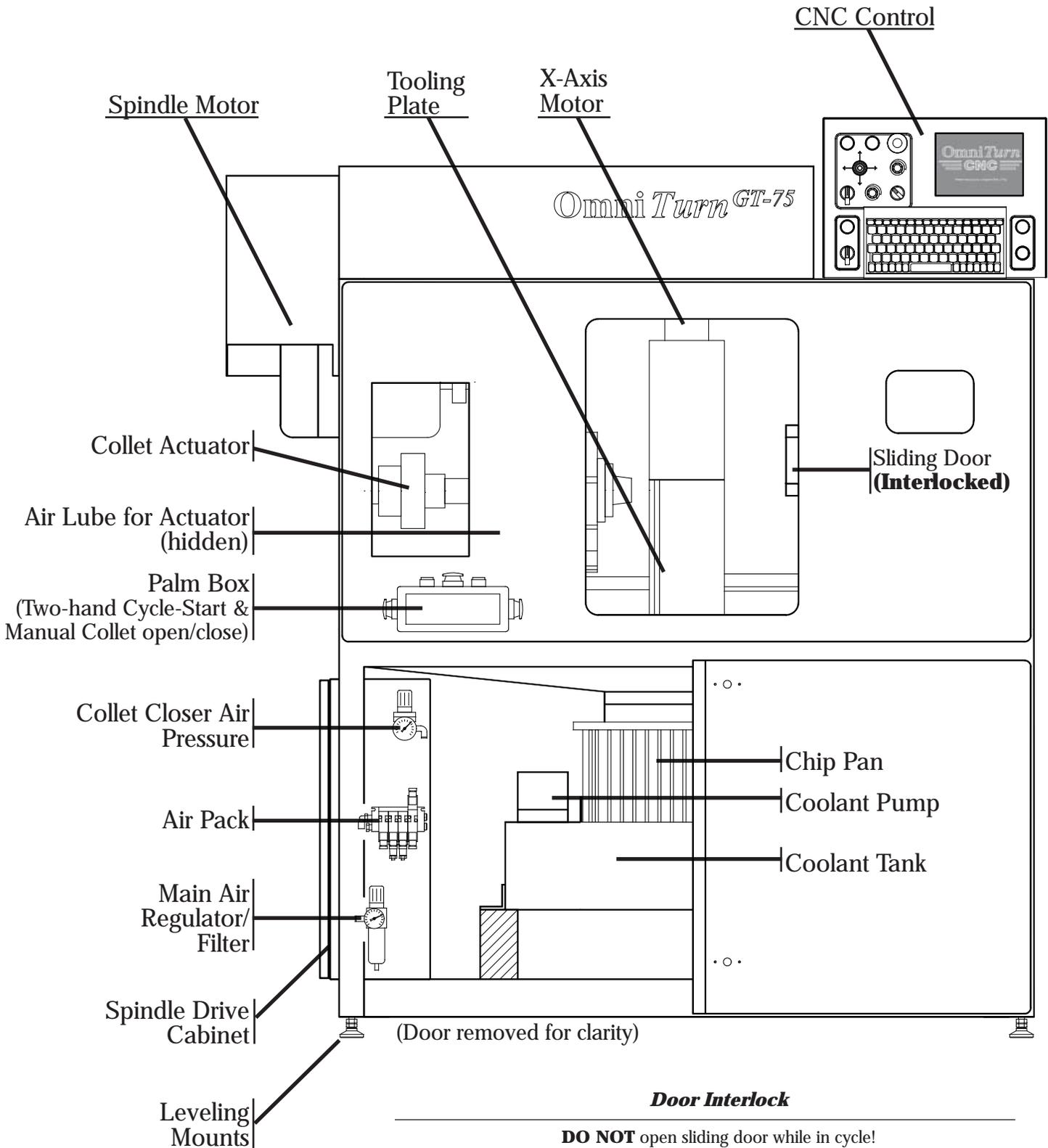
Website: www.OmniTurn.com

OmniTurn - Startup

Table of Contents

Description.....	Page
GT-75 Major Components	1.2
GT-Jr Major Components	1.3
CNC Front Panel; Knobs & Switches	1.4
CNC Front Panel; Keyboard	1.5
CNC Front Panel & Operator's Station	1.6
CNC Rear Panel	1.7
Floppy Drive and USB Stick	1.8
Startup: Applying Power	1.8
Spindle Drive Cabinet & Door Interlock Bypass	1.8
Startup: Turn on Servos	1.9
Startup: Establish Home	1.10
Establish Home, Detail	1.11
Establish Home, Screens	1.12
Establish Home, Explanation.....	1.13
Jog Mode	1.14
MDI Mode	1.16
File Handler, Introduction	1.17
File Handler, Copy Programs.....	1.18
USB Stick	1.18
Copying Programs from USB stick.....	1.18
File Handler, Delete Programs	1.19
Floppy Drive and USB Stick.....	1.19
File Handler, Create Programs.....	1.20

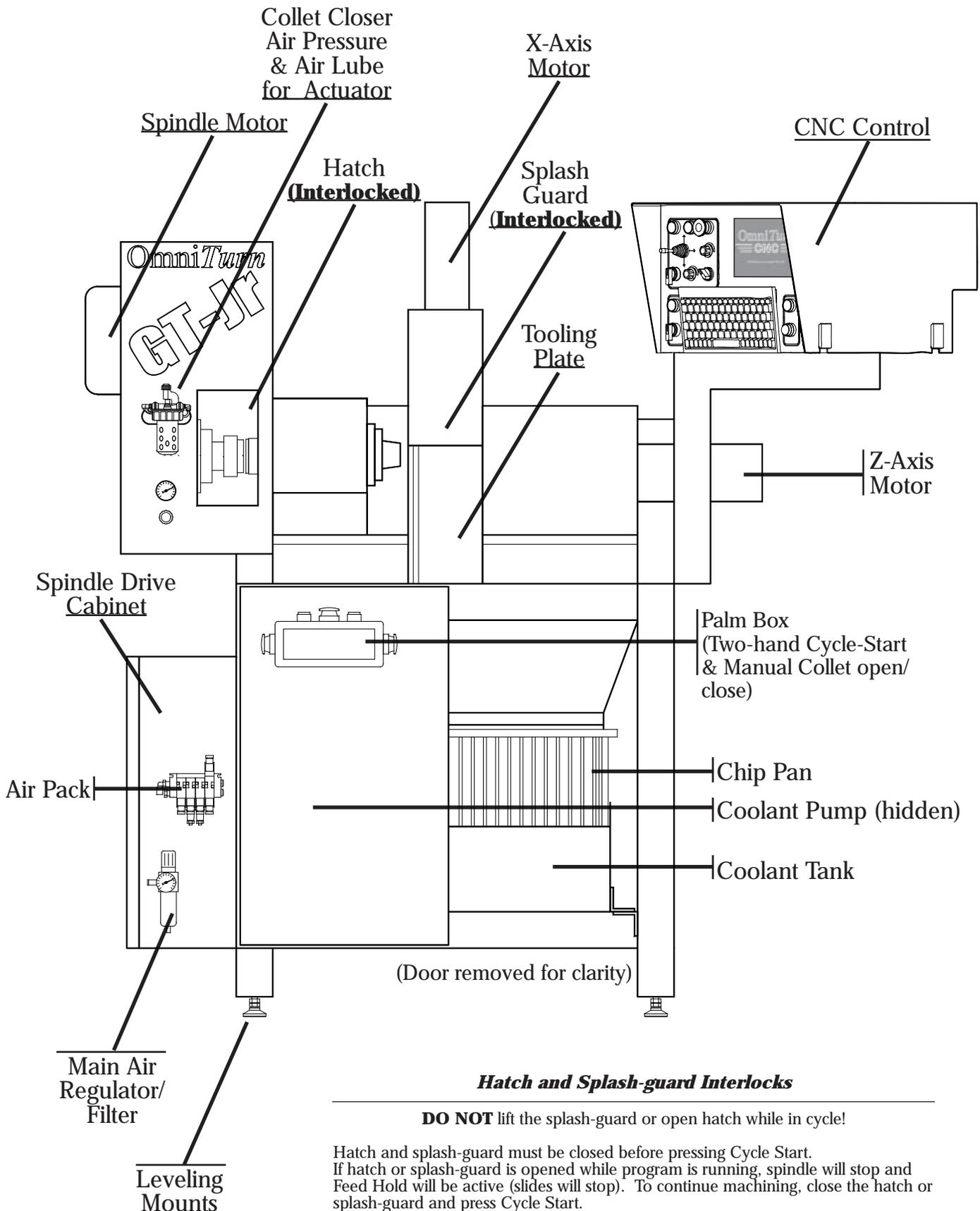
OmniTurn GT-75 Major Components



DO NOT open sliding door while in cycle!

Sliding door must be closed before pressing Cycle Start. If door is opened while program is running, spindle will stop and Feed Hold will be active (slides will stop). To continue machining, close the door and press Cycle Start.

OmniTurn GT-Jr Major Components



OmniTurn Front Panel: Knobs & Switches



Sets Control ON or OFF. Illuminates with power ON. Press ON/Press OFF

Sets Servos ON when Control is ready. Illuminates when Servos are ON. Momentary action

Halts all machine movement and re-sets all M-Functions. Press ON/Twist OFF

Manually move the slide when in Jog Mode. Continuous or incremental according to keyboard selection.

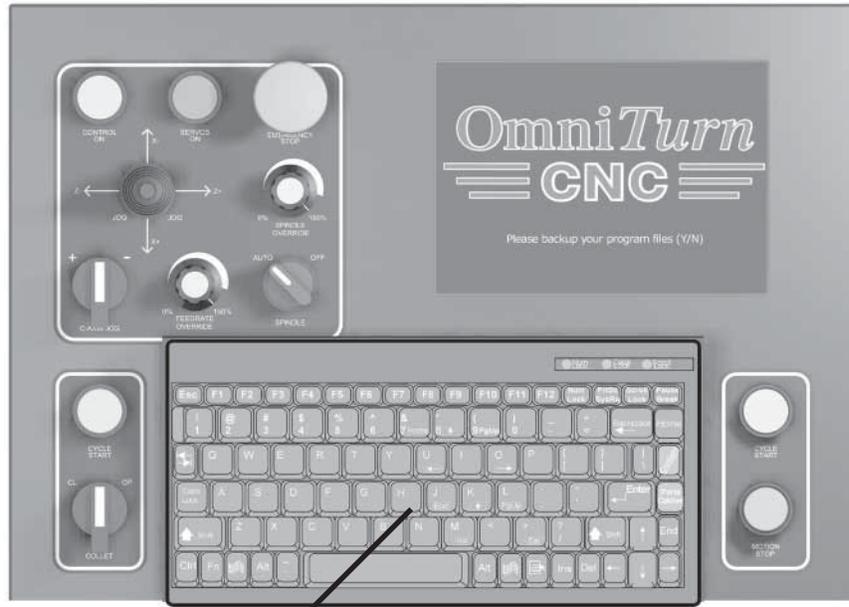
Adjusts speed of spindle from 0 to 150% of programmed rate. Spindle won't run when this is full CCW. Use M84 to lock speed at 100%; use M85 to re-enable; M30, M02 and Tool call will also re-enable.

Manually set spindle OFF and ON as required during setup. Lever action.

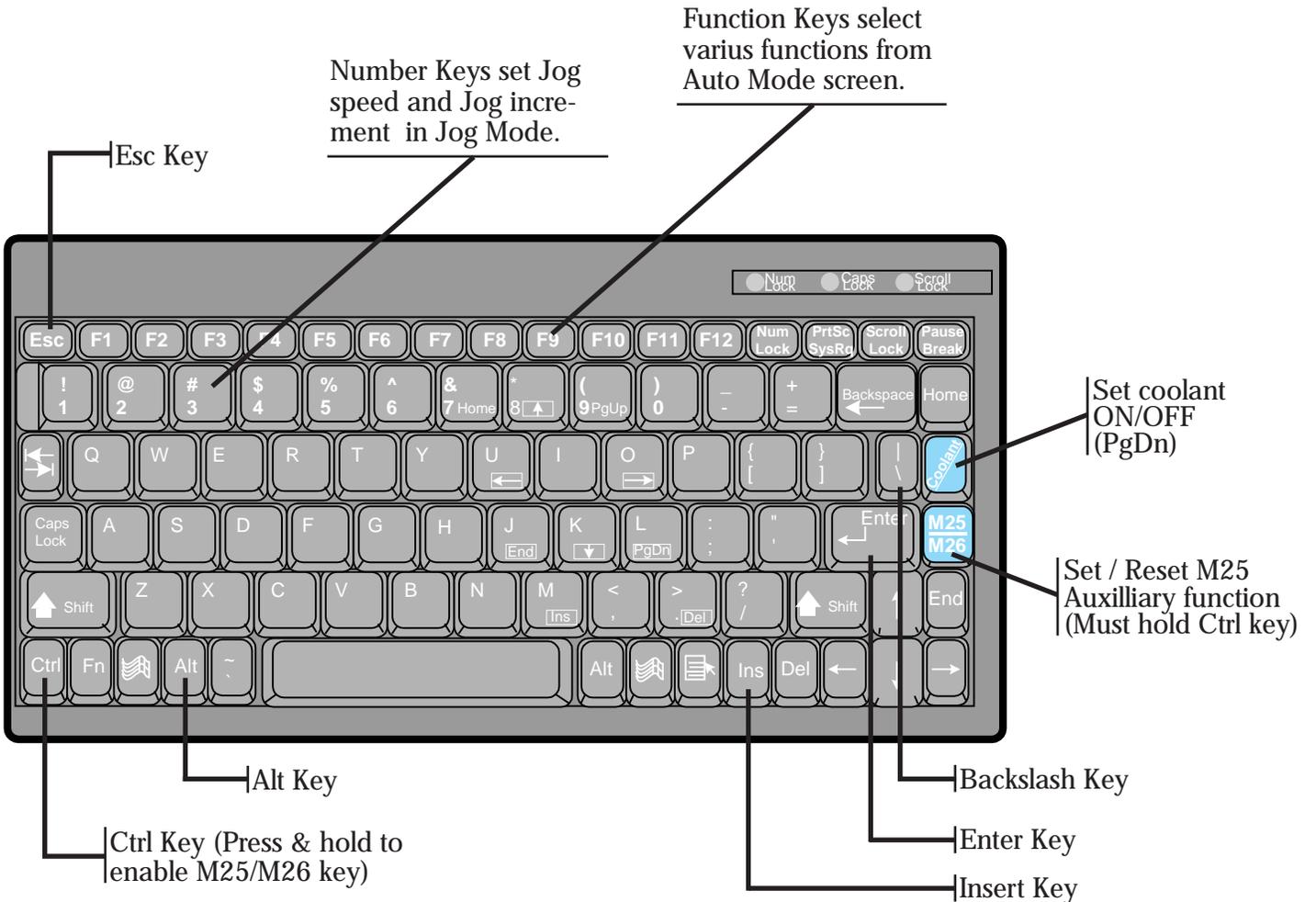
Jogs the spindle (C-Axis option only) according to setting on jog-screen (slow, medium, fast or .01° to 90° per step).

Adjusts feedrate percentage from 0 to 150% of programmed rate. NOTE: The feedrate will not exceed 300ipm. Also, feedrate defaults to 100% with Ctrl-E diagnostic. NOTE ALSO: Use M84 to lock Feedrate Override at 100%. Use M85 to re-enable. M30, M02 and Tool call will also re-enable.

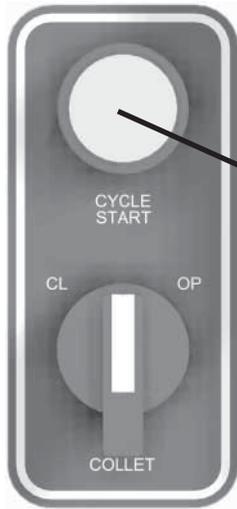
OmniTurn Front Panel: Keyboard



OmniTurn Keyboard



OmniTurn Front Panel and Operator's Station



Cycle start from Front Panel or from Operator's Station

Starts program in Auto Mode. Executes command in MDI Mode. Illuminates while program is running.

Must press both buttons at once.

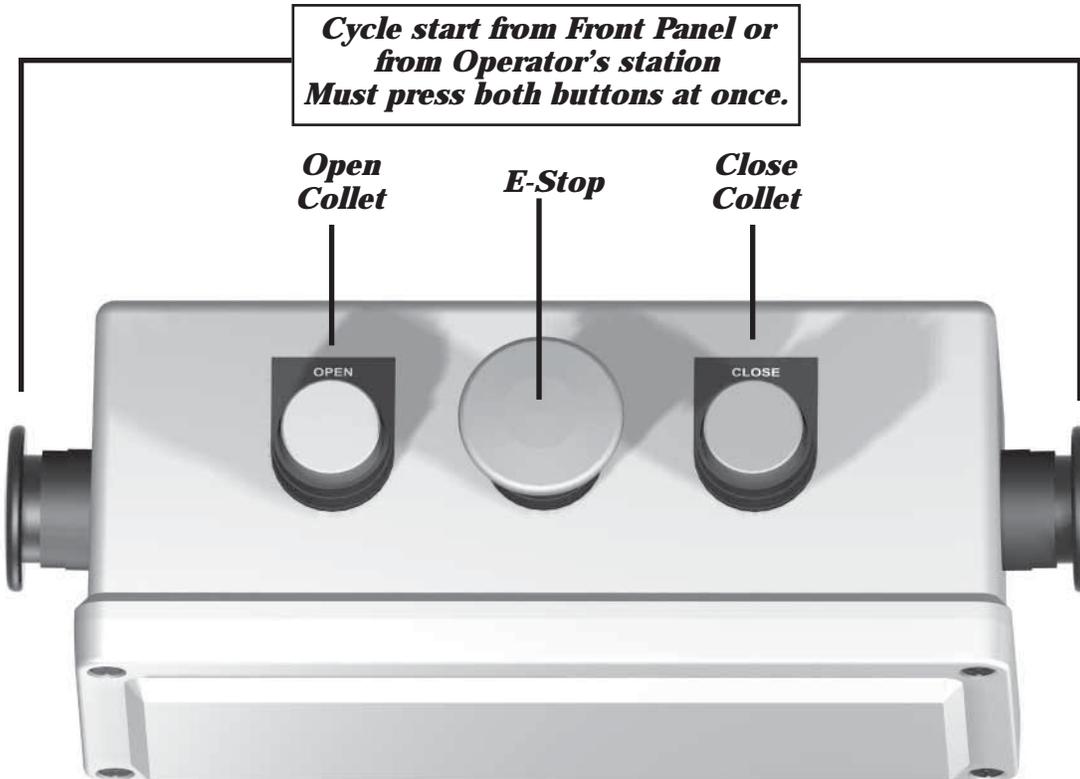


Motion Stop

Stops slide movement during program. Illuminates when active. Press Cycle Start to continue or Esc Key to stop program.

Open/Close Collet from Front Panel or from Operator's Station

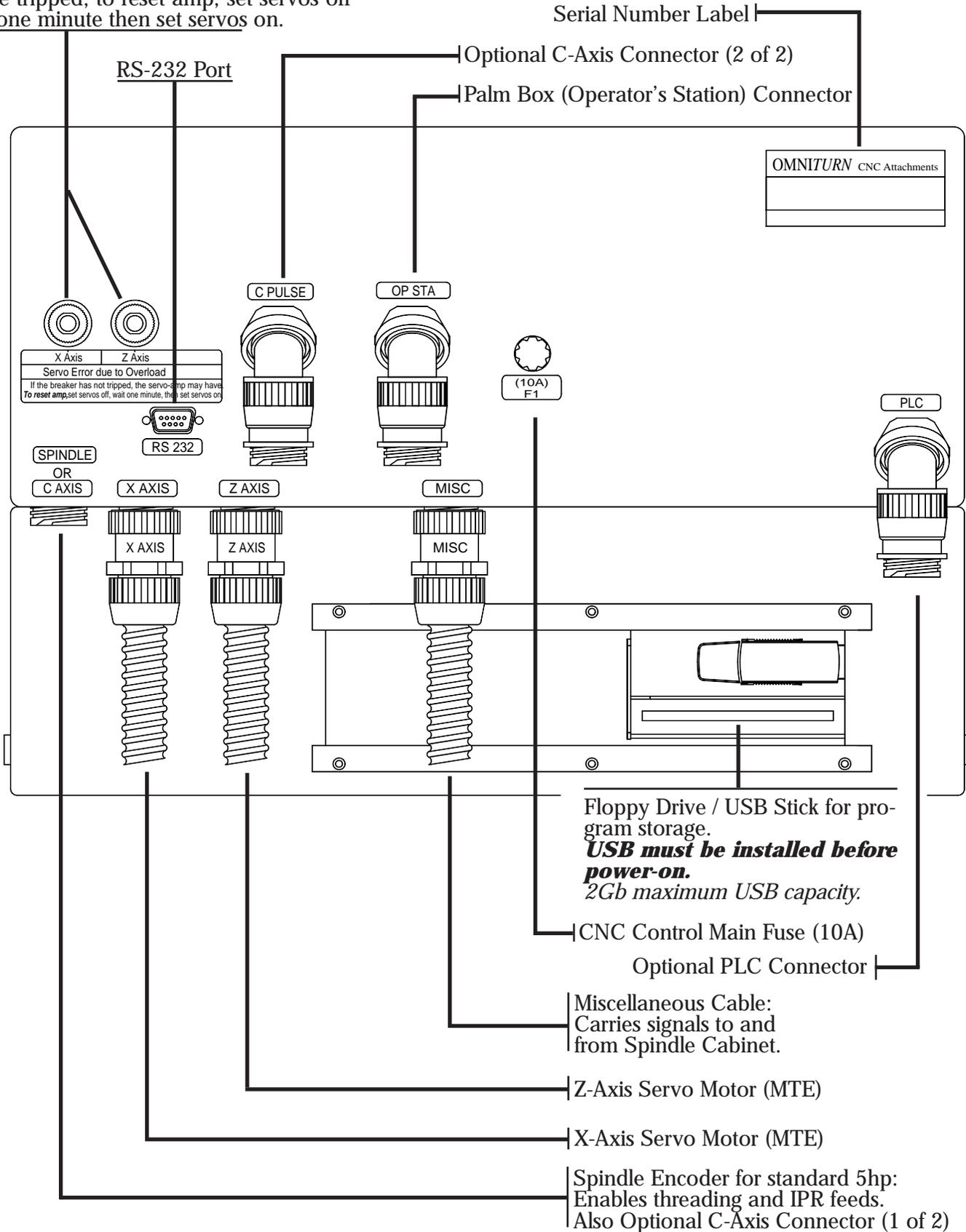
Manual control of collet only effective when spindle is stopped. M13/M12 will operate collet if spindle is running or stopped.



Operator's Station (also called "Palm Box")

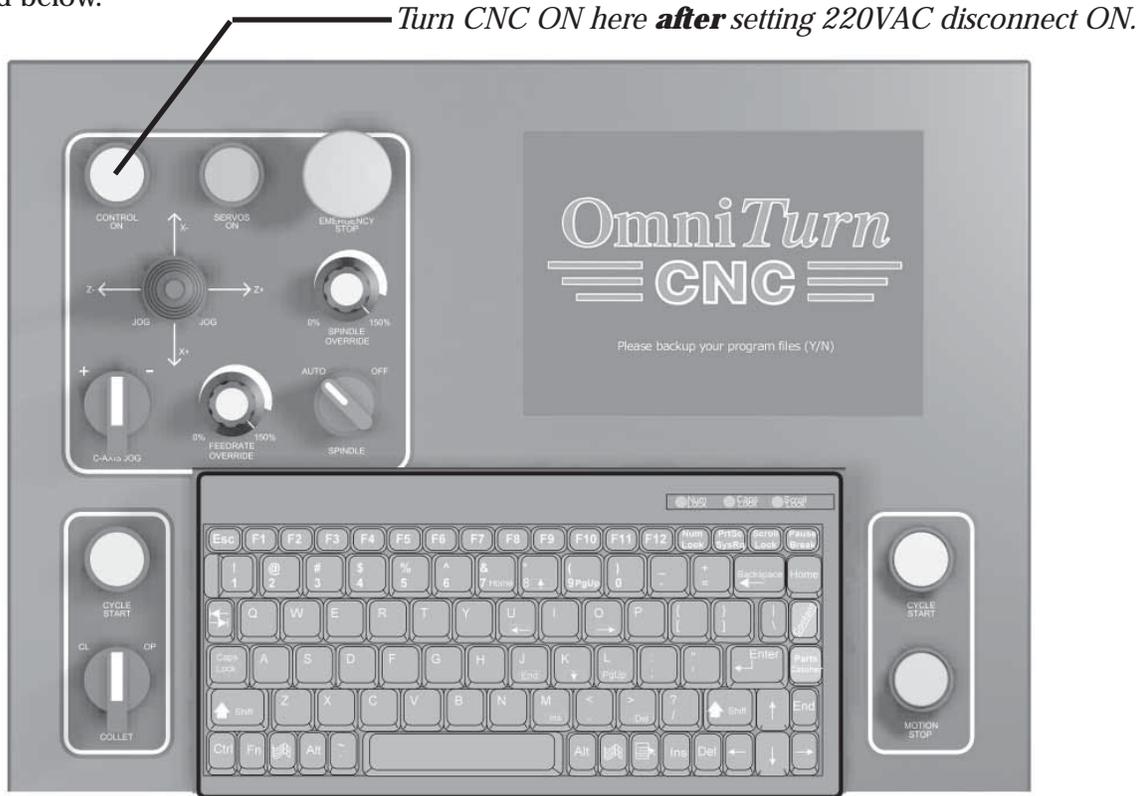
OmniTurn Rear Panel

Servo overloads. Press to reset.
 Note that servo amp can also cause overload. If breaker hasn't tripped, amp may have tripped; to reset amp, set servos off for one minute then set servos on.



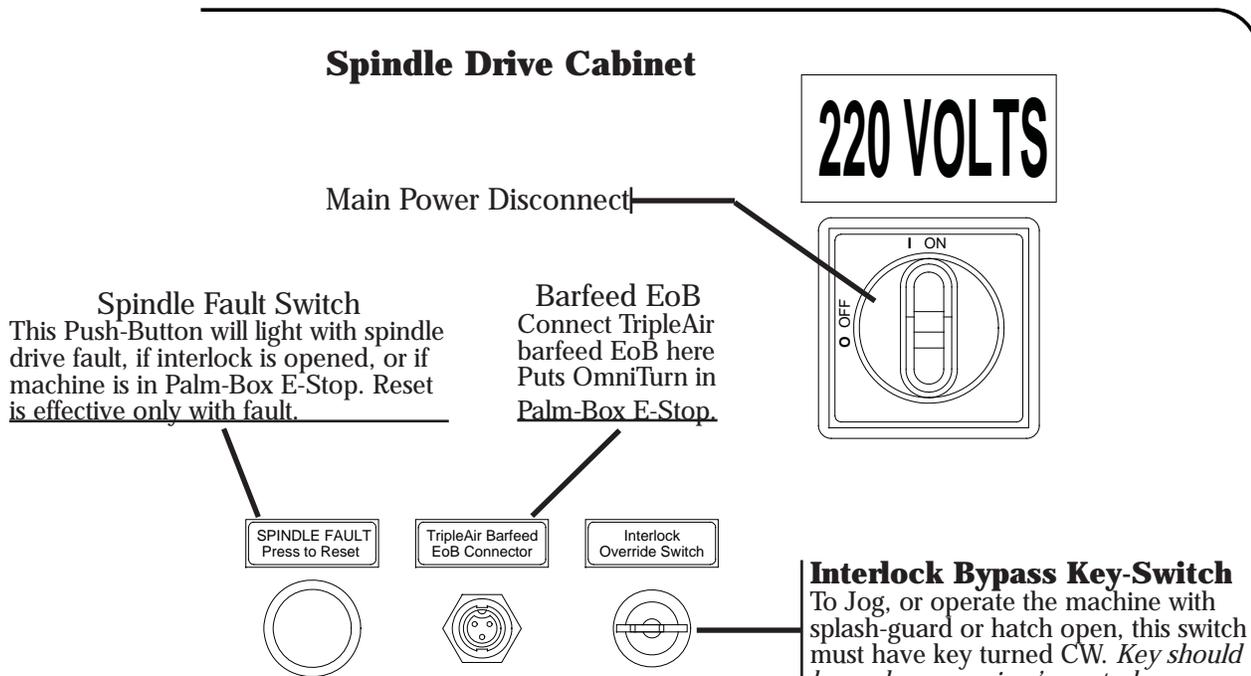
Start - Up: Apply Power; Backup Programs

After the OmniTurn has been installed and all of the cables are connected set Main Power disconnect ON. The Main Power disconnect is located on the Spindle Drive Cabinet at left-hand side of the GT-75 or GT-Jr, as illustrated below.



After boot-up, the first screen you see will ask you to back-up your program files. Always select “Y” to backup. Only those programs which have been edited will be copied to the USB stick installed at rear of CNC control. Programs will be backed up to USB memory stick (supplied) in the Programs folder. Before a new USB stick is used, you should create a Programs folder on it to maximize program storage.

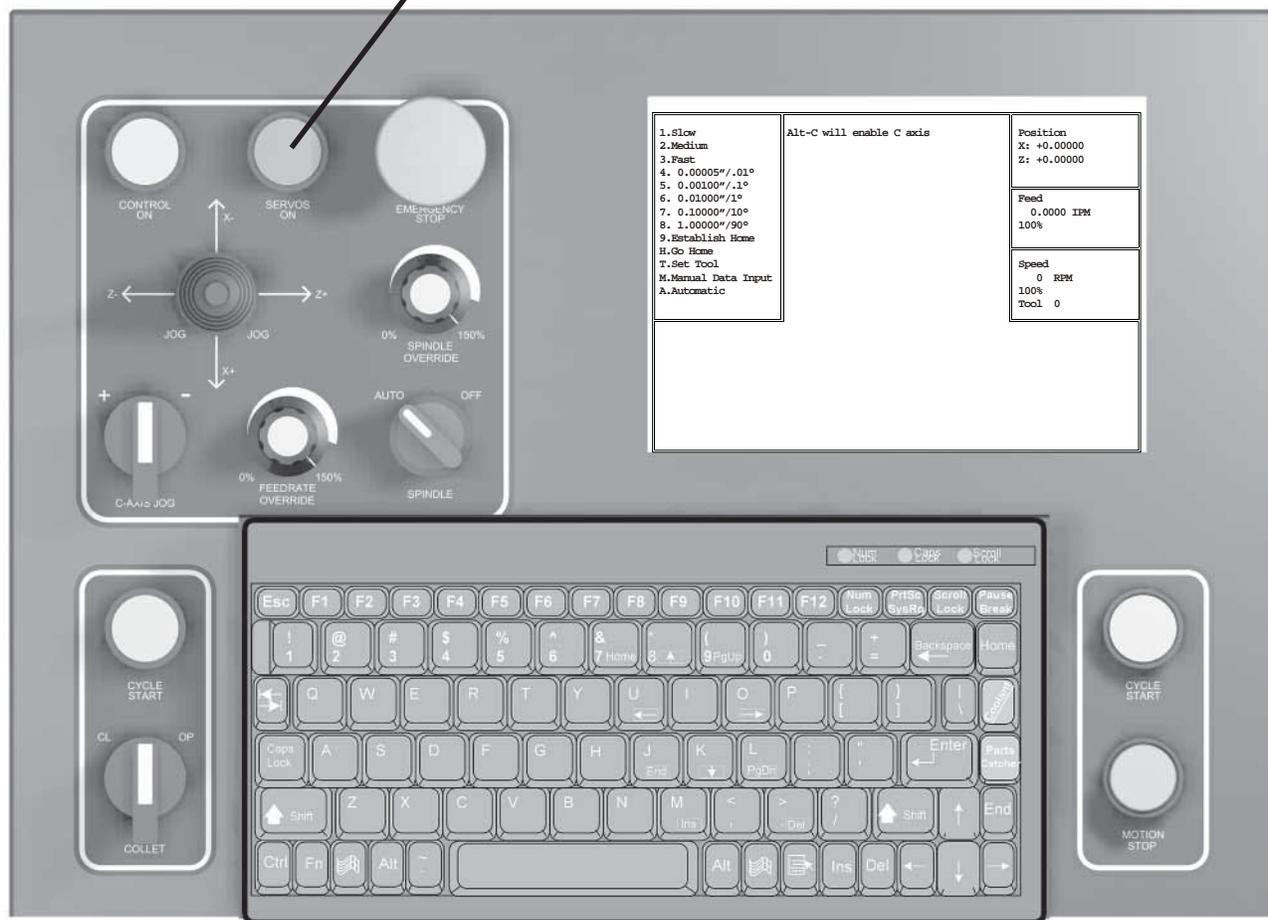
NOTE: If you have OmniTurn programs saved to floppy disk, use your desktop computer to copy them from floppy to USB stick. It may be necessary to purchase an external USB floppy drive to accomplish this.



Start - Up: Turn on Servos

After the programs have been backed up you will be prompted to turn the servos on.

Turn servos ON with blue button.

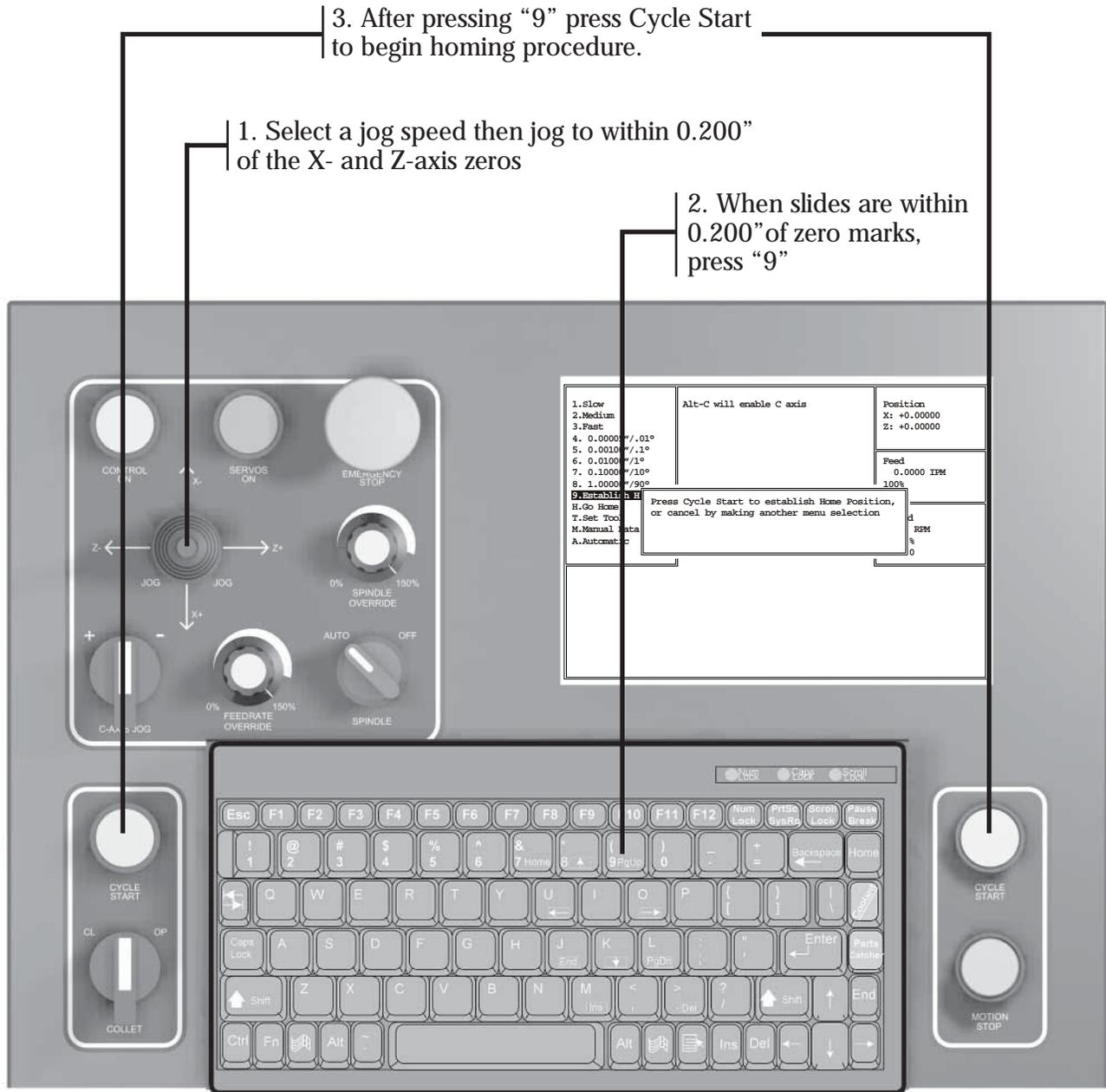


Press the blue “Servos On” pushbutton. It should illuminate and you should hear a slight hum as the axis motors engage. Press any key to clear the ‘Servos off’ prompt. The OmniTurn is now in “Jog” mode. The only other mode is “Automatic”. Manual Data Input (MDI) is part of Jog mode; a dialog box opens to allow program commands to be manually entered.

<pre> 1.Slow 2.Medium 3.Fast 4. 0.00005"/.01° 5. 0.00100"/.1° 6. 0.01000"/1° 7. 0.10000"/10° 8. 1.00000"/90° 9.Establish Home H.Go Home T.Set Tool M.Manual Data Input A.Automatic </pre>	<pre> Alt-C will enable C axis Servos off or Ext. Halt detected- Clear fault or turn servos on, then Press any key to proceed </pre>	<pre> Position X: +0.00000 Z: +0.00000 Feed 0.0000 IPM 100% Speed 0 RPM 100% Tool 0 </pre>
---	---	--

Prompt area indicates that servos are off. This screen also appears when servo error has occurred. Set servos on, then press a key to clear prompt and continue.

Start - Up: Establish Home



Establish "Home"

After the servos are turned on, press any key to clear the prompt, then press "2" (Medium Jog) or "3" (Fast Jog) on the Keyboard to enable the Jog stick. Jog the slide away from the X & Z axis zeros, but not more than 0.200" away. Note that the control will not allow you to leave the Jog mode until the homing procedure has been completed.

Note that the slide will not jog if a door, hatch or splash-guard is open. If it is necessary to operate the machine with door, hatch or splash-guard open, there is an interlock by-pass keyswitch on the spindle drive cabinet door. Only personnel specially trained by your company should have access to the interlock by-pass key.

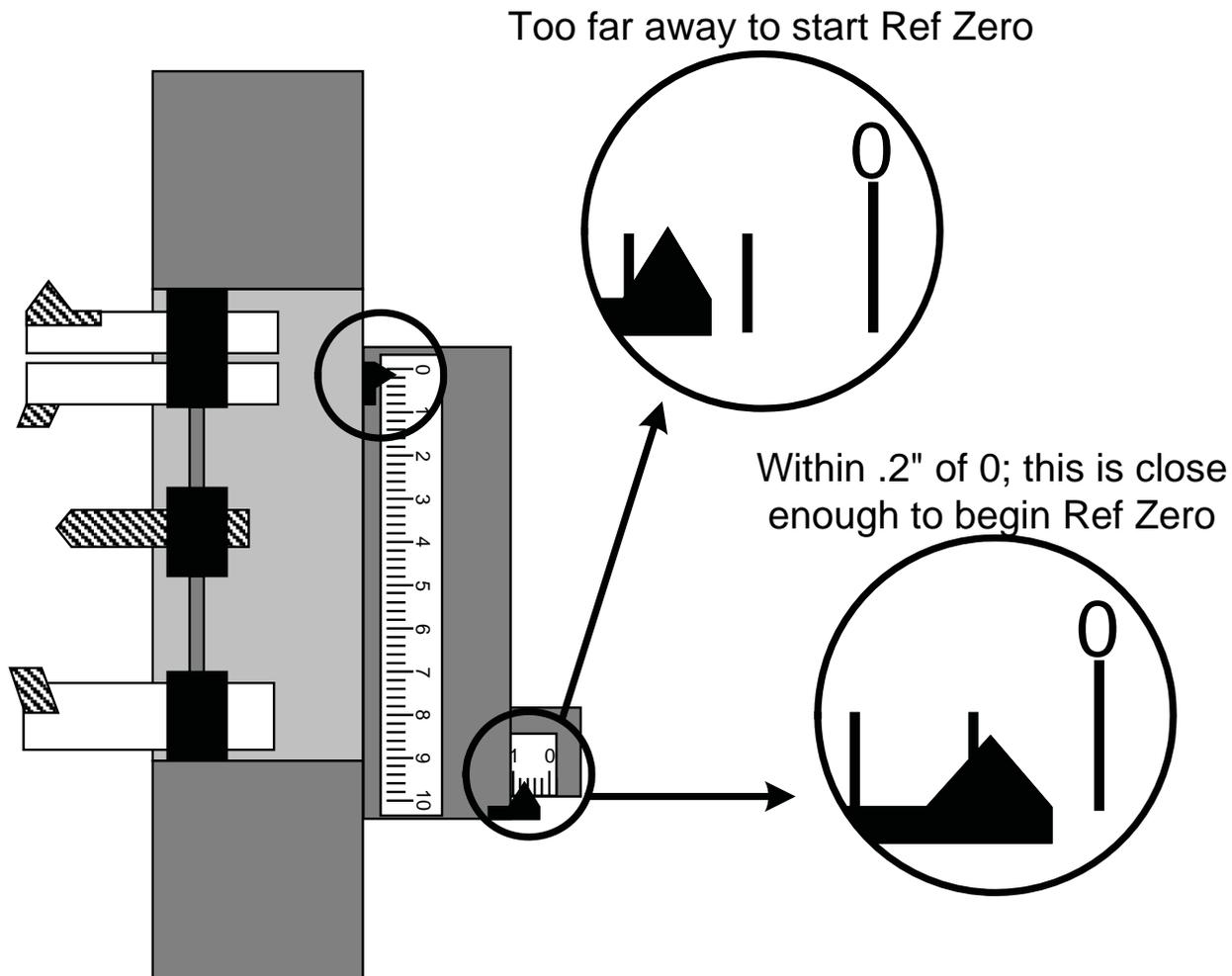
When the slides are within 0.200" of the zeros, press "9" (Establish Home) on the keyboard, then press Cycle Start to start the homing procedure. The slide will move slowly in +Z, then in -X, seeking the home marks on the motor encoders. When homing is complete, X & Z position indicators will read 0.00000.

The following three pages describe the homing procedure in detail.

Establish Home: Detail

Press “2” (Medium Jog) or “3” (Fast Jog) on the Keyboard, and use the Jog Stick to move slide to within 0.200” from Zero in X and Z on scales, as illustrated below. The X axis should be on the plus side of zero, and the Z axis should be on the minus side.

If you are too far away from zero you will be off by .200” when the Ref Zero is found because the axis has a “home” every revolution.



When the slide has been jogged to within 0.200” of zero for **both** axes, press the “9” key on the keyboard, then press Cycle Start pushbutton to complete the homing sequence.

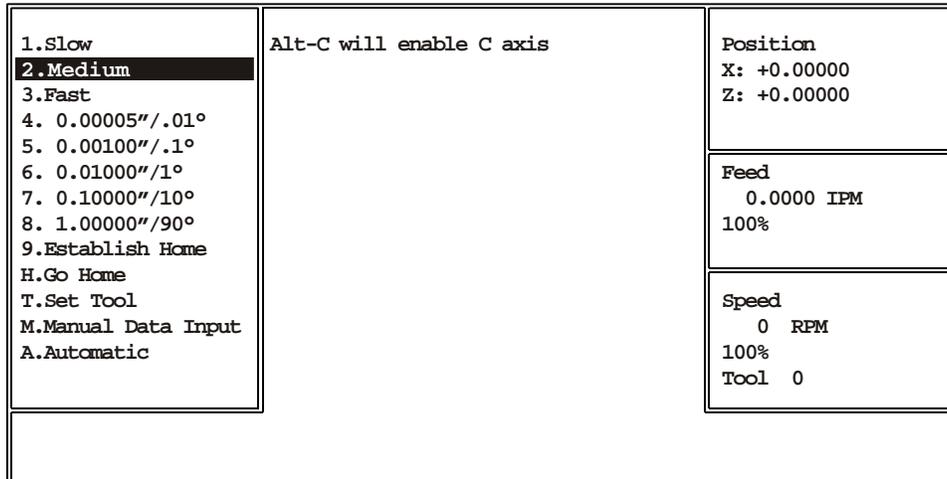
The control will first move the Z and then X axis.

After the procedure, check that the location for zero is where you want it. If you have missed the zero, press the “2” key (Med Jog) then jog the slide to the correct location, then press “9” then Cycle Start again.

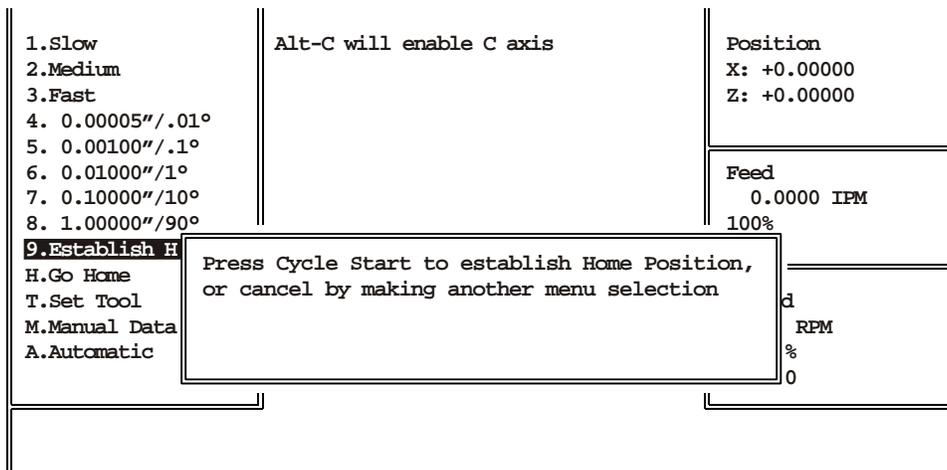
Verify that the position display at the top of the screen now reads zero for both axes.

See following page for the screens which display the process.

Establish Home: Screens



It's a good practice to send the slide "home" before shutting down the OmniTurn. (To do this, press "H", then "Z", then "X") This makes "Establish Home" very quick when machine is turned on: set Servos ON, then press any key, then press "2" for Medium Jog, then move the slide off the zeros with a quick "left" then "down" on the Jog stick. If the slide is not at the zeros, press "3" for Fast Jog to get close, then "2" to put the pointers within 0.200" of zero. The jog screen above is shown with Jog Medium selected.



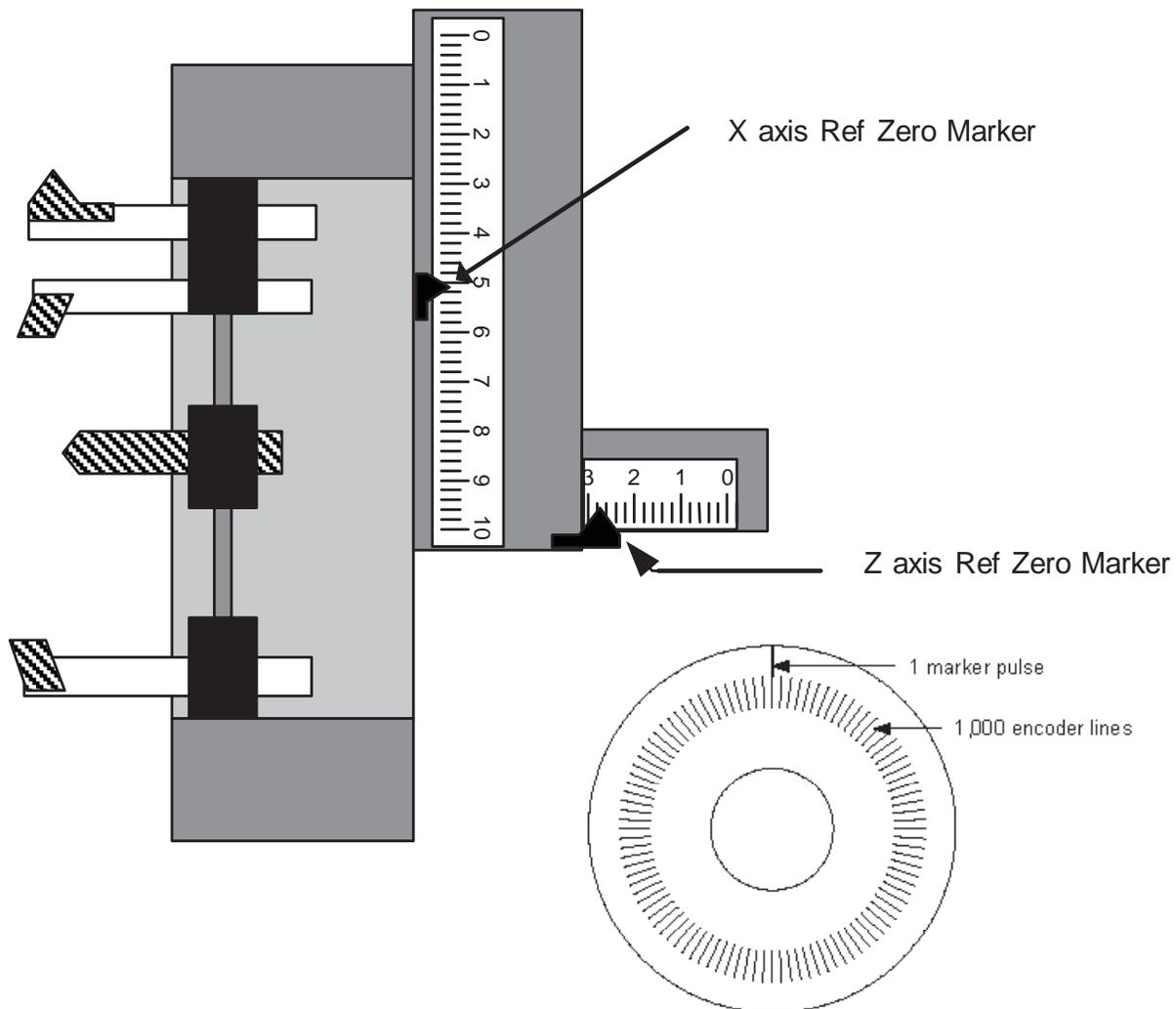
When the pointers are within 0.200" of the zeros on the scales, press the "9" key, then Cycle Start, to establish Home. If "9" is ever pressed by mistake, press the "Esc" key to cancel the selection. When Cycle Start is pressed the slide will move to right in Z-axis, then up in X-axis.

The jog screen above is shown with Establish Home selected

After Home has been established, you can leave the Jog Mode by pressing the "A" key on the keyboard to go to the Auto Mode, or you can stay in the Jog Mode and establish the tool offsets. Pressing the "M" key enables MDI (Manual Data Input), which allows M-functions and simple moves. On the following pages the Jog Mode is discussed in detail.

Establish Home: Explanation

On each axis is a scale that is marked with .2" graduations. These scales are used to establish HOME. Home is a reference point that can be easily found and repeated. The .2" scales are just an aid to find this home point.



How HOME is established

The HOME location will repeat itself from start-up to start-up within .00005". So when you start up the machine from one day to the next it will repeat itself very closely. The point is established by the following:

On the end of the servo motor that drives the axis is an encoder that tells the control how far the slide has traveled. This encoder follows the slide travel incrementally. The encoder does not know exactly where the slide is, only how far it has traveled. The encoder works by counting fine lines printed on a disk.

The drawing above is a sketch of this type of rotary encoder. The inside ring of lines is counted to maintain location. The outside line is used to find a definite location. The encoder will "see" this line once every revolution of the ball screw used to position the slide. The ball screw used on the OmniTurn moves the slide .200" / revolution. So if the slide is positioned within .200" of what we want to call HOME and then tell the control to slowly rotate the screw while looking for the single pulse we can establish this as an easily repeatable location. This is how HOME is established. For consistency and safety, ALWAYS use the zeros on the scales for HOME.

Jog Mode

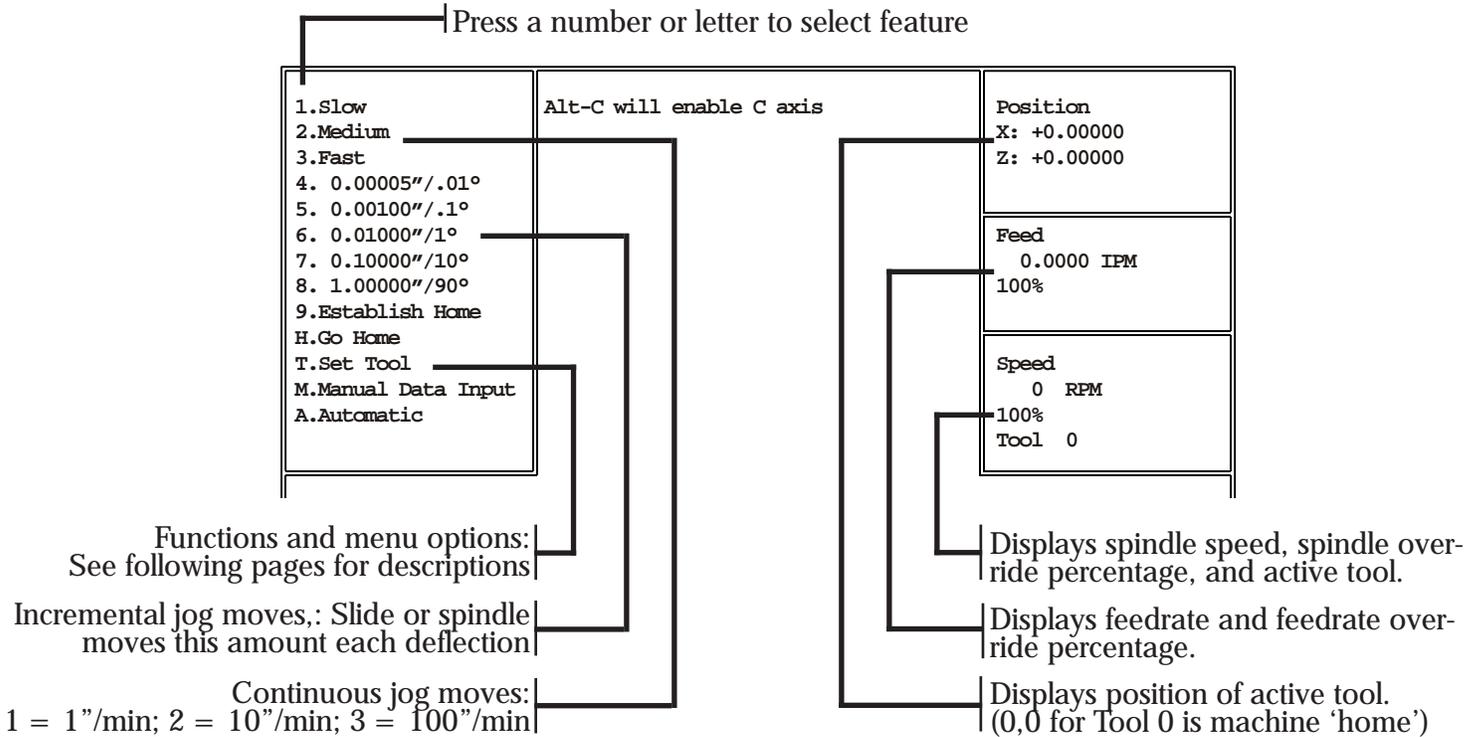
The purpose of the jog mode is to:

- Establish the "HOME" position. This is always done after starting up the control.
- Set tool offsets
- Move the slide manually
- Bore soft jaws or collets
- Orient the C-Axis (optional) an/or establish arbitrary C-Axis home

To enter the jog mode:

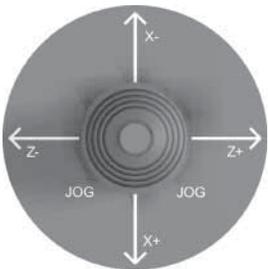
Press the "J" key on the keyboard from Auto Mode screen.

At initial turn-on, Jog Mode is entered after turning on servos then pressing any key.



Jogging the Slide

The slide can be manually positioned with the "Jog stick" actuator on the operator's panel. Deflecting this actuator in the four directions available produces the following motions:



- | | |
|-------|--|
| Up | X axis minus = up on GT-75 & Jr; away from you on attachment |
| Down | X axis plus = down on GT-75 & Jr; toward you on attachment |
| Left | Z axis minus = toward spindle |
| Right | Z axis plus = away from spindle |

Jogging the Spindle (opt. C-Axis only)

To enable the C-Axis, press Alt-C (press and hold the Alt key, then press the "C" key); the spindle will rotate to its home position and lock. The spindle can be manually positioned with the C-Axis Jog control. Slow, medium and fast rotation is accomplished by choosing 1, 2 or 3; incremental rotation is accomplished by choosing 4 - 8.

To set a custom C-Axis home, position the spindle as desired, then press Alt-H; the spindle will orient to this position $\pm .02^\circ$ whenever C-Axis is enabled, manually or by M19. Deflecting this actuator "+" rotates the spindle in M03 direction; "-" rotates M04.



Jog Mode

To send the slide HOME

Pressing the "H" key will bring up the message "Press X to home X, Z to home Z...". It's a good practice to press Z first, in order to move the tools back out of the way. Then press X and the slide will home in the X axis. Note that you don't need to home both axes; pressing any other key will cancel the function. This function is used to check that the Home you have set is still where you think it should be. This function will not work until you have first gone through the "Establish Home" procedure after start-up.

1.Slow 2.Medium 3.Fast 4. 0.00005"/.01° 5. 0.00100"/.1° 6. 0.01000"/1° 7. 0.10000"/10° 8. 1.00000"/90° 9.Establish H.Go Home T.Set Tool M.Manual Dat A.Automatic	Alt-C will enable C axis	Position X: +0.00000 Z: +0.00000
		Feed 0.0000 IPM 100%
	Press X to home X, Z to home Z, or cancel by making another menu selection	RPM Tool 0

To set Tool Offsets or to Select Tool

To set tool offsets, goto Section 3.

To select a previously set tool, press the "T" key, enter the tool number (1 - 32), then press Enter. You will be prompted to press X or Z to set tool position; *press Esc instead*. The Position counter will now show the location of the tool in diameter mode. Use this technique to quickly but accurately turn or bore a single part. By selecting jog speed and using the Feed Rate override pot you can easily set any feed rate.

To start the spindle, press the "M" key to open MDI dialog box, type m03 or m04 and s-number for rpm, press Enter then Cycle Start. Selecting jog speed (or any key) closes MDI dialog box.

For more MDI details see next page.

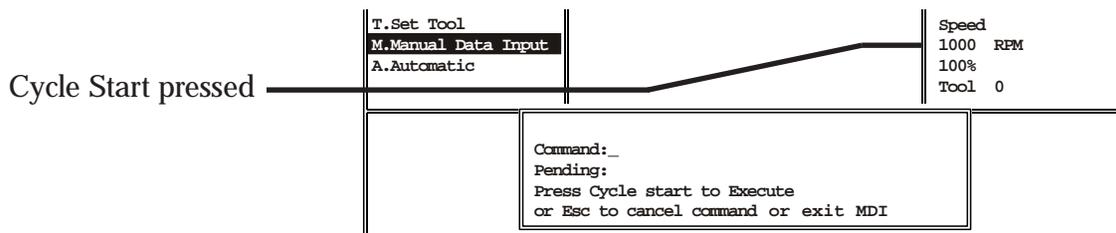
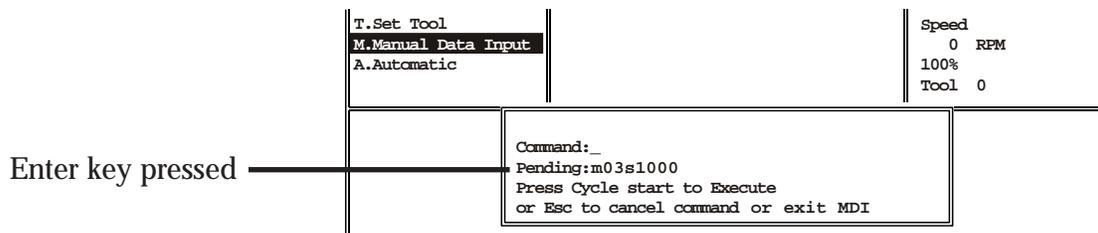
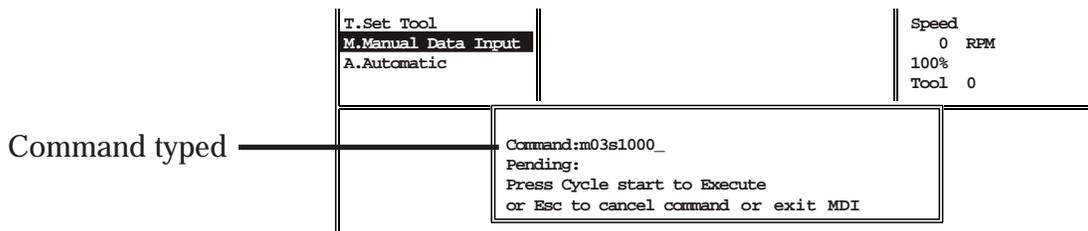
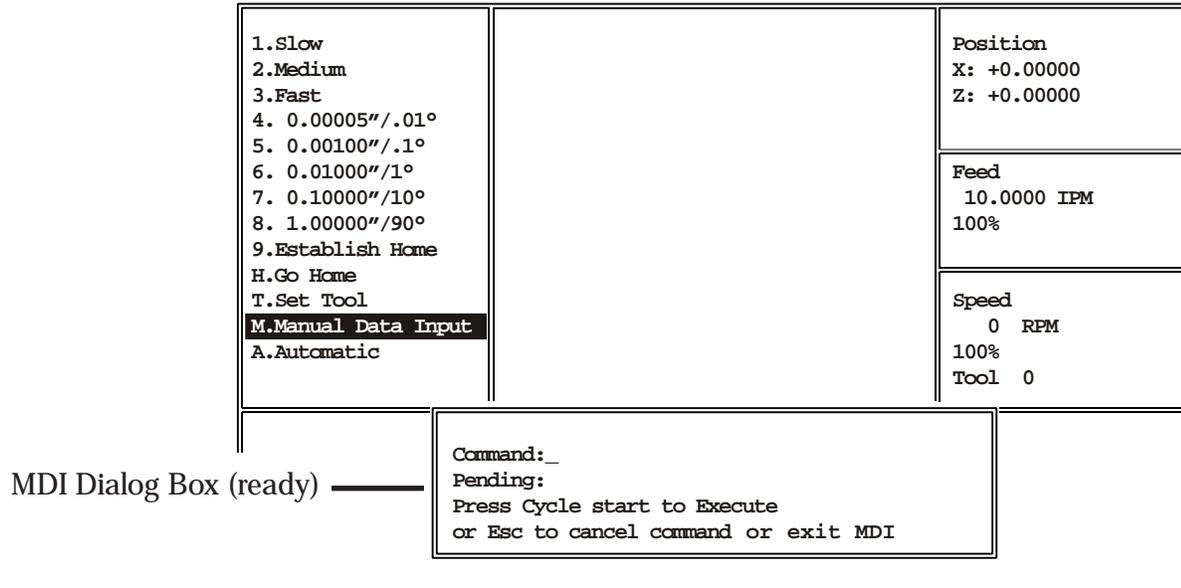
1.Slow 2.Medium 3.Fast 4. 0.00005"/.01° 5. 0.00100"/.1° 6. 0.01000"/1° 7. 0.10000"/10° 8. 1.00000"/90° 9.Establish H.Go Home T.Set Tool M.Manual Data I A.Automatic	Alt-C will enable C axis	Position X: +0.00000 Z: +0.00000
		Feed 0.0000 IPM 100%
	Tool Number:_ Press Esc to cancel tool setting	d RPM Tool 0

MDI Mode

Manual Data Input (MDI) Dialog Box

Pressing "M" key opens the MDI Dialog Box. MDI allows basic operations such as exercising M-functions and simple positioning moves. Canned cycles such as threading and drill cycles are not supported in MDI, but feed rates, spindle speeds and tool numbers can be specified.

To use this feature, type the command on the 'Command' line, then press Enter to move it to the 'Pending' line. then press Cycle Start to execute the command. Press Esc to cancel anytime before Cycle Start. After Cycle Start is pressed, the status will show in the applicable 'Position', 'Feed' or 'Speed' box(s).



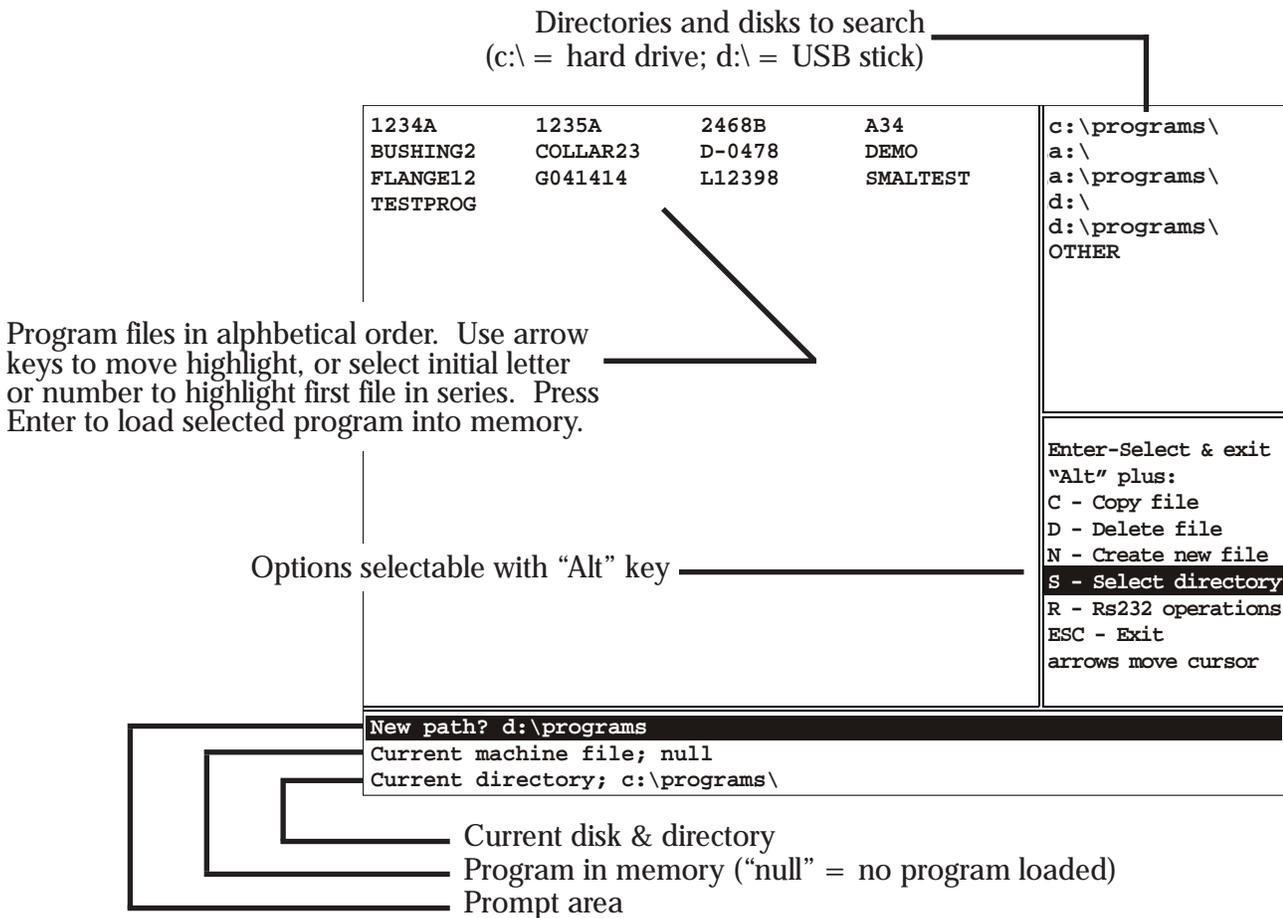
File Handler

Automatic Mode

Pressing the "A" key switches the OmniTurn to Automatic mode. If no program has been selected, a file handler screen will come up and you must select an existing program or create a new program to continue. Pressing Esc will **not** return you to Jog mode; *you must select or create a program to continue.* Automatic Mode (and the program editor) is covered in Section Five; the file handler is covered below.

File Handler

The file handler is invoked from Jog mode only if Auto mode is selected when no program is loaded. The file handler can be invoked from Auto mode three ways: select File Operations (F8); edit an existing program (F3), or load a new program (F5). (See Section Five for Automatic mode details.) The file handler allows you to select, copy or delete programs, or to create new programs.



Options Selectable with "Alt" key

Alt-C = Copy file: You will be prompted for a location to copy to. This option is most useful for copying programs written at your desktop. You must first use Alt-S to select directory. See next page for details.

Alt-D = Delete file: You will be prompted "Delete *filename* Y/N" Press N if you change your mind.

Alt-N = Create New file: You will be prompted "Name of file to create?" Type the name then press Enter. The OmniTurn editor will start. When you leave the editor, you will have opportunity to load this program into memory (make it the current machine file). The editor is described in Section Five.

Alt-S = Select directory: Use the up/dn arrow keys to highlight desired directory. Press Enter to display program files in that directory. Pressing the "D" key will remove the highlighted directory from the list. Highlight "OTHER" to choose a directory not on the list.

Alt-R = RS232 operations: This option allows you to send/recieve programs via the serial port.

File Handler: Copy Programs

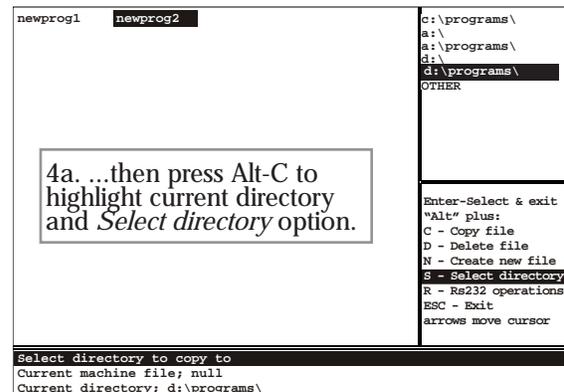
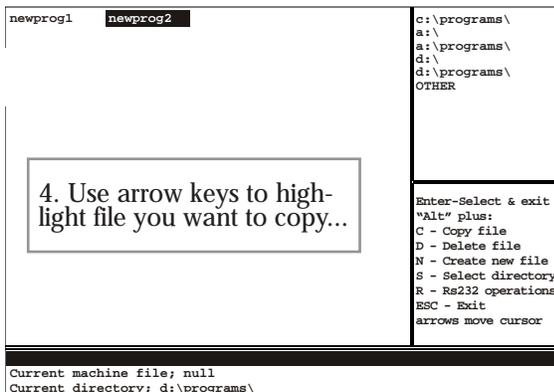
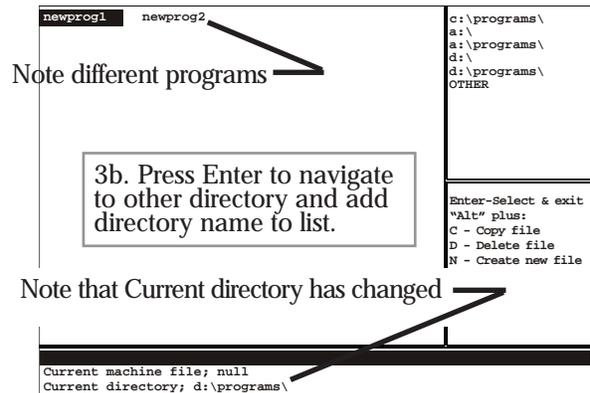
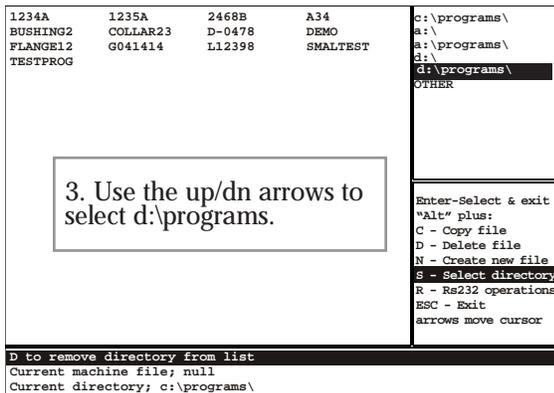
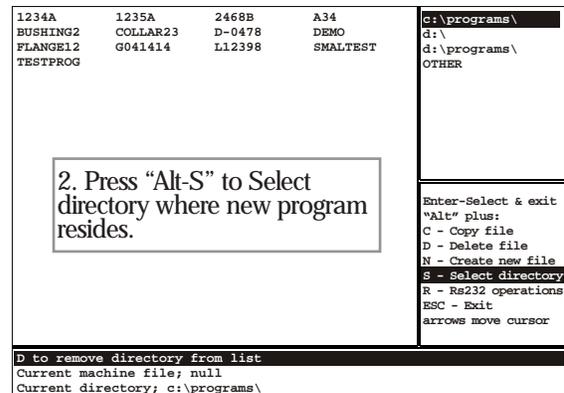
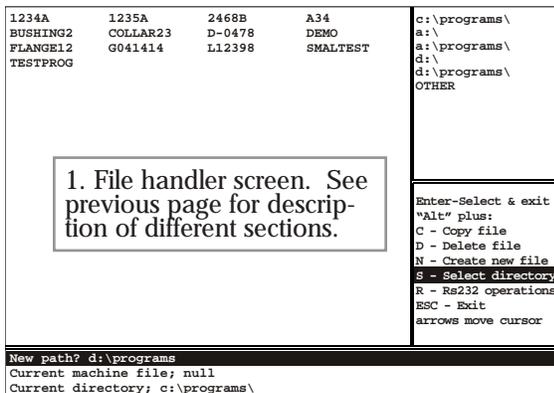
Alt-C: To copy a program to the hard drive from floppy drive or USB stick*

The steps that follow assume that no program is loaded into memory; home has just been established and the program to load is on a USB stick in 'programs' folder.

1. Select "A" from Jog mode to open file handler screen. NOTE: If program has already been loaded, select File Ops (F8) from Auto mode. "Current machine file;" will display the program name instead of 'null'.
2. Press Alt-S to Select directory where new program resides.
3. Use the up/dn arrows to select the directory. USB stick is d:\ If the desired directory is not displayed, select OTHER and type the full 'path', including the back slash symbol (\). See keyboard illustration on page 1.5. Note that if your programs are in a sub-directory (folder) on the USB, you must type the name of the folder as well - d:\programs, in this example. Press Enter.
4. Use arrow keys to highlight file you want to copy, then press Alt-C.
5. Use up/dn arrow keys to highlight c:\programs directory, then press Enter; prompt will indicate "Copying d:\programs\filename to c:\programs\filename".
6. Press Alt-S again, to return to c:\programs directory.

*USB stick 2Gb maximum

Screens below and on next page illustrate these steps.



File Handler: Delete Programs

Alt-C: To copy a program *to* the hard drive *from* USB stick, *continued*

```
newprog1  newprog2
c:\programs\
a:\
a:\programs\
d:\
d:\programs\
OTHER

Enter-Select & exit
"Alt" plus:
C - Copy file
D - Delete file
N - Create new file
S - Select directory
R - Rs232 operations
ESC - Exit
arrows move cursor

Copying d:\programs\newprog2 to c:\programs\newprog2
Current machine file; null
Current directory; d:\programs\
```

NOTE
USB memory stick will not be 'found' by CNC unless installed before power-on.

Computer must boot with USB stick installed; after boot-up, memory stick may be removed and replaced without issue.

To copy a program *from* the hard drive *to* USB stick:

The process is simply the reverse of what has just been illustrated. Again, the steps assume that no program is loaded into memory; home has just been established and the program will be copied to a USB stick in 'programs' folder, which is not in current directory list.

1. Select "A" from Jog mode to open file handler screen. NOTE: If program has already been loaded, select File Ops (F8) from Auto mode. "Current machine file;" will display the program name instead of 'null'.
2. Use arrow keys to highlight file you want to copy, then press "Alt-C".
3. Use the up/dn arrows to select the directory of the USB stick (d:\) you are copying to. If the desired directory is not displayed, select OTHER and type the full 'path', including the back slash symbol (\). See keyboard illustration on page 1.5. Note that if you want to copy the program to a sub-directory (folder) on the USB, you must type the name of the folder as well - d:\programs, in this example.
4. Press Enter; prompt will indicate "Copying c:\programs\filename to d:\programs\filename".

To copy all files from floppy to hard drive:

If you are replacing an earlier version OmniTurn CNC with the new G4 CNC you may have many part programs on one floppy disk. The File Handler has no way to copy them all with one command, but you may 'shell out' to DOS and copy them all as follows:

- 1 - Edit any program. Press F1 to go to 'help' screen, then press F4 to shell to DOS.
- 2 - Put floppy with part programs in drive a:
- 3 - Type copy a:*.* c:\programs (note space between second * and c:) then press Enter; you will see a list of program names scrolling on screen; followed by prompt saying how many file(s) were copied.
- 4 - Type exit then press enter to return to the editor

Alt-D: To delete a program:

1. Use arrow keys to highlight file you want to delete, then press Alt-D; prompt will indicate "Delete c:\programs\filename? Y/N;" NOTE: To delete file from floppy or USB stick, use Alt-S to select dirctory, then highlight the file, then press Alt-D.
2. Press Y to delete, N if you change your mind.

```
1234A    1235A    2468B    A34
BUSHING2  COLLAR23  D-0478    DEMO
FLANGE12  G041414  L12398    SMALTEST
TESTPROG

c:\programs\
a:\
a:\programs\
d:\
d:\programs\
OTHER

Enter-Select & exit
"Alt" plus:
C - Copy file
D - Delete file
N - Create new file
S - Select directory
R - Rs232 operations
ESC - Exit
arrows move cursor

Delete c:\programs\collar23? Y/N;
Current machine file; null
Current directory; c:\programs\
```

File Handler: Create Program

Alt-N: To create a new program:

Selecting “Create new file” (Alt-N) opens the OmniTurn program editor. The editor is described in Section Five, Automatic Mode. The following steps describe entering and exiting the editor.

1. Press Alt-N. You will be prompted “Name of file to create?”
2. Type the name then press Enter. Note: the name of the program can be any combination of letters and numbers from one to eight characters long. If a program with that name exists, you will not be allowed to create new program; you must choose another name or delete the existing program (*see Alt-D on previous page*).
3. The OmniTurn editor will start. Using the editor is described in Section Five, Automatic Mode.
4. When you leave the editor, you will have opportunity to load this program into memory (make it the current machine file). If you choose Y, the “Current machine file;” prompt will change to this program. If you choose N, the prompt will not change. In either case you will remain in the file handler screen.

```
1234A      1235A      2468B      A34
BUSHING2   COLLAR23   D-0478   DEMO
FLANGE12   G041414    L12398   SMALTEST
TESTPROG

c:\programs\
a:\
a:\programs\
d:\
d:\programs\
OTHER

Enter-Select & exit
"Alt" plus:
C - Copy file
D - Delete file
N - Create new file
S - Select directory
R - Rs232 operations
ESC - Exit
arrows move cursor

Name of file to create?
Current machine file; null
Current directory; c:\programs\
```

1. Press Alt-N to create new program; you will be prompted for name.

```
1234A      1235A      2468B      A34
BUSHING2   COLLAR23   D-0478   DEMO
FLANGE12   G041414    L12398   SMALTEST
TESTPROG

c:\programs\
a:\
a:\programs\
d:\
d:\programs\
OTHER

Enter-Select & exit
"Alt" plus:
C - Copy file
D - Delete file
N - Create new file
S - Select directory
R - Rs232 operations
ESC - Exit
arrows move cursor

Name of file to create? newprog3_
Current machine file; null
Current directory; c:\programs\
```

2. Type name (less than 9 alpha-numeric characters), then press Enter

```
Press Esc for no backup, F1 to make backup copy "c:\programs\newprog3.&"

3. The OmniTurn editor will start. Using the editor is described in Section Five.
```

```
1234A      1235A      2468B      A34
BUSHING2   COLLAR23   D-0478   DEMO
FLANGE12   G041414    L12398   NEWPROG3
SMALTEST   TESTPROG

c:\programs
a:\
a:\programs\
d:\
d:\programs\
OTHER

Enter-Select & exit
"Alt" plus:
C - Copy file
D - Delete file
N - Create new file
S - Select directory
R - Rs232 operations
ESC - Exit
arrows move cursor

Make c:\programs\newprog3 current machine file? (y/n)
Current machine file; null
Current directory; c:\programs\
```

Note program added to list

4. You are prompted to make new program current machine file.

Alt-R: RS-232 operations

Selecting “RS-232 operations” (Alt-R) takes you to a screen that enables sending and receiving files via the serial port on the rear panel of the OmniTurn CNC.

You are prompted as follows:

- Press 1 To receive a file from another computer
- Press 2 To send a file to another computer
- Press 3 To exit the RS-232 handler

Codes Honored by the OmniTurn control

(Sort by Code)

Code Usage	Description	Pages
G00 G00	Rapid move	11,12
G01 G01Fn	Feed move	12,13
G02 G02XnZnInKnFn	Arc -Clockwise	6,15,17-24
G02 G02XnZnRn	Arc -Clockwise	6,17-24
G03 G03XnZnInKnFn	Arc -Counter Clockwise.....	6,17-24
G03 G03XnZnRn	Arc -Counter Clockwise.....	6,17-24
G04 G04Fn	Dwell	6,25,62
G10 G10XnZn	Work Shift.....	6,26-28,73
G33 G33XnZnInKnAnCaPO	Threading cycle.....	6,29-36
G40 G40	Cancels Tool Nose Radius Compensation	37-43
G41 G41	Left hand Tool Nose Radius Compensation	6,37-43
G42 G42	Right hand Tool Nose Radius Compensation	6,37-43
G70 G70	Inch mode	6,44
G71 G71	Metric mode.....	6,44
G72 G72	Diameter programming mode... 6,10,14,16,21,22,29,38, 44 ,46,49,59	
G73 G73	Radius programming mode.. 6,10,14,16,21,20,22,29,38, 44 ,46,49,59	
G74 G74XnZnInUnFn	Box Roughing cycle	45-47
G75 G75InUnFnPn	Box Contour Roughing cycle	48-52,54
G76 G76Sn	Minimum spindle speed for constant surface feet	6,60
G77 G77Sn	Maximum spindle speed for constant surface feet.....	6,60
G78 G78UnFnPn	Rough Contour Cycle	51-55
G81 G81ZnFn	Drill cycle.....	6,56
G83 G83ZnKnFn RnLnCn	Peck drill cycle	6,57,58
G90 G90	Absolute mode selection	5,6,10,12,17,19,21,20,56,57, 59
G91 G91	Incremental mode selection.....	5,6,10,17,56,57, 59
G92 G92XnZn	Preset axis position	36, 59 ,74
G94 G94Fn	Inches per minute mode	6,7,11,12,20,45,49,56, 59
G95 G95Fn	Inches per revolution mode.....	6,7,11,12,45,49,56,59
G96 G96Sn	Spindle speed set as surface feet	6,60
G97 G97	Spindle speed set as RPM	6,60
M00 M00	Program stop - does not cancel active "M" functions.....	61
M01 M01	Optional stop	61
M02 M02	End program - does not cancel active "M" functions.....	26,62,65
M03 M03Sn	Spindle on, CW	16,62,65,74
M04 M04Sn	Spindle on, CCW.....	62,65,74
M05 M05	Spindle off, stop.....	62,65,74
M08 M08	Coolant on	16,62,65
M09 M09	Coolant off	62

Codes Honored by the OmniTurn control (Sort by Code)

Code Usage	Description	Pages
M12 M12	Collet clamp.....	62
M13 M13	Collet unclamp.....	62
M19 M19	Spindle Positioning; lock at home (optional C-Axis only).....	62,74
M25 M25	User assigned on.....	62
M26 M26	User assigned off.....	62
M30 M30	End of program - cancels all active "M" functions.....	26,62,65
M31 M31	Cancels Cycle Repeat mode.....	62
M84 M84	Locks feedrate & spindle speed override pots at 100%.....	<i>Section1: 1.4</i>
M85 M85	Re-enables override pots (M30, M02, T# also re-enables).....	<i>Section1: 1.4</i>
M88 M88	Spindle Positioning; lock at home; high res (optional: C-Axis only)	63,74
M89 M89	Spindle Positioning; lock at arbitrary home (optional: C-Axis only)	63,74
M91 M91	Wait for HDR111 to be open circuit	63
M92 M92	Wait for HDR111 to be shorted.....	63
M93 M93	Wait for HDR110 to be open circuit	63
M94 M94	Wait for HDR110 to be shorted.....	63
M95 M95	Jump to subroutine 1 if HDR112 is shorted.....	63
M97 M97InCnPn	Jump to subroutine, conditional (optional: PLC option only)	63
M98 M98Pn	Jump to subroutine	63
M99 M99	End subroutine.....	63
CI CInnn.nn	Incremental spindle angle (optional: C-Axis only)	74
CA CAnnn.nn	Absolute spindle angle (optional: C--Axis only)	74
C XnZnCn	Automatic chamfer at intersection	15,16,67
D Dn	Secondary offsets, axis correction or TNR comp value	68-71
F Fn	Feedrates, dwell	48,56,74
LS LSn	Loop start	72
LF LF	Loop finish.....	73
R XnZnRn	Automatic radius at intersection	15,16
S Sn	Spindle speed selection, SFM or RPM	60,65
T Tn	Tool offset call command.....	9
/ /	Block delete	<i>Section5: 5.5</i>
} }n	Begin subroutine	63

Codes Honored by the OmniTurn control

(Sort by Description)

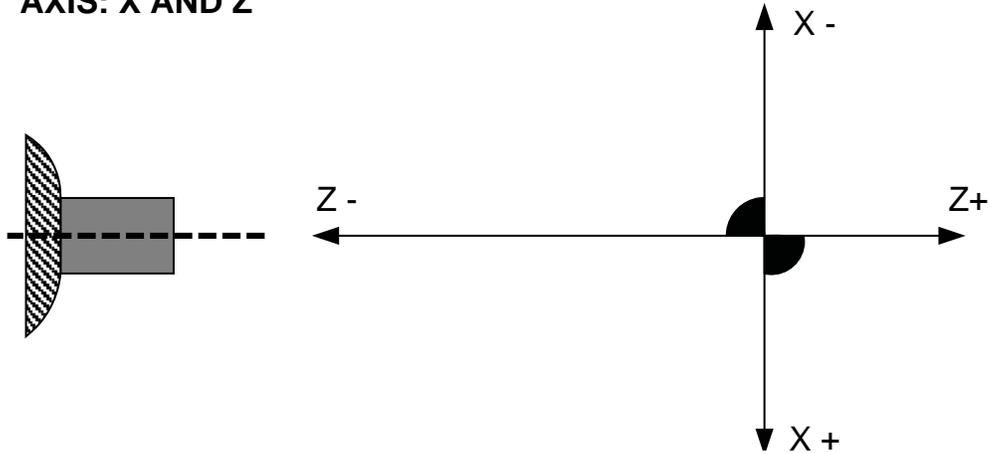
Code	Usage	Description	Pages
G90	G90	Absolute mode selection.....	5,6,10,12,17,19-21,56,59
CA	CAnnn.nn	Absolute spindle angle (optional: C—Axis only)	74
G02	G02XnZnInKnFn	Arc -Clockwise.....	6,1517-24
G02	G02XnZnRn	Arc -Clockwise.....	6,17-24
G03	G03XnZnInKnFn	Arc -Counter Clockwise	6,17-24
G03	G03XnZnRn	Arc -Counter Clockwise	6,17-24
C	XnZnCn	Automatic chamfer at intersection.....	15,16,67
R	XnZnRn	Automatic radius at intersection.....	15,16
}	}n	Begin subroutine	63
/	/	Block delete.....	<i>Section 5.5</i>
G75	G75InUnFnPn	Box Contour Roughing cycle.....	48-52,54
G74	G74XnZnInUnFn	Box Roughing cycle.....	45-47
M31	M31	Cancels Cycle Repeat mode	62
G40	G40	Cancels Tool Nose Radius Compensation.....	16,37-43
M12	M12	Collet clamp	62
M13	M13	Collet unclamp	62
M09	M09	Coolant off	62
M08	M08	Coolant on	16,62,65
G72	G72	Diameter programming mode ...	6,10,14,16,21,22,29,38,44,46,49,59
G81	G81ZnFn	Drill cycle.....	6,56
G04	G04Fn	Dwell.....	6,25,62
M30	M30	End of program - cancels all active M functions	26,62,65
M02	M02	End program - does not cancel active M functions.....	26,62,65
M99	M99	End subroutine.....	63
G01	G01Fn	Feed move	12,13
F	Fn	Feedrates, dwell.....	48,56,74
G70	G70	Inch mode	6,44
G94	G94Fn	Inches per minute mode	6,7,11,12,20,45,49,56,57,59
G95	G95Fn	Inches per revolution mode	6,7,11,12,45,49,56,57,59
G91	G91	Incremental mode selection	5,6,10,17,56,57,59,70
CI	CIinn.nn	Incremental spindle angle (optional: C-Axis only)	74
M98	M98Pn	Jump to subroutine	63
M95	M95	Jump to subroutine 1 if HDR112 is shorted.....	63
M97	M97InCnPn	Jump to subroutine, conditional (optional: PLC option only).....	63
G41	G41	Left hand Tool Nose Radius Compensation.....	16,37-43
LF	LF	Loop finish	73

Codes Honored by the OmniTurn control (Sort by Description)

Code Usage	Description	Pages
LS LSn	Loop start.....	72
G77 G77Sn	Maximum spindle speed for constant surface feet.....	6,60
G71 G71	Metric mode.....	6,44
G76 G76Sn	Minimum spindle speed for constant surface feet.....	6,60
M84 M84	Locks feedrate & spindle speed override pots at 100%....	<i>Section 1:</i> 1.4
M01 M01	Optional stop.....	61,62
G83 G83ZnKnFn RnLnCn	Peck drill cycle.....	6,57,58
G92 G92XnZn	Preset axis position.....	36,59,74
M00 M00	Program stop - does not cancel active M functions.....	61,62
G73 G73	Radius programming mode ...	6,10,17,18,20,29,31,32,38,44,46,49,59
G00 G00	Rapid move.....	11,12
M84 M84	Re-enables override pots (M30, M02, T# also re-enables). <i>Section 1:</i>	1.4
G42 G42	Right hand Tool Nose Radius Compensation.....	37-43
G78 G78UnFnPn	Rough Contour Cycle.....	51-55
D Dn	Secondary offsets, axis correction or TNR comp value.....	68-71
M05 M05	Spindle off, stop.....	62,65,74
M04 M04Sn	Spindle on, CCW.....	62,65,74
M03 M03Sn	Spindle on, CW.....	16,62,65,74
M19 M19	Spindle Positioning; lock at home (optional C-Axis only).....	62,74
M88 M88	Spindle Positioning; lock at home; high res (optional: C-Axis only) ..	63
M89 M89	Spindle Positioning; lock at arbitrary home (optional: C-Axis only) ..	63
S Sn	Spindle speed selection, SFM or RPM.....	60,65
G97 G97	Spindle speed set as RPM.....	6,60
G96 G96Sn	Spindle speed set as surface feet.....	6,60
G33 G33XnZnLnKnCnPO	Threading cycle.....	6,29-36
G41, G42	Tool Nose Radius Compensation.....	6,37-43
T Tn	Tool offset call command.....	9
M26 M26	User assigned off.....	62
M25 M25	User assigned on.....	62
M91 M91	Wait for HDR111 to be open circuit.....	63
M92 M92	Wait for HDR111 to be shorted.....	63
M93 M93	Wait for HDR110 to be open circuit.....	63
M94 M94	Wait for HDR110 to be shorted.....	63
G10 G10XnZn	Work Shift.....	6,26-28,73

Nomenclature

AXIS: X AND Z



The slide has two axes of travel.

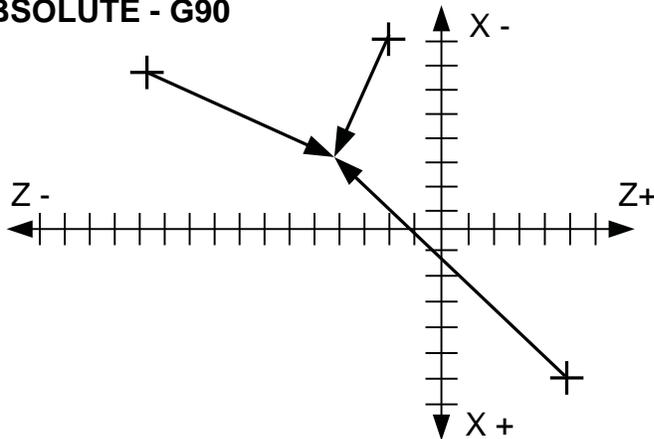
X: Up and down (on GT) or towards and away from you (on Attachments).

Travel up or away from you is (-) minus; down or towards you is plus (+).

Z: Towards or away from the spindle.

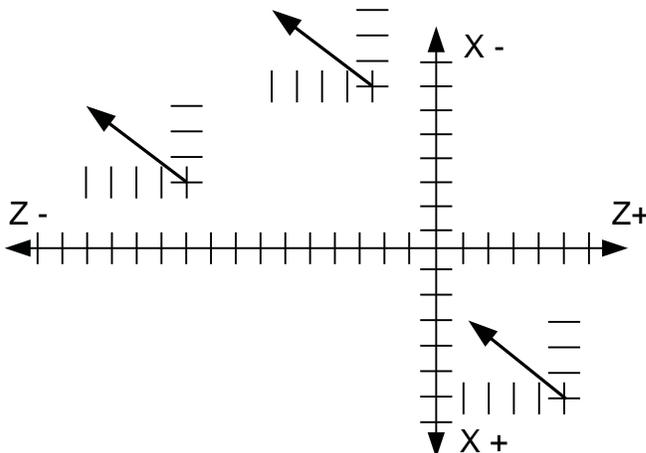
Going towards the spindle is (-) minus. Away from the spindle is (+) plus.

ABSOLUTE - G90



IN ABSOLUTE (G90) THE FOLLOWING
MOVE BRINGS YOU TO THE SAME POINT
NO MATTER WHERE YOU START
X-3 Z-4

INCREMENTAL - G91



IN INCREMENTAL (G91) THE FOLLOWING
MOVES YOU THE SAME AMOUNT FROM
EVERY START.

X-3 Z-4

Programming Format

- **The default** mode for X moves is G73 - radius moves, to program in diameters you must use G72 in the beginning of the program.
- **The first command** of a program must be G90 or G91 to define if the program is in absolute or incremental.
- **No blank** lines are allowed in a program, blank spaces are OK.
- **Comments** are any text or data enclosed in parentheses“()”. Their purpose is to convey to the operator any information that the programmer might think is useful. Comments are displayed in the lower left corner of the screen. They stay on the screen till the comment is changed. As an example, you may want to use the comment to tell the operator what action to take when the spindle stops. For an example, the slide is told to go "HOME" and then the comment is displayed on the screen. Then the slide stops with the message on the screen.
 - Do not put text on lines by itself. Comments must be on a line with a command!
 - Keep the amount of text to a minimum, too much text can cause problems.
 - A good place to put comments is on a line with a tool call ie: T1(LH turn tool)
 - Use only text, do not use periods or commas or any other symbol such as i.e.:

! @ # \$ % ^ & * “ ‘ ? > < / \ | = - .

- **Do not put** any text in a loop.
- **Commas are not allowed anywhere in the program**
- **Dimensional** data is interpreted with a resolution of .00005". The fifth digit to the right of a decimal point must be a 0 or a 5. NOTE: when programming in diameter mode the X axis resolution is .0001 ", not .00005".
- **Decimal** point programming is used. Leading and trailing zeros need not be entered. For example "X1" is interpreted as 1 inch. X1 = X1.00000
- **G and M codes** must be programmed as two digit codes. "G2" is not a legal code and it will be ignored. Also be sure to use the zero and not the letter O as part of the G and M codes. **NOTE:** To avoid this kind of mistake, use always lowercase letters (*x, z, t, etc*)
- **Modal commands:** These are commands that remain active until canceled:
All "M" codes, G10, G70, G71, G72, G73, G76, G77, G90, G91, G94, G95, G96, G97
- **One shot commands:** These act only on the statement they are programmed in:
G02, G03, G04, G33, G34, G81, G83, G92
- **Conflicting commands:**
There can be only one "M" command per line of code
There can be only one "one shot" G code per line of code
There can be more than one nonconflicting modal G code per line
The S and F commands can be with any other command
- **N sequence** are not allowed. They can cause intermittent problems.

Programming Format

- **Feedrate:**

This command specifies the speed at which the tool will travel. Once a Feedrate has been established, it remains until it is changed. Feedrates are specified as either Inches per minute (IPM) or Inches per revolution (IPR).

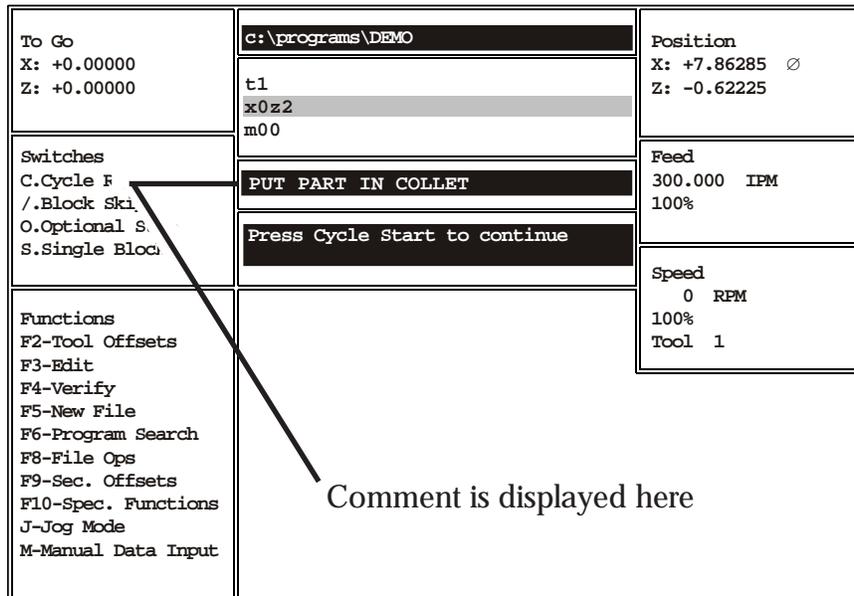
X.1Z-.4F7 This line of code has a feedrate of 7 IPM when in G94 (F300 max)

X1Z-.4F.005 This line of code has a Feedrate of .005 IPR when in G95

- **Tool Selection:**

This command is used to make a new tool offset active. The letter "t" identifies a tool command. For example, **t2** makes the tool offset for t2 active. After this command is executed the absolute position display will show the distance from the tip of tool two to the absolute zero of the work piece.

```
t1
x0z2
m00 (PUT PART IN COLLET)
```



- **Secondary offsets:**

These are activated by the letter D. See the section of secondary offsets.

X.2Z-.35D2 Calls secondary tool offset number two

Creating a new program

A new program can be created at the OmniTurn, or 'off-line' at a computer. When the program is created at the OmniTurn, it is automatically saved to the hard drive. To save a backup to the floppy drive, use the File Handler (Section 1.19). You also have opportunity to backup programs when the OmniTurn is first turned on: selecting 'Y' at the first prompt will backup any programs that have been created or changed since last turn-on.

To create a new program on the OmniTurn:

Press "A" from the main menu to go to the Automatic mode menu.

If no program is loaded, you will see the prompt "File to be processed:". Enter the name of the new file and press enter.

If a program is already loaded, press F8 key to get the prompt. Enter the name of the new file and press enter.

If there was already a program with the name you just entered, the control would now be ready to run it. If this is the case, then you would have to select a new name or change the program of the existing one already there.

If there was no other program that had the new name then the program editor will start. You will be asked to press F1 to create a backup (not necessary) or to press Esc to continue. Press Esc. The editor will provide a new blank screen to type your program.

To create a new program at your computer:

Create a new folder called OmniTurn Programs.

Open NotePad or Write or any word processor you're familiar with.

Enter the program blocks.

Save the file as a text file, with a name that does not exceed eight characters. NotePad saves as text by default; use the 'save-as' feature with other word processors, and save as text.

Rename the file to remove the ".txt".

Copy the file to floppy disk and put the floppy in the OmniTurn.

Use the File Handler (Section 1.19) to copy the file to the OmniTurn hard drive

Tool call statements

The “**tool call statement**” is done in two blocks:

Tn
XnZn

The first block of the T command must have a tool number in the range of 1 through 32. The slide does not move when this command is executed, but the Position display will change to show the how far the tool being used is from the absolute zero of the part. (In order to use this command the tool must have its tool offset established first, otherwise the tool is referenced to machine Home).

The line following must be an X and Z move. You must have both values given! This is the location that the tool will move to. To be safe, you should first move the tool come to a location that is relatively far from the part in Z, and then move the tool into the work. This will help eliminate some of the tool interference problems new users encounter. Later you can call the tool closer in Z to shorten the cycle time.

G00.....Rapid move
T1.....Tool call for tool# 1
X.25Z1move the tool to a location of X= .25 and Z= 1
Z.1move the tool closer to the work, still in rapid mode

Linear moves X n Z n XnZn

Linear one- and two-axis moves are accomplished by giving the axis and the value to move. The result of a command will depend on which mode is active: G90 or G91, and G73 (radius) or G72 (diameter). The examples that follow assume absolute mode (G90).

The X default mode is **radius** mode (G73), not diameter. To specify moves in diameter you must put a G72 in the first block of your program, as shown below. Feed and Rapid moves are accomplished as follows:

Movement in X:

Xn	Single axis (X) move
example: X.25	In radius mode (G73) this will move to .25 from X0 setting for this tool; in diameter mode (G72), this will move to .125 from X0 setting for

this tool.

Movement in Z:

Zn	Single axis (Z) move
example: Z-.1	This will move the tool toward the spindle 0.1" from Z0 setting for this

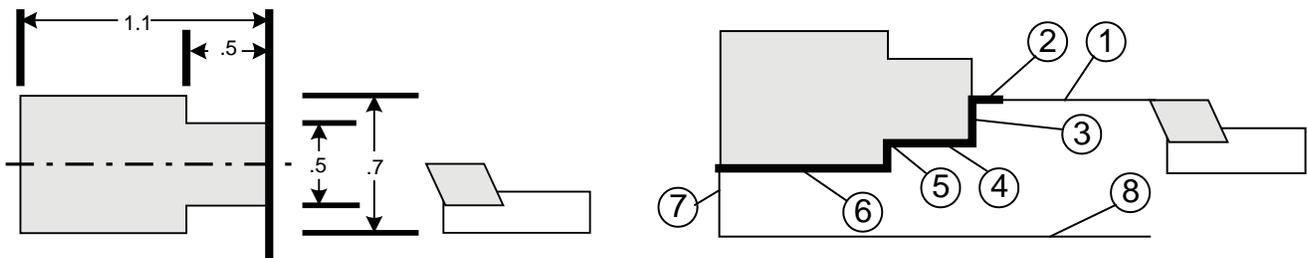
tool.

Z moves can be programmed directly off the print.

Movement in X and Z simultaneously

XnZn	Two axis linear move (X & Z)
------	------------------------------

Example for linear moves



The moves to generate this tool path could be as shown below. The tool path given does not show any of the codes needed to determine the feedrates or spindle speeds etc. The code is shown in the;

	diameter mode:	radius mode:
	G90G94F300G72	G90G94F300G73
	M03S2000	M03S2000
	M08	M08
	T1	T1
	X0 Z1	X0 Z1
(1)	Z.1	Z.1
(2)	Z0 G95f.003	Z0G95f.003
(3)	X.5	X.25
(4)	Z-.5	Z-.5
(5)	X.7	X.35
(6)	Z-1.1	Z-1.1
(7)	X 1	X .5
(8)	Z1G00	Z1G00
	M30	M30

Linear moves X n Z n XnZn

A note about feedrate modes:

The feedrate commands are modal. Once they are set they stay in effect until they are changed. So once you set the mode you do not have to change it again until you change the mode. Once the feedrate mode is selected you can change the feedrate by adding the new feedrate to the line of code where you need it.

G94 -Inch per minute mode

The G94 command will make all feedrate commands effective in inch/minute. If you have a feedrate command of F4 in this mode it will move the slide at 4" per minute. F300 would be 300" per minute, which is the fastest feed. G94 mode is generally used for calling tools, or for moving the slide when the spindle is stopped.

G95 -Inch per revolution mode

This mode has the tool move in inches/revolution. *If the spindle is not turning the slide will not move.*

The feedrate used is the last one that was given. If a new feedrate is needed it can be added to the movement command. The type of feedrate mode can be determined at or before the line. The mode established stays in effect until it is changed. At 4000 rpm the fastest G95 setting is F.0075.

Example formats are:

G94XnF4 Single axis (X) move with new feedrate, at 4"/minute
G95XnF.003 Single axis (X) move with new feedrate, at .003"/rev

Rapid travel (G00):

Rapid travel is 300 inches per minute, which is also the fastest *feed rate* also. OmniTurn accelerates quickly to 300 ipm, and tool-to-tool distances are generally an inch or two, so transit time between tools is very quick, typically 1/4 - 1/2 second.

Feed-rate override is effective for rapid travel; set the override knob for a low value when single-blocking through any new program.

Linear moves X n Z n XnZn

Feedrate mode (G01):

You can go from rapid travel mode to feedrate mode by either specifying a new feedrate and mode, IE: G94F300 for rapid travel, and G95F.n for feedrate. Or using G01 to revert to the last described feedrate mode in the program. This eliminates having to use the G95F.n over and over. The other advantage of using G01 verses specifying the feedrate every time is that if you want to globally change the feedrate you only have to change one line.

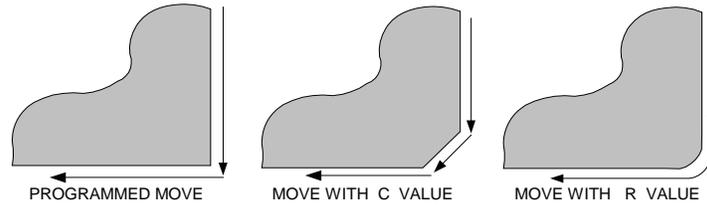
To use the G01 command you must specify the first feedrate move in the program with the G95F.n format. This way the program knows what feedrate to use. Here is an example showing establishing the rapid and feedrate modes and making some moves, then changing to rapid, and then back to the feedrate with G01:

```
G94F300X0Z1  
Z.1  
G95F.003Z.005  
X.5 G00Z.05  
X0  
G01Z0  
X.5  
G00Z1
```

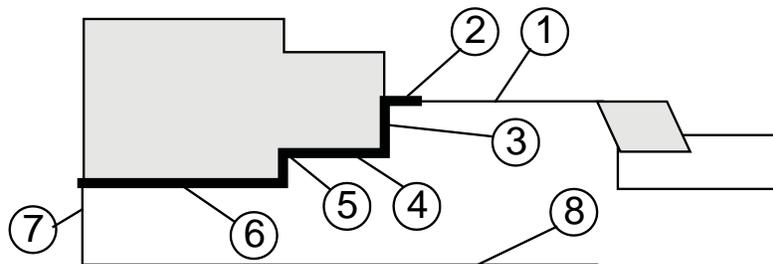
Linear moves X n Z n XnZn

In the example below we will be 'finish turning' a sample part with a left hand tool. The part dimensions are given in diameters: *put G72 in the first block of your program to set OmniTurn to Diameter Mode.*

Z zero is set at the face of the part, X zero is set at the center.



The next diagram shows the rapid moves as a lighter line, feed moves are shown as the heavier lines.



Example in diameter mode

G90 G72 G94 F300 (Establish Absolute positioning, Diameter mode, Feed IPM @ 300" per minute)

T1 (turn tool)	(Call Tool #1 offset)
X0 Z1	(Moves the tool to a safe start location)
Z.1	(Rapid to .1" from the face)
G95 F.003 Z0	(Set feedrate to IPR @ .003" per revolution, Feed to the face)
X.5	(Feed out to .5" diameter)
Z-.5	(Feed to Z -.5)
X.7	(Feed to .7" diameter)
Z-1.1	(Feed to -1.1 in Z)
G00 X1	(Set feedrate to IPM @ 300"/minute, Pull the tool away in X)
Z 1	(Withdraw the tool in Z)
M30	(End of program command)

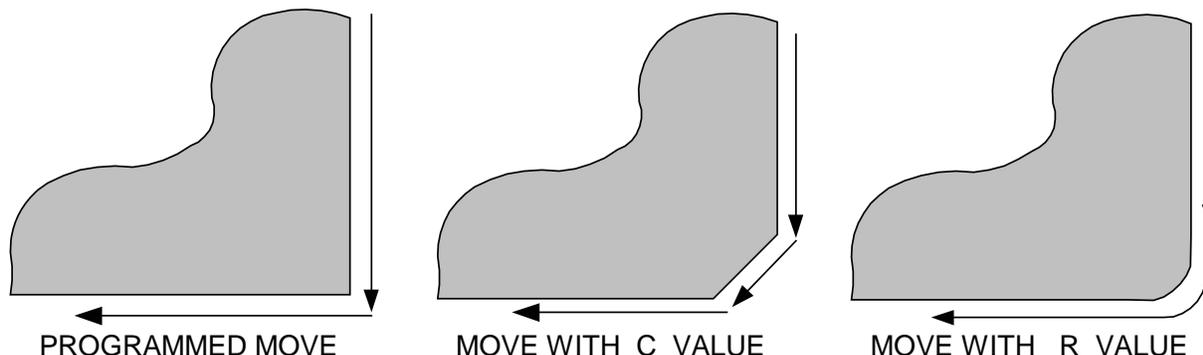
The same example in radius mode, for reference only

G90 G94 F300	(Establish Absolute positioning, Feed IPM @ 300" per minute)
T1	(Call Tool #1 offset)
X0 Z1	(Moves the tool to a safe start location)
Z.1	(Rapid to .1" from the face)
G95 F.003 Z0	(Set feedrate to IPR @ .003" per revolution, Feed to the face)
X.25	(Feed out to .25" radius)
Z-.5	(Feed to Z -.5)
X .35	(Feed to .35" radius)
Z-1.1	(Feed to -1.1 in Z)
G00 X.5	(Rapid feedrate, Pull the tool away in X)
Z1	(Withdraw the tool in Z)
M30	(End of program command)

This page intentionally blank

Automatic corner radiusing (R) and chamfering(C)

It is possible to automatically generate a chamfer or radius between two connecting linear moves. Just program the lines to the theoretical intersection point of the two moves, and put a C or R with the absolute amount of the radius or chamfer needed.



Format

X nZnRn -ZnRn -X nRn

X nZnCn -ZnCn -X nCn

XnZn The linear move leading to the intersection point of two lines

Rn The n is the absolute value of the radius used to blend the two lines

Cn The n is the absolute value of the chamfer used to blend the two lines

RULES

The moves that are connected by the auto chamfer or radius must be linear moves. The C or R command will not work with blending arcs or arcs and lines. If you want to blend these use G02 and G03.

- The moves do not have to be at right angles
- A chamfer created is set back equally from the intersection point of the two lines.
- A radius created is made tangential to the two intersecting lines. The direction (CW or CCW) of the radius is determined automatically by the OmniTurn. It looks ahead to the next move.
- The n value must be the absolute (+) value

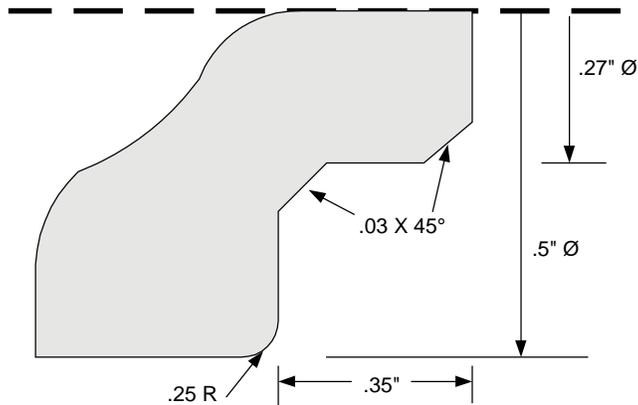
Running programs using C or R

When you use the automatic corner radius or chamfer commands the OmniTurn creates a number of moves to generate what you want. If you look at the command line while you run a program you will notice lines of code that you did not write. In the single block mode you can see arc (G02 or G03) commands. This is normal. When you leave the editor the OmniTurn automatically recreates the new moves. The program is also recreated whenever you change the secondary offset table.

Automatic corner radiusing (R) and chamfering(C)

Example

Note: The example program shown uses more codes than shown yet to this point in the book. M03, M08, G41, Dn and G40 are covered in other sections.

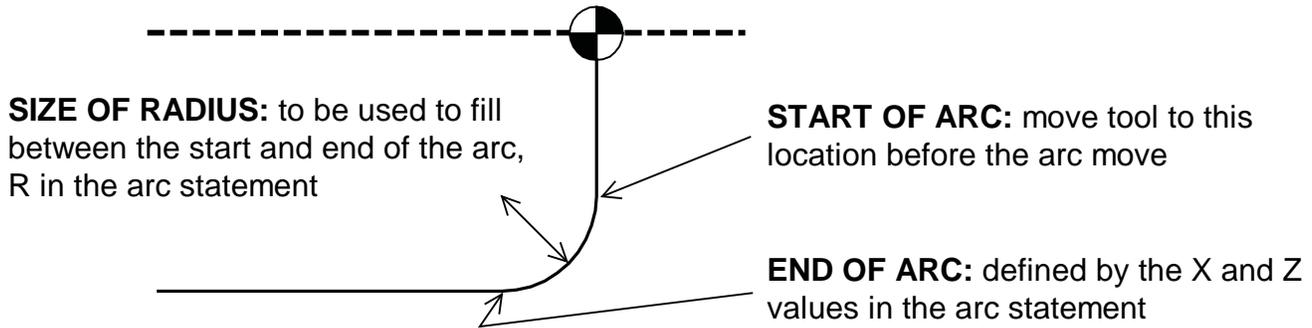


G90G72G94F300	G72 Program is in diameter mode
M03S2500	Turn the spindle on at 2500 rpm
M08	Turn the coolant on
T1(LH turn tool)	
X0Z1	
Z.05	
G95E001	
G41	Turn on LH cutter compensation
Z0D1	Use the radius value found in D1 for compensation
X.27 C.03	Set chamfer amount
Z-.35 C.03	Set chamfer amount
X.5 R.025	Set radius amount
Z-.45	
X.54	Move off the part more than the compensation value
G00Z1	
G40	Turn cutter compensation off
M30	

Arc statements G02 and G03

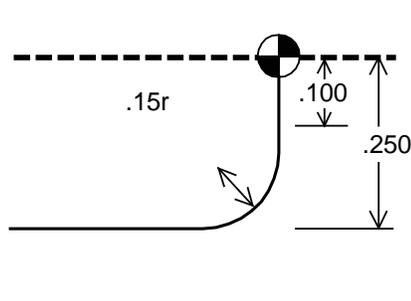
Using R version:

Before the arc statement is used the tool must be moved to the start location of the arc. Then the arc statement follows with the end of the arc location (X and Z) and the length of the connecting arc's radius.

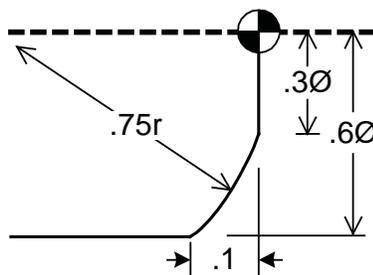


Examples of arc statements using R (NOTE: G73 = Radius Mode)

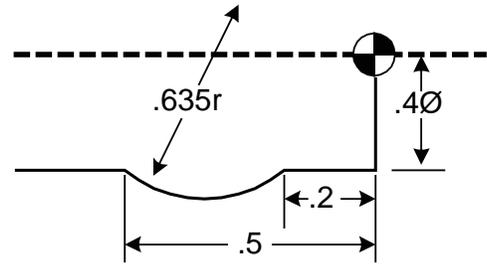
Following are three examples of arc statements using the arc statement with R. These sample programs are showing only a finish pass. **These are shown in the radius mode (G73) see 2.21 for diameter.**



```
g90g94f300g73
m03s2500
m08
t1
x0z1
z.1
g95f.003z0
x.1
g02x.25z-.15r.15
z-.3
x.275
g00
z1
m30
```



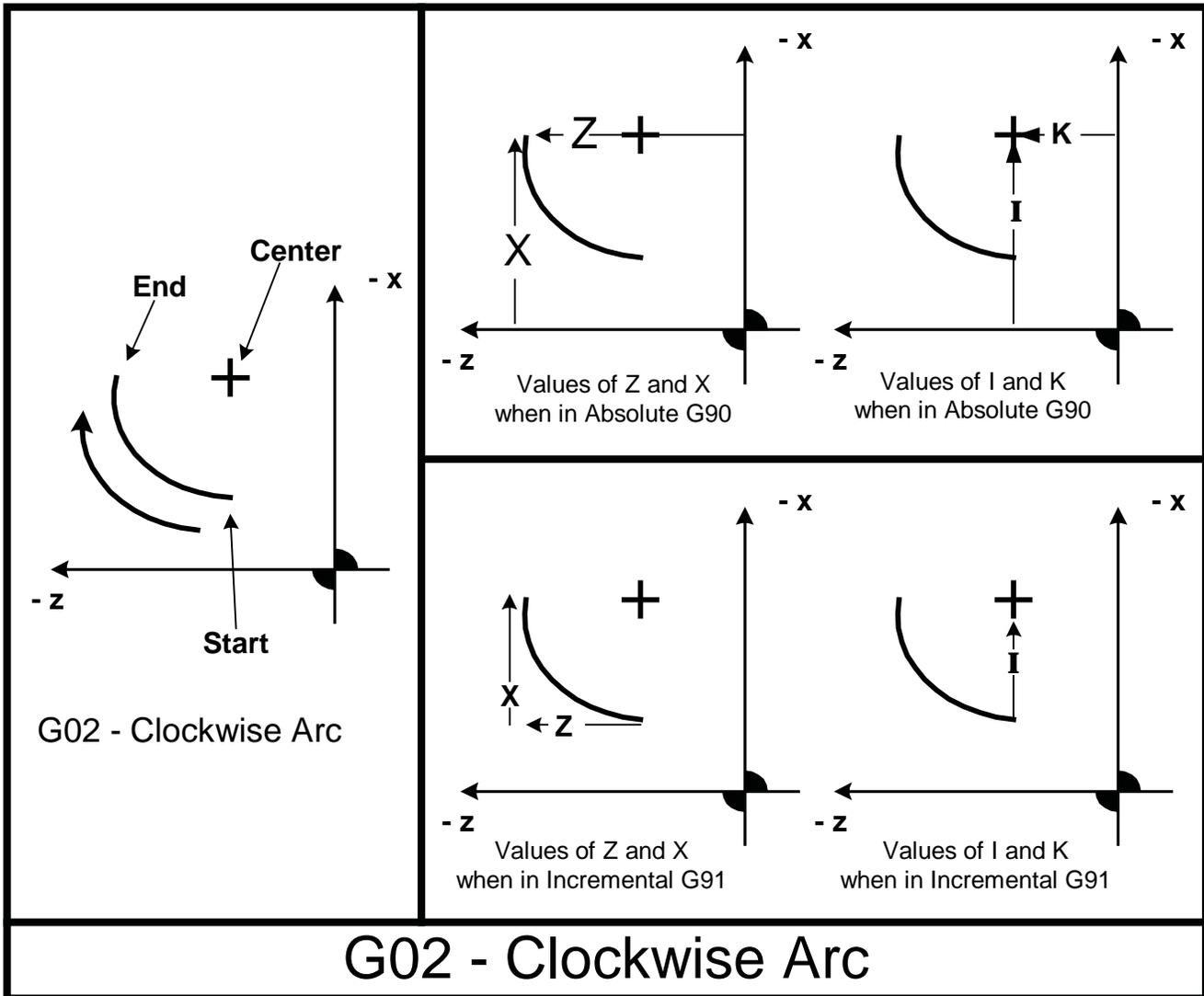
```
g90g94f300g73
m03s2500
m08
t1
x0z1
z.1
g95f.003z0
x.15
g02x.3z-.1r.75
z-.3
x.275
g00
z1
m30
```



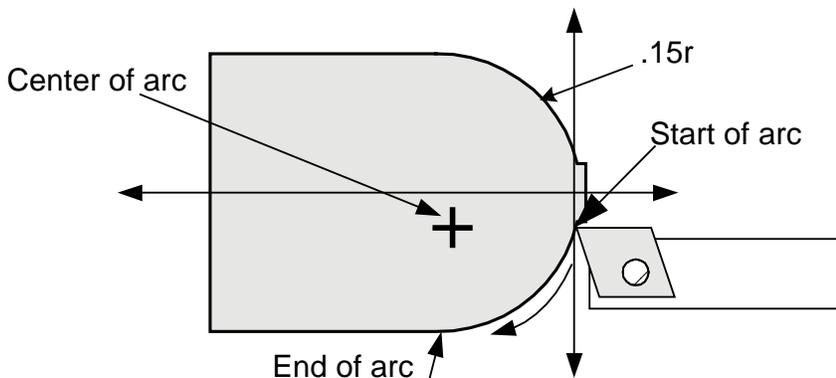
```
g90g94f300g73
m03s2500
m08
t1
x0z1
z.1
g95f.003x.2
z-.2
g02x.2z-.5r.635
z-.7
g00
x.3
z1
m30
```

Arc statements G02 and G03

Description of arcs using I and K in G73 (radius mode)



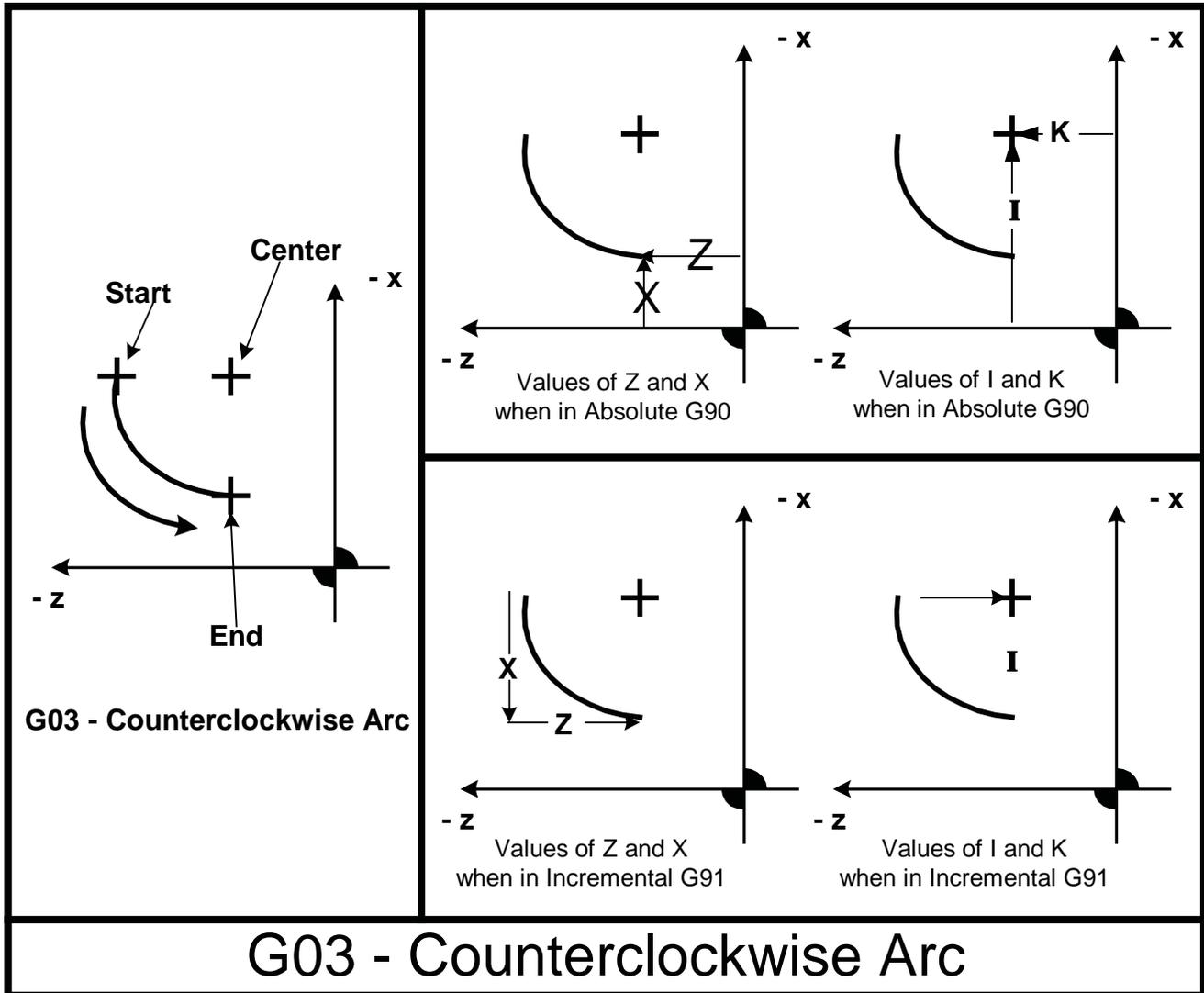
Using G02 - Clockwise Arc



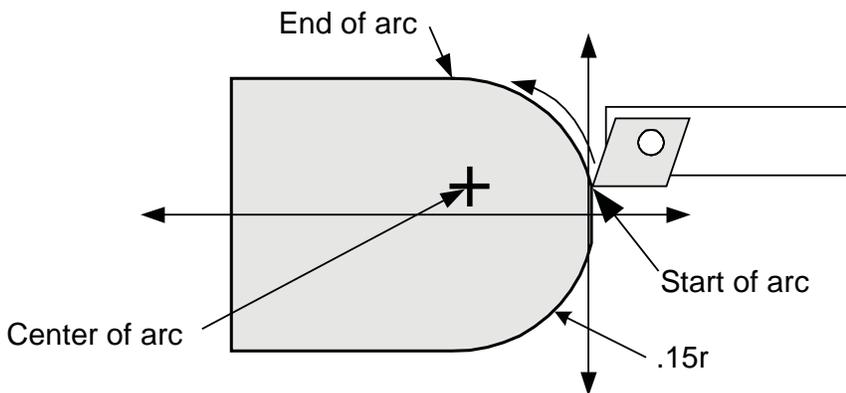
```
G90 G94 G73
T1F300
X0 Z.1
X.1 Z0
G02 X.25 Z-.15 I.1 K-.15
Z-.5
or the arc statment could be:
G02 X.25 Z-.15 R.15
```

Arc statements G02 and G03

G73 - Radius Mode



Using G03 - Counterclockwise Arc while in G73 - radius mode



```
G90 G94 G73
T1F300
X0 Z.1
X.1 Z0
G03 X.25 Z-.15 I.1 K-.15
Z-.5
or the arc statment could be:
G03 X.25 Z-.15 R.15
```

Arc statements G02 and G03

Diameter mode

Arc moves in diameter programming have minor differences from radius programmed arcs.

G02 and G03 arc moves in diameter mode (G72) and absolute (G90)

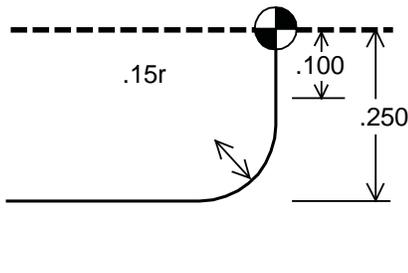
Note: -Using arc statements in the diameter mode (G72) be sure you are in absolute (G90)
-Position tool at start point before using arc move
-This format follows the Fanuc format more-closely than previously

G02XnZnInKn • G03XnZnInKn

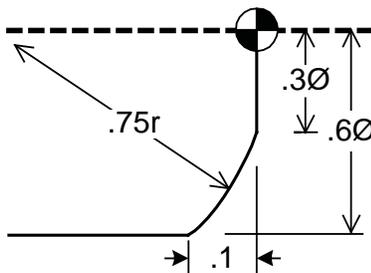
G02 Clockwise arcs
G03 Counter Clockwise arcs
Xn Diameter value at the end of the arc
Zn Location of end of the arc in Z from the part zero
In Incremental distance from arc start to the arc center in X
Kn Incremental distance from arc start to the arc center in Z
R The length of the radius to be used to connect the start and end points

Examples of arc statements using R in diameter mode:

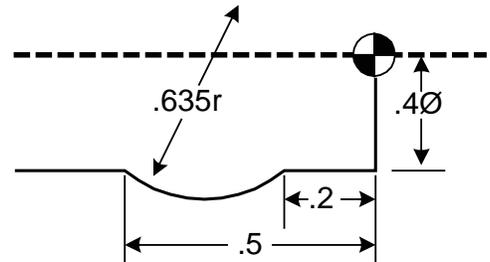
For an explanation on usage please refer to the section at the beginning of this chapter on using the R in the radius mode. The format is the same except the values of X are given in diameters (G72 mode).



```
g90g94f300g72
m03s2500
m08
t1
x0z1
z.1
g95f.003z0
x.2
g02x.5z-.15r.15
z-.3
x.55
g00z1
m30
```



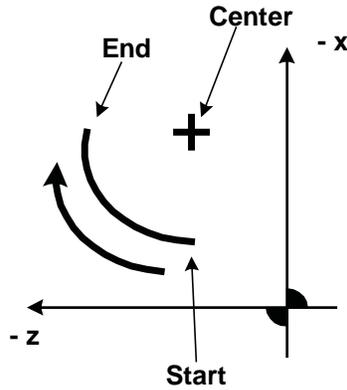
```
g90g94f300g72
m03s2500
m08
t1
x0z1
z.1
g95f.003z0
z0
x.3
g02x.6z-.1r.75
z-.3
x.65
g00
z1
m30
```



```
g90g94f300g72
m03s2500
m08
t1
x0z1
z.1
g95f.003
x.4
z-.2
g02x.4z-.5r.635
z-.7
g00
x.6
z1
m30
```

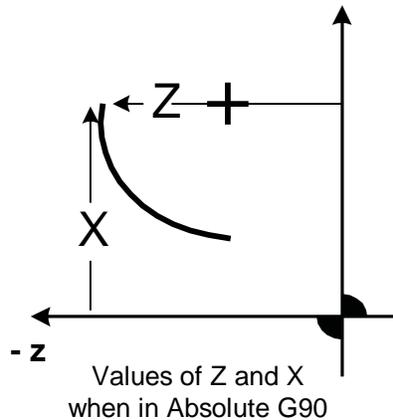
Arc statements G02 and G03

Arc statements using I and K in diameter mode (G72):

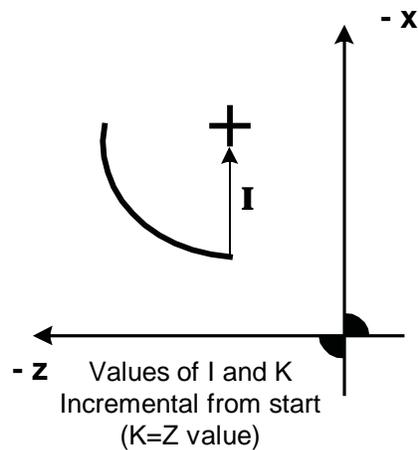


G02 - Clockwise Arc

End of arc ($X_n Z_n$): This is the same. This is the location of the end of the arc.

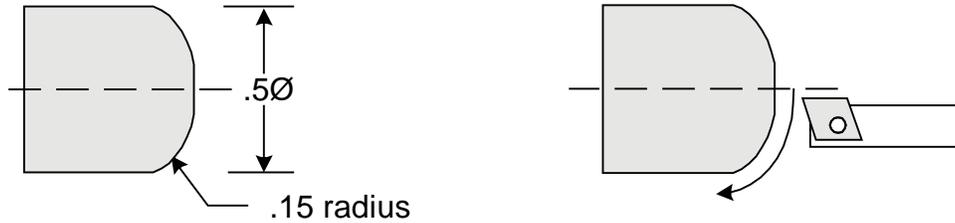


Arc center ($I_n K_n$): This is different. With diameter programming the arc center is defined as the incremental distance from the arc start to the center.

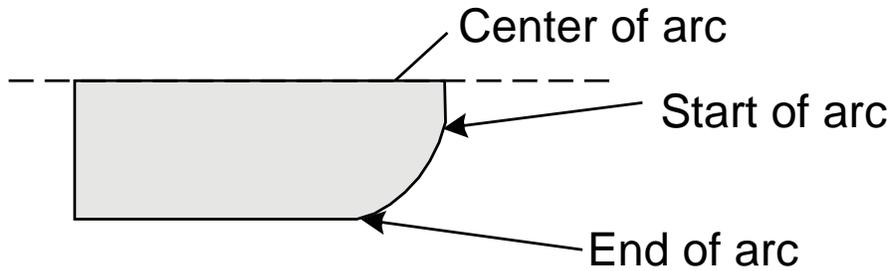


Arc statements G02 and G03

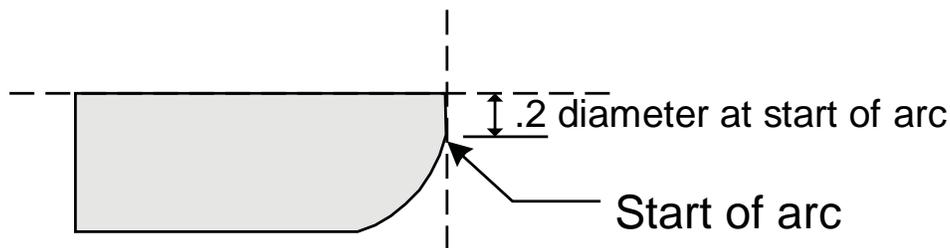
The following picture shows an example of an arc that is machined with G02 -CW using I & K



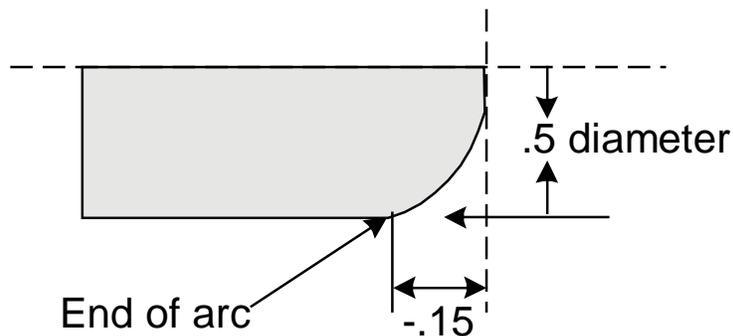
For this example first we show the three important locations that must be defined to write the arc statement:



The starting point of the arc is where $X = .2$ (diameter value) and $Z = 0$. So these values will be used to write the position move before the arc statement: **X.2Z0 (start of arc)**

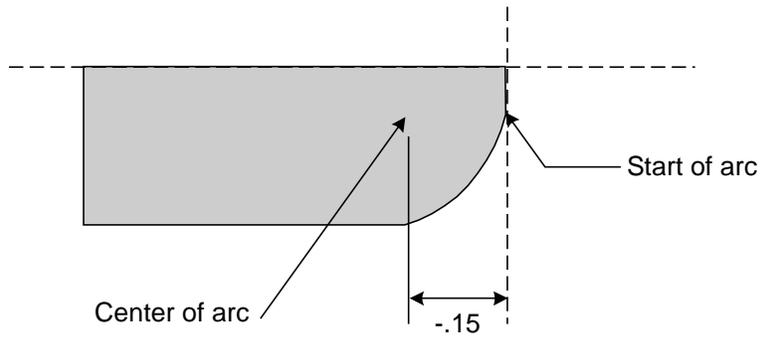


The end of the arc is defined from the absolute zero for the part. As shown on the following figure $X = .5$ and $Z = -.15$. These are used for the X and Z values in the G02



The I and K values are the incremental distances from the start of the arc to the center. In the following figure you can see that start and arc center in X are at the same diameter value so the distance between them is zero (I0). In the Z axis the distance is $-.15$. Note: the value of this is minus because of the direction, not the end location.

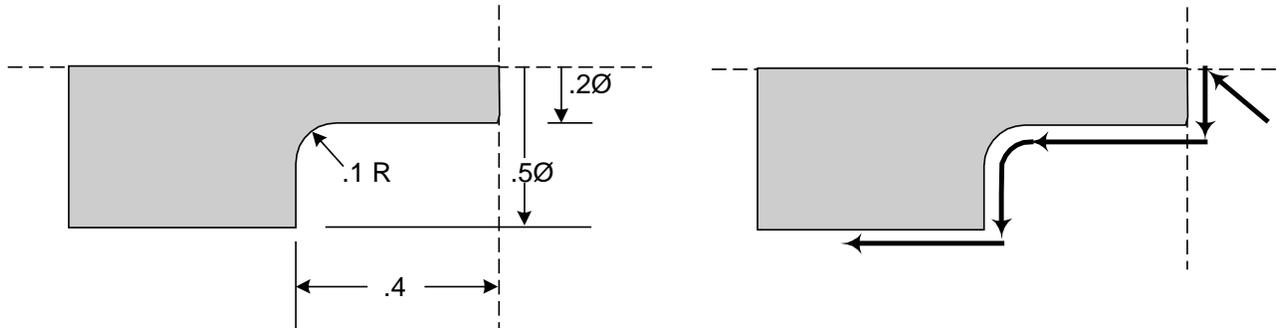
Arc statements G02 and G03



So the program lines could be:

```
XOZO  
X.2  
G02X.5Z-.1SIOK-.15
```

For the next example we will show a G03:



```
XOZO  
X.2  
Z-.3  
G03X.4Z-.4I.1K0  
X.5  
Z-.5
```

Dwell - G04

The dwell statement format is:

G04Fn

Note:

- The 0 in the G04 must be the number 0, not the letter o.
- The "n" after the F is the number of seconds needed to dwell.
- The shortest dwell is .1 seconds
- There is no limit to the length of the dwell. However we suggest you only use it for machining dwells. DO NOT use it for loading and unloading operations in the program. This would be unsafe!

Example

G04F5 This statement will dwell the slide .5 seconds.

G10 Work shift

Work shift is used to offset a program from the original starting point. Typical applications are:

- Machining multiple parts off a single shootout of a bar.
- Shifting a program away from the spindle the first time it is run

G10XnZn

G10 will shift the reference of the slide incrementally. If G10 is put into a loop the program will shift each time the command is used. The shift will take place on the next tool call. If you put the shift after a tool call the effect will take place the next time through the loop.

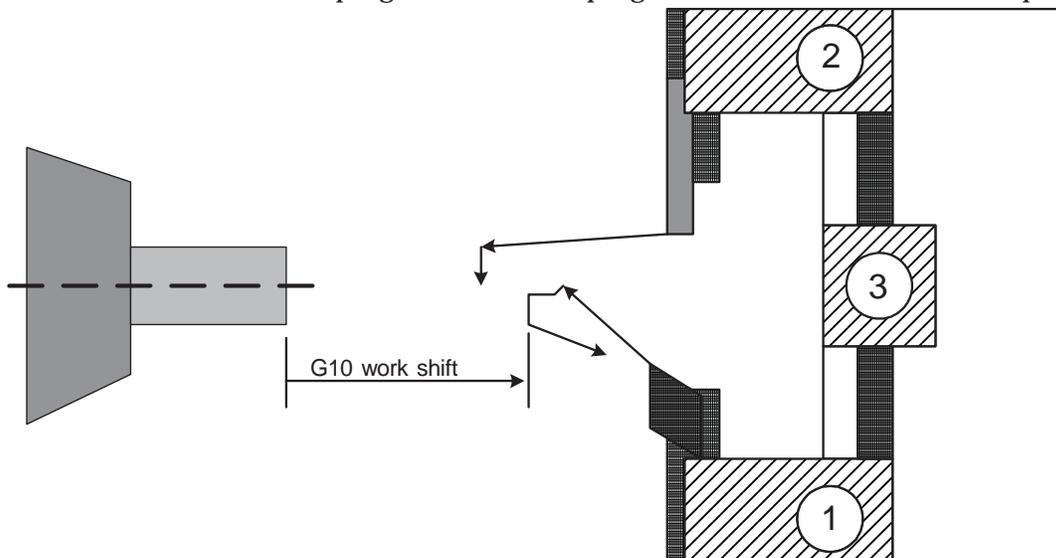
Note: The shift is executed on tool calls! Use the G10 command before tool calls otherwise there will be no effect.

The shift will be canceled any of these commands: TO - M30 - M02

The command must have a value for both X and Z.

Example of shifting a program for test running

In the following example we show using the G10 work shift for running a program the first time way from the work to make sure that the program looks like it will run OK. In this example you would set the tools to make the part. After the program is run a few inches away with the work shift the G 10 command would be removed from the program. Then the program would be run to make a part.



G90G72G94F300

M03S2000

G04F2

G10X0Z3 Shifts program 3” to the right

T1(LH TURN TOOL)

X0Z.2

ZOG95F01

X.1 Z-.05F.002

Z-.15

X.2

G94F300Z1

T2(PART OFF TOOL)

X0Z1

X-.4

Z-.35

G10 Work shift

G96S150
G76S500
G95F001 X.005
G94F300Z1
G97
M30

Example of shifting a program for bar work

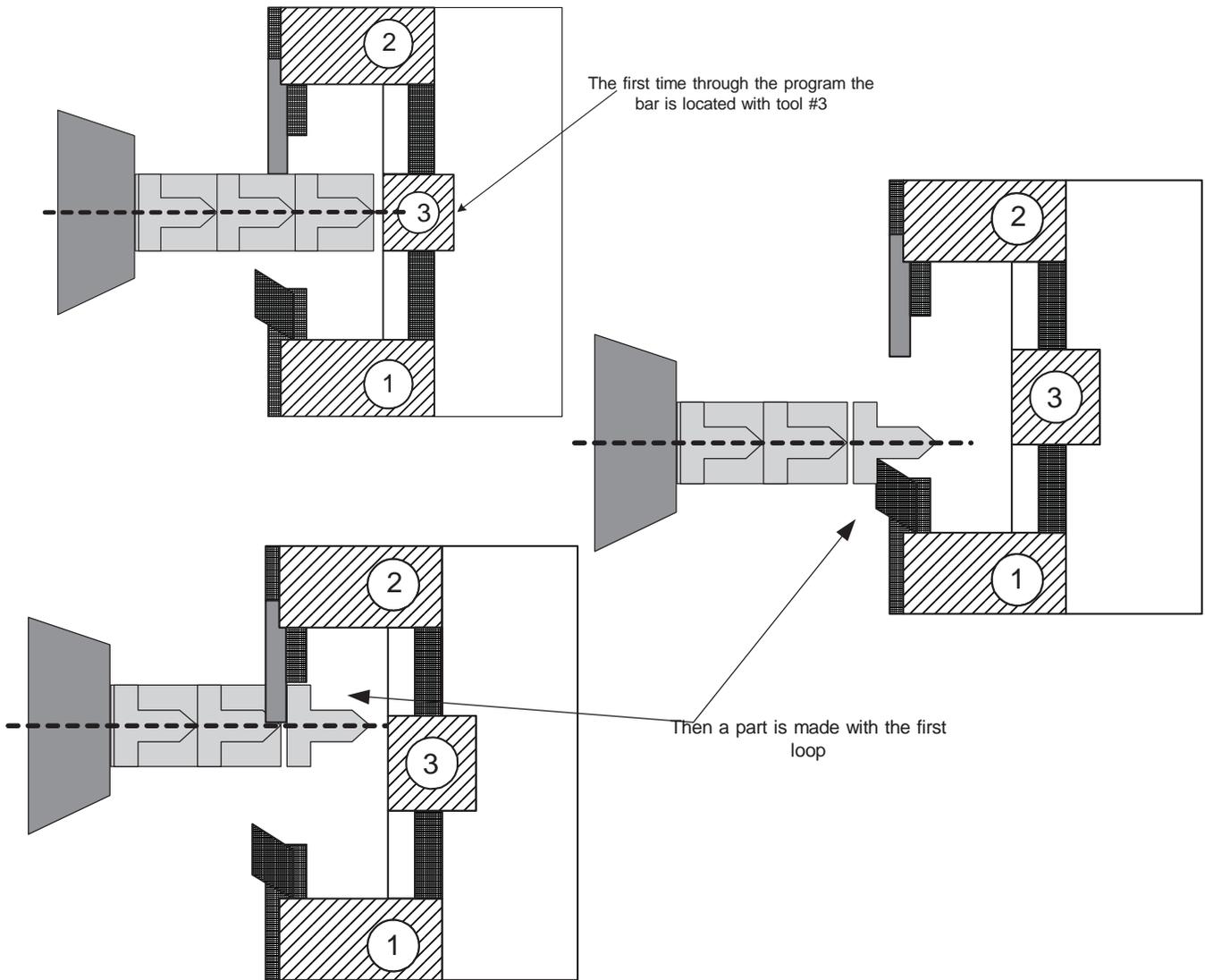
With the G10 work shift you can take a program and loop it with the shift so that you get multiple parts done on a single feedout. In the next example three parts will be made with one barfeed sequence:

G90G72G94F300
M03S2000
T3(WORK STOP)
XOZ1
Z-1
M13
Z.025F50
M12
Z 1 G94F300
LS3
T 1(LH TURN TOOL)
XOZ.2
S2000
ZOG95F01
X.1 Z-.05F002
Z-.15
X.2
G94F300Z1
T2(PART OFF TOOL)
XOZ 1
X-.4
Z-.35
G96S150
G76S500
G95F001 X.005
G94F300Z1
G97
G10XOZ-.35
LF
M30

The loop starts here and will repeat 3 times. Notice it is before a tool call.

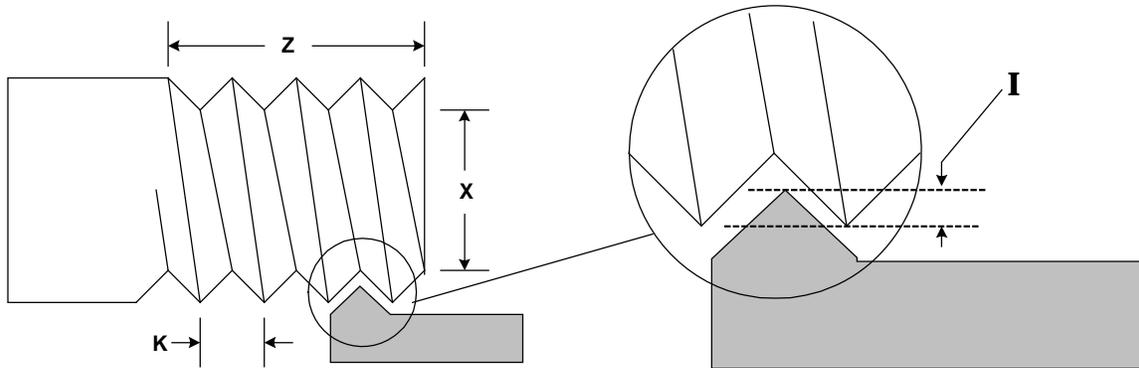
**Here is the work shift, shifting the next loop
End of loop**

G10 Work shift



G33 Threading

The format is: **G33XnZnInKnAnCnPO**



- X The X axis location (as a radius) of the final pass of the cycle in G72 mode this is the final pass as a diameter.
- Z The Z axis location of the end of the thread
- I The starting incremental amount of material to be removed after the first pass.
This is to be defined as the diameter removal in diameter mode
- K The lead of the thread, amount per revolution. Note: there is no practical limit to lead; spindle rpm must be slow enough to limit feed rate to 300 ipm.
- A Used for tapered threading, it specifies the amount the X axis will move over the length of a tapered thread
- C Causes the infeed to be at an angle, the default is 29°
- P Used when you want the tool to keep traveling forward while it pulls out of the work. This will leave no undercut
- O Including the letter O makes a single pass at the finished depth

Notes:

Diameter or radius mode

The use of the threading cycles is the same for either diameter (G72) or radius (G73) mode. Only the the values of X will be different. The values will correspond to the mode.

Starting position in Z

The tool in most cases will be started at least .1" away from the start of the thread to allow the slide to get up to speed before it makes contact with the material. This number will vary depending on the spindle speed and the pitch of the thread. The courser the thread and faster the spindle speed, the farther away you will need to start. Under worst case conditions the slide can get up to full threading speed in about 1/2 revolution of the ball screw. In most cases this does not matter, however if you are threading from an undercut and the tool has very little room to ramp up to speed, this is very important. You will have to slow the spindle down until the thread gauge goes on.

Starting position in X

The tool should be positioned to take the first pass. The farther away you start the tool, the more passes will be needed. In production runs it pays to experiment a little for the best results and speed.

Depth of each pass: I

The control will start with removing the amount given as I. Then the control will automatically reduce the depth of the cut as the tool gets deeper. This is a fixed procedure that cannot be changed, it keeps the amount of material removed constant. Start the tool so that it takes a full cut on the first pass.

G33 Threading

Retraction position between passes:

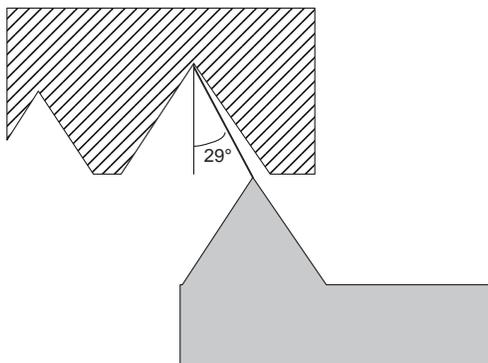
The tool will back away from the starting position plus two times the amount of I. Even as the tool gets deeper into the material it will always retract to the same point.

Pullout position in Z when using P option:

The tool will start to pull out at the location given in Z. It will travel beyond Z the same amount as it has to travel in X to reach the retraction position.

Angle infeed C option

If C is included in the G33 command the tool will feed in at an angle. This defaults to 29°. The maximum angle is 30° (based on standard 60° tool geometry) the min is 0°. If you wanted the tool to angle in at 27°, add C27 to the threading cycle command.



The single pass option O can be used for a cleanup pass:

When a single pass is needed write the same threading pass as used for cutting the thread. Just add a O to the command. Be sure to start the thread at the same point and at the same spindle speed. This option can be used with all variations of the threading command.

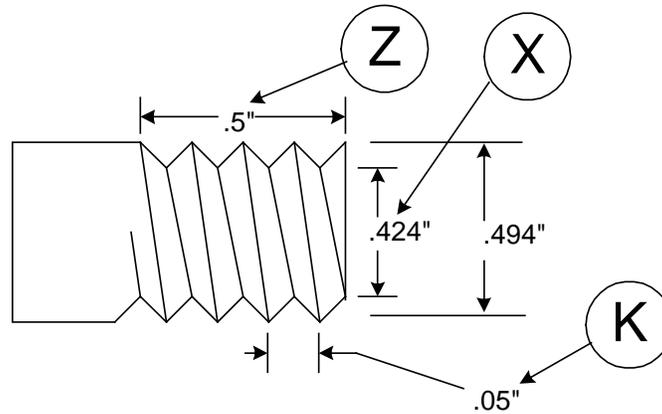
End of cycle position:

At the end of the threading cycle the tool will return to the starting point.

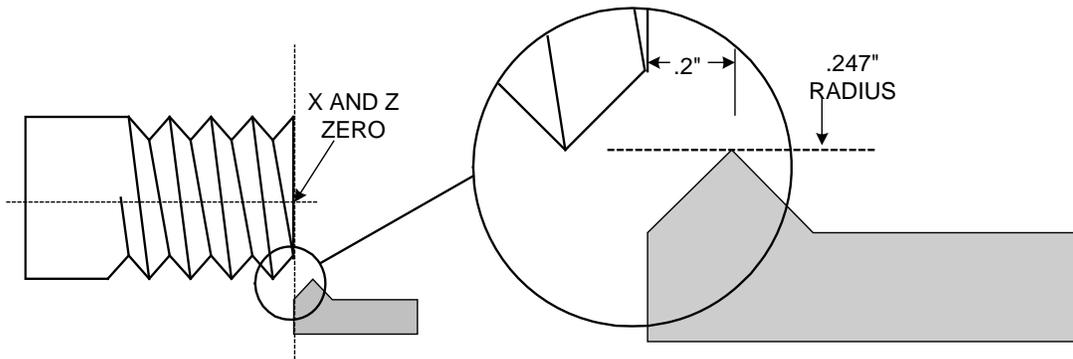
G33 Threading

Threading example

Example: A straight thread, Male, 20 pitch, Minor $\text{\O} = .424''$, Major $\text{\O} = .494''$, length of thread = $.5''$, and there is no undercut.



For this part the X zero is at the center of the part. The Z zero is at the face of the part.



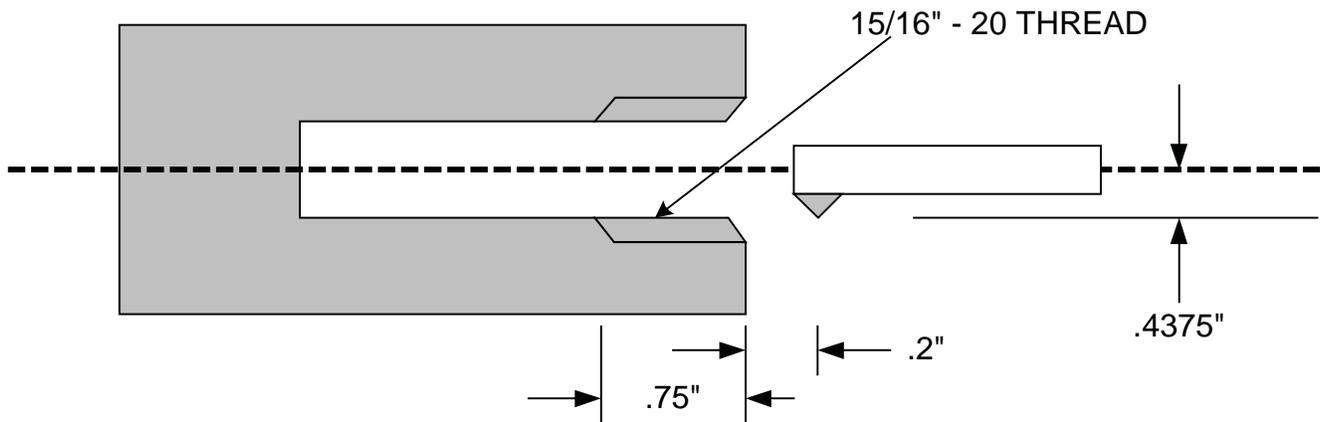
The locations shown above are given in radiuses. The starting location in X is the major diameter, less the amount of the first pass. In this case the major diameter is $.494''$, half of this is $.247''$. Subtract $.01$ to be taken for the first pass: $.237$

For the above example the program commands are (written in radius mode, G73):

T4F200 (Call the threading tool and position it at tool offset)
X0Z.2 (The offset puts the tool at the part center, $.2''$ from the face in Z)
X.237 (Positions the tool at the starting radius to cut the thread)
G33X.212Z-.5I.012K.05PC (Cut thread)

G33 Threading

Internal threading example



In the above example we will be cutting a 15/16-20 internal thread. (written in radius mode, G73):

T8	Call threading tool into position
X.4375Z.2	Give a value to the tool call location
X.4425	Move tool out to take a .005" cut on the first pass
G33X.46875Z-.75I.01K.05C	Threading cycle for internal thread

Threading, Single Pass

The format is: G33XnZnInKnAnCnPO

The single pass cycle is the same format as the regular cycle except it has the addition of the letter O in the command. The cycle will automatically move the tool to the finish depth in X and perform a single pass, then pull out to 3 times I in the direction from where the tool was before the command was started. When doing a single pass cycle to clean up a thread that has already been cut, copy the exact same code that you had in the original threading cycle and start the tool in the same spot as before.

It is possible to do single pass threading with the G33 command. The tool will follow the same lead that has already been cut as long as:

- the part has not moved in the collet
- the cycle is started in the same location in Z and X.
- the spindle speed has not changed
- If P or C has been used in the first threading cycle be sure to include it with the single pass

This is useful for:

Deburring: To deburr you can cut the thread and then use the turning tool to top the thread and chamfer the entry and exit of the thread. Then reposition the threading tool and take a clean up pass.

Control: The single pass technique can be used to have greater control over the amount to be removed with each pass.

G33 Threading

As an example we will go back and take a cleanup pass on the first external example. We will start the tool in Z at the same position as we did with the first cycle, .2". The X starting location will be the same diameter.

T4F300

X0Z.2

X.237

G33X.212Z-.5I.008K.05PCO

Call the threading tool and position it at tool offset

The offset puts the tool at the part center, .2" from the face in Z

Positions the tool at the starting radius to cut the thread

Cut a single pass thread

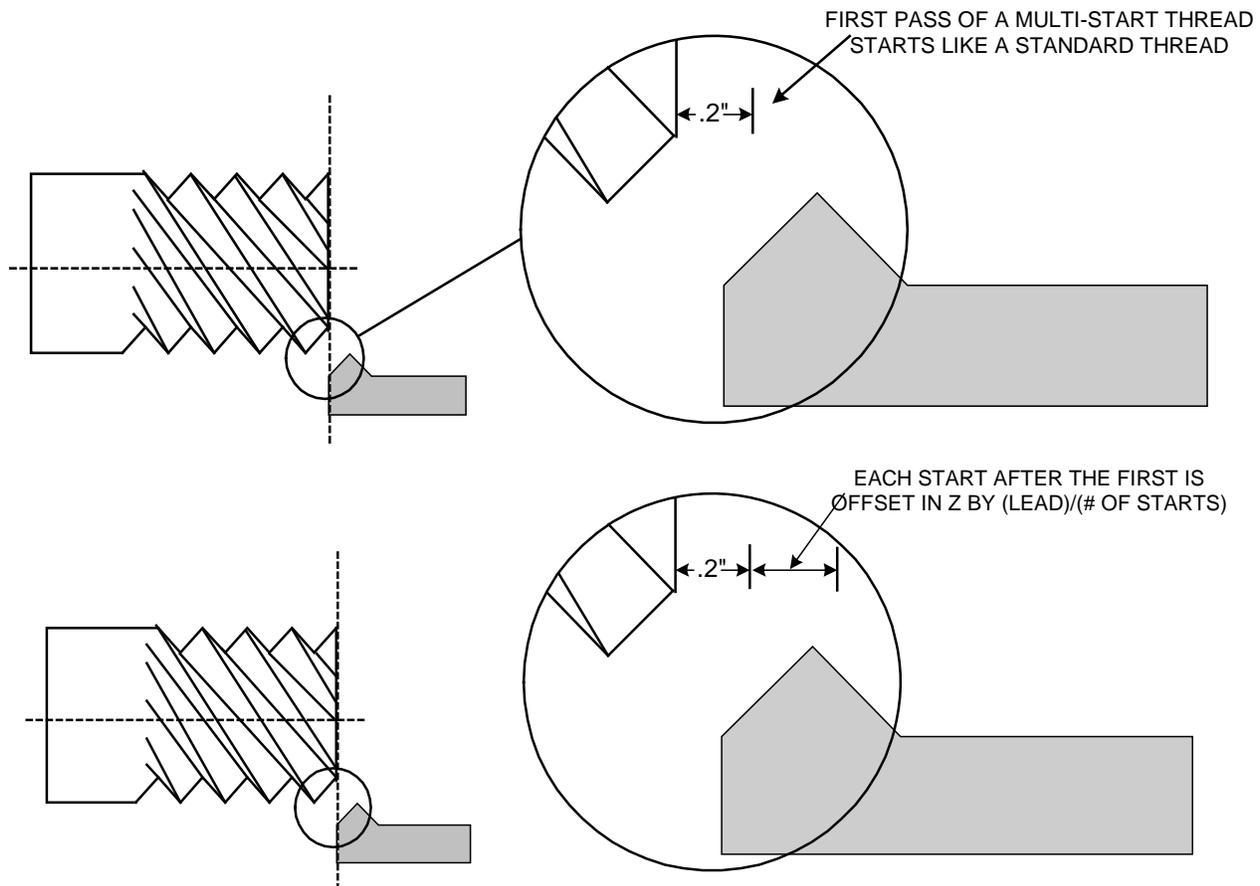
G33 Threading

Threading, Multi Start

Multi start threading can be done by using the regular G33 command. Each start has to be its own G33 command, ie. a three start thread would require three G33 commands. The differences between each of the commands would be:

Value for K:

This value would be multiplied by the number of starts. As an example if we had a three start 20-pitch thread, single-start value for K is .05". With the three-start the value is three times that, .150".



Value of Z:

The starting position of Z is offset for each start by (pitch)/(number of starts). If we have a three start thread the first start is at the normal .2" from the face of the material. The second start is offset by .2" + (.15/3) .25. The third start is offset by .25 + (.15/3) .3

Code for the above example:

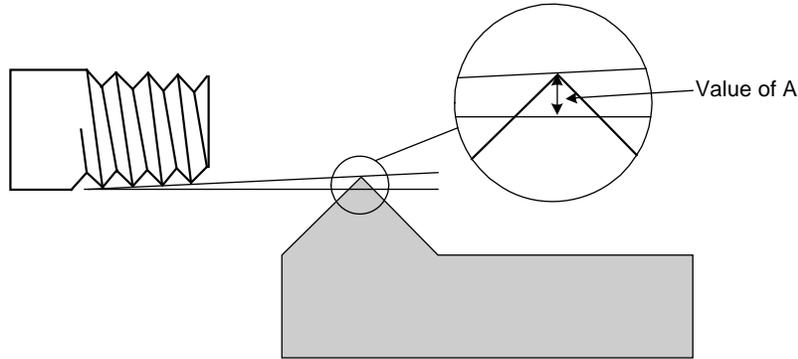
```
G90G94 F300  
T4  
X0Z.2  
X.247  
G33X.212Z-.5I.01K.15P  
Z.25  
G33X.212Z-.5I.01K.15P  
Z.3  
G33X.212Z-.5I.01K.15P
```

G33 Threading

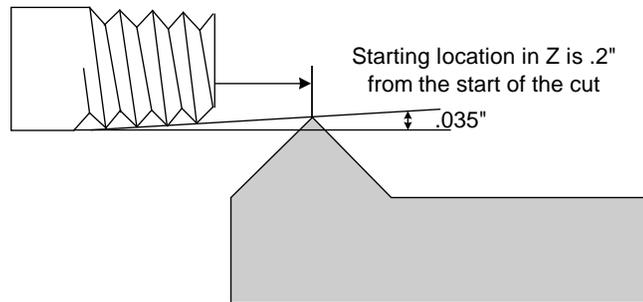
Threading, Tapered

G33XnZnInKnAnP

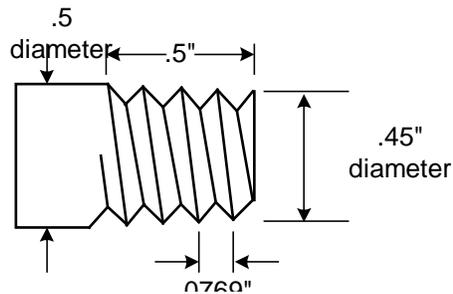
Tapered threading is done with the G33 command that includes an "A". This is the amount traveled in X over the distance traveled in Z.



Remember that A is the amount over the total distance traveled, this has to include the .2" used to get up to speed.



Example



g90g94f300g72

t5

x0z.2

x.415

g33 x.365z-.5i.005k.0769a.035

Move to start corrected for taper (remember .2 in Z)

Taper thread

This page intentionally blank

TOOL NOSE RADIUS COMPENSATION G41,G42, G40

Tool nose radius compensation

Notes on use:

When radii or angles are programmed and you need a very accurate reproduction, you have to take into account the size of the tool nose radius. Otherwise there will not be enough material removed in the area of the radius or angle. The tool nose radius compensation is very helpful when programming any moves that are not parallel to the axis's. With the G41 and G42 codes you can compensate for the size of the tool nose radius without any complicated computations. The amount of compensation can be changed by correcting a radius value stored with the secondary tool offset table. The direction of the offset correction is also done with the secondary tool offset values of X and Z.

Format

Right Compensation	G42
Left Compensation	G41
Cancel Compensation	G40
Compensation Value Location	Xn.nnnnZn.nnnnDn

G41 or 42	Specifies the type of compensation to be turned on
G40	Turns the compensation off
Dn	Is the secondary offset that stores the value of the tool nose radius value to be used. This value is taken from the R register in that offset table. This also can be used to shift the tool path to fit a previously completed path.

Sequence

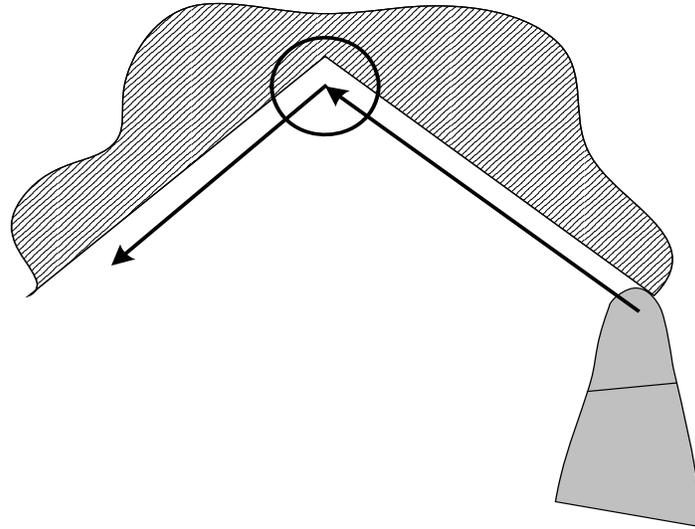
G42	Turn compensation on
XnZnDn	Move with secondary offset radius value used to turn on comp
XnZn	Move used to turn off compensation
G40	Turn compensation off

Rules

- The compensation must be turned on before a linear move, the command must be on a line by itself.
- The secondary offset (Dn) must be with a linear move on the line after either the G41 or G42
- The compensation must be turned off after a linear move, the command must be on a line by itself. To turn the compensation off put the G40 on the line after you make the move to clear the work. The turning off of the compensation will be done on this move. Be sure the move off the work is larger than the size of the tool nose radius being compensated.
- Compensation must be turned off before it can be turned on again. If you have to go from right to left compensation you must have a move off the part to turn one off before the other is turned on.
- The compensation can be used on all types of moves.
- The value of the R in the secondary offsets must be (+). It is the incremental value of the tool nose radius. ie: a .007" radius tool has a compensation value of .007

TOOL NOSE RADIUS COMPENSATION G41,G42, G40

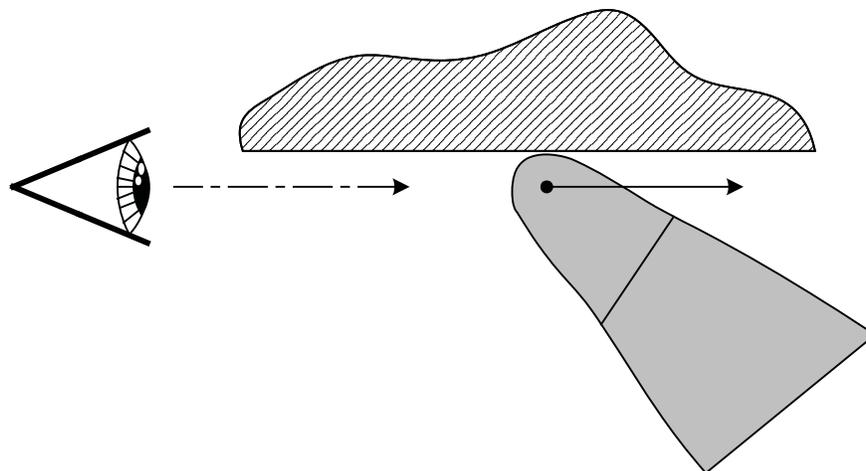
- Tool changes automatically turn off compensation
- Tool nose radius compensation can be used in either Radius (G73) or Diameter (G72) modes
- When the compensation is turned on or off the tool must be off the part by no less than the size of the radius being compensated. The clearance move off the part must be to a distance off the part by at least twice the TNR value.



When using the TNR compensation the tool path gets shifted off the finished size. This does not matter if the tool being used to take a finish pass is different than the roughing tool. The tool is shifted in the setup to give a correct finished size. If the same tool is used to do the rough and finish pass then the tool path must be shifted to correct for the error created with the TNR comp. Next is a sample of what would happen without correction for size.

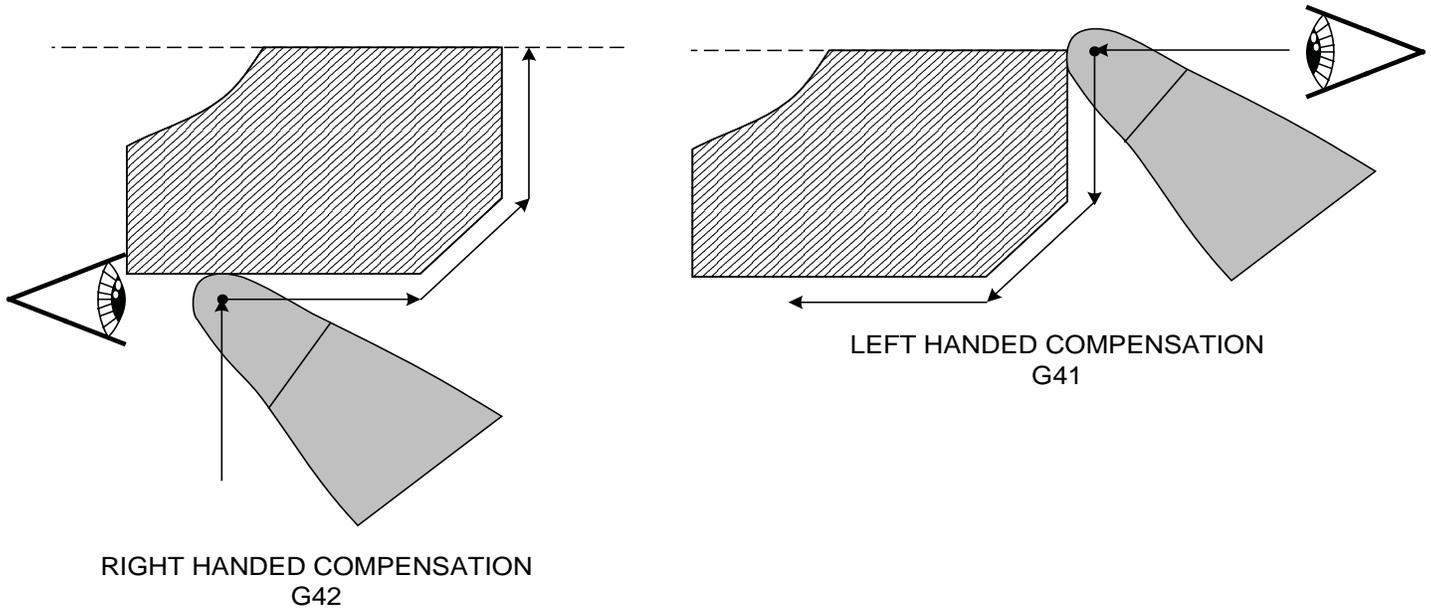
Right or left?:

The right and left compensations are based on the type of move you are performing, not the type of cutter. The type of compensation is described by looking at what side of the cut the center of the tool nose radius is. Imagine that you are sitting at the center of the tool nose radius, looking in the direction of the cut. The type of compensation that you have to apply is determined by whether the center of the tool is on the right or left of the material. In the following example you would want to apply G42 -right handed compensation:



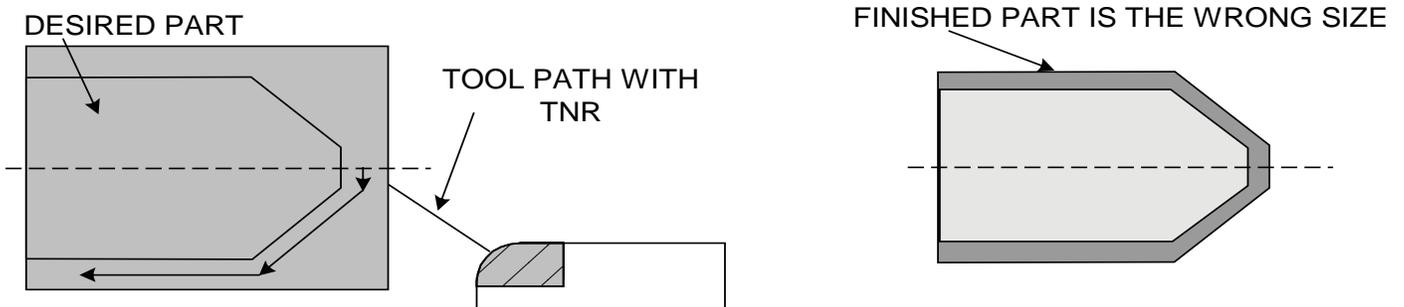
TOOL NOSE RADIUS COMPENSATION G41,G42, G40

In the following examples we use the same cutters, and the part geometry is the same. The only difference is the direction of the tool path:



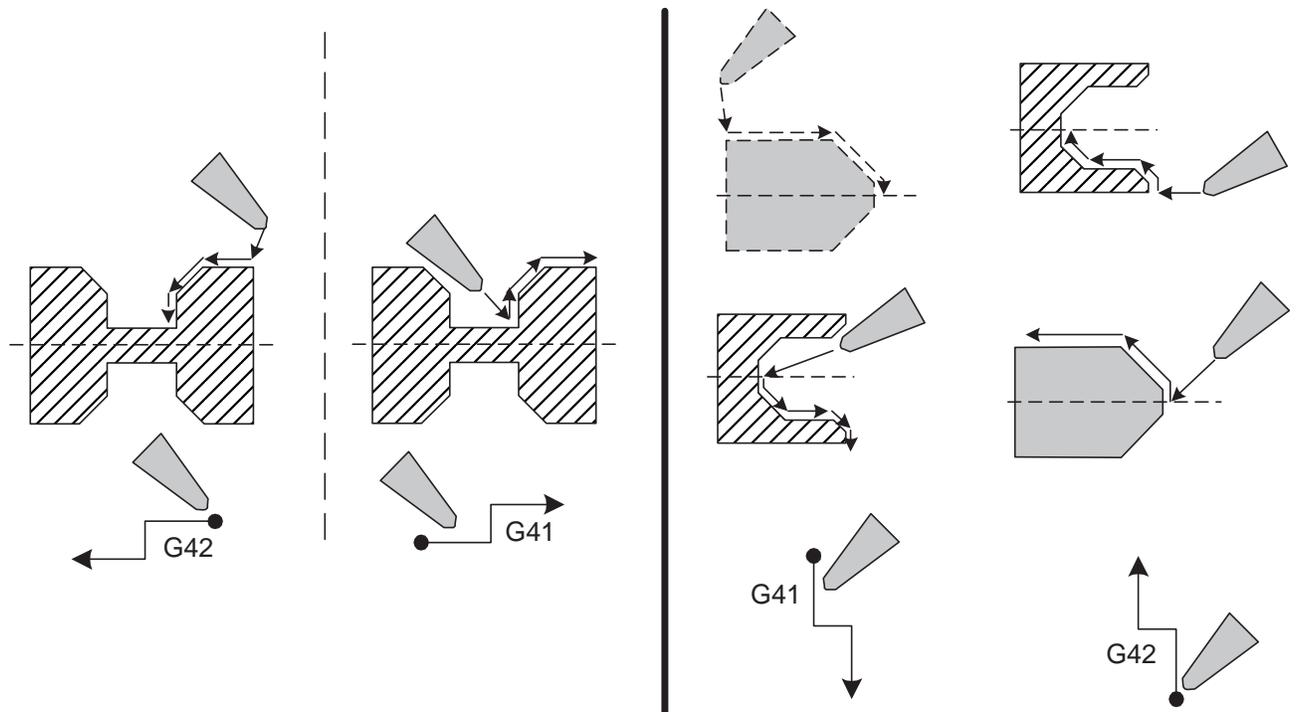
Shifting the TNR compensation

The direction of the correction will depend on the direction of the tool path and desired TNR compensation.

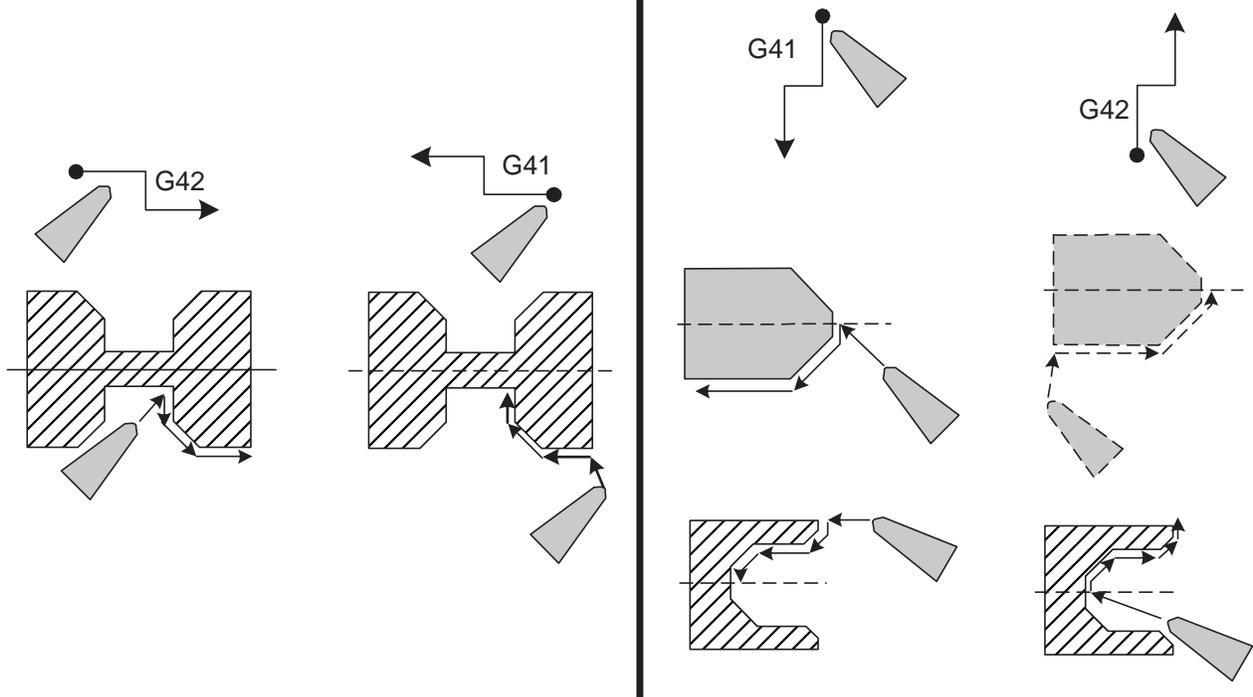


Notice the following table for the direction of the corrections to be added to the same secondary offset as the tool nose radius.

TOOL NOSE RADIUS COMPENSATION G41, G42, G40



<p>(+) X (+)Z</p>	<p>Correction needed in secondary offsets:</p>	<p>(+) X (-)Z</p>
<p>(-) X (+)Z</p>		<p>(-) X (-)Z</p>



TOOL NOSE RADIUS COMPENSATION G41,G42, G40

Setting the TNR value:

The value used for the compensation of the tool nose radius is stored in the secondary offset table. To enter a value in the table press F9 -SECCMP from the automatic page. This will bring up the secondary offset table:

1	X: +0.00000 Z: +0.00000 R: 0.00000	17	X: +0.00000 Z: +0.00000 R: 0.00000
2	X: +0.00000 Z: +0.00000 R: 0.00000	18	X: +0.00000 Z: +0.00000 R: 0.00000
3	X: +0.00000 Z: +0.00000 R: 0.00000	19	X: +0.00000 Z: +0.00000 R: 0.00000
4	X: +0.00000 Z: +0.00000 R: 0.00000	20	X: +0.00000 Z: +0.00000 R: 0.00000
5	X: +0.00000 Z: +0.00000 R: 0.00000	21	X: +0.00000 Z: +0.00000 R: 0.00000
6	X: +0.00000 Z: +0.00000 R: 0.00000	22	X: +0.00000 Z: +0.00000 R: 0.00000
7	X: +0.00000 Z: +0.00000 R: 0.00000	23	X: +0.00000 Z: +0.00000 R: 0.00000
8	X: +0.00000 Z: +0.00000 R: 0.00000	24	X: +0.00000 Z: +0.00000 R: 0.00000
9	X: +0.00000 Z: +0.00000 R: 0.00000	25	X: +0.00000 Z: +0.00000 R: 0.00000
10	X: +0.00000 Z: +0.00000 R: 0.00000	26	X: +0.00000 Z: +0.00000 R: 0.00000
11	X: +0.00000 Z: +0.00000 R: 0.00000	27	X: +0.00000 Z: +0.00000 R: 0.00000
12	X: +0.00000 Z: +0.00000 R: 0.00000	28	X: +0.00000 Z: +0.00000 R: 0.00000
13	X: +0.00000 Z: +0.00000 R: 0.00000	29	X: +0.00000 Z: +0.00000 R: 0.00000
14	X: +0.00000 Z: +0.00000 R: 0.00000	30	X: +0.00000 Z: +0.00000 R: 0.00000
15	X: +0.00000 Z: +0.00000 R: 0.00000	31	X: +0.00000 Z: +0.00000 R: 0.00000
16	X: +0.00000 Z: +0.00000 R: 0.00000	32	X: +0.00000 Z: +0.00000 R: 0.00000

Secondary offset :
Press Esc to exit offset adjustment screen
Press C to clear all Secondary offsets

First: Select a secondary offset number

Next: Enter the tool path correction. Enter the value with the correct sign. Refer to the previous table. If the value should be -use the sign. If the value is + just enter the value.

X value: Enter twice the value of the tool tip radius. i.e. if TNR= .007 enter .014

Z value: Enter the value of the tool tip radius.

Then: Enter the value of tool nose compensation, IE .007 and then press ESC

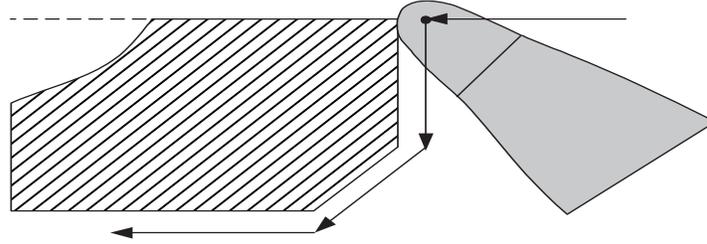
Changing a compensation value:

When you put a value in the R offset table it writes over the old value. So if you have a number already in the register that you want to use and it is not the value needed, all you have to do is enter the correct value. As an example if you have a value of .032 in the offset and you want to change it to .008, just enter the new value. You do not have to clear the register first. If you want to correct a value slightly, you must enter in the final value needed. ie: if you have .007 and want to increase it by .001 you must enter .008. **Do not enter -.001**

TOOL NOSE RADIUS COMPENSATION G41,G42, G40

Worked examples

In the first example a turning tool is used in one direction.



G90G94F300

M03S2000

T1 (LH turn tool with .015 tnr)

X0Z 1

Z.05

G95F003

G41

Turn on left hand tool nose radius compensation

X0Z0**D1**

Use the radius value found in secondary offset #1

X.22

X.25Z-.03

Z-.3

X.27

G94F300Z2

G40

Turn off the TNR compensation on the Z2 move

M30

Before running the program the setup person must make sure that there are the correct values in the secondary offset D 1.

$X = -.01500$ $Z = -.01500$ $R = .01500$

If the values are not correct, then clear them and enter new ones. Remember when entering the X value you must enter twice the TNR value, i.e. -.03 for the above example.

TOOL NOSE RADIUS COMPENSATION G41,G42, G40

Running a program that used Tool Nose Radius Compensation

When you write a program with TNR compensation there is another program that is created automatically that has all of the moves that make up the compensated program. When you run the program you will see extra moves in your program that you did not write. This is normal. If you run a program in single block mode you will see the newly created moves. You will not see the G40, G41, or G42 code in the executed program. There will be moves that get the tool ready and in place for the rest of the compensated moves. The values of the X and Z's will be changed to compensate for the TNR.

When you go to the editor you will be working on your original program. After you leave the editor the program will automatically be rewritten and stored so it is ready to run. Also every time you leave the secondary offset table the program will be rewritten to compensate for the new tool nose radius values given.

G70 - G71 Inch - Metric Modes

G70 (default) sets the control so that moves and feed rates are in “Inch” mode.

It is not necessary to use the G70 command to set the control to Inch Mode at initial turn-on, but if you run an “inch-mode” job after a “metric-mode” job you must turn the control OFF, then ON, to clear the G71. If your shop switches between modes often, it is best to include the G70 at the beginning of all Inch programs, and G71 at the beginning of all Metric programs to avoid errors.

G71 sets the control so that moves and feed rates are in “Metric” mode. The Position Counter changes from five-place display to four-place, and the feed rates are prefixed by “MM” indicating millimeters per minute (G94 mode) or millimeters per revolution (G95 mode).

NOTE: Use the G71 command at the beginning of the program before you call any tools.

G72 - G73 Diameter and Radius Programming

G72 sets the control so that X moves will now be read as diameter values. This follows more closely to the Fanuc style format.

G72 Diameter Programming Mode

The G72 command is modal. Once it is used this mode stays active until it is turned off with a G73 command. The default mode is G73 -radius. So be sure to use the G72 command in your program if you want to program in the diameter mode.

NOTE: Use the G72 command in the beginning of the program before you call any tools.

NOTE: If you plan on using SEARCHTO (F6) in the automatic page be sure to put a G72 after the tool call statement. The search to command will not pick up the call out for the diameter mode at the beginning of the program. If you have it on the first line after the tool call it will work, IE:

```
T1  
G72X.5Z.1
```

G74 - Box Roughing Cycle

G74 is a box roughing cycle where a rectangular area of material is removed in many passes.

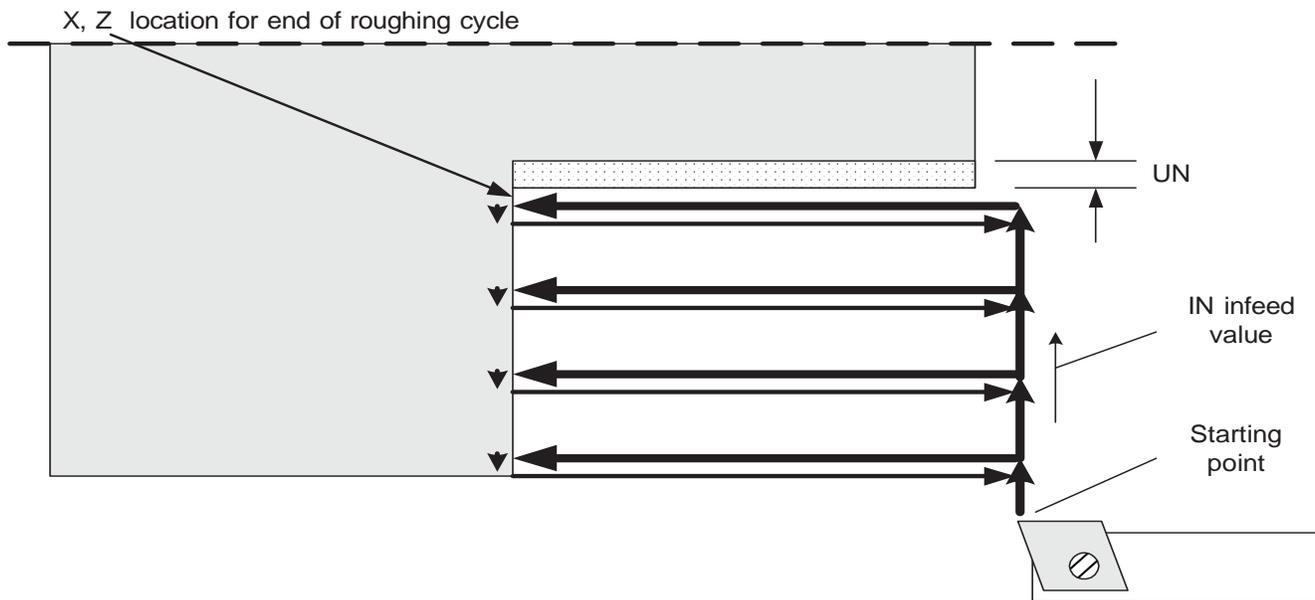
G74XnZnlUnFn

X and **Z** is the corner of the box area to be cleared out

In is the maximum amount to be roughed per pass, defined as the depth of cut per side

Un amount of material to be left by the cycle for a finish pass in X only.
(depth of cut, as a radius)

Fn is the feedrate



The box cycle starts at the current position, then makes cutting passes parallel to the Z axis at a cutting depth no greater than the I ending at X, Z. At the end of the cycle the tool is returned to the start point.

If you want to leave material for a finish pass the X and Z values must be offset for this.

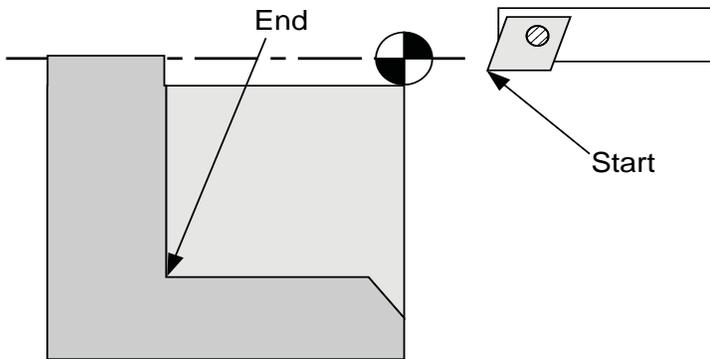
The feedrate is IPM (G94) or IPR (G95), depending on the mode when the cycle is started.

The X, Z coordinate may be absolute or incremental, based on the current mode of the control.

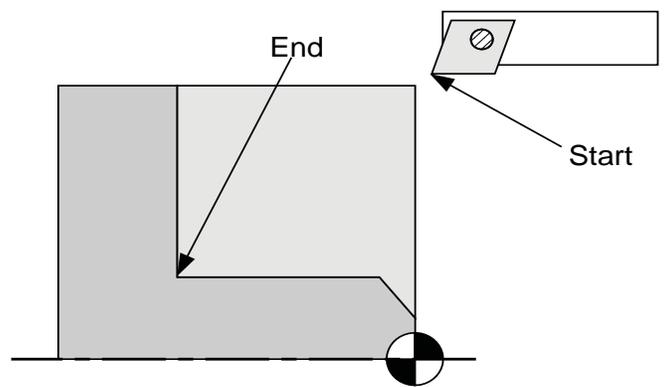
The return passes are at a fixed clearance distance (.02") from the last cutting pass.

G74 - Box Roughing Cycle continued

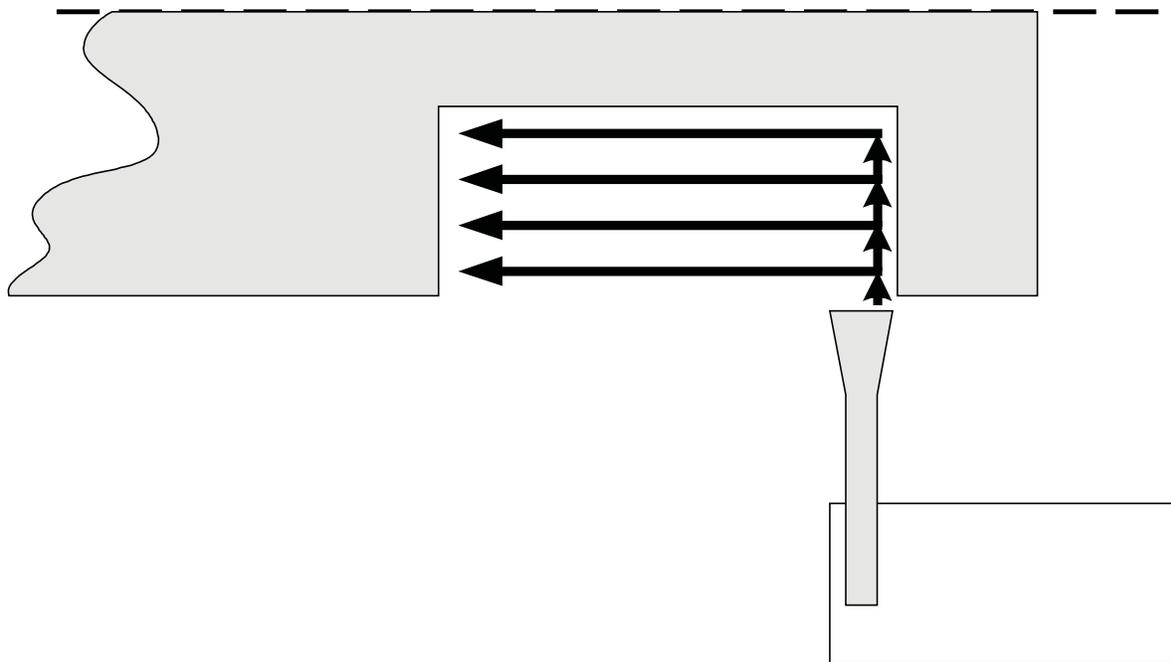
The G74 cycle can be used for either internal or external removal. It can also be used from the front (x+) or back (x-) of the part.



Internal example



External example from the back



Example behind a shoulder

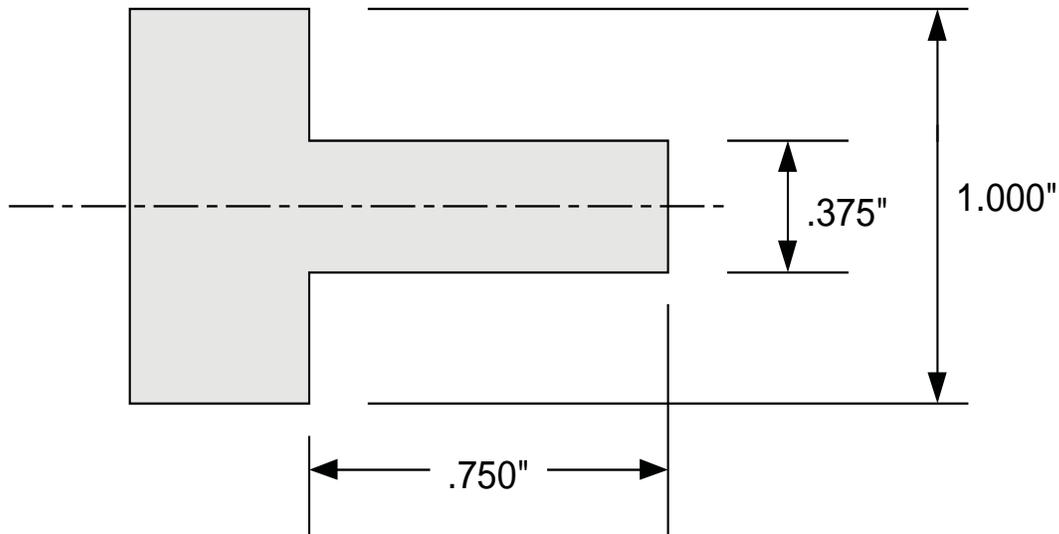
The G74 cycle can be used in radius (G73) or diameter (G72) mode. In both cases the In value is the amount to be taken off on a side. An In value of .05 would take .1 off the diameter of the part with each pass.

Tool nose radius compensation **cannot be** active when using this box roughing cycle.

G74 - Box Roughing Cycle continued

Worked example for G74

In the following example we will rough the 3/8" stem out from the solid 1" diameter bar. The G74 statement is written to leave .005" on X surface and .003" on the Z surface for a finish pass. The material to be left on the Z axis must be done with the end location value in the G74 code. The finish pass is not shown on this example.



G90G94F30OG72

M03S2000

T1(LH Turn tool for roughing)

X1.05Z.5

Z.1

G95

G74X.375Z-.747I.075U.005F.003

programmed to leave .005 on X and Z for finish cut

M05

G00Z 1

X-1

M30

G75 - Box Contour Roughing Cycle

G75 is the start of a box contour cycle. This cycle serves to rough out an area rounded in part by a contour defined in the part program

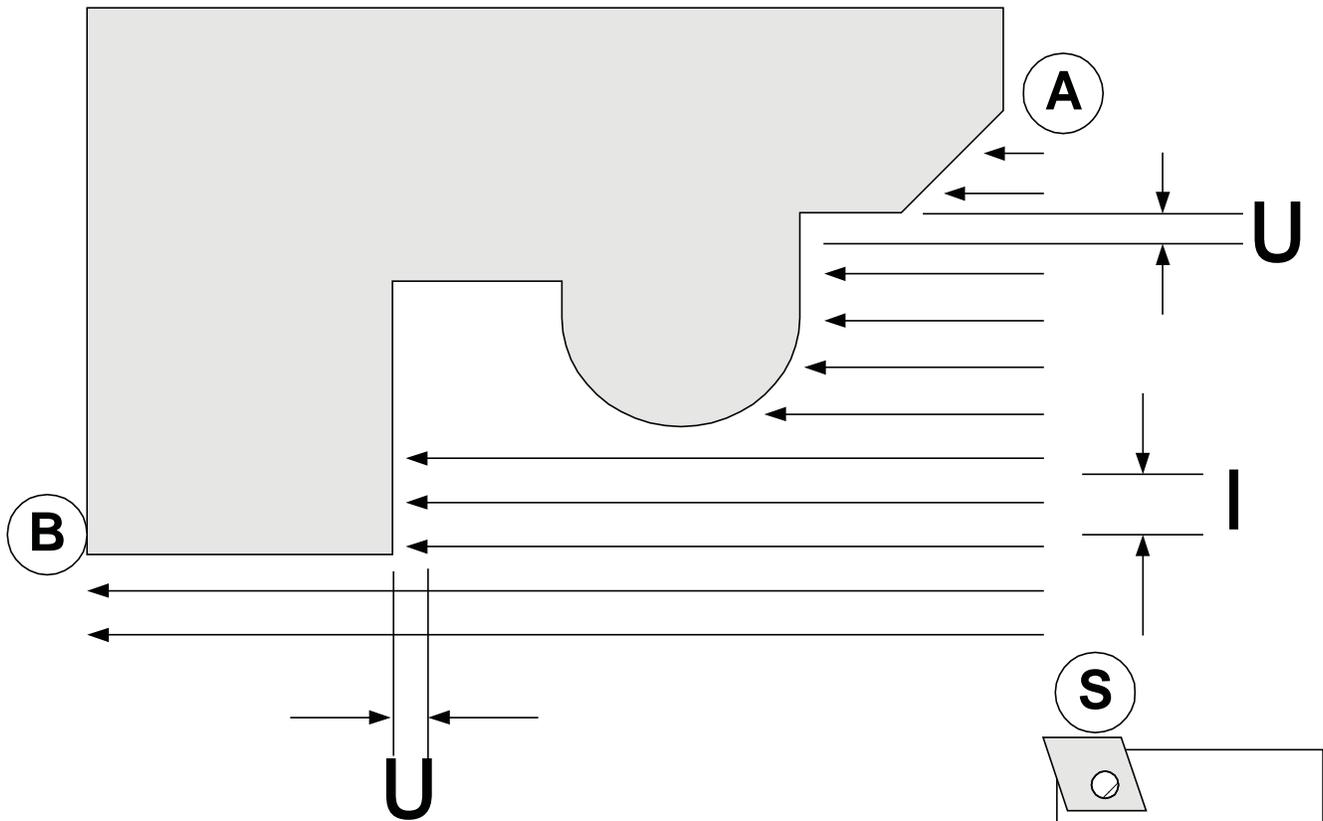
G75InUnFnPn

Un is the amount to be left on the part for the a finish pass

In is the maximum amount to be roughed per pass, defined as the depth of cut per side

Fn is the feedrate

Pn (optional) is a subroutine number



The box cycle starts at the current position, then makes cutting passes parallel to the Z axis at a cutting depth no greater than the I value until the last pass which is a U amount off the part. The area of material that is removed is rounded by the Z and X axis through the tool start point and the contour from A to point B. At the end of the cycle, the tool is returned to the start point.

The location of the contour code to rough to:

- If the P code is present, the contour is defined in a subroutine n.
- If the P code is not present, the contour is defined in the following blocks until a block with an "RF" code is encountered

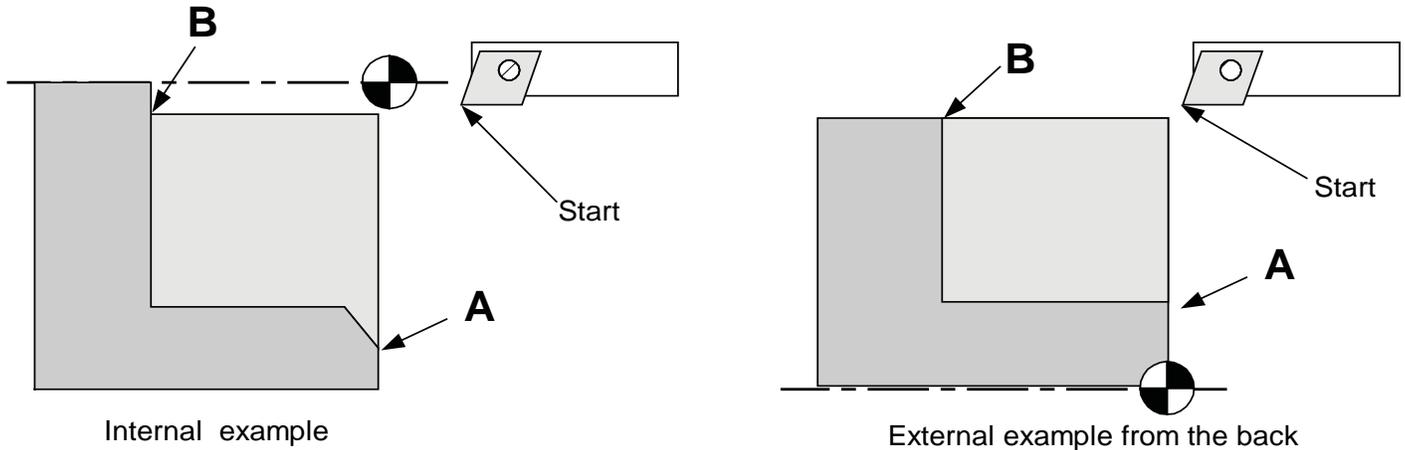
G75 - Box Contour Roughing Cycle continued

The feedrate is IPM (G94) or IPR (G95), depending on the mode of the control when the cycle is started.

The RF code must be on a line by itself.

The return passes are at a fixed clearance distance (.02") from the last cutting pass.

The G75 cycle can be used for either internal or external removal, and from the back.

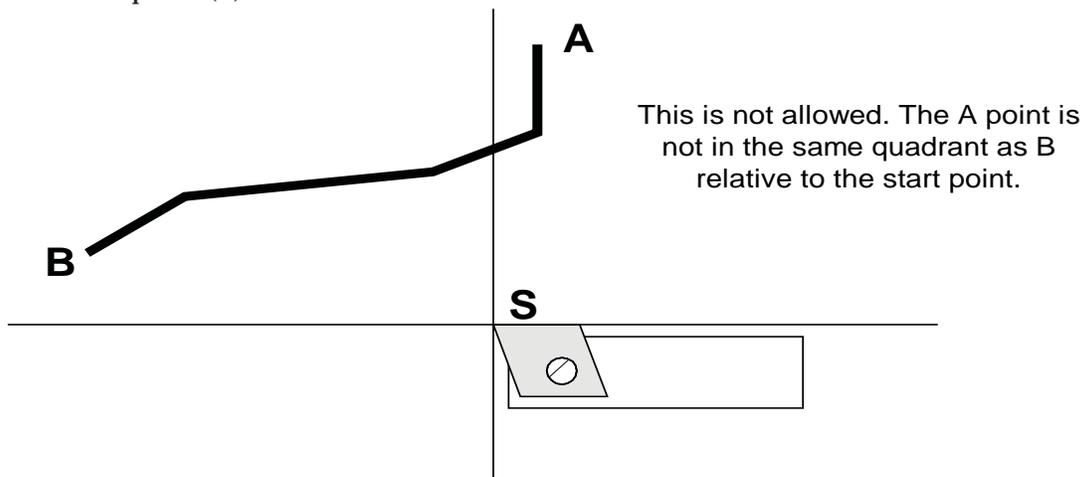


The G75 cycle can be used in radius (G73) or diameter (G72) mode. In both cases the In value is the amount to be taken off on a side. An In value of .05 would take .1 off the diameter of the part with each pass.

Tool nose radius compensation can not be active when using the roughing cycle.

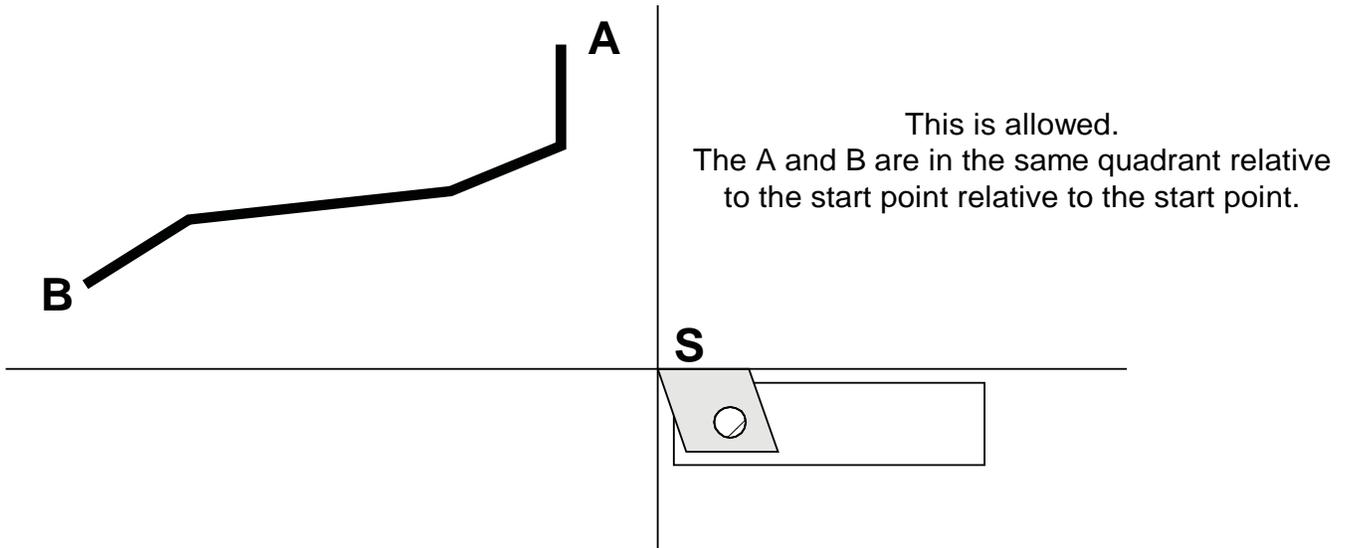
In calculation of the contour, the control honors those codes which affect the geometry of the contour (presets, absolute/incremental, etc.) but ignores any other information which might be present (T & D codes, M codes, feedrates, etc.)

An error will be declared if the start and end point of the contour (A & B) are not in the same quadrant relative to the start point (S)

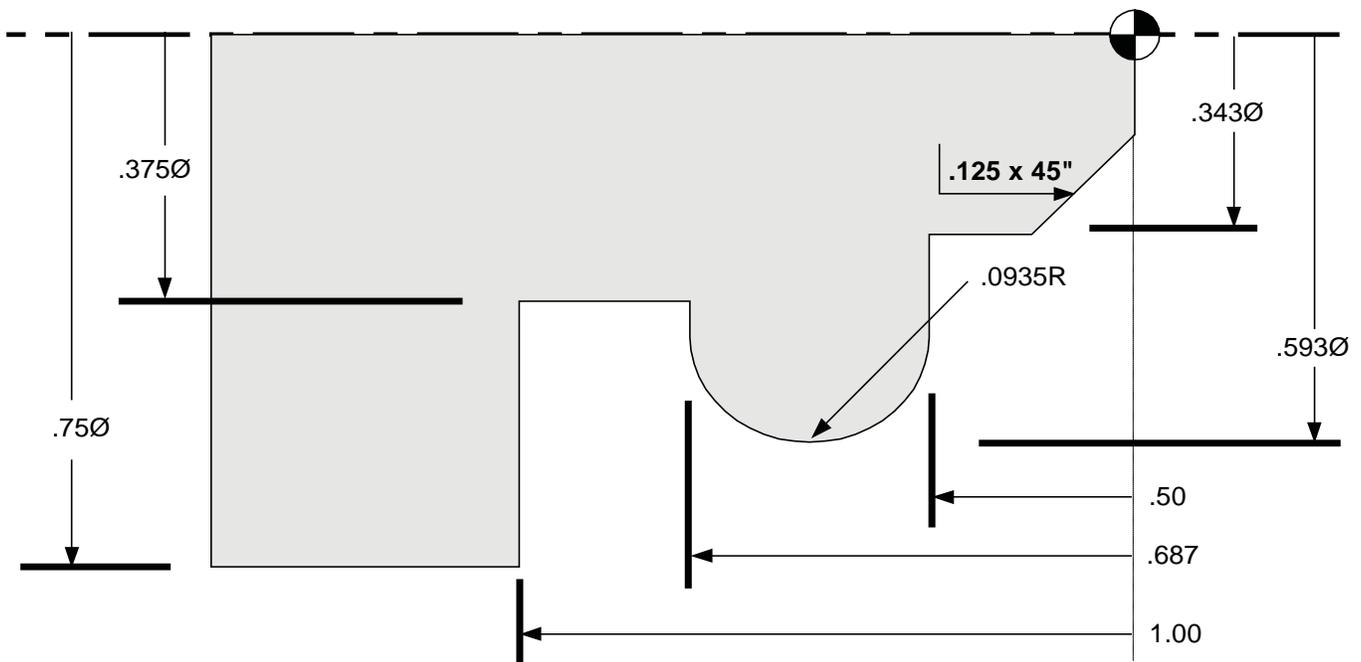


If the start point is moved over to include both A and B the cycle will work.

G75 - Box Contour Roughing Cycle continued



Worked external examples for G75



Above is a part that can be roughed. The part is defined in radius values in X. The code that can be used to generate the contour could be:

```
X0Z0
X.343C.125
Z-.5
X.406
G02X.406Z-.687R.0935
X.375
Z-1
X.75
```

The first example has the finished contour to be roughed immediately after the G75 command. This program would just rough the part and stop.

G75 - Box Contour Roughing Cycle continued

Written in Diameter mode

```
G90G94F300G72
M03S2500
T 1(LH TURN TOOL)
X0Z1
X.8Z.1
G95F003
G751.05U.02E.003
X0Z0
X.343C.125
Z-.5
X.406
G02X.406Z-.687R.0935
X.375
Z-1
X.75
RF
M05
G00Z 1
X-1
M30
```

Written in Radius mode

```
G90G94F300G73
M03S2500
T 1(LH TURN TOOL)
X0Z1
X.4Z.1
G95F003
G751.05U.02E.003
X0Z0
X.1723C.125
Z-.5
X.203
G02X.203Z-.687R.0935
X.1875
Z-1
X.375
RF
M05
G00Z 1
X-. 5
M30
```

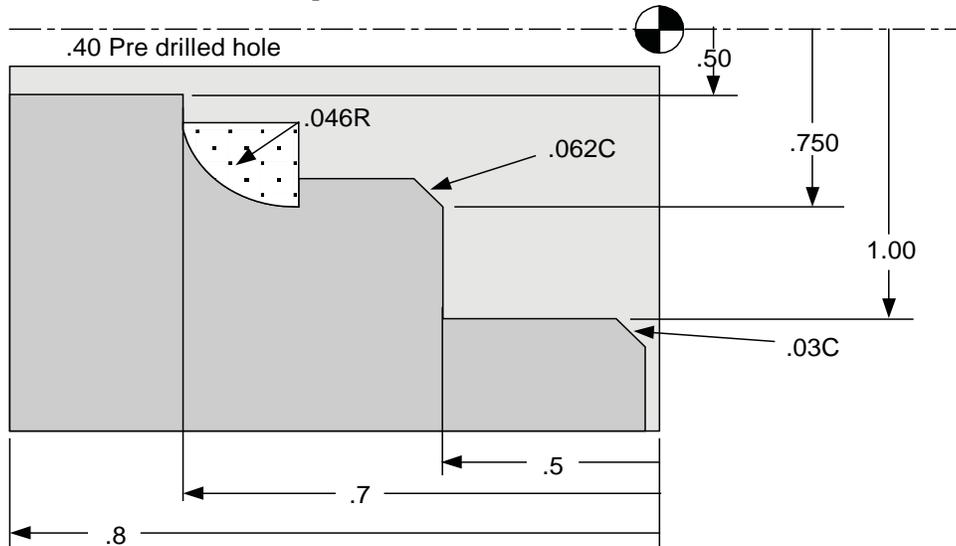
In this example the contour code is given as a subroutine. You can use the same subroutine for the G75 and G78 roughing passes. If you have to make a correction to the contour, it can be done in one place instead of for each cycle. You can also use it for the finishing pass if you are not going to use TNR compensation. If you want to use the compensation then you must leave the finish pass out of the subroutine since they will not accept the compensation.

```
G90G94F300G72
M03S2500
T 1(LH TURN TOOL)
X0Z 1
X.4Z.1
G95F003
G751.05U.02E.003P1
M05
G00Z1
X-1
M30
}1
X0Z0
X.343C.125
Z-.5
X.406
G02X.406Z-.687R.0935
X.375
Z-1
X.75
M99
```

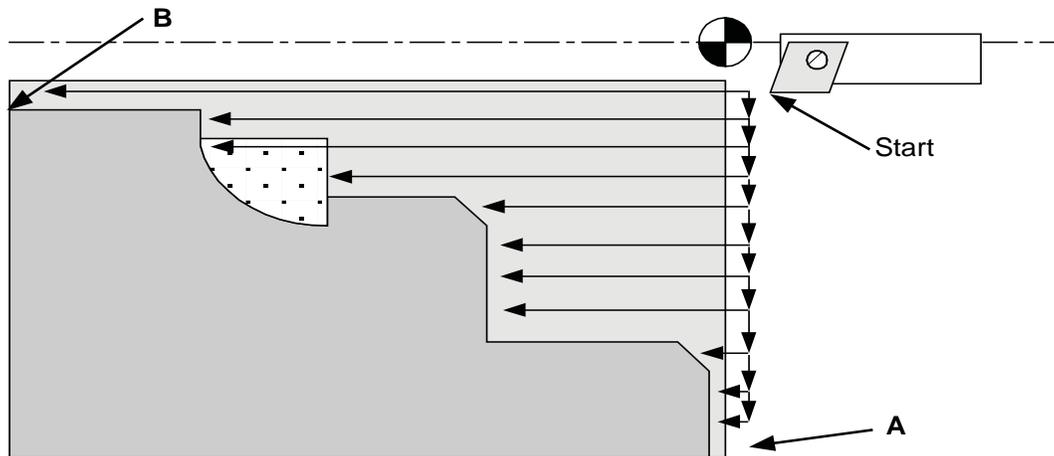
G75 - Box Contour Roughing Cycle continued

Worked internal example for G75

In this example there is a blank with a predrilled .4" hole.



Notice the starting point is at the minor diameter of the finished bore, and the A and B points are at the starting and ending points of the finish contour. Also notice that the finish contour does not use C for the .03" chamfer. If this were needed in this box cycle you could add a facing move and chamfer to the contour pass.



```
G90G94F300G72
M03S2000
T1
X.4Z.1
G95F003
G751.05U.02F003
X 1.2Z0
X1.06
X 1 Z-.03
Z-.5
X.75C.062
Z-.704
G03X.658Z-.75R.046
X.5
Z-1.
RF
G00Z 1
M30
```

G78 - Rough Contour Cycle

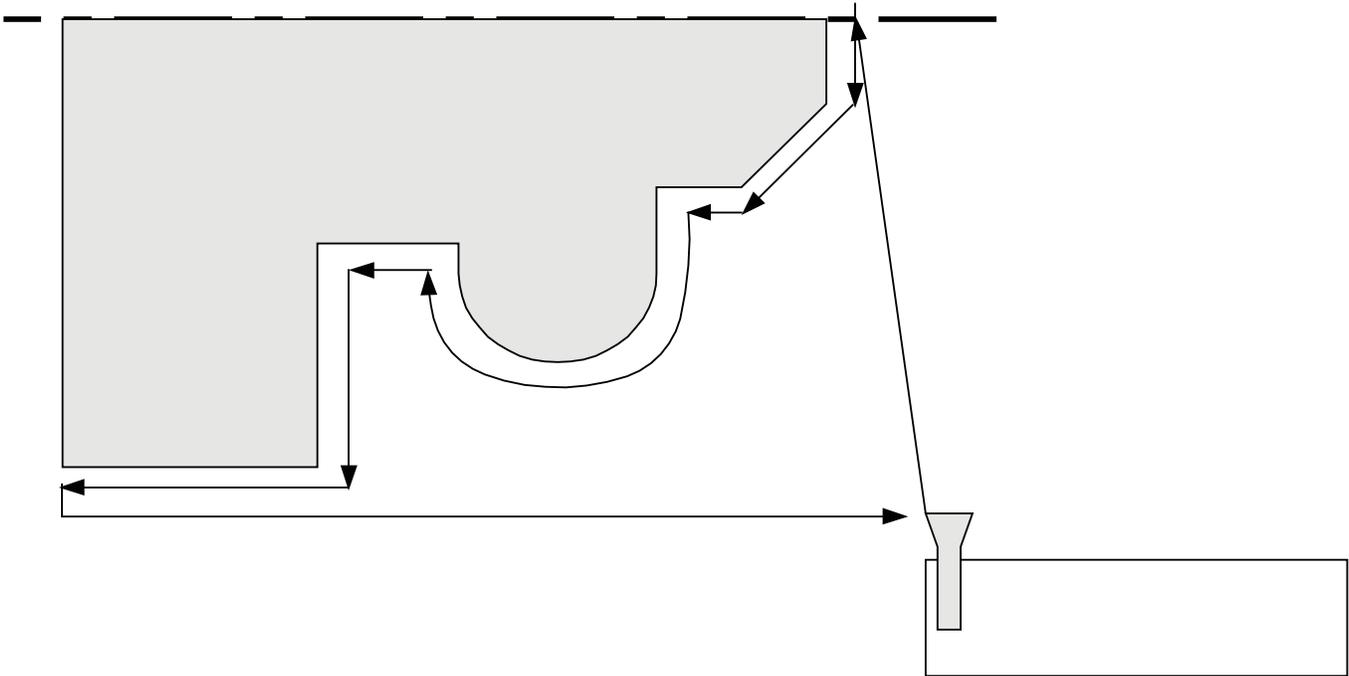
G78 is the start of a rough contour cycle. This cycle serves to rough a contour based on a section of program code describing a finish contour.

G78UnFnPn

Un is the amount to be left on the part for the a finish pass, (amount per side)

Fn is the feedrate

Pn (optional) is a subroutine number



The box cycle starts at the current position, then makes a cutting pass parallel to the final contour, but away from the part at a distance Un . At the end of the cycle, the tool is returned to the start point.

The rules and general usage commands are the same as with the G75 command. Please refer to that command for these notes.

Note about finish passes, subroutines, and tool nose radius: Tool nose radius compensation can not be used within a subroutine. So if you need to use TNR compensation for the finish pass you will have to copy the contour pass into the program for the last pass.

G78 - Rough Contour Cycle continued

Worked example for G78

The following code is a finish pass for the same example for G75. Please refer there for the part layout. In this example we are using a different tool to take the finish contour pass, T2.

```
G90G94F300G73
M03S2500
T2(LH Finish TURN TOOL)
X.8Z.1
G95F003
G78U.01E.003
X0Z0
X.343C.125
Z-.5
X.406
G02X.406Z-.687R.0935
X.375
Z-1
X.75
RF
G00Z 1
M30
```

The same example shown using two tools, three passes with two types of cycle and subroutines.

The subroutine is used so that there is only one contour pass in the program. If there are any changes to the finish pass they can be done in one place and then the rest of the program is also changed.

```
G90G94F300G73
M03S2500
T1(LH TURN TOOL)           Call roughing tool
X0Z1
X.14Z.1
G95F003
G75I.05U.04F003P1         Box rough cycle
G00Z1
T2(LH FINISH TURN TOOL)   Call finish tool
X0Z1
X0 Z.1
G95F003
G 78 U.01 E.003 P1       Rough contour cycle
G00Z.1
X0
G95F.002
```

.....continued on next page

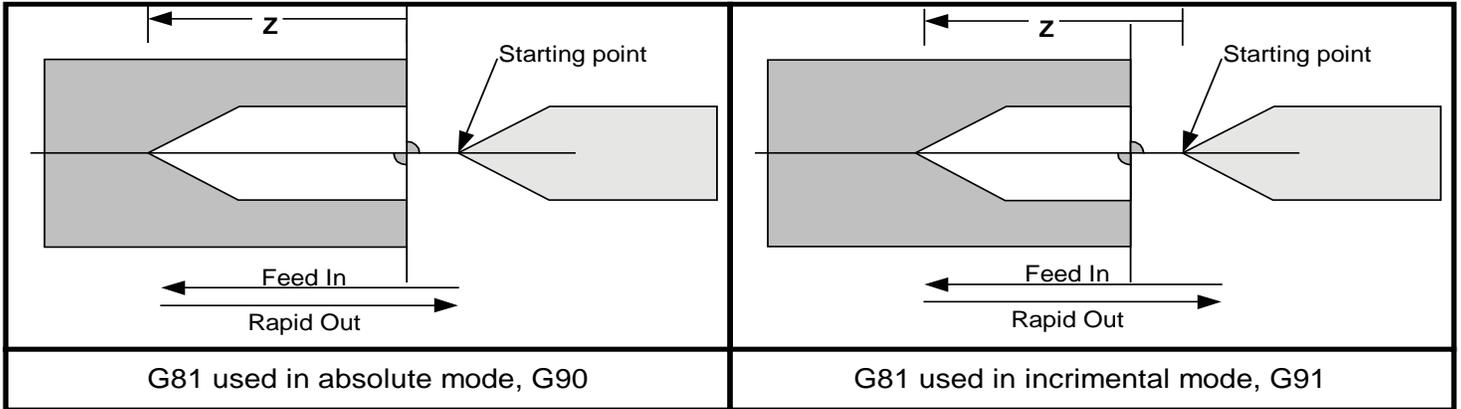
G78 - Rough Contour Cycle continued

P1 Finish pass to depth with same tool
G00Z1
X-1
M30
}1
X0Z0
X. 343C.125
Z-. 5
X.406
G02X.406Z-. 687R. 0935
X.375
Z-1
X.75
M99

G81 Drill Cycle

G81 is a one shot command. It is used to feed to a drill a specific distance in Z and then rapid back to the starting point. The format is:

G81 Zn Fn

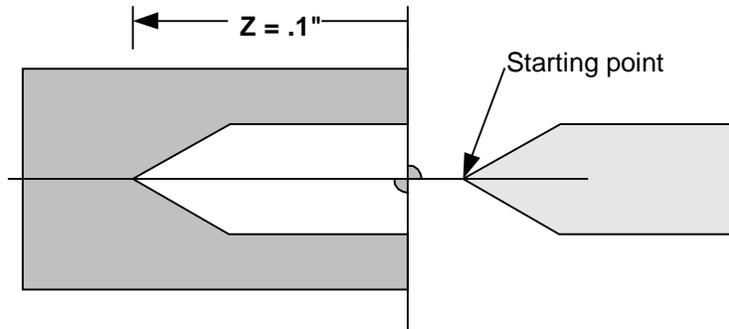


In G90, absolute mode: **Z** specifies the end of the hole from absolute Zero.

In G91, incremental mode: **Z** specifies the distance the tool will travel from the starting point.

F is the drilling feedrate in inches per rev or minute depending on if you are in G94 or G95.

G81 Example



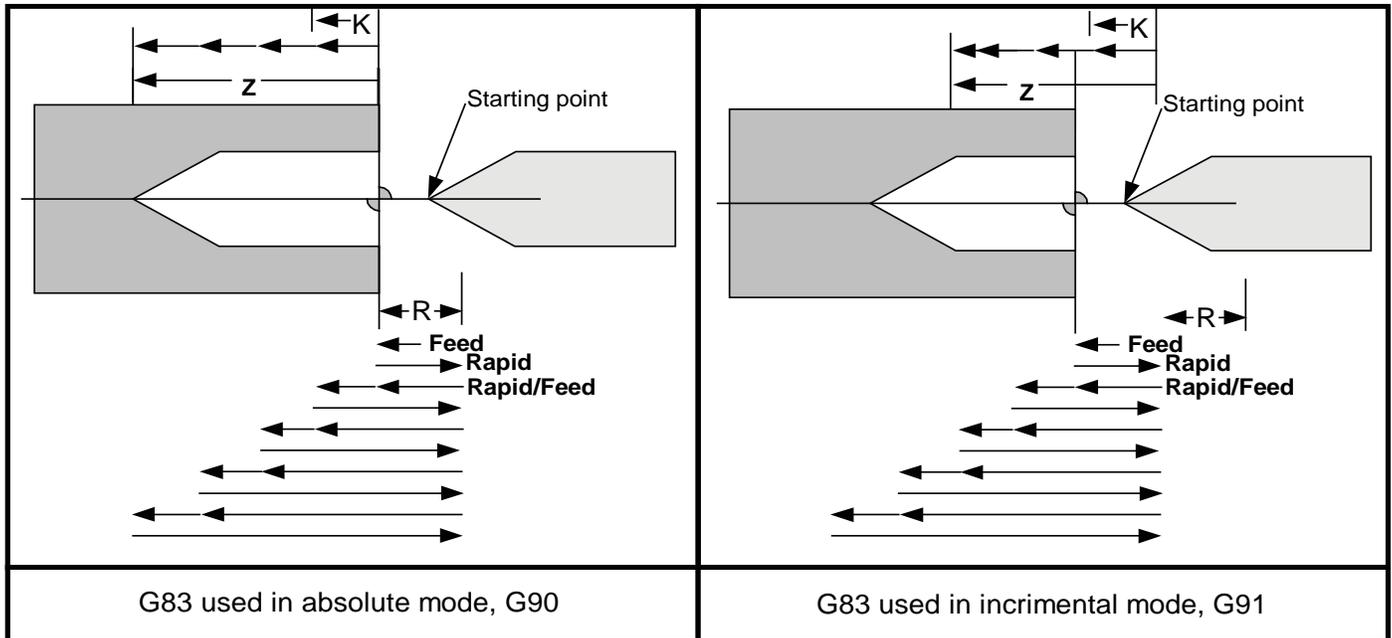
To drill a part .1" deep at a feed of .005" per revolution. The program would be:

```
G90 G94F300      Puts the control into absolute mode
TI              Calls tool #1 offset
X0Z.01          Positions tool at x= 0, Z= .01
G95             Sets ipr mode
G81Z-.1F005     Drills the hole .1" deep at .005 ipr.
```

G83 Peck Drill Cycle

G83 is a one shot command. It is used to peck drill to a specific distance in Z and then rapid back to the starting point. The format is:

G83 Zn Kn Fn Rn Ln Cn



In G90, absolute mode: **Z** specifies the end of the point of the hole from the part Zero.

In G91, incremental mode: **Z** specifies the distance the tool will travel from the starting point.

Start location: Position the drill where you want the first drill peck to start. After the first peck the drill will rapid out to the **R** location, and then back to where it started less the **C** value.

K specifies the depth of cut per peck.

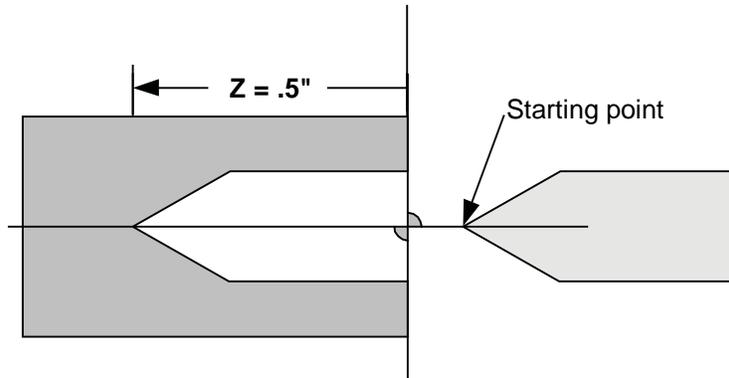
F is the drilling feedrate in inches per rev or minute depending on whether you are in G94 or G95.

R is the retraction plane, the tool will rapid back to this location at the end of each peck. **Default is the starting point of the cycle**

L is the rapid travel feedrate for the retraction move, noted in **IPM. Default is 200ipm**

C is the clearance distance left when the drill returns to the cut. **Default is .02"**

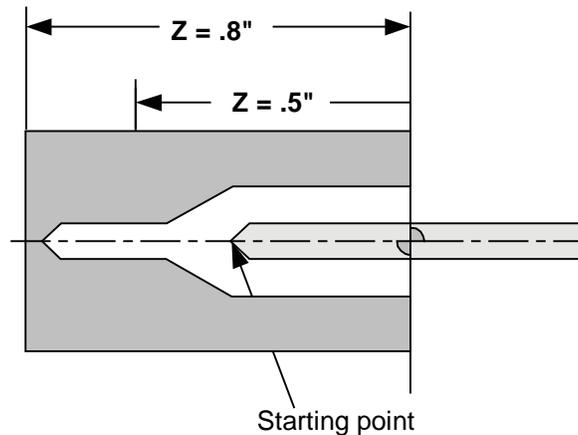
G83 Peck Drill Cycle



To drill a part .5" deep at a feed of .003" per revolution, and .1" pecks. The program would be:

G90 G94F300	Puts the control into absolute mode
T1	Calls tool #1 offset
X0Z.01	Positions tool at x= 0, Z= .01
G95	Sets ipr mode
G83Z-.5K. 1F003R.5C.05L300	Z Drills the hole .5" deep
	F at .003 ipr
	K with .1" pecks
	R retract to .5 in Z to clear the chips
L	peck moves done at 300IPM
	C return to the cut less .05" after peck

In the next example we have added a second drill that will peck drill a smaller hole at the bottom of the first. Notice that the drill will start a little off the bottom of the first hole and then retract clear off the hole to remove chips and get coolant before the next peck.



G90, G91 - G92 - G94, G95

G90 and G91 Absolute and Incremental mode selection

G90 and G91 set the mode of operation of the control. These commands are used in the program. Once one of these commands are used it stays in that mode until it is changed. There should be one of these commands in the first line of your program.

G90 -Absolute mode: distances given will move the tool relative to an absolute zero.

G91 -Incremental anode: distances given will move the tool relative to where the tool is.

NOTE: You will find that almost all (if not all) of your programs will be done in G90.

G92 is a position preset.

This is not a move command. When this command is executed the tool does not move, it changes where the control thinks it is.

After a tool change, a G92 can be used to set the location value to what you want the tool to be moved to. After the tool is called and it is in position use:

G92 X0 Z.2 This will set the position display to X= 0, Z.2

The G92 command does not effect the reference zero, "HOME". Even after you preset a location with the G92 the control knows where the reference zero is.

NOTE: If the tool is not starting at X= 0 then the values will be either radius(G73) or diameter(G72) depending on the mode you are in.

G94 and G95 Feedrate selections

These set the feedrate mode. After a feedrate is selected, it remains until it is changed.

G94 -feed set at inches per minute

Slowest: .00005" per minute

Fastest: 300" per minute

G95 -feed set at inches per revolution

Slowest: .00005" per revolution

Fastest: depends on spindle speed, the limit has to be calculated to 300 ipm

As an example, the fastest feedrate at 3000 rpm is. 1"/rev

Constant Surface feet spindle speeds - G96,G97,G77,G76

To use the following codes the OmniTurn must be equipped with a spindle control package. There are two types of spindle speed control modes that the OmniTurn control can use:

Spindle speed in RPM -(G97). In this mode the S value will set the spindle speed in turns per minute, "RPM". The speed will stay at this value until it is changed. If the spindle is turned off and then back on in the program the speed will still be the previously set value.

This mode is good for drilling and fixed spindle speed operations.

Constant Surface Feet -G96. In this mode the S value will set the amount of surface feet the tool will see. The speed of the material passing the tool will stay constant, no matter what the tool's distance from center is. As the tool gets closer to center the speed of the spindle will increase. Many tool and material suppliers give suggested feeds and speeds in terms of surface feet. This mode is good for turning and facing operations. **(See notes on use below)**

Minimum spindle speed -G76: Sets the minimum spindle speed, G76Sn.

Maximum spindle speed -G77: Sets the Maximum spindle speed, G77Sn

Notes: The default spindle speed mode is G97, RPM mode.

•M03, M04, and M05 operate the same for both modes of spindle control

Important Note

Notes on use:

The constant surface speed control is not intended to be turned on at the beginning of the program and then left on. If you do this the spindle speeds will vary greatly every time the machine moves! This will create excessive wear on the spindle motor and drive. Turn the constant surface feet mode on just after the tool has been positioned for the cut. Estimate the spindle speed that the CSF mode will start at and have the spindle turned on before you make the positioning moves. After the cut has been finished turn the constant surface feet mode off. Then use RPM commands. **DO NOT LEAVE THE G96 ACTIVE FOR TOOL CHANGES.**

Simple formulas to convert these values are:

$$rpm = \frac{12 * sfm}{3.14 * dia} \qquad sfm = \frac{3.14 * dia * rpm}{12}$$

Sample program showing constant surface feet:

```
G90G94F300
M03S1500 (Turn spindle speed on)
T1(LH TURN TOOL .008 RADIUS)
X.25Z.2
G96S250 (Set spindle to SFM mode @250sfm)
G76S500 (Establish minimum spindle speed at 500 rpm)
G77S2500 (Establish maximum spindle speed at 2500 rpm)
Z0
G95E002X0
G94F300Z2
G97 (Switch to RPM mode)
S2000 (Set spindle speed at 2000 rpm)
T2(DRILL)
X0Z.2
G95E.003Z-.5
G94F300Z2
M30
```

Program commands - "M-Codes"

"M" codes are commands that control operations other than slide movements. These commands are Modal. That is, once turned on they stay active until turned off.

Some M codes control optional attachments that have to be wired into your lathe. The wiring schematics and example uses are included in the "Options Documentation" section at the back of this manual.

The M code must have a two digit number. M04 is not the same as M4. The control will not see M4. Also be careful that the 0 in the command is a number zero and not the letter O.

There can be only one M code per line of the program. If you need to perform two M functions, they must be done one line after the other.

M00 Program Stop

This program stop is not optional. When the control encounters the M00 it will stop and wait for the operator to press "CYCLE START". As an example: This command can be used to stop the program when the slide has a stop in position to allow an operator to push a piece of bar stock to the stop. Once the collet is closed the operator can push the "CYCLE START" and allow the program to continue.

M01 Program Stop - Optional

Optional stops can be put into the program, M01. This stop command is one that can be skipped over. To turn the optional stops on go to Automatic mode, once the program is selected and before the program is run press "O". This will cause the program to stop like a M00. To get past the stop press cycle start. To turn the optional stop off press "O" again.

Uses for the optional stop:

-insert a M01 after a statement for a new tool. This will help when running a new program to be sure that the tool offsets have been entered correctly. Once the program is tested you can turn off the stop and let the program run automatically.

-Have a M01 at the beginning of a program that is going to use an automatic bar feed or parts loader. This way you can have the optional stop activated when you are setting up the machine. Once the cycle and program are proven correct you can turn off the stop and let the machine run automatically.

To Go X: +0.00000 Z: +0.00000	c:\programs\newprog3	Position X: +0.00000 Z: +0.00000
Switches C.Cycle Repeat /.Block Skip O.Optional Stop S.Single Block	Press Cycle Start to Begin	Feed 0.000 IPM 100%
Functions F2-Tool Offsets F3-Edit F4-Verify F5-New File F6-Program Search F8-File Ops F9-Sec. Offsets F10-Spec. Functions J-Jog Mode M-Manual Data Input		Speed 0 RPM 100% Tool 0

See Section Five, Automatic Mode, page 5.6 for more detail about these switches

Program commands -"M-codes" continued

M03, M04, M05 - Spindle control

These commands will control the direction of the spindle.

M03 Spindle on, forward (top coming)
M04 Spindle on, reverse (top going)
M05 Spindle off

Speed in rpm is given by the S command. (For Constant Surface Feet per Minute see G96).

The correct format is:

M03S2000 This turns the spindle on forward, at 2000 RPM

M08, M09 - Coolant control

M08 Coolant on
M09 Coolant off

M12, M13 - Collet clamp control

M12 clamps the collet
M13 unclamps the collet

M19 - Spindle Positioning: Stop the spindle and lock it at repeatable home position.

M25, M26 -User assigned spare M functions (parts catcher on GT-75)

A 3 pole double throw relay in the spindle cabinet is controlled by these M-codes. You can wire to this relay for optional live tools or air blast or any other on/off device. The relay will handle up to 5amps at 220vac three phase. Machines with optional parts catcher use this M-code for that function.

M30 & M02 -End of program

One of these commands must be the last line of your program. M30 will turn off all active M functions and reset the control back to the beginning of the program, and wait for the operator to press "CYCLE START". If you have the Automatic mode on "Continuous" it will restart the program from the beginning. M02 will end the program and reset it to the beginning without resetting M functions.

M31 - Cancel Cycle Repeat

If you want a subroutine to stop the program, as in an end of bar subroutine, for example, you must cancel the Cycle Repeat mode before the M30 command.

Program commands -"M-codes" continued

M88 - Stop the spindle and lock it at high resolution home (optional: C-Axis only)

This code is used to stop the spindle at the C-Axis home to a resolution of 0.01°.

M89 - Stop the spindle and lock it at arbitrary home (optional: C-Axis only)

This code is used to quickly stop the spindle to make holes or slots. This is quickest C-Axis positioning, but all C-Axis machining must be done before returning to normal spindle function because the spindle does not search for the true C-Axis home as it does with M19 and M88.

M91, M92, M93, M94 - Wait for input

These M-codes stop the program until an input is "on" or "off". This is useful for coordinating activity for an auto-loader. The OmniTurn 'waits' (the program stops, like M00 or M01) until the input is in the correct state.

Relay closure to 0VDC (COM) sets the input "on".

The input is "off" when the relay is open.

The inputs are located on headers on the circuit card assembly (CCA) in the spindle cabinet. (see page 6.23 or 6.30 CCA layout and functions).

The commands are as follows:

M91 Wait for HDR111-1 & -2 to be open circuit

M92 Wait for HDR111-1 & -2 to be shorted together

M93 Wait for HDR110-1 & -2 to be open circuit

M94 Wait for HDR110-1 & -2 to be shorted together

M95 - Conditional jump to subroutine

This M-code will cause the program to jump to subroutine 1 if HDR112-1 & -2 are shorted together.

The condition must exist *before* the command is executed. Use dwell (G04) if necessary to insure that the state of the input is stable *before* the program executes the M95 command

M97 - Conditional jump to subroutine (optional: Mitsubishi Fx PLC only)

This command will cause the program to jump to any subroutine if any available PLC input is either "on" or "off". The syntax is M97InCnPn. (Memory locations in the PLC that end in 8 or 9 can also be used).

In is the *input* which is being tested

Cn is the *condition*; either 1 ("ON") or 0 = ("OFF")

Pn is the *subroutine* which will be executed

The condition must exist *before* the command is executed. Use dwell (G04) if necessary to insure that the state of the input is stable *before* the program executes the M97 command

M98 - Jump to subroutine (unconditional)

When this command is executed, the program will jump to the specified subroutine.

The syntax is M98Pn, where n is the subroutine number.

}n - Begin subroutine n

The first line in any subroutine must be the brace } followed by the subroutine number. No other text is allowed on that line.

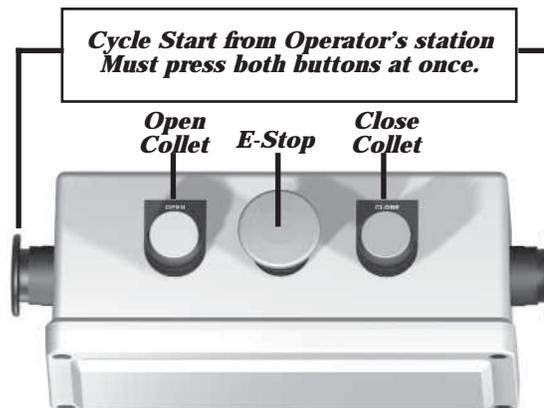
M99 - End subroutine

The last line in any subroutine. The *next* line which will execute will be the line ***immediately after the line that called the subroutine.*** No other text is allowed on that line.

Spindle Control

Cycle Start

The Operator's Station (also called Palm Box) provides convenient access to Cycle Start and collet control.

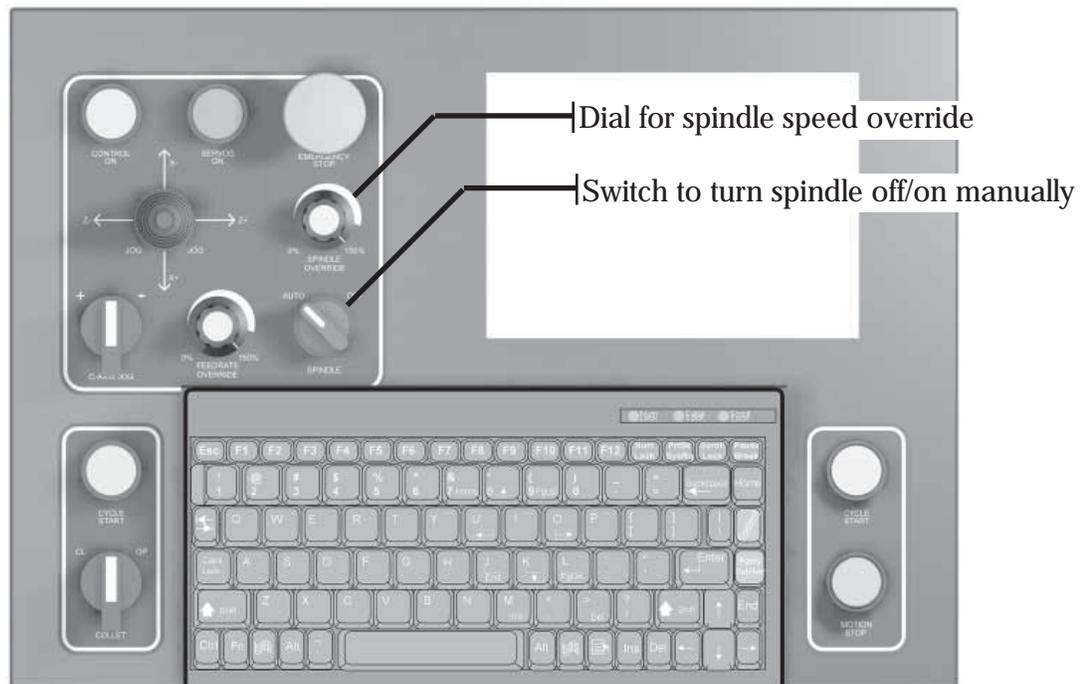


Controlling the spindle:

Manual control of the spindle

You can turn the spindle on manually. This is useful for setting up tools and testing the drive. First go to the MDI mode. (See Section 1.16 for details) Then enter a command to turn the spindle on at a speed you want: M03S2000. This will turn the spindle on. To turn the spindle off and leave the command for spindle on still active turn the switch on the face of the control for "SPINDLE OVERRIDE" from AUTO to OFF. Now leave MDI by pressing F1 and go to the Jog mode. Now you can turn the spindle on and off with the override switch. If you want a lower speed you can turn the dial for a lower speed.

Note: If you go the Automatic mode and enter the editor the spindle on command is cancelled.



To review the functions of all the front panel controls, refer to pages 1.4, 1.5 and 1.6 in Section 1.

Spindle Control

From the program:

Turning on the spindle.' You can turn the spindle on forward (top coming) or reverse (top going):

Spindle on, forward: **M03S2000**

Spindle on, reverse: **M04S2000**

Turning the spindle off:

Spindle off automatically at the end of the program: **M30**

Spindle off by command in the program: **M05**

Manual override: There is a switch on the face of the OmniTurn control that will turn the spindle off.

Leaving the spindle on at the end of the program:

If you are running a job automatically (bar work or automatic loaders) and want to leave the spindle running at the end of the program use M02 for end of program. This is like a M30 but it will not reset the M functions that have been turned on, like M02 and M08.

Setting a spindle speed:

S is used to set a spindle speed. This can be either RPM or SFM (Constant Surface Feet per Minute. Please refer to G96 for instructions on using SFM.

When you turn the spindle on it is good practice to set a speed at that time: M03S2000.

If you already have the spindle on and you want to change the speed, you do not have to use the M03 command again. Note: If you want to put the M03 in again it will not do any harm. An example would be: X. 125Z-.25S 1500

Manual override: There is a dial (pot) on the face of the OmniTurn control that will override the spindle speed selected. It changes from 0 -100%.

Tapping with the spindle control:

It is possible to tap with the spindle control option. You will need either a self releasing or pitch compensating (floating) tap holder. The following is a short program that will tap at 20 pitch approx .5" deep:

```
g90g94f300
m03s1000          Turn spindle on at 1000rpm
t4 (1/4-20 tap) Call tap into 'safe' position
z.1              Move tap almost to face of part
g95f.048z-.4    Feed in at about 95% of pitch to allow some play in holder, stop short of full
depth
m04              Reverse the spindle
g04f.5         **Dwell to allow time for spindle reversal
z.1              Feed tap out to clear the part
g00z1           Rapid to 'safe' location
:
:
```

** This dwell is used to give the spindle some time to reverse before starting the tap back out. During this dwell, the slide is stopped, but the spindle is slowing down in the m03 direction, then speeding up in the m04 direction. The floating tap holder allows the tap to move in the thread as the spindle turns. This dwell time will vary depending on the system you have and the spindle speed you are running. The lower the spindle speed, the shorter the dwell time.

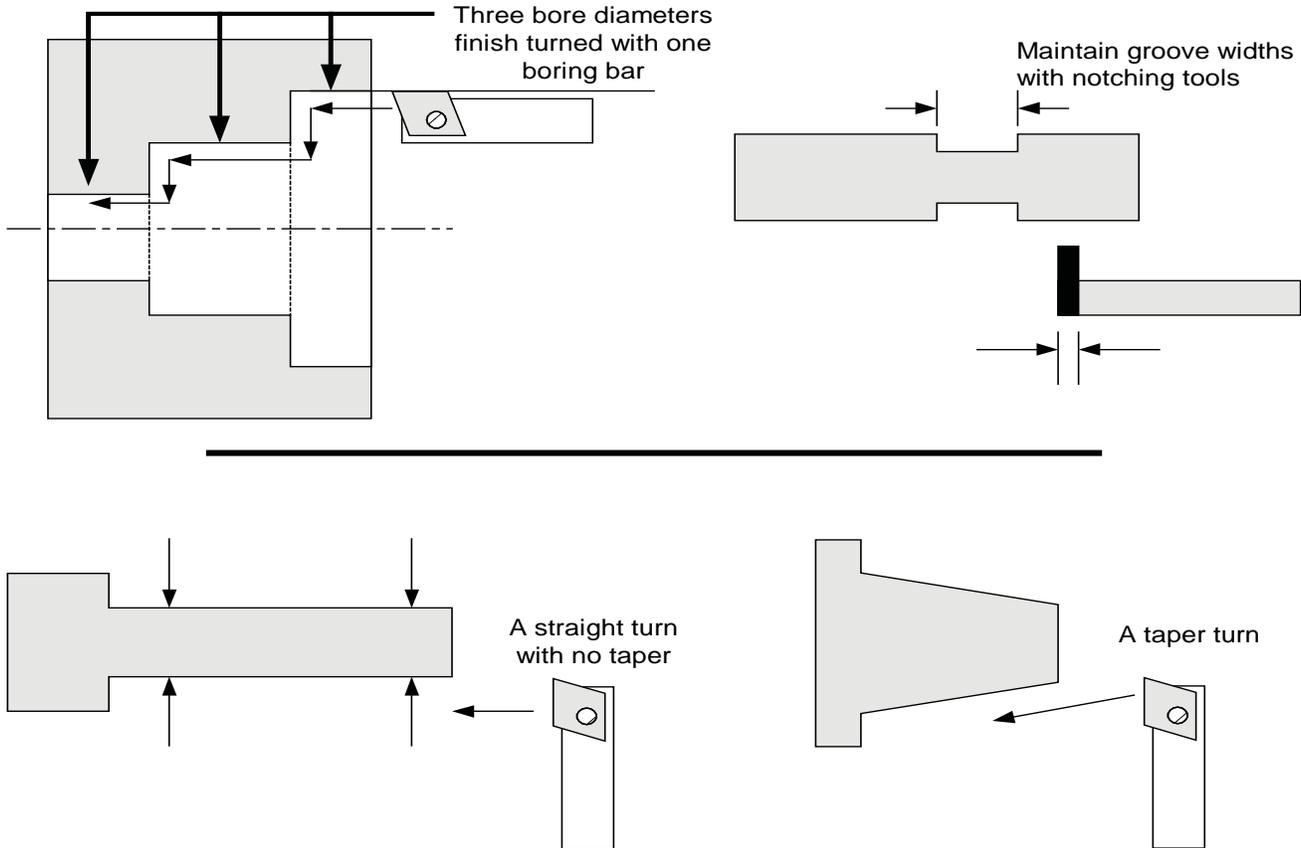
This page is intentionally blank

Secondary Offsets

What are secondary offsets?

Secondary offsets are corrections that you can put into your program that the operator can adjust when running the program without having to go into the program to edit it. Once the program has been written with the secondary offsets incorporated, these corrections are made by pressing F9 while in the Automatic mode and inputting the amounts. This procedure is very similar to adjusting tool offsets. The big difference with secondary offsets is that there can be more than one correction to a tool.

There are a number of ways that they can be used. Below are a few examples of typical applications.



In all three cases it would be very advantageous to be able to have the operator make corrections to the parts that entail more than just moving the tool by changing the tool offset (T). If you made a change to the tool offset, the overall size of the part would change in each of the above examples.

If, however, you had a taper in the long thin part (lower left sample) and had to correct it to get the part straight, offset changes would not help. The secondary offset allows you to add or subtract a little, to any move, at any point in the program. So the correction of the taper can easily be taken care of.

NOTE: Clear secondary offsets before using them!

Before you run a program that uses secondary offsets be sure that you have reset the secondary offsets that you are using to zero! This can be done by pressing C when asked to make a correction to the offset table (See F9 in the Automatic section)

Secondary Offsets

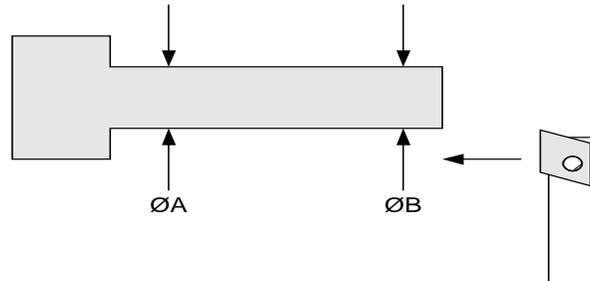
Using Secondary Offsets

Secondary offsets are used with D commands. The format is the same as a T command. Add the D with a number, ie: "D2" for #2 secondary offset, to the line of code to be corrected. This command will call up the value located in the secondary offset table and add it to the move. The secondary table looks like this:

Look at the following example:

1	X: +0.00000 Z: +0.00000 R: 0.00000	17	X: +0.00000 Z: +0.00000 R: 0.00000
2	X: +0.00000 Z: +0.00000 R: 0.00000	18	X: +0.00000 Z: +0.00000 R: 0.00000
3	X: +0.00000 Z: +0.00000 R: 0.00000	19	X: +0.00000 Z: +0.00000 R: 0.00000
4	X: +0.00000 Z: +0.00000 R: 0.00000	20	X: +0.00000 Z: +0.00000 R: 0.00000
5	X: +0.00000 Z: +0.00000 R: 0.00000	21	X: +0.00000 Z: +0.00000 R: 0.00000
6	X: +0.00000 Z: +0.00000 R: 0.00000	22	X: +0.00000 Z: +0.00000 R: 0.00000
7	X: +0.00000 Z: +0.00000 R: 0.00000	23	X: +0.00000 Z: +0.00000 R: 0.00000
8	X: +0.00000 Z: +0.00000 R: 0.00000	24	X: +0.00000 Z: +0.00000 R: 0.00000
9	X: +0.00000 Z: +0.00000 R: 0.00000	25	X: +0.00000 Z: +0.00000 R: 0.00000
10	X: +0.00000 Z: +0.00000 R: 0.00000	26	X: +0.00000 Z: +0.00000 R: 0.00000
11	X: +0.00000 Z: +0.00000 R: 0.00000	27	X: +0.00000 Z: +0.00000 R: 0.00000
12	X: +0.00000 Z: +0.00000 R: 0.00000	28	X: +0.00000 Z: +0.00000 R: 0.00000
13	X: +0.00000 Z: +0.00000 R: 0.00000	29	X: +0.00000 Z: +0.00000 R: 0.00000
14	X: +0.00000 Z: +0.00000 R: 0.00000	30	X: +0.00000 Z: +0.00000 R: 0.00000
15	X: +0.00000 Z: +0.00000 R: 0.00000	31	X: +0.00000 Z: +0.00000 R: 0.00000
16	X: +0.00000 Z: +0.00000 R: 0.00000	32	X: +0.00000 Z: +0.00000 R: 0.00000

Secondary offset number:
Press Esc to exit offset adjustment screen
Press C to clear all offsets:



With this shaft, ØA diameter = ØB diameter. In order to do this, put an offset command at the end of the turn.



The code to face and turn the shaft is:

Without correction
X0Z0

With correction
X0Z0

move to the center and face

Secondary Offsets continued

X. 125	X. 125	move to the diameter
Z-.75	Z-.75 D4	turn the diameter, corrected
X.2	X.2 DO	move to the major diameter, turn offset off

If there is no problem with a taper on the part, the X and Z values of D4 are set to zero. If, however, there is a taper, say .001" oversize at the base of the turn, this can now be corrected. Call up the secondary offset table "F9". The control will ask "Offset Number?" Type 4 and return. Then it will ask "X DIAMETER CORRECTION?". Type -.001 and Return. You will notice that the value of #4 -X will now be -.0005. This is because the offsets affect the radius, not the diameter. Then the control will ask "Z CORRECTION?". If there is no correction just hit Return. Then to return to the automatic mode press ESC. This will bring you back to Automatic mode ready to run the program again. To start the program again, press "CYCLE START".

You will notice that the operator does not have to go into the program to adjust for the taper. With this feature you can have personnel make corrections without having to understand programming.

The correction of -.0005 will be added to the third line move and it will behave like the line was:

X.1245Z.75

Since there was no correction in X, there is no X value.

Canceling a secondary offset

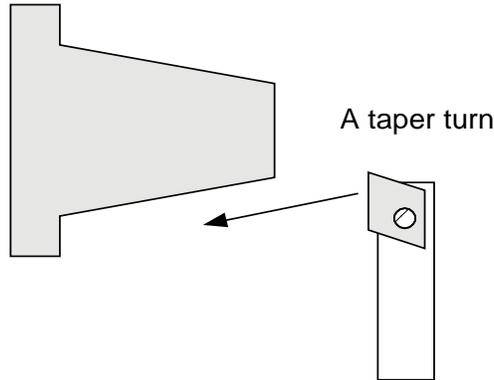
The secondary offset correction stays in effect until:

- There is a tool change. This cancels an offset
- A secondary offset DO will turn off the offset.
- Calling up another offset will cancel the original offset and enact the new offset

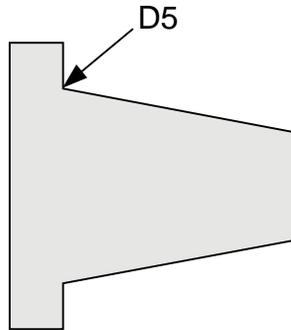
Secondary Offsets continued

Secondary offset examples:

A taper that has to be single point turned and then maintain the major diameter.



The actual taper that a tool cuts will depend on the toolnose radius. If this is not an easy tool to maintain then the taper will vary as the tool changes. Normally this would be a big problem. However with the secondary offsets this is very simple. We could put a secondary offset at the end of the taper and then turn the offset off. This would look like:



The commands might look like this:

X0Z0	Move to the center and face of the part
X.2	Move to the minor diameter of the taper
X.3Z-.5 D5	Mover to the major diameter of the taper, corrected with D5
X.4 D0	Mover to the major diameter of the part, correction off, D0
Z-.7	Turn the major diameter of the part

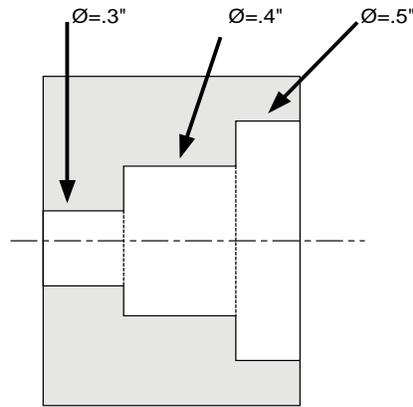
In this case D5 can be used to adjust two features:

- The diameter at the end of the taper -D5 X value
- The location in Z of the taper. -D5 Z value

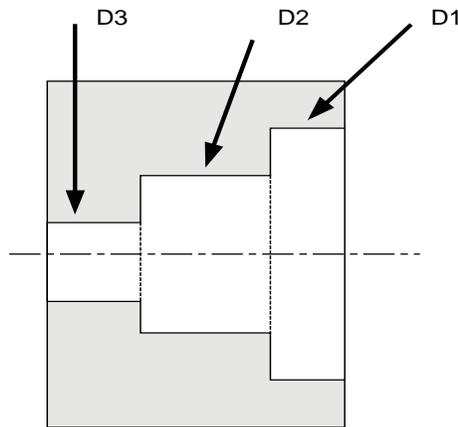
Neither, either, or both values can be entered. Each will effect the angle generated.

Secondary Offsets continued

Here is another example:



For this example, three diameters have to be turned with only one tool. Each of the diameters can have an individual offset.



The coding for this is:

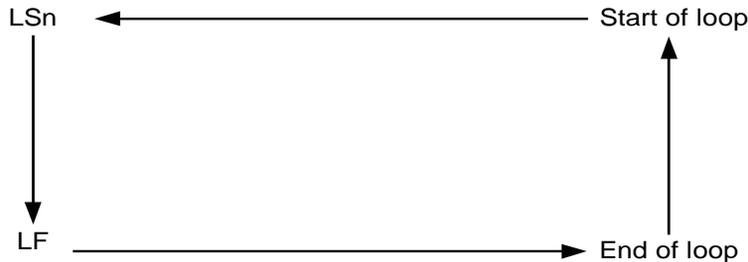
X-.35Z0	Move to the face of the part at -.7 diameter
X.25 D1	Move to .5" diameter, corrected
Z-.2	Turn the first bore
X.2 D2	Move to the second bore diameter, corrected
Z-.5	Bore the second diameter
X.15 D3	Move to the last bore diameter, corrected
Z-.7	Bore the last diameter

Here we have used the secondary offsets to help position the tool at the beginning of each of the bores. The offset correction will be added to the positioning move and then be maintained until a new offset is changed. For the .5" diameter we have added D1 to the move that brings the tool to the .5" diameter. Then, when we bore the first diameter, the correction is continued and the hole is on size. If we have no correction on the remaining two bores they are not corrected since there are zero values in each of the offsets. The next move to .4" diameter and .3" diameter will not be corrected. When the next tool is called, the D3 offset will be canceled.

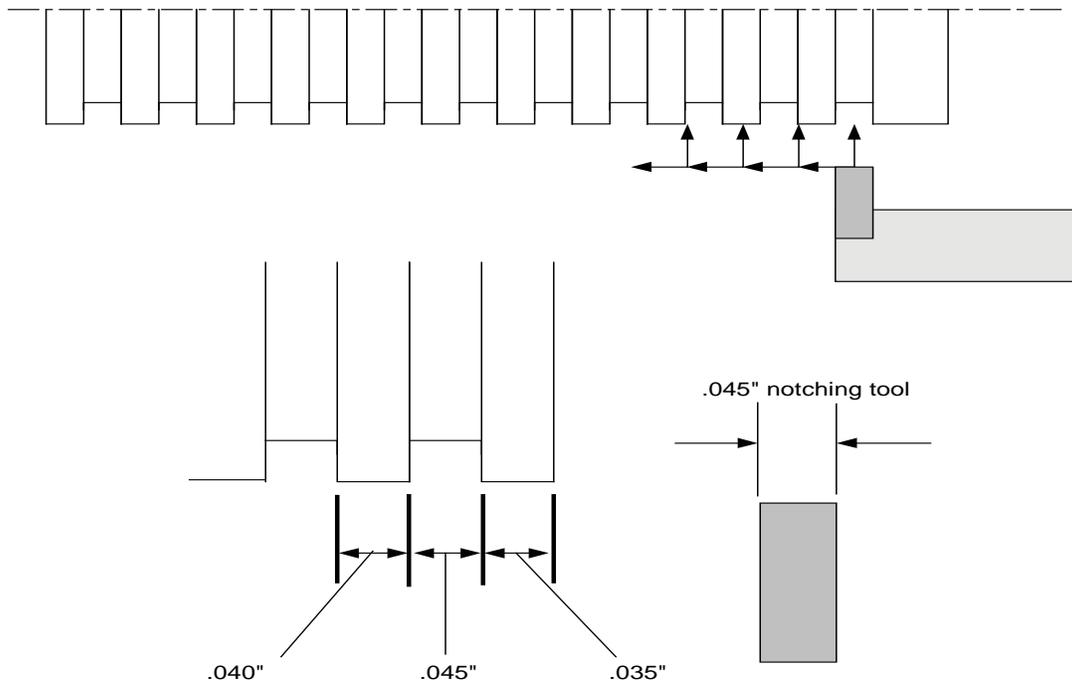
Looping

Looping is used to perform repetitive moves without having to write long programs. The start of a loop is defined by LS and then the number of times you want to execute the loop. IE: LS35 will start a loop with 35 repetitions. This command has to be on a line by itself. As the end of the loop put a LF on a line by itself.

NOTE: Text statements can not be used inside the loop!!!!



An example of this is having to have to make lots of notches on a part that are evenly spaced:



For this example the could be:

G90G94F300

T 1 (notch tool 045 wide)

X.25Z.2

Z-.035

G91-----NOTE THIS LOOP IS DONE IN INCRIMENTAL

LS16

G95E001 X-.1

G04E05

G00X.1

Z-.085

LF

G90-----BACK TO ABSOLUTE MODE

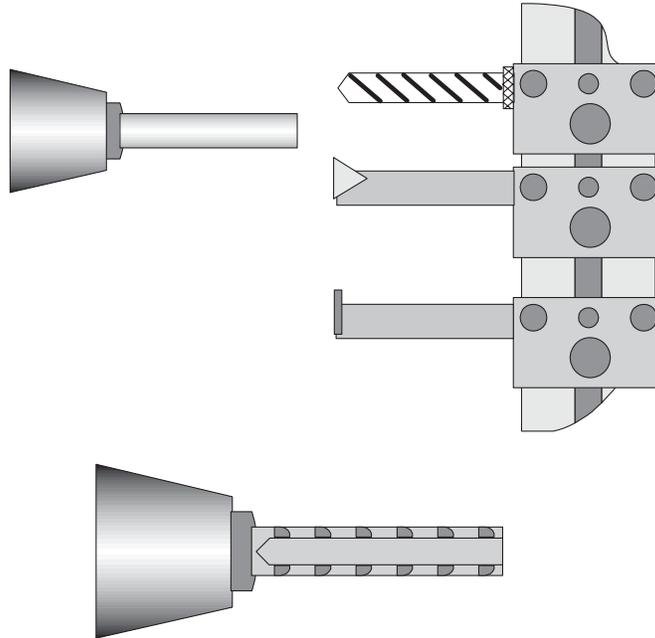
ZI

M30

Looping

Looping with Work Shift (G10)

It is easy to loop a portion of a program and have it shift over using work shift -G10. This enables you to do many parts with only one feed out of a bar. In the following example it will show how to drill one deep hole and then turn and part off 6 thin rings.



```
G90G94F300G72
M03S2000
M08
T 1 (DRILL)
X0Z.5
Z. 1
G95F003
G83Z-1 K.3L300C.200
G00Z1
LS6
T2 (TURN TOOL)
X.3Z.5
Z. 1
G95F.003Z0
X.35Z-.025
Z-.15
G00X.36
Z1
T3 (PART OFF TOOL)
X.4Z.5
Z-.15
G95F001 X.25
G00Z 1
G10X0Z-.15
LF
M30
```

Spindle Positioning

Spindle positioning system specifications (Option)

Spindle power: 5HP
Voltage: 200 -230V 3 phase
Resolution: .01°
Max Speed: 4000 rpm
Min Speed: 0 rpm

M19 Programmed by itself causes the spindle to quickly position at C-Axis “home” (0°) within about $\pm 0.2^\circ$ accuracy. For highest accuracy use M88, below. After the command is executed the spindle is locked in position. To release the spindle use M05.

M89 Programmed by itself causes the spindle to stop and lock immediately, creating an arbitrary C-Axis “home”. This is the fastest way to begin machining C-Axis features, but all the features referenced to the first must be completed before exiting C-Axis mode because the “home” is arbitrary and cannot be accurately repeated.

M88 Highest accuracy C-Axis “home” location. Programmed by itself causes the spindle to position at C-Axis “home” (0°). Takes longer than M19, but locates to about 0.05° accuracy.

CI(-)nnn.nn This moves the spindle an *incremental* amount of degrees from it’s current position.

CA(-)nnn.nn This moves the spindle move to an *absolute* angular location relative to “home”.

Fnnn.nnn This sets the feed rate for the C-axis. Like the X and Z-axes, the C-axis encoder provides 20,000 counts per inch, so a programmed feed rate of one inch per minute (1ipm) translates to 1ipm *surface* feedrate, only at 0.573” diameter. To calculate programmed feedrate, divide 0.573 by the diameter of the part and multiply by desired feedrate:

$$F = (0.573/D) * F_{\text{desired}}$$

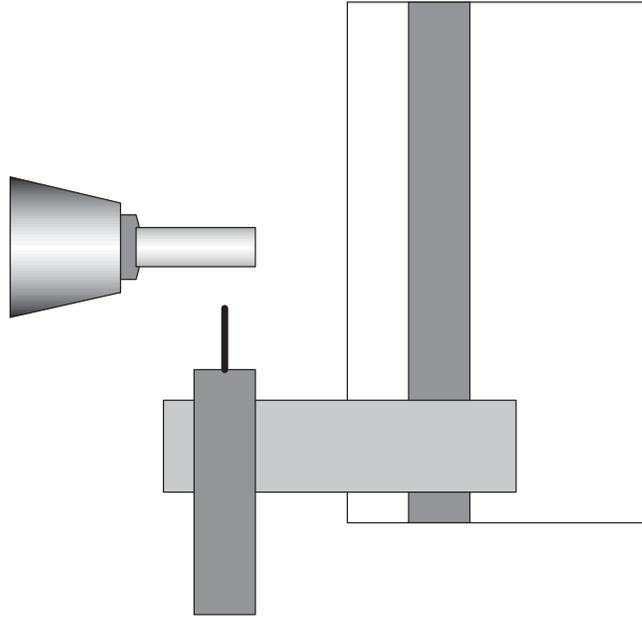
Notes on use:

The C-Axis is fully coordinated with the X & Z axes; the CNC sees it as a third linear axis. Simply program how far in Z and/or X the slide should move for a given angular rotation in C and the CNC will calculate the necessary feed rates.

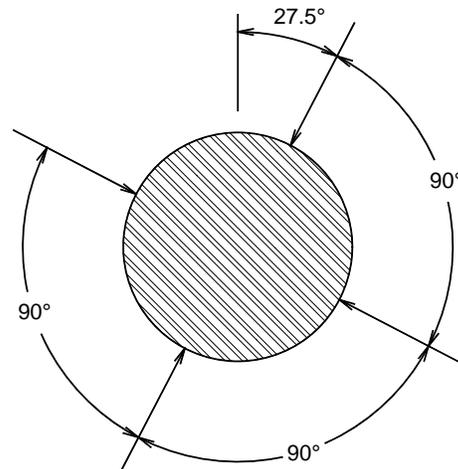
Spindle Positioning

Example showing positioning and cross drilling

In the following example we show a drill mounted on the slide.



The slide will be used to drill the holes. We will drill (4) holes 90° apart, the first hole is located at 27.5° from a reference 0°.

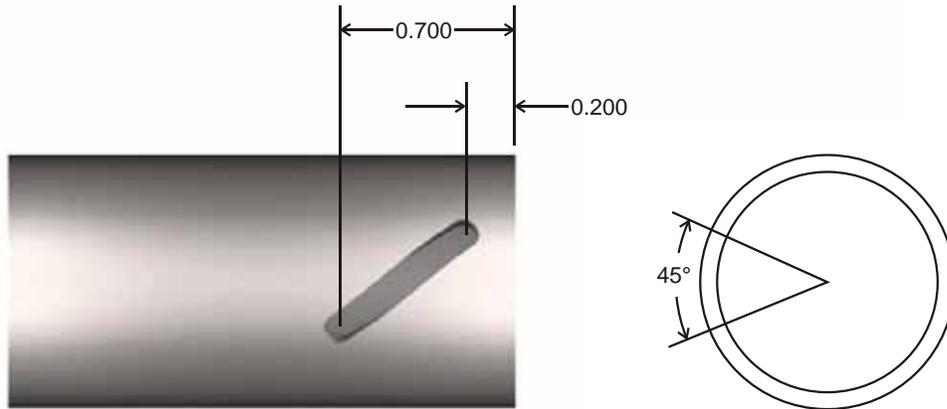


```
g90g94f300g72  
t32 (live drill from the side)  
x.75z-1  
z-1.3  
m27 (turn drill on)  
m19 (orient the spindle to 0°) →  
ca27.5 (position spindle at 27.5° absolute)  
ls4 (start of loop)  
g94f10x1 (drill a hole)  
x.75f10  
f300 (position spindle at max feedrate)  
ci90 (rotate 90° incrementally)  
lf (end of loop)  
z0  
m30
```

NOTE: If the four holes don't need to be located relative to another feature, use m89 instead of m19 and eliminate the ca27.5. M89 stops the spindle without homing, which saves some time. The four holes will be at 90° relative to each other, but independent of any other features.

Spindle Positioning

C-Axis rotational milling (Mill a slot in a tube)



Use C-Axis mode to mill the slot as a lead:

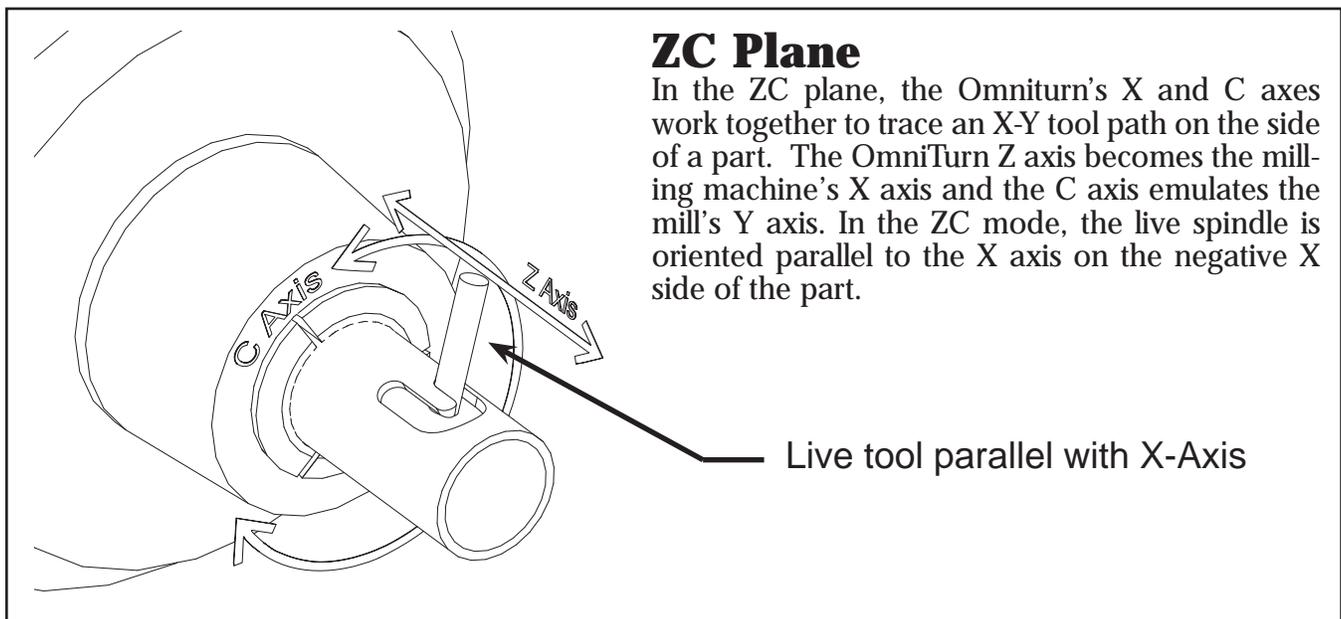
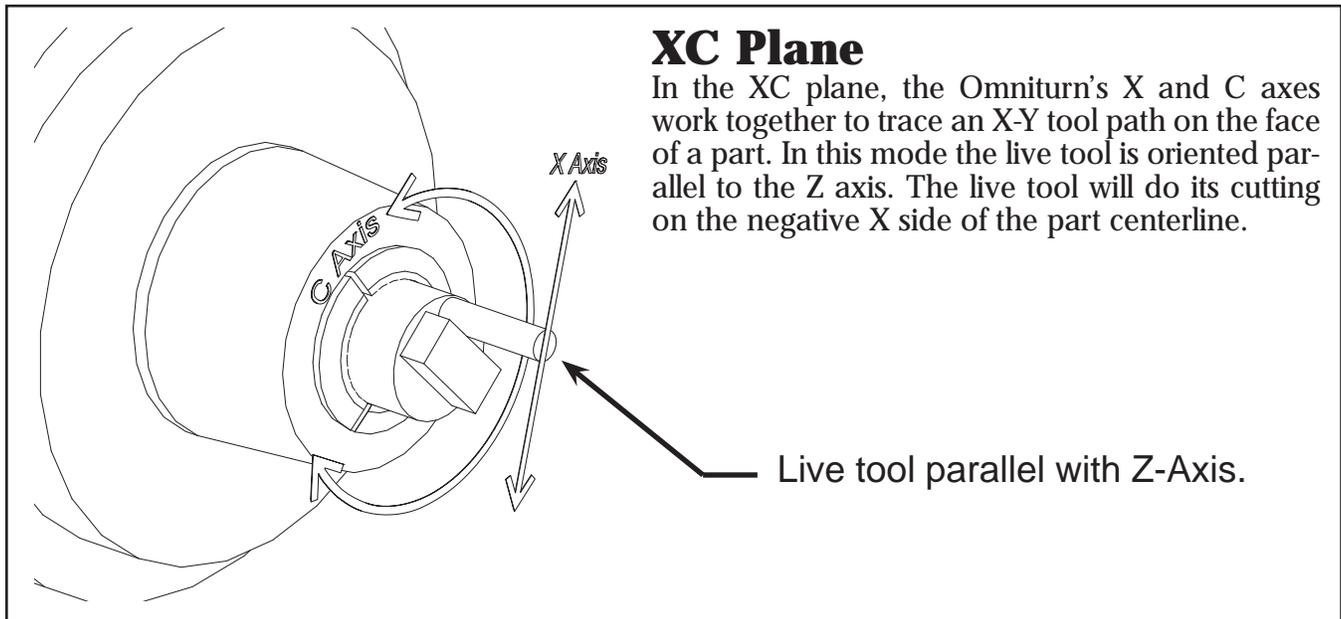
```
g90g94f300g72  
t32 (Live mill from the side, below part)  
x.75z-1 (Move to safe position)  
z-1.2 (Move to Z dimension)  
m27 (Turn mill on)  
m19 (Positioning Mode: locate spindle at 0°)  
x.5f3 (Feed mill into part at 3ipm)  
ca-45z-.7f5 (Rotate spindle m04 direction 1/8 turn and move 0.500 in Z at 5ipm)  
f100x.75 (Clear tool)  
m28 (Turn mill off)  
g00z0 (Clear tool)  
m30 (End program)
```

Spindle Positioning

C-Axis Plane Switching

Plane switching allows the programming of parts using the C axis in combination with the X or Z axis as if you were programming a milling machine.

There are two plane-switched modes: XC and ZC:



Spindle Positioning

C-Axis Plane Switching: Commands

The programming details for plane switching are not yet finalized, so currently plane-switching commands need to be hidden from the error checking software by placing them in parentheses. The only commands permitted in the plane-switched part of the program are axis moves and feedrate commands. There is no cutter compensation, so it is up to you to write the program for the endmill you are using. All dimensions in the plane-switched mode are in absolute coordinates.

The commands are as follows:

(~PLXC): Start XC mode. The tilde character “~” in a (comment) alerts the Omniturn’s parsing program to scan for a plane switching command.

(~PLZC): Start ZC mode. The tilde character “~” in a (comment) alerts the Omniturn’s parsing program to scan for a plane switching command.

(Xnn): moves the X axis to the specified coordinate

(Ynn): moves the Y axis to the specified coordinate

(G02XnnYnnInnJnn): moves X and Y to the specified coordinates clockwise along a circular path with its center located at the I dimension in X and the J dimension in Y.

(G03): same as G02 but travels CCW along the arc.

(Fnnn): feedrate specified in inches per minute

(Znn): moves the virtual mill’s Z axis.

In the ZC plane, the Z dimension is specified as the diameter of the cylinder the tool’s tip would touch. In the XC plane, the Z axis is the same as it would be if plane switching was not in effect.

(Dnn)(ZC plane only): specifies the part diameter at which the Y dimensions are translated to C axis rotation.

This value defaults to the virtual mill’s Z position (that is, the diameter at the tool’s tip) at the start of the plane-switched operation.

It should only be changed when the Y position is 0, since it causes a re-scaling of the Y dimensions on the first move after it is programmed.

The reason this parameter exists is that you may want the Y dimensions to be calculated at the surface of the part rather than at the depth of cut.

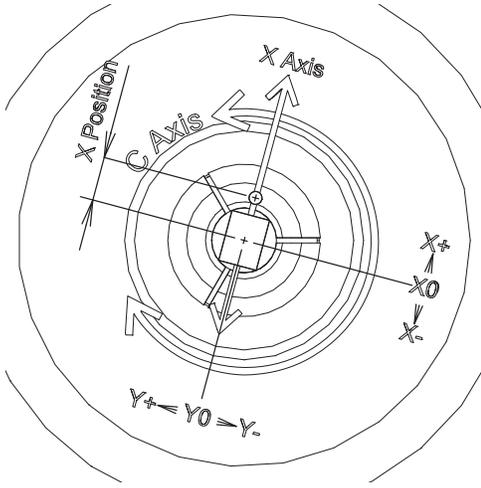
(END): cancel plane-switch mode

Note: *if you copy the plane-switch programs, do not include comments in the plane switched section. Nothing is allowed in the plane-switched program except the program statements described above.*

Spindle Positioning

C-Axis Plane Switching: Using the XC Plane

In this mode the live tool works on the face of the part.



The XC mode is started with the command (~PLXC).

Before entering this command the live tool must be located on the negative X side of the part, with its spindle centerline parallel to the lathe's Z axis.

On entry, the Y position is 0, and the X position is the distance from the spindle center to the center of the tool.

The computer constantly calculates the tool's X-Y position along the programmed path, and the X-Y coordinates are converted to polar coordinates, with X axis motion controlling the distance from center and C axis rotation controlling the angle.

As seen from the tool's point of view, motion to the right is X+ and up is Y+.

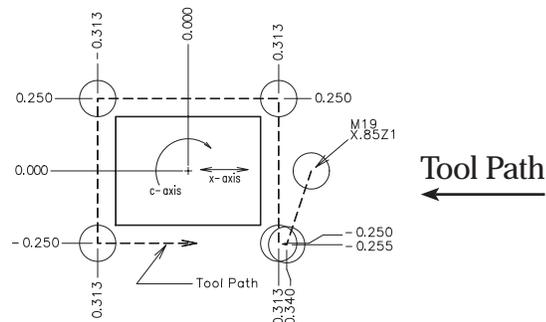
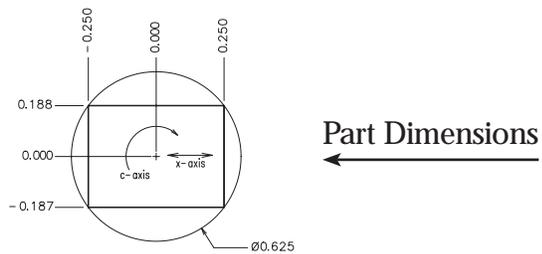
The Z axis remains the same as in normal turning mode.

C-Axis XC Plane Switching Programming

The following programs use a 1/8" endmill to cut various shapes on the end of a 5/8" bar. Note that the programmed dimensions allow for the cutter size

The first program cuts a 3/8" x 1/2" square (5/8" diagonal measurement). Note that it begins cutting at a corner of the square and goes around the outside of the square in a CCW direction. Starting at the corner minimizes the initial chip load, which reaches its maximum at the center of each side. Going around the part CCW (conventional cutting) avoids the tendency of small cutters to dig in and snap off when climb cutting.

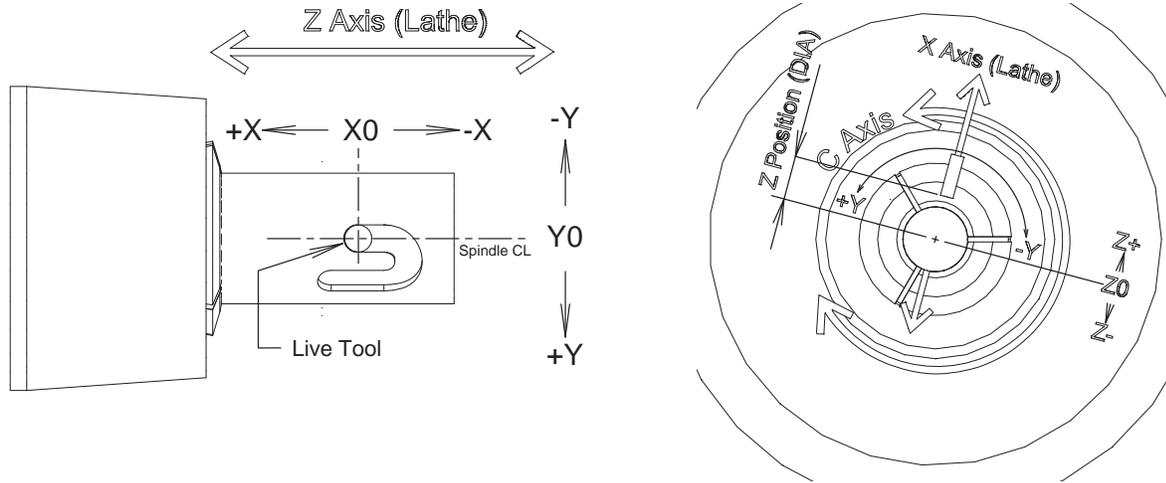
```
g90g94g72f300
m19 (c-axis at 0° absolute)
t21 (live tool)
x-.85z1 (safe location)
f100z.05
f10z-.1 (at depth)
m27 (turn on live tool)
(~PLXC) start XC mode
(x.340y-.255) safe near corner
(x.313y-.250) at first corner
(x.313y.250) next corner
(x-.313y.250) third corner
(x-.313y-.250) fourth
(x.313y-.250) back at first
(end) end plane-switch mode
f200Z1.5
m30
```



Spindle Positioning

C-Axis Plane Switching: Using the ZC Plane

In this mode the live tool works on the side of the part.



The ZC mode is started with the command (~ PLZC).

Before entering this command the live tool must be located on the negative X side of the part, with its spindle centerline parallel to the lathe's X axis.

On entry, the tool is centered at X0 and Y0 position in the virtual milling machine, so before switching planes you need to locate the Z and C axes to the location you want to be X0, Y0 on the part surface. **NOTE:** *You must accommodate the tool diameter, and program accordingly.*

X positive motion is to the right (toward the spindle face) as viewed from the live spindle's point of view, and Y positive motion is up from the live tool's point of view.

The virtual mill's Z axis is set to the X position prior to the plane switch, but the sign is inverted, so that a positive Z move moves the tool tip away from the part center.

Because most lathe programming refers to radial positions as diameters, Z dimensions are given as the diameter of the cylinder the tool tip would touch.

For example, in this code:

```
g72g94f300  
m19  
T11 (call up live tool)  
x-1z.5 (tool is at a 1inch diameter on -X side of part)  
z-.2  
(~ plzc) plane switch mill position is x0,y0,z1  
(f5z.85) tool tip moves in to cut at a diameter of .85
```

Note that the z.85 command moves the tool tip **in** toward the center of the part.

Spindle Positioning

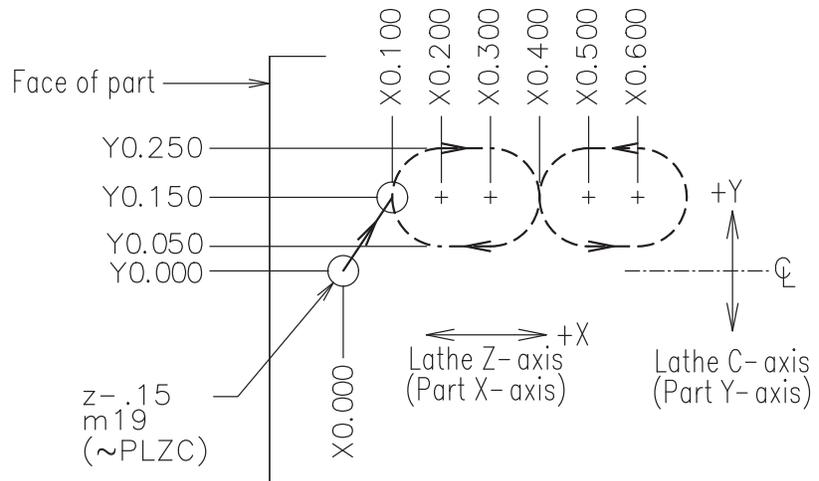
C-Axis ZC Plane Switching: Programming *continued*

These examples were cut in a 7/8" delrin bar with a 1/16" end mill.

Note: if you copy these programs, do not include comments in the plane switched section. Nothing is allowed in the plane-switched program except the program statements described above.

This one does a figure-8 pattern consisting of 2 oval "racetrack" patterns joined together.

```
g90g72g94f100
m27 (turn on live tool)
t11
x-1z1
z-.15 (x0y0 on virtual mill)
m19 (turn on C axis)
(~PLZC) start CZ mode
(d.875) set Y axis scaling to 7/8" diameter
(x.1y.15f20) move to starting X-Y position
(z.775f10) plunge tool in to a depth of .05"
(g02x.2y.25i.2j.15) cw 90 deg arc moving right .1", up .1" centered at x.2, y.15
(x.3) straightaway to the right .1"
(g02x.4y.15i.3j.15) another 90 deg. cw arc
(g03x.5y.05i.5j.15) now a ccw 90 deg arc
(x.6) bottom straight on right-hand oval
(g03x.6y.25i.6j.15) ccw 180 deg arc
(x.5) top straight on right oval
(g03x.4y.15i.5j.15) ccw arc back to start of right oval
(g02x.3y.05i.3j.15) cw arc to bottom straightaway of left oval
(x.2) bottom straight
(g02x.1y.15i.2j.15) cw arc back to start point
(z1) withdraw cutter
(end) cancel plane-switch
x-1
z1
m30
```



Spindle Positioning

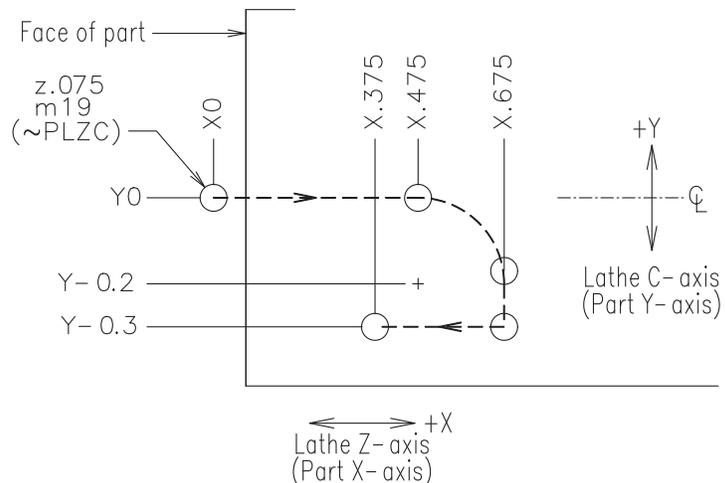
C-Axis ZC Plane Switching: Programming *continued*

These examples were cut in a 7/8" delrin bar with a 1/16" end mill.

Note: if you copy these programs, do not include comments in the plane switched section. Nothing is allowed in the plane-switched program except the program statements described above.

This cuts a J-slot like those found on electrical connector housings.

```
g90g72g94f100
m27 (turn on live tool)
t11
x-1z1
z.075 (x0y0 on virtual mill)
m19 turn on C axis
(~ PLZC) start ZC mode
(d.875) set Y axis scaling to 7/8" diameter
(z.775f10) move in to a cut depth of .05"
(x.475) cut straight side of J
(g02x.675y-.2i.475j-.2) right turn at end of long side
(y-.3) short straight segment
(x.375) move back in x to create detent notch
(z1) withdraw tool
(end) cancel plane switch
x-1
z1
m30
```



Tool Offsets

Tool Offsets

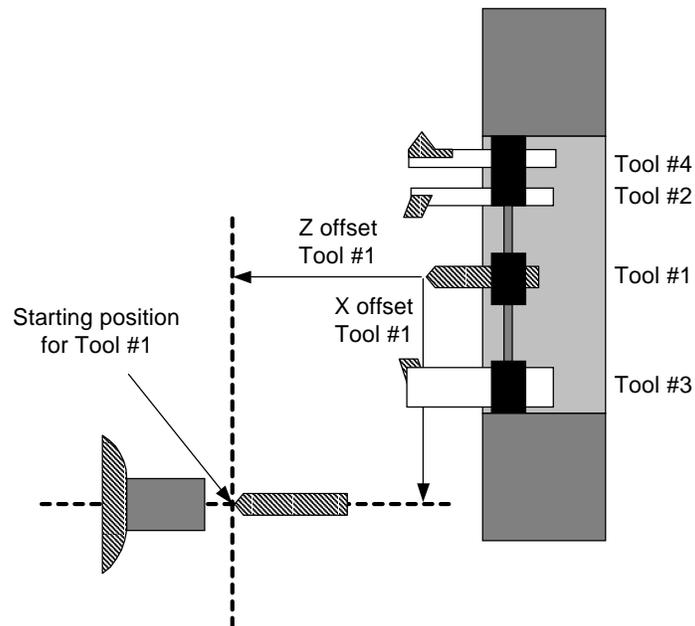
After a program is written, the control has to be told where the tools are on the slide. Tool offsets are used by the control to position the tool and know where it is. There are three steps when using the offsets.

1. **Establishing the offsets:** This is done in the Jog mode before the program is run
2. **Call the tool offset and establish what the location is:** This is done in the program you have written and used when the program is run
3. **Adjust the offset:** This is done while you are running a program and find the tool is not located where you want it. Fine adjustments are available down to .0001 "/diameter in X and .00005" in Z.

What are tool offsets?

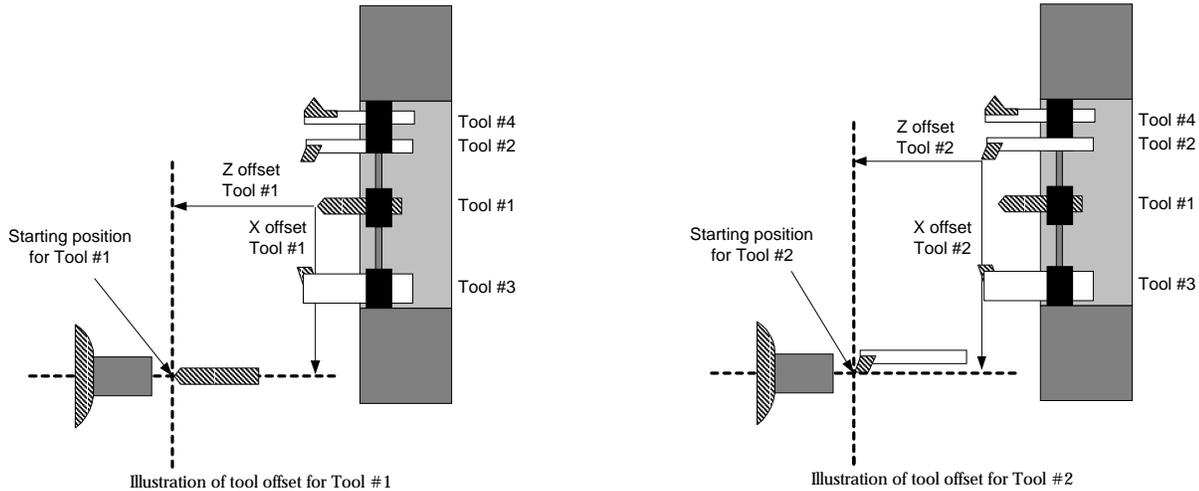
When you turn on the OmniTurn it does not know where it is. However the position commands used in the program assumes that it does. In order to run a program the control needs to know where the slide is and where the tools are. So the tool offset is part of a two step procedure. First the slide has to establish a HOME, Reference Zero (See Establish HOME procedure). Then from HOME it is possible to find the distance of the tool from the work piece. This location will be the starting point for the tool. These procedures are required for all CNC machines, not just the OmniTurn.

In the example below, we show a slide that has four tools. The slide has been sent "Home" and is now in that location. The tool offsets that are shown are the distance from HOME to the starting point for tool #3. When the program calls tool #3 it will move the slide to the distance from the reference zero given in the offset table. It does not matter where the slide is when the offset is called.



When the control is turned off the offsets are not forgotten, they are remembered on the disk that holds the OmniTurn software. When you turn on the OmniTurn, it automatically loads the tool offsets from the last time the machine was setup. This means that once a job is setup, and the tools have not been moved, you can run it without having to setup again. The only thing to be done is Reference Home the slide so that it has the correct starting point with which to apply the offsets. Each tool will have its own offsets. See the fig below as examples of additional offsets.

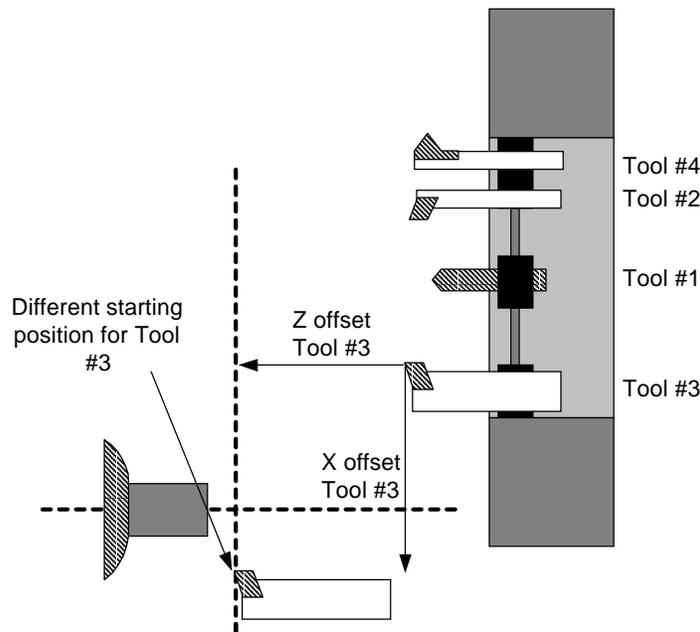
Tool Offsets



For all of the above examples, the tool is started at the center of the part and some distance from the face. This is being done in the examples only by choice. The tool can be started anywhere. To save moves and time you might have wanted to start tool #3 close to the diameter of the blank and not at the center. While learning to program, it might be helpful to always start the tool from the center of the part and then move it where ever it is needed. This will make the setup and establishing the tool offset a little simpler.

After you have told the control where the tool is going to start (Tool Offset) then the program tells the control what the value of the starting location is.

Look at the examples below. In this example we are assuming that Z zero is at the face of the part and X zero is at the center. We have picked a different starting location for tool #3. Both are valid starting points. The only requirement is to tell the control what the value of the starting point is. This is done in the program on the line after the tool call.



The starting point for the tool is the X and Z values immediately after the Tn command. This is where the tool will move to.

Tool Offsets

How tool offsets are used

There are now two different ways to establish the location of the tool after a tool call (tool offset):

G92 statement -original system software available on all OmniTurn's

Non G92 statement -This code is only available on system disks dated after 11/96.

This manual covers only the new method. It is suggested that if you are programming for the first time, or have experience with Fanuc type controls that you use the Non G92 type tool calls. The documentation for the older system is available on request from the factory. If you need these pages please just ask for them, they are available for no charge.

The tool offset is automatically used when a tool call is made, i.e. T1. The slide does not move. Instead the absolute position display changes to show distance of the tool to the absolute zero of the part. Then the next line has to be a move to location with both a X and Z value.

```
T1  
X.5Z 1
```

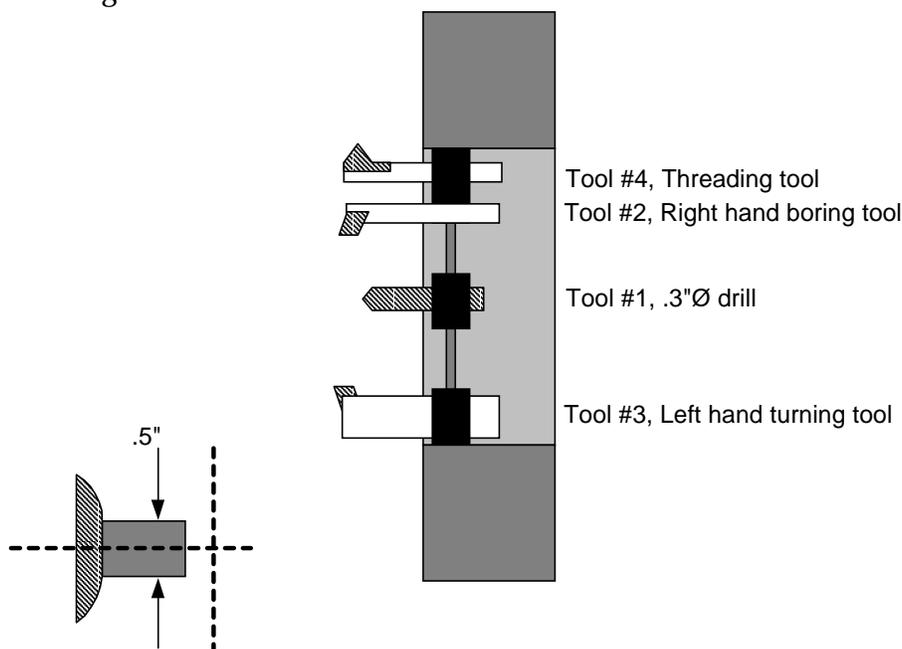
This will move the tool to 1/2" or 1" diameter depending on whether the program is in the diameter or radius mode.

Setting Tool Offset Examples

Tool offsets can be established only after the slide has been HOMED Once that has been done, the offsets can be setup.

There are a few types of tools that are common. Below, you will find a few examples covering these common types.

Tool #1	.3" drill
Tool #2	Right hand boring tool
Tool #3	Left hand turning and facing tool
Tool #4	Threading tool



There is no order that the tools must be set up on the slide. It is not important in what order you do the offsets. It is possible to only do one offset and call it #3. The control would not care that there were no values in unused offsets.

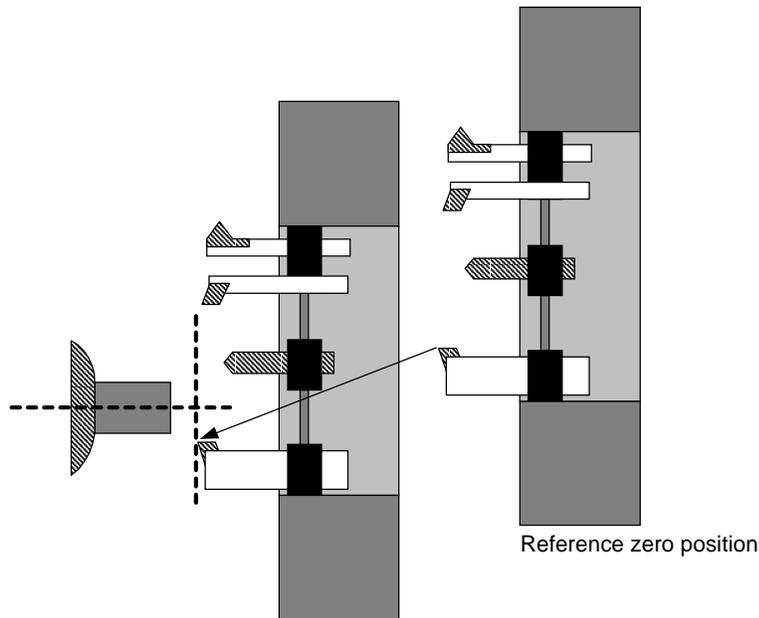
Tool Offsets

If however you call a tool that has not been set, there may be a collision. For the following examples we will assume:

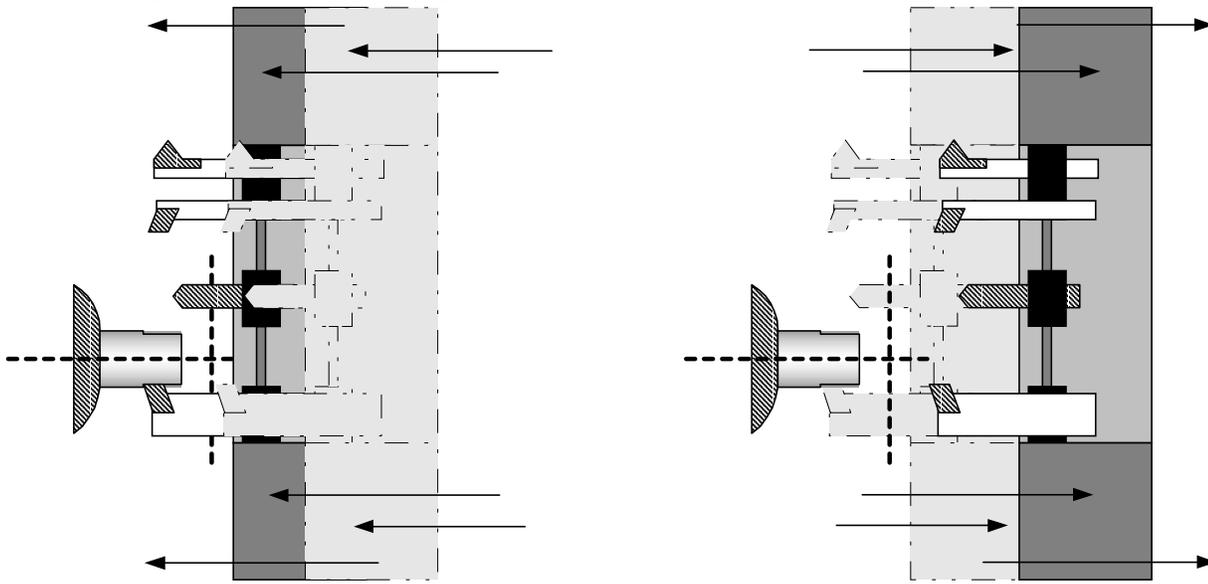
- The material is approx. .5" diameter.
- The part will be programmed so that all of the tools will start at the center of the part in X, and .1" away from the face in Z. Follow this format for the first few programs you write. Later you can be more efficient with time and movements after you have more experience!

To Set Left Hand Turning Tools (Tool #3)

- Be sure that the slide has been HOMED.
- From the main screen go to the Jog mode by typing "J"
- Using the jog keys and joystick move the cutting tool until it is just off the material and slightly smaller.



- Turn the spindle on, select the jog speed for slow, and take a skim pass of the material as shown next.



- Then, move the slide back in Z. **Do not move the slide in X.** This cut will be used to establish the offset.

Tool Offsets

To Set Left Hand Turning Tools continued

1.Slow 2.Medium 3.Fast 4. 0.00005"/.01° 5. 0.00100"/.1° 6. 0.01000"/1° 7. 0.10000"/10° 8. 1.00000"/90° 9.Establish Home H.Go Home T.Set Tool M.Manual Data Input A.Automatic	Alt-C will enable C axis	Position X: +0.00000 Z: +0.00000
		Feed 0.0000 IPM 100%
		Speed 0 RPM 100% Tool 0
Select "T" to start entering tool offset		

At this point, you are still in the jog mode. Instead of selecting a new jog speed now, select "T". The control will now begin the sequence for entering a tool offset. See the next screen.

1.Slow 2.Medium 3.Fast 4. 0.00005"/.01° 5. 0.00100"/.1° 6. 0.01000"/1° 7. 0.10000"/10° 8. 1.00000"/90° 9.Establish H.Go Home T.Set Tool M.Manual Data I A.Automatic	Alt-C will enable C axis	Position X: +0.00000 Z: +0.00000
		Feed 0.0000 IPM 100%
	Tool Number:3_ Press Esc to cancel tool setting	d RPM % Tool 0
Enter tool number you are setting		

After you have selected T, the control will ask what tool it is that you are about to enter. Type the number tool, in this case 3. Then hit the "RETURN" key.

Tool Offsets

To Set Left Hand Turning Tools -continued

1.Slow 2.Medium 3.Fast 4. 0.00005"/.01° 5. 0.00100"/.1° 6. 0.01000"/1° 7. 0.10000"/10° 8. 1.00000"/90° 9.Establish H.Go Home T.Set Tool M.Manual Data Inp A.Automatic	Alt-C will enable C axis	Position X: +0.00000 Z: +0.00000
	Press X or Z to set X or Z tool Position or press Esc to cancel tool setting	Feed 10.0000 IPM 100%
		Speed 0 RPM 0% ol 0
Enter tool number you are setting		

After you have selected the #3 tool offset, the control will ask you whether you want to enter the Z or X offset. In this case we have set the tool on the diameter of the material and we are ready to enter the X offset, so hit X.

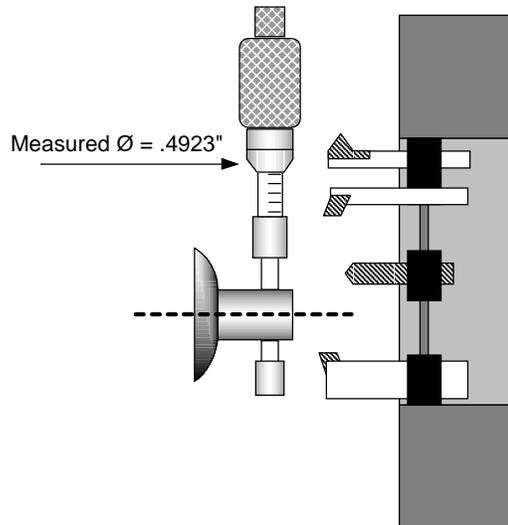
1.Slow 2.Medium 3.Fast 4. 0.00005"/.01° 5. 0.00100"/.1° 6. 0.01000"/1° 7. 0.10000"/10° 8. 1.00000"/90° 9.Establish H.Go Home T.Set Tool M.Manual Data Inp A.Automatic	Alt-C will enable C axis	Position X: +0.00000 Z: +0.00000
	Press X or Z to set X or Z tool Position or press Esc to cancel tool setting Enter current diameter: .4923_	Feed 10.0000 IPM 100%
		Speed 0 RPM 0% ol 0
Enter diameter of material just cut		

Then measure the diameter of the material you just cut accurately with a micrometer. Enter this diameter when the screen asks for it. Remember that this will be a **diameter** measurement.

NOTE: If the tool was touched off on the back side of the part (-X), then enter the diameter as a negative.

Tool Offsets

To Set Left Hand Turning Tools -continued



Take some care but do not be overly careful since any error made here can be easily corrected with the tool offset correction later when you are making the first piece. After typing .4923 hit "RETURN".

Now establish the Z offset, for tool #3

The setting of the Z offset is a little different.

- Touch the tool off in the Z axis and then press T like you did with the X axis.
- The control will now ask for a tool number. In this example you would press 3 and then enter.
- Then press Z when asked which axis you are setting.
- Then the control asks what the location of the tool is from absolute zero in Z.

1.Slow 2.Medium 3.Fast 4. 0.00005"/.01° 5. 0.00100"/.1° 6. 0.01000"/1° 7. 0.10000"/10° 8. 1.00000"/90° 9.Establish H.Go Home T.Set Tool M.Manual Data Inp A.Automatic	Alt-C will enable C axis	Position X: +0.00000 Z: +0.00000
		Feed 10.0000 IPM 100%
		Speed 0 RPM 0% ol 0
Press X or Z to set X or Z tool Position or press Esc to cancel tool setting Enter current Z position:0_		
Enter location of tool relative to zero on the part		

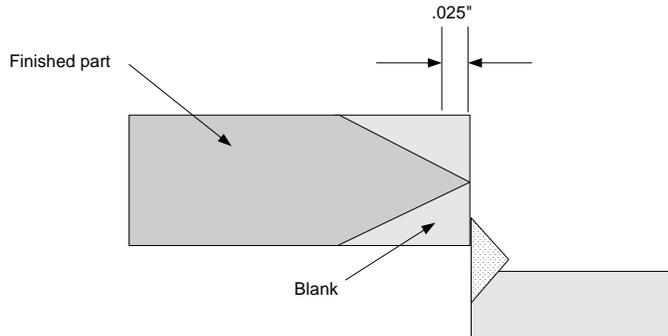
Using a finished part to touch off in Z:

If you put a finished part in the collet against a stop this would give you the absolute face of the part. The parts you machine should have the same location in Z when they are done. So you can jog the tool over the face of the part and touch off and when asked what the location is in Z you could enter 0.

Tool Offsets

Using an unmachined part to set Z:

If you are using a rough part that has material left on the face to be removed you can still use it. Before you put the part in the collet measure how much you have to make the part to size. Then touch off on the known face.



Then when asked what the location is you can enter in the known amount. In the above example you would enter in $.025$.

Working with bar stock:

If you are working with bar stock then the location of the Z start is only important in a relative way. Set the tool starting point far enough from the collet or chuck so there will not be a collision.

Tips - Tool Offset

A short cut for setting both X and Z offsets at the same time:

If you can set your tool in the required location in both X and Z before you go to set the tool with the "T" command it is possible to set both tools at one time without having to ESC and re-enter the T command.

To do this follow the procedure to get the tools into location, then press T and set the X tool, then before pressing ESC, press Z. This will set the Z tool and then exit back to Jog.

<pre> 1.Slow 2.Medium 3.Fast 4. 0.00005"/.01° 5. 0.00100"/.1° 6. 0.01000"/1° 7. 0.10000"/10° 8. 1.00000"/90° 9.Establish Home H.Go Home T.Set Tool M.Manual Data Input A.Automatic </pre>	<p>Alt-C will enable C axis</p> <p>Position of selected tool</p> <p>Selected tool</p>	<pre> Position X: +0.49230 Ø Z: +0.00000 Feed 10.0000 IPM 100% Speed 0 RPM 100% Tool 3 </pre>
---	---	---

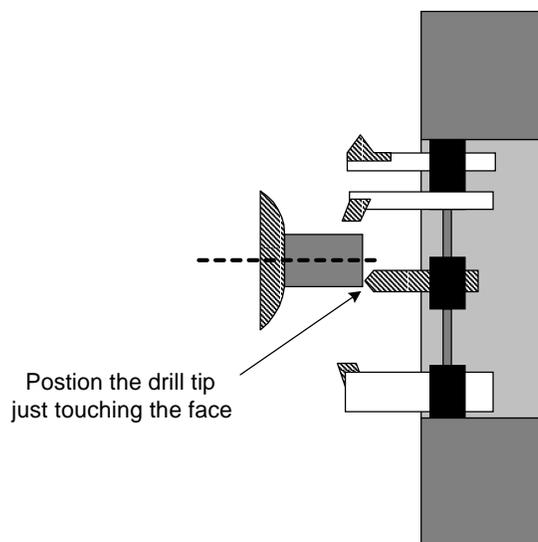
After setting Z position, Set Tool dialog box closes and the Position box shows true position of selected tool.

Tool Offsets

- The screen will show "Z OFFSET ENTERED". You are now done with the Z offset for tool #3. Press "ESC" to return to the Jog menu.

Setting a Drill (Tool #1)

- Start the same as with the left hand turning tool, be sure that the slide has been HOMED
- If you are not in the jog mode, enter it now.
 - Put a 5/8" collet in the headstock.
 - Get a 5/8" pin and put it in the collet
 - Put a 5/8" bushing holder on the tool slide
 - Jog the slide until it is located where you want the drill to be.
 - With the 5/8" pin still in the collet, slide the bushing holder onto the pin.
 - Tighten the bushing holder to the slide.
 - Without moving the slide start the tool offset procedure
 - Start with the X offset
 - When asked, Set the diameter measured as 0
- Press "ESC" to go back to the Jog mode.



Setting a Drill (Tool #1) -continued

The Z Offset for the drill is done the same way the left hand tool was done:

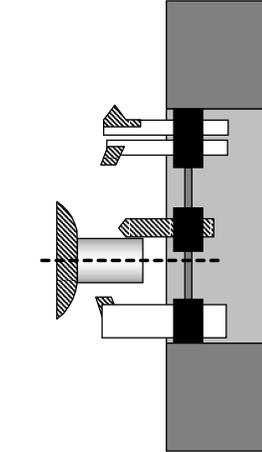
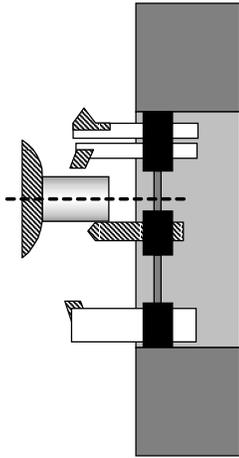
- Touch the drill tip to the face of the part
- Select "T" from the jog menu
- Enter the tool number, in this case it is still # 1, then "RETURN"
- Select the axis being set, "Z".
- Now enter the location of the tool from zero. This could be 0 when touching off on a finished part.
Then
- Press "ESC" to get back to the jog mode.

Tool Offsets

An alternative method of setting drill:

Position the tool as show below. Either side of the tool will do.

Setting up a drill on the + side of the spindle



Setting up a drill on the - side of the spindle

Above there are two ways to setup a drill. In both cases the tool is not used to cut as we did in the first example with the left hand turning tool. "Touch" off the material with a feeler gauge or piece of paper. The sign of the diameter entered will tell the control which side of the spindle your tool is on.

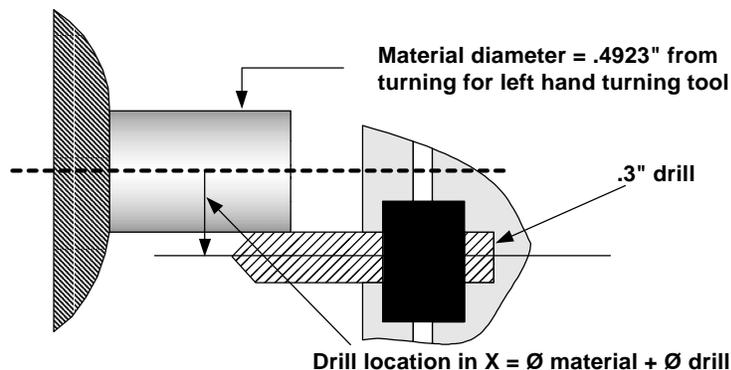
- (+) side of spindle: enter diameter measured as a positive number
- (-) side of spindle: enter diameter measured as a negative number

With the tool just touching the material select "T" to start entering the tool offset.

- Next the control will ask "OFFSET NUMBER ?". Type the Tool #, in this case it will be 1, and then "RETURN",

-The next question is "PRESS X OR Z TO STORE PRESENT X OR Z-AXIS OFFSET", type X

- **"WHAT WAS THE TURNED DIAMETER?"**. This question can be answered two ways. The answer depends on what side of the spindle the tool is located. The sign (+) or (-) will change with the tool location. In either case the value of the offset will be the same.



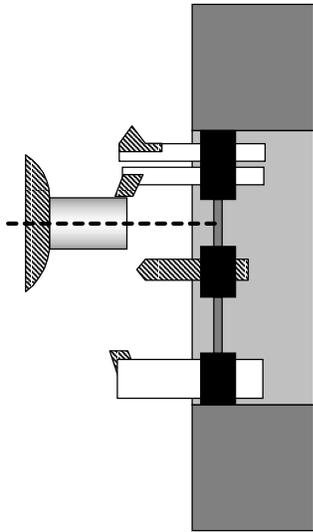
The value is equal to the diameter of the material plus the diameter of the drill. In the above figure you can see that the location of the center of the drill is the sum of these two numbers. .3" drill + .4923" material = .7923" location of drill center in X. For this example we are on the positive side so type .7923 and "RETURN".

- Then set the Z as you have been shown before.

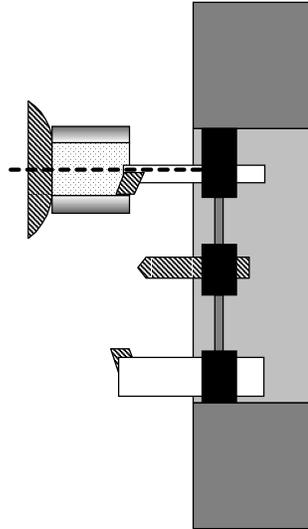
Tool Offsets

Setting ID Tools, ie Boring tools & Threading tools

The procedure for setting ID tools is similar to the two previous tools. The only difference is how you will touch off to determine the turned diameter.



Setting a boring tool on the OD.
The measured \varnothing will be entered as a
NEGATIVE

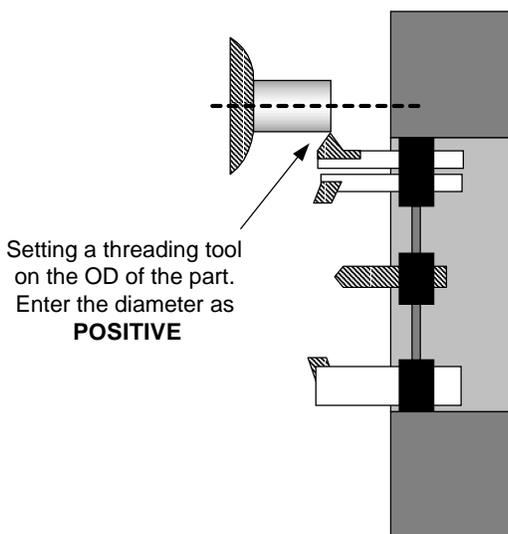


Setting a boring tool on the ID.
The measured \varnothing will be entered as a
POSITIVE

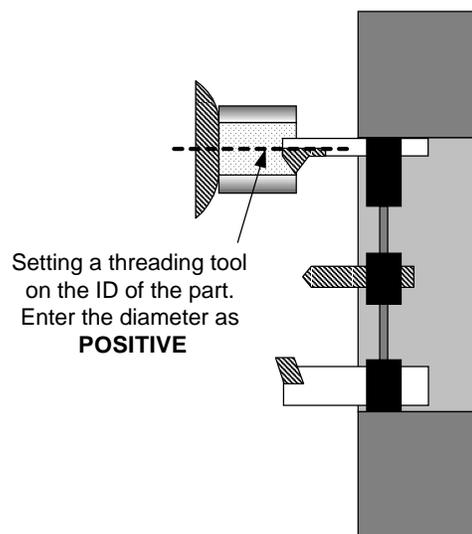
Setting Threading tools

The threading tool is set similar to the other tools.

Setting X: The turned diameter is set like the other tools you have done. The offset can be set on the OD or ID.



Setting a threading tool
on the OD of the part.
Enter the diameter as
POSITIVE

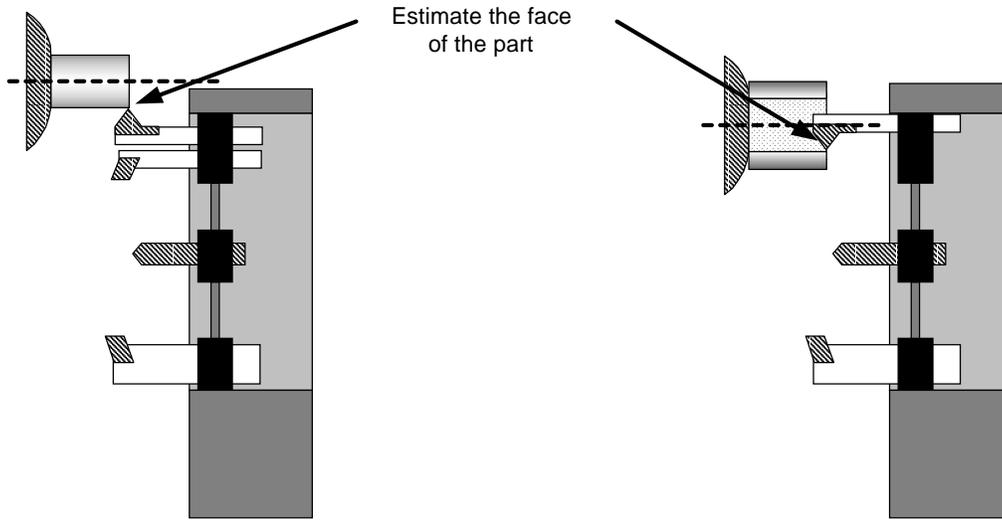


Setting a threading tool
on the ID of the part.
Enter the diameter as
POSITIVE

Tool Offsets

Setting Threading tools, continued

Setting Z: The approximate location of the tool can be done by eye with the corner of the piece.



OmniTurn Start-up sample part

OmniTurn Sample Part

Welcome to the OmniTurn. This document is a tutorial used to run a first program with the OmniTurn. It is suggested before you try to work with this tutorial that you spend some time reading the manual and gain a basic understanding of the programming and operations of the system.

The goal of this tutorial:

- first looking at the print
- tooling
- programming the part
- checking the tool offsets
- test running the program
- cutting a part
- correcting errors in the program
- adjusting the part to size with offsets

Materials needed for this tutorial:

1/2" collet
1/2" x 3" aluminum, brass, or other easily machined material
LH turning tool and holder
LH threading tool and holder
1/4" drill
5/8" bushing holder
1/4" bushing
#2 center drill
5/8" bushing holder bushing for center

Starting the sample part

A note about the options for the OmniTurn

The OmniTurn has a number of options. Such as:

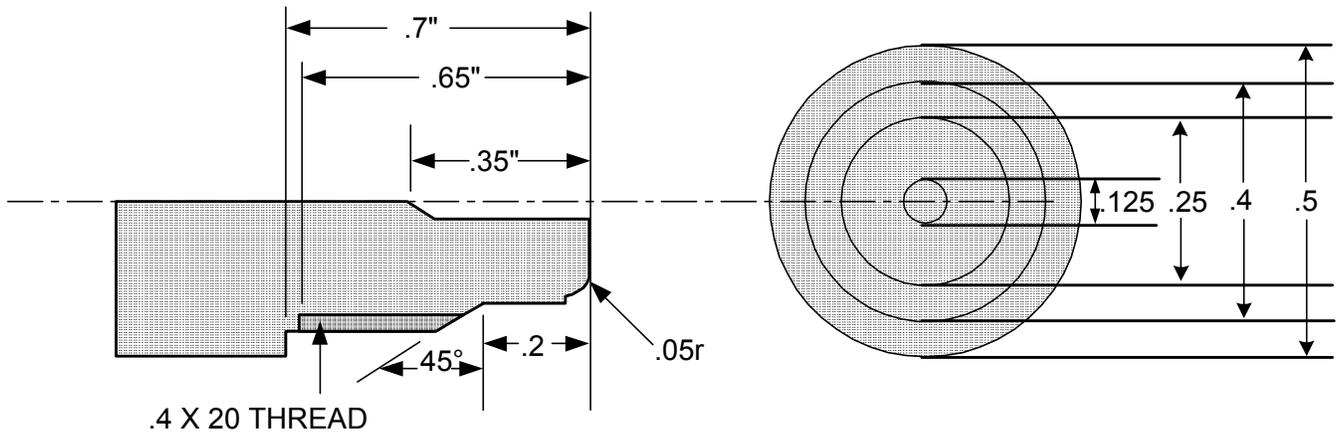
Spindle control - On / off
Spindle speed control - infinitely
Threading encoder
I/O additional "M" functions

Sample part layout

Please take a look at the following part. You will notice that there are a number of operations that must be performed (otherwise what fun would this teaching part be). We will be doing the following:

Roughing the OD
Center Drill
Peck drill to depth
Finish the face and OD
Single point threading
Deburr with the turning tool
Make a single pass on the thread for cleaning and deburring

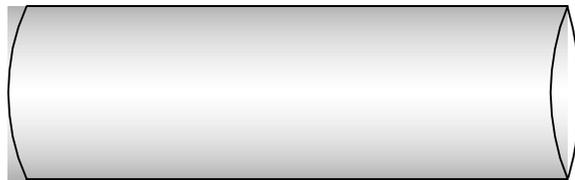
OmniTurn Startup sample part



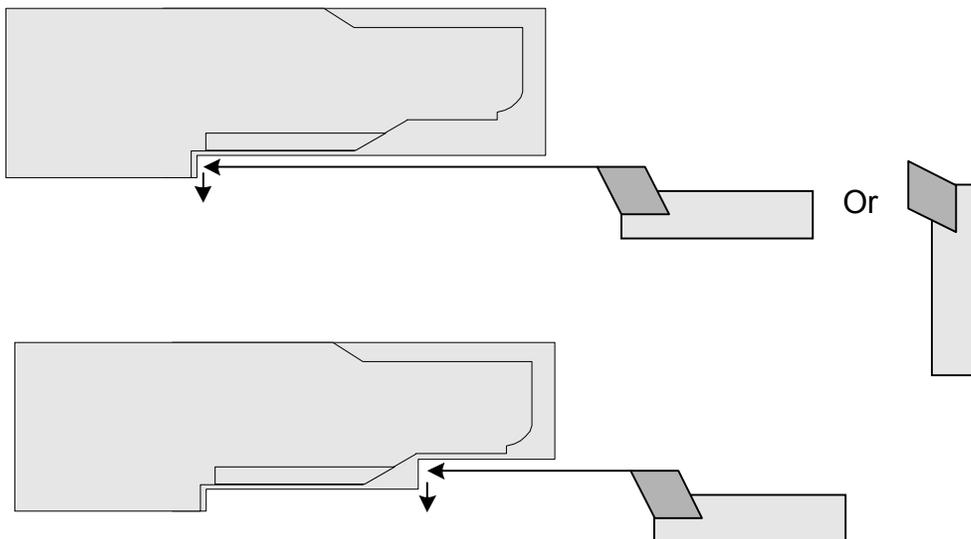
If you are familiar with machining you might want to skip to the next section, "TOOLING"

OPERATIONS

We will start with a solid piece of material 1/2" in diameter, 3" long.

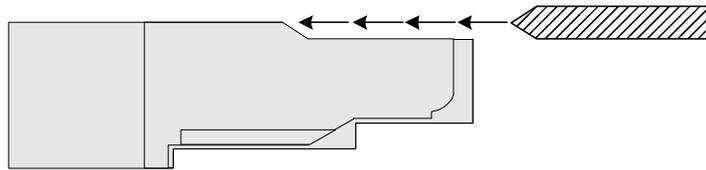


In machining this part we will use the same tool for roughing and finishing the OD. This does not have to be the case. You could setup another tool for the finishing pass. We will be doing the roughing first.

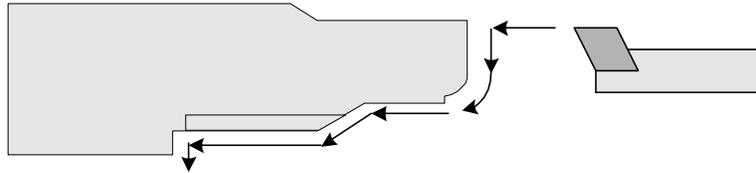


We will take two passes to rough the part. After we rough the OD and remove most of the material we change tools and center (spot) drill. Then we change tools again and peck drill the hole to finish depth.

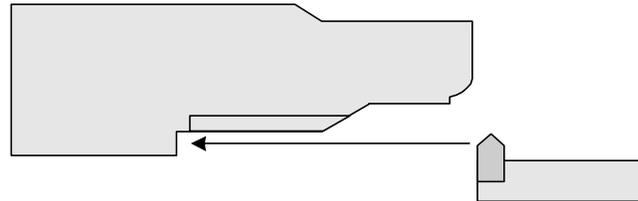
OmniTurn Startup sample end



After the drilling we change back to the turning tool to face and finish turn the OD.



With the OD finished we can thread. (if your attachment does not have an encoder for threading you will have to skip over this operation)



After the threading is complete we will change back to the turning tool to turn the major diameter of the thread to help deburr the OD. Then to make sure that all of the threads are clean and burr free we can take a single pass with the threading tool at it's finished depth. If the appropriate threading insert is used, you would not have to do the last 2 operations.

Selecting Tooling

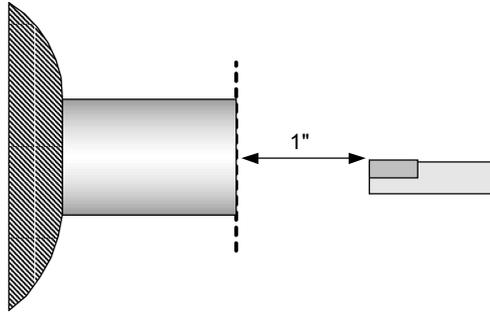
Before we start to program there are a number of decisions that must be made. Tooling is one. First you have to select the tools to be used. This has been gone over in the previous section but they have not been formally listed yet. Now we can assign tool numbers:

Tool #1	Left hand turning tool
Tool #2	Center tool
Tool #3	1/8" drill
Tool #4	Threading tool

Then we figure out where we want them to start. This does not mean where they will be on the slide. When you write the program, the physical location of the tools does not matter, this will be taken care of during the "setup". What we need to take care of at this point is the "logical" starting point. This is where we will take over the movement of the tool with the program. It is important to note this information down for the person that will be doing the setup at the machine. Even if you are the one that will be doing the setup it is good practice to make a setup sheet and write down the starting locations for the tools.

For this sample part we will start tool #1 a little bit in front of the part in Z and on center in X. Please note the next diagram. We will start the tool 1" in front of the part. This is a very safe distance to rapid the tool too and it is not too far away so as to waste too much time. This amount will vary depending on the type of tooling used and preferences of the programmer. As you get used to the system you can make this distance smaller to gain cycle time. Also note that we have all the tools starting 1 inch away, you can vary this. We made them all the same so that setup will be a little easier at first.

OmniTurn Startup sample part



For each of the tools we have to establish a starting location. This location is not that important to worry about much. Just pick a location for each tool that will be safe to rapid to without worrying about collisions. Now we will add some information to the tool table that we started before:

Tool #1	Left hand turning tool	X=0	Z=1
Tool #2	Center tool	X=0	Z=1
Tool #3	1/8" drill	X=0	Z=1
Tool #4	Threading tool	X=0	Z=1

Programming

Now that we have the part layout and a general machining sequence we can begin writing the program. If you want to try and write a program before you read ours, now is the time to try. Do not get upset if your program is not the same as ours. Each machinist will program the part differently, this is normal. Only concern yourself with understanding what we have done and how our program works so you can work with the general format to create your own.

In order to enter the part into the OmniTurn you will have to enter a new program:

Refer to pages 1.10 - 1.12 for startup and establish home procedure.

Refer to pages 1.19 - 1.22 for entering Automatic Mode and File Handler

At the File Handler screen, press Alt-N (Hold the Alt key down, then press 'n' key) to start editor.

To enter Automatic Mode there must be a file in memory; you must select from list or create a new program before Auto Mode screen will appear.

Now you should be ready to enter the new program. Try typing in the program for the practice. If you do not want to spend the time entering in the program, load the disk supplied with the start-up tooling kit. The sample part is named SAMPLE 1

NOTE: Code in Italics are optional. Not all controls will support these lines.

- G90G94F300G73 • This program is in absolute, feed rate is set at 300 ipm. Start every program like this. There are very few parts that require G91 programming. Note that this program is written in radius mode -G73
- T1 (LH turning tool) • Makes tool 1 offsets active
- X0Z1 • Moves the tool to a safe distance from the face
- M01 • I put in an optional stop so when I run the program the first time I can

OmniTurn Startup sample part

have the tool automatically stop before it takes a cut and look to see if it came to the right location.

X.1 Z.025

- We will use the first tool to act as a material length stop. Here we are positioning the tool a little off center and in front of zero of Z. There are two reasons we stop the tool a little on the plus side of zero: when the collet is closed the part will pull back some, the other reason is we want to leave a little material to be faced off.

M00(close the collet)

- Stop the program to allow the operator time to pull the material out to the stop and close the collet. The comment will alert the operator to perform this.

M03S2000

- Turn the spindle on at 2000rpm.

This command operates an optional function. If your machine does not have spindle control please do not enter this command. What you might want to do instead is to add a comment on the previous line "and turn the spindle on".

X.27

- This moves the tool beyond the major diameter so it can move in Z for a rough facing cut. This move is still in rapid

G96S250

- This changes the spindle speed to "constant surface feet" mode. We will be cutting at 250 sfm. If you have a spindle drive option you can choose to cut in this mode or just leave it in RPM.

G77S3000

- Sets maximum spindle rpm to 3000 rpm so the spindle won't go to fast while facing off the part. If you had the part in the chuck you would probably make the G77 speed much lower.

G04F1

- This is a dwell to allow the spindle to come up to speed before cutting

Z.015

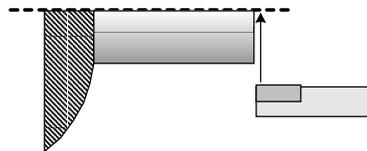
- This positions the tool in Z for the rough facing move. (rapid feed rate)

G95F.003X-.015

- Here we change to IPR at .003" per rev. We move the tool past zero in X to face the part. We move past zero since there is a radius on the tip of the tool. This move assumes that the radius is .015". Please adjust this move to coincide with the tool you are using.

X.13

- Move the tool for the second roughing pass, still in rapid



G94F300Z.025

- We move the tool off the face of the part in rapid

G97S2000

- Changes spindle speed to rpm mode, and sets speed to 2000 rpm.

X.205

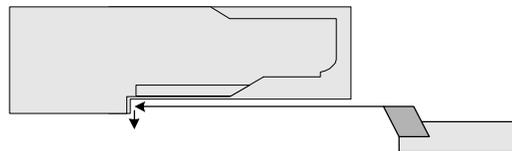
- Position the tool to take the first roughing pass, this is still in rapid

G95F.003Z-.75

- Change the feed rate to IPR and move the tool to take the first rough pass

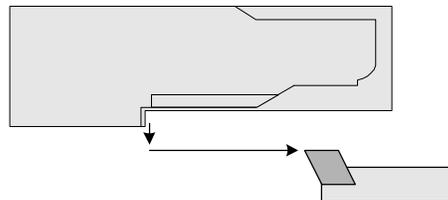
X.26

- This moves the tool out in X at the same feed rate of .003"



G94F300Z.025

- The tool is moved at rapid to clear the face of the part



S2500

- Change the spindle rpm to a higher speed for the next smaller diameter pass.

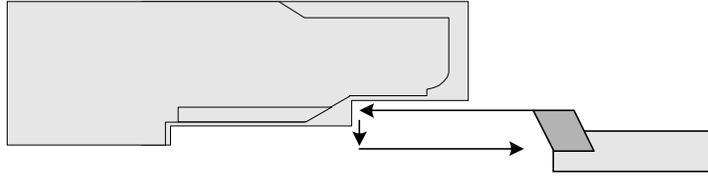
G95F.003Z-.195

- Change to feed mode, and make the second pass

OmniTurn Startup sample part

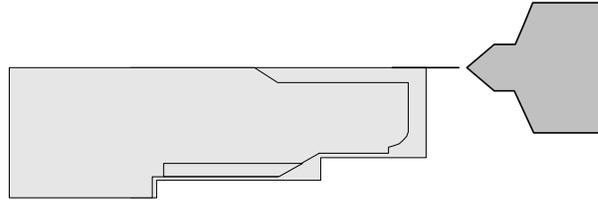
X.225
G94F300Z1

- This clears the tool, still in the feed mode
- Change to rapid and move the tool to Z 1, this is clearance for a tool change



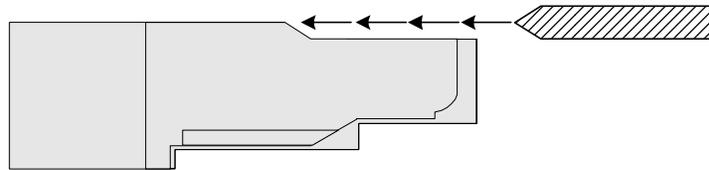
T2S2500(Center drill)
X0Z1
M01
Z .025
G 95 F.002Z-.025

- The center drill is called into position and the spindle speed is changed.
- The tool is moved to a safe location
- Optional stop for tool check on the first run through
- The tool is rapidly sent to the face of the part with about .02 clearance
- Feed the center tool into the material at a feed rate of .002"/rev



G94F300Z 1
T3S2800(Drill)
X0Z1
M01
2.025
G95F.003
G83Z-.35K.1L300

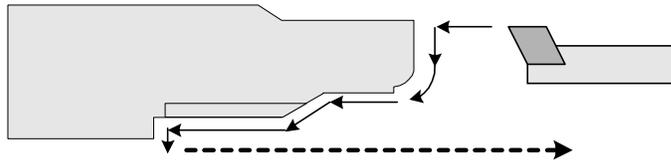
- Rapid the tool clear of the part for a tool change
- Call the drill into position and change the spindle speed
- Move the drill to a safe location
- Optional stop for the tool check on the first run through
- Rapid the drill to the face of the part with a little clearance
- Change to the feed mode of IPR
- Peck drill the hole, .1" per peck, feed rate of .003 IPR, rapid at 300 ipm



G94F300Z1 S2400
T1 (LH tool .007 TNR)
X0Z1
M01
G77S3000
G96S250
G04F1
G41
Z.025D1
G95Z0F.002
X.075
G02X.125Z-.051.075K-.05
Z-.2
X.2Z-.275
Z-.75
X .26
G97S2000
G94F300Z1
G40

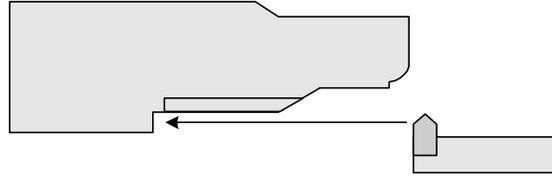
- Move the tool to clear the part for a tool change and change spindle speed.
- The turning tool is brought into position again for the finish pass
- Move the tool to a save location
- Optional stop for the tool check on the first run through
- Set maximum spindle speed to 3000
- Change spindle speed mode back to sfm @250
- Dwell 1 second to allow the spindle to change RPM
- Turn on tool nose radius compensation
- The tool is rapidly sent close to the face of the part & turn on comp with D1
- The tool is fed to the face of the part to begin the finish contour
- The next few lines turn the part
- Spindle back to RPM mode and set speed to 2000
- This clears the tool in Z to 1 at rapid feed rate
- Turn tool nose radius compensation off with move off the part

OmniTurn Startup sample part



T4(Threading tool)
X0Z1
M01
G04F1
X.2
Z0
G33X.175Z-.65K.05I.004

- Calls the threading tool offset
- Moves the threading tool to a safe location
- Optional stop for the tool check on the first run through
- This is a dwell used to allow the spindle to change speed
- Moves the tool to the starting diameter of the threading cycle
- Moves the tool to .2” from the start of the thread in Z
- This is the threading command

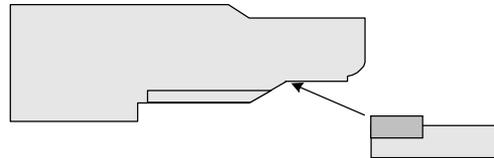


Z1
T 1(LH Turn tool)
X.15Z.1

- This clears the tool in Z to 1 for the tool change
 - Call the turning tool for the deburr pass
 - First move to location for the tool, more aggressive location than before
- Note: I did not put a M01 after this tool call since this tool was already called and it was OK. So why stop it again?

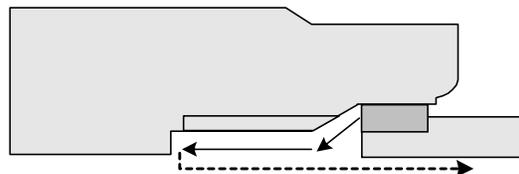
G41
Z.01D1
G95F01 X.125Z-.2

- Turn on TNR comp
- Move tool to a spot so it can start the deburr pass
- Move tool to start of deburr (IPR), faster than normal cut pass



X.2Z-.275F.003
Z-.75

- Start deburr pass



X.26
G40
G94F300Z1
T4S2000(threading tool)
X0Z1
G04F1
X.2Z0
G33X.175Z-.65K.05I.004O
M05

- Clear tool
- Call the threading tool for a single pass to cleanup thread
- Establish tool location
- Dwell for 1 second to allow the spindle to change speed
- Moves the tool to same start location as first threading cycle
- This is the threading command for a single pass (note O)
- This turns the spindle off. We are not waiting for the M30 at the end of the program to turn the spindle off, this saves a second or two
- this is a clearance move
- End of the program, the control will now be ready to run another part

Z1
M30

OmniTurn Startup sample part

Saving the program to the disk

- Now to save the program press F1 to get back to the main page of the word editor
- Then press F3 to save the program to the disk
- Then press F2 to exit the program

Verify the program

Before you run the program it is best to look for possible mistakes. Even the most experienced programmers make simple typing errors. The verification software will point out possible errors.

First make the program active by entering in the name when asked "FILE TO BE PROCESSED"

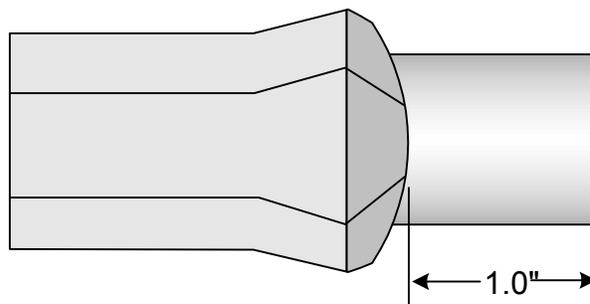
Then press F4 - Please see the notes on using the verification software in chapter 5

Setting the tools on the slide

Now that you have written and entered the program it is time to set the tools on the slide. There are a few considerations that must be made when you are putting the tools on the slide, remember:

- Set the tools so that they will not interfere with one another during tool change or during a cutting cycle. Be sure to consider both the length and width clearances.
- Keep the tools as close together as reasonable so there is not too much time wasted during tool changes.

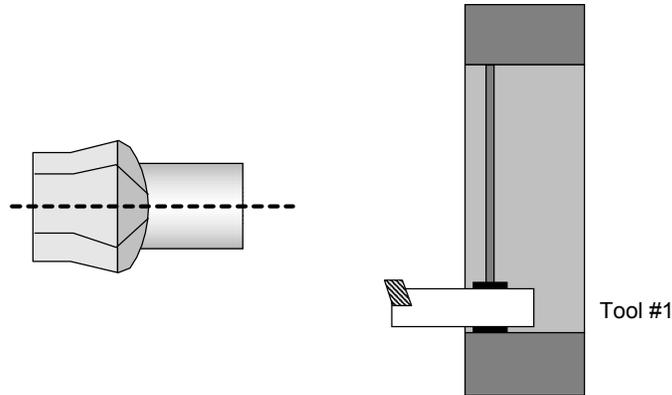
To set the tools we first have to put a blank in the collet. Set the blank so that it sticks out about 1 inch. This will be enough to machine the part and not hit the collet or spindle.



Then put the turning tool on the slide. For this example put it on the near side.

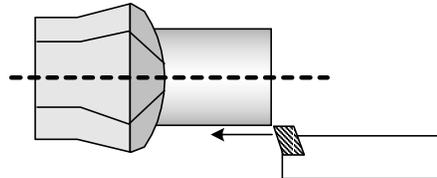
OmniTurn Startup sample part

Setting X axis

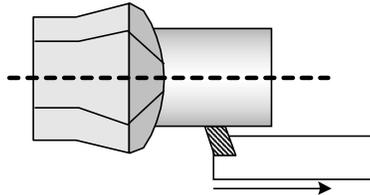


Now go to the jog mode and follow the procedure on setting up tool. Turn the spindle on, (if you have a unit with spindle control do this in the MDI mode first).

Move the slide and take a light cut on the diameter.

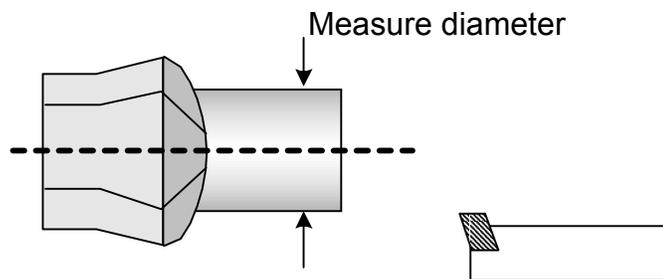


Then move the slide in Z only to clear the material, do not move in X!



Turn the spindle off. If you have the spindle control use the switch on the OmniTurn panel.

Then measure the diameter with a micrometer



Now press T - "SET TOOL" on the keyboard to start inputting the tool offset.

After you press T the control will ask you to input a tool number. Now press 1 and return.

Next you have to tell the control that you are setting the X axis for the tool offsets, press X.

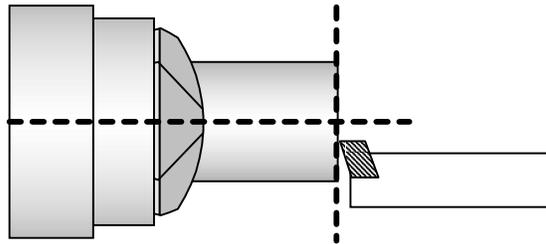
OmniTurn Startup sample part

Now the control asks for the **diameter** of the part you just cut. Enter this measurement and press return.

To go back to the jog mode and continue with the entering of the offsets press the ESC key.

Setting Z axis

Now you have to set the Z offset for the tool. Turn the spindle on. Jog the tool until it touches the face of the part. Once you get close to the face of the part select the #5 setting for a jog speed of .001" per jog stick movement. This allows you to just "touch" the face of the part. Then press T to start the input of the Z offset for T 1.



Then press 1 when asked for the tool number, and RETURN

Then select Z

Then the control asks the location of the tool in Z. Since we are going to use the face of the bar as Z=0 you can enter 0 and press RETURN

Press ESC to continue in the jog mode.

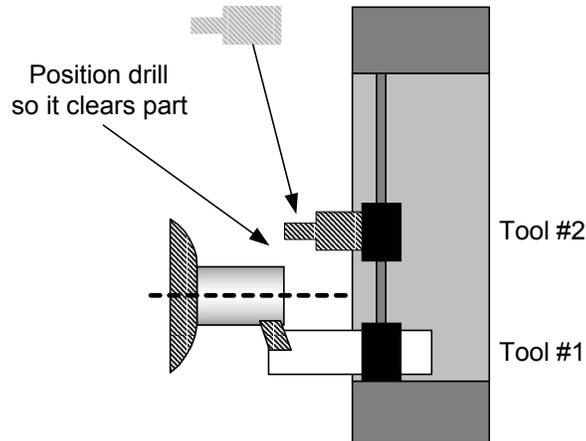
Now you can setup the center drill and drill. There are a number of ways to setup drills. We will describe one technique. You will need a 1/2" collet, bushing, bushing holder, and pin.

The first concern for this type of tooling is to establish where you want the next tool to be on the tooling plate. This is important since there is a possibility that if you position the new tool too close to the last tool that there can be a collision. Remember this is a gang tool machine. It does not have a turret. When you move a tool you are also moving all the other tools at the same time. You have to set the tools so that while you are working with a tool all of the others do not collide with the: part, collet, spindle, etc. A simple way to set the tooling is:

Put the control into the jog mode and move the first tool to its worst case position and then put the next tool on the slide so there is enough clearance with all obstacles. This does not have to be very exact with most examples since it is only for clearance. When you get into high production jobs like this can be very important since if the tools are too far apart there is a lot of wasted movement and the cycle time will be too long. For this example the tools should be positioned so there are no collisions and the amount of wasted moves is limited.

OmniTurn Startup sample part

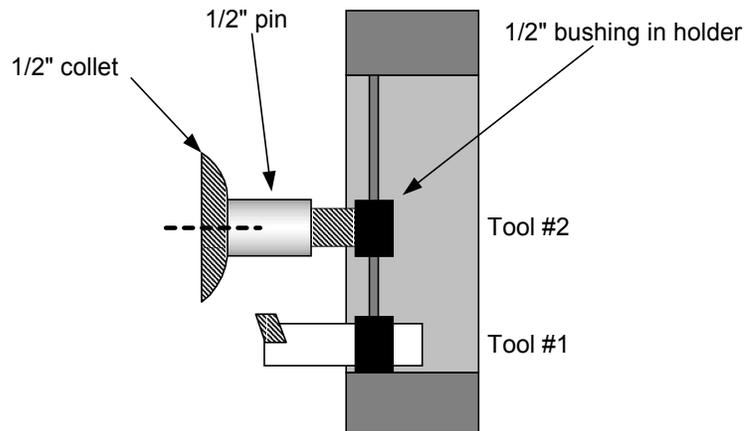
Setting X for a Drill



With the slide and tooling set as shown above lightly secure the center drill and holder to the slide.

Then put a 1/2" bushing into the holder and a 1/2" collet in the spindle. Jog the slide until the tool holder with the 1/2" bushing is in line with the spindle. Now move the slide close enough to the collet so you can put a 1/2" pin into the collet and bushing holder at the same time. Loosen the bushing holder and line it up exactly with the collet. Then lock the collet holding the pin so we know that it is directly in line with the pin and bushing. Then lock the bushing holder in place.

Now we can establish the tool offset location for the center drill in X.



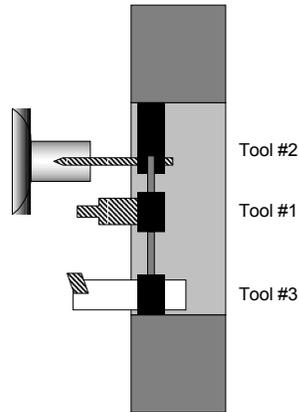
- In the jog mode select "T"
- When asked for the tool number, enter "2" and return
- Press X for the axis you are entering
- Enter "0" for the diameter measured and return
- Press "Esc" to return to the jog mode

Now move the slide back in Z and put the correct bushing and center drill into the holder. The Z location for the tool is next.

- Jog the tool until it just clears the part, with the spindle off loosen the tool and slide it forward until it touches the face of the material. Then tighten it in place
- Press T to start the tool offset input procedure.
- When asked for the tool number, enter 2 and return
- Press Z for the axis to be set.
- Enter 0 for the current location, and return.
- The Z is set, press the "Esc" key to return to the jog menu.

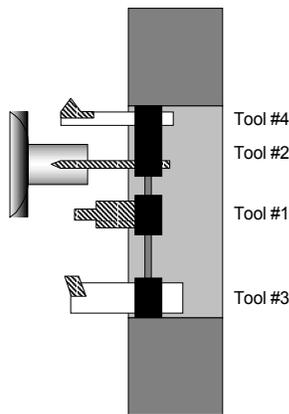
OmniTurn Startup sample part

For Tool #3 follow the same procedure with setting the drill.

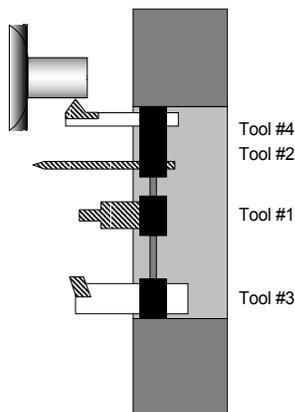


Next is the threading tool. Here we will use another technique. Instead of cutting with the tool we can use the cut surface from setting the first tool to establish the location of this tool.

First, while the slide is still on center with the drill, mount the threading tool so it does not collide with the spindle or the work piece.

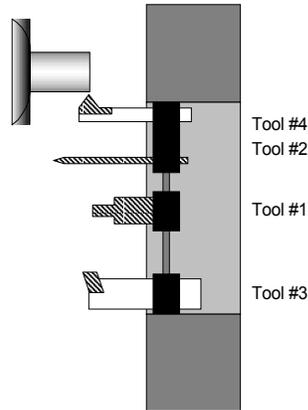


Then jog the slide until the threading tool just clears the turned diameter, it would be best to use a feeler gauge or slide the tool until it just touches the turned diameter. You can do this with the spindle off. If the spindle is on, then you would jog until the tool just touches the material and makes a chip. With the tool touching the turned diameter select "T" from the jog menu to enter the X tool offset for tool #4. Then when the control asks what the turned diameter was enter the value given for this turn from tool #1. Then press RETURN.



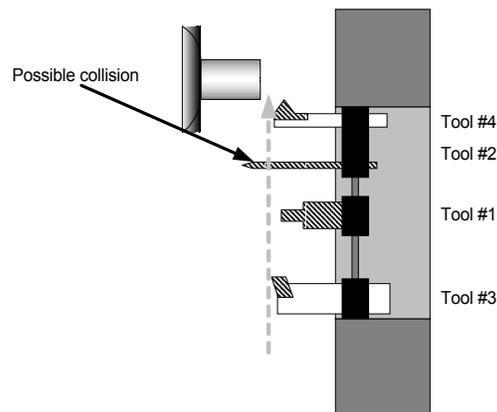
OmniTurn Startup sample part

Then press ESC and jog the tool until it is just even with the end of the material. This location is generally not that critical so don't waste too much time.

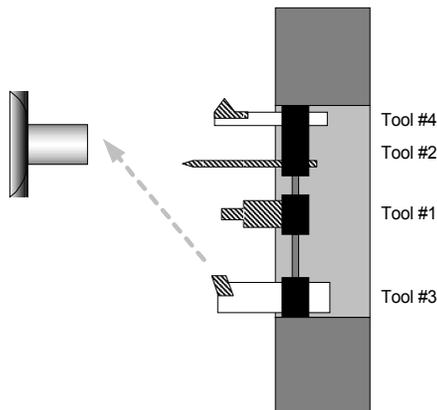


Now that you have the tool where you want it in Z you can establish the tool offset. Press T to start the entry and then select Z. Then enter 0 for the current location and press RETURN. When the control tells you that the offset is entered press Esc to get back to the jog mode.

Now you are done setting the tools. The next step will be to test run the program and make sure that it does what you expect it to do. First before we leave the Jog mode move the tools into a position that will allow the program to run without colliding on it's first move. If we left the slide as shown above we might hit the long drill on the way to the LH turning tool.



If we move the slide back and over a little there will be no problem.

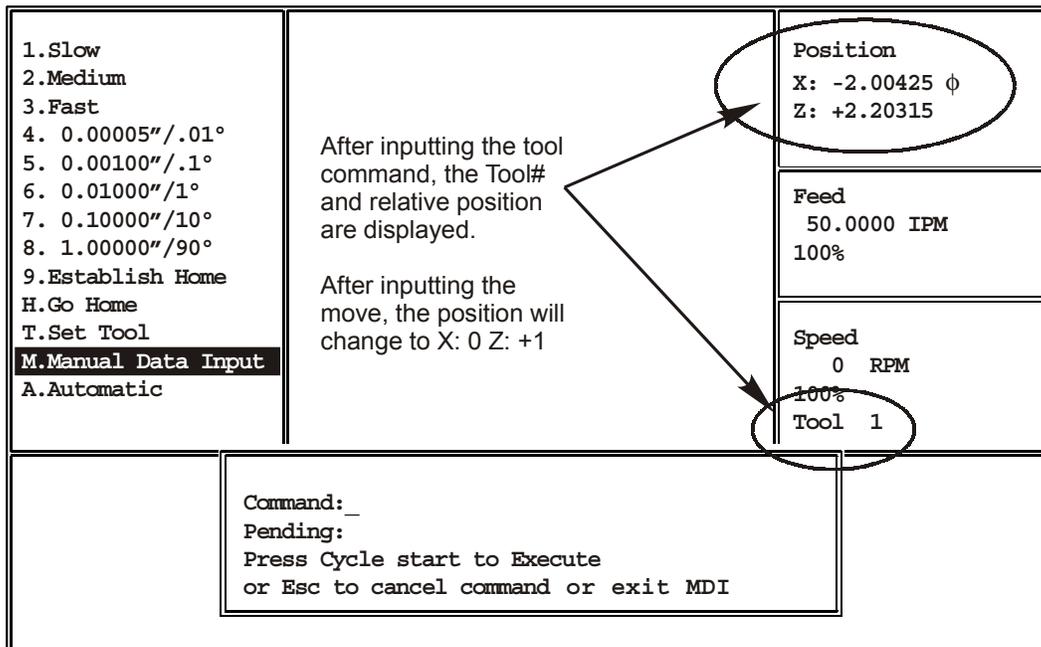


OmniTurn Startup sample part

Checking the tool offsets

Once you have entered the tool offsets you might want to check and make sure the tools go where you expect them to. One way to do this is in the MDI mode. Here you can call the tools and they will go to the set tool offset positions. To get to the MDI mode press M. **Refer to Page 1.18 for a description of the MDI Mode.** In the MDI mode you can enter one command at a time and execute it.

To call the tool into position, first enter the command that calls the tool: **T1**, then press enter, then cycle start. (If you make a mistake, press the ESC key to redo.) The tool will not move, but the position display will show the distance from the face and center of the work piece to the current tool location. If this distance does not look correct you might want to reset the tool. To move the tool, type the command **XOZ1F50**, press enter, then Cycle Start. The tool will move to center and 1" from the face of the work piece at a feed rate of 50 ipm. If you want to stop the motion press the Motion Stop button on the face of the control. To continue, press the Cycle Start again.



After you check the first tool you can enter the command to locate the second tool: T2. If there is a problem with the location of a tool you can go back to the jog mode and reset the tool. To exit the MDI mode press J, this will bring you back to the Jog menu.

Inputting secondary offsets for TNR Compensation

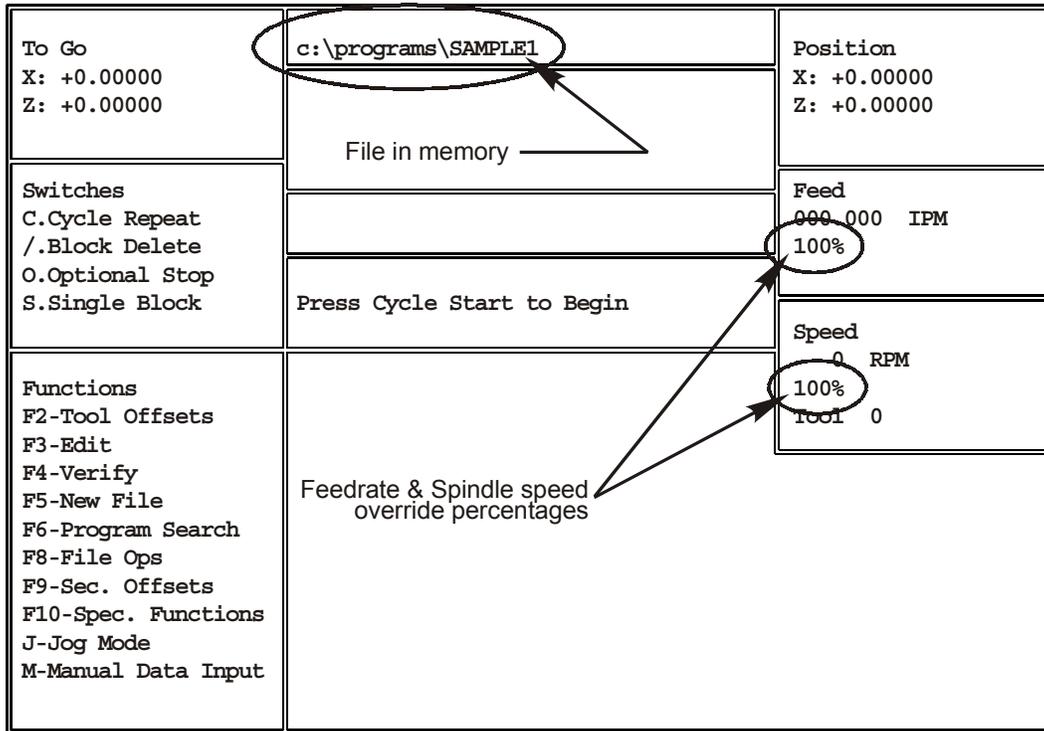
Before you run the program the values to be used for the TNR compensation must be entered. Go to the Automatic mode and then press F9 Sec. Offsets

- When asked what secondary offset you want to adjust press 1 (for D1 in the program) and RETURN
- If there is a value in X, press C to clear it. Then enter -.014 and RETURN. This value is twice the TNR, and the direction is (-).
- If there is a value in Z, press C to clear it. Then enter -.007 and RETURN. This value is twice the TNR, and the direction is (-).
- Then enter the size of the TNR into R. .007 and press RETURN.
- Then press ESC to get out of the secondary offset entry page.,

OmniTurn Startup sample part

Testing the program

Now that you have made sure that the tools go to where you think they should it is time to test the program you have written. Go to the Automatic mode screen. If you were still in the MDI mode you would press **A** to go to Automatic Mode. Then at the Automatic mode screen you have to enter the name of the program to be run. If it is already listed as **the FILE IN MEMORY** and the control asks you to press cycle start, then you are ready to go on. If there is no file in memory then enter the name now. If there is another program active press F5 to allow you to enter the new name now.



There are a few ways to test a program. Making the program run in slow motion is one of the ways to confirm what the program will do. If the slide starts to take off in a direction you did not intend you will have time to stop the slide and correct the program without having a collision.

Feed rate overrides:

Use the Feedrate Override Pot lower the feedrate as a percentage. As this pot is adjusted, the percentage in the Feed box changes. This reduction is applied to both feed and rapid moves. You do not have to wait for the program to start in order to reduce the feed rates; adjust the pot to 0%, start the program, then bring up the feedrate as desired.

Feed Hold:

If you want to stop the motion of the slide press "FEED HOLD" on the control panel. This will stop the motion of the slide. If you want to continue press CYCLE START. If you want to exit the program and make some corrections press ESC. This will turn off M functions and stop the program.

Optional Stop (M01):

In the example program we put M01, optional stops after each tool change. This enables you to run the program with slower feedrates as describe above, and have the program automatically stop after each tool change so that you can check to make sure the tool is in the correct position. If the tool is not where you expected it to be, you can press ESC and then go to set the tool again. If the tool is in the expected location, then press cycle start and the program will continue.

To activate the Optional stop mode press the letter O when in the automatic mode before you start the program.

OmniTurn Startup sample part

Testing the Program, con't

Single Block mode

This mode will execute one line of the program and wait for you to press cycle start again to execute the next line. This way you can see what will happen with your program before it happens. This is really good to do if you are not sure what the program is going to do. After you gain some experience with the control and are comfortable with stopping the control you might skip using this option.

To activate this mode press "S" from the Automatic mode page.

To Go X: +0.00000 Z: +0.00000	c:\programs\SAMPLE1	Position X: +0.00000 Z: +0.00000
Switches C.Cycle Repeat /.Block Skip O.Optional Stop S.Single Block	Press Cycle Start to Begin	Feed 0.000 IPM 100%
Functions F2-Tool Offsets F3-Edit F4-Verify F5-New File F6-Program Search F8-File Ops F9-Sec. Offsets F10-Spec. Functions J-Jog Mode M-Manual Data Input		Speed 0 RPM 100% Tool 0

Press 'S' key to activate Single Block mode

This mode will show you the line of code that you are currently executing and the next line that will be run. This way you can make sure that the code you are running will go where you think it should. If you find an error and want to edit the line before you run it you can press ESC and then go back and edit the program before you run the line.

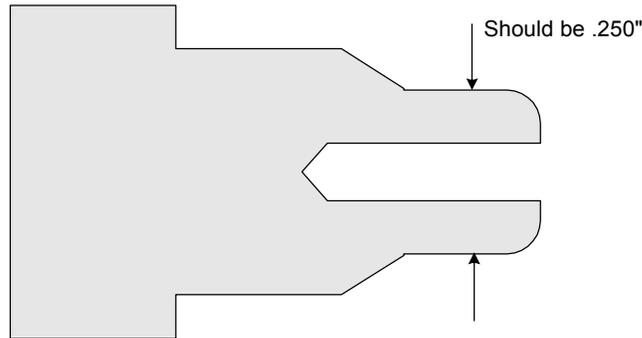
To Go X: +0.00000 Z: +0.00000	c:\programs\SAMPLE1 G95F.003Z-.75 X.26 G94F300Z.025	Position X: +0.00000 Z: +0.00000
Switches C.Cycle Repeat /.Block Skip O.Optional Stop S.Single Block	Press Cycle Start to Begin	Feed 0.000 IPM 100%
Functions F2-Tool Offsets F3-Edit F4-Verify F5-New File F6-Program Search F8-File Ops F9-Sec. Offsets F10-Spec. Functions J-Jog Mode M-Manual Data Input		Speed 0 RPM 100% Tool 0

Three blocks of the program are displayed:
Current block is highlighted with reverse text
Previous block is above it
Next block is below it

OmniTurn Startup sample part

Making corrections

Once the program has been tested and it is making the correct motions it is time to make sure that the part is the correct size. In our example we have a close diameter to maintain. It is the .250" turn.



If we now make a part and measure it there will probably be variations from what you program and the size of the finished part. These differences can come from a number of sources, minor errors in establishing tool offsets, tool deflections, material deflection. If the error is small it is possible to change the tool offset to try and correct for the error. The correction is made by pressing F2 from the automatic mode page. This will bring up the tool offset correction screen:

1	X: +0.86480 Z: -1.25340	17	X: +0.00000 Z: +0.00000
2	X: +1.65025 Z: -1.99200	18	X: +0.00000 Z: +0.00000
3	X: +2.91130 Z: -0.93885	19	X: +0.00000 Z: +0.00000
4	X: +0.00000 Z: +0.00000	20	X: +0.00000 Z: +0.00000
5	X: +0.00000 Z: +0.00000	21	X: +0.00000 Z: +0.00000
6	X: +0.00000 Z: +0.00000	22	X: +0.00000 Z: +0.00000
7	X: +0.00000 Z: +0.00000	23	X: +0.00000 Z: +0.00000
8	X: +0.00000 Z: +0.00000	24	X: +0.00000 Z: +0.00000
9	X: +0.00000 Z: +0.00000	25	X: +0.00000 Z: +0.00000
10	X: +0.00000 Z: +0.00000	26	X: +0.00000 Z: +0.00000
11	X: +0.00000 Z: +0.00000	27	X: +0.00000 Z: +0.00000
12	X: +0.00000 Z: +0.00000	28	X: +0.00000 Z: +0.00000
13	X: +0.00000 Z: +0.00000	29	X: +0.00000 Z: +0.00000
14	X: +0.00000 Z: +0.00000	30	X: +0.00000 Z: +0.00000
15	X: +0.00000 Z: +0.00000	31	X: +0.00000 Z: +0.00000
16	X: +0.00000 Z: +0.00000	32	X: +0.00000 Z: +0.00000

OFFSET NUMBER:
Press Esc to exit offset adjustment screen

OmniTurn Startup sample part

If we measured the diameter in question and found it to be .252” we would have to correct the diameter by making it .002” smaller. To do this we would have to remember what tool was used to cut it. In this case it was tool #1, the LH turning tool. So we would enter ”1” when asked what offset number. Then for the correction in X we would enter ”-.002” to make the diameter smaller. You will notice that the value of the tool offset location will be modified.

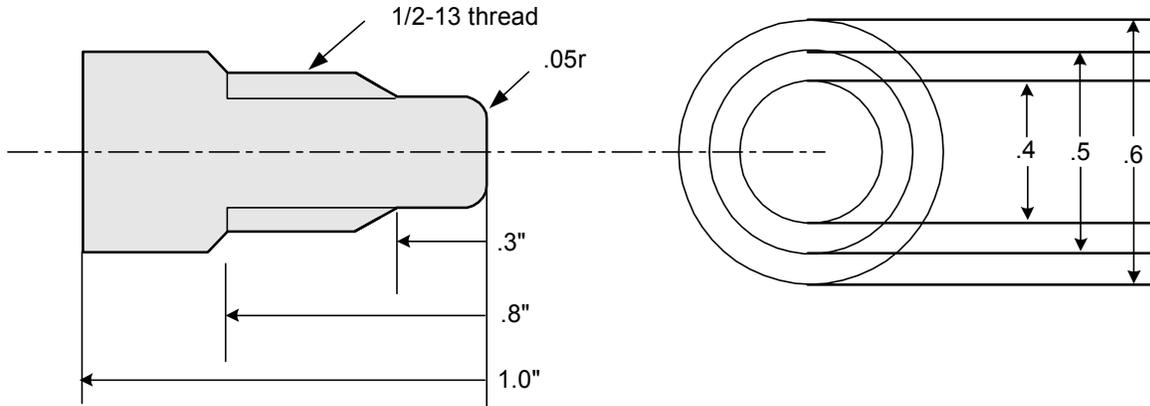
1	X: +0.86480 Z: -1.25340	17	X: +0.00000 Z: +0.00000
2	X: +1.65025 Z: -1.99200	18	X: +0.00000 Z: +0.00000
3	X: +2.91130 Z: -0.93885	19	X: +0.00000 Z: +0.00000
4	X: +0.00000 Z: +0.00000	20	X: +0.00000 Z: +0.00000
5	X: +0.00000 Z: +0.00000	21	X: +0.00000 Z: +0.00000
6	X: +0.00000 Z: +0.00000	22	X: +0.00000 Z: +0.00000
7	X: +0.00000 Z: +0.00000	23	X: +0.00000 Z: +0.00000
8	X: +0.00000 Z: +0.00000	24	X: +0.00000 Z: +0.00000
9	X: +0.00000 Z: +0.00000	25	X: +0.00000 Z: +0.00000
10	X: +0.00000 Z: +0.00000	26	X: +0.00000 Z: +0.00000
11	X: +0.00000 Z: +0.00000	27	X: +0.00000 Z: +0.00000
12	X: +0.00000 Z: +0.00000	28	X: +0.00000 Z: +0.00000
13	X: +0.00000 Z: +0.00000	29	X: +0.00000 Z: +0.00000
14	X: +0.00000 Z: +0.00000	30	X: +0.00000 Z: +0.00000
15	X: +0.00000 Z: +0.00000	31	X: +0.00000 Z: +0.00000
16	X: +0.00000 Z: +0.00000	32	X: +0.00000 Z: +0.00000

X DIAMETER ADJUSTMENT:
Press Esc to exit offset adjustment screen

After you correct the X adjustment then just press return to get past the Z input. Then press Esc to go back to the Automatic mode to run the next part.

Now you are ready to run parts.

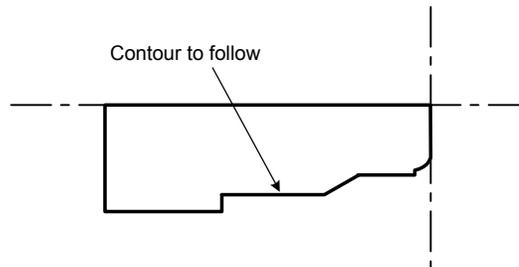
Worked Examples



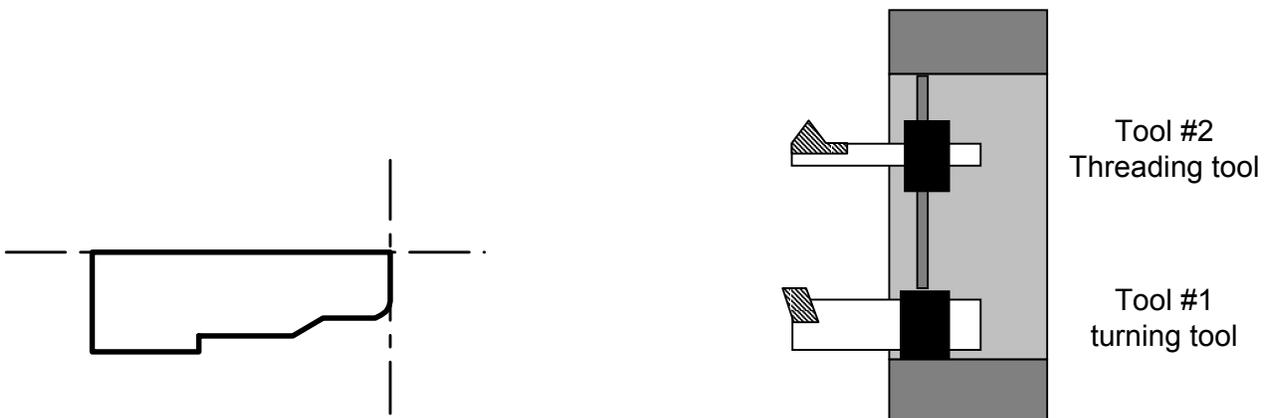
In the above example, the finished part will be made from a cutoff blank .61" diameter by 1.1" long. The first item to take care of is the layout of the job. This will entail what our sequence of operations will be. Then select the tools to accomplish this sequence. Once the tooling is set it is possible to write the program and then cut the part.

The machining operations that we will be performing are:

- Rough turn the OD twice, Tool #1
- Face the part to length, Tool #1
- Profile the contour to finished size, Tool #1
- Thread the part, Tool #2



Now, the tools can be selected for the operations listed above. See below the tool selections that will be used:



Worked Examples

Before the program is written we will setup the tools and establish the starting point of the tools from the T command. The tools will start at:

Tool #1 The turning tool will start at the center of the part in X and .2" away from the face in Z. This .2" in Z will give enough room in Z so that the tool will not crash into the face if the part is a little long.

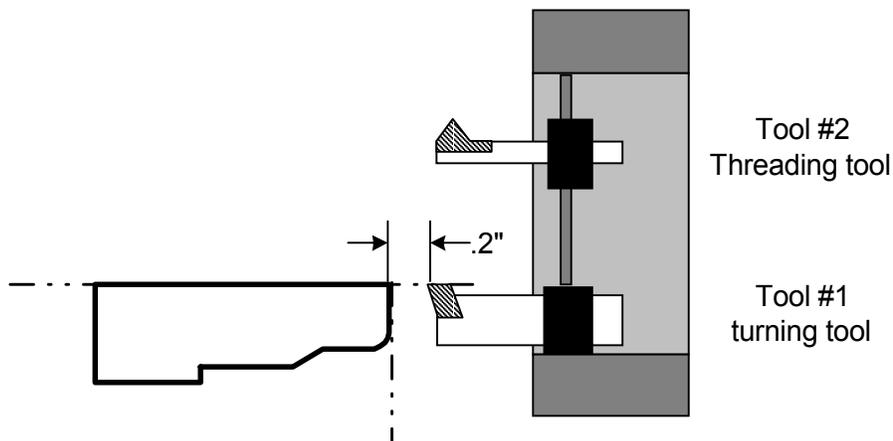
Tool #2 This tool will start at a radius in X equal to the major diameter of the thread less the amount for the first cut.

$$\text{Diameter } .50" \div 2 = .25" \text{ radius}$$

This radius less the first cut of .005" is $.25" - .005" = .245"$ starting location in X

In Z we will start at least .2" away from the start of the thread to allow the servos time to ramp up to speed.

We need these starting dimensions now because these locations are used in:



1. The setting of the tool offsets in the Manual mode.
2. The writing of the G92 statements after the Tool change in the program

Note: Care should be taken that the offset location will not cause collisions during tool changes.

Above is an illustration of where the T1 command will locate the tool. We have set the location so that the tool will come to the center ($X = 0$) and .2" from the face of the finished part, or .1" from the face of the rough blank. The setup can be done with either the finished or blank part and compensated for during the establishing of the Tool offsets.

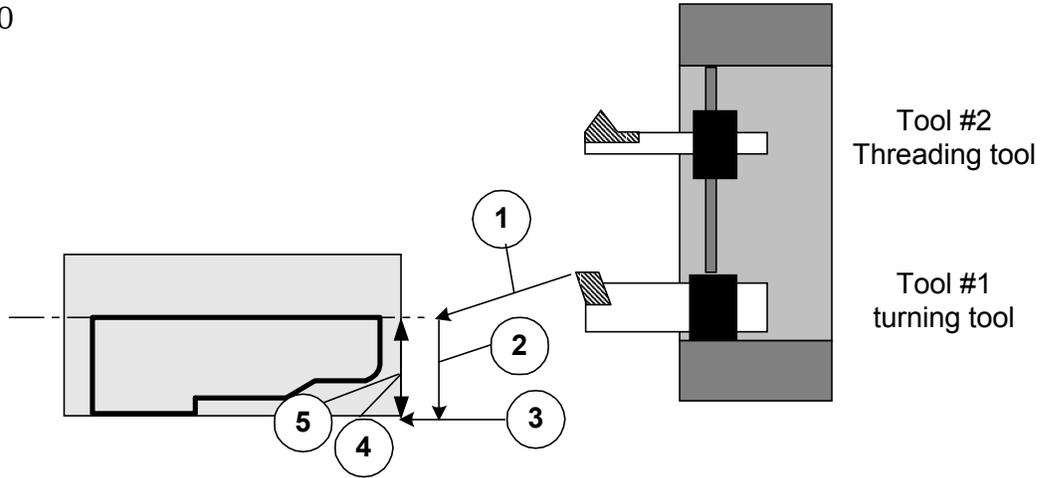
Before we start to program there are a few more comments to be made:

G90 or G91: we will be programming in G90 (absolute). This means that all dimensions given will be from the Reference zero for the part. The location of this Zero is established by the G92 statement after the Tool change.

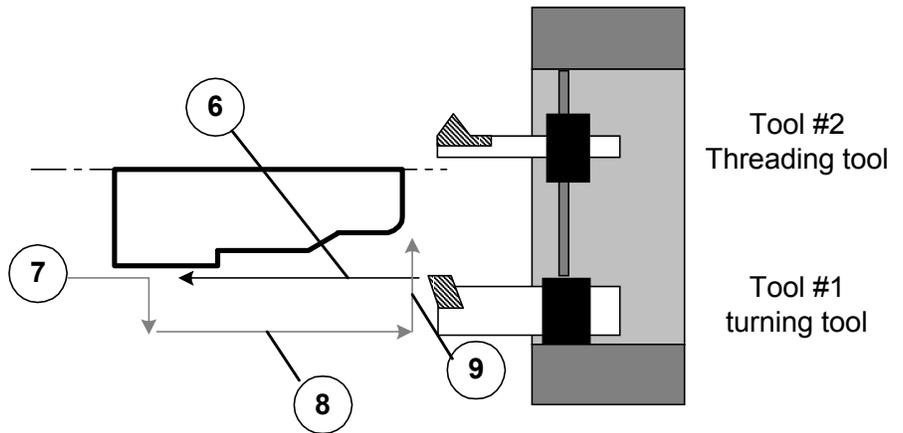
G94 or G95: This part will be programmed in G94, Inches per Minute.

Worked Examples

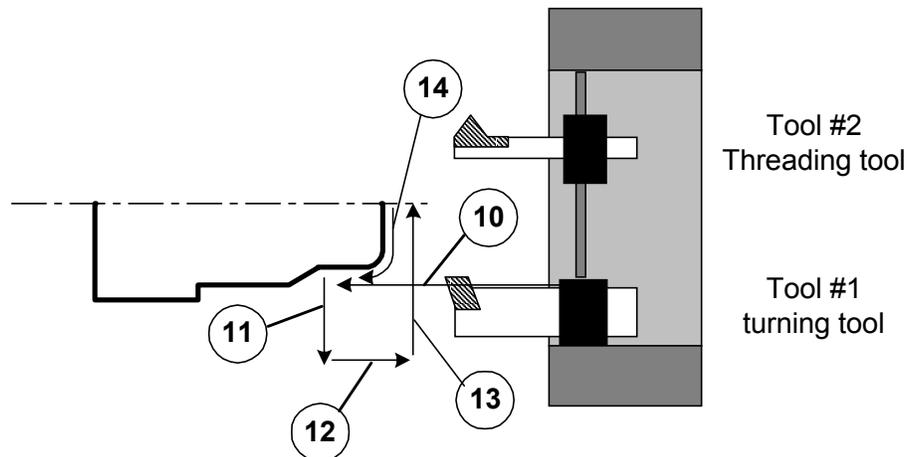
- (1) G90 G94 F200
- (2) T1
- (3) G92 X0 Z.2
- (4) M01
- (5) X.35
- (6) Z.01
- (7) X0F3
- (8) X.26F200



- (6) Z-.79F3
- (7) X.27F200
- (8) Z.1
- (9) X.21

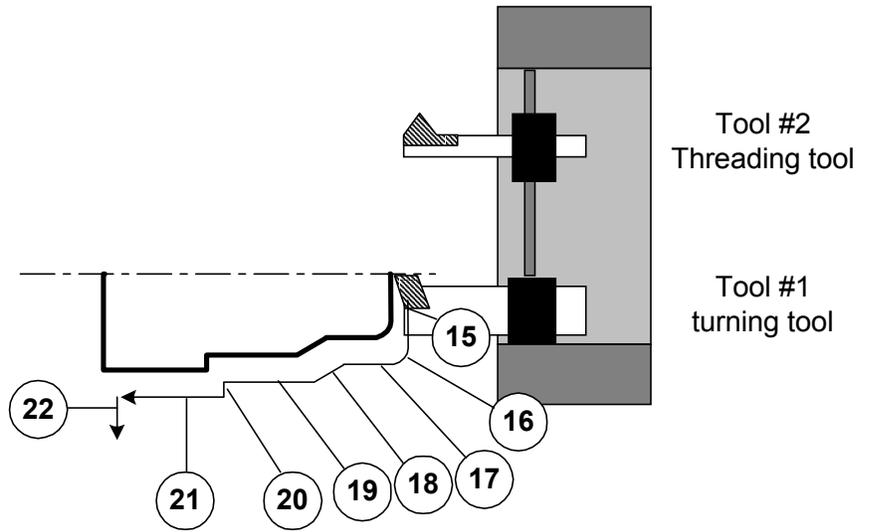


- (10) Z-.29F3
- (11) X.22
- (12) Z.1 F200
- (13) X0
- (14) Z0F3

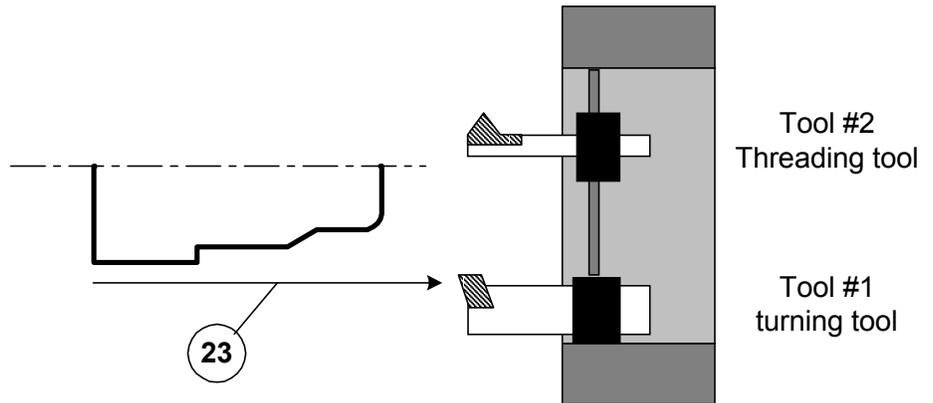


Worked Examples

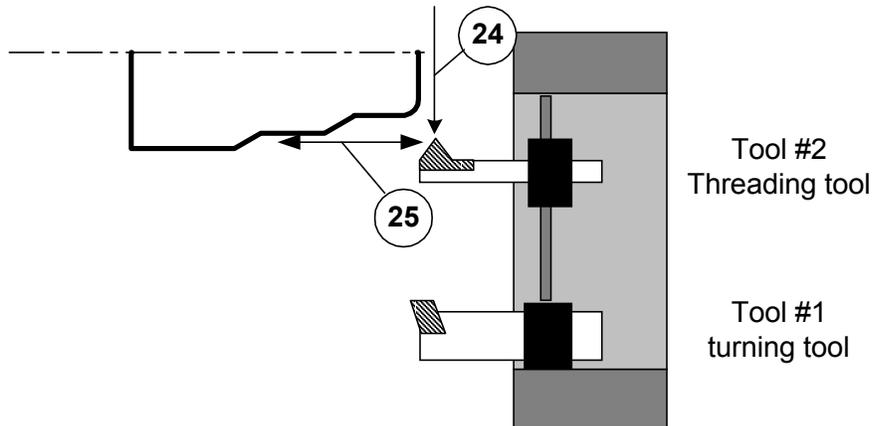
- (15) X.15
- (16) g02x.2z-.05i.15k-.05
- (17) Z-.3
- (18) X.25Z-.35
- (19) Z-.8
- (20) X.3Z-.85
- (21) Z-1
- (22) X.32



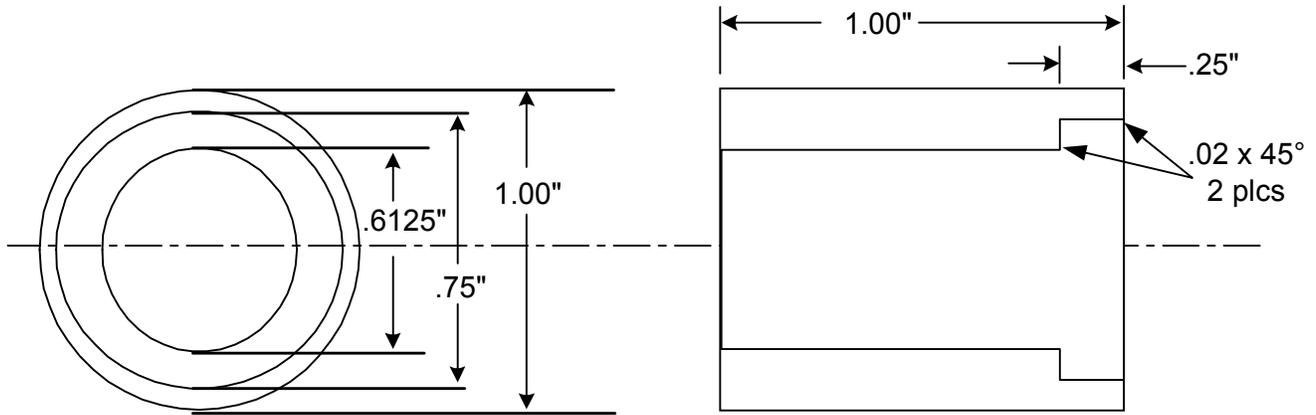
- (23) Z.5F200



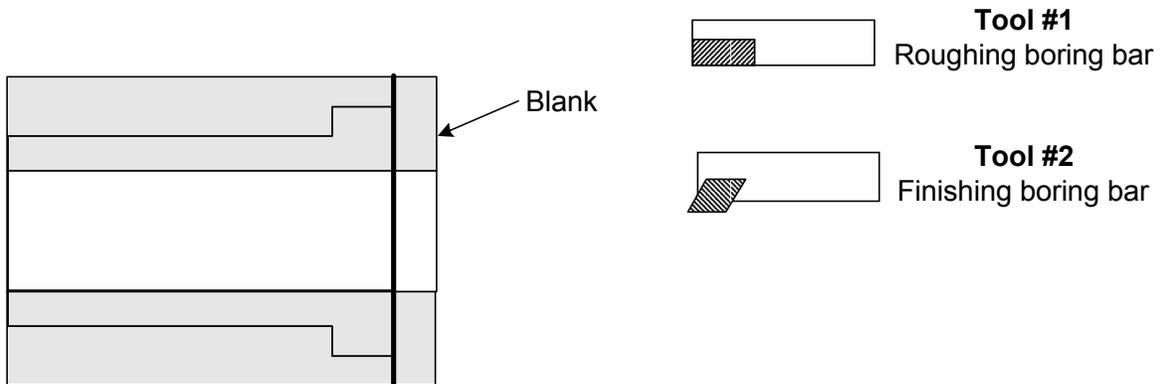
- (24) T2
- G92X.245 Z.2
- M01
- (25) g33x.22z-.8i.005k.0769p
- T0
- M30



Worked Examples



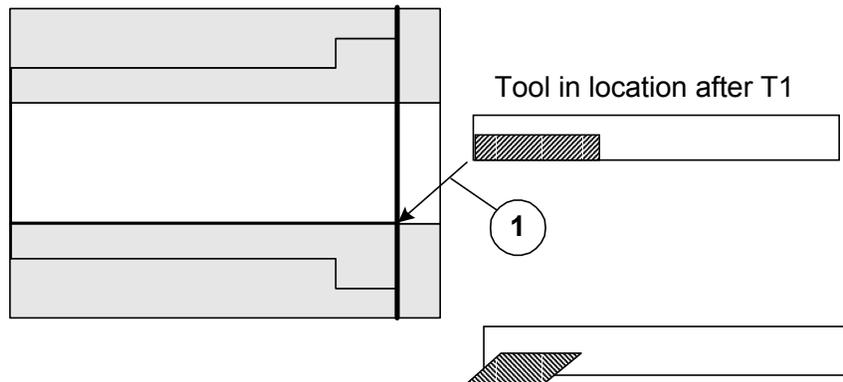
In the above example we will be using two boring tools to finish the face and ID.



The blank has been predrilled and is a little longer than the finished part.

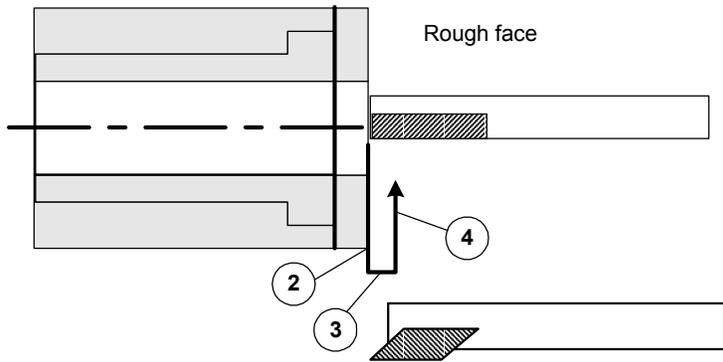
G90 G94 F200
T1
G92 X0 Z.2
M03S2000

(1) X.28 Z.01

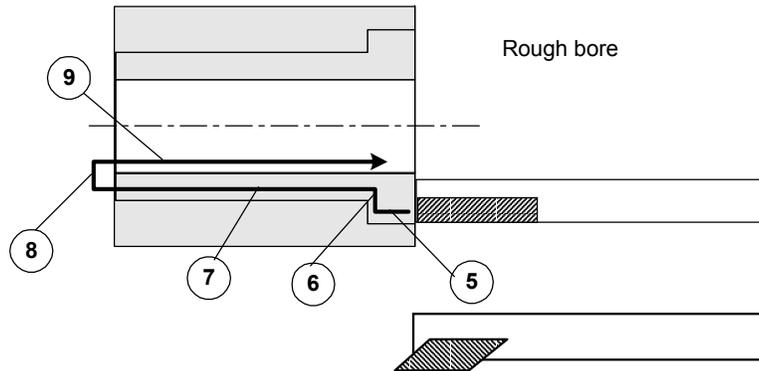


Worked Examples

- (2) X.55 G95 F003
- (3) G94 F200 Z.015
- (4) X.34



- (5) G95 F002 Z-.235
- (6) X.3
- (7) Z-1.05
- (8) X.29
- (9) G94 F200 Z.5
- T2
- G92 X.55 Z.1
- S3000
- Z0

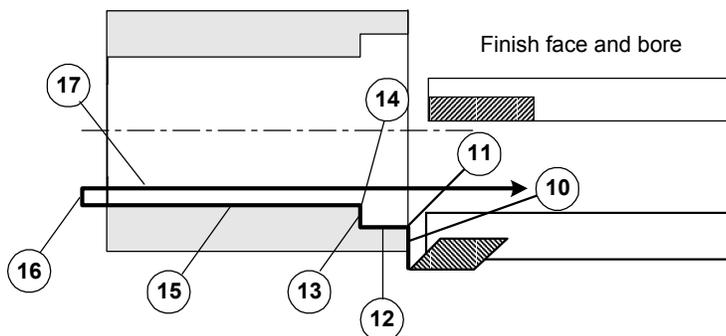


- (10) G95 F002 X.395
- (11) X.375 Z-.02 D1
- (12) Z-.25 D2
- (13) X.32625
- (14) X.30625 Z-.27 D3
- (15) Z-1.05 D4
- (16) X.29
- (17) Z.5 G94 F200
- M05
- T3
- M30

Generate chamfer with correction to effect first bore
Bore first diameter with correction for shoulder length

Generate chamfer with correction to effect second bore
Bore second diameter with correction for back of bore

Position the tools away from the part for loading, it also
Turns off the secondary offsets



Automatic Mode, Table of Contents

Description.....	Page
Introduction.....	5.2
Initial Program Verify.....	5.3
Functions and Switches	5.5
Cycle Repeat	5.6
Block Skip	5.7
Optional Stop.....	5.7
Single Block	5.7
Tool Offsets	5.8
Program Verification: Cycle Time	5.11
Program Verification: Tool Path	5.12
Running Different Program.....	5.14
Search To Tool in Program	5.14
Save Offsets with Program.....	5.17
Cycle Repeat Counter	5.17
Parts Counter	5.17
M-Functions on Keyboard.....	5.18
Editor	5.19

Automatic Mode

Running programs

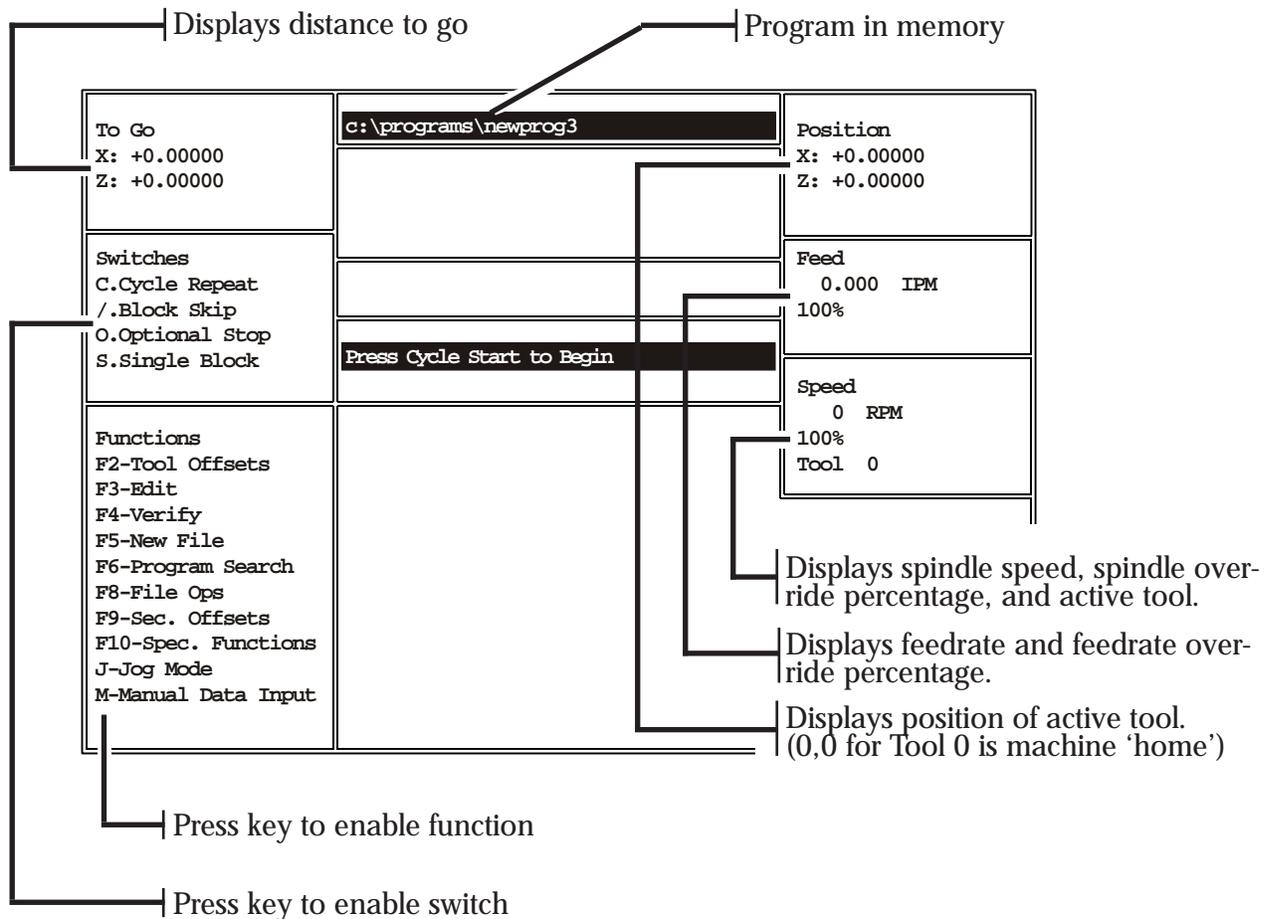
In the Automatic mode the control displays the program that it is currently running. When Auto Mode is first selected the File Handler screen appears. **You must select a program to run before Auto Mode screen will appear.** See Section One, Jog Mode for details (Page 1.17).

Syntax error-checking

When a program is first loaded into memory (or reloaded after editing) it is checked for errors such as missing or wrong codes, typing errors (O's for zeros, etc) and faulty arc geometry. If errors are found, the syntax error screen appears, showing the error and allowing you to edit the program. Details are on the next two pages.

Offsets

Be sure that the tool offsets are correct for the program to be run. If this program is the same as when the control was last shut down, the offsets should still be the same and the program will run without resetting the tools. For example, if you are running a program and shut the control down for the night. When you start up the control the next morning all you have to do is enter the program name in the file to be run once you enter the Automatic mode the next morning. See below. To save and recall tool offsets according to program, see Special Functions (F10), in this section.



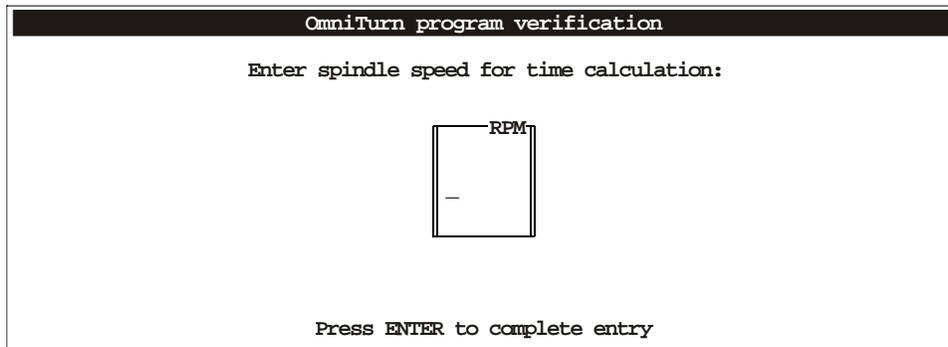
Automatic Mode, Initial Program Verification

Syntax error-checking

When a program is first loaded into memory (or reloaded after editing) it is checked for errors such as missing or wrong codes, typing errors (O's for zeros, etc) and faulty arc geometry. If errors are found, the syntax error screen appears, showing the error and allowing you to edit the program.

No spindle speed

If no spindle speed is found by the verification program, the screen below will appear.



OmniTurn program verification

Enter spindle speed for time calculation:

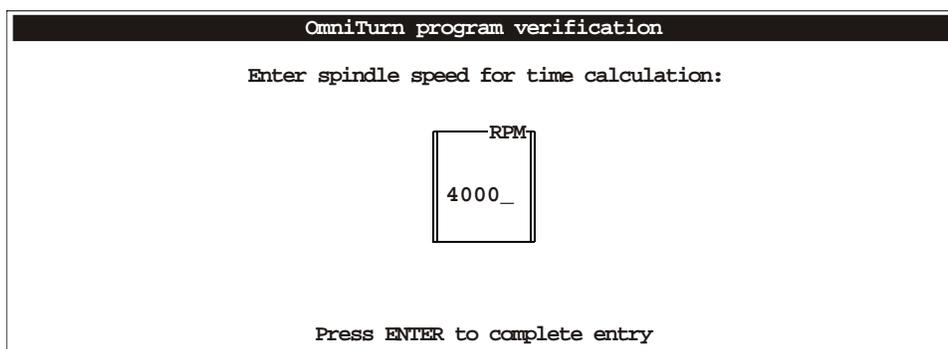
RPM

—

Press ENTER to complete entry

Enter a value for spindle speed in RPM, then press enter.
The program will estimate cycle time for that spindle speed.

The speed is NOT written to your program. You must edit your program to include spindle speed and direction.



OmniTurn program verification

Enter spindle speed for time calculation:

RPM

4000_

Press ENTER to complete entry

Automatic Mode, Initial Program Verification, con't

```
OmniTurn program verification

c:\programs\NEWPROG3
x5
g93f.005
g02x5z-5i5k-2.5

Use Arrow keys to move to error and correct. ESC to Exit

ERRORS
Line 8: Unrecognized G code G
Line 10: Faulty arc geometry
Line 11: Default error code
Line 12: Unrecognized letter
```

Making corrections:

If an error is found in your program it is possible to correct it now. Use the left or right arrow keys to move the cursor under the incorrect character. In the above example the g93 should be g95. Arrow over to the 3 and then type a 4. When you are done with the correction use the down arrow to go to the next comment. Some errors may not be legitimate, but result from a position error on a previous line. Note that to *replace* a bad character, the default, underscore cursor (as shown) works fine, but to *insert* characters or to *add* characters to the end of the line you must press the Ins key (insert) to get the box cursor (not shown).

Refer to Section One, page 1.7 (Front Panel: Keyboard) for location of Ins key.

Saving the corrections:

Press ESC key when you're finished making corrections. The screen below will appear:

```
OmniTurn program verification

File has been edited. Save it? Yes
                               No

Use Cursor keys; Press Enter to make a selection. ESC to Exit
```

Press Enter to accept the default (Yes). If you choose not to save the program, use the down arrow to select No, then press Enter. Pressing ESC key will save and exit. If there are no more errors

Automatic Mode, Functions & Switches

The "F" keys have the following functions:

NOTE: These keys are not effective while program is running

F 2	Tool Offsets	Adjust tool offsets.
F 3	Edit	Edit existing programs or create new programs.
F 4	Verify	Verify the program in memory and display the tool paths.
F 5	New File	This will remove the program from active memory and open the file handler screen.
F 6	Prog Search	This enables the program to be started some place other than the start
F 8	File Ops	File Operations: File handler screen Erase programs Create new program Up- or Download programs from another computer over RS232
F 9	SecOffsets	Adjust values of secondary tool offsets and TNR compensation values
F10	Spec Functions	Special functions: Load tool offset values for program (previously saved) Save tool offset values with program in memory Set/reset parts counter Set value of countdown counter
J	Jog Mode	Switch to Jog page
M	MDI	Switch to Manual Data Input page

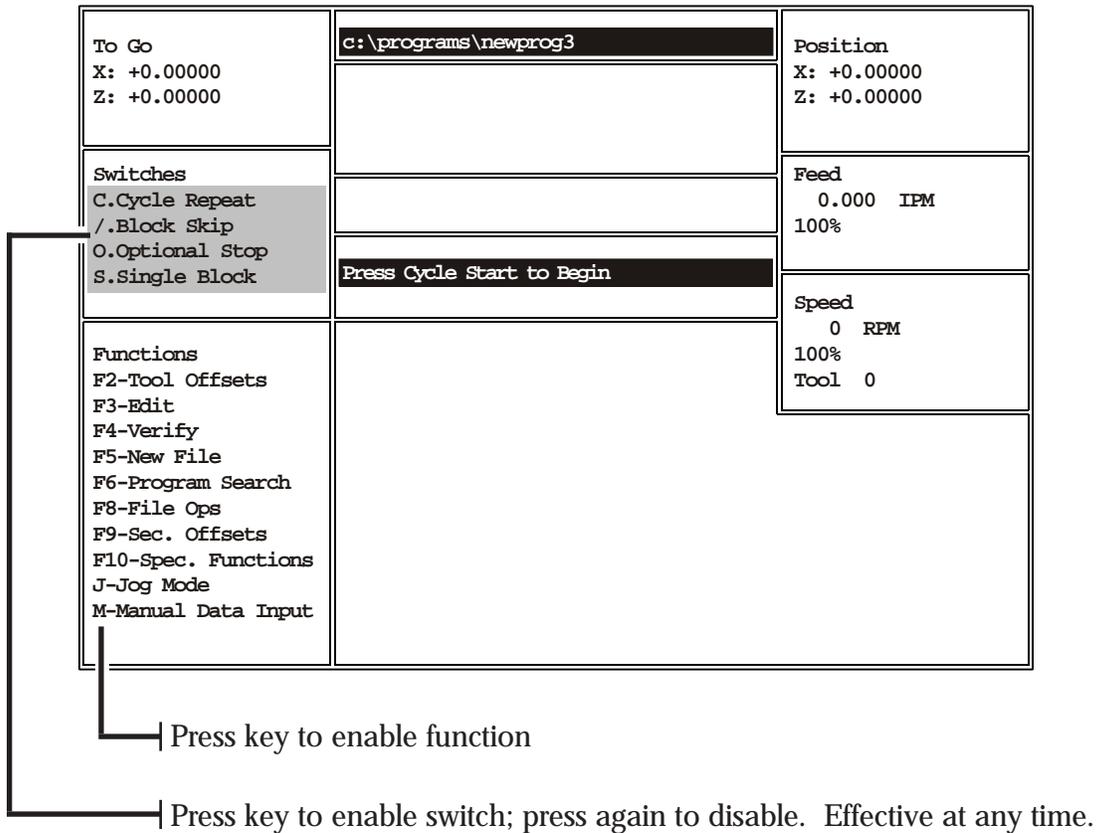
HOT keys on the keyboard while in the automatic mode

NOTE: These keys are effective whether program is running or not

C	Cycle Repeat	The program will run continuously. To cancel, Preass "C" again, or press "A".
/	Block delete	With this active the control will skip over program lines starting with "/". In your program, put the "/" symbol as the first character in any block you might want to skip. (Press "/" again to cancel.)
O	Optional stop	This makes the M0l act as a stop program command (M00). In your program, put M01 wherever you might want to stop. (Press "O" again to cancel.)
S	Single block	The program will run one line at a time with each press of the cycle start button.
Pg Up		Coolant on/off (M08/09). Press to turn coolant on, press again to turn coolant off.
Pg Dn		Parts catcher Out/In (M25/26). Press to turn M25 on, press again to turn M26 off. Note that M25 is an auxiliary output on all OmniTurn machines; if a parts catcher is not installed, this M-function is available for any auxiliary function you need. See page 5.19 for keyboard.

Automatic Mode, Switches

Functions and switches available from Automatic Mode



To Go X: +0.00000 Z: +0.00000	c:\programs\newprog3	Position X: +0.00000 Z: +0.00000
Switches C.Cycle Repeat /.Block Skip O.Optional Stop S.Single Block		Feed 0.000 IPM 100%
Functions F2-Tool Offsets F3-Edit F4-Verify F5-New File F6-Program Search F8-File Ops F9-Sec. Offsets F10-Spec. Functions J-Jog Mode M-Manual Data Input	Press Cycle Start to Begin	Speed 0 RPM 100% Tool 0

Cycle Repeat Mode

To run the program over and over without pressing Cycle Start, select Cycle Repeat by pressing the C key. This is the preferred mode for bar jobs and loaders, which automatically supply material for machining. To cancel the Cycle Repeat mode, press C again, or press A.

Automatic Mode, Switches, con't

To Go X: +0.00000 Z: +0.00000	c:\programs\newprog3	Position X: +0.00000 Z: +0.00000
Switches C.Cycle Repeat /.Block Skip O.Optional Stop S.Single Block	Press Cycle Start to Begin	Feed 0.000 IPM 100%
Functions F2-Tool Offsets F3-Edit F4-Verify F5-New File		Speed 0 RPM 100% Tool 0

Block Skip Mode

If there are lines in your program that need to be skipped over sometimes, add the forward slash character “/” which is under the question mark to the beginning of each line to skip. The entire program will run normally unless you press the “/” key; then only lines not marked with “/” will run. The marked lines will be skipped over. Press “/” again to return to normal automatic mode.

To Go X: +0.00000 Z: +0.00000	c:\programs\newprog3	Position X: +0.00000 Z: +0.00000
Switches C.Cycle Repeat /.Block Skip O.Optional Stop S.Single Block	Press Cycle Start to Begin	Feed 0.000 IPM 100%
Functions F2-Tool Offsets F3-Edit F4-Verify F5-New File		Speed 0 RPM 100% Tool 0

Optional Stop Mode

If there are specific lines in your program where you need to stop sometimes, put the M01 (Optional Stop) code on a line by itself. The program will run without stopping at these places unless you press O. The Optional Stop switch will highlight and the program will stop at each line with M01. Press Cycle Start to continue. Press O again to return to normal automatic mode. See Section Two, page 2.61 for more about Optional Stop.

To Go X: +0.00000 Z: +0.00000	c:\programs\newprog3	Position X: +0.00000 Z: +0.00000
Switches C.Cycle Repeat /.Block Skip O.Optional Stop S.Single Block	Press Cycle Start to Begin	Feed 0.000 IPM 100%
Functions F2-Tool Offsets F3-Edit F4-Verify F5-New File		Speed 0 RPM 100% Tool 0

Single Block Mode

When running a program for the first time it is very useful to step through the program one line at a time. The tool positions can be checked and offsets can be verified. Press S to initiate Single Block mode. Cycle Start must be pressed to execute each line. Press S again to return to normal automatic mode.

Automatic Mode, Function Keys

On the top of the keyboard is a group of "F" keys. These are used differently throughout the control software. Notations are made on the screen to help the operator remember how the keys are being used with the different sections of software. Care should be taken to remember that these keys change depending on the "Mode" the control is in. Following will be the description of how the Function keys are used in the Automatic mode.

Automatic Mode, F2: Tool Offsets

F2 Tool offset screen, used to modify tool offsets

This function key brings up the screen to adjust the tool offsets. Tool offsets are used to correct the starting location of the tools, and they will effect the finished part dimensions. These values are created when the tools are setup in the jog mode. When the F2 key is pressed the screen will then ask what tool number you want to adjust. The distances shown are the amount needed to travel from the Home position to the offset location. See below:

In the example above you see the offset screen with three tools being used. When it is necessary to correct a tool offset, enter the amount of change that is required. As an example, we will assume that tool 2 in the above example is a turning tool and is cutting apart .001" too large. So enter the offset change of -.001" for the X Diameter offset. When this value is entered you will notice that the total value of X has changed. This addition does not have to be done by the operator.

1	X: +0.86480 Z: -1.25340	17	X: +0.00000 Z: +0.00000
2	X: +1.65025 Z: -1.99200	18	X: +0.00000 Z: +0.00000
3	X: +2.91130 Z: -0.93885	19	X: +0.00000 Z: +0.00000
4	X: +0.00000 Z: +0.00000	20	X: +0.00000 Z: +0.00000
5	X: +0.00000 Z: +0.00000	21	X: +0.00000 Z: +0.00000
6	X: +0.00000 Z: +0.00000	22	X: +0.00000 Z: +0.00000
7	X: +0.00000 Z: +0.00000	23	X: +0.00000 Z: +0.00000
8	X: +0.00000 Z: +0.00000	24	X: +0.00000 Z: +0.00000
9	X: +0.00000 Z: +0.00000	25	X: +0.00000 Z: +0.00000
10	X: +0.00000 Z: +0.00000	26	X: +0.00000 Z: +0.00000
11	X: +0.00000 Z: +0.00000	27	X: +0.00000 Z: +0.00000
12	X: +0.00000 Z: +0.00000	28	X: +0.00000 Z: +0.00000
13	X: +0.00000 Z: +0.00000	29	X: +0.00000 Z: +0.00000
14	X: +0.00000 Z: +0.00000	30	X: +0.00000 Z: +0.00000
15	X: +0.00000 Z: +0.00000	31	X: +0.00000 Z: +0.00000
16	X: +0.00000 Z: +0.00000	32	X: +0.00000 Z: +0.00000

OFFSET NUMBER:
Press Esc to exit offset adjustment screen

Automatic Mode, F2: Tool Offsets, con't

After selecting a number and pressing Return the screen will ask

1	X: +0.86480 Z: -1.25340	17	X: +0.00000 Z: +0.00000
2	X: +1.65025 Z: -1.99200	18	X: +0.00000 Z: +0.00000
3	X: +2.91130 Z: -0.93885	19	X: +0.00000 Z: +0.00000
4	X: +0.00000 Z: +0.00000	20	X: +0.00000 Z: +0.00000
5	X: +0.00000 Z: +0.00000	21	X: +0.00000 Z: +0.00000
6	X: +0.00000 Z: +0.00000	22	X: +0.00000 Z: +0.00000
7	X: +0.00000 Z: +0.00000	23	X: +0.00000 Z: +0.00000
8	X: +0.00000 Z: +0.00000	24	X: +0.00000 Z: +0.00000
9	X: +0.00000 Z: +0.00000	25	X: +0.00000 Z: +0.00000
10	X: +0.00000 Z: +0.00000	26	X: +0.00000 Z: +0.00000
11	X: +0.00000 Z: +0.00000	27	X: +0.00000 Z: +0.00000
12	X: +0.00000 Z: +0.00000	28	X: +0.00000 Z: +0.00000
13	X: +0.00000 Z: +0.00000	29	X: +0.00000 Z: +0.00000
14	X: +0.00000 Z: +0.00000	30	X: +0.00000 Z: +0.00000
15	X: +0.00000 Z: +0.00000	31	X: +0.00000 Z: +0.00000
16	X: +0.00000 Z: +0.00000	32	X: +0.00000 Z: +0.00000

X DIAMETER ADJUSTMENT:
Press Esc to exit offset adjustment screen

Now enter the value of change (ie: -.001) and press Return. The value of X will update and then ask you about Z.

1	X: +0.86480 Z: -1.25340	17	X: +0.00000 Z: +0.00000
2	X: +1.64975 Z: -1.99200	18	X: +0.00000 Z: +0.00000
3	X: +2.91130 Z: -0.93885	19	X: +0.00000 Z: +0.00000
4	X: +0.00000 Z: +0.00000	20	X: +0.00000 Z: +0.00000
5	X: +0.00000 Z: +0.00000	21	X: +0.00000 Z: +0.00000
6	X: +0.00000 Z: +0.00000	22	X: +0.00000 Z: +0.00000
7	X: +0.00000 Z: +0.00000	23	X: +0.00000 Z: +0.00000
8	X: +0.00000 Z: +0.00000	24	X: +0.00000 Z: +0.00000
9	X: +0.00000 Z: +0.00000	25	X: +0.00000 Z: +0.00000
10	X: +0.00000 Z: +0.00000	26	X: +0.00000 Z: +0.00000
11	X: +0.00000 Z: +0.00000	27	X: +0.00000 Z: +0.00000
12	X: +0.00000 Z: +0.00000	28	X: +0.00000 Z: +0.00000
13	X: +0.00000 Z: +0.00000	29	X: +0.00000 Z: +0.00000
14	X: +0.00000 Z: +0.00000	30	X: +0.00000 Z: +0.00000
15	X: +0.00000 Z: +0.00000	31	X: +0.00000 Z: +0.00000
16	X: +0.00000 Z: +0.00000	32	X: +0.00000 Z: +0.00000

Z ADJUSTMENT:
Press Esc to exit offset adjustment screen

Enter the amount of change in Z and then press Return.

To correct another tool, enter the tool number now and press Return. **To exit the tool offset** correction screen press ESC and press return. This will tell the control that you are done and bring you back to the Automatic mode.

Automatic Mode, F2: Tool Offsets, con't

Notes on Tool Offsets:

1. The control will allow you to clear all the offsets by pressing C (for clear). Please only do this when you have had experience with the control and understand what you are doing. Clearing offsets can cause you to crash tools if it is done incorrectly!
 2. The smallest offset changes are:
0.00005" in Z
0.0001" in X, (this is equal to .00005" on the radius)
 3. Tool Offset changes greater than .05" will invoke "*Do you really want to move [distance] (y/n)?*" prompt to alert the you of possible mistake. This is a safety feature to ensure that you do not put in a large correction in error, *ie* 0.1" instead of .01"
 4. If you have no change to a offset value just press Return without inputting a value. The control assumes that you want zero change.
 5. Tool offsets can be changed at any program stop: m00, m01 or Motion Stop.
-

Automatic Mode, F3, Program Editing

F3 Program Editing

This function starts the text editor, used to change existing programs, or to create new ones.

The editor is described starting on page 5.19.

Automatic Mode, F4: Program Verification

Verification (F4)

This function will check the current program. Note that the syntax is checked when a new program is loaded into memory, and when you exit the editor. The verification program does the following:

1. Shows estimated cycle time
2. Lists all the tools used
3. Show graphically the tool path created by the G code program

Note that this example uses the 'demo' program for illustration.

Running Verification

Press F4 and the verification software will be called up.

1. Estimated cycle time

The first operation performed by this section is the calculation of cycle time. The cycle time is calculated and a screen similar to the illustration below will appear:

```
OmniTurn program verification

Cycle Time
-----
Estimated part cycle time is 30 seconds

Press any key to continue_
```

2. Tools listed

Pressing any key after viewing the cycle time brings up a screen showing the tools used similar to the one below. The tool definitions included in the program as comments are shown as well. This screen is primarily for choosing which tool to use for the tool path generation. Use the up/dn arrow keys to highlight the desired tool, then press enter to display the tool path. The default is ALL tools; pressing enter will display tool paths for all the tools.

```
OmniTurn program verification

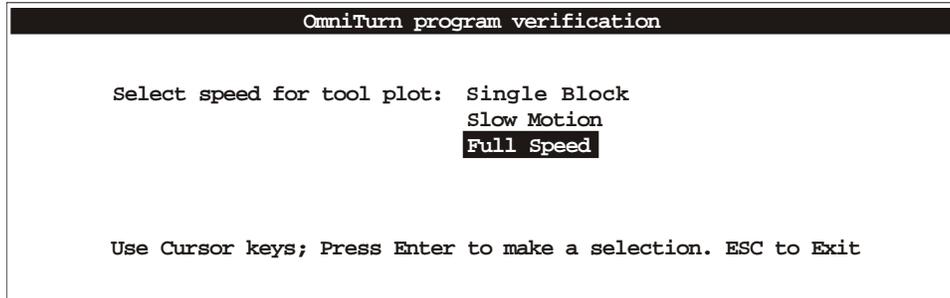
Select tool for verification ALL
                          T1 TURNING
                          T2 THREADING TOOL

Use Cursor keys; Press Enter to make a selection. ESC to Exit
```

Automatic Mode, F4: Program Verification, con't

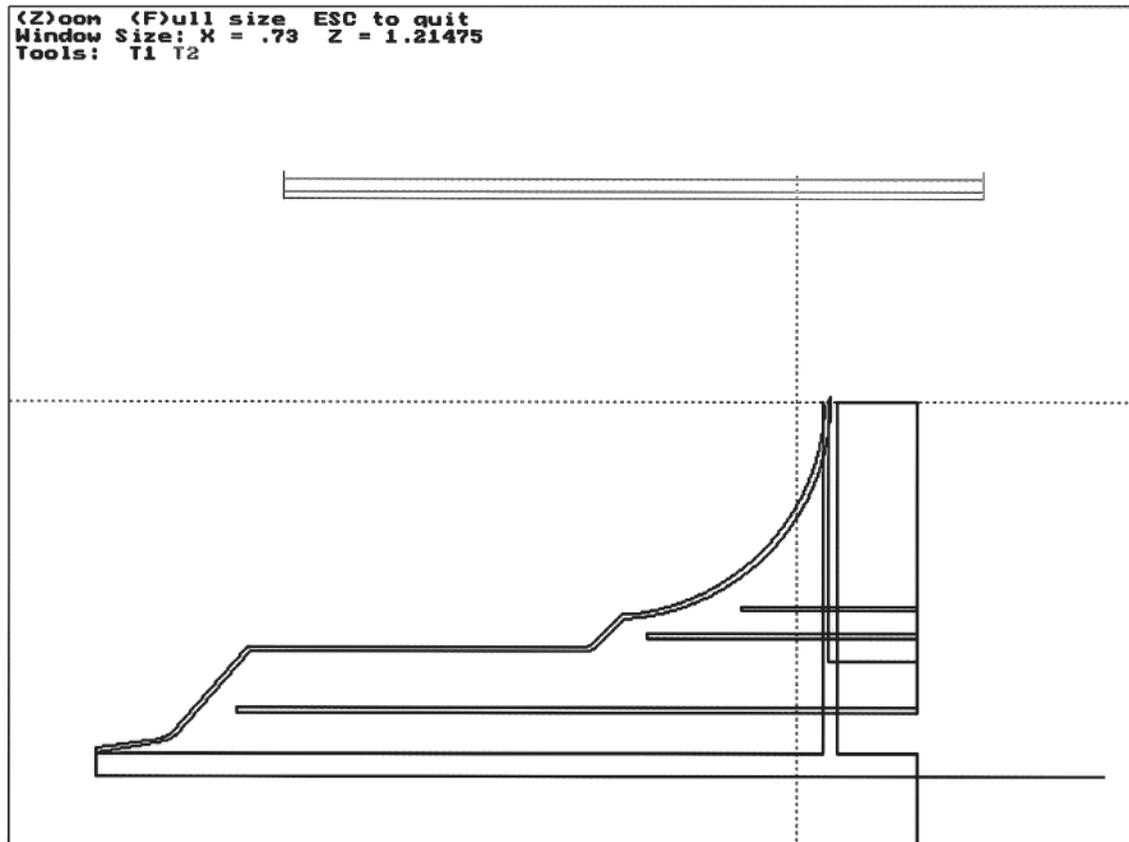
3. Tool path graphics

Select the speed of the graphics display. If you choose single step you can advance through the program using the Enter to go to the next move. The default is Full Speed, which is instantaneous on screen.



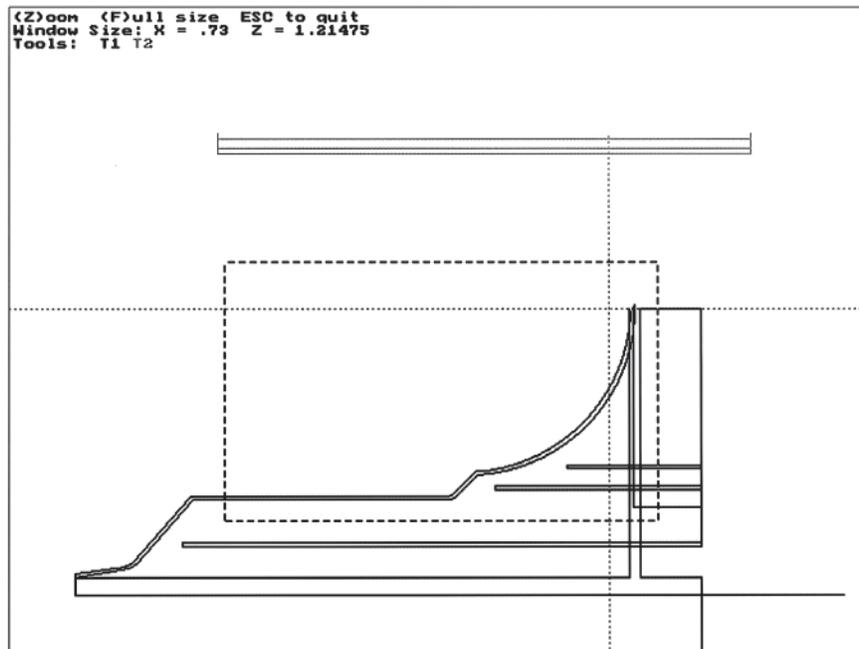
Press enter to display the tool paths.

Each tool is shown in a different color. The part contour for each tool is shown as the tool would cut. In this case the turning tool (T1) is a left-hand tool, which cuts on the plus side of the part, and the threading tool (T2) is a right-hand tool, which cuts on the minus side of the part.

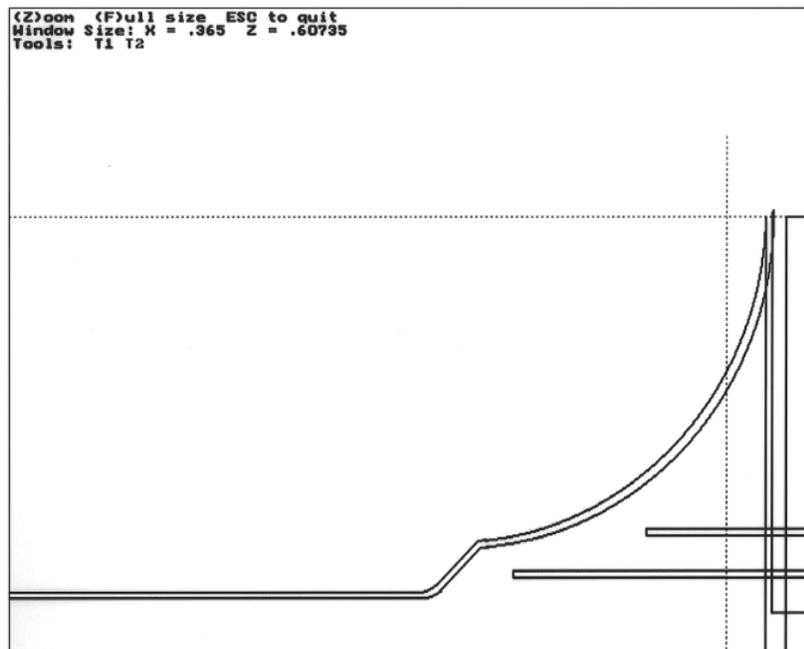


Automatic Mode, F4: Program Verification, con't

Zooming You can look closer at a section of the graphics by zooming in. Press the Z key. A window locator will appear as shown below. Use the arrow keys to move the window over the area that you want to examine.



When the window covers the desired area, press Z again (or press Enter) to fill the screen with that section of the tool path, as shown below. If you want to get closer, press Z again to obtain window. Press Esc to return to Automatic screen



Automatic Mode, F5: New File, F6: Program Search

F5 New Program; used to change the running program to a new one

This function is used to change the program that is in the current file and allow the operator to enter in a new program name to be run. When F5 is pressed the file handler screen will appear. You will see the list of programs available. Use the arrow keys to highlight the desired program. See page 1.17 - 1.20, Jog Mode, File Handler for details.

F6 Program Search; used to start the program at a certain tool instead of the beginning.

The program "DEMO" is used in the following example.

This function will allow you to start the program at a certain tool instead of the beginning.

NOTE: If there are less than two tools in your program, pressing F6 will generate an error as follows:

```
Program contains no Search Points
Press any key to return to Auto mode
```

If there is more than one tool in your program they will be listed, with highlight on first tool. Use arrow keys to select desired tool, then press Enter.

```
T 1
T 2
T 0

Use arrow keys to highlight Search Point
Enter to select, or ESC to cancel search
```

When Enter is pressed, another screen will appear defining the tool and the line number, and any M-functions that are active at that point:

```
Searched to tool 2 ,line 46
The following conditions will be established:
M03S2500
M08
Press cycle start to proceed or Esc to cancel
```

Pressing Cycle Start will return you to Auto screen, in Single Block mode, with highlight on selected tool:

To Go X: +0.00000 Z: +0.00000	C:\programs\DEMO	Position X: +7.86285 ∅ Z: -0.62225
	G94F50 T2 X-.5Z.2	
Switches C.Cycle Repeat .Block Skip O.Optional Stop S.Single Block	THREADING TOOL	Feed 000.000 IPM 100%
	Press Cycle Start to continue or Esc to terminate program	Speed 0 RPM 100% Tool 2
Functions F2-Tool Offsets F3-Edit F4-Verify F5-New File		

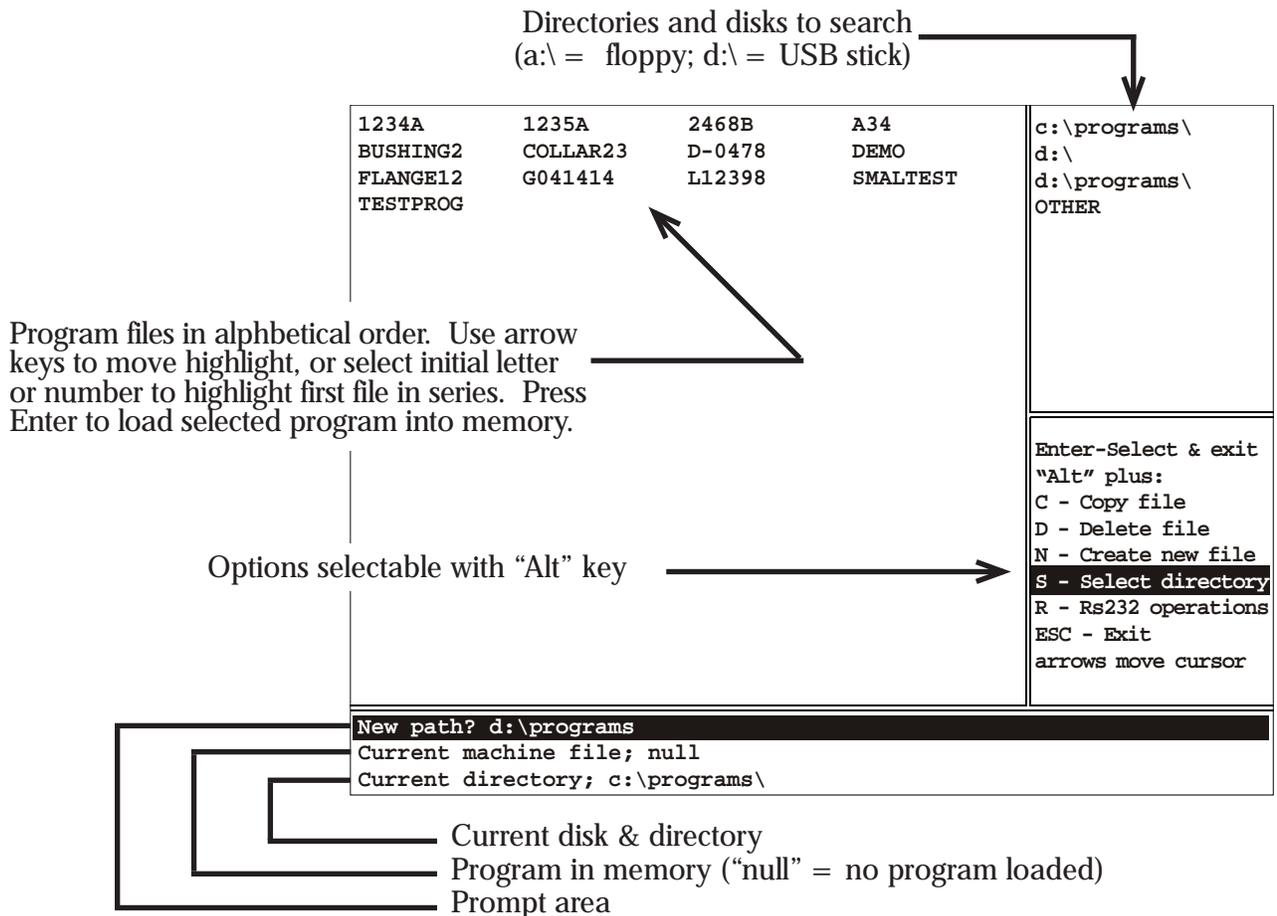
The Program Search function reads every block of the program up to the tool selected, so the machine will start running as if it has just executed all the previous blocks. When Cycle Start is pressed, the current feed rate will be displayed and the tool will move to the part. Note that no collision avoidance calculation is done, so be sure that the tool can move safely to the part before pressing Cycle Start.

Automatic Mode, F8: File Operations

F8 File Ops; File Operations. This Function allows you to select, copy, delete and create new programs, and to send or receive programs via RS232 interface.

See page 1.17 - 1.20, Jog Mode, File Handler for details.

The screen is described below:



Options Selectable with "Alt" key

Alt-C = Copy file: You will be prompted for a location to copy to. This option is most useful for copying programs written at your desktop. You must first use Alt-S to select directory. See next page for details.

Alt-D = Delete file: You will be prompted "Delete filename Y/N" Press N if you change your mind.

Alt-N = Create New file: You will be prompted "Name of file to create?" Type the name then press Enter. The OmniTurn editor will start. When you leave the editor, you will have opportunity to load this program into memory (make it the current machine file). The editor is described in Section Five.

Alt-S = Select directory: Use the up/dn arrow keys to highlight desired directory. Press Enter to display program files in that directory. Pressing the "D" key will remove the highlighted directory from the list. Highlight "OTHER" to choose a directory not on the list.

Alt-R = RS232 operations: This option allows you to send/recieve programs via the serial port.

Automatic Mode, F9: Secondary Offsets

F9 Secondary tool offset screen; used to modify secondary tool offsets

This function will call up the secondary offset table. There are 32 offsets available and 32 tool nose radius compensation offsets. Please refer to the section on secondary offsets for their use. Notice that this differs from the offset table screen in that almost all of the offset values are set to 0.00000. Secondary offsets are corrected like offsets.

1	X: +0.00000 Z: +0.00000 R: 0.00000	17	X: +0.00000 Z: +0.00000 R: 0.00000
2	X: +0.00000 Z: +0.00000 R: 0.00000	18	X: +0.00000 Z: +0.00000 R: 0.00000
3	X: +0.00000 Z: +0.00000 R: 0.00000	19	X: +0.00000 Z: +0.00000 R: 0.00000
4	X: +0.00000 Z: +0.00000 R: 0.00000	20	X: +0.00000 Z: +0.00000 R: 0.00000
5	X: +0.00000 Z: +0.00000 R: 0.00000	21	X: +0.00000 Z: +0.00000 R: 0.00000
6	X: +0.00000 Z: +0.00000 R: 0.00000	22	X: +0.00000 Z: +0.00000 R: 0.00000
7	X: +0.00000 Z: +0.00000 R: 0.00000	23	X: +0.00000 Z: +0.00000 R: 0.00000
8	X: +0.00000 Z: +0.00000 R: 0.00000	24	X: +0.00000 Z: +0.00000 R: 0.00000
9	X: +0.00000 Z: +0.00000 R: 0.00000	25	X: +0.00000 Z: +0.00000 R: 0.00000
10	X: +0.00000 Z: +0.00000 R: 0.00000	26	X: +0.00000 Z: +0.00000 R: 0.00000
11	X: +0.00000 Z: +0.00000 R: 0.00000	27	X: +0.00000 Z: +0.00000 R: 0.00000
12	X: +0.00000 Z: +0.00000 R: 0.00000	28	X: +0.00000 Z: +0.00000 R: 0.00000
13	X: +0.00000 Z: +0.00000 R: 0.00000	29	X: +0.00000 Z: +0.00000 R: 0.00000
14	X: +0.00000 Z: +0.00000 R: 0.00000	30	X: +0.00000 Z: +0.00000 R: 0.00000
15	X: +0.00000 Z: +0.00000 R: 0.00000	31	X: +0.00000 Z: +0.00000 R: 0.00000
16	X: +0.00000 Z: +0.00000 R: 0.00000	32	X: +0.00000 Z: +0.00000 R: 0.00000

Secondary offset number:
Press C to clear all offsets:
Press Esc to exit offset adjustment screen

First: Select a secondary offset number

Next: Use the return key to enter past the X and Z inputs.

Then: Enter the value of tool nose compensation, IE .007 and then press ESC

Changing a TNR compensation value:

When you put a value in the R offset table it writes over the old value. So if you have a number already in the register that you want to use and it is not the value needed, all you have to do is enter the correct value. As an example if you have a value of .032 in the offset and you want to change it to .008, just enter the new value. You do not have to clear the register first. If you want to correct a value slightly, you must enter in the final value needed. ie: if you have .007 and want to increase it by .001 you must enter .008. **Do not enter -.001**

Clearing secondary offsets to Zero.

It is possible to clear all of the secondary offsets by pressing C when asked for a secondary offset number. This will set the entire table to

Individual offsets can be set to zero by pressing C when asked to enter a correction amount.

Automatic Mode, F10: Special Functions

F10 Special Functions, used to call up a list of special functions.

To Go X: +0.00000 Z: +0.00000	c:\programs\newprog3	Position X: +0.00000 Z: +0.00000
Switches C.Cycle Repeat /.Block Skip O.Optional Stop S.Single Block	"L" to load offsets for this program "S" to save offsets for this program "C" to set cycle repeat counter "P" to enable/reset parts counter	Feed 000.000 IPM 100%
Functions F2-Tool Offsets F3-Edit F4-Verify F5-New File F6-Program Search F8-File Ops F9-Sec. Offsets F10-Spec. Functions J-Jog Mode M-Manual Data Input		Speed 0 RPM 100% Tool 0

- **Save and recall tool offset tables.**

This function enables you to load or save the tool offsets with the program. Press L at this screen to load the offsets for the current program *from* the disk; press S to save the offsets for the current program *to* the disk. Use the OmniTurn 805 eight-position tool-holder (p/n 805PFB), which allows you to remove and replace tooling exactly as it was, and this function cuts setup time for periodic jobs to just a few moments. Also works well for re-loading offsets for different programs which use same set of tooling.

- **Set the number of cycles to repeat:**

Press C at this screen, then enter the number of cycles to repeat, then press Enter to return. Press C at Auto screen to enable Cycle Repeat. This is good for use with a barfeed. As an example you could set the machine up and tell it you need 20 pieces. The OmniTurn will count down to zero and then stop.

Switches C.Cycle Repeat /.Block Skip O.Optional Stop S.Single Block	Press Cycle Start to Begin	Feed 0.000 IPM 100%
Functions F2-Tool Offsets F3-Edit F4-Verify	Cycles remaining: 20	Speed 0 RPM 100% Tool 0

- **Turn on the Parts Counter:** Press P at this screen. You will return to the Auto mode screen, and there will be a counter on the screen. The counter will count up each time an M30 or M02 is executed. To clear the counter come back into the F10 screen and press P again.

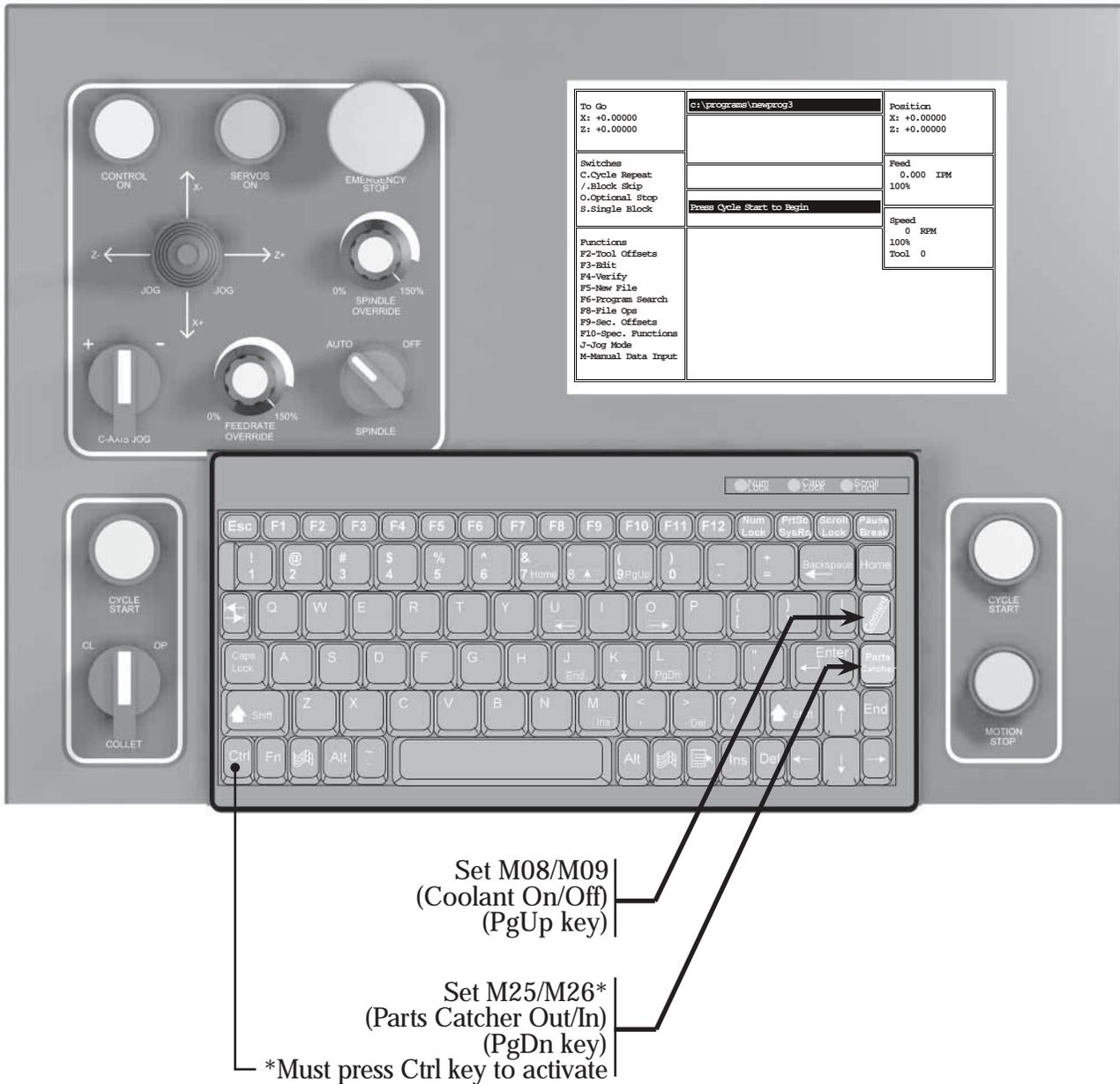
Switches C.Cycle Repeat /.Block Skip O.Optional Stop S.Single Block	Press Cycle Start to Begin	Feed 0.000 IPM 100%
Functions F2-Tool Offsets F3-Edit F4-Verify	Part Count: 0	Speed 0 RPM 100% Tool 0

Automatic Mode

M function keyboard controls

Toggle M functions on and off with keyboard controls.

- Press the key once to turn the function on, Press again to turn it off
- Works only in Jog or Automatic mode



To review the functions of all the front panel controls, refer to pages 1.4, 1.5 and 1.6 in Section 1.

Automatic Mode, F3: Edit Program

F3 Edit; used to change existing programs, or create new ones

Note: To create new programs, see pages 1.17 to 1.20 in Section One, Jog Mode.

Editing an existing program:

The editor is accessed from the Automatic mode by pressing F3. The program currently in memory will be edited. If you want to edit a program not in memory, press F5 to invoke File Operations screen, then highlight desired program and press Enter to load it into memory (see 1.17 - 1.20 for details).

You will be prompted to press Esc for no backup, or F1 to make a backup copy with “.&” extension. The backup is unnecessary if you have consistently answered “Y” to backup your program files when OmniTurn is first turned on. (See page 1.8). After pressing Esc or F1 the editor will open with your program.

To add lines to a program:

- Move the cursor to where you need to add the line.
- Type the new text and press Enter

To correct existing text:

- move the cursor to the text to be corrected
- type the new text in
- delete the text being removed

To save corrections without exiting editor (do this often while editing)

- Press F1 to access special functions screen
- Press F3 to save the corrections that have been made

To exit the editor and save corrections made

- Press F1 to access special functions screen
- Press F2 to save the corrections that have been made and exit the editor

To exit the editor and NOT save the corrections made

- Press F1 to access special functions screen
- Press Esc to halt the automatic saving function of the exit routine
- Press F2 to exit the editor without saving the corrections

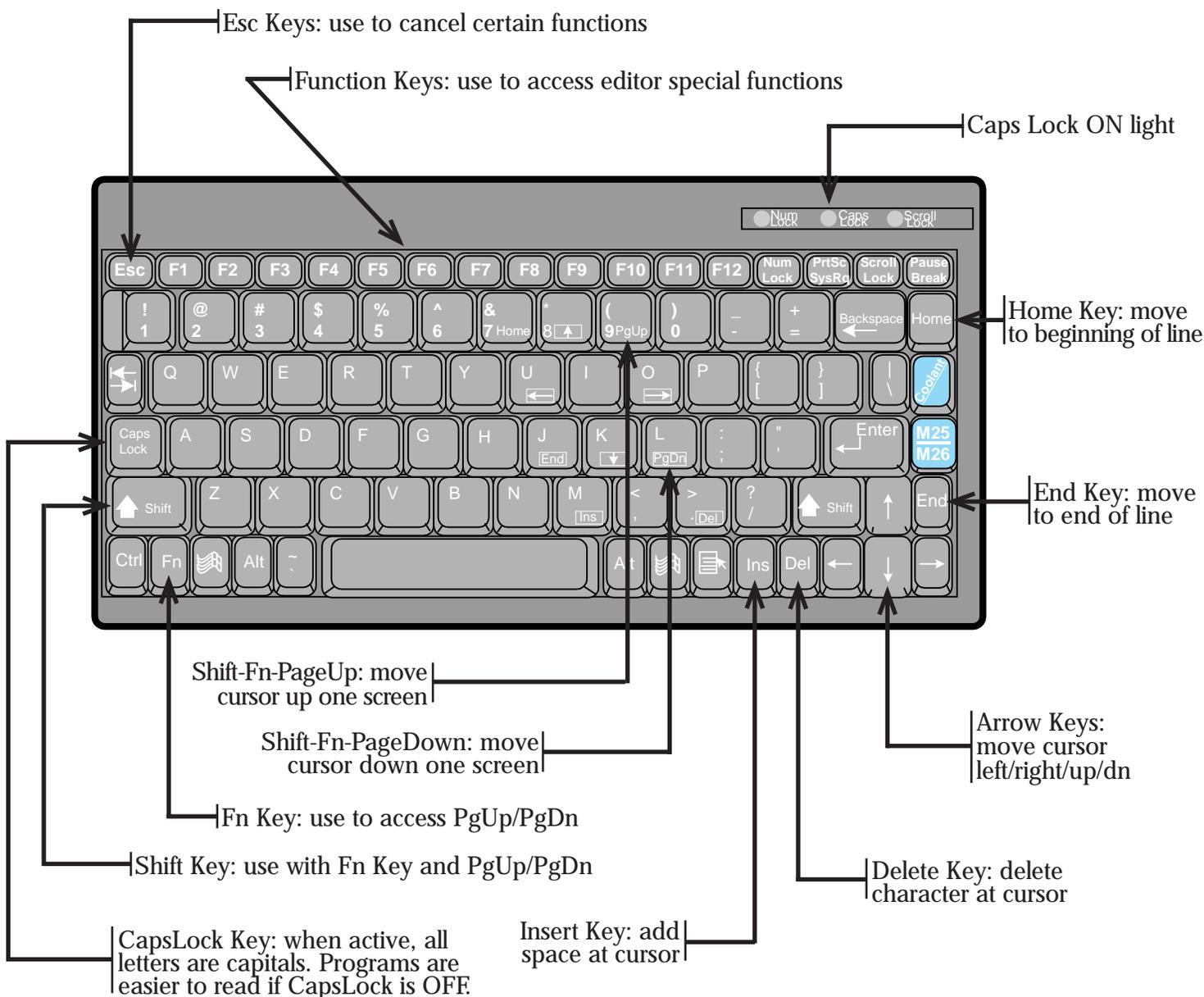
To save program with different name (useful for creating ‘family’ of parts)

- Press F1 to access special functions screen
- Press F5 to rename program; type new name. Note that “c:\programs\” before the program name is the disk drive and folder where the program is stored. Press Enter to return to editing. Program is NOT saved yet.
- Press F1, then F3 to save the program and continue editing

Some special functions while editing

- F1: access special functions screen. Includes Help Menu
- F3: mark beginning and end of block of text to copy, and location to copy to
- F4: mark beginning and end of block of text to delete
- F5: clear block marking set by F3, F4 or F6
- F6: mark beginning and end of block of text to move, and location to move to
- F8: change case of letter at cursor

Automatic Mode, Editing Keys



Function keys: Use to access help screen, highlight text to copy or delete, etc

Arrow keys: Use to move the cursor one character or one line at a time.

Home: moves the cursor to the beginning of the line it's on

End: moves the cursor to the end of the line it's on

Ins: adds spaces in front of the cursor location

Del: deletes the character at the cursor

Back Space: erases the character to the left of the cursor

Fn: use to access Page Up & Page Down keys

Fn-PgUp: moves the cursor one screen back per hit

Fn-PgDn: moves the cursor one screen forward per hit

Automatic Mode -Notes on the Editor

The text editor can help you manipulate and modify your programs. The editing functions are:

F4 -Block delete

This function is used to erase sections of the program. To use this function:

- move the cursor to the beginning of the code that you want to erase
- press F4 to start the text marking
- move the cursor to the end of the text that you want to erase. The text that you are going to erase is now being marked.
- if you want to un-mark the code and cancel the delete command press F5
- to complete the erase press F4 again

F3 -Block copy

This function is used to copy sections of the program.

- move the cursor to the beginning of the code that you want to copy
- press F3 to start the text marking
- move the cursor to the end of the text that you want to copy. The text that you are going to copy is now being marked.
- if you want to unmark the code and cancel the copy command press F5
- to end the marking of the code to be copied press F3 again.
- move the cursor to the location that you want the code copied to, then press F3
- you can copy the code more than once, move the cursor again and press F3 again
- to end the marking press F5

F9 -Find, F10 -Replace

This function is used to find text in the program and then replace it with new text. An example for using this is to change a tool number, or a feed rate.

- go to the text editing screen
- move the cursor to the beginning of the program
- press F9, at the top of the screen you will see FIND ". " type the text that you want to replace. In the case of the example you could type T12
- press F10, at the top of the screen you will see REPLACE ". " type the text that you want to put in place of the found text. In the case of the example you could type T23
- press RETURN get back to the edit mode
- press F10 to find and replace, repeat pressing the F10 key until you have gone through the entire program

Renaming programs

In the editor you can duplicate a program and give it a new name. This is very useful! You can make changes to an existing program and still have the original to fall back to if the changes don't work the way you want, or if you have a family of parts that use the same basic program with only minor changes. With renaming you could make a custom program for each part without having to type the entire program again.

- make the program active in the automatic mode
- press F3 to go to the editor
- press ESC to bring the text to the screen, then press F1 to get to the HELP screen
- at the HELP screen press F5 for RENAME. You will see the original file name at the top of the screen with c:\programs\ in front of the name. An example of this would be "c:\programs\BUSHING". Use the arrow keys to move the cursor to the right of the c:\programs\. Then type the new name of the file and delete the old one. The new name could be "c:\programs\BUSHING2".
- after the name has been typed press RETURN. This will take you back to your program.
- when you are done working with the program, save and exit the editor as you normally would, press F1, then F2. Now you will still have the old program, and a new one.

This page intentionally blank

OmniTurn - Trouble shooting guide

Table of Contents

Description.....	Page
Trouble Shooting Guide	6.2
Ctrl-S Spindle Encoder Counts	6.4
Alt-I Input Monitor Function	6.5
Positioning Problems	6.6
Ctrl-H Diagnostic Mode	6.6
Servo Motor Replacement	6.8
CNC Control Disassembly	6.10
Top Chassis Components	6.11
Connect Card.....	6.12
Front Panel Connect Card	6.12
“5AX” Motion Card.....	6.13
Input / Output (I/O) Assignments	6.13
Computer Chassis	6.14
Setting Servo Amps.....	6.15
Ctrl-E Diagnostic	6.16
OmniTurn Block Diagram (Inverter Drive)	6.17
CNC Control Block Diagram	6.18
Front Panel Interface	6.20
Misc Cable & Operator’s Station (Inverter Drive).....	6.21
Servo Drive Interface.....	6.22
Spindle Panel, Inverter Drive	6.23
Spindle Cabinet TB1 Wiring, Inverter	6.24
Spindle Drive Wiring, Inverter	6.25
Inverter Drive Illustration	6.26
GT Air Pack.....	6.27
OmniTurn Block Diagram (C-Axis)	6.28
Misc Cable & Operator’s Station (C-Axis)	6.29
Spindle Panel (C-Axis)	6.30
Spindle Cabinet TB1 Wiring (C-Axis)	6.31
Spindle Drive Wiring (C-Axis)	6.32
Collet Closer Installation and Maintenance	6.33
Lubricator Operation & Service.....	6.37
Lubrication & Maintenance Schedule.....	6.38

OmniTurn - Trouble shooting guide

The Yellow power light won't come on

The yellow power light indicates that the computer and auxiliary power supplies are working. The 12V should come on when the system is turned on and the computer is working. If the monitor displays the start-up information but the yellow power light is off, replace the bulb (#53 miniature bayonet).

- Be sure there is power to the machine.
- Check that disconnect on spindle cabinet is ON.
- Check that the CNC Control is plugged into socket on back of machine.
- Check the fuse on the back of the CNC Control.
- Check fuses on step-down transformer in spindle cabinet.
- Check fuse on auxiliary power supply in CNC Control.

The Blue servo light won't come on

The Blue servo light indicates that the servos have been turned on. If this won't illuminate that means that the E-Stop string is open, or there is a problem with the drives or their control system.

If you hear the servos engage when the button is pressed, but the blue light is off, replace the bulb (#53 miniature bayonet).

- Check that the E-stops on CNC and Operator's Station (palm box) are released.
- Check that the Operator's Station is plugged into the CNC.
- Check the 1CR relay on the connect card in the control, be sure it is plugged in.
- Check voltage at CRDY test-point on Connect Card (see page 6-12). At "Please turn servos on" prompt, CRDY should stay near zero volts when Servos On button is pressed. If voltage at CRDY goes up to 12V ribbon cable to motion card could be bad, or problem could be on motion card. If CRDY stays low, the E-Stop string could be open (see schematic on page 6.18) or ribbon cable to front panel connect card could be bad or Servos On switch could be bad.

The Monitor won't come on

- If the rest of the system works (the yellow light comes on and, after 1 minute, you can turn on Caps Lock Light on keyboard) but you do not get an image on the screen there could be a problem with the monitor. Make sure 12V cable from CRT, is attached to CN10 on connect card.
- If the Caps Lock Light won't light, the computer is probably not booting up.

The Red Motion Stop light come on every time you try to move the slide

- Check that the Spindle Drive is not faulted; press the reset button on the the spindle drive cabinet located at the left side of GT-75 and GT-Jr.
- Check the hatch on GT-Jr, or any interlocked doors on your system. To jog the machine with door open during setup, get the interlock bypass key from your supervisor.
- If you have a barfeeder, check that the end of bar circuit is not activated

"Servo Axis Error" when you try to jog the slide:

Most common cause is from over-travelling the slide and hitting a "hard stop".

- Press "ESC", this will reset the control. If the control shuts down again when you try to jog the axis go to the next step.
- Check the two thermal overloads. These are located on the rear panel of the CNC. It can take a few minutes for the overload to cool enough to be reset. Press "Esc" and try to jog again.
- Turn the control off, wait 15 seconds, turn the control on again. This will reset the Servo driver card. If the control shuts down again when you try to jog, go to the next step.
- Check for loose motor cable; set power off, then disconnect and reconnect.

OmniTurn - Trouble shooting guide

If only one axis is giving you problems, do the following:

- Turn the control off and swap the cables to the motors, Z to X and X to Z. When you try to jog in Z, the X motor should move; when you try to jog in X, the Z motor should move.
- Turn the control back on and try to jog in the good axis; if the “bad” motor moves, the motor is probably ok and the motion control card is bad; if the motor doesn’t move, it’s probably bad. Try the other axis and verify your conclusions.
- If you have questions about this procedure, call the factory (541-332-7004)

The slide crashes whenever a program is run

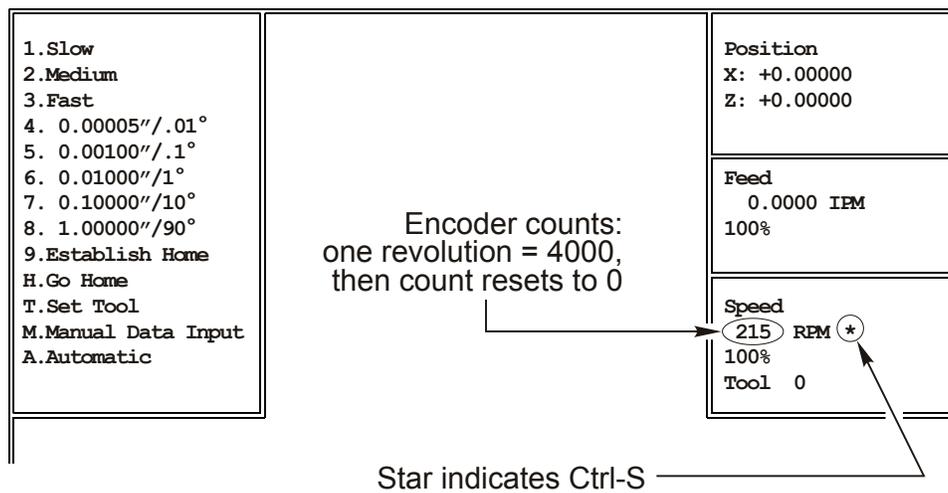
- | | |
|-----------|--|
| Problem | <ul style="list-style-type: none">• The HOME position has been improperly set• Either the tool offsets have not been set or have been lost• Your program is incorrect• Check the XnZn statements after a tool change, be sure you have one for each and they are correct. |
| Solutions | <ul style="list-style-type: none">• Reset HOME• Check your tool offset table• Check your program• Reset the tool• Reload the tool offset table, see F10 in the automatic mode |

Spindle won’t come on

- Check that both E-stops are released, on CNC control and operator’s station
 - Turn the ”spindle override” pot on the CNC control full CW
 - Turn the spindle switch on the CNC control to ”AUTO”
 - Check that the spindle lock pin is pulled out
 - Check that MISC cable from the OmniTurn CNC to Spindle Cabinet is connected
 - Check that Operator Station Cable is connected to Spindle Cabinet
 - Spindle drive box must be turned on (attachments only)
 - Cables to the spindle drive box must be plugged in
-
- Spindle drive may be in ‘fault’ condition. Push the red reset button on the spindle drive box to reset the spindle drive. This could be tripped for a number of reasons:
 - Too low or too high line voltage
 - The duty cycle is too much; cycle time is too short and too often.
 - Noise from coolant pump or other contacts
 - Acceleration or deceleration are too short (standard 5hp drive). *Parameters can be set to change these: see “AC Spindle Drive” pg 6.18*

OmniTurn - Trouble shooting guide

Spindle Encoder Counts (Ctrl-S Diagnostic)



Spindle Encoder Counts (Ctrl-S Diagnostic)

Pressing Ctrl-S in Jog Mode enables display of spindle encoder counts and marker.

If the slide doesn't move, or moves erratically in any "inches per revolution" mode (g95, g33, g83 etc) Ctrl-S is a quick way to determine if the spindle encoder is at fault.

When Ctrl-S is active, a star (*) appears to the right of RPM in the spindle speed box. The numbers to the left of RPM are encoder counts. Turning the spindle by hand will increase or decrease the numbers until the marker is encountered. There are 4000 encoder counts and one marker. When the marker is encountered, the numbers will increase to 3999, then reset to zero.

To verify that the spindle encoder is functional, go to Jog Mode, press Ctrl-S, then slowly turn the spindle while watching the encoder count.

Initially, turn the spindle first one way, then the other less than 1/8 turn: unless you are right at the marker, the numbers should count up in one direction and count down in the other. If the numbers don't change, or if they only change in one direction, the encoder is faulty.

If the numbers increase and decrease, turn the spindle to increase the numbers. If the marker is working properly, the numbers should count to 4000, then start over at zero. If the numbers don't reset, the encoder is faulty. To alert you to this, the color of the numbers will reverse at about 4100 counts.

OmniTurn - Trouble shooting guide

Input Monitor Function (Alt-I Diagnostic)

1.Slow 2.Medium 3.Fast 4. 0.00005"/.01° 5. 0.00100"/.1° 6. 0.01000"/1° 7. 0.10000"/10° 8. 1.00000"/90° 9.Establish Home H.Go Home T.Set Tool M.Manual Data Input A.Automatic	X+ Jog: 1 X- Jog: 1 Z+ Jog: 1 Z- Jog: 1 C+ Jog: 1 C- Jog: 1 Left CS: 1 Right CS: 1 Motion Stop: 1 Collet Open: 1 Collet Close: 1 Spin. Off/Auto: 1 Guards: 0 Inhibit: 1 M91 input: 1 M93 input: 1 M95 input: 1 0 indicates active input	Position X: +0.00000 Z: +0.00000 Feed 0.0000 IPM 100% Speed 0 RPM 100% Tool 0
--	--	--

Input Monitor Function (Alt-I Diagnostic)

Pressing Alt-I in Jog Mode displays status of inputs.

Active inputs (on) are displayed as 0, inactive inputs (off) are shown as 1.

This diagnostic is helpful for quickly determining if the states of the various switches and M-functions are being communicated to the motion control card.

For example: if the machine won't jog, you can quickly verify that the motion control card is reading the state of the Jog Stick by moving the Jog Stick and watching the 1's change to 0's.

Note that the Spindle Off/Auto switch is only monitored when the spindle is commanded to be on.

Also, when door interlock switch is open (1), the Spindle Off/Auto switch is shown as off (1) regardless of its actual condition, and Motion Stop is shown as on (0) regardless of its actual condition.

If guard input is disabled by the presence of the noguards.txt file, the guards input will be shown as active(0) whether the door is open or not.

To cancel Alt-I press Alt-I again.

OmniTurn - Trouble shooting guide

Positioning Problems:

Your parts are not maintaining size; measurements are different part to part.

Using Ctrl-H to diagnose repeatability problems

This test will quickly determine if the problem is electrical or mechanical. It is not necessary to load any special program, or to re-home the machine.

Go to Jog Mode.

Press and hold Ctrl Key, then press H Key: the following box will appear:

1.Slow 2.Medium 3.Fast 4. 0.00005"/.01° 5. 0.00100"/.1° 6. 0.01000"/1° 7. 0.10000"/10° 8. 1.00000"/90° 9.Establish Home	Alt-C will enable C axis Encoder Count Captured X: 0 Z: 0 C: 0	Position X: +0.00000 Z: +0.00000 Feed 0.0000 IPM 100%
---	--	--

Select Jog 1 (Jog Slow) and jog the slide toward the nearest mark on the scale in the axis you're having trouble with. When the "Position" counter passes 0.20000, the "Encoder Count Captured" number should change to 4000 (either plus or minus, depending on the direction you jogged). Illustration below assumes you started at home; if you started at other location, the Position will be at any multiple of 0.200".

1.Slow 2.Medium 3.Fast 4. 0.00005"/.01° 5. 0.00100"/.1° 6. 0.01000"/1° 7. 0.10000"/10° 8. 1.00000"/90° 9.Establish Home	Alt-C will enable C axis Encoder Count Captured X: 4000 Z: -4000 C: 0	Position X: +0.20535 Z: -0.20615 Feed 0.0000 IPM 100%
---	---	--

If the both axes read 4000 , the problem is mechanical: loose coupler, excess backlash, entire slide loose on ways, bad spindle, tooling, etc. Review the suggestions on the next page for help in determining the mechanical cause of the repeatabilty or size problem.

If either axis reads other than 4000, the problem could be a bad bad motor encoder, or some trouble on the motion card.

At this point, to narrow it down, you must load a test program that can run with the motor cables swapped. Controls ship with a program named *smaltest* on the harddrive. *Smaltest* only moves the slide one inch from home in X & Z, so it is safe to run with tooling on the slide. Run five or ten cycles of *smaltest*, then do the Ctrl-H test again. Note which axis is not 4000. Set the E-Stop to turn off servos, and swap the motor cables at the back of the control. Run five or ten cycles of *smaltest* and do Ctrl-H test again: If the same axis doesn't read 4000, the problem is on the motion card. If that axis reads 4000, but the other axis is now not reading 4000, the problem is in the motor encoder.

OmniTurn - Trouble shooting guide

Positioning Problems:

The slide has mechanical problems with repeating a size

- The part is moving, check your work holding fixture
- Be sure the tooling is held tightly
- If you have an **attachment**, see if the slide is loose on the lathe
- The ball screw nut has come loose in it's housing
- Loose encoder at the end of the axis motor
- Loose coupling between the Servo motor and the ball screw for axis movement

To see if it is a mechanical problem position the slide close to the spindle, mount an indicator on the slide so it touches the headstock, zero out the indicator and then push and pull the slide with servos on. The indicator should show some movement as you push and pull on the slide. However it should return to Zero when you let go. If the slide does not come back to Zero then there is something loose, (Slide, Ballscrew nut, Ballscrew taper roller bearing, etc).

To check if the slide for repetition try running a simple program that will show the type of error that you are getting clearly. Run the program in "cycle repeat":

```
g90g94f300(TEST PROGRAM FOR Z AXIS)  
t30(Move the slide so indicator is 1 inch away from headstock)  
x0z1  
z.1  
f20z0 (Creep up on the indicator)  
g04f1(Read the indicator during the dwell)  
f300z1  
m30
```

```
g90g94f300(TEST PROGRAM FOR X AXIS)  
t31(Move the slide so indicator is 1 inch away from headstock)  
xlz0  
x.1  
x0f20  
g04f1(Read the indicator during the dwell)  
f300x1  
M30
```

After running the program and studying the way the indicator repeats, or doesn't, you should have an idea of what your mechanical problem could be. If not, call the factory and describe the type of error that you are seeing, for example: constant creeping in one direction, random movement in both directions, jumping.

OmniTurn - Trouble shooting guide

SERVO MOTOR REPLACEMENT INSTRUCTIONS

Removal:

Set main disconnect OFF and lock out for safety.

1. Disconnect the cable for the faulty motor at rear of CNC and pull it clear of wireways cable clamps and panel holes. Push the slide away from the motor to access the motor coupling. For X-Axis, the slide brake assembly must be removed. Two 6-32 cap screws secure the cylinder to the saddle.
2. Remove the sheet metal covering the slide nearest the motor you wish to change. On X-Axis this cover is held with acorn nuts on 1/4-20 all-thread; on Z-Axis the cover is held by the three phillips head screws through the scale. Remove the motor cover, which is held with one screw.
4. Loosen the cap screw holding the clamp on the coupling on the motor side.
5. Remove two 3/8-18 cap screws that hold the motor mount and motor to the base. You may have to lightly tap the motor mount to remove it from the machine, as it is pinned for alignment.
6. Remove four 10-32 cap screws that hold the motor to the motor mount.

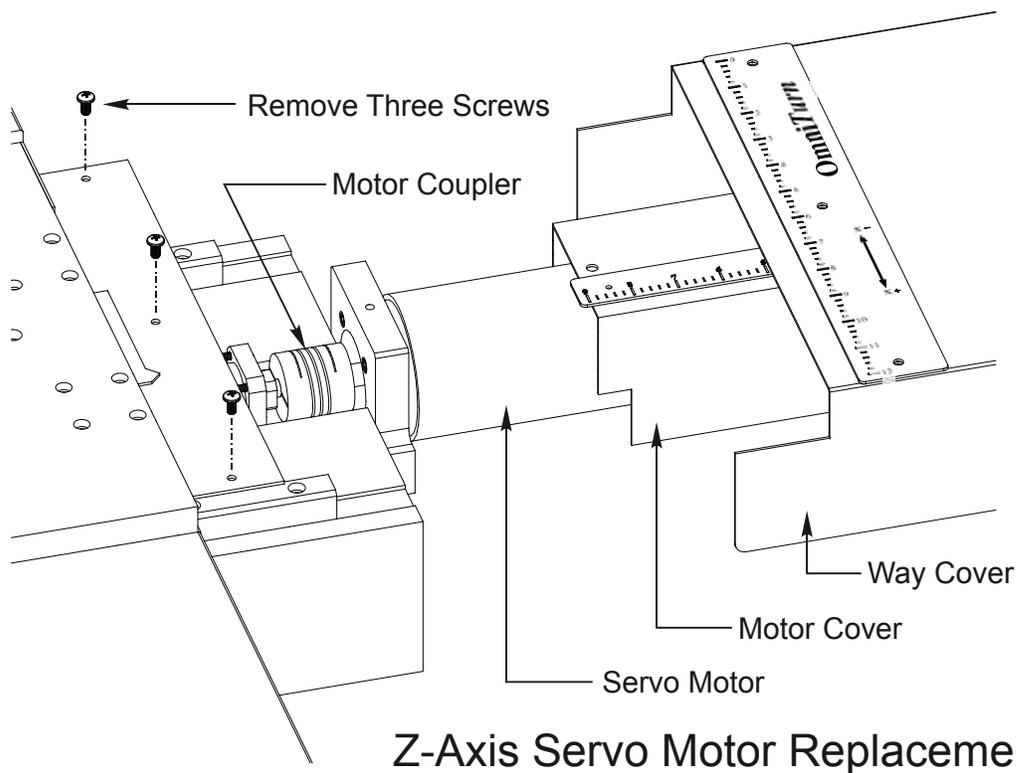
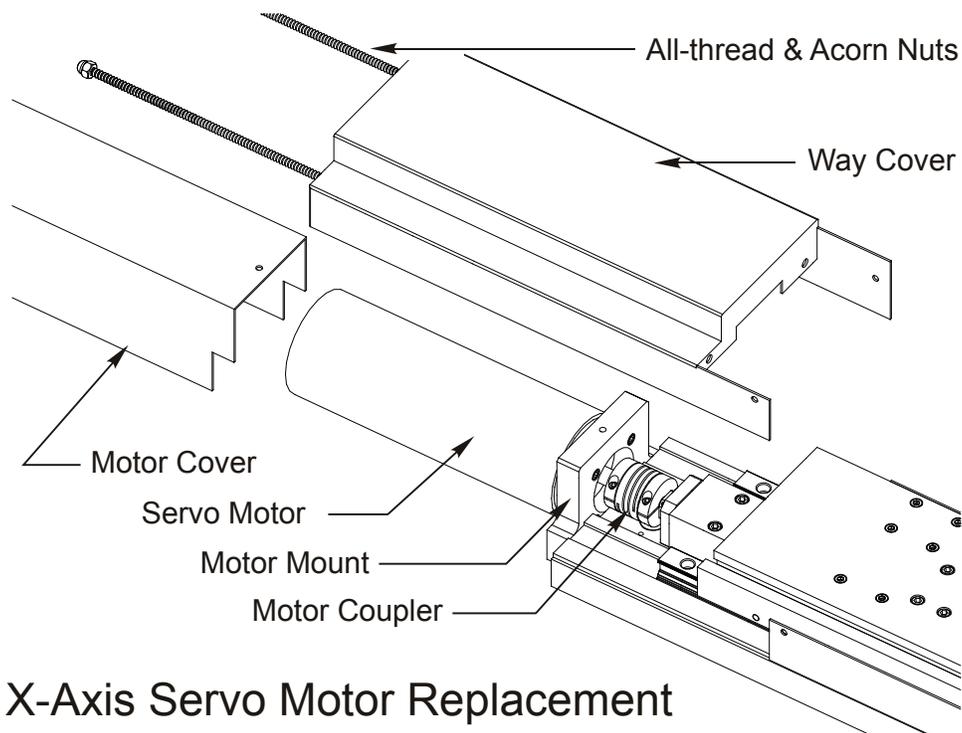
Replacement:

1. Attach the replacement motor to the motor mount with four 10-32 cap screws.
2. Notice the mark on the end of the motor shaft and another on the face of the motor. These marks are aligned when the motor is at "home". Turn the motor shaft so that the marks are 180° apart; that is 1/2 turn. This provides about 0.100" clearance past home.
3. For X-Axis, push the slide all the way down, against the Bellville stack; for Z-Axis, push the slide all the way to the left, against the Bellville stack.
4. Attach the motor mount to the base, slipping the shaft into the coupler. Don't let the shaft turn much. Tighten the motor mount to the machine before tightening the clamp on the coupler.
5. Replace the sheet metal as required.
6. After re-assembly, jog the axis to both ends and verify that the pointer will go just slightly past "0"; jog back to the other side of "0", then establish Home as usual. If the pointer does not indicate "0", loosen it and move it to zero.

CAUTION - YOUR TOOL OFFSETS HAVE CHANGED. The slide will not home *exactly* where used to: if you are set up on a job you must re-set all offsets.

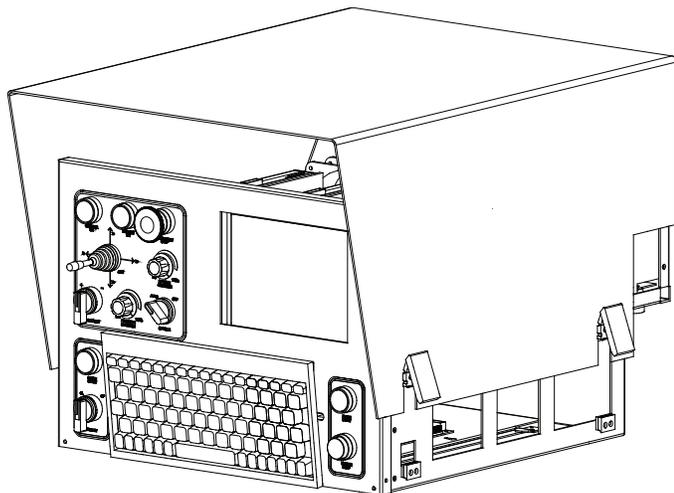
OmniTurn - Trouble shooting guide

SERVO MOTOR REPLACEMENT INSTRUCTIONS

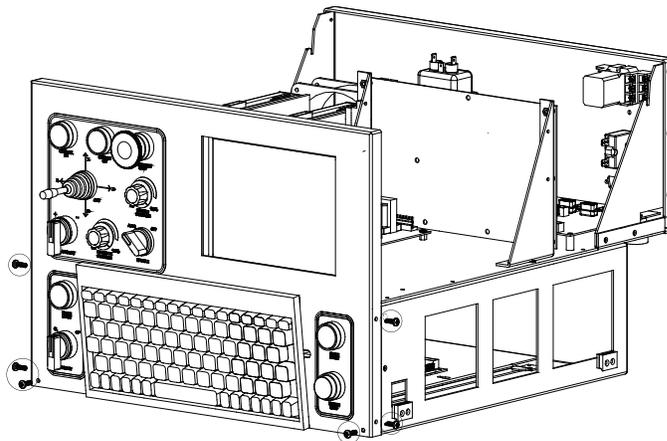


OmniTurn - Trouble shooting guide

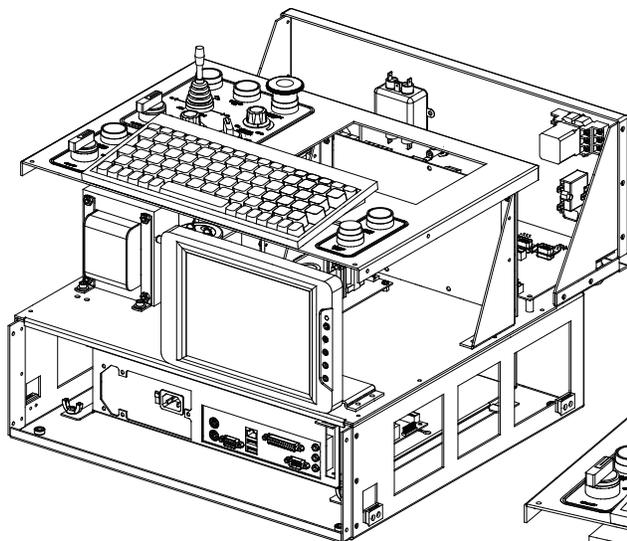
To Disassemble the CNC Control



Un-plug power cord, then remove the blue cover.

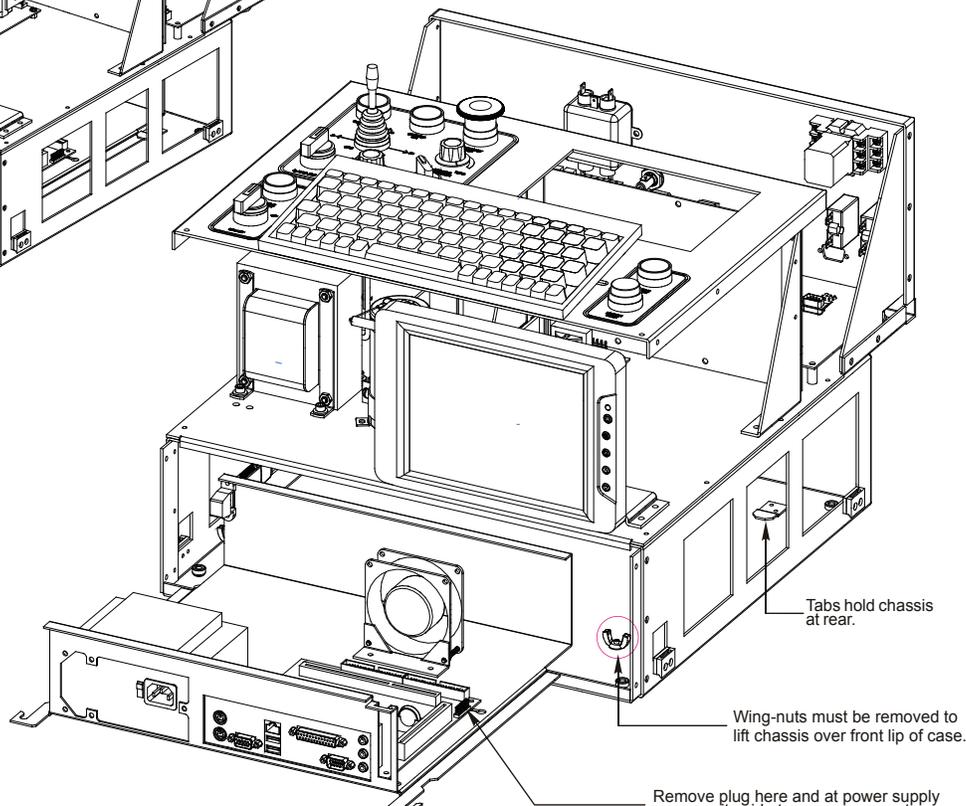


Remove six screws holding front panel.



The computer is held in with wing-nuts and the front; it slides under tabs at the rear.
NOTE: Two plugs must be disconnected before removing the computer, one at header indicated at right, and one power supply connector near powersupply.

It is not necessary to disconnect cables to front-panel. Stow panel gently on top of components as shown.



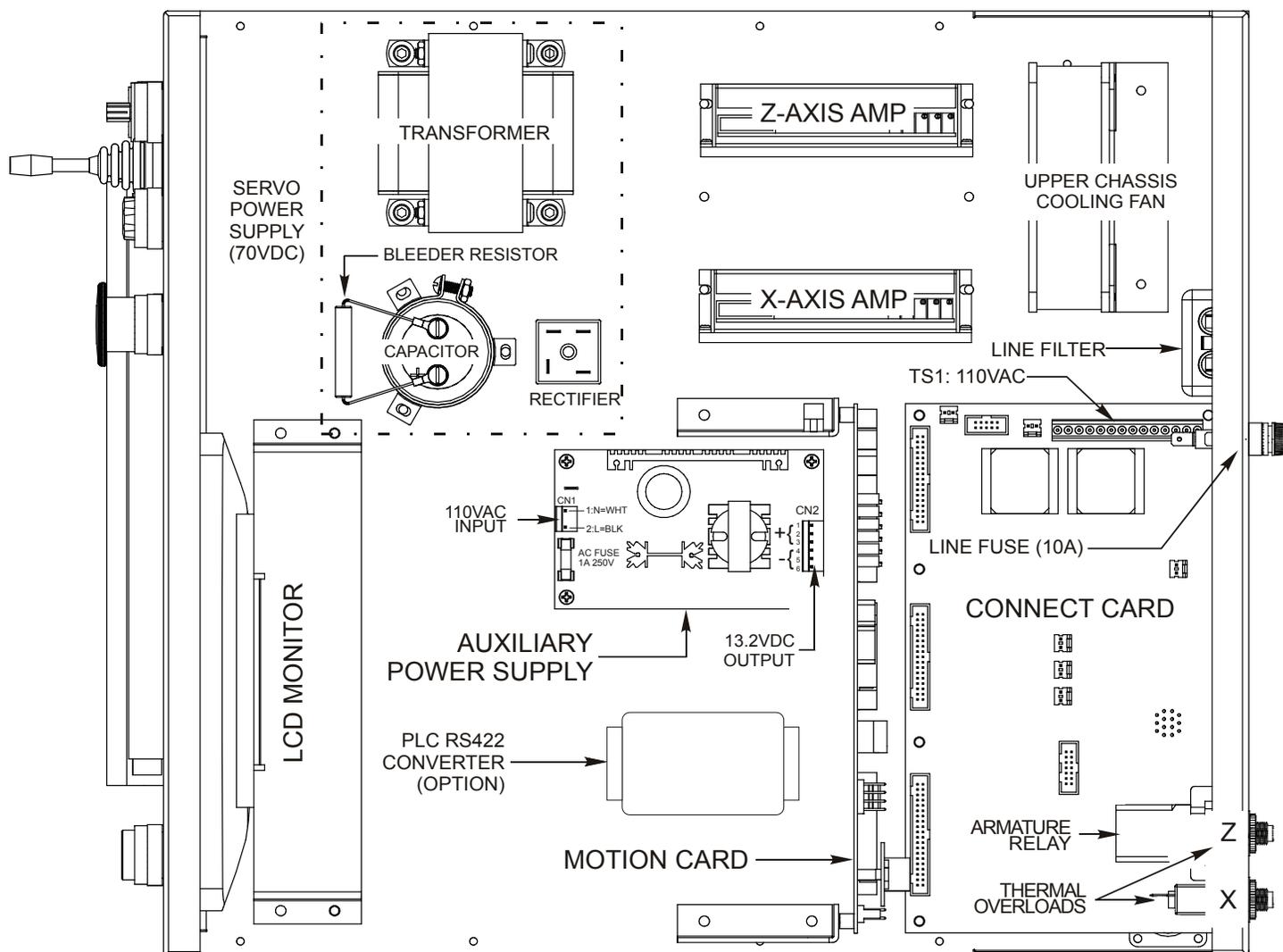
↑
Tabs hold chassis at rear.

↑
Wing-nuts must be removed to lift chassis over front lip of case.

↑
Remove plug here and at power supply

OmniTurn - Trouble shooting guide

CNC Top Chassis



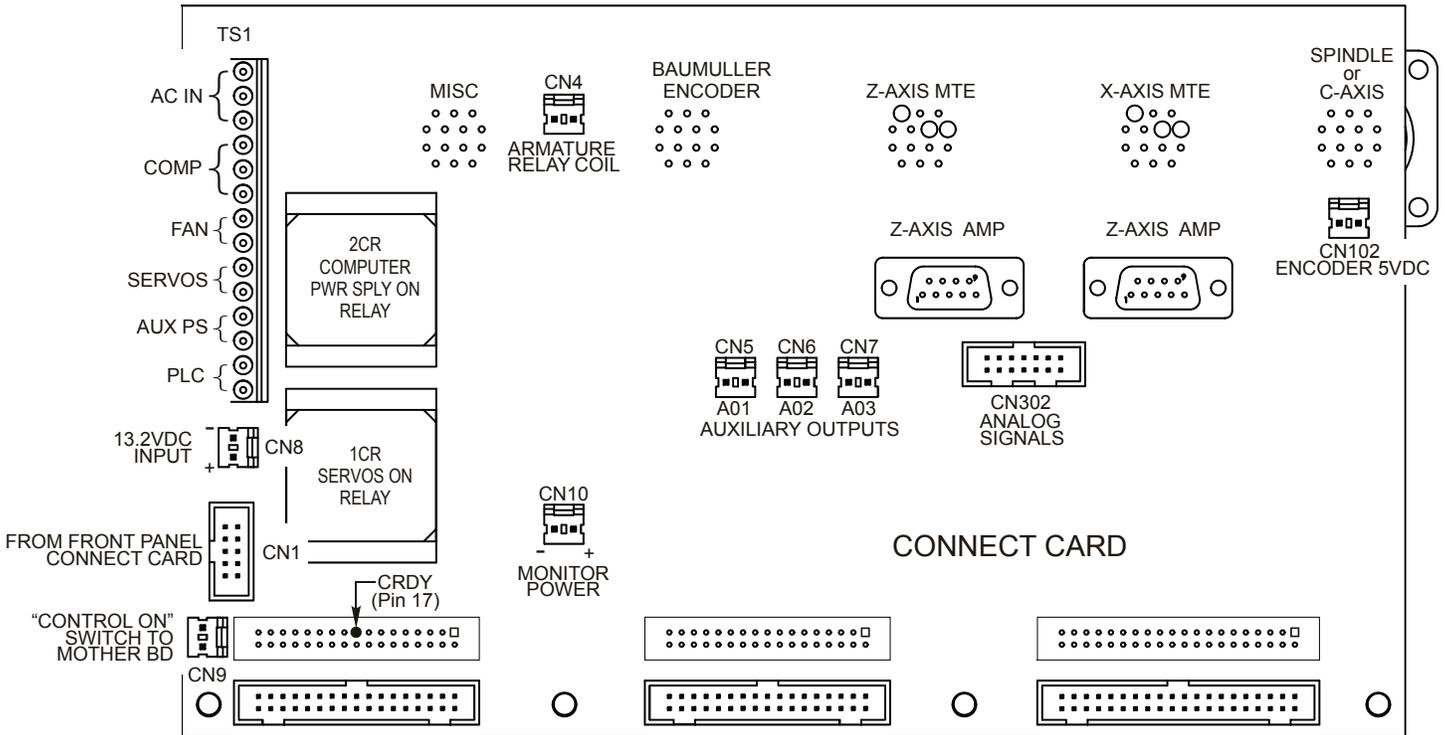
The illustration above shows the location of the main components. Detailed drawings of the Servo Amps, Connect Card and Motion Card will be found on the following pages.

The new generation motion card no longer communicates with the computer via accessory card slots so it has been moved from the computer chassis to the top chassis. It is mounted vertically adjacent to the connect card.

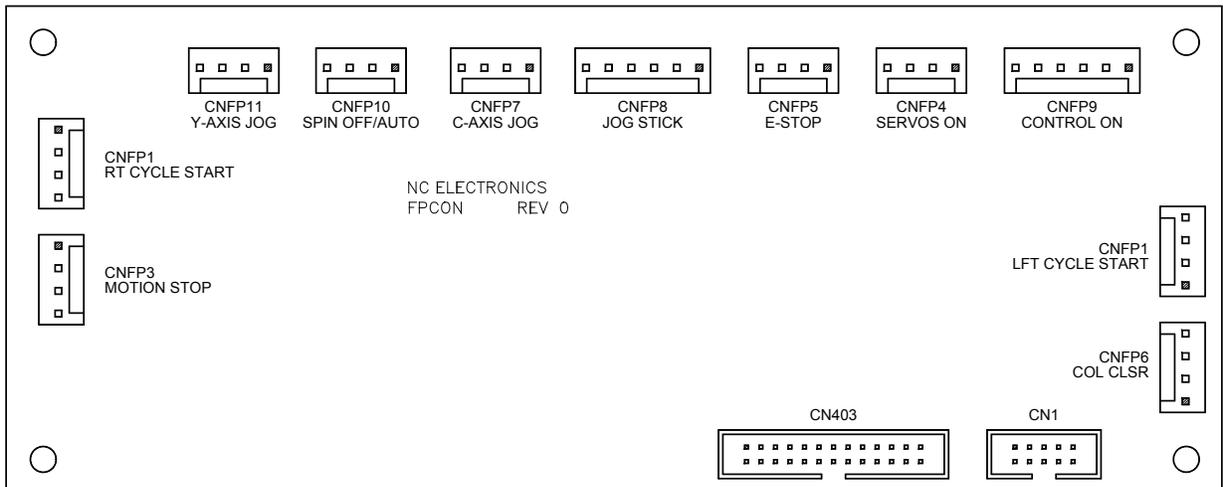
The servo amps and cooling fan have been rearranged to provide better cooling.

OmniTurn - Trouble shooting guide

Connect Card



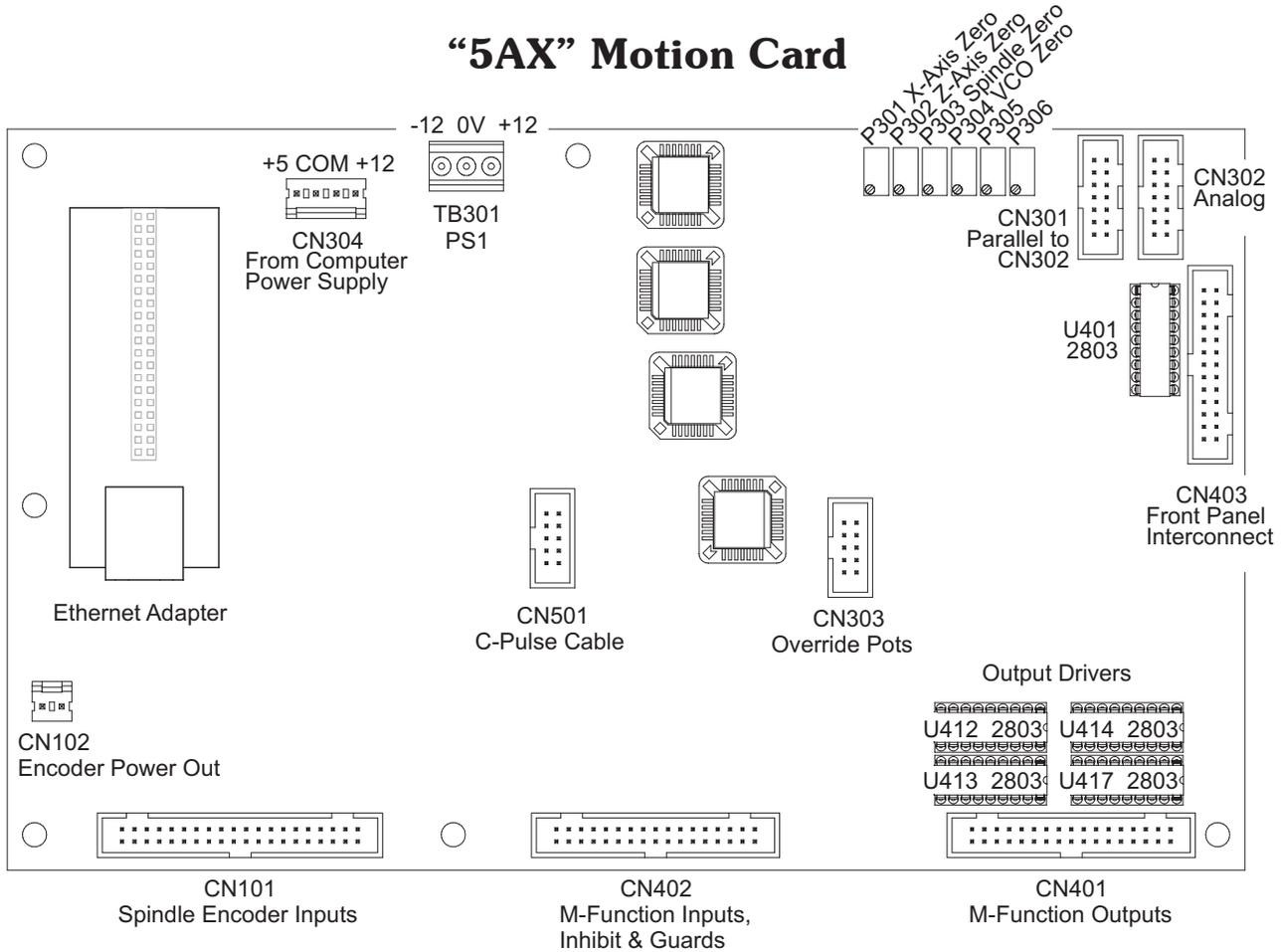
CNC Front-Panel Connect Card



front_panel_connect_card.dcd

OmniTurn - Trouble shooting guide

“5AX” Motion Card



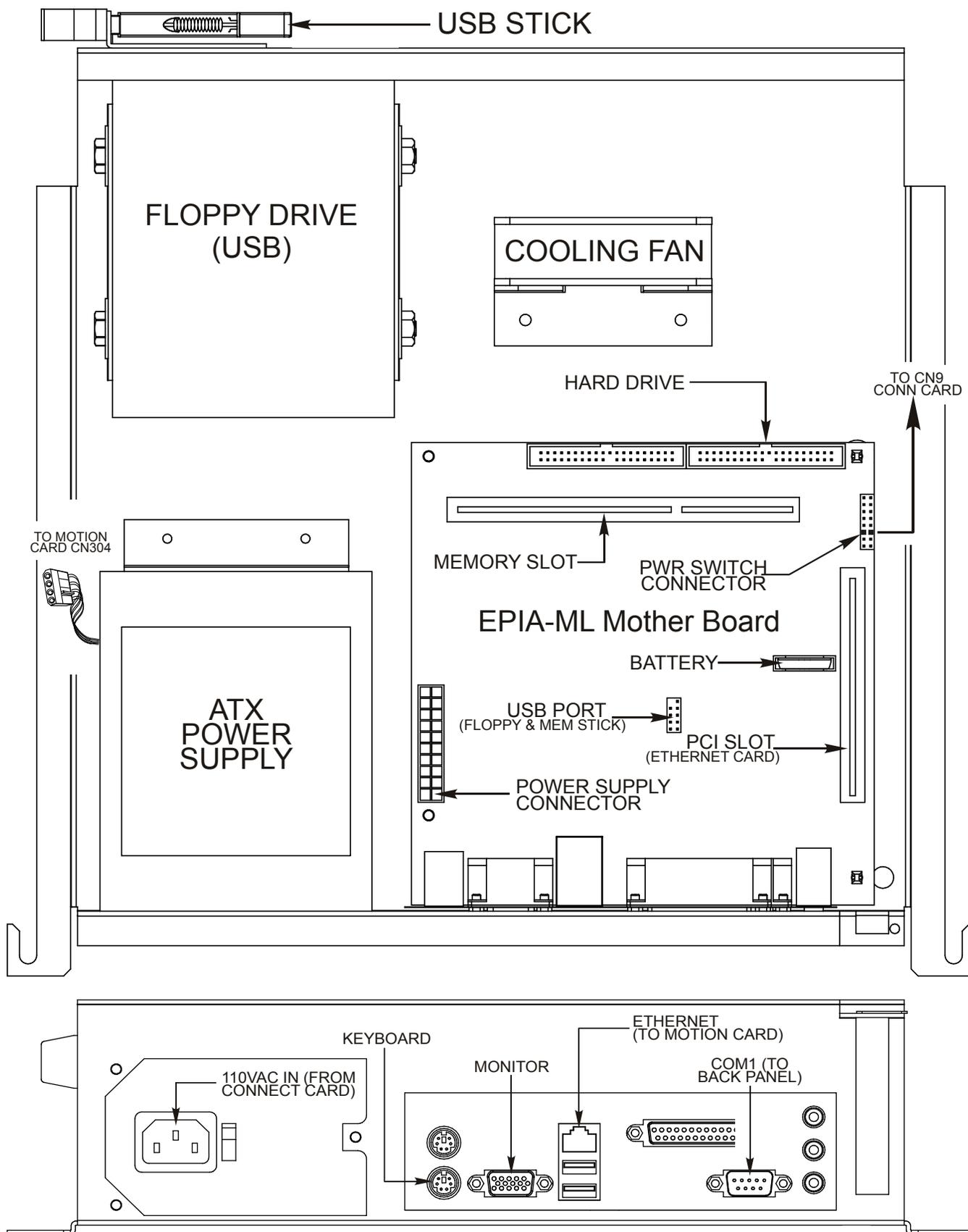
To remove the Motion Card from the chassis, un-screw the brackets from the chassis, not from the card. Do not attempt to un-plug cable to Ethernet Adapter: un-plug entire adapter from Motion Card.

I/O ASSIGNMENTS: M-FUNCTION, CABLE, HEADER, DRIVER IC PIN

OUTPUTS	CABLE	5AX CONN	DRIVER PIN
M03	MISC-6	CN401-6	U413-13
M04	MISC-4	CN401-4	U413-12
M08	MISC-5	CN401-3	U412-12
M12	MISC-7	CN401-5	U412-13
M13	MISC-3	CN401-1	U412-11
M17	CAX-13	CN401-9	U412-15
M21	CAX-9	CN401-8	U413-14
M23	CAX-10	CN401-7	U412-14
M25	MISC-1	CN401-2	U413-11
M27	CAX-11	CN401-10	U413-15
CRDY	N/A	CN401-17	U417-11
AO1(HR. METER)	N/A	CN402-33	U401-14
AO2	N/A	CN402-32	U401-13
AO3	N/A	CN402-31	U401-12
INPUTS (STATUS CAN BE MONITORED IN JOG BY ALT-I)			
GUARDS	MISC-13	CN402-29	
INHIBIT	MISC-8	CN402-26	
M91	CAX-14	CN402-27	
M93	CAX-7	CN402-25	
M95	CAX-12	CN402-28	

OmniTurn - Trouble shooting guide

CNC Computer Chassis (EPIA-ML Mother Board)



OmniTurn - Trouble shooting guide

Setting OMNITURN Servo Amplifiers

NOTE: The amplifiers are pre-set at the factory and should only rarely require slight adjustments to following error. Steps 2 & 3 on pages 6.13 & 6.14 are the only adjustments normally required.

If for some reason it becomes necessary to re-set the amplifiers from scratch, pre-set the pots as follows:

Signal Gain: Full CCW *then four turns CW* (“Full CCW” means 20 turns)
Tach Gain: Full CW (“Full CW” means 20 turns)
Comp: Full CCW *then three turns CW*
Current Limit: Full CW

Tools required: A digital voltmeter (DMM or DVM)
Fine tipped probes, or paper clips
Jewellers common screwdriver or “tweaker”

To correctly setup the servo amps, it is necessary to adjust pots on the Servo Amps and on the Motion Card.

Set Servos ON (*All adjustments are made with servos ON.*)

The steps are as follows:

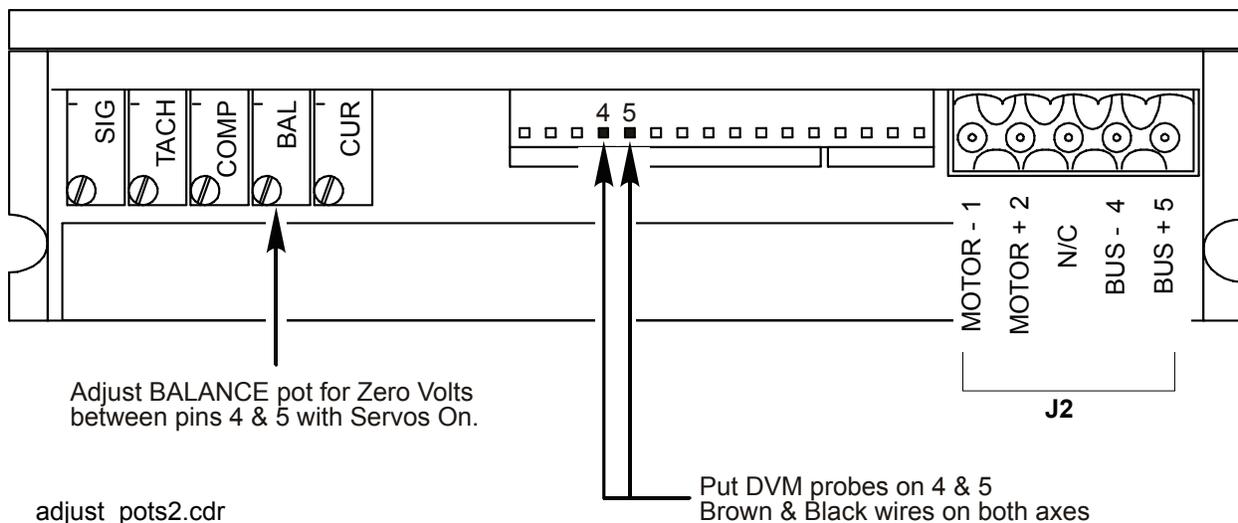
1. Servo Amplifier Balance Adjustment
2. Motion Card Zero Adjustment
3. Servo Amplifier Following Error Adjustment
4. Re-check Servo Amp balance and re-adjust as required
5. Re-check Servo Amp following error

1. Servo Amplifier Balance Adjustment

Put probes of DVM between Pins 4 & 5 on servo amp signal connector. If the probe-tips are too large, cut pieces of paper clip and insert into connector holes to make contact.

Adjust BALANCE pot on Servo Amplifier for 0V +/- 0.005V

SERVO AMPLIFIER BALANCE ADJUSTMENT



OmniTurn - Trouble shooting guide

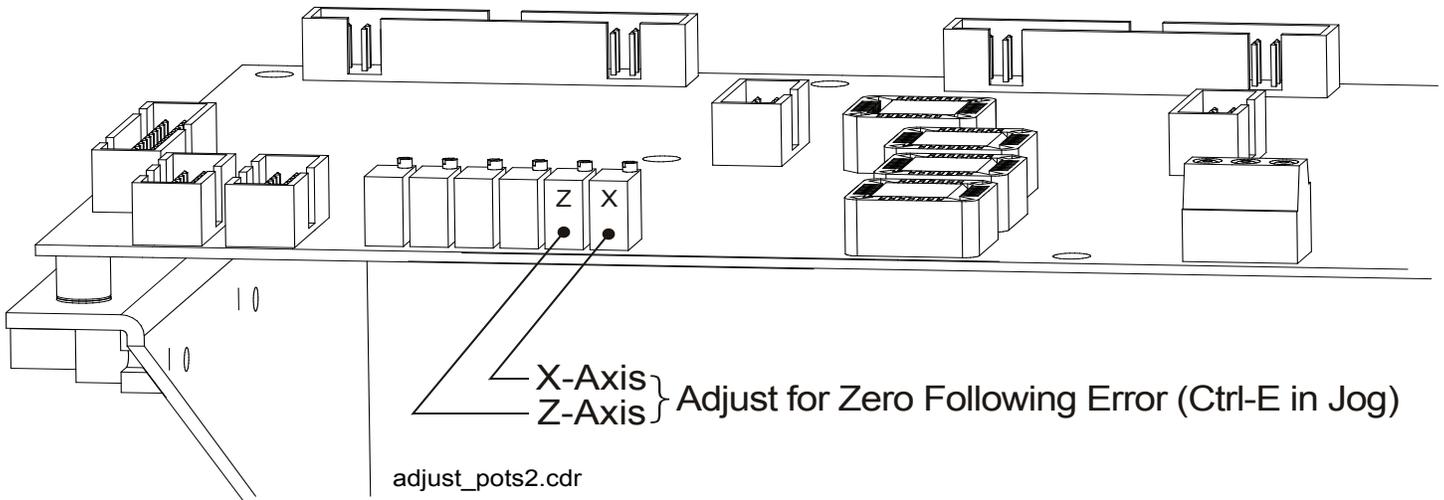
Setting OMNITURN Servo Amplifiers, con't

2. Motion Card Zero Adjustment (Ctrl-E)

In Jog Mode, open Following Error display box. (Press and hold Ctrl key, then press 'E' key).
On the Motion Card, adjust the X and Z Axis Zero pots for zeros at the X: and Z: locations.
Note that +/- 1 is acceptable; the numbers may alternate between 0 and 1.

1.Slow 2.Medium 3.Fast 4. 0.00005"/.01° 5. 0.00100"/.1° 6. 0.01000"/1° 7. 0.10000"/10° 8. 1.00000"/90° 9.Establish Home H.Go Home T.Set Tool	Alt-C will enable C axis	Position X: +0.00000 Z: +0.00000
	Following Error X: 0 Z: -1 C: 0 Set X and Z to 70 at Medium jog	Feed 0.0000 IPM 100%
	adjust_pots2.cdr	Speed

MOTION CARD ZERO ADJUSTMENT



OmniTurn - Trouble shooting guide

Setting OMNITURN Servo Amplifiers, con't

3. Servo Amplifier Following Error Adjustment (Ctrl-E)

In Jog Mode, open Following Error display box. (Press and hold Ctrl key, then press 'E' key).

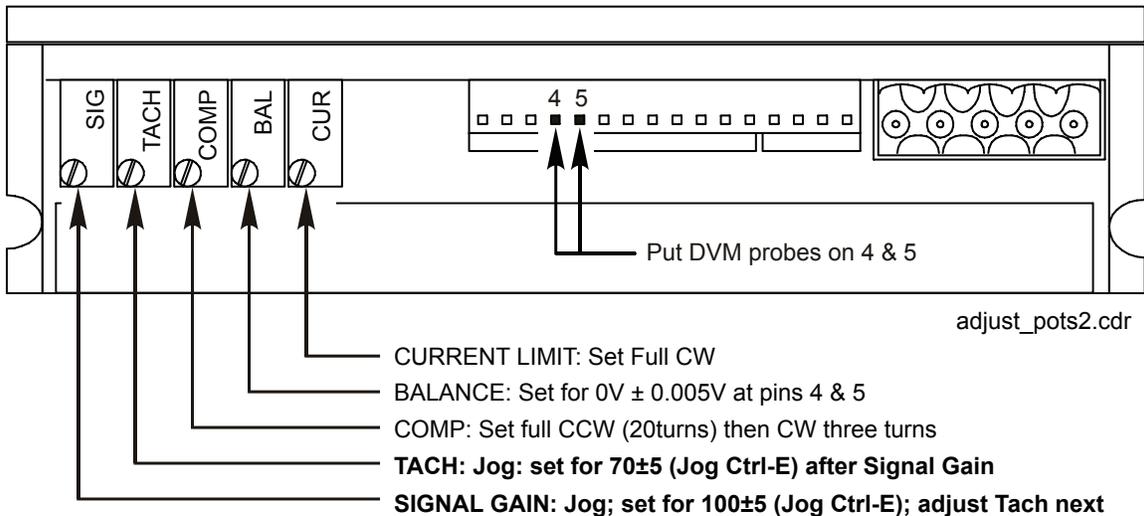
NOTE: The Feedrate Override defaults to 100% in Ctrl-E mode.

Select Jog 2 (Medium Jog)

On the Servo Amplifier, adjust the Signal Gain pot **while jogging**; adjust for 100 counts \pm 5 at the X and Z locations. Next, adjust the Tach pot **while jogging**; adjust for 70 ± 2 counts at the X and Z locations. It doesn't matter which direction you jog; the polarity will change, but the numbers will be effective.

1.Slow 2.Medium 3.Fast 4. 0.00005"/.01° 5. 0.00100"/.1° 6. 0.01000"/1° 7. 0.10000"/10° 8. 1.00000"/90° 9.Establish Home H.Go Home T.Set Tool	Alt-C will enable C axis Following Error X: 70 Z: 71 C: 0 Set X and Z to 70 at Medium jog adjust_pots2.cdr	Position X: +0.00000 Z: +0.00000 Feed 0.0000 IPM 100% Speed
--	--	---

SERVO AMPLIFIER FOLLOWING ERROR ADJUSTMENT



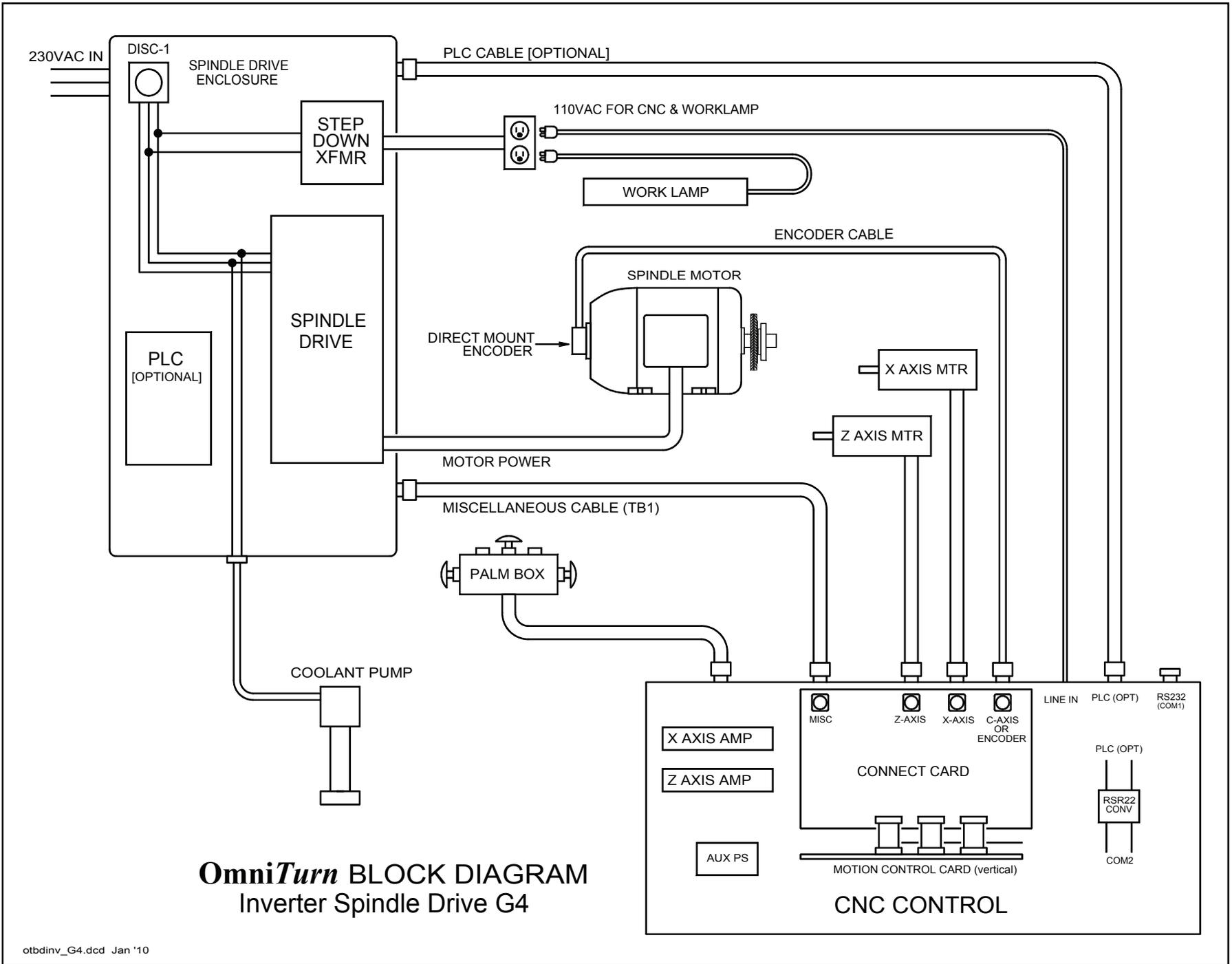
4. Re-check Servo Amp balance and re-adjust as required

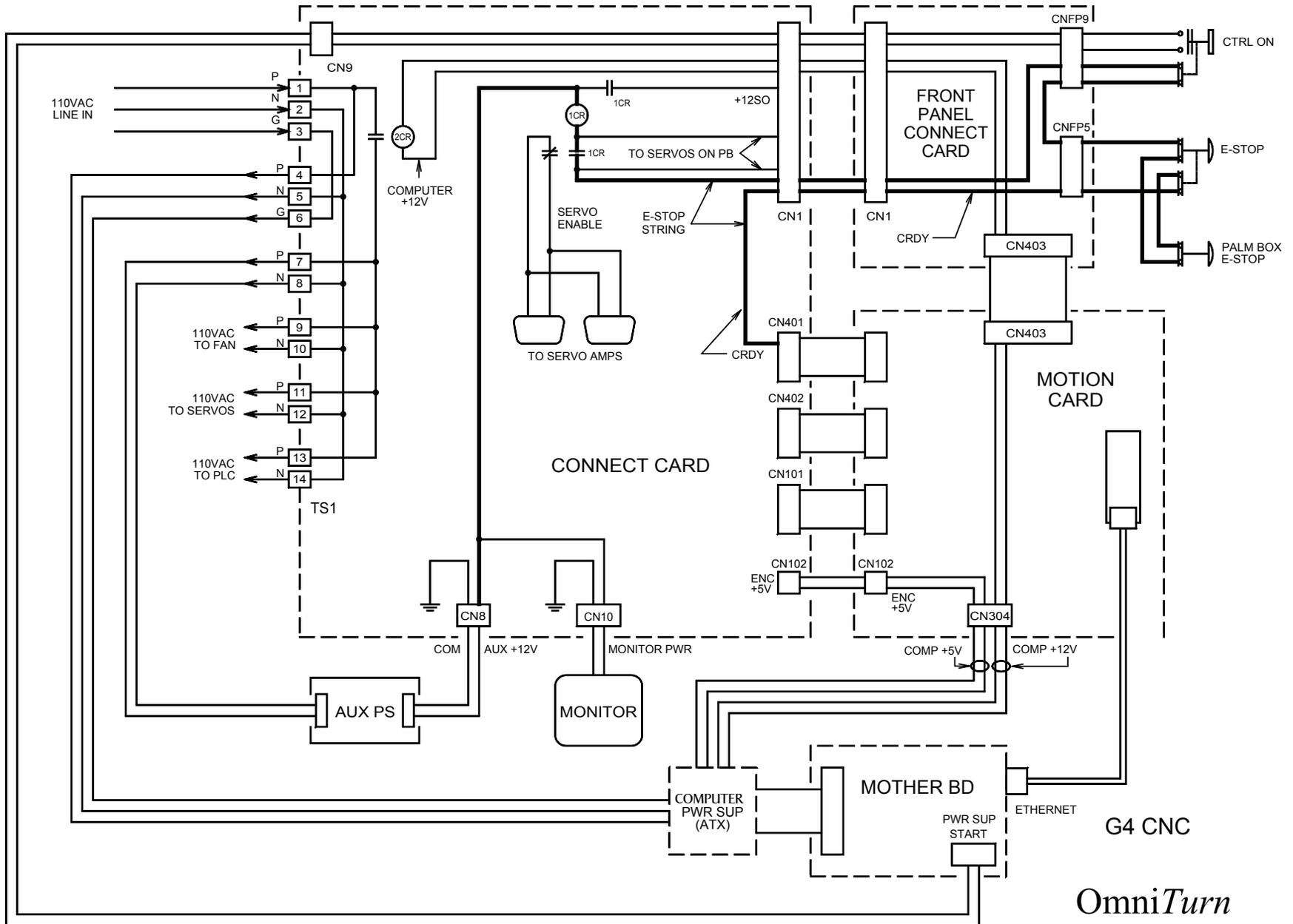
Verify that the voltage between pins 4 & 5 is still $0V \pm 0.005$ at rest (not jogging). Adjust Balance pot as required to set 0V.

5. Re-check Servo Amp following error

After re-adjusting the balance, be sure to jog back & forth at Jog 2 and re-adjust the Tach pot for 70 ± 2 counts. Ctrl-E again to close the Following Error box.

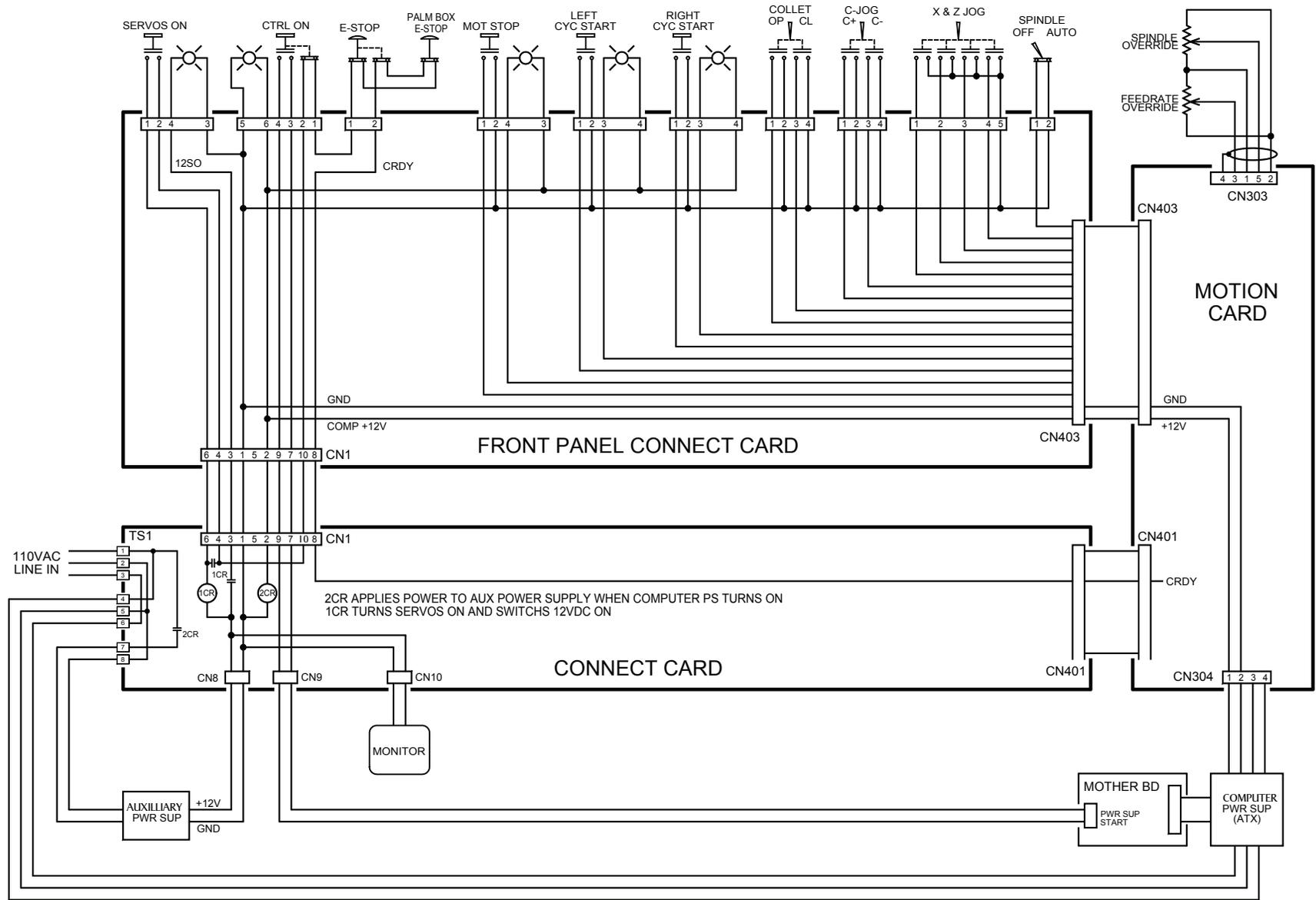
This concludes the Servo Amplifier setup procedure.





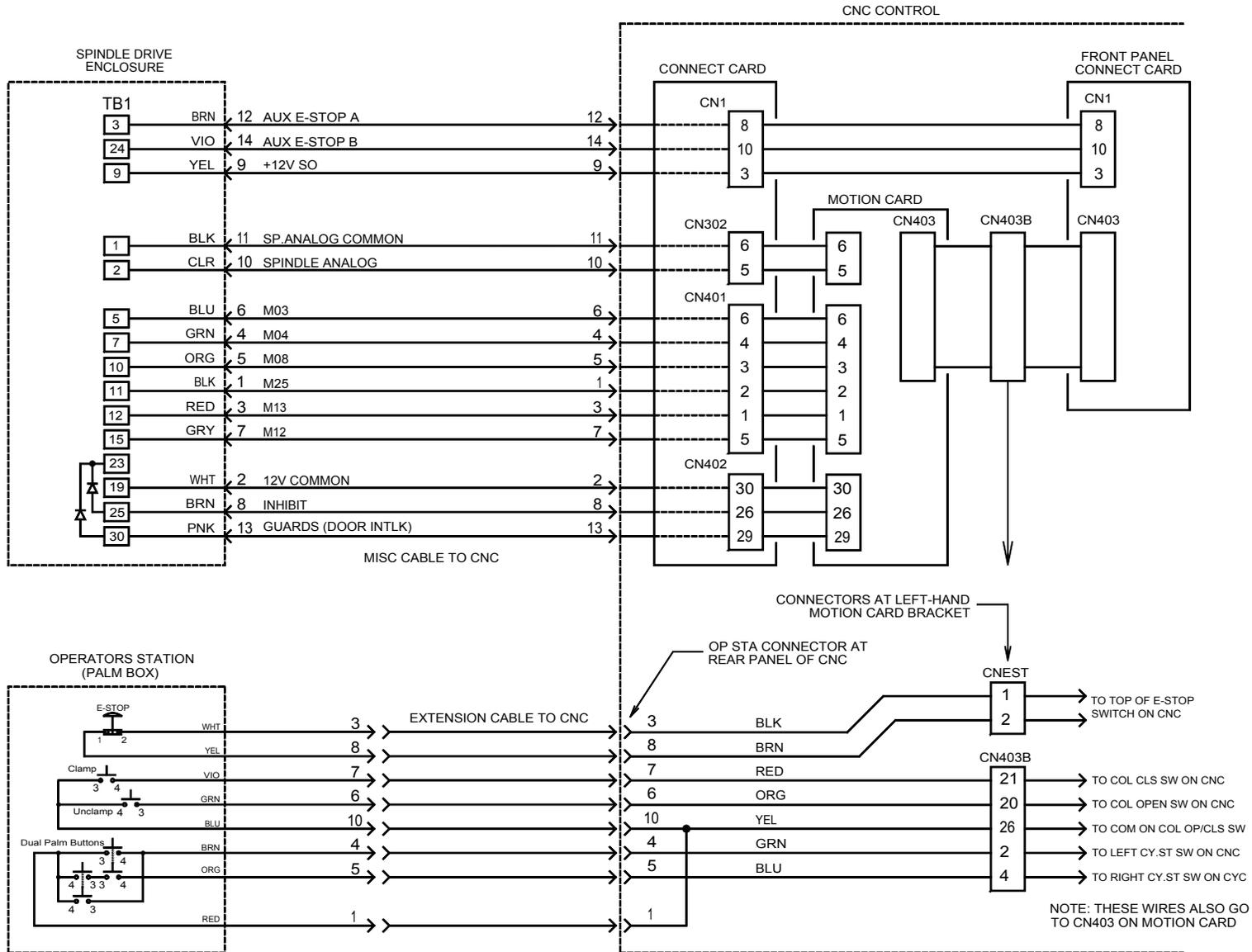
OmniTurn
CNC BLOCK DIAGRAM
E-STOP STRING

cnc_block-diagram-G4.dcd SEPT 2010



G4 CNC

OmniTurn
FRONT PANEL INTERFACE

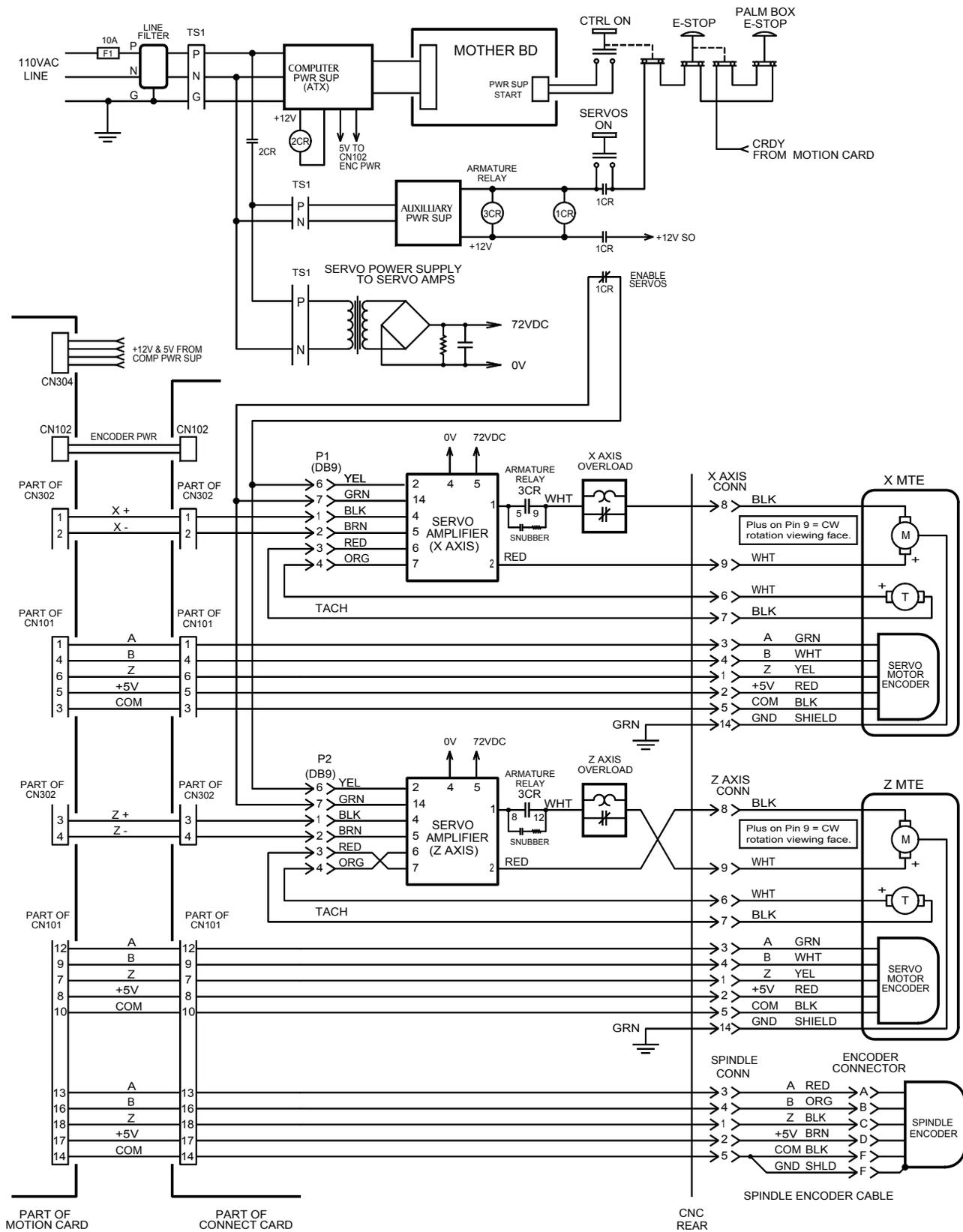


INVERTER DRIVE

G4 CNC

OmniTurn
MISCELLANEOUS CABLE
AND OPERATORS STATION

OmniTurn - Trouble shooting guide



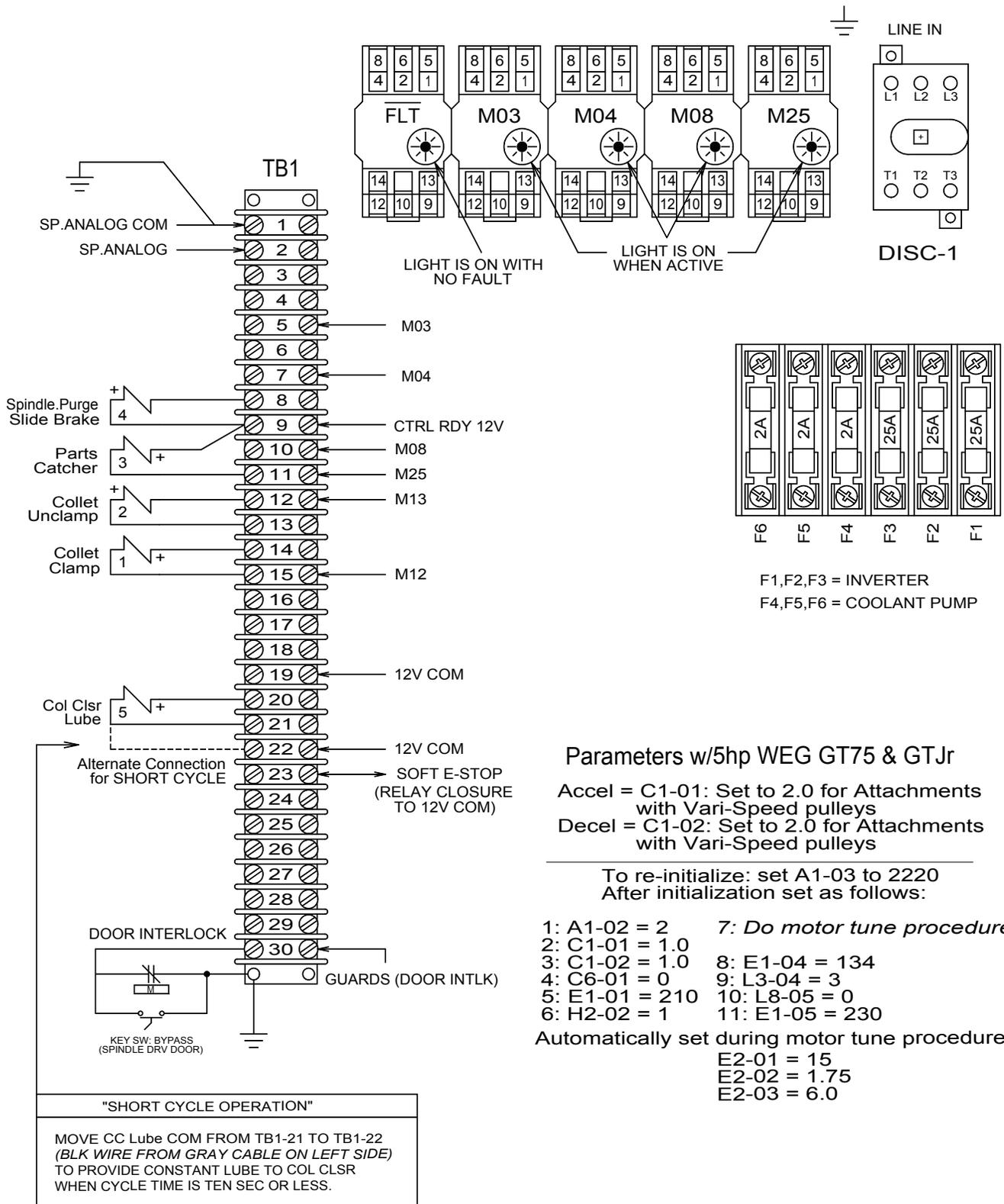
G4
INVERTER DRIVE

OmniTurn
SERVO DRIVE INTERFACE

srvo_sch-G4.dcd 8/2010 (chgd sp.encoder wire colors)

OmniTurn - Trouble shooting guide

NOTE: This drawing includes optional components and references not on all panels.



Parameters w/5hp WEG GT75 & GTJr

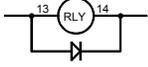
Accel = C1-01: Set to 2.0 for Attachments with Vari-Speed pulleys
 Decel = C1-02: Set to 2.0 for Attachments with Vari-Speed pulleys

To re-initialize: set A1-03 to 2220
 After initialization set as follows:

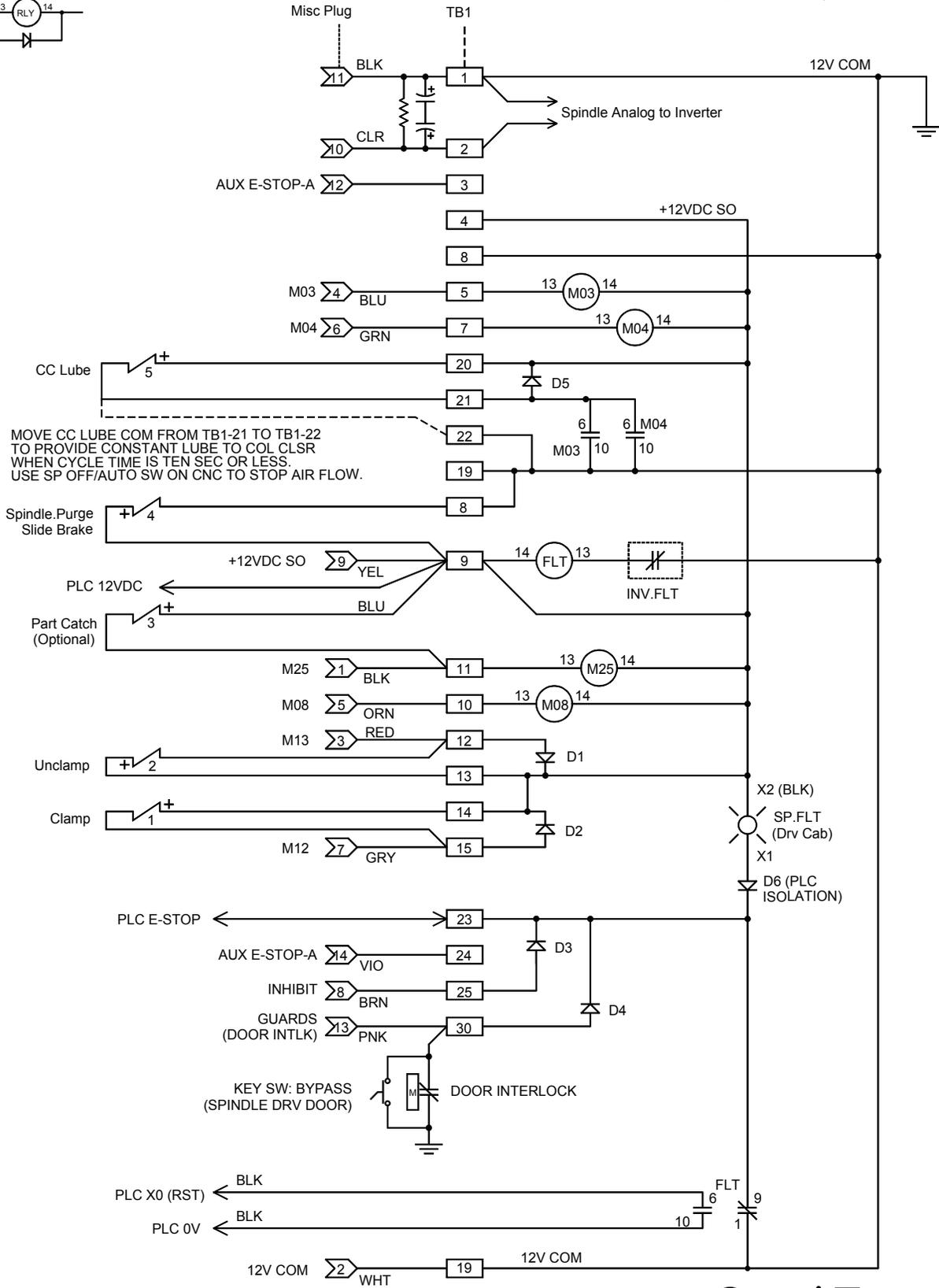
- | | |
|----------------|----------------------------|
| 1: A1-02 = 2 | 7: Do motor tune procedure |
| 2: C1-01 = 1.0 | 8: E1-04 = 134 |
| 3: C1-02 = 1.0 | 9: L3-04 = 3 |
| 4: C6-01 = 0 | 10: L8-05 = 0 |
| 5: E1-01 = 210 | 11: E1-05 = 230 |
| 6: H2-02 = 1 | |
- Automatically set during motor tune procedure:
- E2-01 = 15
 - E2-02 = 1.75
 - E2-03 = 6.0

OmniTurn - Trouble shooting guide

NOTE: All relay coils have diodes:



NOTE: This drawing includes optional components and references not on all panels.



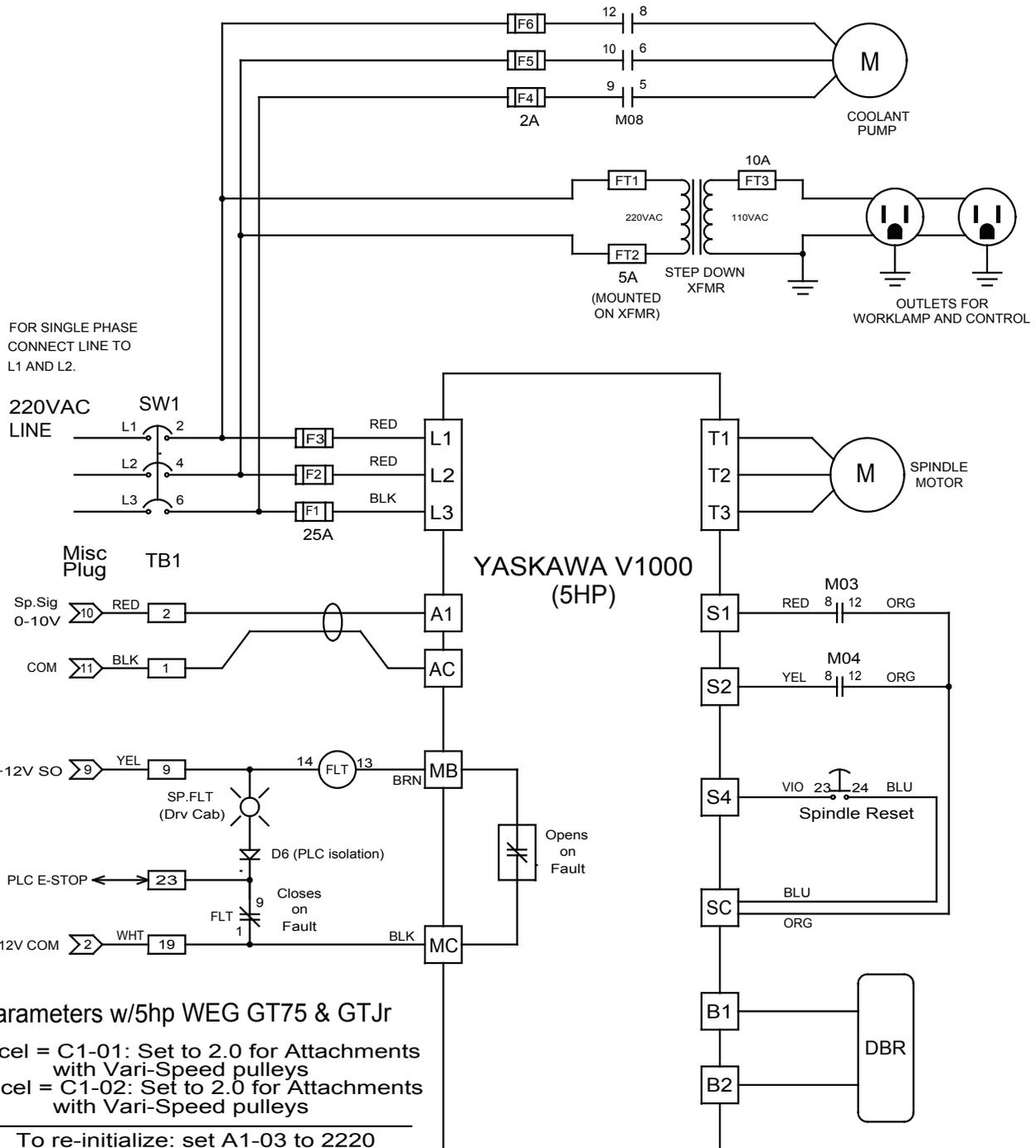
ac_sch1b-G4.dc may 2010
COMBINED SP.PURGE/SLIDE BRAKE

G4 CNC

OmniTurn
AC SPINDLE CONTROL LOGIC

OmniTurn - Trouble shooting guide

NOTE: This drawing includes optional components and references not on all panels.



Parameters w/5hp WEG GT75 & GTJr

Accel = C1-01: Set to 2.0 for Attachments with Vari-Speed pulleys
 Decel = C1-02: Set to 2.0 for Attachments with Vari-Speed pulleys

To re-initialize: set A1-03 to 2220
 After initialization set as follows:

- | | |
|----------------|----------------------------|
| 1: A1-02 = 2 | 7: Do motor tune procedure |
| 2: C1-01 = 1.0 | 8: E1-04 = 134 |
| 3: C1-02 = 1.0 | 9: L3-04 = 3 |
| 4: C6-01 = 0 | 10: L8-05 = 0 |
| 5: E1-01 = 210 | |
| 6: H2-02 = 1 | 11: E1-05 = 230 |
- Automatically set during motor tune procedure:
- | |
|--------------|
| E2-01 = 15 |
| E2-02 = 1.75 |
| E2-03 = 6.0 |

DBR = 40 OHMS 500W

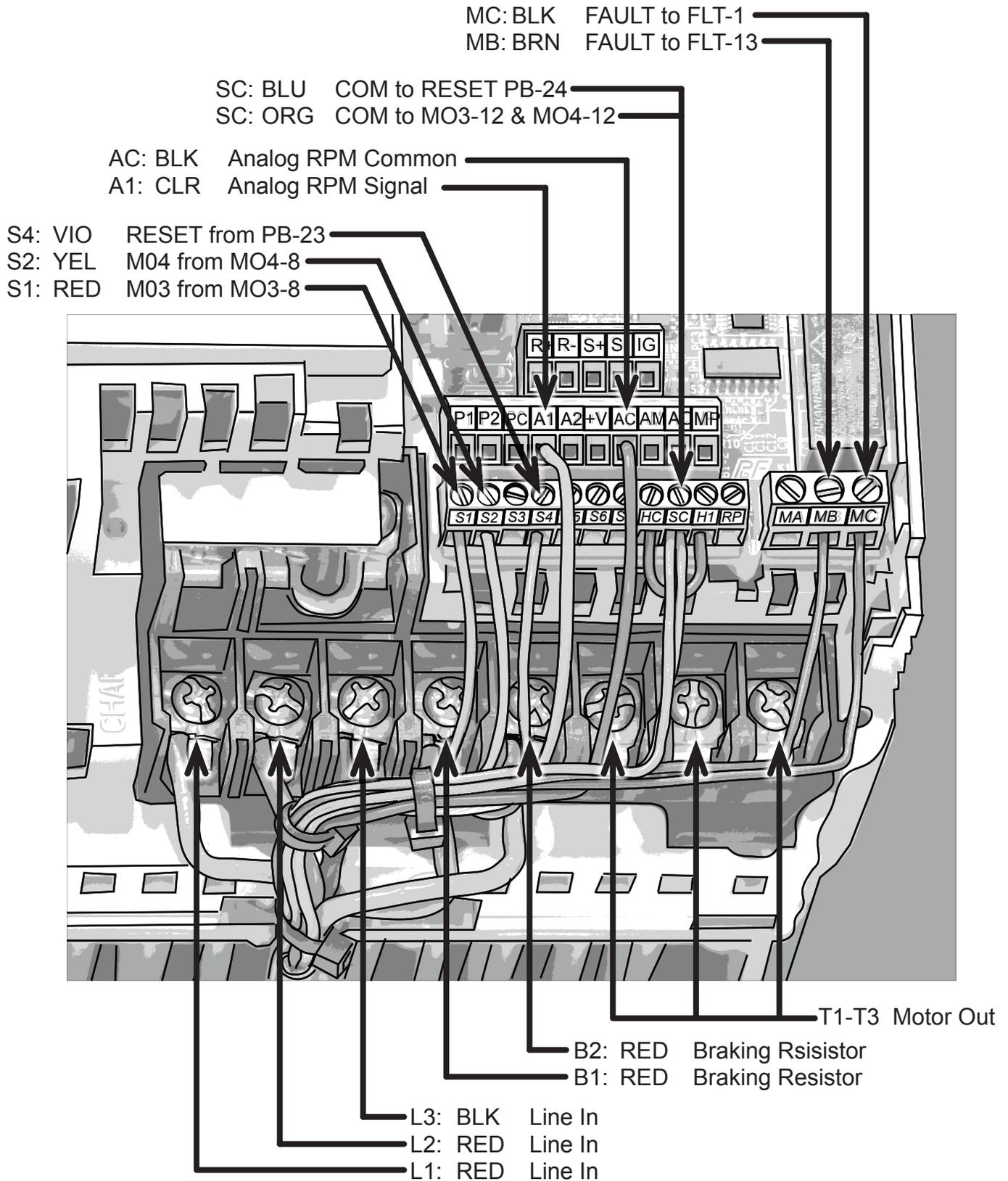
G4 CNC

OmniTurn JULY 2010
 AC SPINDLE DRIVE (V1000 5HP)

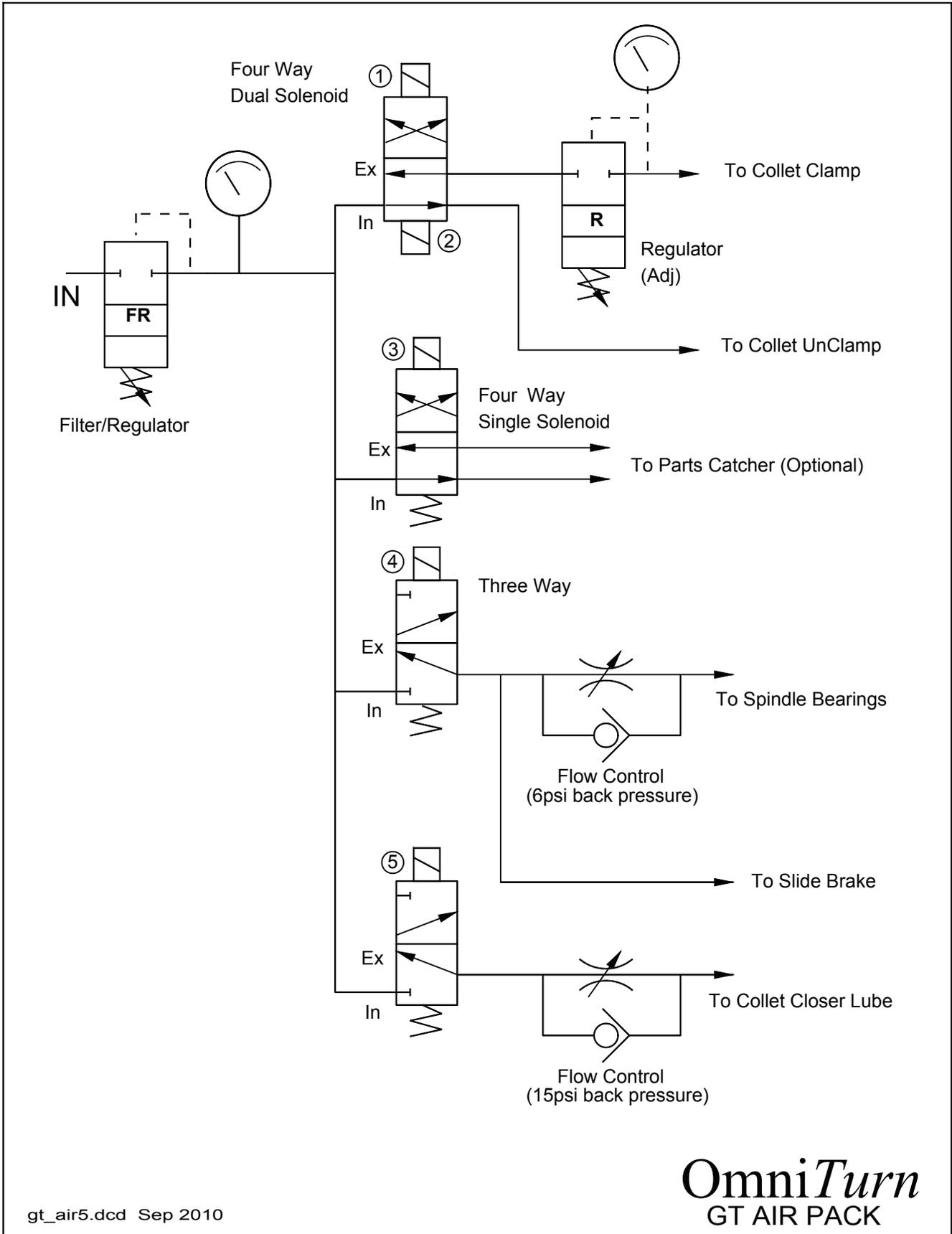
v1000_sc-G4.dcd

OmniTurn - Trouble shooting guide

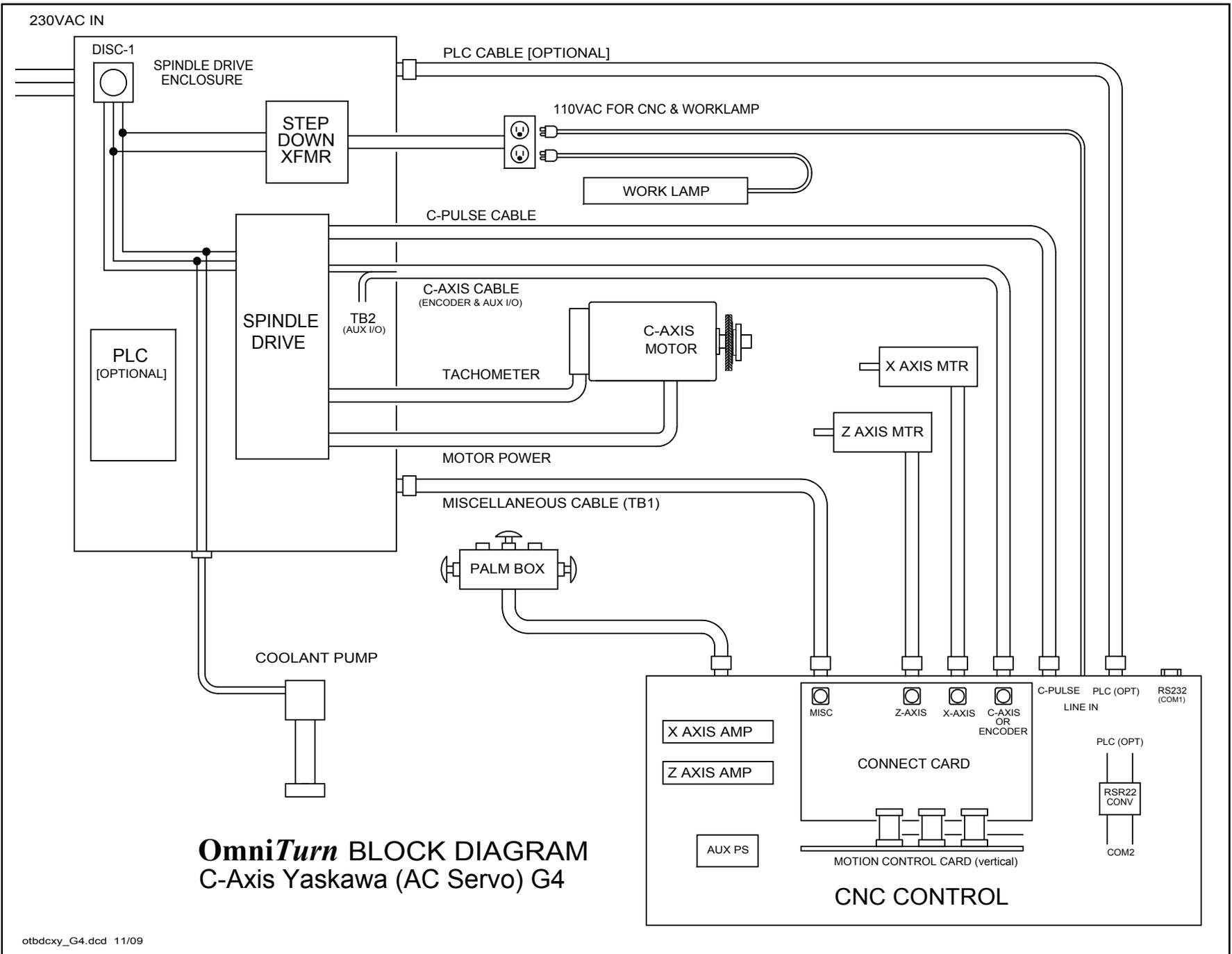
5hp Spindle Drive Wiring (Yaskawa V1000)



OmniTurn - Trouble shooting guide

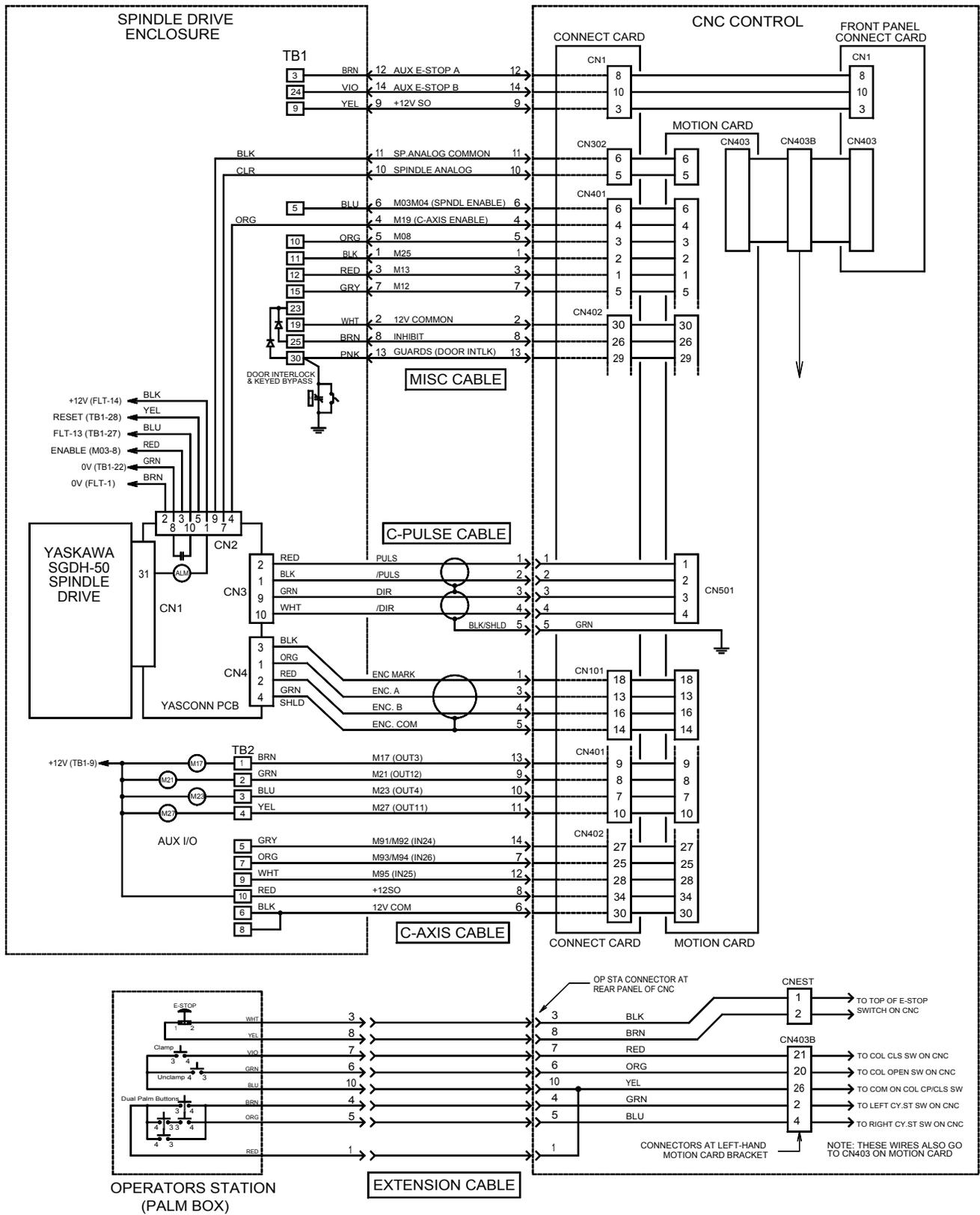


gt_air5.dcd Sep 2010



OmniTurn BLOCK DIAGRAM
C-Axis Yaskawa (AC Servo) G4

OmniTurn - Trouble shooting guide



C-AXIS DRIVE

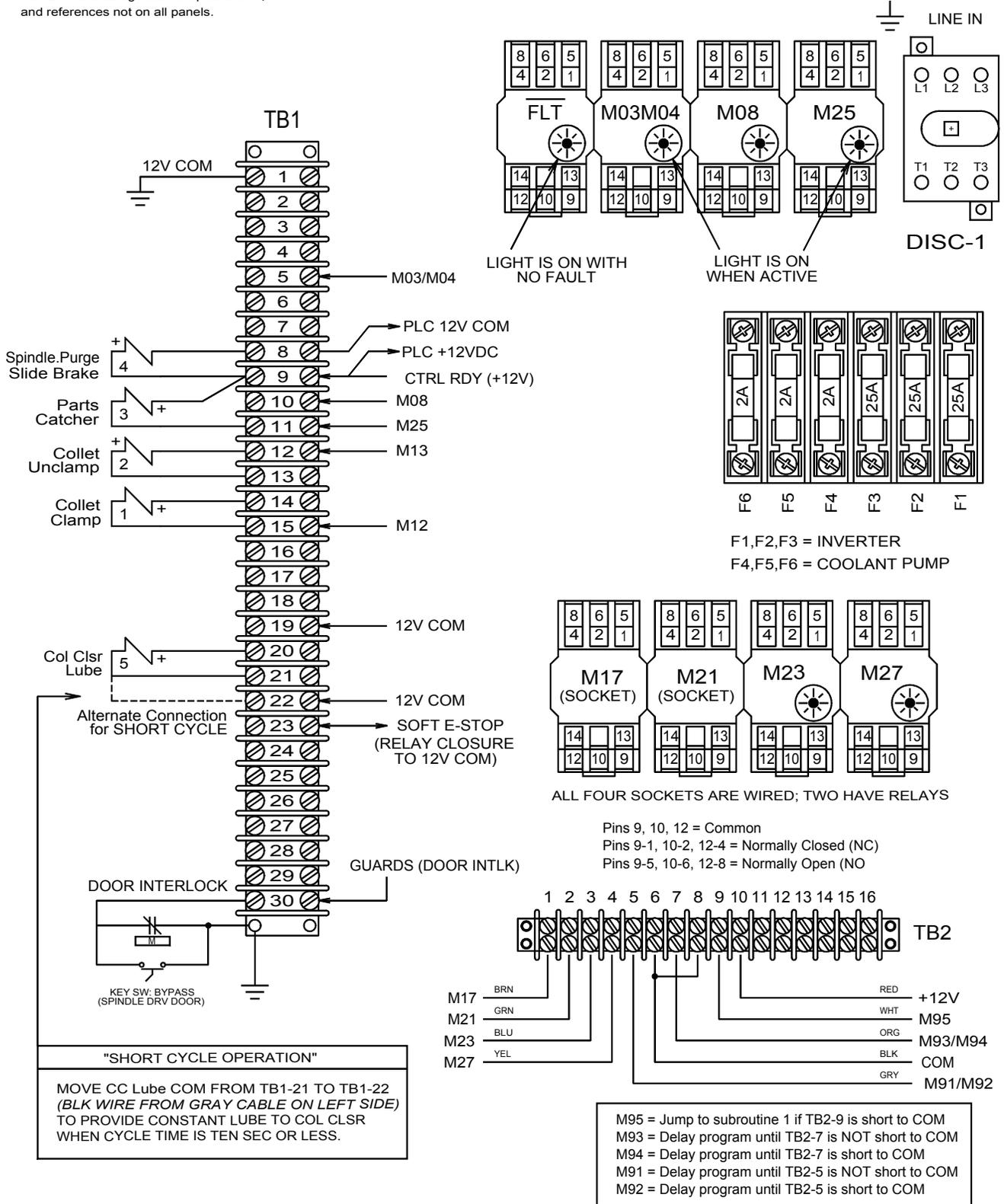
OmniTurn

MISCELLANEOUS CABLE

AND OPERATORS STATION

OmniTurn - Trouble shooting guide

NOTE: This drawing includes optional components and references not on all panels.



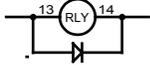
acsG4_LBL.dcd jun 2011

G4 CNC

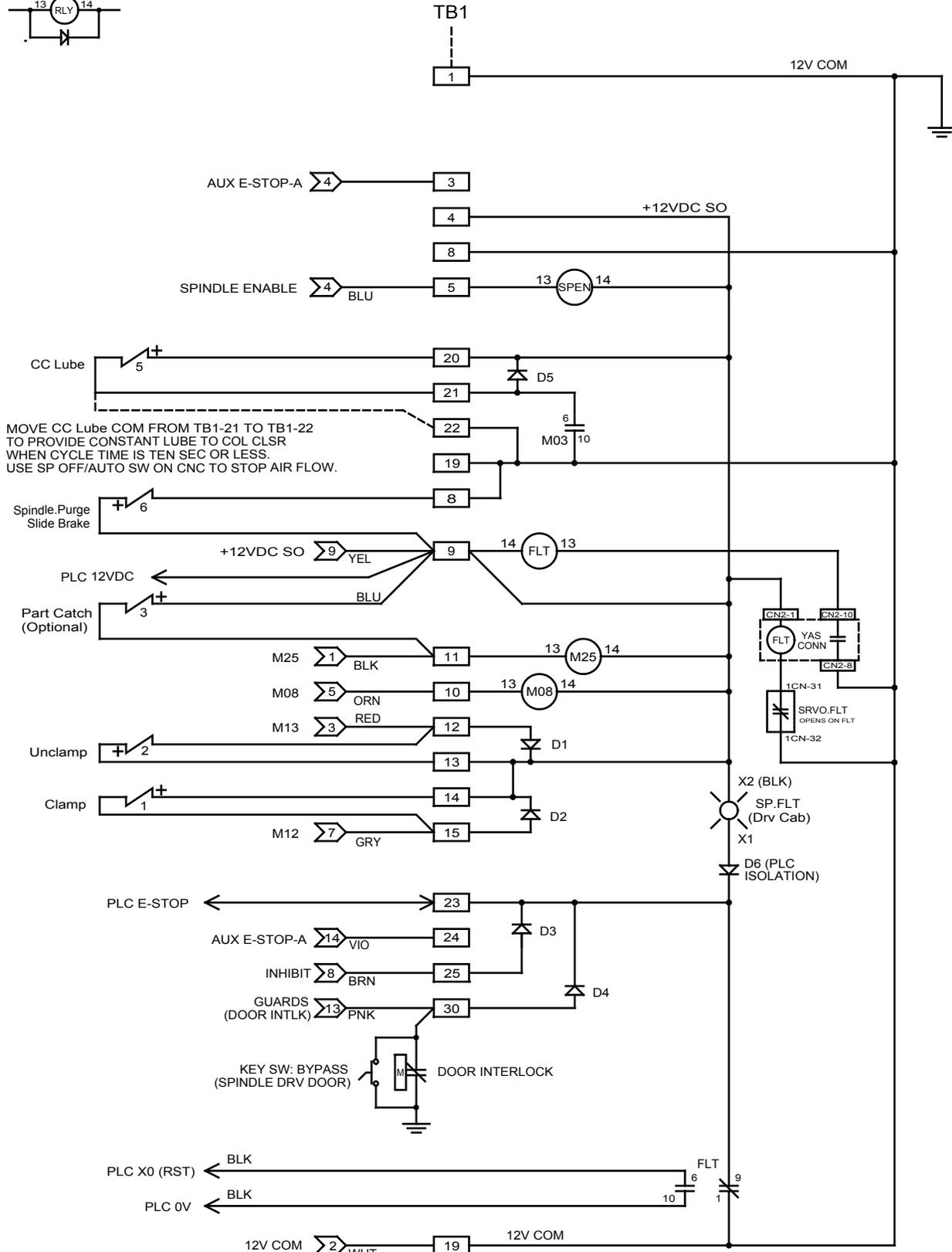
OmniTurn
C-AXIS SPINDLE DRIVE PANEL

OmniTurn - Trouble shooting guide

NOTE: All relay coils have diodes:



NOTE: This drawing includes optional components and references not on all panels.



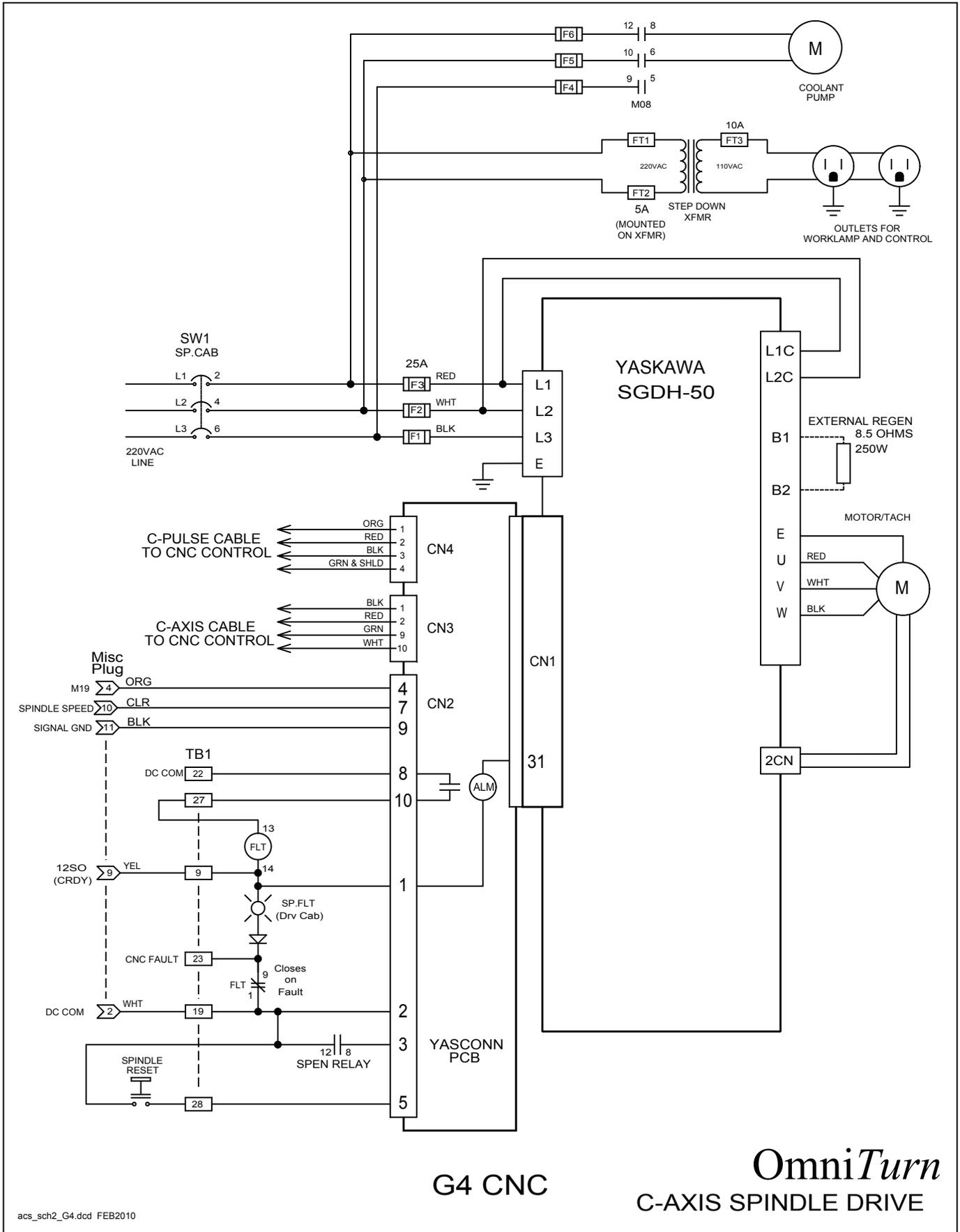
MOVE CC Lube COM FROM TB1-21 TO TB1-22 TO PROVIDE CONSTANT LUBE TO COL CLSR WHEN CYCLE TIME IS TEN SEC OR LESS. USE SP OFF/AUTO SW ON CNC TO STOP AIR FLOW.

acs_sch1-G4.dcd may 2010
COMBINED SP.PURGE/SLIDE BRAKE

G4 CNC

OmniTurn
C-AXIS SPINDLE CONTROL LOGIC

OmniTurn - Trouble shooting guide



G4 CNC

OmniTurn
C-AXIS SPINDLE DRIVE

acs_sch2_G4.dcd FEB2010

OmniTurn - Trouble shooting guide



Collet Usage and Collet Closer Maintenance

Inserting Collet:

- 1) With piston (item 2) in forward or released position, back off locking key (item 17) , enabling drawtube (item 7) to be turned by knurled collet adjusting knob (item 15).
- 2) Place collet in spindle. Turn knurled knob (item #15) clockwise until taper of collet engages with taper of spindle.
- 3) Align locking key (item 17) with adjusting slot (4 slots provided) on draw tube, engage locking key (item 17). This prevents drawtube from turning and becoming loose during operation.

How to control holding force:

- 1) By using the regulator and gage, start with air pressure of 20 lbs.; increase pressure gradually until part is held sufficiently for machining operation.

USE MINIMUM PRESSURE NECESSARY FOR HOLDING PART.

If part still slips at pressure greater than 60psi, try a serrated collet; serrations increase gripping force by 10% without increasing pressure.

- 2) The Dunham Air Actuator permits even gripping pressure although workpiece may have diameter variation. Holding force can be adjusted for any type of machining by increasing or decreasing the air pressure.

Maintenance:

- 1) Little or no maintenance is required. Bearings are self lubricating. All components such as bearings, “O” ring seals and retaining rings are standard. (See drawing #MC545 when ordering replacements.)

Installing an air cleaner and lubricator in line is advisable, but not essential.

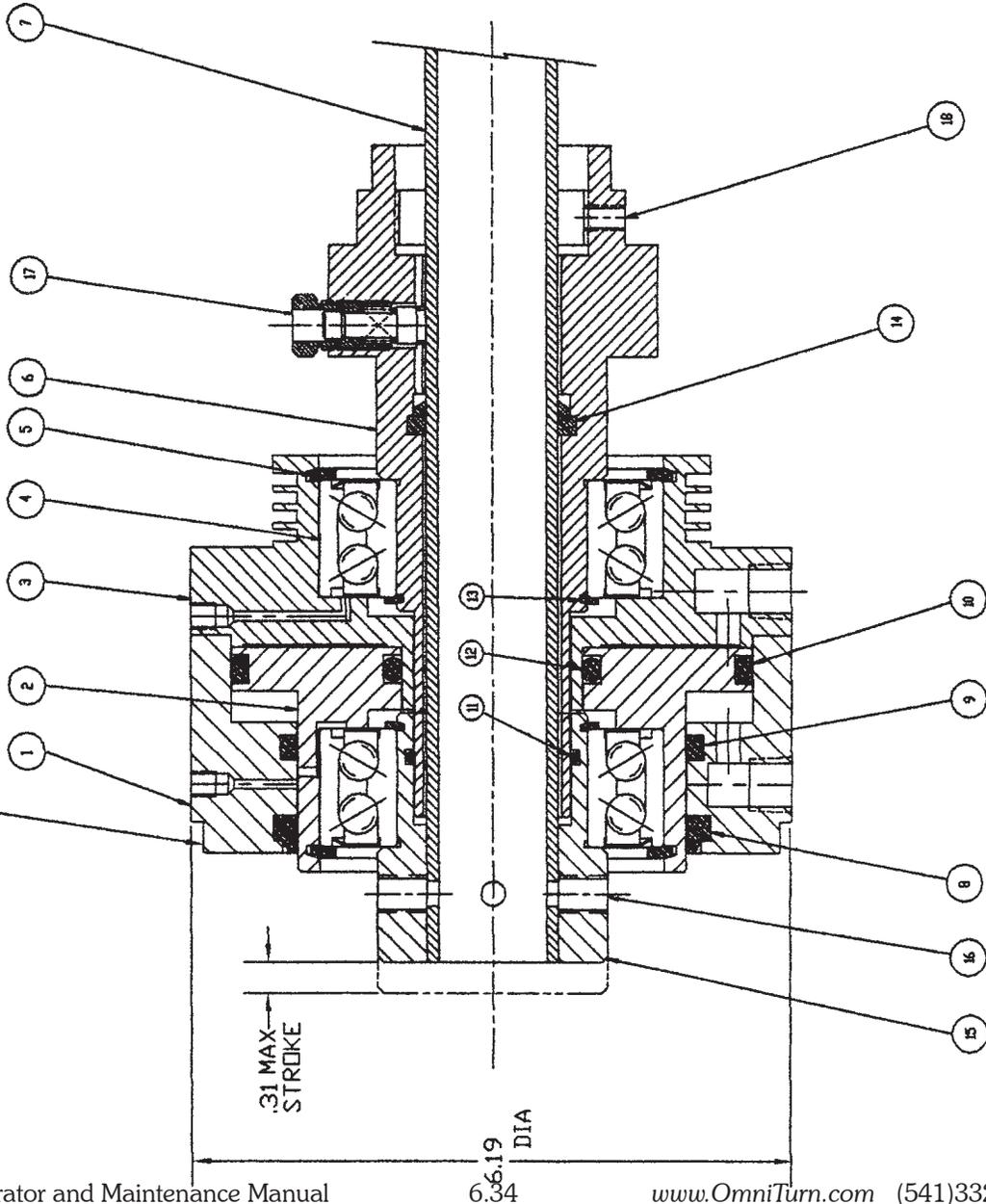
The purchaser or end user of this retrofit unit is fully responsible for the fabrication and application of special protective guarding to cover this unit's moving parts. Drawtube cutoff must be performed as instructed above. Oil must be available for the oil-mist system.

8 Parklawn Drive· Bethel· CT· 06801 • USA· 203-739-0415· FAX: 203-739-0461 www.dunhamtool.com preferred@dunhamtool.com

OmniTurn - Trouble shooting guide

REVISIONS		DATE	APPROVED
REV.	DESCRIPTION	DATE	APPROVED
4	ENGINEERING RELEASE	11/97	MG

POSITION INDICATOR HERE



ITEM	PART NO.	DESCRIPTION	QTY.
19	#10-24 X 1" LG.	SOCKET HEAD CAP SCREW	6
18	1/4-20 X 3/8 LG	BRASS POINT SOC SET SCREW	3
17	MA299-H	LOCKING KEY ASSEMBLY	1
16	3/8-16 X 5/8 LG	HALF DOG POINT SET SCR	4
15	MC544	KNURLED ADJUSTING KNOB	1
14	00602100A	DISCOGRIN SEAL	1
13	5102-196	TRU-ARC SNAP RING	2
12	D-328	"D" - RING	1
11	D-130	"D" - RING	1
10	04353	GUARD RING	1
9	04345	GUARD RING	1
8	006-044	DISCOGRIN SEAL	1
7	MB244-H6	DRAWTUBE	1
6	MC543-A	MAIN BRG MOUNT	1
5	5002-354	TRU-ARC SNAP RING	2
4	D-55510ND	BACC BEARING CONE SHIELD	2
3	MC542	MAIN BRG HOUSING	1
2	MC541	PISTON	1
1	MC540	CYLINDER	1

UNLESS OTHERWISE SPECIFIED -
 ANGLES MAY VARY
 DIMENSIONS MAY VARY
 DECIMAL DIM MAY VARY
 FRACTIONAL DIM MAY VARY
 ROUND EDGES
 CONCORD TO HAVE FILLETS
 SURFACE FINISH

THE BUNHAM TOOL COMPANY
 REV. 11/97
 AIR COLLET ACTUATOR - OIL MIST
 NO. ELECTRONICS
 DATE 11/97/MAIL
 CHK. DATE NEXT ASSN
 SCALE 1:1
 DIMS MG
 MC586
 REV 4

OmniTurn - Trouble shooting guide

Collet Closer Installation and Maintenance

MODEL #DACCOM-NC for OMNITURN AIR POWERED ROTARY ACTUATOR (REF. DWG. MCS86)

Maximum pull force: 17911bs. @ 90 PSI (19.9 X line pressure).
Maximum push force: 1458 lbs. @ 90 PSI (16.2 X line pressure).
Maximum stroke : 5/16"
Maximum RPM : 4500 (continuous)
6000 (intermittent)
Largest diameter : 6.18"
Length : 7.06" open (without spindle adapter)
6.75" closed
Weight : 14 lbs 11 oz. (without spindle adapter)
Through hole : 1.06"

Compressed air requirements:

Actuation: Variable 15 to 90 P.S.I.; clean, dry air required.

USE MINIMUM PRESSURE NECESSARY FOR HOLDING PART

Cooling : 15 P.S.I.; clean, lubricated

Description:

This Dunham air powered rotary actuator operates at up to 4500 RPM, even when powered with maximum air pressure. Special consideration has been given to enhance its durability; this includes hardened wear surfaces, complete sealing of rotating components, and urethane wipers. The bearings are shielded, positive air pressure moves throughout the actuator, and the drawtube mount acts as a seal.

The air actuator is designed to mount to rotating devices, most commonly spindles. The unit will provide a push or pull force capable of opening or closing collets, fixtures, and short stroke power chucks.

Installation: (reference Assembly Drawing MC586)

1) Spindle Mounting Adapter (Main Bearing Mount):

Spindle Adapter (item 6) does not need to be further machined to fit spindle end. This special mounting adapter has been specifically designed and machined for the OmniTurn Lathe so that it threads directly onto the back end of the Omniturn spindle. The actuator mounting surface of the adapter must measure at least .001" T.I.R. perpendicular with the spindle when mounted.

2) Secure actuator assembly by tightening set screws onto the spindle. Attach air lines to ports on actuator. Insert collet into spindle; power the actuator to its forward position.

8 Parklawn Drive · Bethel · CT · 06801 · USA · 203-739-0415 · FAX: 203-739-0461 www.dunhamtool.com preferred@dunhamtool.com

OmniTurn - Trouble shooting guide

Collet Closer Installation and Maintenance

Insert drawtube (item 7), through the actuator, with its threaded end toward the front of the spindle. Be certain that drawtube lock (item 17) is retracted to allow insertion of drawtube. Rotate drawtube to fully engage the drawtube threads with collet threads; back off 1-1/2 revolutions. Measure the remaining length extending from the knurled knob, if any. Remove the drawtube to cut off excess length. The drawtube must be flush with the knurled knob (item 15) when later reinstalled.

3) Fastening Drawtube to Actuator:

The drawtube must have four, equally spaced, 1/4" holes drilled 11/16" from the draw tube end. These holes must be drilled within .003" of drawtube centerline.

The holes must be chamfered burr free. The drawtube is now reinstalled into the air actuator. Tighten the four 3/8"-16 socket head set screws (item 16) so their half dog points engage the newly drilled 1/4" holes. The screws must then be loosened 1/8 turn. Drawtube installation is now complete.

4) Adjusting Actuator Concentricity:

Adjust three brass point set screws (item 18) to general tightness; please note that this mounting adapter has been designed for ease of mounting and alignment with its ground front face, bore face and bore diameter. Reduce air pressure supplied to forward port of actuator to 15 P.S.I. Using a dial indicator reach to indicator point identified on drawing MC586. Rotate spindle slowly to measure the rise and fall of this portion of the actuator. Adjust concentricity with the use of the three set screws (item 18) lightly. If actuator is in good alignment with the spindle, the indicator should measure .0005"-.0010" concentricity. Tighten three set screws to spindle securely.

5) Attached Oil-Mist Lubrication:

To operate actuator to full potential, attaching the oil-mist lubricator assembly is required. The unit supplied must be mounted close to actuator while providing easy access to the oil reservoir. Attach oil-mist to ports labeled on drawing #MC545. A shut off valve MUST be used to turn off air supplied to oil-mist assembly when not in use.

FAILURE TO USE THIS LUBRICATION MAY RESULT IN UNIT FAILURE. The air regulator should be adjusted to 15 P .5.1. The Norgren "Type L07 Lubricator" should be adjusted to supply 1 drop every 40 seconds. The reservoir should be replenished with Mobil "DTE" light (or equivalent) as required. A small amount of grease is displaced out of the bearings by the rotation and the oil-mist lubricator; this should be considered normal.

FAILURE TO COMPLETE THE PROPER INSTALLATION AS INSTRUCTED ABOVE CAN RESULT IN OPERATOR INJURY OR PREMATURE WEAR ON THE UNIT.

8 Parklawn Drive • Bethel, CT • 06801 • USA • 203-739-0415 • FAX: 203-739-0461 www.dunhamtool.com preferred@dunhamtool.com

OmniTurn - Trouble shooting guide

L606 Lubricator

Installation

1. The lubricator should be installed with reasonable accessibility for service whenever possible - repair service kits are available. Keep pipe or tubing lengths to a minimum with inside clean and free of dirt and chips. Pipe joint compound should be used sparingly and applied only to the male pipe - never into the female port. Do not use PTFE tape to seal pipe joints - pieces have a tendency to break off and lodge inside the unit, possibly causing malfunction. Also, new pipe or hose should be installed between the lubricator and equipment being lubricated.
2. The upstream pipe work must be clear of accumulated dirt and liquids.
3. Select a lubricator location as close as possible to the equipment being lubricated and downstream of any pressure regulator.
4. Install lubricator so that air flows in the direction of arrow on body.
5. Install lubricator vertically with bowl drain mechanism (if supplied) at the bottom.

Operation and Service

1. Filling - Lubricators can be filled while under pressure and without shutting down equipment. Slowly remove either fill plug and fill to 1/4" to top of bowl using correct oil. For proper automatic fill operation, the oil inlet pressure to lubricator must be maintained between 10 and 200 PSI above air pressure to lubricator.

Suggested Lubricant: F442

Petroleum based oil of 100 to 200 SSU viscosity at 100°F and an aniline point greater than 200°F. (Mobil DTE24 and Sun Company Sunvis 932 are good examples). Do not use oils with adhesives, compound oils containing solvents, graphite, detergents or synthetic oils.

2. Replace the Fill Plug (by turning clockwise) and seat firmly. Excessive torque is not required. Turn on air supply, if leakage occurs, **DO NOT OPERATE** - conduct repairs again. The lubricator is now ready for setting.

3. Oil Delivery Adjustment - To adjust oil delivery, turn Adjustment Knob on top of the lubricator.

Leaner - Clockwise

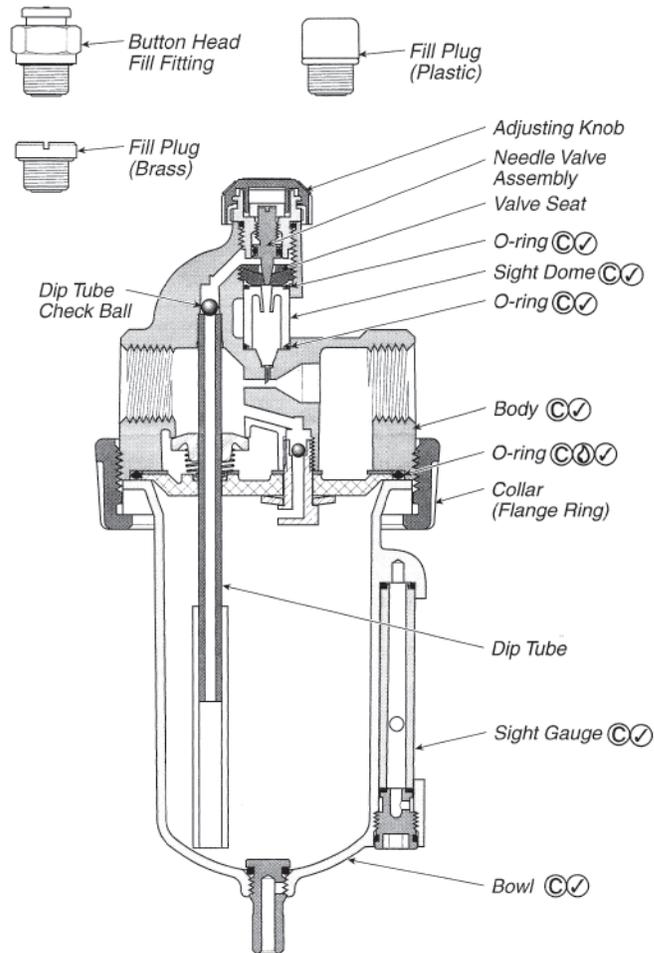
Richer - Counterclockwise

By counting the number of drops per minute in the Sight Dome, you can adjust to your requirements. Generally, one drop per minute downstream for every 10 - 15 SCFM flow is satisfactory. 25 drops per minute equals one (1) ounce per hour - volume of oil passing through the Sight Dome.

NOTE: This is a constant density type lubricator which delivers a constant ratio of oil air flow. Therefore, if air flow increases or decreases, oil delivery will be adjusted proportionately. **ONLY IF A DIFFERENT RATIO IS DESIRED SHOULD YOUR ADJUSTMENT KNOB SETTING BE CHANGED AFTER YOUR INITIAL SETTING.**

4. Cleaning - Erratic lubricator operation or loss of lubrication is almost always due to dirt (rust, pipe tape, etc.) in the needle valve or venturi area. To clean, shut off and vent all air line pressure to the unit being cleaned. In most cases cleaning is needed only in the oil metering area. Pull off Adjusting Knob and remove Needle Valve Assembly by turning out large hex nut. Remove Needle Valve Seat and clean removed parts with alcohol making sure hole in seat is clear. With a #57 drill, make sure hole in bottom of sight gauge area is open. Remove Bowl. Clean parts with soapy water or denatured alcohol **but do not use denatured alcohol on plastic bowl, sight dome or sight gauge.** If using compressed air to blow dry, be sure to wear appropriate eye protection.

5. After servicing, apply system pressure and check for air leaks. If leakage occurs, **Do Not Operate** - conduct servicing again.



- Lightly grease with provided lubricant.
- Inspect for nicks, scratches, and surface imperfections. If present, reduced service life is probable and future replacement should be planned.
- Clean with lint-free cloth.

Bowl			
Polycarbonate	BK606Y	B	1/4", 3/8"
Zinc with Sight Gauge	BK605WY	W	1/4", 3/8"
Polycarbonate	BK606A	B	1/2"
Aluminum	BK603A	E	1/2"
Zinc with Sight Gauge	BK605WA	W	1/2"
Aluminum with Sight Gauge	BK606X30A	G	1/2"
Aluminum	BK603B	E	3/4" thru 1-1/2"
Zinc with Sight Gauge	BK6D5WB	W	3/4" thru 1-1/2"
Aluminum with Sight Gauge	BK606X30B	G	3/4" thru 1-1/2"
Repair Kit			
Dip Tube Replacement Kit	DTK606	AI	All Sizes
Needle Valve Assembly	RK606Y	All	All Sizes
Sight Dome Repair Kit	RK606SY	All	All Sizes
Sight Gauge Bowl Repair Kit	RBK605WY	W	11/4", 3/8"
Sight Gauge Bowl Repair Kit	RKB605WA	W	1/2"
Sight Gauge Bowl Repair Kit	RKB606X30A	G	1/2"
Sight Gauge Bowl Repair Kit	RKB606WB	W	3/4" thru 1-1/2"
Sight Gauge Bowl Repair Kit	RKB606X30B	G	3/4" thru 1-1/2"
Button Head Fill Fitting (3/4 Hex.)	SAA606C109-1	-	-
Button Head Fill Fitting (11/16 Hex.)	L606C14	-	-
Fill Plug (Brass)	SA606B4	-	-
Fill Plug (Plastic)	SAP04113	-	-

OmniTurn - Trouble shooting guide

OmniTurn GT75 & GTJr

MAINTENANCE & LUBRICATION SCHEDULE

NOTE: Preventative maintenance reminders will periodically appear on the control monitor to prompt the operator of various lubrication requirements. The schedule below lists these requirements in greater detail.

DAILY

1. Check main air pressure: 30psi minimum for stand-alone GT, depending on collet pressure requirement. If any OmniTurn loader is installed, then 90psi.
2. Check collet closer lubricator (mounted on bulkhead in spindle motor compartment). With spindle running observe the sight glass for oil flow. If necessary, adjust oil flow to one drop per 30 seconds. Use Mobile Light DTE oil or equivalent.
3. Remove any heavy chip build up from guards and way covers.

WEEKLY

1. Check air regulator/filter
 - Drain main regulator bowl (loosen plug CW)
 - Drain coalescing filter bowl (loosen plug CW)

MONTHLY

1. Examine main regulator element; replace with Watts EKF31 if it shows serious contamination.
2. Examine coalescing filter element; replace with Watts EK504VY if it looks oil-coated.

EVERY FOUR TO SIX MONTHS (Water-Based Coolant)

Lubricate the ways and ball screws.

NOTE: If CUTTING OIL is used, no lubrication is necessary, ever.

The linear guide manufacturer recommends charging the bearing blocks with lithium grease every 4,000,000 inches of travel. On the Z-axis, this is about 96,000 parts with two inches of thread done in 8 passes.

LUBRICATING the WAYS and BALL SCREWS

There is a fitting on each of the eight bearing blocks. Three pieces of sheet metal and the accordion bellows must be removed to access the fittings and ball screws. The accordion bellows is removed by lifting each end off its holder, exposing the spindle-end Z-axis fittings. The other Z-axis fittings are exposed by removing the wide piece of sheet metal with the X-axis scale affixed. Three screws through the X-axis scale hold this in place. The X-axis covers are held in place by four long all-thread rods and acorn nuts, two on each end of the tooling plate. See page 6.7 for illustration.

With the sheet metal removed, thoroughly grease each bearing block with lithium grease. A “needle tip” is required to fit the fitting on the blocks. Tips are available from Napa (pn 715-1215) or MSC (pn 48527378).

The ball screws should be lubricated with a drizzle of Slick-50 over the entire exposed length.