⚠️ Safety

1 Startup

2 Programming

3 Tool Offsets

4 Sample Part

5 Automatic Mode

6 Troubleshooting

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 or gloves 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.

OmniTurn Safety

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.

- Check door interlock operation before operating the machine: With servos ON, slide the door open and verify that the red Motion Stop lamp lights. If not, 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 recom.mended to reduce the risks of sight damage and hearing loss. Do not wear loose clothing or gloves.

- Do not operate the machine unless the doors are closed and the door inter.locks 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; do not wear loose clothing or gloves.**

- **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.

Before starting cycle – Check door interlock operation: With servos ON, slide the door open and verify that the red Motion Stop lamp lights.

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.

OmniTurn Safety

Practical Safety Advice for All Owners & Operators

Safety violations will continue to be a top priority for OSHA. This is especially true for companies that have had a previous accident. This is a huge deal because each year over 18,000 American workers are injured while using or doing maintenance on machinery. What kind of injuries? Amputations (cut off) of hand and fingers, crushed hands and fingers, abrasions, gouges, cuts, etc. Plus over 800 deaths! And these numbers have stopped declining. Why is this? Well, one reason is safety guarding. In 2016, 88 percent of machinery guarding citations were called "serious" and where the employer should have know that there was a substantial possibility of a worker being injured or killed. Machinery guarding standards are outlined by OSHA and ANSI (American National Standards Institute). They currently believe that half of all machine shops and fab shops currently have dangerously inadequate machinery guarding. Is your shop one of them?

Avoid losing an employee and avoid getting sued for huge sums of $! If an employee is injured at your shop, they might never return to work and they and their family may be entitled to financial compensation.

Nine things you must do:

1. Check all your machines at least once a year to make sure that each one still has its original safety guards and emergency stop buttons that still function.
2. Find and replace all missing safety guards. If necessary, buy or fabricate a new one.
3. OSHA's number one machine safeguarding violation is for grinding machines. The grinding wheel constantly needs to be checked for cracks to avoid anyone being injured when a cracked grinding wheel shatters, which usually happens shortly after the grinder is turned on. The shield, the cover, the resting plate are all important devices that each serve a purpose to allow safe use of the grinding machine.
4. Make sure that older machines have automatic shutoff just like the newer machines. This is very important with machines for sheet metal rolling, plate rolling, tube bending and structural shape rolling. This means that when you step on the foot pedal the machine goes but when you release your foot the machine stops.
5. Replace old mechanical press brake machines and ironworker machines with new model hydraulic powered machines that are designed to automatically reverse back up once the foot pedal has been released. Old mechanical type machines that have a flywheel are nearly impossible to safely stop once the downstroke has been started.
6. Install safety light curtain guarding systems on each press brake machine that does not already have them.
7. Install your own custom made safety guards to protect operators that will be doing "repeat work" in which the path of the metal can be left open while the sides, top and bottom areas around the path of the metalwork can be shielded.
8. Also pay attention to preventing unexpected injuries from forklifts, cranes, hoists and other possibilities that are specific to your operation.
9. If your company has many employees, buy or rent a safety video that has re-enactments of actual common industrial accidents and have your shop employees watch the video and then have a discussion session afterwards.

Remember:

* Older machines are not exempt from OSHA requirements and may need their safety guarding to be updated.
* Not all new machines are safe to use for the purpose you intend and may need additional guarding added.
* Visit the OSHA Machinery Guarding Webpage to learn more about your responsibilities and how to follow through on those responsibilities.

OmniTurn Safety

Metalworking machines can be very dangerous, which means they may cause injury or death if used carelessly or incorrectly: Buyers shall require employees to use all safety devices, guards and proper safe operating procedures as set forth in manuals and instruction sheets originally furnished by the manufacturer. Buyer shall not remove or modify any such device, guard or sign. It is Buyer's responsibility to provide all the means that may be necessary to effectively protect all their employees from serious bodily injury which otherwise may result from the method of particular use, operation, set-up or service of the equipment. It is Buyer's responsibility to provide to provide proper training to each employee using the machine or to use only employees that are sufficiently experienced in the use of such machine. It is the responsibility of Buyer to comply with any and all national and local codes including OSHA. If Buyer fails to comply with such provisions of this paragraph or the applicable standards or regulations aforementioned, Buyer shall indemnify and save NC ELECTRONICS & OMNITURN harmless from and against any and all claims, losses or damages arising there from. Those previous two sentences are very important to understand.

Because it is impossible for machinery suppliers to predict exactly how you are going to use the machine over the years, it is impossible for machinery suppliers to guard against every danger without ruining useful machine productivity, so you need to be responsible and take this seriously. OSHA reports that machinery users suffer 18,000 amputations, lacerations, crushing injuries, and abrasions. OSHA also reports over 800 machinery users are killed per year. Only YOU can prevent accidents.

Common Sense: Now that you have read all of the above, it is time to think about this from a practical standpoint. There is no substitiute for using common sense. No amount of time saving is worth the risk of cutting off a finger or worse. And that is what will eventually happen if you take dangerous shortcuts. Don't remove the safety guards before using the machine. If you do take one off for some reason, put it back before you use the machine. Don't let distractions, laziness, being in a hurry, or your ego, get you injured.

Be smart: Always be on the lookout for possible safety hazards. Watch out for pinch points where your finger could get crushed or severed. It is difficult to go through life without a finger. It is hard to turn handles, play sports, type, etc. Women will think it looks gross. Watch out for moving parts. You dont want to be too close if you lose your balance or bump into something that pulls you in. Keep hands, fingers, hair and loose clothing away from moving parts.

Knowledge: Please visit the OSHA Machinery Safety website (https://www.osha.gov/Publications/Mach_SafeGuard/chapt1.html). It is worth the time to brush up on the safety precautions OSHA expects from every machine owner. If you dont know how to use a machine you purchased, hire someone who does. If you can't to afford to hire someone to use the machine for you, then temporarily hire someone to train you on how to properly use the machine. Don't try to learn by yourself. It is best for machinery users to go through a Machine Tool Apprenticeship.

Safety Guards: Safety devices such as Light Curtains are strongly recommended to prevent fingers and hands from getting crushed especially for predictable repeat usesf you buy a Brake Press machine, OSHA wants you to install a light curtain guarding system on it. If you are going to use a machine for a repeat operation, OSHA prefers that you use dual palm buttons if they can be reasonably accommodated, OSHA also wants you to figure out extra safety guards to reduce the danger from pinch points of the machine that your repeat part does not pass thru. This can take the form of a plexiglass/Lexan guard, metal mesh guard or electric light curtain. Visit the OSHA website for more info.

Remember: People have died or had hands cut off from past mistakes that happen in the blink of an eye. Don't let that happen to you, your co-workers or your employees.

Safety Labels on OmniTurn Machines

⚠ WARNING

Moving parts can crush and cut.

Set E-Stop *before* putting hands into cutting area!

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.

⚠ WARNING

Belt and pulleys can crush and cut.

Do not operate with any hatch or panel open.

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.

⚠ WARNING

Impact Hazard.

Unsupported bar can whip with deadly results.

Never allow unsupported stock to extend past drawtube end.

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.

Omni*Turn* 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

OmniTurn - Startup: GT-75 Technical Specifications

GENERAL SPECIFICATIONS:

Travel in X:12"
Travel in Z:9"
Tooling Height:1/2" (2" optional)
Tooling Plate Size:Gang type (quick change)
Repeatability of Slide Travel:.00005" (50 millionths)
Resolution of Slide Travel:.00005" (50 millionths)
Accuracy of Slide Travel:.0001" (one tenth)
CNC Control:OmniTurn PC Based
Rapid Travel:300ipm
Feed Rate:.00005ipr to 300ipm
Threading:240tpi to unlimited (useful for long-lead multi-start)
Thrust:500lb continuous; 750lb intermittent
Axis Motors:Closed Loop Servo with velocity feedback
Acceleration:.04sec 0 to 300ipm

CONSTRUCTION:

Linear Ways:Precision Linear Guides
Ball Screws:Precision ground, Zero Backlash
Base Billet:MJ7 Composite Casting
Headstock:Cast Iron
Frame Finish:Heavy Duty Powder Coat
Coolant Pump:1/8hp (1/4hp optional)

SPINDLE:

Cartridge Type:Class 7 Angular Contact
Spindle Taper, ID:5C, .000025" TIR
Spindle Taper, OD:4° Cam Lock
Collet Actuator:Pneumatic

SPINDLE DRIVE, STANDARD:

Spindle RPM:100 - 4000rpm (5000 rpm optional)
Power:5hp
RPM Programmable:RPM and Constant Surface Feed

SPINDLE DRIVE, C-AXIS:

Spindle RPM:0 - 4000rpm (5000rpm optional)
Power:5hp
RPM Programmable:RPM, CSF, C (abs), C (incr), C/X, C/Z, C/X/Z
Spindle Positioning Resolution:.02°

REQUIREMENTS:

Electrical:208 - 240 VAC 1 or 3 phase 20A
Air:Compressed Air 3.0 cfm @ 60 - 100 psi
Floor Space:34" x 66 " (75" high)
Machine Weight:1650lbs

OmniTurn - Startup:GT-75 Initial Setup

REMOVE CHIP PAN
The chip pan is usually shipped in the cutting area of the GT-75. It holds the coolant pump, leveling feet, and any special tooling or accessories.

REMOVE 4x4's
Replace the 4x4's with the leveling feet (or optional bolt-down feet). Absolute leveling is not necessary; adjust the feet so that the machine is stable on your floor.

INSTALL COOLANT SYSTEM
Remove white cardboard box behind the lower front doors (CNC control & user's manual).
The coolant tank is shipped bolted to the right side of the frame with two bolts; the other two bolts are threaded into mounting holes in cross member. Remove these bolts and unbolt the coolant tank. Set coolant tank on cross members with pump chamber on left side; bolt securely to cross members. CNC mounting wedge is shipped in coolant tank; set this aside for now.
The coolant pump flange holds coolant pump to tank w/o hardware. Set pump atop pump chamber. Connect coolant hose to pump; tighten clamp securely.
Plug coolant pump power cord into outlet on spindle drive cabinet (blue cabinet at left).

INSTALL CNC CONTROL
Remove CNC control and OmniTurn User's Manual from white cardboard box. *USB stick, door-interlock bypass key, and any technical documents specific to your machine, are added to the front of the User's Manual.* Place CNC on table upside-down. Remove four 1/4-20 SHCS from bottom of Control. Mount wedge to control. Attach wedge and control to GT75 using 3/8-16 SHCS. Control can be rotated if desired, by using only one bolt. Cut zip-ties holding cables and connect cables to control: ***insure that labels on cable plugs agree with labels on control sockets.*** Plug CNC power cord and work lamp plug box into outlets in box at top right-rear of GT75. Locate USB stick inside front-cover of OmniTurn Manual, and install in socket on front panel at right of monitor screen. This is the parts program backup and emergency hard drive restore.

CONNECT AIR
*The GT75 has a pneumatic brake on the X axis slide: **do not move slide without air!***
Connect dry air to 1/4NPT port on regulator input. The regulator is set for 90 psi. When servos are on there is positive air pressure on spindle bearings to prevent coolant migration.

CONNECT POWER
Route 220VAC line to junction-box at rear of spindle cabinet on GT-Jr. Connect three-phase power to the black, white and red wires. Phasing is not necessary. For single-phase operation (not available for C-Axis) connect hot wires to the black and white wires. Cap or tape the end of the red wire. Connect ground wire to green ground screw in box. Power for CNC is supplied by transformer in spindle cabinet.

TURN-ON
Insure that E-Stop switches on CNC Control and Dual Palm-Box are disengaged. Set Spindle Drive Cabinet disconnect switch to ON. Press Control On pushbutton on Control panel. Switch will light and computer will begin boot-up sequence. When boot-up is complete, computer will prompt "Please backup your program files (Y/N)". Always Press "Y" to backup any files which were edited. You may see some test files copied this time, or no files at all. Press any key to continue. The next screen will prompt "Servos off or Ext. Halt detected-". Press the blue Servos On button.

DOOR INTERLOCK and BYPASS KEY
To insure operator safety, the sliding front door is interlocked to the CNC. The sliding door must be closed for Cycle Start to function. If the door is opened while program is running, spindle will stop and Feed Hold will be active. To continue, close door and press Cycle Start switches.

PHASING
Phasing is not necessary unless equipped with **CONVEYOR** or **3-PHASE LIVE-TOOL** options. Adjust phasing for these options at line input; multiple options have been phase-matched at factory before shipping.

BYPASS KEY
The interlock bypass key-switch is located on the Spindle Drive Cabinet door at left side of machine. A pair of keys are inside front-cover of User's Manual. These keys should be in the possession of supervisory personnel specially trained and cognizant of the potential hazards involved with working on machinery that is powered up and/or in cycle.

Your GT-75 is now initialized and ready to Jog. Refer to your Manual for detailed instructions to jog and establish Home.

OmniTurn GT-75 Major Components

CNC Control

Cutting Area
*(see next page
for details)*

Spindle Motor

Collet Closer
Lubricator

Omni *Turn* GT-75

Sliding Door
(Interlocked)

Collet Closer

Palm Box
(Two-hand Cycle-Start &
Manual Collet open/close)

Collet Closer Air
Pressure

Chip Pan

Coolant Pump

Air Pack
*(see next page
for details)*

Coolant Tank

Main Air
Regulator/
Filter

Spindle Drive
Cabinet

(Door removed for clarity)

Leveling
Mounts

Door Interlock

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.
Check door interlock operation before operating the machine: With servos ON, slide the door open and verify that the red Motion Stop lamp lights. If not, contact your shop supervisor.

OmniTurn GT-75 Cutting Area

Loc-Line coolant hose w/flow control adjustment

X-Axis Motor

X-Axis Scale

Z-Axis Motor

Z-Axis Scale

Spindle
(5C w/5° cam lock)

Tooling Plate

OmniTurn GT-75 Air Pack

Unclamp (2)

Clamp (1)

Collet Closer Pressure Gauge and Adjustment
Set pressure while holding part. Use minimum pressure required to securely hold part.

Spindle Purge (4)

Collet Closer Lube Flow Ctrl (5)
(Factory Set @ 15psi)

Spindle Purge Flow Ctrl (4)
(Factory Set @ 6psi)

X Axis Slide Brake (4)

Collet Closer Lube (5)

'Clamp' Solenoid (1)

Main Air Regulator: Set to 90psi

'UnClamp' Solenoid (2)

Aux Air Conn (1/4 NPT/F)

⑤ ④ ②

Door Interlock Cable

Shop Air In (1/4 NPT/F)

Manual Drain

OmniTurn Front Panel: Knobs & Switches

See Page 1.8 for description of these controls.

See Page 1.8 for description of these controls.

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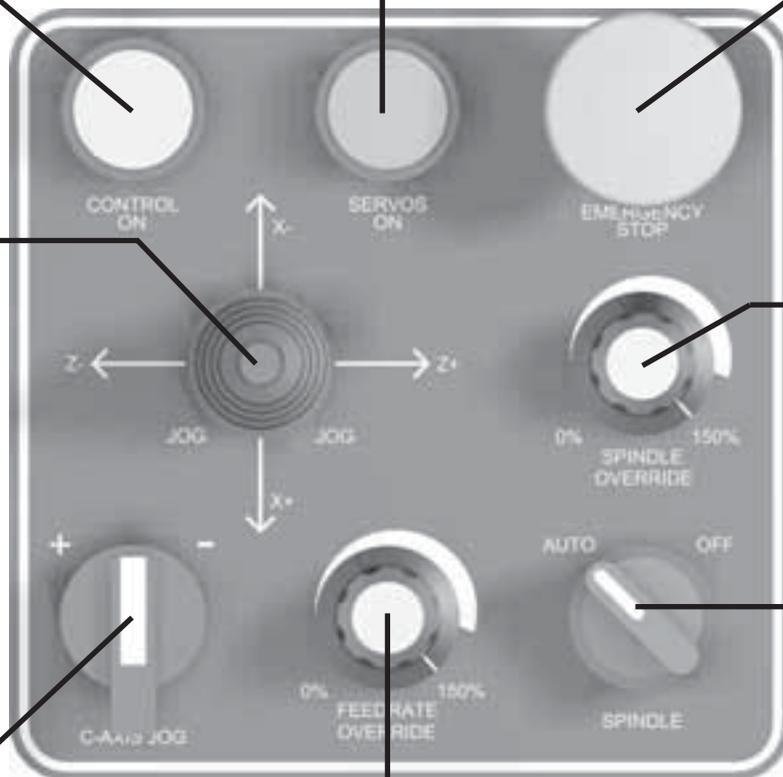
Jogs the spindle (C-Axis option only) according to setting on jog-screen (slow, medium, fast or .01° to 90° per step).

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

USB Stick for program storage. ***USB must be installed before power-on.*** *Do not remove USB while powered up.*

OmniTurn Keyboard

Number Keys set Jog speed and Jog increment in Jog Mode.

Function Keys select varius functions from Auto Mode screen.

Esc Key

Insert Key

Set coolant ON/OFF (PgUp)

Set / Reset M25 Auxilliary function (PgDn + Ctrl key)

Alt Key

Backslash Key

Enter Key

Ctrl Key (Press & hold to enable M25/M26 key)
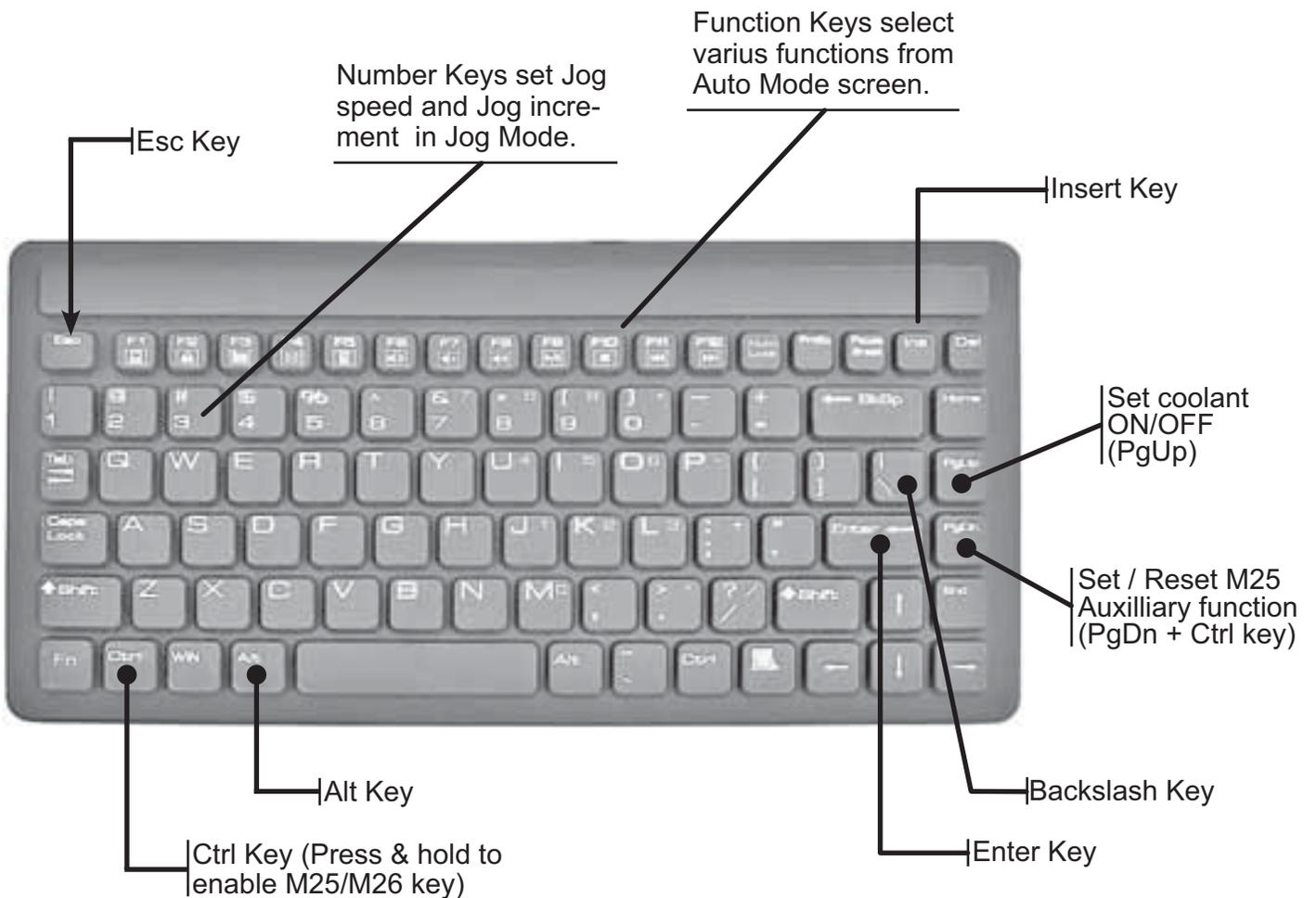
OmniTurn Front Panel and Operator's Station

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.

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.

Motion Stop

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

Cycle start from Front Panel or from Operator's station
Must press both buttons at once.

Open Collet   **E-Stop**   **Close Collet**
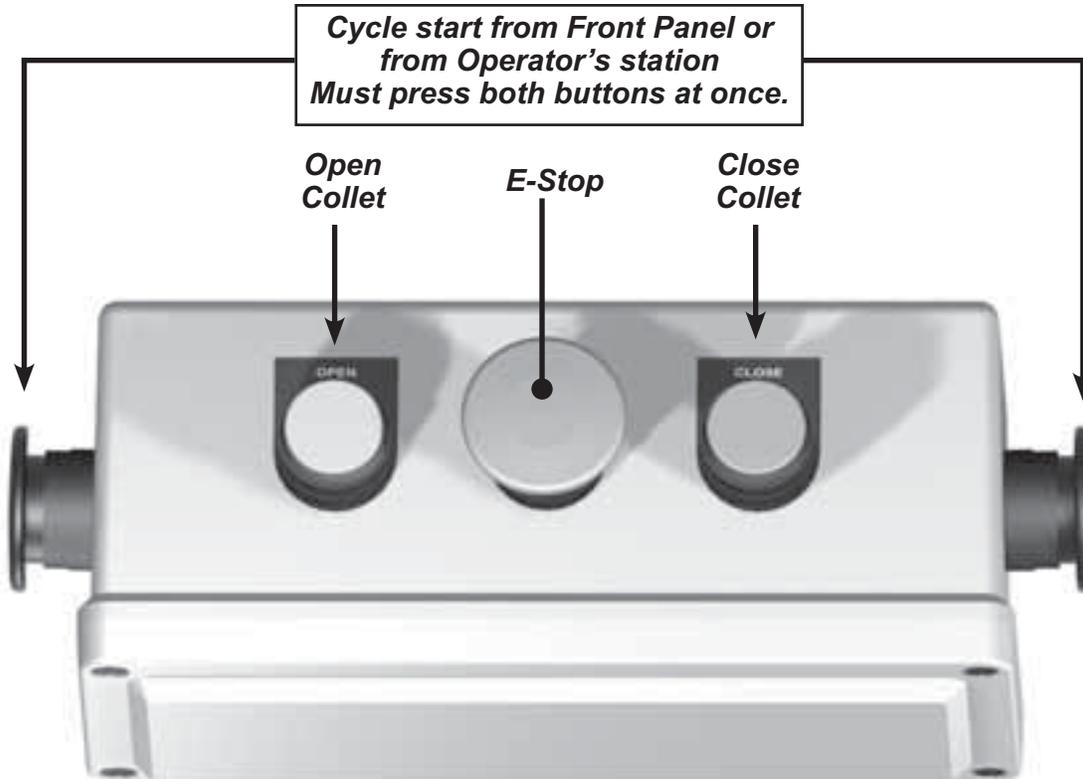
Operator's Station (also called "Palm Box")

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.

RS-232 Port

Serial Number Label

Optional C-Axis Connector (2 of 2)

Palm Box (Operator's Station) Connector

OMNI*TURN* CNC Attachments

C PULSE

OP STA

X Axis | Z Axis
Servo Error due to Overload
If the breaker has not tripped, the servo-amp may have.
To reset amp, set servos off, wait one minute, then set servos on.

(10A)
F 1

PLC

RS 232

SPINDLE
OR
C AXIS

X AXIS

Z AXIS

MISC

X AXIS

Z AXIS

MISC

CNC Control Main Fuse (10A)

Optional PLC Connector

Miscellaneous Cable:
Carries signals to and
from Spindle Cabinet.

Z-Axis Servo Motor (MTE)

X-Axis Servo Motor (MTE)

Spindle Encoder for standard 5hp:
Enables threading and IPR feeds.
Also Optional C-Axis Connector (1 of 2)

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.

*Turn CNC ON here **after** setting 220VAC disconnect ON.*

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 in sockeet on front-panel 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.

Spindle Drive Cabinet

220 VOLTS

Main Power Disconnect

Spindle Fault Switch
This Push-Button will light with spindle drive fault. Press button to reset.

SPINDLE FAULT
Press to Reset

Interlock
Override Switch

Interlock Bypass Key-Switch
To Jog, or operate the machine with splash-guard or hatch open, this switch must have key turned CW. *Key should be under supervisor's control*

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.

```
1.Slow                    Alt-C will enable C axis        Position
2.Medium                                                  X:  0.00000
3.Fast                                                    Z:  0.00000
4. 0.00005"/.01°
5. 0.00100"/.1°
6. 0.01000"/1°                                            Feed
7. 0.10000"/10°                                            0.0000 IPM
8. 1.00000"/90°                                           100%
9.Establish Home
H.Go Home
T.Set Tool                                                Speed
M.Manual Data Input                                         0  RPM
A.Automatic                                               100%
                                                          Tool  0
```
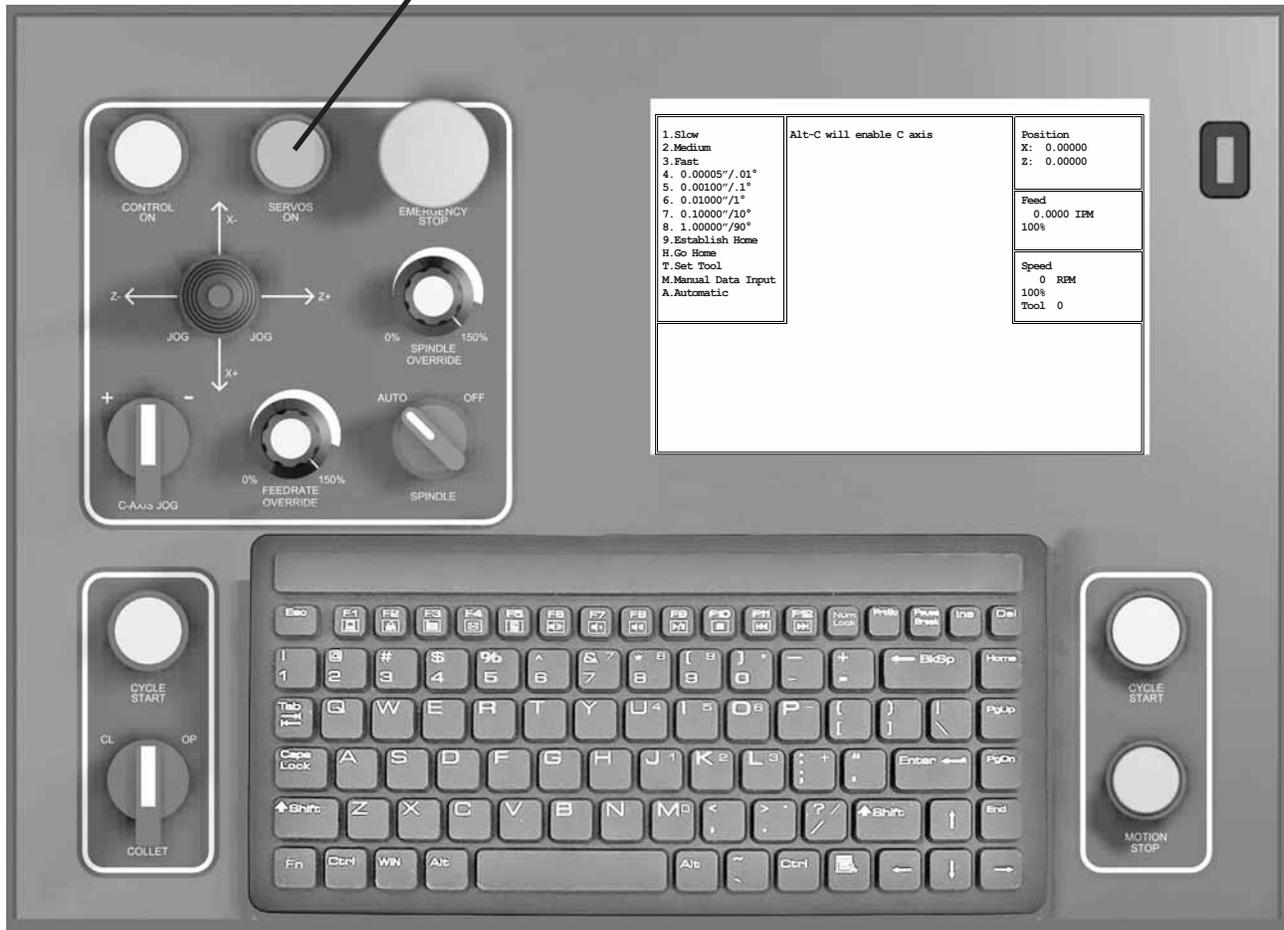
Press the blue "Servos On" pushbutton. It should illuminate and you should hear a slight hum as the axis motors engage. Press any key to clear the 'Servos off' prompt. The OmniTurn is now in "Jog" mode. The only other mode is "Automatic". Manual Data Input (MDI) is part of Jog mode; a dialog box opens to allow program commands to be manually entered.

```
1.Slow                    Alt-C will enable C axis          Position
2.Medium                                                    X: +0.00000
3.Fast                                                      Z: +0.00000
4. 0.00005"/.01°
5. 0.00100"/.1°           Servos off or Ext. Halt detected-
6. 0.01000"/1°            Clear fault or turn servos on, then   Feed
7. 0.10000"/10°           Press any key to proceed              0.0000 IPM
8. 1.00000"/90°                                              100%
9.Establish Home
H.Go Home
T.Set Tool                                                  Speed
M.Manual Data Input                                           0  RPM
A.Automatic                                                 100%
                                                            Tool  0
```
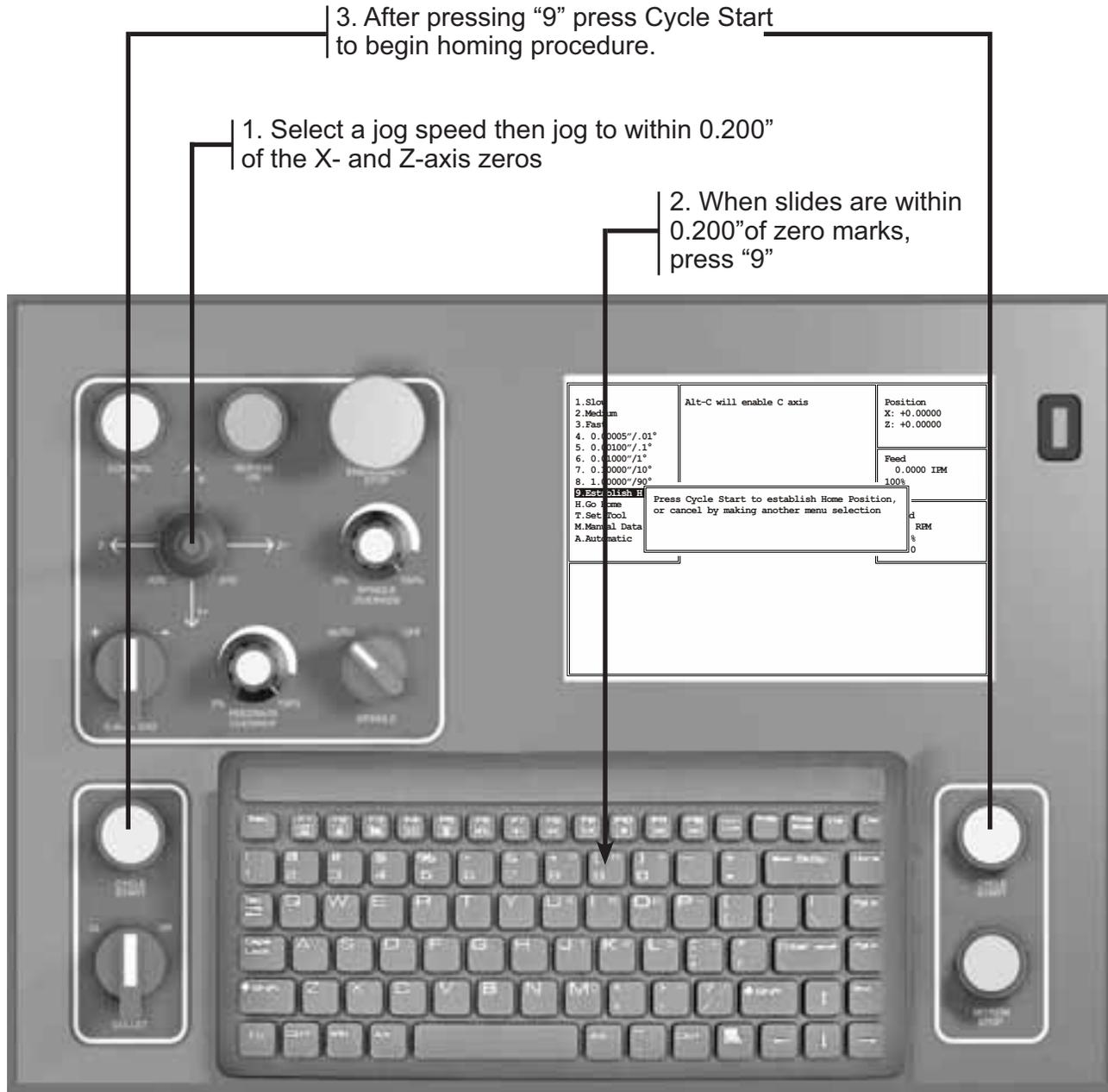
Prompt area indicates that servos are off. This screen also appears when external fault has occured. Set servos on, *then press a key to clear prompt and continue.*

Start - Up: Establish Home

3. After pressing "9" press Cycle Start to begin homing procedure.

1. Select a jog speed then jog to within 0.200" of the X- and Z-axis zeros

2. When slides are within 0.200"of zero marks, press "9"

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

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.

Too far away to start Ref Zero

Within .2" of 0; this is close enough to begin Ref Zero
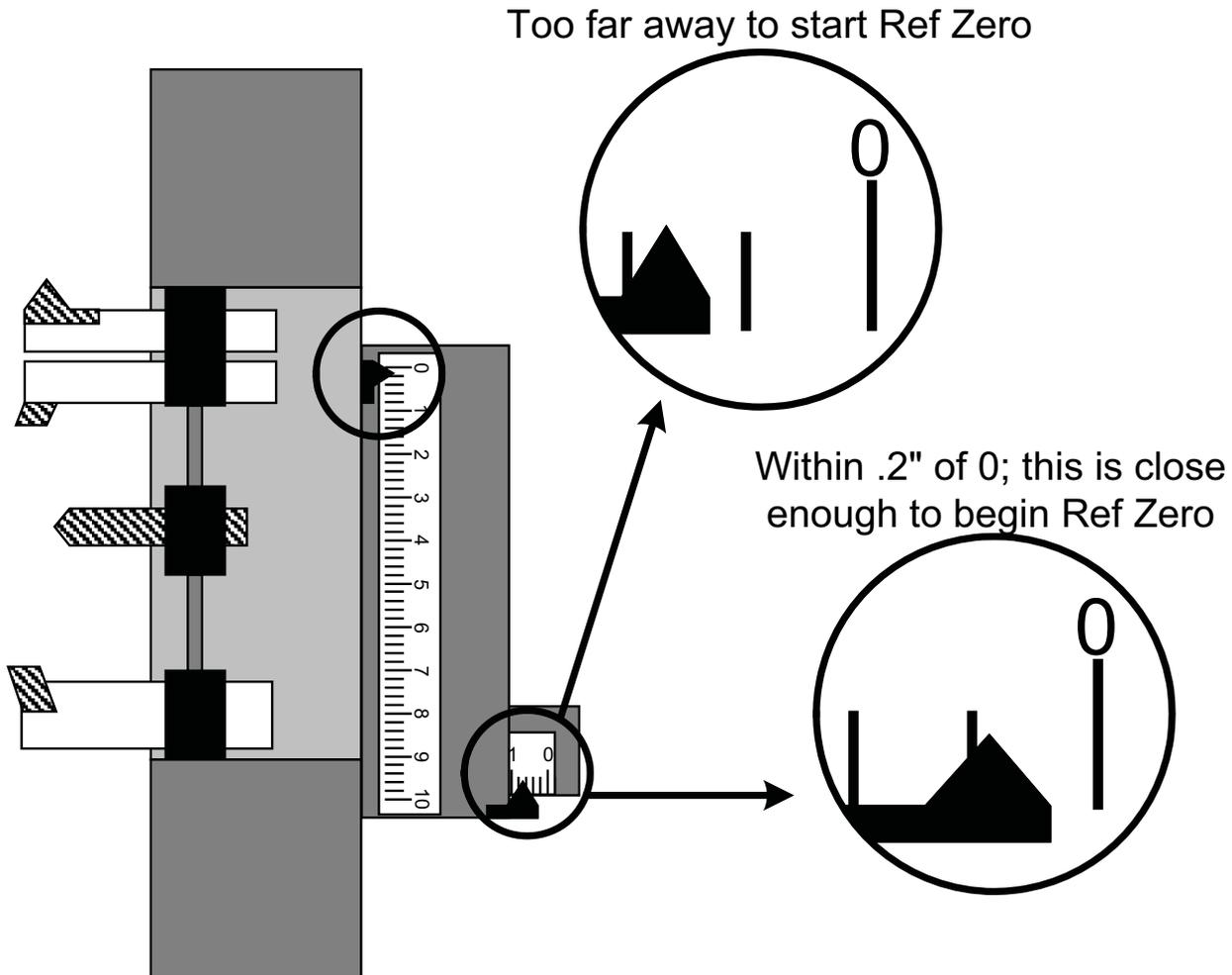
When the slide has been jogged to within 0.200" of zero for *both* axes, press the "9" key on the keyboard, then press Cycle Start pushbutton to complete the homing sequence.

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

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

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 machne 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.

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

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.

X axis Ref Zero Marker

Z axis Ref Zero Marker

1 marker pulse

1,000 encoder lines
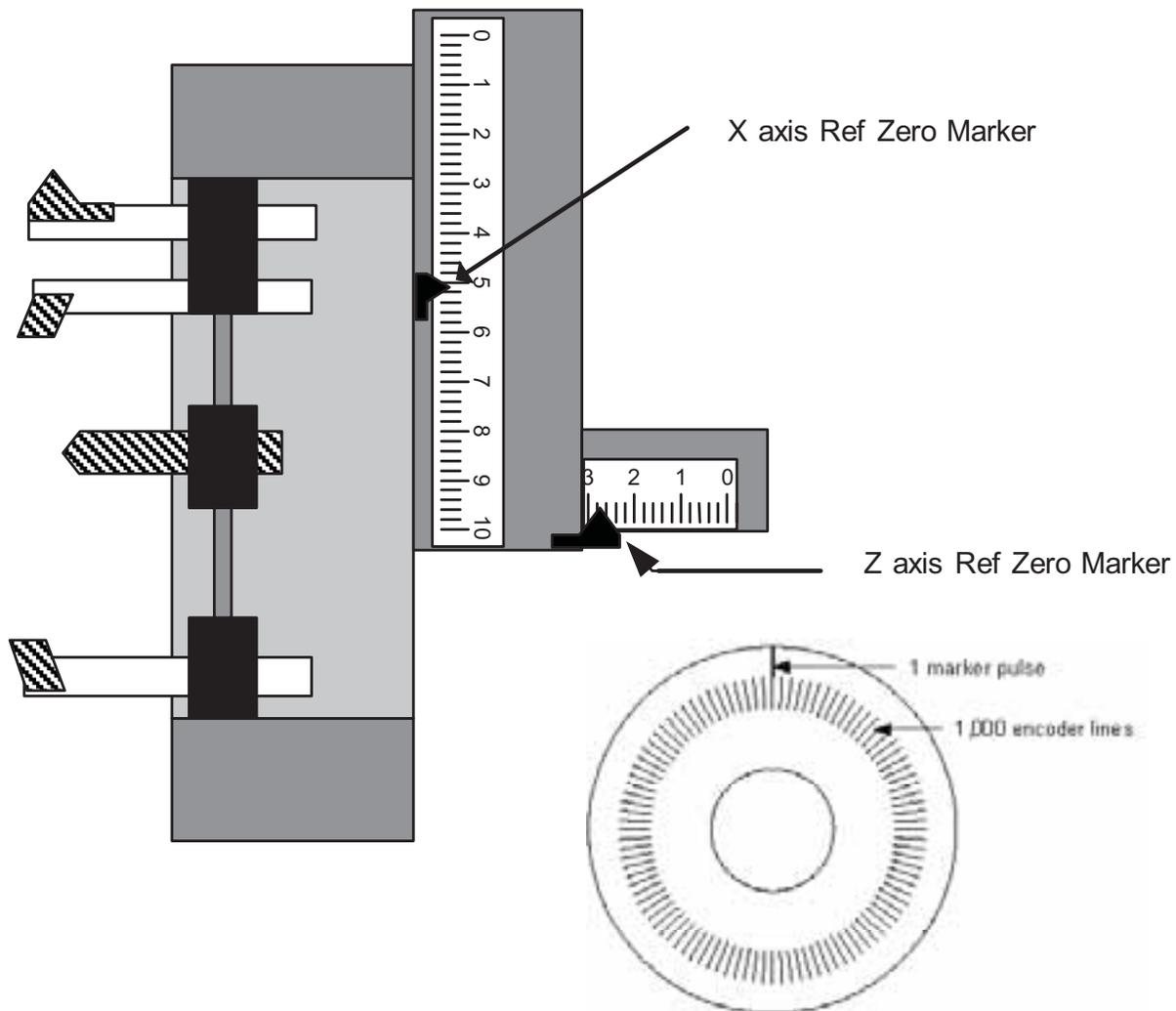
How HOME is established

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 Omni-Turn 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.
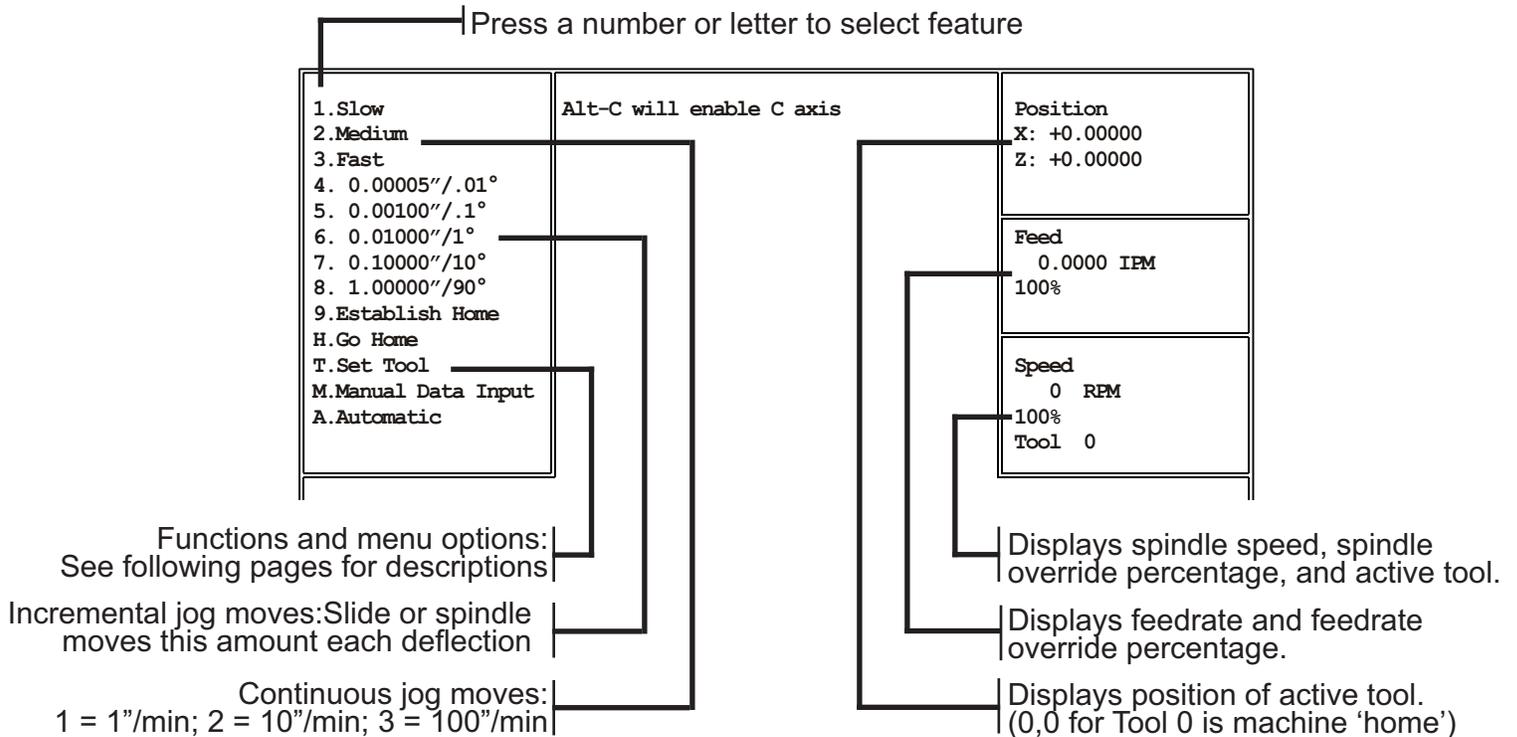
Jog Mode

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.

Press a number or letter to select feature

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

Functions and menu options:
See following pages for descriptions

Incremental jog moves:Slide or spindle
moves this amount each deflection

Continuous jog moves:
1 = 1"/min; 2 = 10"/min; 3 = 100"/min

Displays spindle speed, spindle
override percentage, and active tool.

Displays feedrate and feedrate
override percentage.

Displays position of active tool.
(0,0 for Tool 0 is machine 'home')

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. Deflecting this actuator "+" rotates the spindle in M03 direction; "-" rotates M04.
To set a custom C-Axis home, position the spindle as desired, then press Alt-H; the spindle will orient to this posisition ±.02° whenever C-Axis is enabled, by Alt-C (jog) or by M19.

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.

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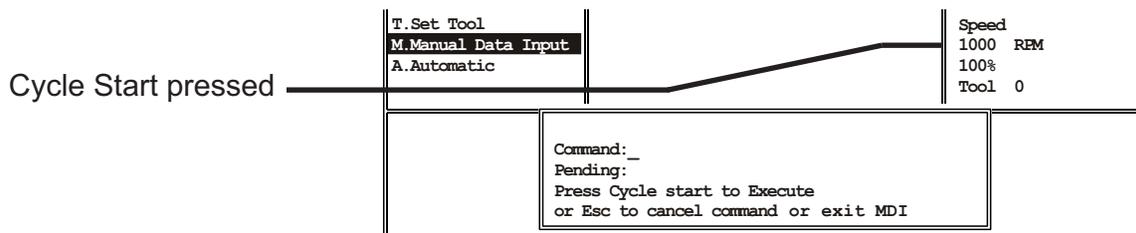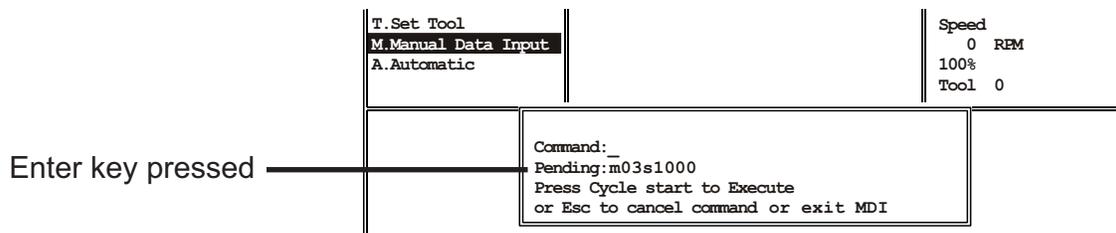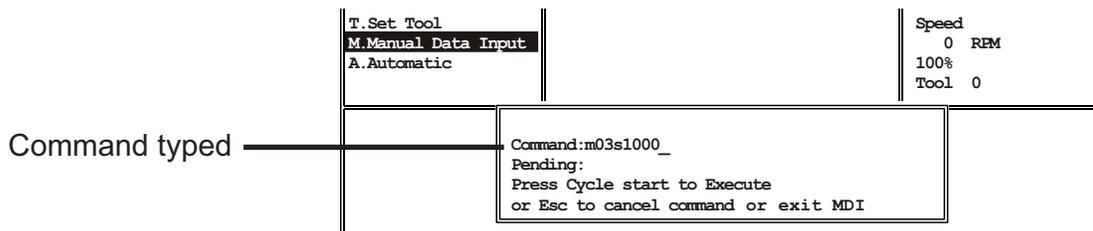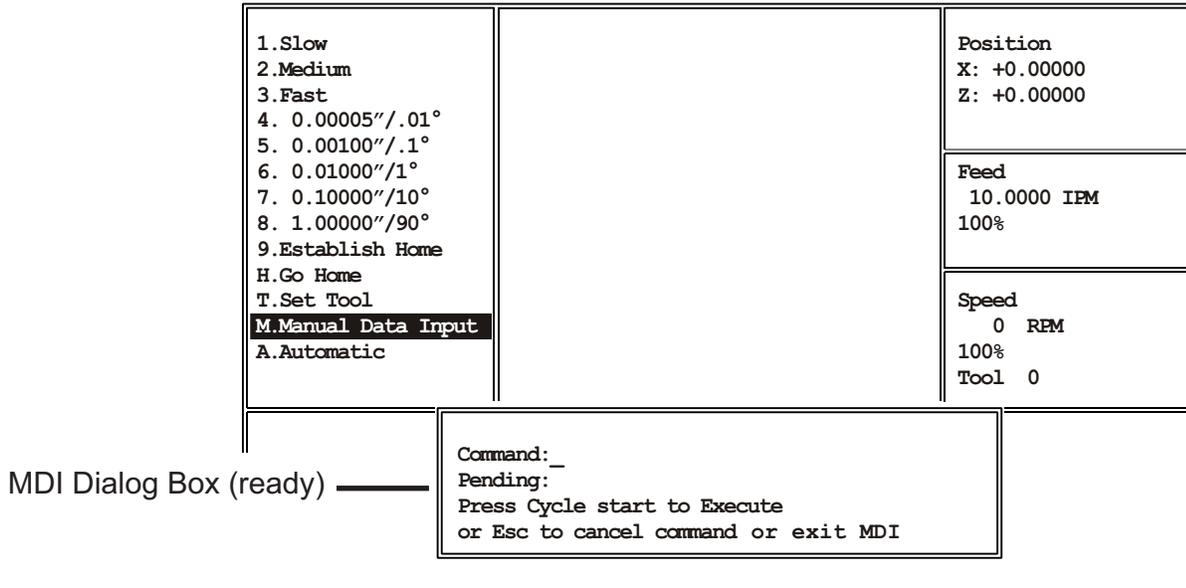For more MDI details see next page.

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

```
1.Slow                          Position
2.Medium                        X: +0.00000
3.Fast                          Z: +0.00000
4. 0.00005"/.01°
5. 0.00100"/.1°
6. 0.01000"/1°                  Feed
7. 0.10000"/10°                  10.0000 IPM
8. 1.00000"/90°                 100%
9.Establish Home
H.Go Home
T.Set Tool                      Speed
M.Manual Data Input               0  RPM
A.Automatic                     100%
                                Tool  0
```

MDI Dialog Box (ready)
```
Command:_
Pending:
Press Cycle start to Execute
or Esc to cancel command or exit MDI
```

Command typed
```
T.Set Tool                      Speed
M.Manual Data Input                0  RPM
A.Automatic                     100%
                                Tool  0

Command:m03s1000_
Pending:
Press Cycle start to Execute
or Esc to cancel command or exit MDI
```

Enter key pressed
```
T.Set Tool                      Speed
M.Manual Data Input                0  RPM
A.Automatic                     100%
                                Tool  0

Command:_
Pending:m03s1000
Press Cycle start to Execute
or Esc to cancel command or exit MDI
```

Cycle Start pressed
```
T.Set Tool                      Speed
M.Manual Data Input             1000  RPM
A.Automatic                     100%
                                Tool  0

Command:_
Pending:
Press Cycle start to Execute
or Esc to cancel command or exit MDI
```

File Handler: Overview

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. Presssing 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.

Directories and disks to search
(c:\ = hard drive; d:\ = USB stick)
To delete unused directories see page 1.22

```
1234A      1235A      2468B      A34          c:\programs\
BUSHING2   COLLAR23   D-0478     DEMO         d:\
FLANGE12   G041414    L12398     SMALTEST     d:\programs\
TESTPROG                                      OTHER
```

Program files in alphbetical order. Use arrow keys to move highlight, or select initial letter or number to highlight first file in series. Press Enter to load selected program into memory.

```
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 curser
```

Options selectable with "Alt" key

```
F5 to type filename,F8 to load from RS232
Current machine file; null
Current directory; c:\programs
```

Current disk & directory
Program in memory ("null" = no program loaded)
Prompt area *(Note that file name can be typed (F5) or scanned (F8)*

Select Program to Run
1. Press initial letter or number of program, then use arrow keys to highlight desired program; press Enter
2. Press F5, then type complete name or number of program; then press Enter
3. Press F8 with barcode scanner connected to RS232 port on rear panel of CNC; then press Enter

---

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.   Highlight "OTHER" to choose or create a directory not on the list.
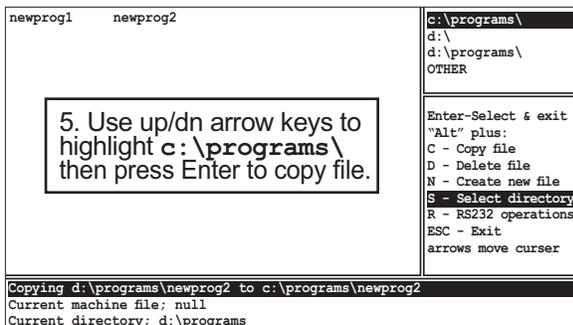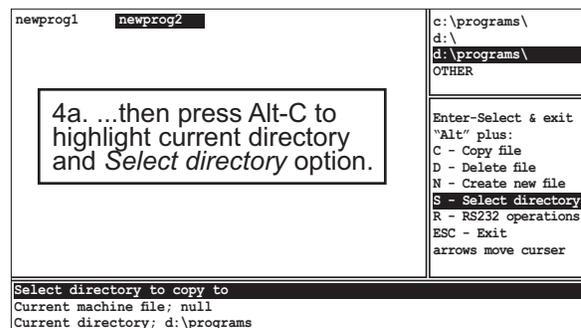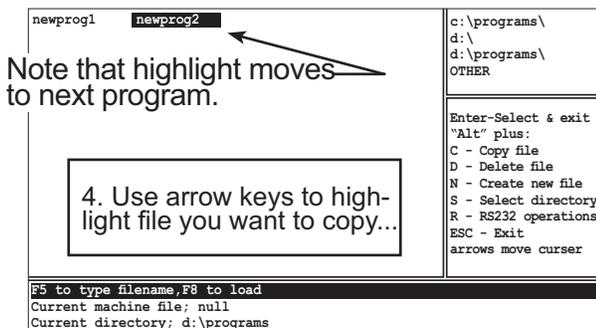***Alt-R*** = *RS232 operations:* This option allows you to send/recieve programs via the serial port.
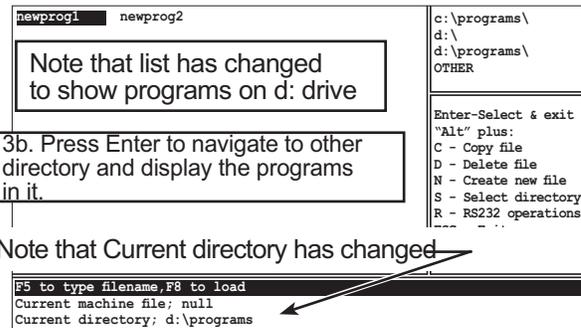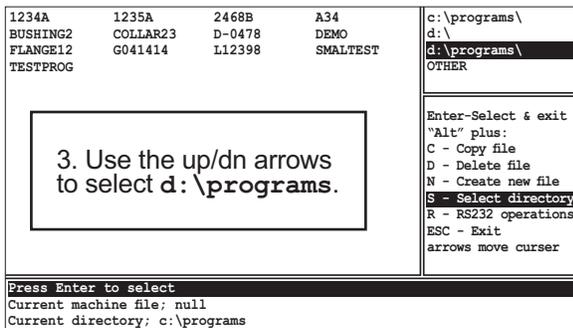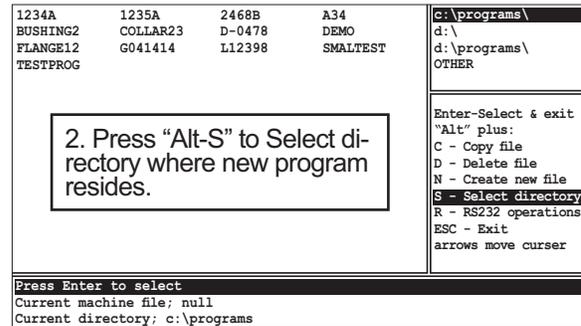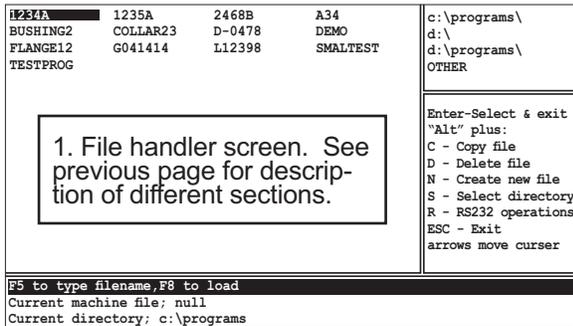
File Handler: Copy Program from USB stick

Alt-C: To copy a program *to* the hard drive *from* 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.

Screens below and on next page illustrate these steps

1234A	1235A	2468B	A34	c:\programs\
BUSHING2	COLLAR23	D-0478	DEMO	d:\
FLANGE12	G041414	L12398	SMALTEST	d:\programs\
TESTPROG				OTHER

1. File handler screen. See previous page for description of different sections.

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 curser

F5 to type filename,F8 to load
Current machine file; null
Current directory; c:\programs

1234A	1235A	2468B	A34	**c:\programs**
BUSHING2	COLLAR23	D-0478	DEMO	d:\
FLANGE12	G041414	L12398	SMALTEST	d:\programs\
TESTPROG				OTHER

2. Press "Alt-S" to Select directory where new program resides.

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 curser

Press Enter to select
Current machine file; null
Current directory; c:\programs

1234A	1235A	2468B	A34	c:\programs\
BUSHING2	COLLAR23	D-0478	DEMO	d:\
FLANGE12	G041414	L12398	SMALTEST	**d:\programs**
TESTPROG				OTHER

3. Use the up/dn arrows to select **d:\programs**.

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 curser

Press Enter to select
Current machine file; null
Current directory; c:\programs

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

Note that list has changed to show programs on d: drive

3b. Press Enter to navigate to other directory and display the programs in it.

Enter-Select & exit
"Alt" plus:
C - Copy file
D - Delete file
N - Create new file
S - Select directory
R - RS232 operations
ESC - Exit

Note that Current directory has changed

F5 to type filename,F8 to load
Current machine file; null
Current directory; d:\programs

newprog1 **newprog2**	c:\programs\
	d:\
	d:\programs\
	OTHER

Note that highlight moves to next program.

4. Use arrow keys to highlight file you want to copy...

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 curser

F5 to type filename,F8 to load
Current machine file; null
Current directory; d:\programs

newprog1 **newprog2**	c:\programs\
	d:\
	d:\programs
	OTHER

4a. ...then press Alt-C to highlight current directory and *Select directory* option.

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 curser

Select directory to copy to
Current machine file; null
Current directory; d:\programs

newprog1 newprog2	**c:\programs**
	d:\
	d:\programs\
	OTHER

5. Use up/dn arrow keys to highlight **c:\programs** then press Enter to copy file.

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 curser

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. *Do not remove USB stick when powered up.*

File Handler: Copy Programs

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 copyied 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 programs from floppy to hard drive:

If you are replacing an earlier version OmniTurn CNC with the new G4 CNC you will have part programs on one floppy disks. *The newest OmniTurn G4 CNC does not have a floppy drive installed.*

Use your desktop computer to copy the programs from your floppies to the USB drive that came with the G4 CNC. If your desktop has no floppy drive, you must purchase an external USB floppy drive. External USB floppy drives are available at most local computer stores and Walmart.

To copy all programs from old-style CNC hard drive to floppy:

1. On the G3 OmniTurn, drop to DOS. (If the machine is already on, press and hold the left-shift key, then press Esc key; if you're just starting up, press and hold Ctrl key, then press C key (Ctrl-c) at the "Do you want to backup..." prompt).
2. At the C:\OMNITURN> prompt, type CD\PROGRAMS then press Enter.
3. Replace your User Programs floppy with the blank floppy.
4. At the C:\PROGRAMS> prompt, type A: then press Enter to log onto the floppy.
5. At the A:\> prompt, type MD A:\PROGRAMS then press Enter to create a new folder (directory) on floppy to hold your programs.
6. At the A:> prompt, type CD PROGRAMS then press Enter to change to the programs folder (directory).
7. At the A:\PROGRAMS> prompt, type COPY C:*.* (C colon star dot star) then press Enter to begin copying all your program files from the hard drive to the "programs" folder on the floppy. This could take a while if you have hundreds of programs.

Replacement USB stick for OmniTurn CNC

The USB stick that shipped with your OmniTurn is primarily for program backup, but it also can be used as emergency 'boot' drive if internal hard drive should fail.

Replacement USB stick is available from the factory: p/n 995-16-018.

Alternatively, you can create your own replacement USB stick by performing the procedure described at www.omniturn.com/bin/G4_USB_image1.htm

or by going to www.omniturn.com and selecting "3-Downloads", "Software", "G4 CNC USB".

File Handler: Delete Program

To copy all programs from G4 CNC hard drive to USB stick:

The best way to insure that all your program files are backed up to the USB stick is to consistently select "Y" at the "Please backup your program files" prompt each time you power up your Omni-Turn.

If this has not been done consistently and you want a complete copy of all program files, follow these instructions:

1. On your desktop computer, create a new folder named "PROGRAMS" on the USB stick.
2. On the OmniTurn, drop to DOS. (If the machine is already on, select Jog, press and hold the Ctrl key, then press Q key (Ctrl-Q); if you're just starting up, press and hold Ctrl key, then press C key (Ctrl-C) at the "Please backup your program files" prompt).
3. At the K:\CNC> prompt, type C: then press Enter.
4. At the C:\RUNFILES> prompt, type CD\PROGRAMS then press Enter.
5. At the C:\PROGRAMS> prompt, type D: then press Enter to log onto the USB stick.
6. At the D:> prompt, type CD PROGRAMS then press Enter to change to the programs folder (directory).
7. At the D:\PROGRAMS> prompt, type COPY C:*.* (C colon star dot star) then press Enter to begin copying all your program files from the hard drive to the "programs" folder on the floppy.

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          c:\programs\
BUSHING2    COLLAR23   D-0478     DEMO         d:\
FLANGE12    G041414    L12398     SMALTEST     d:\programs\
TESTPROG                                       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 curser

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

To delete unused directories:
A list of directories is stored in `C:\RUNFILE\DIRECT.TXT`
To remove unused directories, drop to DOS and edit the file as required.
The steps to do this are as follows:
1. Drop to DOS (Ctrl-C at OmniTurn CNC screen, or Ctrl-Q from Jog)
2. Type `C:` then press Enter to change to `C:\RUNFILES`
3. Type `EDIT DIRECT.TXT` then press enter to open direct.txt in the program editor
4. Delete entire line with unused directory
5. Save and exit the editor, then set CNC control power off, then on to reboot.

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          c:\programs\
BUSHING2    COLLAR23    D-0478      DEMO         d:\
FLANGE12    G041414     L12398      SMALTEST     d:\programs\
TESTPROG                                         OTHER


          ┌──────────────────────────┐     Enter-Select & exit
          │ 1. Press Alt-N to create │     "Alt" plus:
          │ new program; you will be │     C - Copy file
          │ prompted for name.       │     D - Delete file
          └──────────────────────────┘     N - Create new file
                                            S - Select directory
                                            R - RS232 operations
                                            ESC - Exit
                                            arrows move curser

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

```
1234A       1235A       2468B       A34          c:\programs\
BUSHING2    COLLAR23    D-0478      DEMO         d:\
FLANGE12    G041414     L12398      SMALTEST     d:\programs\
TESTPROG                                         OTHER


          ┌──────────────────────────┐     Enter-Select & exit
          │ 2. Type name (less than 9│     "Alt" plus:
          │ alpha-numeric characters),│    C - Copy file
          │ then press Enter; program│     D - Delete file
          │ editor will start.       │     N - Create new file
          └──────────────────────────┘     S - Select directory
                                            R - RS232 operations
                                            ESC - Exit
                                            arrows move curser

Name of file to create? newprog3
Current machine file; null
Current directory; c:\programs
```

```
    File   Edit   Search   Options   Window   Help
                    C:\PROGR AMS\NEWPROG3
 g90g72g94f300
 m30


          ┌──────────────────────────┐
          │ 3. The OmniTurn editor will│
          │ start.  Using the editor is│
          │ described in Section Five. │
          └──────────────────────────┘
```

```
1234A       1235A       2468B       A34          c:\programs\
BUSHING2    COLLAR23    D-0478      DEMO         d:\
FLANGE12    G041414     L12398      NEWPROG3     d:\programs\
SMALTEST    TESTPROG                             OTHER

Note program added to list ─────→

                                            Enter-Select & exit
          ┌──────────────────────────┐     "Alt" plus:
          │ 4. After creating new pro-│    C - Copy file
          │ gram, you are prompted to │    D - Delete file
          │ make new program current  │    N - Create new file
          │ machine file.             │    S - Select directory
          └──────────────────────────┘     R - RS232 operations
                                            ESC - Exit
                                            arrows move curser

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

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
See also page 1.19, using RS232 barcode scanner to enter name of program to run.

The RS-232 transfer parameters are stored in a file C:\RUNFILES\PRM.COM The contents of this file is COM1:2400,n,8,1,rs,cs,ds,cd which allows file transfer without any 'hand-shake' jumpers.

NOTE: For WiFi or Ethernet connection, a Network Interface Kit is available: P/N 990-39-100.

Metric Mode Parameter Setting

G71 sets the control so that moves and feed rates are in "Metric" mode. Complete instructions for using G71 and G70 to switch between "Metric" and "Inch" mode can be found in Section 2, page 2.44.

If the CNC is used in a metric-only environment, the parameter file can be changed so G71 is not necessary.

To change the parameter file:

1. Drop to DOS (Ctrl-Q from Jog screen or Ctrl-C from "OmniTurn CNC" screen).

2. At K:\CNC> prompt, type C: then press enter.

3. At C:\RUNFILES> prompt, type edit filepath.cfg then press Enter to start OmniTurn editor.

NOTE: on older OmniTurns, you may have to type ew filepath.cfg.

The configuration file will be displayed in the editor. The file has 17 lines of configuration data, plus some explanatory text. The configuration data looks like this:

```
c:\pb35\utils\tcmp.~tool offset file
c:\pb35\utils\sectcmp.~secondary offset file
c:\programs\~directory of part programs
XZ~axes recognized(in alpha order)
100~max feedrate in enc. cts./ms
4000~max rpm
1~0 for single, maintained collet signal, 1 for pulsed
1~background color
15~foreground color
8~secondary foreground color
15~background color for active selections
1~foreground color for active selections
4~background color for banners
15~foreground color for banners1~1 for bipolar spindle ref,0 for + only
0~0 for default inch mode,1 for metric    ⬅
inv~for yaskawa;inv or baum for other spindles
```

Change the 0 at the start of the high-lighted line above to a 1. Be careful not to delete the ~ character, because the control software looks for it to mark the end of the data. Press F1, then F2 to save the file and exit the editor. Reboot the machine by turning it off, then back on. When it restarts, it will be in metric mode.

Codes Honored by the OmniTurn control
(Sort by Code)

Code	Usage	Description	Pages
G00	G00	Rapid move	11
G01	G01Fn	Feed move	12
G02	G02XnZnInKnFn	Arc -Clockwise	16-19
G02	G02XnZnRn	Arc -Clockwise	16-19
G03	G03XnZnInKnFn	Arc -Counter Clockwise	16-19
G03	G03XnZnRn	Arc -Counter Clockwise	16-19
G04	G04Fn	Dwell	20
G10	G10XnZn	Work Shift	20-21
G33	G33XnZnInKnAnCaPO	Threading cycle	23-30
G40	G40	Cancels Tool Nose Radius Compensation	31-35
G41	G41	Left hand Tool Nose Radius Compensation	31-35
G42	G42	Right hand Tool Nose Radius Compensation	31-35
G70	G70	Inch mode	37
G71	G71	Metric mode	37
G72	G72	Diameter programming mode	37
G73	G73	Radius programming mode	37
G74	G74XnZnInUnFn	Box Roughing cycle	39-40
G75	G75InUnFnPn	Box Contour Roughing cycle	41-45
G76	G76Sn	Minimum spindle speed for constant surface feet	52
G77	G77Sn	Maximum spindle speed for constant surface feet	52
G78	G78UnFnPn	Rough Contour Cycle	46-47
G81	G81ZnFn	Drill cycle	48
G83	G83ZnKnFn RnLnCn	Peck drill cycle	49-50
G90	G90	Absolute mode selection	51
G91	G91	Incremental mode selection	51
G92	G92XnZn	Preset axis position	51
G94	G94Fn	Inches per minute mode	11,51
G95	G95Fn	Inches per revolution mode	11,51
G96	G96Sn	Spindle speed set as surface feet	52
G97	G97	Spindle speed set as RPM	52
M00	M00	Program stop - does not cancel active "M" functions	53
M0l	M0l	Optional stop	53
M02	M02	End program - does not cancel active"M" functions	54
M03	M03Sn	Spindle on, forward (top coming)	54,58
M04	M04Sn	Spindle on, reverse (top going)	54.58
M05	M05	Spindle off, stop	54,58
M08	M08	Coolant on	54
M09	M09	Coolant off	54
M10	M10	Stop feed; clamp collet (Opt. bar/shaft loader (ZipLoader)	56
M11	M11	Collet unclamp, start feed (Opt. bar/shaft loader (ZipLoader)	56
M12	M12	Collet clamp	54
M13	M13	Collet unclamp	54
M17	M17/M18	One of four optional outputs to be assigned by user	54

Codes Honored by the OmniTurn control
(Sort by Code)

Codes Honored by the OmniTurn control
(Sort by Description)

Code	Usage	Description	Pages
G90	G90	Absolute mode selection. 51
CA	CAnnn.nn	Absolute spindle angle (optional: C--Axis only) 64
G02	G02XnZnInKnFn	Arc -Clockwise 16-19
G02	G02XnZnRn	Arc -Clockwise 16-19
G03	G03XnZnInKnFn	Arc -Counter Clockwise 16-19
G03	G03XnZnRn	Arc -Counter Clockwise 16-19
C	XnZnCn	Automatic chamfer at intersection. 14-15
R	XnZnRn	Automatic radius at intersection 14-15
}	}n	Begin subroutine. 55
/	/	Block delete (block skip) 6
G75	G75InUnFnPn	Box Contour Roughing cycle 41-45
G74	G74XnZnInUnFn	Box Roughing cycle 39-40
M31	M31	Cancels Cycle Repeat mode 54
G40	G40	Cancels Tool Nose Radius Compensation 31-36
M12	M12	Collet clamp 54
M13	M13	Collet unclamp 54
M11	M11	Collet unclamp, start feed (Opt. bar/shaft loader (ZipLoader). . . . 56
M09	M09	Coolant off. 54
M08	M08	Coolant on. 54
G72	G72	Diameter programming mode 37
G81	G81ZnFn	Drill cycle. 48
G04	G04Fn	Dwell 20
M30	M30	End of program - cancels all active M functions 54
M02	M02	End program - does not cancel activeM functions 54
M99	M99	End subroutine 55
G01	G01Fn	Feed move 12
F	Fn	Feedrates, C-Axis Feedrate, Dwell. 7, 64
G70	G70	Inch mode 37
G94	G94Fn	Inches per minute mode 11,51
G95	G95Fn	Inches per revolution mode. 11,51
G91	G91	Incremental mode selection 51
CI	CInnn.nn	Incremental spindle angle (optional: C-Axis only). 64
M98	M98Pn	Jump to subroutine 55
M95	M95	Jump to subroutine 1 if HDR112 is shorted 55
M97	M97InCnPn	Jump to subroutine, conditional (optional: PLC option only). 55
G41	G41	Left hand Tool Nose Radius Compensation 31-36
M40 thru M59		Loader functions (op. ZipLader, front loader, and rear loader) 56
LS	LSn	Loop start 20-21
LF	LF	Loop finish. 20-21
Spindle Control		Manual Control of the Spindle. 57-58
G77	G77Sn	Maximum spindle speed for constant surface feet 52
G71	G71	Metric mode 37
G76	G76Sn	Minimum spindle speed for constant surface feet. 52

Codes Honored by the OmniTurn control
(Sort by Description)

Code	Usage	Description	Pages
M17	M17/M18	One of four optional outputs to be assigned by user 54
M21	M21	One of four optional outputs to be assigned by user (M17,21,23,27)54
M23	M23	One of four optional outputs to be assigned by user (M17,21,23,27)54
M27	M27	One of four optional outputs to be assigned by user (M17,21,23,27)54
M0l	M0l	Optional stop. 53
M84	M84	Override pots (feedrate & spindle speed) lock at 100% 54
M85	M85	Override pots unlock (M30, M02, Tool call also re-enables). 54
G83	G83ZnKnFn RnLnCn	Peck drill cycle 49-50
(~PL)	(~PLXC)	Plane Switching on face (optional: C-Axis only) 67-74
(~PL)	(~PLZC)	Plane Switching on OD (optional: C-Axis only). 67-74
M32, M33, M38		PLC functions (optional Expanded I/O) 56
M60 thru M65		PLC functions (optional Expanded I/O) 56
G92	G92XnZn	Preset axis position. 51
M00	M00	Program stop - does not cancel active M functions 53
G73	G73	Radius programming mode. 37
G00	G00	Rapid move. 11
M87	M87	Real-time Work Shift from digital indcator. 22, 55
G42	G42	Right hand Tool Nose Radius Compensation 31-36
G78	G78UnFnPn	Rough Contour Cycle 46-47
D	Dn	Secondary offsets, TNR comp value or axis correction 35, 59
M05	M05	Spindle off, stop 54,58
M03	M03Sn	Spindle on, forward (top coming) 54,58
M04	M04Sn	Spindle on, reverse (top going). 54.58
M89	M89	Spindle Positioning; lock at arbitrary home (optional: C-Axis only)55,64
M19	M19	Spindle Positioning; lock at home (optional C-Axis only) 64-66
M88	M88	Spindle Positioning; lock at home; high res (optional: C-Axis only)55,64
S	Sn	Spindle speed selection, SFM or RPM 52,54,58
G97	G97	Spindle speed set as RPM 52
G96	G96Sn	Spindle speed set as surface feet. 52
M25	M25/M26	Standard Output to be assigned by user 54
P	P	Start any move in sync with spindle 11,30
M10	M10	Stop feed; clamp collet (Opt. bar/shaft loader (ZipLoader). 56
Tapping		Tapping with the OmniTurn 58
G33	G33XnZnlnKnAnCaPO	Threading cycle. 23-30
T	Tn	Tool offset call command 9
M82	M82	VCO wait for cutoff completion function (optional Vertical Cutoff) . 54
M80	M80	VCO cutoff cycle (optional Vertical Cutoff) 54
M81	M81	VCO cutoff cycle (optional Vertical Cutoff) 54
M93	M93	Wait for HDR110 to be open circuit. 55
M94	M94	Wait for HDR110 to be shorted 55
M91	M91	Wait for HDR111 to be open circuit. 55
M92	M92	Wait for HDR111 to be shorted 55
G10	G10XnZn	Work Shift 20-21
Custom M-Functions		Writing Custom M-Functions for use with Optional PLC. 56

Nomenclature

AXIS: X AND Z
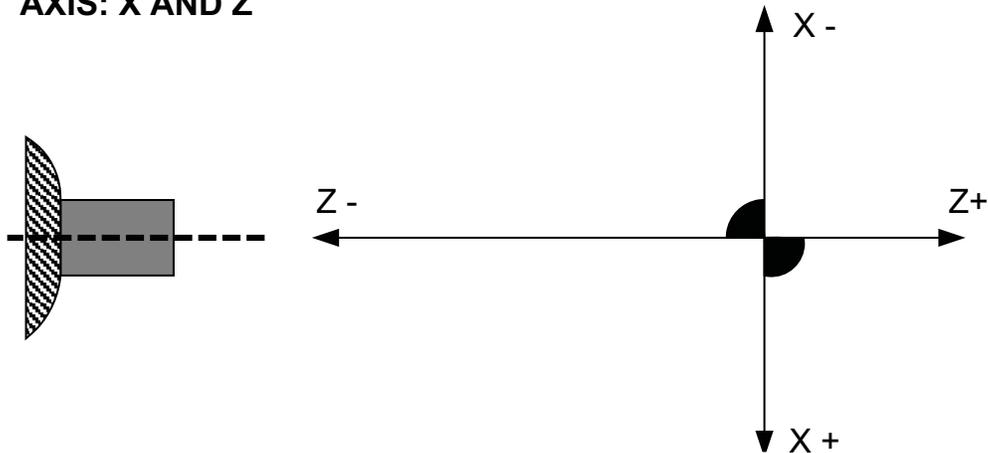
The slide has two axes of travel.
X: Up and down (on GT) or towards and away from you (on Attachments).
Travel up or away from you is (-) minus; down or towards you is plus (+).
Z: Towards or away from the spindle.
Going towards the spindle is (-) minus. Away from the spindle is (+) plus.

ABSOLUTE - G90
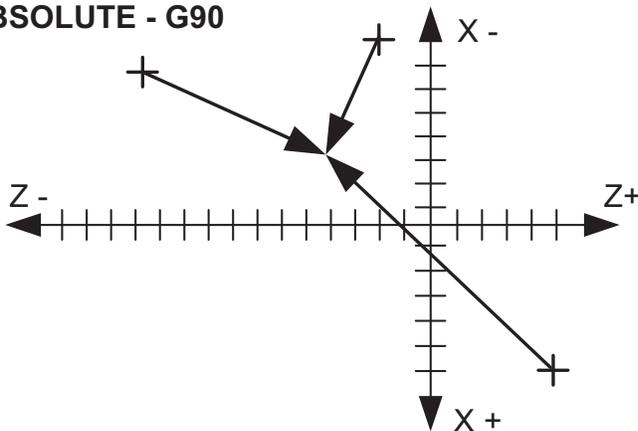
IN ABSOLUTE (G90) THE FOLLOWING COMMAND MOVES YOU THE SAME AMOUNT FROM EVERY START: X-3 Z-4

INCREMENTAL - G91
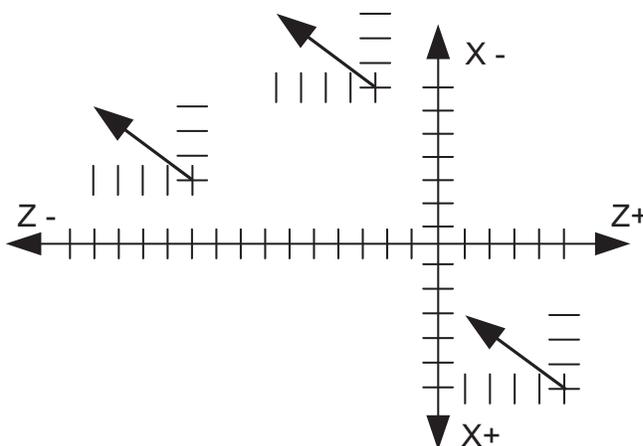
IN INCREMENTAL (G91) THE FOLLOWING COMMAND MOVES YOU THE SAME AMOUNT FROM EVERY START: X-3 Z-4

Programming Format

Header:
It is best practice to put the following block at the start of your program:

g90g72g94f300

g90=absolute; **g72**=diameter; **g94**=ipm; **f300**=fastest cutting speed

A brief description and setup instructions in (comment blocks) may be put at the top of your program.

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 center of the screen. They stay on the screen until the comment is changed. To clear the previous comment put blank parentheses ().
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.
Do not put any comments 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 used with any other command

N sequence line numbers are not allowed. They can cause intermittent problems.

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 at the Auto Mode screen; then only lines not marked with "/" will run. The marked lines will be skipped over. See Automatic Mode section 5.6.

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)*

```
To Go                 c:\programs\DEMO          Position
X: +0.00000                                     X: +7.86285  ⌀
Z: +0.00000           t1                        Z: -0.62225
                      x0z2
                      m00

Switches                                        Feed
C.Cycle F             PUT PART IN COLLET         300.000  IPM
/.Block Ski                                      100%
O.Optional S
S.Single Bloc        Press Cycle Start to continue

                                                Speed
                                                   0  RPM
Functions                                       100%
F2-Tool Offsets                                 Tool  1
F3-Edit
F4-Verify
F5-New File
F6-Program Search
F8-File Ops
F9-Sec. Offsets
F10-Spec. Functions       Comment is displayed here
J-Jog Mode
M-Manual Data Input
```

- **Secondary offsets**:
  These are activated by the letter D. See the section of secondary offsets.

  X.2Z-.35**D2**   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 USB stick, 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:
See Section 1.19ff for a complete description of the OmniTurn File Handler, especially "Alt-N", create new file.

See Section 5.19ff for a description of the editor.

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.
The new OmniTurn File Picker will recognize files with a ".txt" or ".cnc" extension, as well as those with no extension at all.
Copy the file to USB stick (thumb drive) and put the stick 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.

The second block of the T command must be an X and Z move. Both X and Z values must be declared. This is the location that the tool will move to.

To be safe, you should first move the tool 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.

In order to use a tool, the control must know where part X and Z zero are located. This is done in Jog mode by 'touching off' each tool that is used in a program.

Program fragment:

```
G00      (rapid move)
T1       (tool call for tool#1)
X.25Z1   (move the tool to a location of X=.25 and Z=1)
Z.1      (move 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
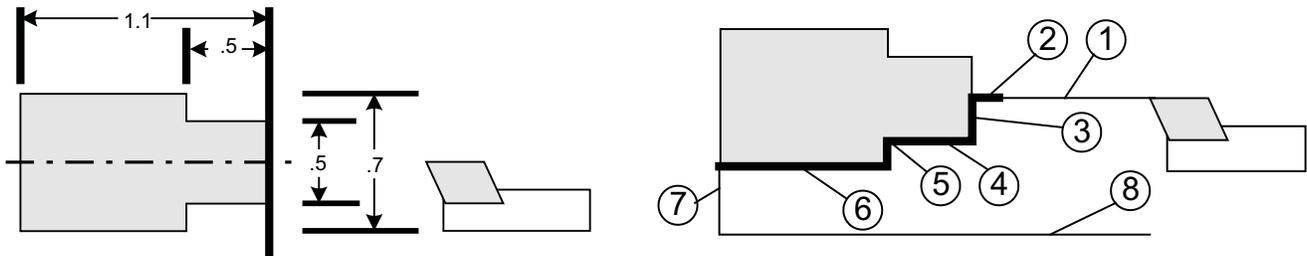
The moves to generate this tool path could be as shown below. The tool path given does not show any of the codes needed to determine the feedrates or spindle speeds etc. The code is shown in the;

	diameter mode:	radius mode:
	G90G94F300G72	G90G94F300G73
	M03S2000	M03S2000
	M08	M08
	T1	Tl
	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.075.

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

"P" Suffix - Sync move with spindle:

Adding the letter P to the end of any move command causes that move to start in sync with the spindle. You can make a do-it-yourself threading cycle by programming each cutting pass to start in sync with the spindle. This gives you more control over the depth of cut on each pass and the pullout position.

See page 2.30 for thread-milling example.

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):

G01 is shortcut for "g94Fnnn" or "g95Fnnn", whichever was the last feedrate called in your part program.

If you are cutting with the same feedrate on different tools, use g00tn to call the next tool at 300ipm, then g01 to switch from rapid to the previous cutting feed rate, either IPM (g94) or IPR (g95).

To use the G01 command you must specify the first feedrate move in the program with the g95fn or g94fn 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:

```
g94f300
x0z1
z.1
g95f.003
z.005
x.5
g00
z.05
x0
g01
z0
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.
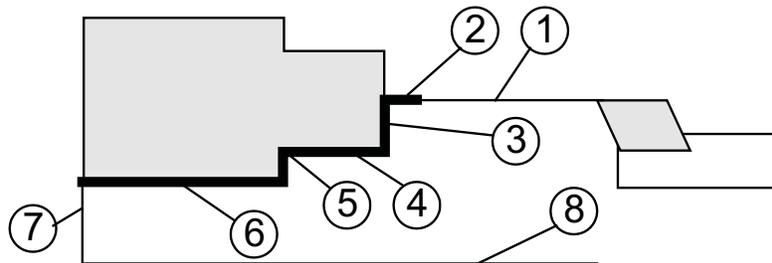
Example in diameter mode

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

Tl (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)

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.
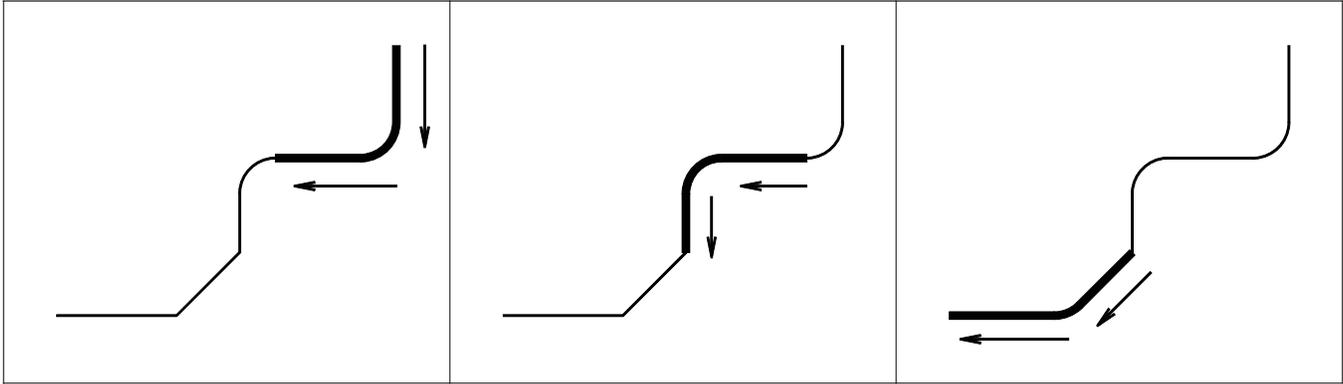
Format

XnRn	**-ZnRn**	**Xn-ZnRn**
-Zn	**Xn**	**-Zn**
XnCn	**-ZnCn**	
-Zn	**Xn**	

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.
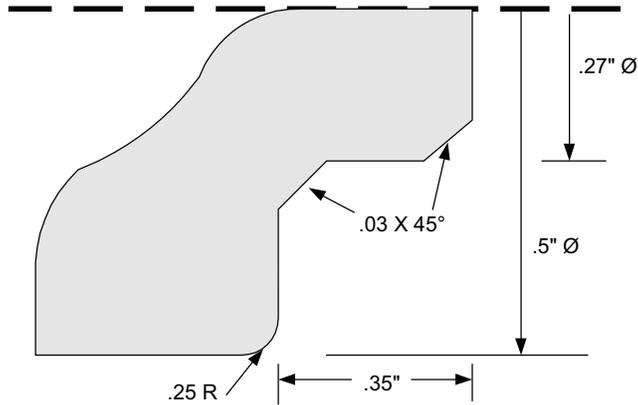
G90G72G94F300	G72 Program is in diameter mode	
M03S2500	Turn the spindle on at 2500 rpm	
M08	Turn the coolant on	
T1(LH turn tool)		
X0Z1		
Z.05		
G95F.001		
G41	Turn on LH cutter compensation	
Z0D1	Use the radius value found in D1 for compensation	
X.27**C.03**	Set chamfer amount	
Z-.35**C.03**	Set 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

G02 is used to generate a clockwise arc.
G03 is used to generate a counterclockwise arc

G02 and G03 Arcs by **Radius (r):**
G02XnZnRn / G03XnZnRn:

G02 and G03 Arcs by **Center (i-k):**
G02XnZnInKn / G03XnZnInKn

Xn: X location (Ø) at the end of the arc
Zn: Z location at the end of the arc
Rn: Radius of arc
In: X location of center of arc (incremental)
Kn : Z location of center of arc (incremental)

Position tool at Start of arc before calling arc move.

Definitions G02
Center of arc
k
i
Start of arc
Radius of arc
End of arc

Definitions G03
Start of arc
Radius of arc
End of arc
i
Center of arc
k

Dimensions G02
0.150
0.235 (k)
0.500
0.200
0.085 (i)
R0.250

Dimensions G03
0.150
0.500
R0.250
0.200
0.235 (i)
0.085 (k)
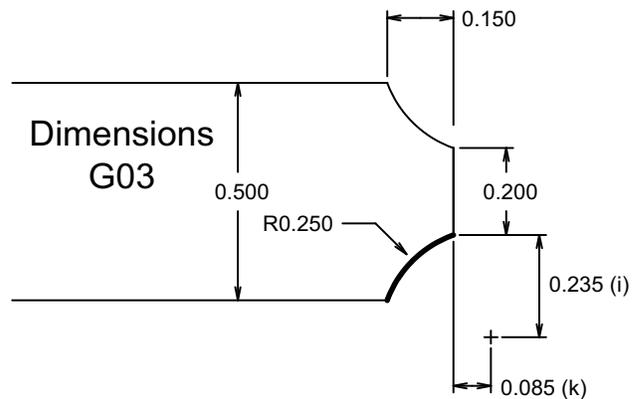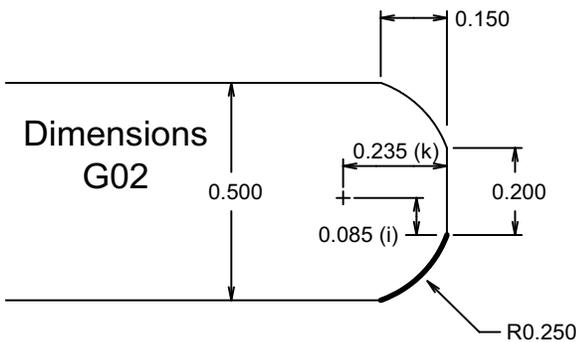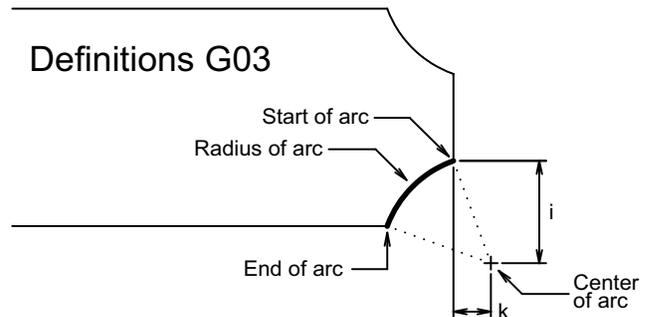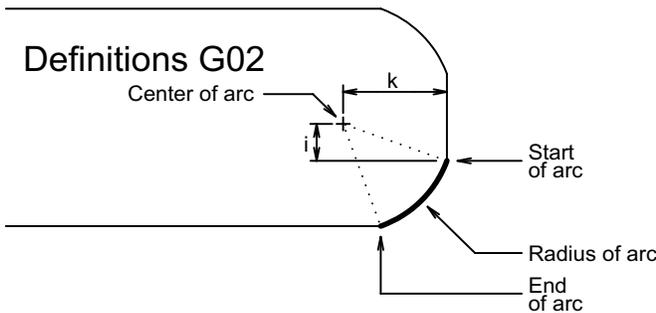
Code, G02
(both 'r' and 'i-k' are shown)

```
.
.
t1
x0z1
z.1
g95f.003
z0
x.2 (Start of arc)
using r:    g02x.5z-.15r.25
using i-k:  g02x.5z-.15i-.085k-.235
z-.3
```

Code, G03
(both 'r' and 'i-k' are shown)

```
.
.
t1
x0z1
z.1
g95f.003
z0
x.2 (Start of arc)
using r:    g03x.5z-.15r.25
using i-k:  g03x.5z-.15i.235k.085
z-.3
```

Arc statements G02 and G03

G02 and G03 Arcs by Radius
G02XnZnRn / G03XnZnRn

G02: Clockwise
G03: Counter clockwise

Xn: X location (Ø) at the end of the arc
Zn: Z location at the end of the arc
Rn: Radius of arc

R0.75

R0.50

0.100   0.400

0.100

0.200

0.600
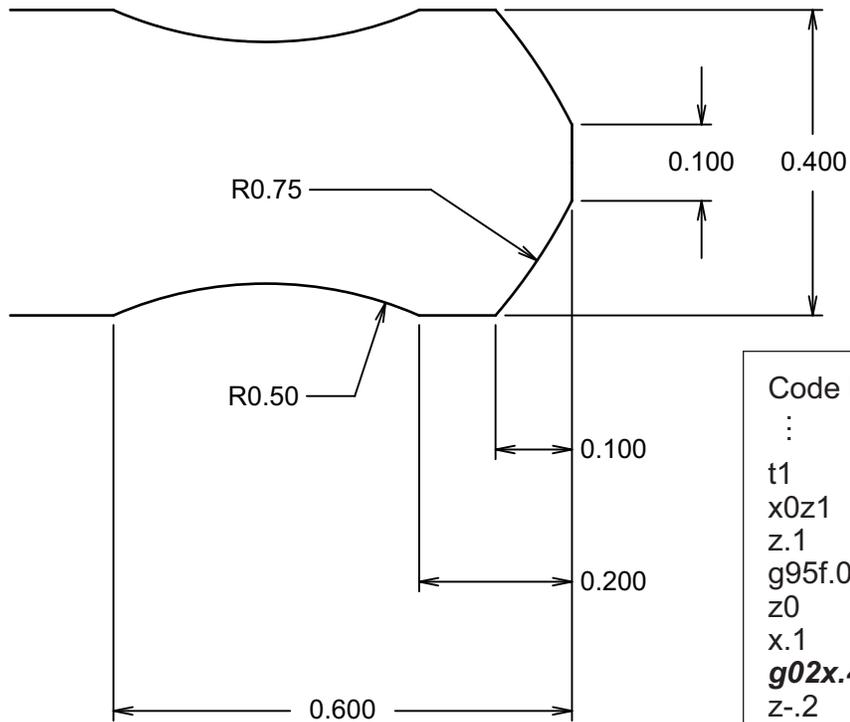
```
Code Fragment
  ⋮
t1
x0z1
z.1
g95f.003
z0
x.1
g02x.4z-.1r.75
z-.2
g03x.4z-.6r.5
z-.7
  ⋮
```

Tool Path

x.1

g02x.4z-.1r.75
(cw)

z-.7   g03x.4z-.6r.5   z-.2
                   (ccw)
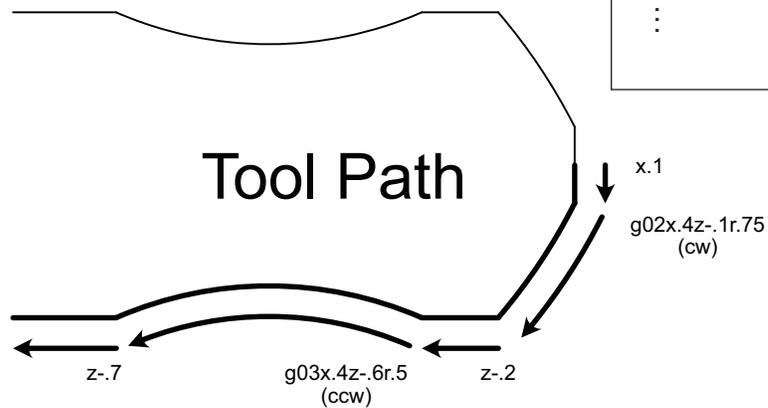
Arc statements G02 and G03

G02 and G03 Arcs by Center
G02XnZnInKn / G03XnZnInKn

G02: Clockwise
G03: Counter clockwise

Xn: X location (Ø) at the end of the arc
Zn: Z location at the end of the arc
In: X location of center of arc (incremental)
Kn: Z location of center of arc (incremental)
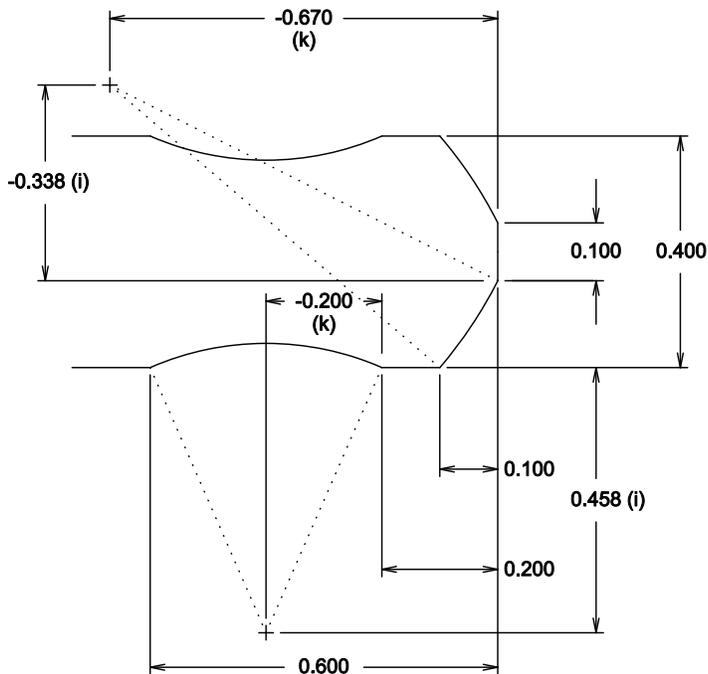
```
Code Fragment
  ⋮
t1
x0z1
z.1
g95f.003
z0
x.1
g02x.4z-.1i-.338k-.67
z-.2
g03x.4z-.6i.458k-.2
z-.7
  ⋮
```

Tool Path
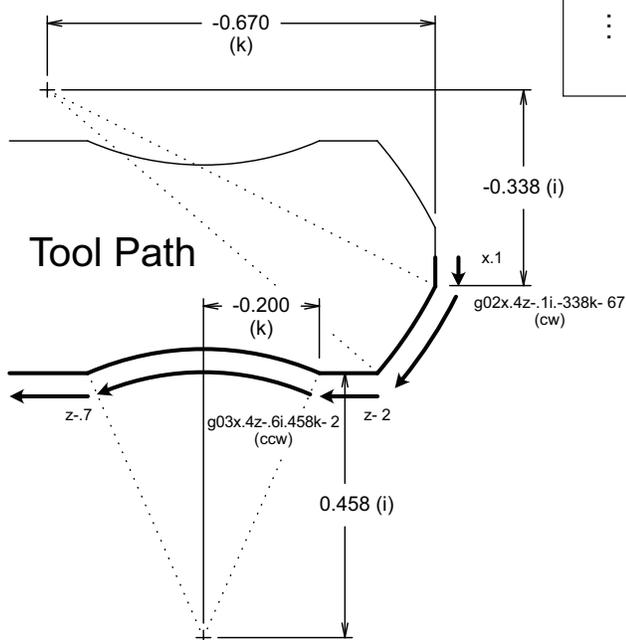
This page intentionally blank

G04 Dwell

The dwell pauses the program for a period of time. The format is:

G04Fn

The "n" after the F is the number of seconds needed to dwell. The shortest dwell is .1 second.

The dwell is useful for brief pause at the bottom of a flat hole or a groove before pulling out, or to delay cutting until the coolant is flowing. DO NOT use it for loading and unloading operations. This would be unsafe!

Example

G04F.5   This statement will dwell the slide .5 seconds.

G10 Work shift, and Looping

Work shift is used to offset a program from the original starting point. Typical applications are machining multiple parts off a single feedout of a bar, or for shifting a program away from the spindle the first time it is run. The format is:

G10XnZn

G10 shifts 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. The shift is canceled by m30 or m02. The command can have a value for X or Z or both.

Looping

Looping is used to perform repetitive moves without having to write long programs. The start of a loop is defined by **LS*nn*** where *"nn"* is number of times to execute the loop. At the end of the loop put a **LF** on a line by itself. ***NOTE: Comments cannot be used inside the loop.***

Here is a loop program for a decorative piece using g10 and g02:

```
g90g72g94f300
t1 (0.031 radius groove tool)
x0z.1
m03s1000
g95f.003
z0
ls9 (LOOP START)
t1 (0.031 radius groove tool)
x.4375z0
g02x.4375z-.125r.0625
g10x0z-.125 (WORK SHIFT)
lf (LOOP FINISH)
t1 (0.031 radius groove tool)
x.4375z0
g02x.5625z-.0625r.0625
z-.25
m30
```

F4 Verify screen showing tool paths
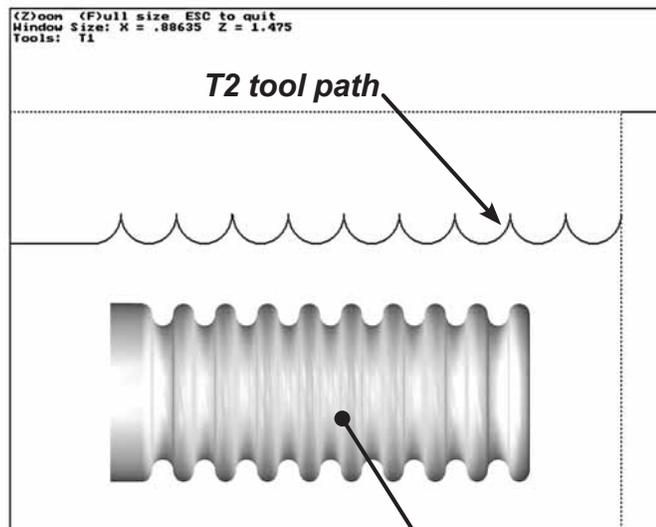
Note that the tool path/tool radius is such that smooth contours result.

G10 Work shift (see also M87 next page)

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:

```
G90G94F300G72
M03S2500
T1 (table stop)
x0z.05
z-1.75
m13
f10z0
m12
LS3  (LOOP START)
T2(lh turn tool)
X0Z.2
x1.05z.05
G95F.003
g75i.05u.02f.003
x0z0
x.5z-.25
z-.5
x1.025
RF
g00z.1
x0z.05
g01
z0
x.5z-.25
z-.5
x1.025
g00z.2
T3 (0.125 partoff)
x-1.05z.2
z-.875
g01
x.05
x-1.05
g00z.1
g10x0z-.875  (WORK SHIFT)
LF  (LOOP FINISH)
M30
```

F4 Verify screen showing tool paths


```
(Z)oom  (F)ull size  ESC to quit
Window Size: X = 1.69765  Z = 2.825
Tools:  T1 T2 T3
```

T3 tool path (part-off)

T2 tool path

G75 tool path
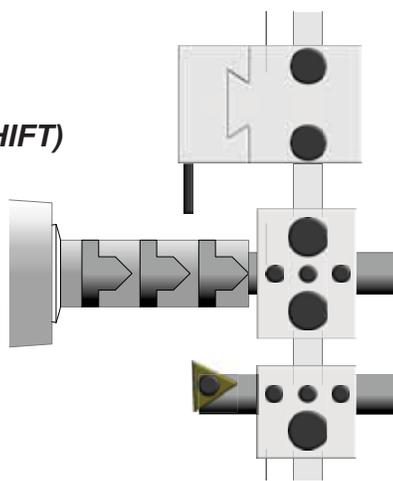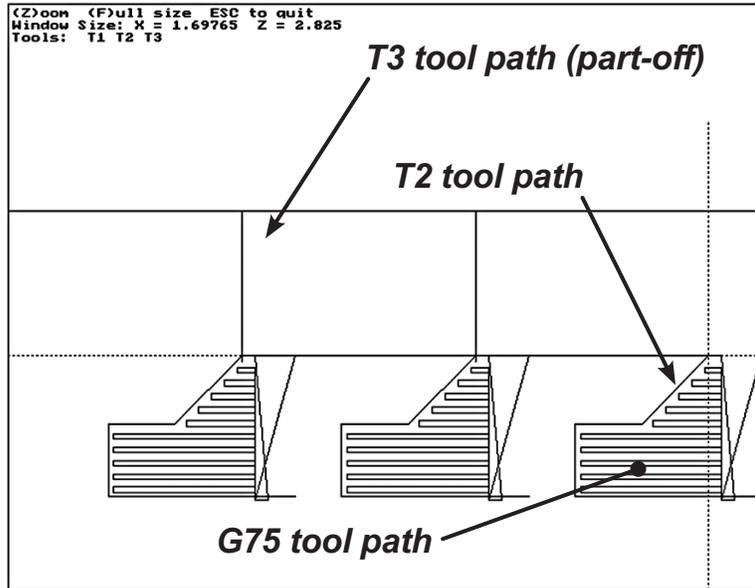
Close collet before
loop begins
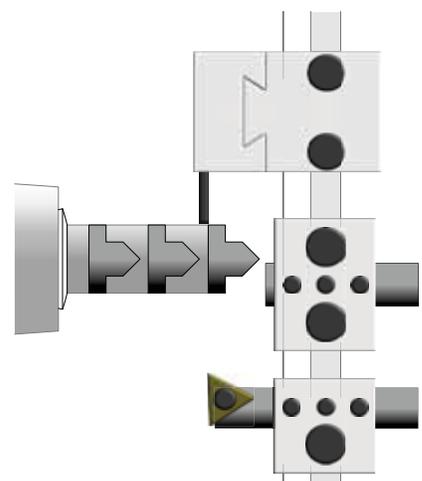
Ready to part-off
first part

M87 External Offset Adjust

The M87 code performs a workshift or tool offset change based on data from an external measuring device.

The device sends its data to the CNC over the RS232 connection on the CNC's COM1 serial port.

The M87 adjustment has 2 different modes of operation: it can work with a simple digital indicator or with a more elaborate PLC-controlled measuring system. In either mode, the communication parameters are set by a user-created file called GAGE.TXT (described below). The measurement units provided by either measurement system must match the inch/metric mode of the part program. X values are always treated as radial, even if the part program is in diameter mode.

In the simple indicator mode, the M87 code applies power to the indicator through the handshake signals in the RS232 connector. The control allows approximately a second for the indicator to stabilize, then signals the indicator to send its reading by toggling one of the handshake lines. The indicator responds with an ASCII text string corresponding to the indicator reading, e.g. "-.0050". This value is applied as a G10 workshift to whichever axis is specified in the GAGE.TXT file, just as if the program contained a G10 statement in place of the M87.

In the PLC-controlled mode, the user's part program must first signal the PLC to start the measurement, then do the M87. As described above, it takes about a second from the time the M87 is executed until the control is ready to receive the measurement from the serial port. The PLC needs to format the measurement readings into a text string to pass to the control. The adjustments may be applied to either a G10 workshift or to a tool offset. The PLC needs to provide either a tool offset number or G10 code in the response string, along with the value of the measurement and the axis (or axes) to which the measurement is to be applied.

Example strings:
    T12X.003
    T10X-.004Z.007
    G10X.0007Z-.002
The PLC program is responsible for assembling the text strings. If there are multiple tools that need to be adjusted, it may be necessary to use a different M-function to signal the PLC for each tool's measurement.

The GAGE.TXT file
This is a file that the user creates in the C:\RUNFILES directory. It needs two lines: the com port parameter string, and the axis to be adjusted in the simple indicator mode.

Example for Z-axis, as used with the Mahr 1075R indicator:
    com1:4800,n,8,1,cs,ds,cd*
    Z

Note that the Mahr indicator uses the DTR line as a signal to transmit the reading, so the cable must include that wire. If you are interfacing to a device that doesn't use a handshake line, you can add "rs" to disable the handshake lines and use a 3-wire cable or rs422 converter.
Your gage.txt file would look more like this:
    com1:9600,n,8,1,cs,ds,rs,cd*
    Z

Mahr 1075
Digital
Indicator

* Actual com port parameter string may vary: see manufacturer's datasheet

G33 Threading

G33XnZnInKnAnCnPO

(First pass only: 2x depth of cut in diameter mode)

Before G33, locate tool at major diameter minus 2x depth of cut)
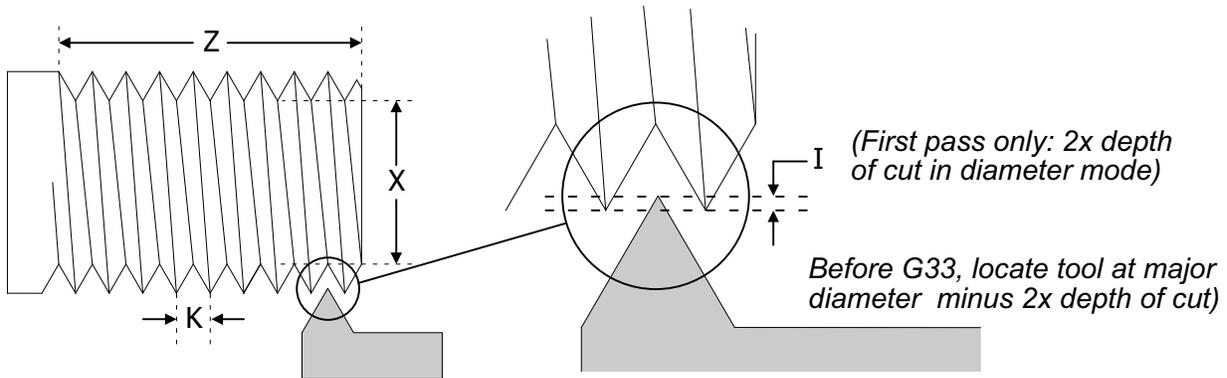
X The minor diameter of the thread

Z The Z axis location of the end of the thread

I Depth of cut on the first pass. In diameter mode, I is 2x the depth of cut

K The lead of the thread, distance in Z per revolution. For single-start thread lead = pitch. 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:

Starting position in Z
The tool in most cases will be started at least 0.10" 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 coarser 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
Start the tool so that it takes a full cut ("I") on the first pass. The control will calculate the amount of material removed, then reduce the depth of the cut as the tool gets deeper to keep the amount of material removed constant. Because the depth of cut reduces with each pass to keep the amount of material constant, the number of passes is not simply total depth of cut divided by "I".

G33 Threading

Calculating the number of passes:
Because the amount of material removed with each pass is kept constant as the tool gets deeper into the thread, the depth of cut is reduced for each pass. The number of passes is equal to the square of the total depth divided by I:

$$\#passes = \left(\frac{\text{tot depth}}{I}\right)^2$$

For example: 1/2-13 od thread with minor diameter=0.4084 and basic major diameter=0.5000 has total depth of 0.0916. If I is set to 0.020 then total depth divided by I ≈ 4.5 and 4.5² ≈ 20, so about 21 passes. Locate your tool so that the very first pass is "I" deep in the part.

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.
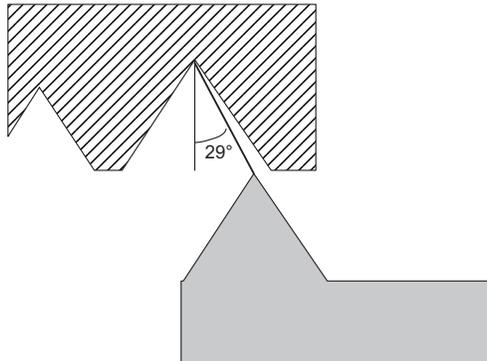
The single pass option O can be used for a cleanup pass:
When a single pass is needed write the same threading pass as used for cutting the thread. Just add a O to the command. Be sure to start the thread at the same point and at the same spindle speed. This option can be used with all variations of 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: 1/2-13 thread, Minor Ø = .4084", Major Ø = .495", length of thread = .5", no undercut.
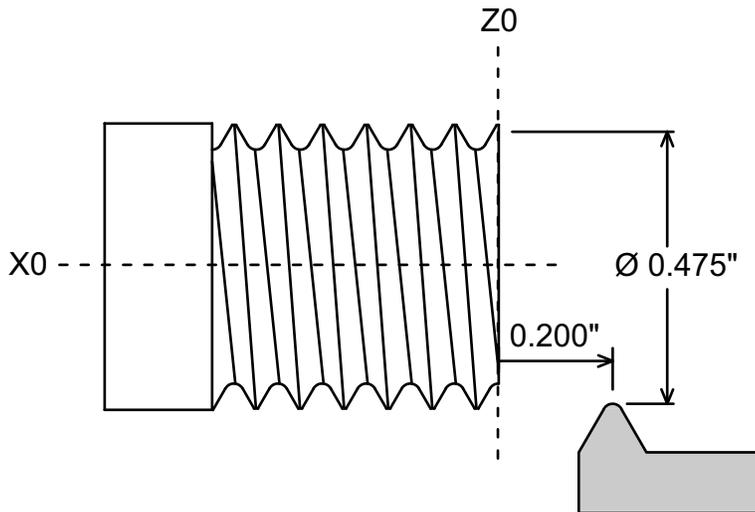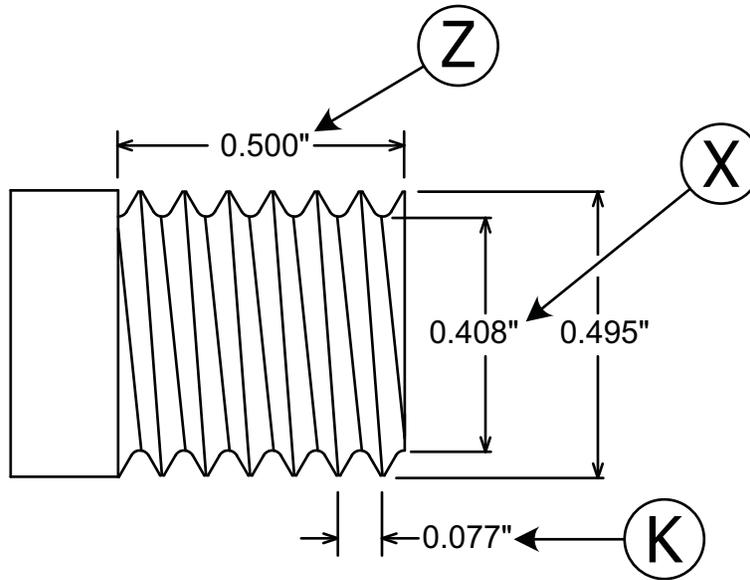
The starting location in X is the major diameter, minus I (twice the depth of cut of the first pass). In this case the major diameter is 0.495", depth of cut is 0.010. I = 0.020", so starting X diameter = 0.475".

Turning is almost always done in diameter mode (G72), so this example is in diameter mode. The only potential confusion is the "I" parameter because depth of cut is incremental, but "I" must be *twice* depth of cut to conform to the diameter mode.

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.475	(Positions the tool at the starting diameter to cut the thread)
G33X.408Z-.5I.02K.077P	(Cut thread)

G33 Threading

Internal threading example

15/16" - 20 THREAD

.4375"

.2"

.75"
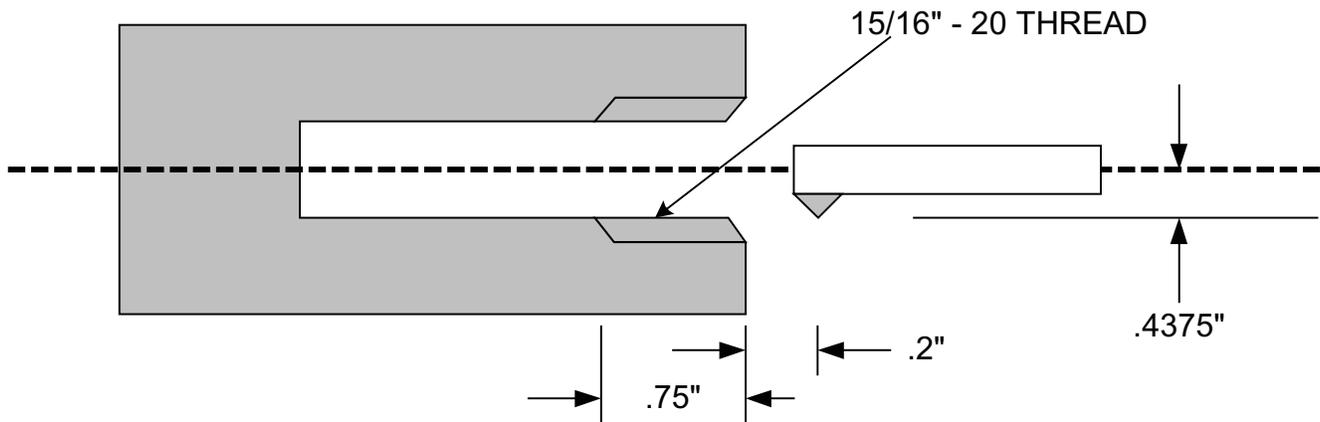
In the above example we will be cutting a 15/16-20 internal thread. (written in radius mode, G73):

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.46875**Z**-.75**I**.01**K**.05**C**	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	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.212**Z**-.5**I**.008**K**.05**PCO**	Cut a single pass thread

Threading, Multi Start

Multi start threading can be done by using the regular G33 command. Each start has to be it's 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".

FIRST PASS OF A MULTI-START THREAD
STARTS LIKE A STANDARD THREAD

← .2" →

EACH START AFTER THE FIRST IS
OFFSET IN Z BY (LEAD)/(# OF STARTS)

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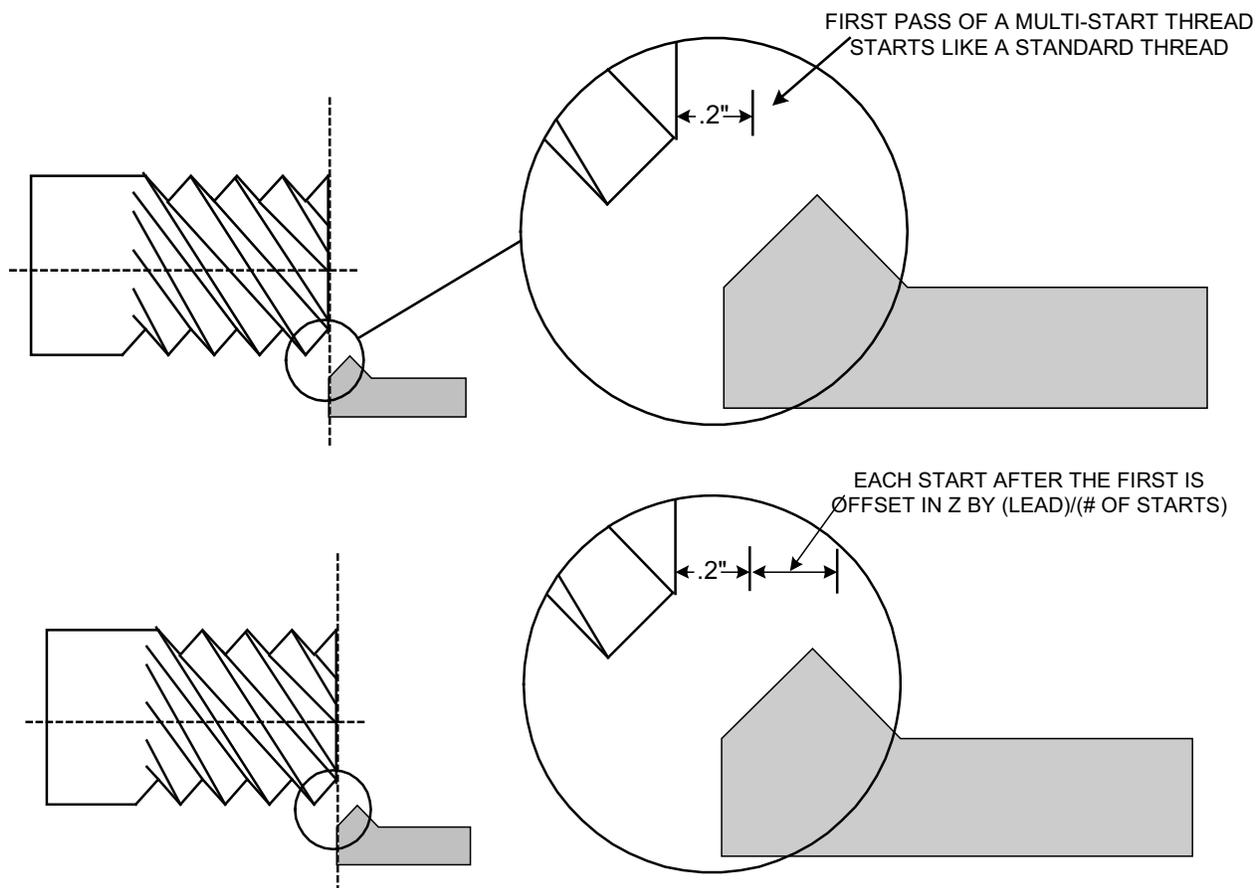
Value of Z:
The starting position of Z is offset for each start by (pitch)/(number of starts). If we have a three start thread the first start is at the normal .2" from the face of the material. The second start is offset by .2" + (.15/3) .25. The third start is offset by .25 + (.15/3) .3

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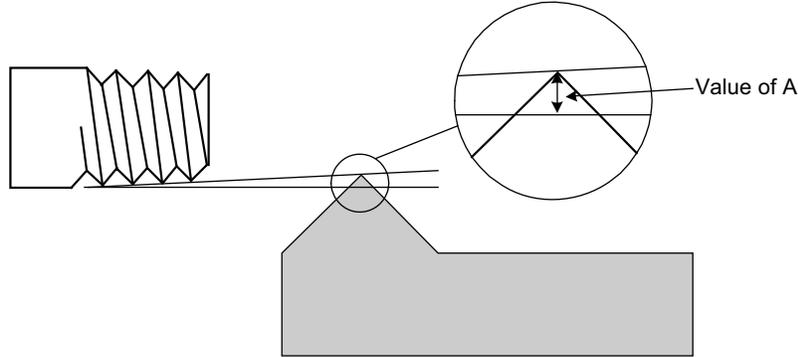
Remember that A is the amount over the total distance traveled, this has to include the .2" used to get up to speed.
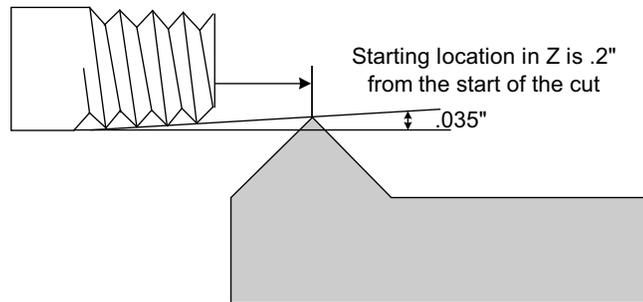
Example
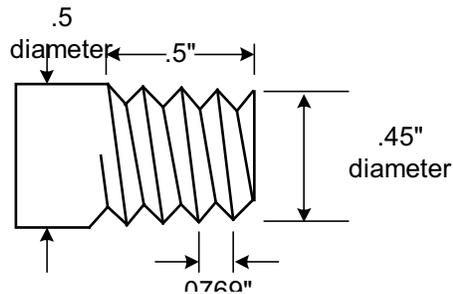

```
g90g94f300g72
t5
x0z.2
x.415                              Move to start corrected for taper (remember .2 in Z)
g33 x.365z-.5i.005k.0769a.035      Taper thread
```

"Do-it-Yourself" Threading w/P Suffix

Adding the letter P to the end of any move command causes that move to start in sync with the spindle.

You can make a do-it-yourself threading cycle by programming each cutting pass to start in sync with the spindle. This gives you more control over the depth of cut on each pass and the pullout position.

Putting the letter P on the end of the command line for any move causes that move to start when the spindle reaches its zero position.

With this feature, you can put threads on an arc if desired. Special 'bone-screws' sometimes require threading on portions of an arc. Also, you can do multi-pass thread-milling.

Here's a thread-milling program for doing a 2-56 thread. The thread mill head was .062 diameter, and it was set so that when the tool was on center the X dimension was .062. The mill is programmed it to take 3 passes just to illustrate how the P function works:

Since the thread-milling cutter is spinning pretty fast, you probably don't want more than 200-300 rpm on the spindle. This example assumes a .067 drilled hole with a depth of cut of .01" on the initial pass, and a thread length of .25".

```
T1
x.062z.5
z.2
x.077 (position to take .01" on 1st pass)
g04f.2(short delay gives a little settling time)
g95f.01785 (feed 1/56" per rev)
z-.25p(1st thd pass)
x.062 (retract back to center)
g94f100z.2(retract Z)
x.083(take a little less on the second pass)
g04f.2(settle for a while)
g95f.01785
z-.25p(2nd thd pass)
x.062(x retract)
g94f100z.2
x.086(last cut takes even less)
g04f.2
g95f.01785
z-.25p(last pass)
x.062
g94f100z.2
```

TOOL NOSE RADIUS COMPENSATION G41,G42, G40

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.

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 the radius value (R) stored with the secondary tool offset table. The direction of the offset correction is done with the X and Z values in the secondary tool offset table.

Format

G41 or G42	Specifies the type of compensation to be turned on (left or right)
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	(First move with secondary offset value)
...	(Other moves with secondary offset value)
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, and 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 twice 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 positive. It is the value of the tool nose radius. ie: a .007" radius tool gets a compensation value of .007
- 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.
- Tool changes automatically turn off compensation

TOOL NOSE RADIUS COMPENSATION G41,G42, G40

G41 and G42 Offsets Explained

Left or right compensation is determined by viewing the toolpath from the cutter's point of view. The tool is moved away from the part to accommodate the tool nose radius (see illustration below):

   If the tool is moved away from the part to the LEFT, G41 is used.
   If the toolis moved away from the part to the RIGHT, G42 is used.

A simple OD part is illustrated below and on the next page, using the same cutter on both sides of part, cutting both toward and away from the spindle. Below those illustrations is a simple back-turning example.
The canned cycle (g41 or g42) creates a toolpath that follows the contour of the part, offset by the "R" value set in the Secondary Offset Table.

To put the centerline of the tool nose radius on that toolpath, the program applies the "X" and "Z" values:

   If the tools is working the frontside of the part, the "X" will be minus.
   If the tool is working the backside of the part, the "X" value will be plus.
   If the tool is working a face away from the spindle (plus-z) the "Z" value will be minus.
   If the tool is working a face toward the spindle (minus-z) the "Z" value will be plus.

The sign (plus or minus) of the "X" and "Z" values is to move the tool toward the part the amount of the TNR.

Secondary offset:_
1 X: 0.04000 Z: -0.02000 R:0.02000

Secondary Offsets:
X *plus* twice TNR
Z *minus* TNR

G42:
Cutter is moved to RIGHT
(as viewed in direction of cut)

RIGHT

Secondary Offsets:
X *minus* twice TNR
Z *minus* TNR

G41:
Cutter is moved to LEFT
(as viewed in direction of cut)

LEFT

Secondary offset:_
1 X: 0.04000 Z: -0.02000 R:0.02000

TOOL NOSE RADIUS COMPENSATION G41,G42, G40

LEFT

Secondary Offsets:
X *plus* twice TNR
Z *minus* TNR

Secondary offset:_
1 X: 0.04000 Z: -0.02000 R:0.02000

G41:
Cutter is moved to LEFT
(as viewed in direction of cut)

Secondary offset:_
1 X: -0.04000 Z: -0.02000 R:0.02000

G42:
Cutter is moved to RIGHT
(as viewed in direction of cut)

RIGHT

Back-turning Example

Secondary offset:_
1 X: 0.02000 Z: 0.01000 R:0.01000

G41:
Cutter is moved to LEFT
(as viewed in direction of cut)

LEFT

Secondary Offsets:
X *plus* twice TNR
Z *plus* TNR

RIGHT

G42:
Cutter is moved to RIGHT
(as viewed in direction of cut)

Secondary Offsets:
X *minus* twice TNR
Z *plus* TNR

Secondary offset:_
1 X: -0.02000 Z: 0.01000 R:0.01000

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-Sec. Offsets 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
```

1. Select a secondary offset number
2. Enter the tool path correction. Enter the value with the correct sign. Refer to the previous pages. If the value should be minus, use the - sign. If the value is plus 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.

3. 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. *eg*: 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

G41 Example Program

The tool nose radius and the amount of offset is entered into the Secondary Offsets Table. Press F9 in Auto Mode to access the table.

An excerpt from the Secondary Offesets Table is illustrated below, along with an example program and toolpath.
In the Secondary Tool Offset Table:

   The "X" value is the diameter value (twice the TNR) to offset the tool in the X-axis
   The "Z" value is the amount to offset the tool in the Z-axis
   The "R" value is the tool nose radius

In the program the "D" value is the secondary offset tool that is called.
In the secondary offset table:

For this example:
X = -2xTNR Z = -TNR   R = TNR

```
1 X: -0.04000   Z: -0.02000 R:0.02000   17 X: +0.00000
2 X: +0.00000   Z: +0.00000 R:0.00000   18 X: +0.00000

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

Program:

```
g90g72g94f300
 m03s2000
t1
x-.04z.2 (call tool -x tnr
Ø)
 g41 (turn on lh tnr)
x-.04Z.05d1 (sec offset #)
 g95f.003
 x-.04z0
 x.190
 x.250z-.030
 z-.300
 x.37 (twice tnr Ø to clear)
g00z.2 g40 (turn off tnr)
 m30
```

0.030

0.300

0.020 (TNR)

0.250   0.190

This move occurs when tnr is turned on (g41)

TOOLPATH

PART

0.020 (TNR)

This move occurs when tnr is turned off (g40)

This page intentionally blank

G70 - G71 Inch - Metric Modes

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

G71 sets the control so that moves and feed rates are in "Metric" mode.
Use the G71 command at the beginning of the program before you call any tools.
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 milllimeters per revoltion (G95 mode).

NOTE: G71 is "modal" so the CNC will stay in metric mode until the control is turned off, then on again, or until G70 is issued. If your shop switches between modes often, it is best to include the G70 at the beginn ing of all Inch programs, and G71 at the beginning of all Metric programs to avoid errors.

I**f the CNC is used in a metric-only environment, the parameter file can be changed so G71 is not necessary.**

TO CHANGE THE PARAMETER FILE:

1. Drop to DOS (Ctrl-Q from Jog screen).
2. At K:\CNC> prompt, type C: then press enter.
3. At C:\RUNFILES> prompt, type edit filepath.cfg then press Enter to start OmniTurn editor.
NOTE: on older OmniTurns, you may have to type ew filepath.cfg.

The configuration file will be displayed in the editor. The file has 17 lines of configuration data, plus some explanatory text. The configuration data looks like this:

```
c:\pb35\utils\tcmp.~tool offset file
c:\pb35\utils\sectcmp.~secondary offset file
c:\programs\~directory of part programs
XZ~axes recognized(in alpha order)
100~max feedrate in enc. cts./ms
4000~max rpm
1~0 for single, maintained collet signal, 1 for pulsed
1~background color
15~foreground color
8~secondary foreground color
15~background color for active selections
1~foreground color for active selections
4~background color for banners
15~foreground color for banners
1~1 for bipolar spindle ref,0 for + only
0~0 for default inch mode,1 for metric
inv~for yaskawa;inv or baum for other spindles
```

Change the 0 at the start of the high-lighted line above to a 1. Be careful not to delete the ~ character, because the control software looks for it to mark the end of the data. Press F1, then F2 to save the file and exit the editor. Reboot the machine by turning it off, then back on. When it restarts, it will be in metric mode.

G72 - G73 Diameter and Radius Programming

G72 Diameter Programming Mode
G73 Radius 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, so be sure to use the G72 command in your program if you want to program in the diameter mode. Put the G72 command at the begining of the program before you call any tools.

G74 - Box Roughing Cycle

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

G74XnZnInUnFn

X*n* is the X corner of the box area to be roughed out
Z*n* is the Z corner of the box area to be roughed out
I*n* is the maximum amount to be roughed per pass, defined as the depth of cut per side
U*n* is necessary but ignored; adjust X and Z values to leave material for finish pass
F*n* is the feedrate
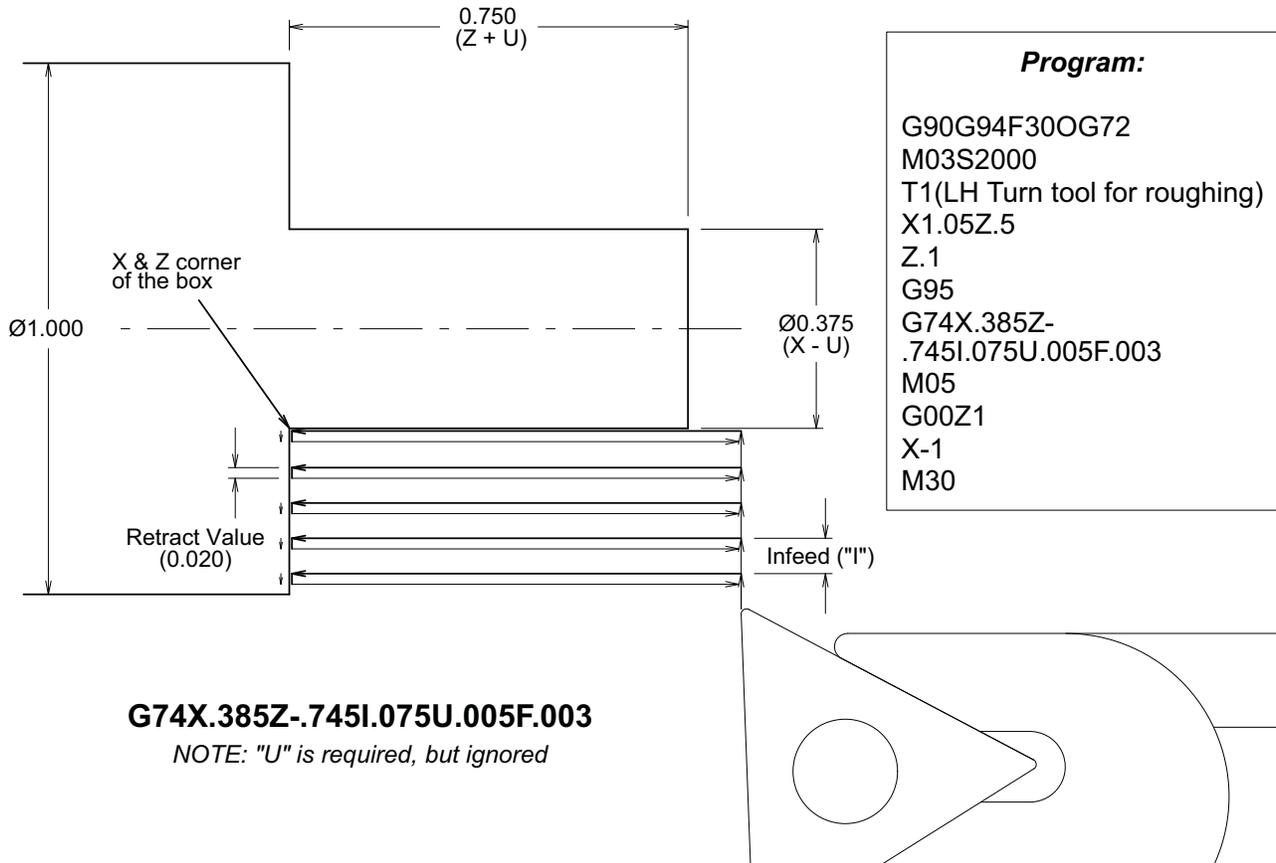
Program:

```
G90G94F30OG72
M03S2000
T1(LH Turn tool for roughing)
X1.05Z.5
Z.1
G95
G74X.385Z-
.745I.075U.005F.003
M05
G00Z1
X-1
M30
```

G74X.385Z-.745I.075U.005F.003

NOTE: "U" is required, but ignored

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.
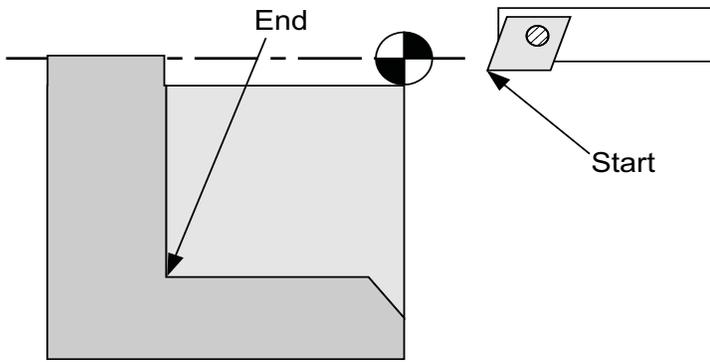
Internal example
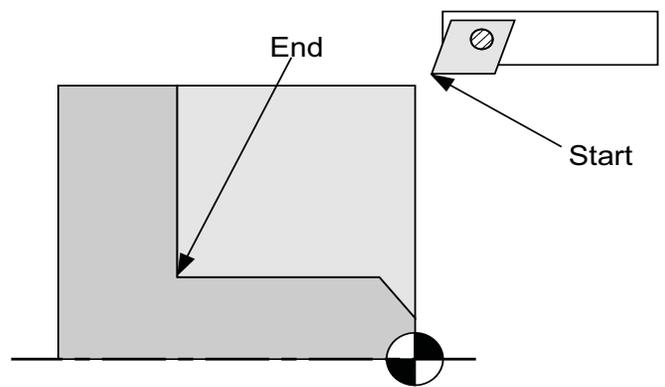
External example from the back
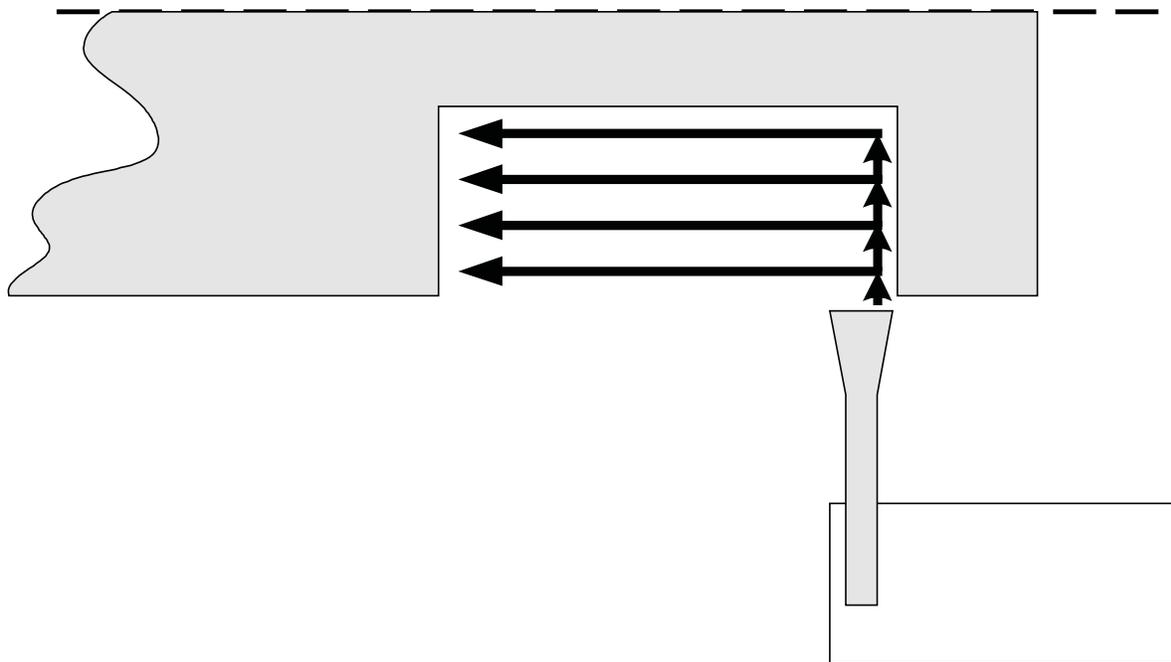
Example behind a shoulder

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.
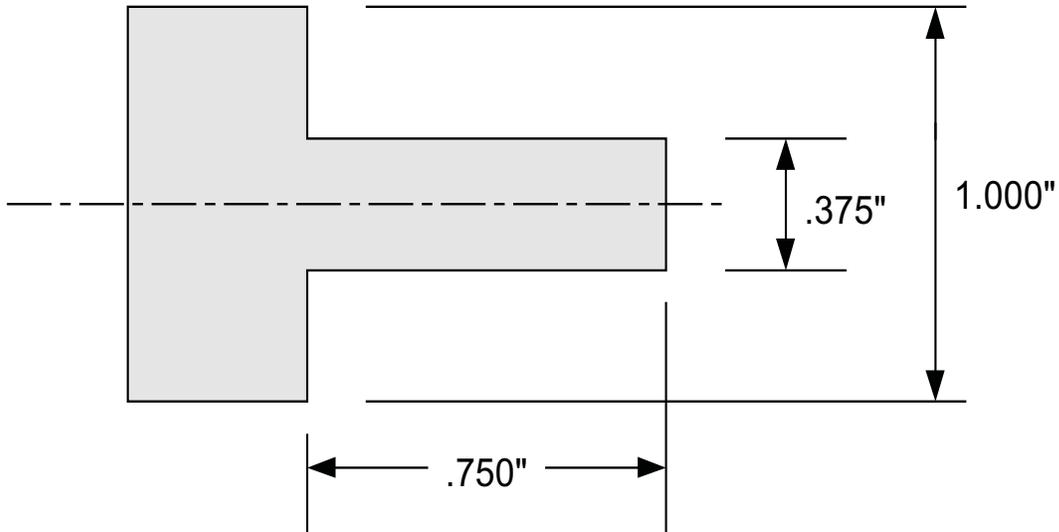

```
G90G94F30OG72
M03S2000
T1(LH Turn tool for roughing)
X1.05Z.5
Z.1
G95
G74X.375Z-.747I.075U.005F.003          programmed to leave .005 on X and Z for finish cut
M05
G00Z 1
X-1
M30
```

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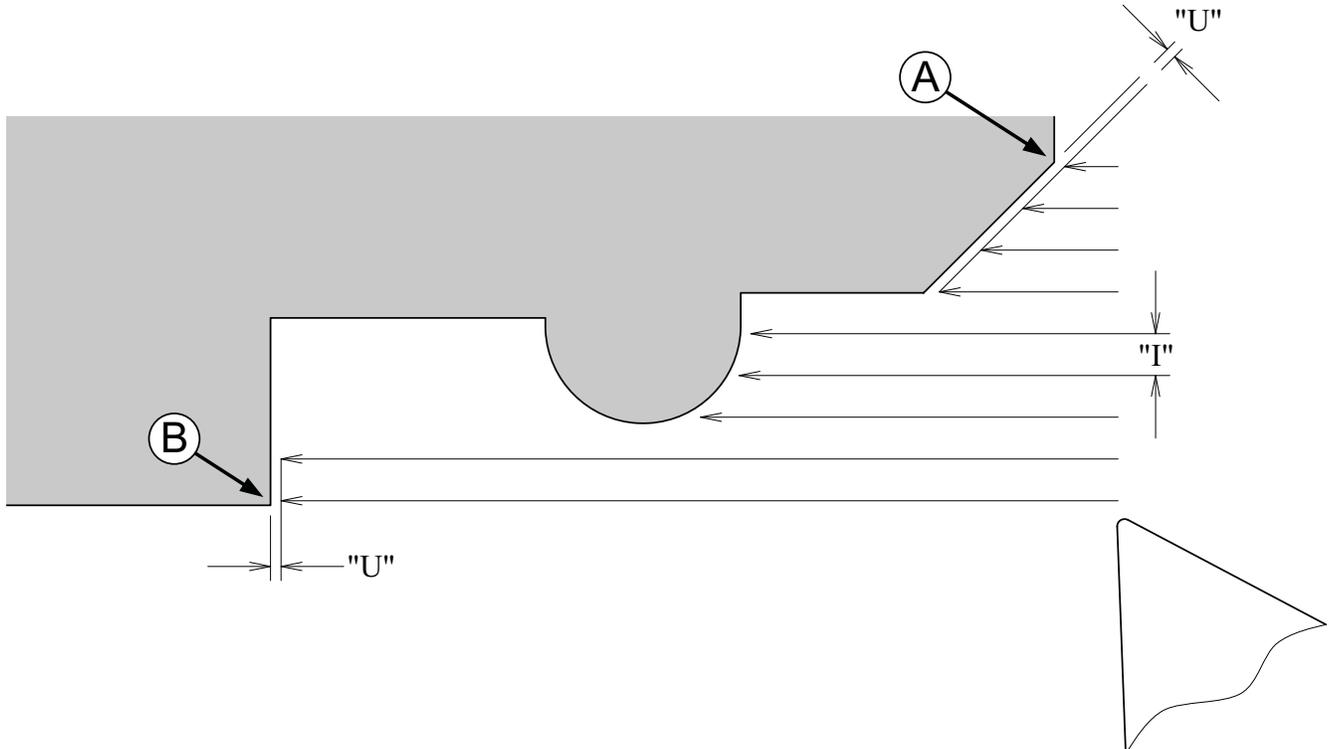
The box cycle starts at the current position, then makes cutting passes parallel to the Z axis at a cutting depth no greater than the I value until the last pass which is a U amount off the part. The area of material that is removed is rounded by the Z and X axis through the tool start point and the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

B   Start

A

Internal example

B   Start

A

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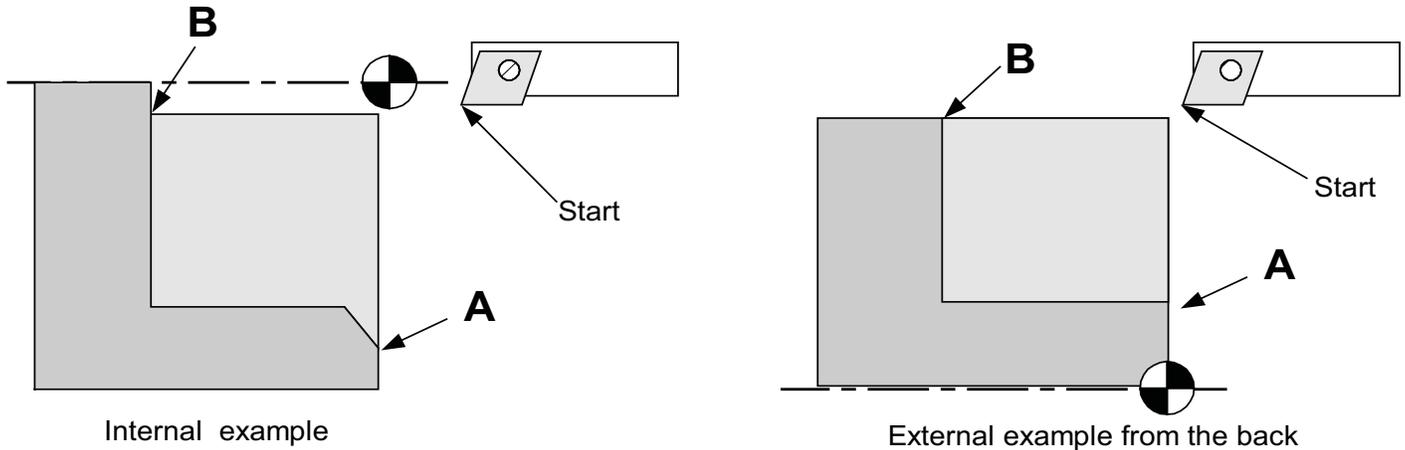
The G75 cycle can be used in radius (G73) or diameter (G72) mode. In both cases the In value is the amount to be taken off on a side. An In value of .05 would take .l off the diameter of the part with each pass.

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)

A

This is not allowed. The A point is
not in the same quadrant as B
relative to the start point.

B

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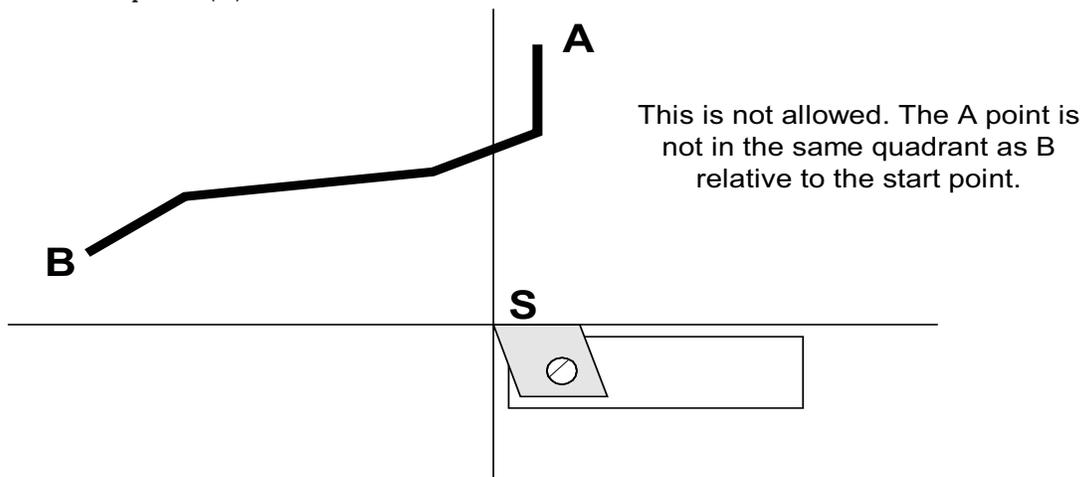
If the start point is moved over to include both A and B the cycle will work.

A

B

S

This is correct:
The A and B are in same quadrant
relative to the start point
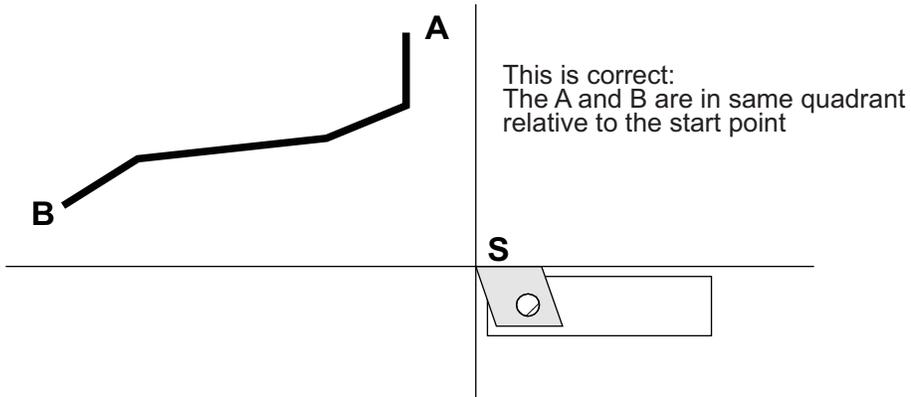
Worked external examples for G75

0.375

0.750

0.125 x 45°

0.343   0.406   0.593

R0.0935

0.300

0.487

0.750
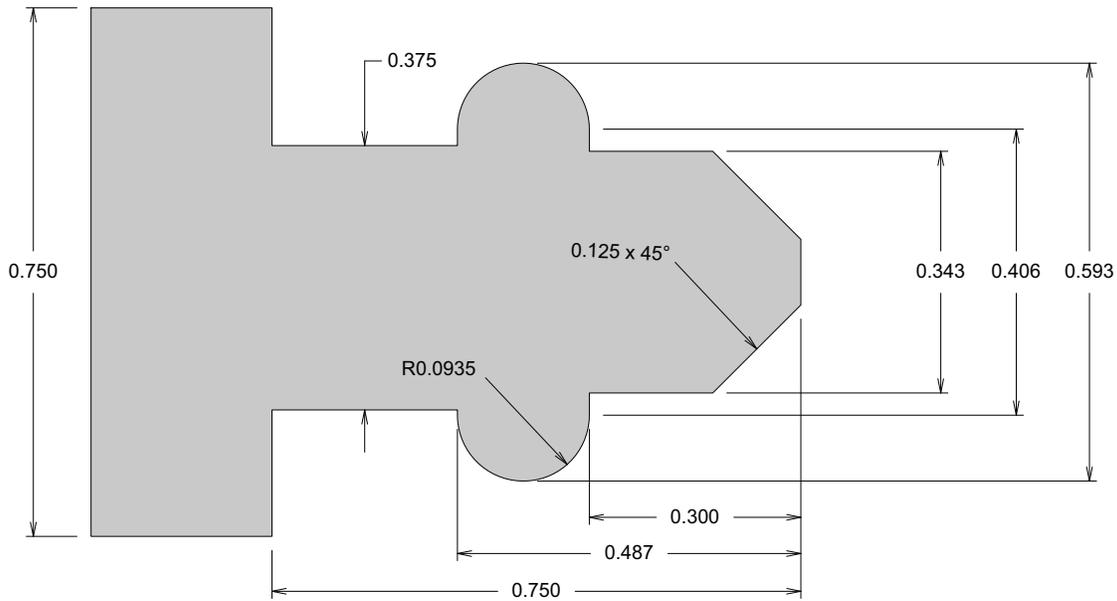
Above is a part that can be roughed. The code that can be used to generate the contour could be:

F4 Verify screen showing tool path

```
G90G94F300G72
M03S2500
T1(LH turn tool)
X0Z.1
g95f.003
X0Z0
X.343C.125
Z-.3
X.406
G02X.406Z-.487R.0935
X.375
Z-.75
X.75
m30
```


T1 tool path
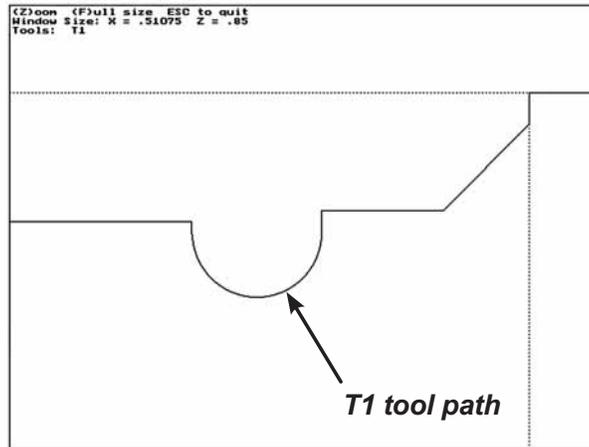
On the next page, the G75 is incorporated, to generate the roughing passes.

G75 - Box Contour Roughing Cycle continued

This example has the finished contour to be roughed immediatly after the G75 command:

```
G90G94F300G72
M03S2500
T1(LH turn tool)
X0Z.1
X.75Z.05
G95F.003
G75I.02U.005F.003 (Box contour cycle)
X0Z0
X.343C.125
Z-.3
X.406
G02X.406Z-.487R.0935
X.375
Z-.75
X.75
RF (End of box contour cycle)
m30
```

In the example below, 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
T1(LH turn tool)
X0Z.1
X.75Z.05
G95F.003
G75I.02U.005F.003P1 (Box contour cycle)
M05
G00Z1
X-1
M30
}1
X0Z0
X.343C.125
Z-.3
X.406
G02X.406Z-.487R.0935
X.375
Z-.75
X.75
M99
```

F4 Verify screen showing tool path

T1 tool path
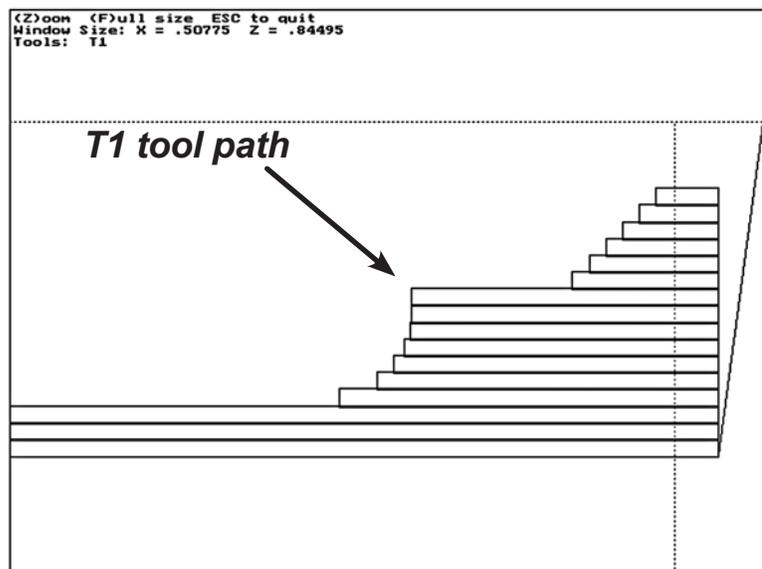
Worked internal example for G75

In this example there is a blank with a predrilled .4" hole.
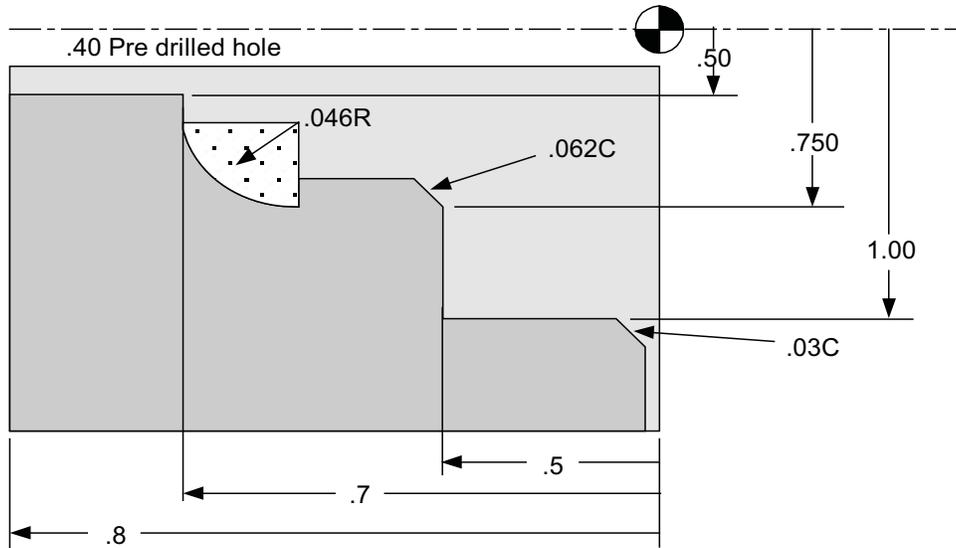
Notice the starting point is at the minor diameter of the finished bore, and the A and B points are at the starting and ending points of the finish contour. Also notice that the finish contour does not use C for the .03" chamfer. If this were needed in this box cycle you could add a facing move and chamfer to the contour pass.
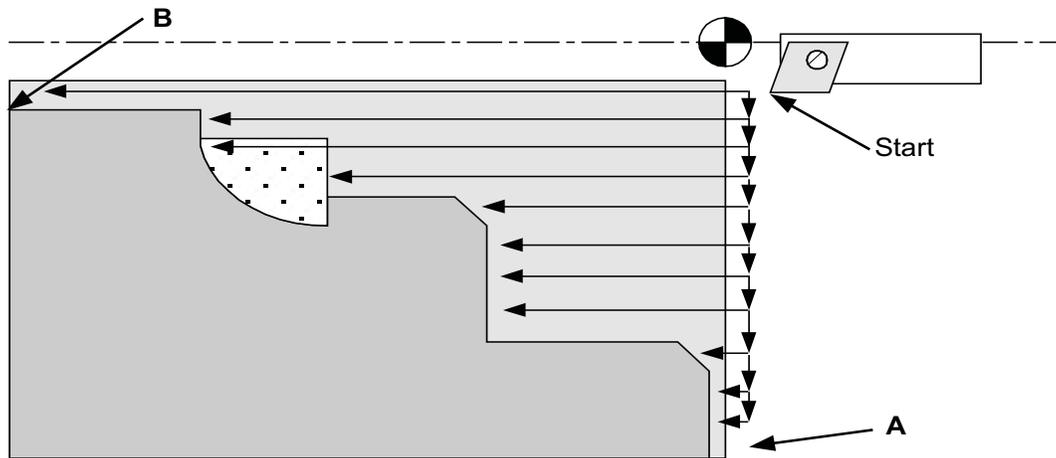

```
G90G94F300G72
M03S2000
T1
X.4Z.1
G95F.003
G75i.05u.02f.003
X 1.2Z0
X1.06
X 1 Z-.03
Z-.5
X.75C.062
Z-.704
G03X.658Z-.75R.046
X.5
Z-1.
RF
G00Z 1
M30
```

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 Rough Contour 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 cannot 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.
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 the same tool to take the rough contour pass, T1.

G90G94F300G72
M03S2500
T1(LH turn tool)
X0Z.1
X.75Z.05
G95F.003
G78 U.01 F.003 (Rough contour cycle)
X0Z0
X.343C.125
Z-.3
X.406
G02X.406Z-.487R.0935
X.375
Z-.75
X.75
RF
m30

F4 Verify screen showing tool paths

Finished part

T1 tool path

G78 - Rough Contour Cycle continued

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.

```
G90G94F300G72
M03S2500
T1(LH turn tool)
X0Z.1
X.75Z.05
G95F.003
G75 I.02 U.005 F.003 P1 (Box contour cycle)
G00Z.1
T3 (box tool)
x.75z.1
z-.497
x.6
g01
g74 x.395 z-.74 i.02 u.01 f.003 (Box roughing cycle)
T2 (LH finish turn tool)
X0Z.1
X.75 Z.05
G95F.003
G78 U.01 F.003 P1 (Rough contour cycle)
G00Z.05
X.75z.05
G95F.002
G78 U.001 F.003 P1 (Finish pass to depth with same tool)
G00Z.1
M30

}1
X0Z0
X.343C.125
Z-.3
X.406
G02X.406Z-.487R.0935
X.375
Z-.75
X.75
M99
```

F4 Verify screen showing tool paths

G78 tool path

G74 tool path

G75 tool path
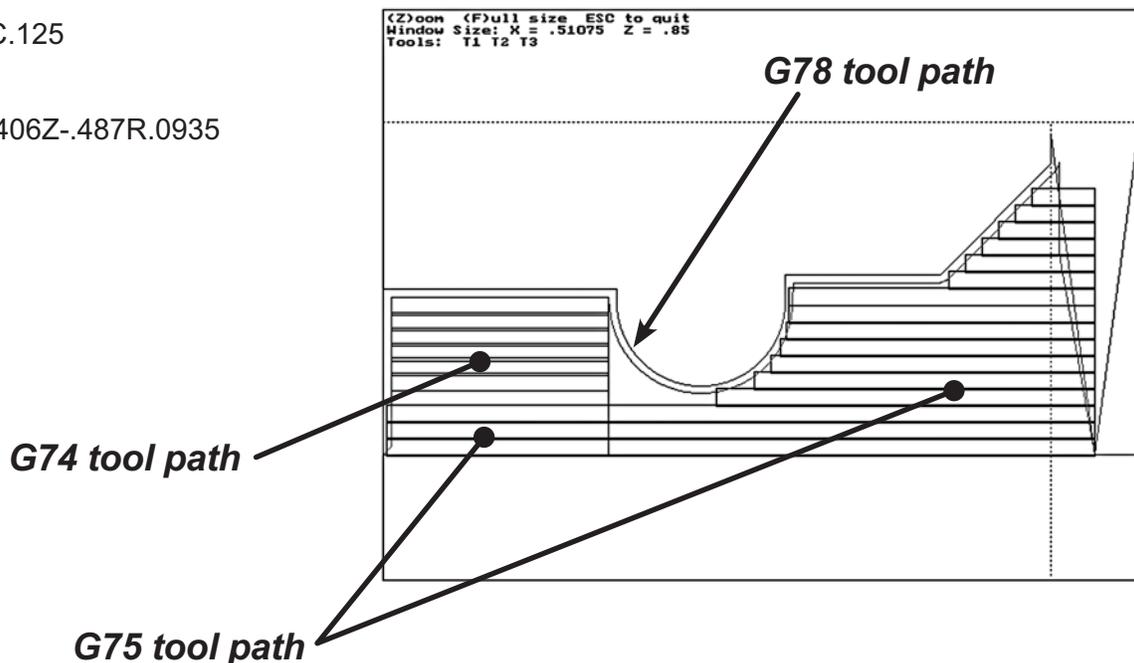
G81 Drill Cycle

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

Zn specifies the bottom of the hole from Z0 (usually the face of the part)

Fn is the drilling feedrate: inches per rev (g95) or inches per minute (g94)
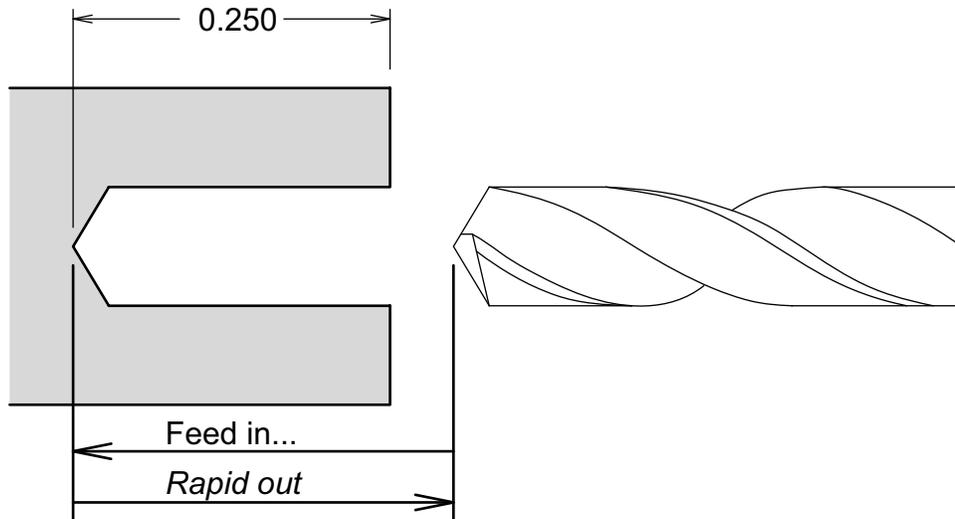
To drill a hole .250" deep at a feed of .005" per revolution:

g90g72g94f300	*Standard header: absolute, diameter, ipm, 300ipm*
m03s3000	*Set spindle on at 3000rpm*
t1	*Calls tool #1 offset*
x0z.05	*Positions tool at x=0, Z=.05*
g95	*Sets ipr mode*
g81z-.25f.005	*Drills the hole .250" 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

Zn specifies the depth of the hole from the Z0 on the part part (usually the face).

Kn specifies the depth of cut per peck.

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

Rn is the location in Z that the tool will retract back to after each peck.
Default is the starting point of the cycle

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

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

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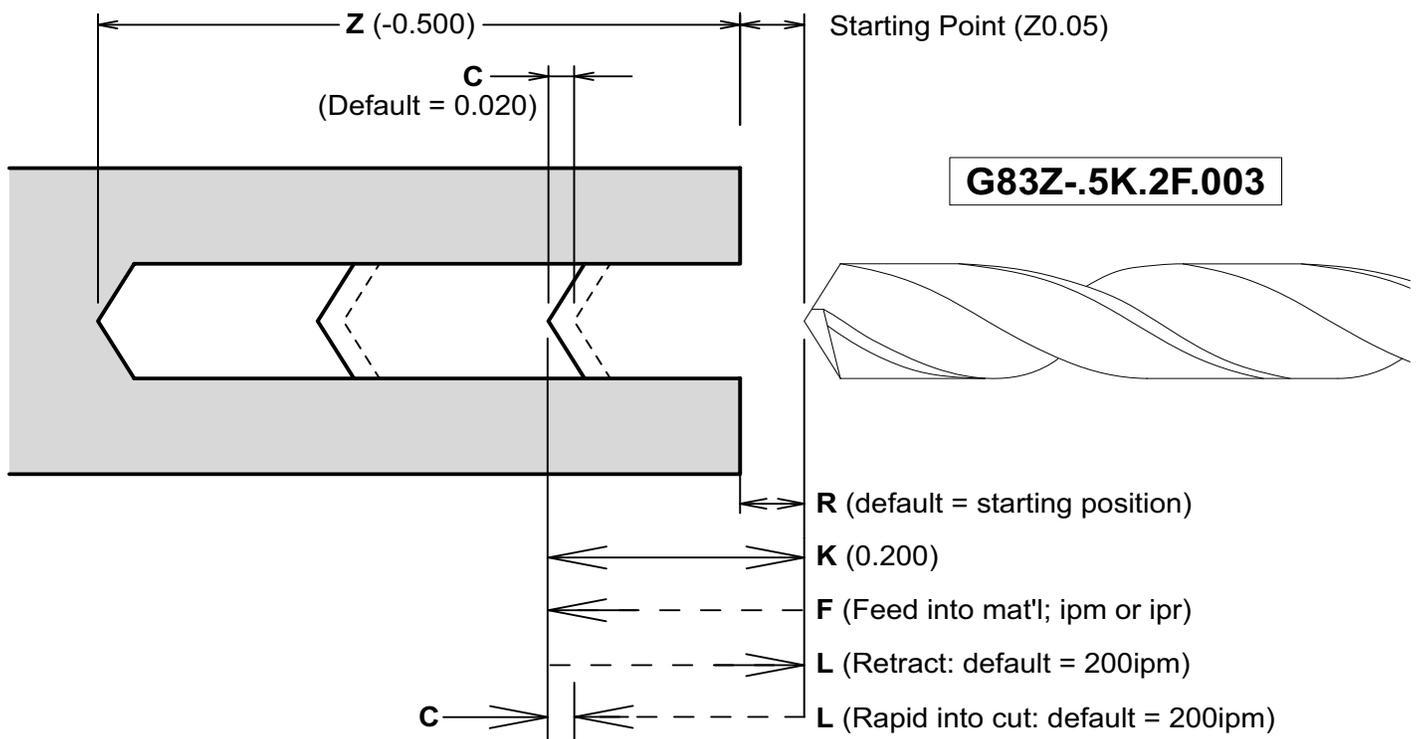
Start location: Position the drill where you want the first drill peck to start. After the first peck the drill will rapid out to the R location, and then back to where it started less the C value.

G83 Peck Drill Cycle

Z (-0.500) — Starting Point (Z0.05)

C (-0.040)

G83Z-.5K.2F.003R.01C.04L300

R (0.01)
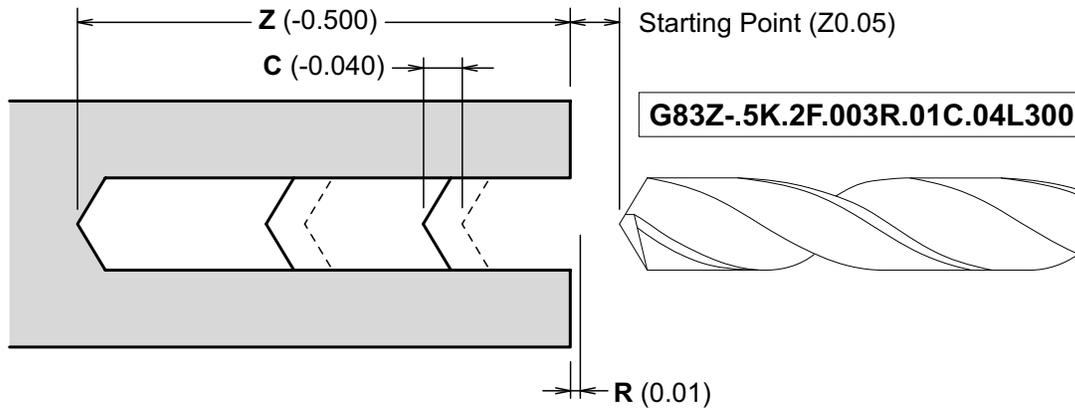
To drill a hole .5" deep at a feed of .003" per revolution, and .2" pecks. The program would be:

g90g72g94f300	Puts the control into absolute mode (g90)
T1	Calls tool #1 offset
X0Z.05	Positions tool at x=0, Z=.05
G95	Sets ipr mode
G83Z-.5K.2F.003R.01C.04L300	Z Drills the hole .5" deep
	K with .2" pecks
	F at .003 ipr
	R retract to .01 in Z to clear the chips
	C return to the cut less .040"
	L rapid moves done at 300IPM

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.

Z (-0.700)

R (0.400)

0.490
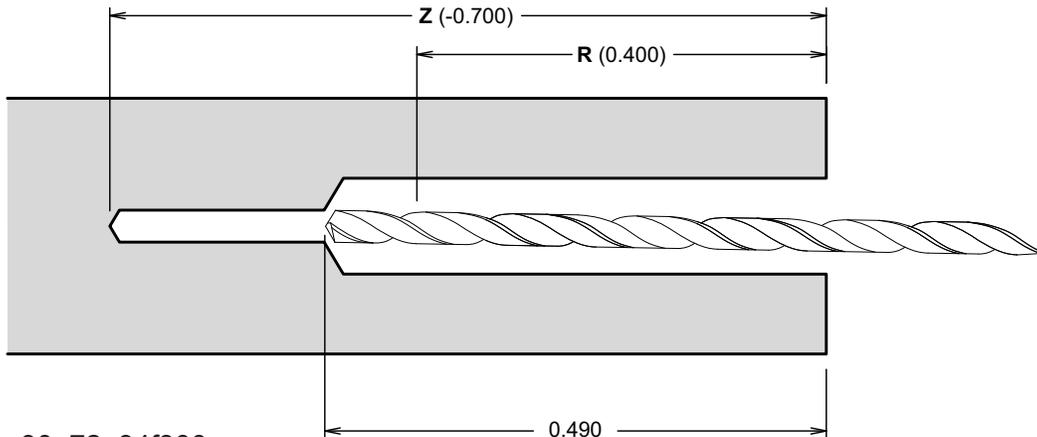
```
g90g72g94f300
T1 (3/32 drill)
X0Z.05
G95
G83Z-.5K.2F.003R.01C.04L300
G00Z1
T2(1/32 drill)
X0Z 1
Z-.490
G83Z-.7K.1F.002R-.4C.05L300
```

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: locations given will move the tool relative to an absolute zero, usually the face of the part (Z) and the center of the part (X).

G91 - Incremental mode: locations 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, but the position s creen will change to agree with the g92 value.

On the earliest OmniTurns, when a tool was called, the position screen showed its position relative to machine "home", g92 was required in order to set the position screen to show the tool position relative to the part.

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 0.1"/rev

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

There are two types of spindle speed control modes that the OmniTurn control can use:

G97: Spindle speed in RPM (default) 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.

G96: Constant Surface Feet 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
G95F.002X0
G94F300Z2
G97             (Switch to RPM mode)
S2000           (Set spindle speed at 2000 rpm)
T2(DRILL)
X0Z.2
G95F.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
When the control encounters the M00 it will stop and wait for the operator to press "CYCLE START". This command is 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
This stop command will be skipped over (ignored) if the "Optional Stop" switch is turned on.
To turn the optional stops on go to Automatic mode once the program is selected and before the program is run, and press "O". This will cause the program to stop like a M00. To get past the stop press cycle start. To turn off the optional stop (to ignore it) press "O" again. *(See illustration below)*

 Uses for the optional stop:
> 1. Insert an 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.
> 2. Have an M0l 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                c:\programs\newprog3          Position
X: +0.00000                                        X: +0.00000
Z: +0.00000                                        Z: +0.00000


Switches                                           Feed
C.Cycle Repeat                                        0.000   IPM
/.Block Skip                                        100%
O.Optional Stop
S.Single Block       Press Cycle Start to Begin

                                                   Speed
                                                      0  RPM
Functions                                          100%
F2-Tool Offsets                                    Tool  0
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
```
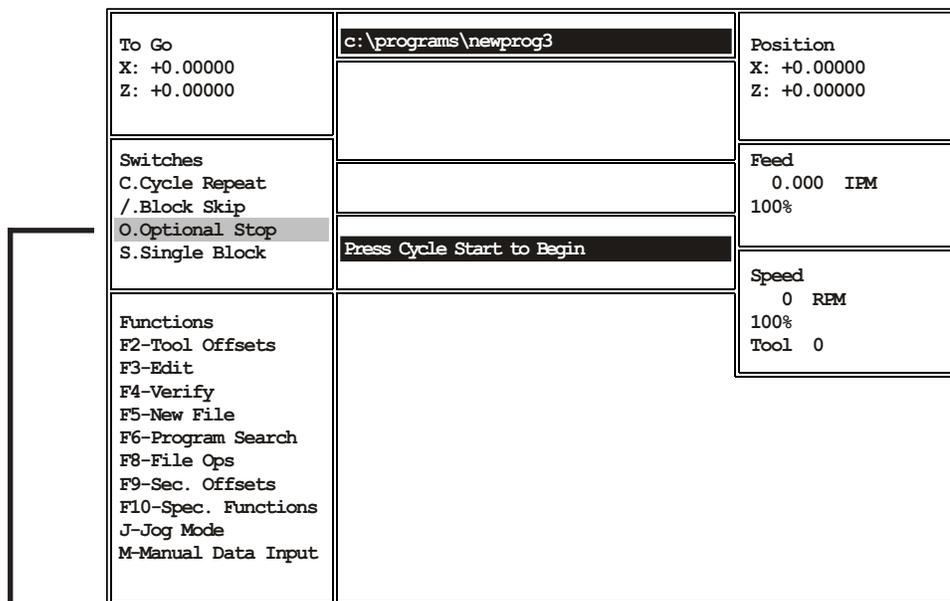
See Section Five, Automatic Mode, page 5.6 for more detail about these switches

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

M08, M09: Coolant control
M08 Coolant on
M09 Coolant off

M12, M13: Collet clamp control
M12 clamps the collet
M13 unclamps the collet

M17, M21, M23, M27: User assigned M functions (optional)
These optional M-codes allow multiple user functions to be CNC controlled. They can work with the M91 - M94 input M-functions (next page) to coordinate auto-loader activity. Three-pole double throw relays in the spindle cabinet are controlled by these M-codes. You can wire to the relays for optional live tools or air blast or any other on/off device. The relays will handle up to 5amps at 220vac three phase.

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.

M80, M81: Cutoff Cycles (Optional VCO)
M80 and M81 are two separately programmable cutoff cycles for use with the optional Vertical Cutoff (VCO). Diameters, feed-rates and dwells are set for each cycle and invoked in program.

M82: Wait for Cutoff Complete (Optional VCO)
Other machining operations can be done concurrent with cutoff cycle. Put M82 before operations that could cause interference untl cutoff cycle is complete; program dwells until cutoff is complete.

M84 / M85: Lock / Unlock Spindle Speed & Feed Override Pots
Use M84 to set both override pots to 100% and to prevent their adjustment. M85 re-enables.

M87 - Real-time Work Shift from digital indcator (optional)
This code is used to set work shift according to input from digital indicator (see page 2.22)

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 pages 6.25 - 6.27 for 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 ar 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: Mitusbishi 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.

M32, M33, M38: M-functions used by PLC Expanded I/O Option
M32 sets PLC ouput Y0 on; M33 sets PLC ouput Y1 on. Can be used to set air tools on.
M38 sets PLC outputs Y0 and Y1 off. Can be used to set air tools off.

Program commands - M-codes for PLC and Auto-loaders

M60 / M61: M-functions used by PLC Expanded I/O Option
M60 sets PLC output Y3 on; M61 sets PLC output Y3 off. Can be used to enable bar pusher.

M62: M-function used by PLC Expanded I/O Option
M62 pulses PLC output Y2 for 1/2 second, then waits for PLC input X7 (End of Bar signal)

M64 / M65: M-functions used by PLC Expanded I/O Option
M64 sets PLC output Y5 on; M65 sets PLC output Y5 off.

M10, M11, M40 - M59: M-functions used by ZipLoader, Rear Loader and Front Loader
The **ZipLoader** is a magazine bar loader that is also a shaft or bar loader.
The **Rear Loader** is a vibratory bowl fed fastener loader for working the shank of headed fasteners.The **Front Loader** is similar to the rear loader, but it puts the parts into the collet from the front, for working the head.

Custom M-Functions

Editing & Writing .usr files to access the additional I/O:

When the optional PLC is installed, you can create custom M-Functions to interact with the PLC. A simple programming language sets, resets or pulses outputs, waits for intput states and sets, resets, pulses and examines internal PLC memory bits.

To edit or create a usr file on the OmniTurn:

1. Drop to DOS (Ctrl-Q from Jog screen).
2. At K:\CNC> prompt, type C: then press enter.
3. At C:\RUNFILES> prompt, type edit m??.usr (where m?? is the number of the M-function) then press enter to start OmniTurn editor. NOTE: on older OmniTurns, you may have to type ew m??.usr.

Edit the program as usual, save and exit, then restart the OmniTurn.

There are only ten commands in the 'usr language':
(*n* is the number of the output, mark, input or length of time)

setbn	Turns an output ON: *setb10 turns on output Y10*
setmn	Turns a mark ON: *setm7 turns on M7 in PLC*
clrbn	Turns an output OFF: *clrb10 turns off output Y10*
clrmn	Turns a mark OFF: *clrm7 turns off M7 in PLC*
waitonn	Stops the usr program until an input turns ON: *waiton14 waits for X14 to go on*
waitmonn	Stops the usr program until a mark in the PLC is set: *waitmon8 waits for M8*
waitoffn	Stops the usr program until an input turns OFF: *waitoff14 waits for X14 to go off*
waitmoffn	Stops the usr program until a mark is cleared: *waitmoff8 waits for M8 to clear*
delayn	This command stops the usr program for a number of seconds, from .1 sec to 10 sec: *delay.3 waits for 3/10 of a second.* Used for pulsing an output, or for waiting a while after seing an input before setting an output .
end	This command is the last command in every usr program.

Spindle Control

Cycle Start

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

**Cycle Start from Operator's station
Must press both buttons at once.**

**Open Collet**   **E-Stop**   **Close Collet**
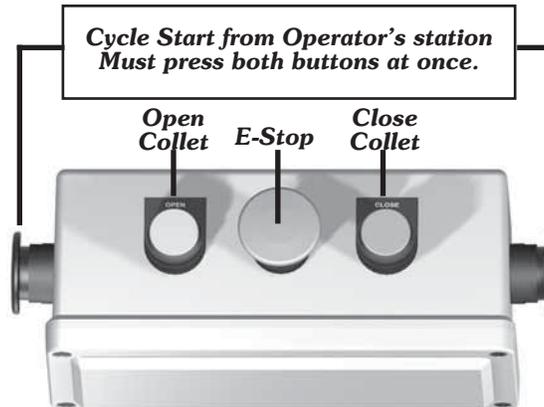
Controlling the spindle:

Manual control of the spindle

You can turn the spindle on manually. This is useful for setting up tools or 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, for example. To turn the spindle off but leave the command active, set the SPINDLE "AUTO" / "OFF" to OFF. You can leave MDI and go to the Jog mode by pressing Esc key. Now you can turn the spindle on and off with the switch. Use the override knob to adjust the speed.

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

Dial for spindle speed override

Switch to turn spindle off/on manually
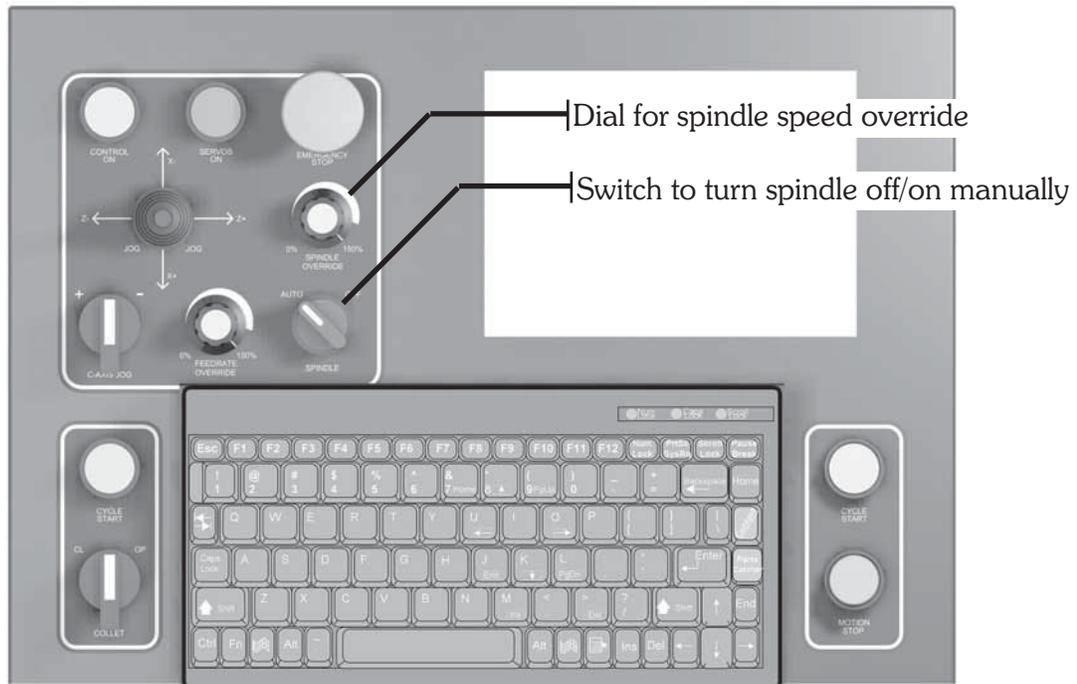
To review the functions of all the front panel controls, refer to pages 1.4, 1.5 and 1.6 in Section 1.

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: **M03**S2000

      Spindle on, reverse: **M04**S2000

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 M03 or 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-.25S1500

Manual override:

      There is a knob on the face of the OmniTurn control that will override the spindle speed selected. It changes from 0 -150%. Note:150% will not exceed maximum rated speed.

Tapping with the spindle control:

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
.
.
```
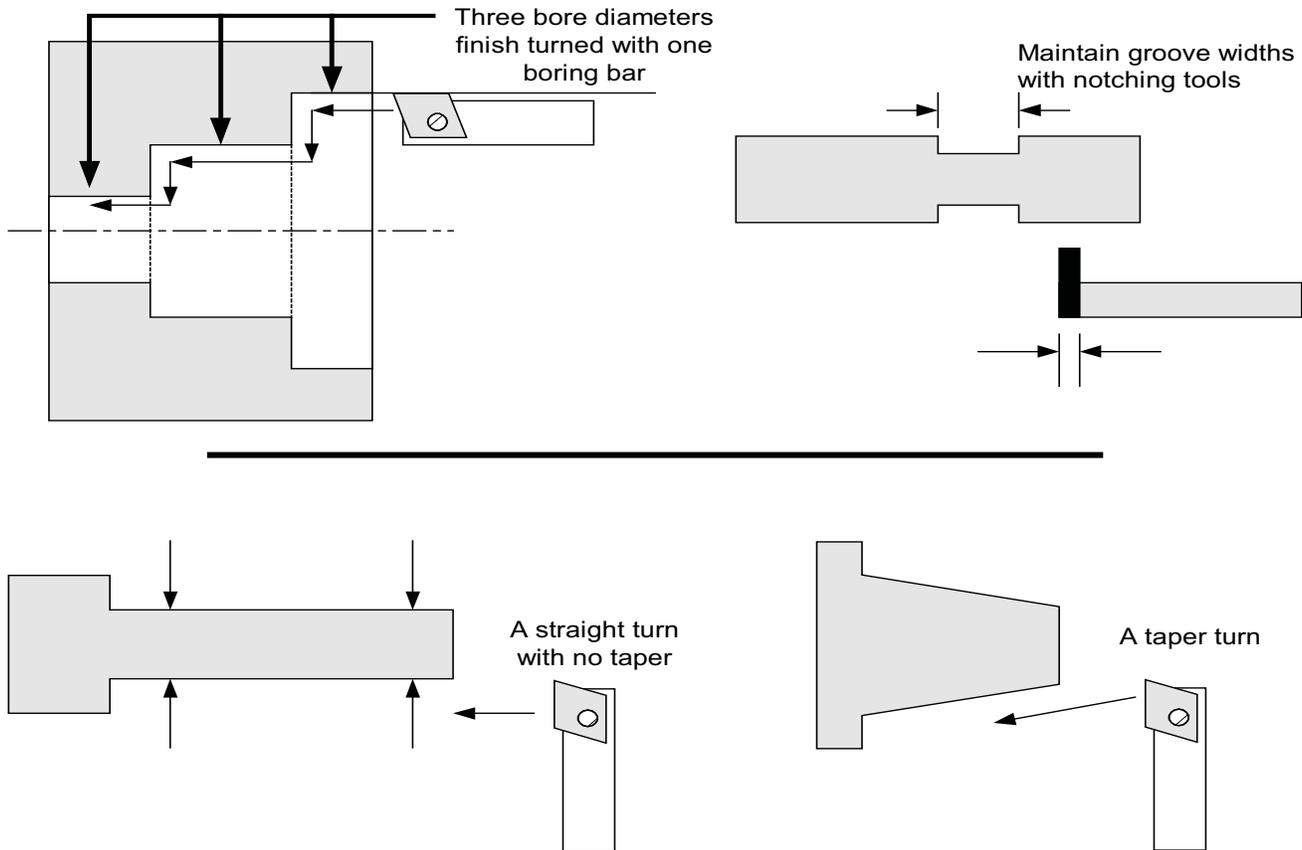
** This dwell is used to give the spindle some time to reverse before starting the tap back out. During this dwell, the slide is stopped, but the spindle is slowing down in the m03 direction, then speeding up in the m04 direction. The floating tap holder allows the tap to move in the thread as the spindle 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.

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.

Three bore diameters finish turned with one boring bar

Maintain groove widths with notching tools

A straight turn with no taper

A taper turn

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)
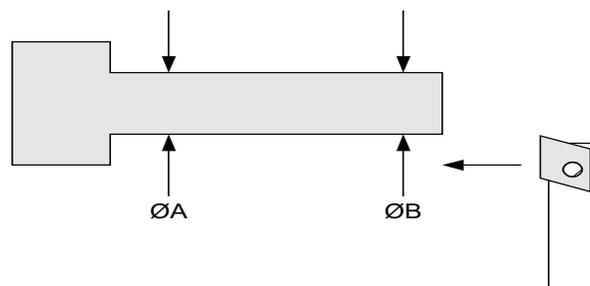
Secondary Offsets

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:

```
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:
```
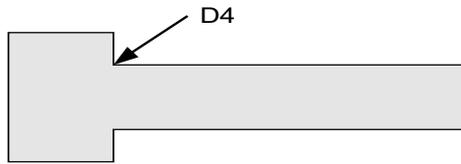
Look at the following example:

ØA   ØB

With this shaft, 0A diameter = 0B diameter. In order to do this, put an offset command at the end of the turn.

Secondary Offsets continued

The code to face and turn the shaft is:

Without correction	With correction	
X0Z0	X0Z0	(move to the center and face)
X. 125	X. 125	(move to the diameter)
Z-.75	Z-.75D4	(turn the diameter, corrected)
X.2	X.2D0	(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 press 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 OmniTurn will re-interpret the line "z-.75d4" as "z-.75x0.1245", reducing the diameter by 0.001 as the tool moves toward Z-0.750.

Canceling a secondary offset

The secondary offset correction stays in effect until:

• There is a tool change. This cancels an offset

• Calling up secondary offset D0 will turn off the offset.

• Calling up another offset will cancel the original offset and set the new offset

Secondary offset examples:

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

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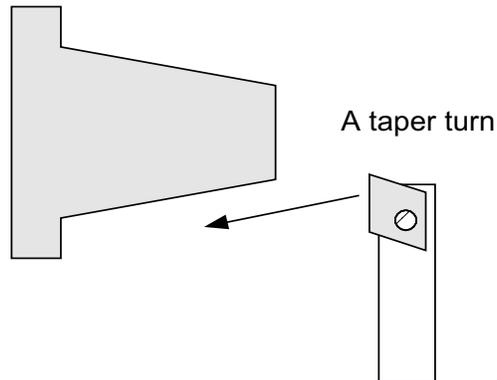
The actual taper that a tool cuts will depend on the toolnose radius. If this is not an easy tool to maintain then the taper will vary as the tool changes. Normally this would be a big problem. However with the secondary offsets this is very simple. We could put a secondary offset at the end of the taper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

D5
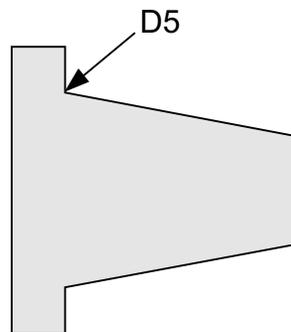
The commands might look like this:

    X0Z0   (Move to the center and face of the part(
    X.2   (Move to the minor diameter of the taper)
    X.3Z-.5**D5**   (Move to the major diameter of the taper, corrected with D5)
    X.4**D0**   (Move 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
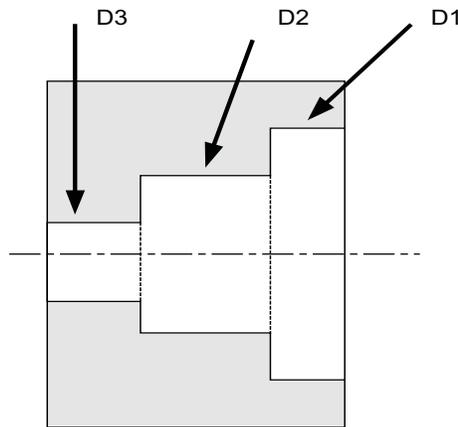• The location in Z of the taper. -D5 Z value

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

Secondary Offsets continued

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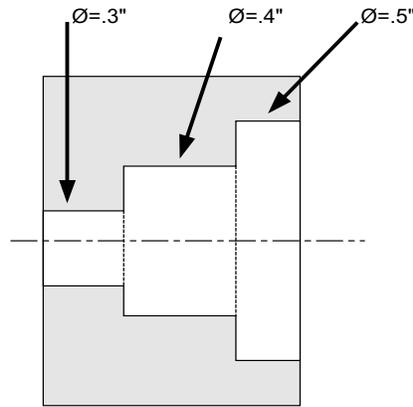
For this example, three diameters have to be turned with only one tool. Each of the diameters can have an individual offset.

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.

Spindle Positioning

Spindle positioning system specifications

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 ±0.2° 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. The C-axis encoder provides 36,000 counts per revolution, but the *feed rate* is still 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:

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

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. Feed must be specified as G94.

Jogging and setting Custom Home Position:
Refer to 1.14 for location of Jog control.

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 posisition ±.02° whenever C-Axis is enabled, manually or by M19.
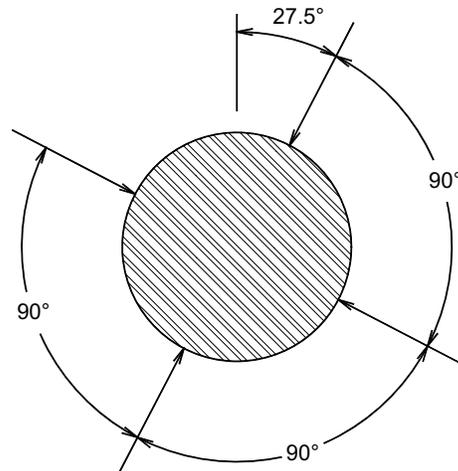
Spindle Positioning

Example showing positioning and cross drilling

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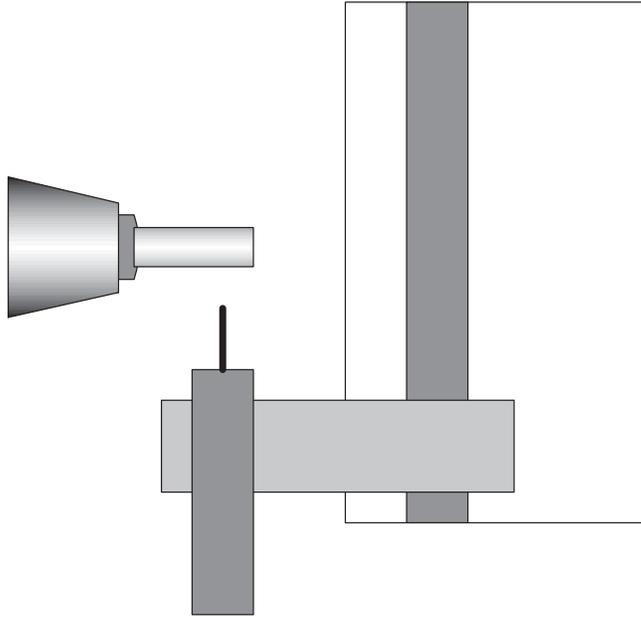
The slide will be used to drill the holes. We will drill (4) holes 90° apart, the first hole is located at 27.5° from a reference 0°.
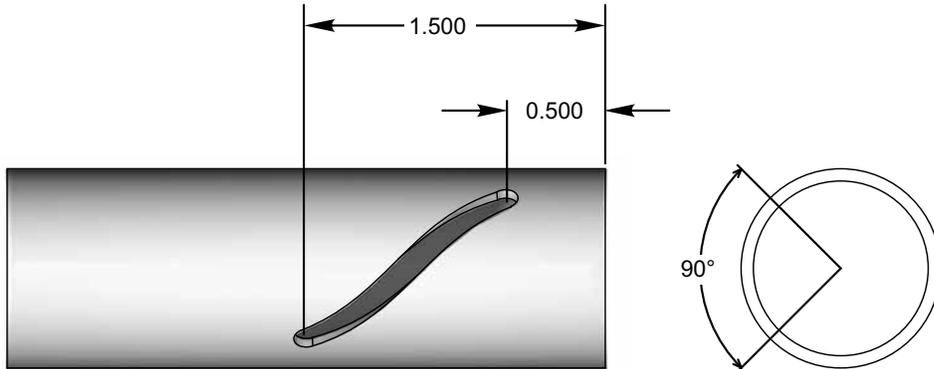

g90g94f300g72
t32 *(live drill from the side)*
x.75z-1
z-1.3
m27 *(turn drill on)*
m19 *(orient the spindle to 0°)* ⟶
ca27.5 *(position spindle at 27.5° absolute)*
ls4 *(start of loop)*
g94f10x1 *(drill a hole)*
x.75f10
f300 (position spindle at max feedrate)
ci90 *(rotate 90° incrementally)*
lf *(end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

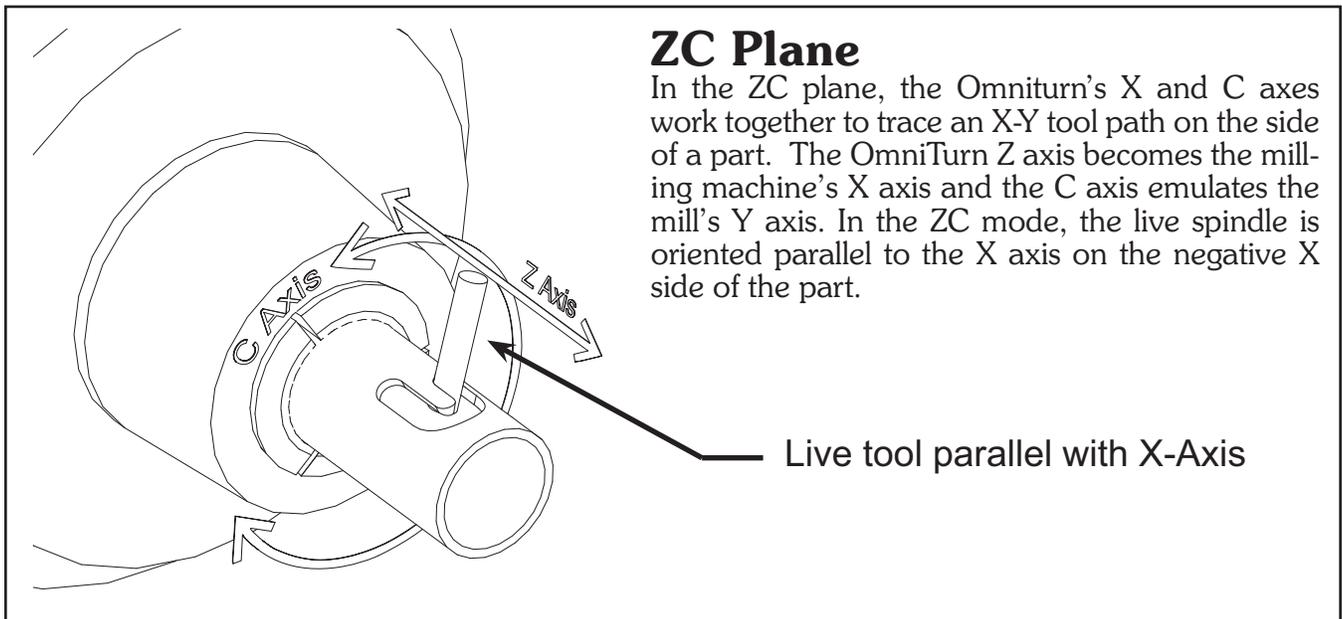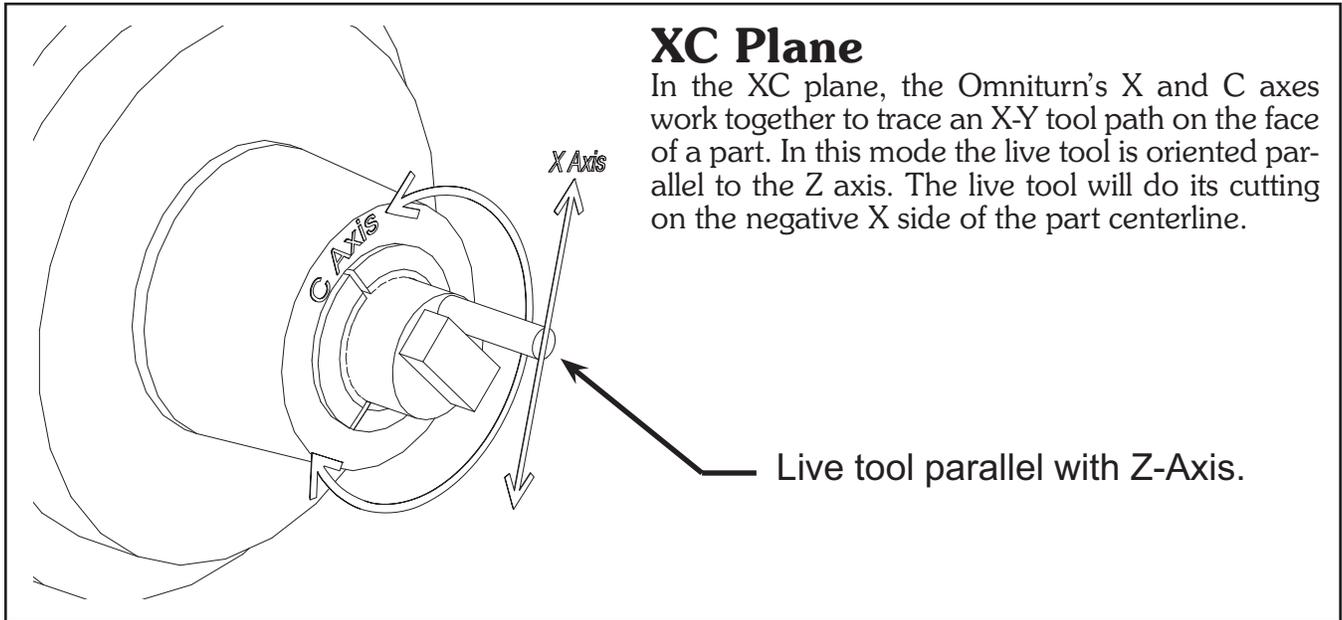
Use C-Axis mode to mill the slot as a lead:

g90g94f300g72
t32 *(Live mill from the side, below part)*
x.75z-1 *(Move to safe position)*
z-0.5 *(Move to Z dimension)*
m27 *(Turn mill on)*
m19 *(Positioning Mode: locate spindle at 0°)*
x.5f3 *(Feed mill into part at 3ipm)*
ca-90z-1.5f5 *(Rotate spindle m04 direction 1/8 turn and move 1.000 in Z at 5ipm)*
f100x.75 *(Clear tool)*
m28 *(Turn mill off)*
g00z0 *(Clear tool)*
m30 *(End program)*

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:

XC Plane

In the XC plane, the Omniturn's X and C axes work together to trace an X-Y tool path on the face of a part. In this mode the live tool is oriented parallel to the Z axis. The live tool will do its cutting on the negative X side of the part centerline.

Live tool parallel with Z-Axis.

ZC Plane

In the ZC plane, the Omniturn's X and C axes work together to trace an X-Y tool path on the side of a part. The OmniTurn Z axis becomes the milling machine's X axis and the C axis emulates the mill's Y axis. In the ZC mode, the live spindle is oriented parallel to the X axis on the negative X side of the part.

Live tool parallel with X-Axis

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.

(X*nn*): moves the X axis to the specified coordinate

(Y*nn*): moves the Y axis to the specified coordinate

(G02X*nn*Y*nn*I*nn*J*nn*): 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.

(F*nn*): feedrate specified in inches per minute

(Z*nn*): 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.

(H*nn*)(XC plane only)**:** how far the the centerline of the live tool is above 1/2"; should be first block after (~PLXC). Easily determined by cutting two slots in face 180° apart. Measure outside distance, subtract tool diameter, *then divide by two.*
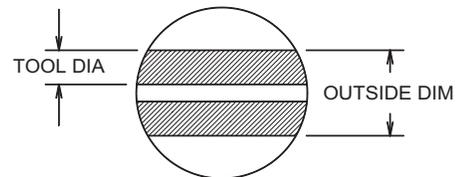
(D*nn*)(ZC plane only)**:** specifies the part diameter at which the Y dimensions are translated to C axis rotation.

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.
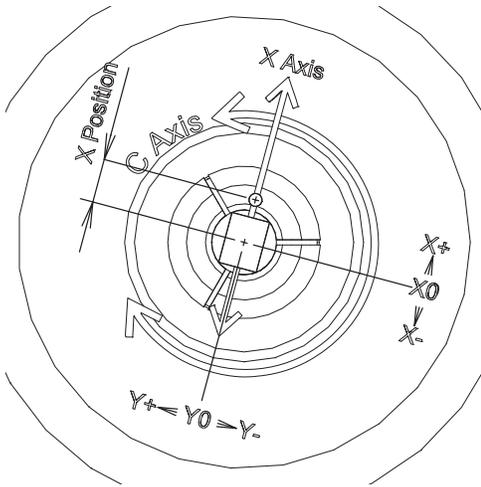
(END): cancel plane-switch mode

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
```


Part Dimensions

Tool Path
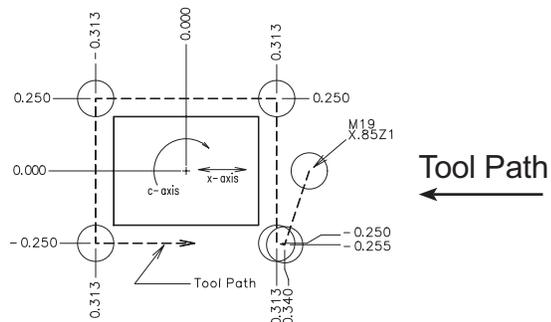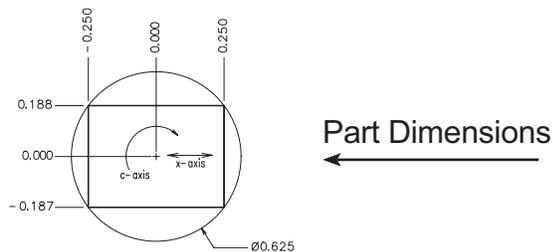
C-Axis XC Plane Switching: Programming continued

These 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

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 program cuts an oval 5/8" long by ¼" wide.

```
g90g94g72f300
m27
m19
t21
x-.85z1
f100z.05
f5z-.1
(~PLXC)
(x.375y0) right end of oval is .3125" from center, +.0625 cutter radius
(g03x.1875y.1875i.1875j0) ccw turn 90 deg w/.1875 radius,
(x-.1875)
(g03x-.375y0i-.1875j0)
(g03x-.1875y-.1875i-.1875j0)
(x.1875)
(g03x.375y0i.1875j0)
(x.4)
(end)
f200Z1.5
m30
```

Part Dimensions →
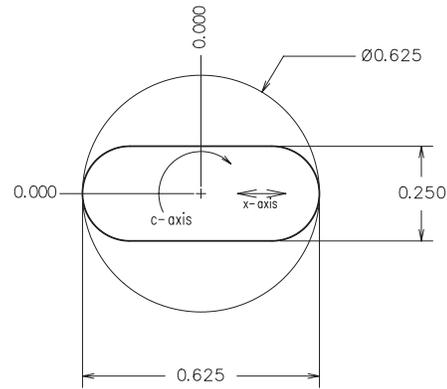
Tool Path →
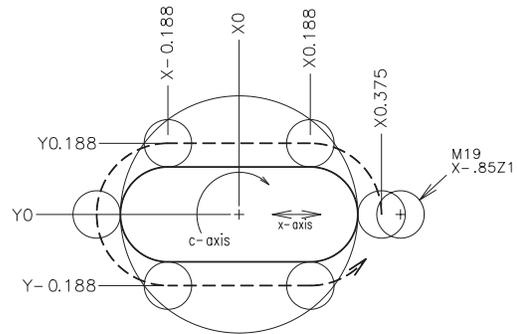
This one uses looping to cut 6 flats.

```
g90g94g72f300
m03s1000
g04f5
m05
m27
m19
t21
x-.85z1
f100z.05
f5z-.1
ls6
(~PLXC)
(x.38465y0)
(x.19235y.33315)
(end)
x-.8
lf
m05
f200Z1.5
m30
```

As each flat is cut, the spindle turns 1/6 of a revolution; canceling plane-switch mode does not cause the spindle to move, so on the next iteration of the loop, the next flat is cut.
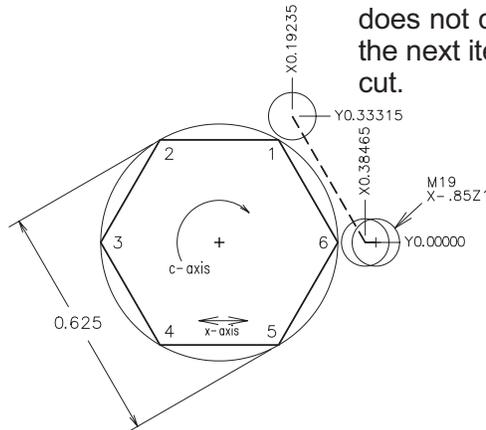
Spindle Positioning

C-Axis Plane Switching: Using the ZC Plane

In this mode the live tool works on the side of the part.
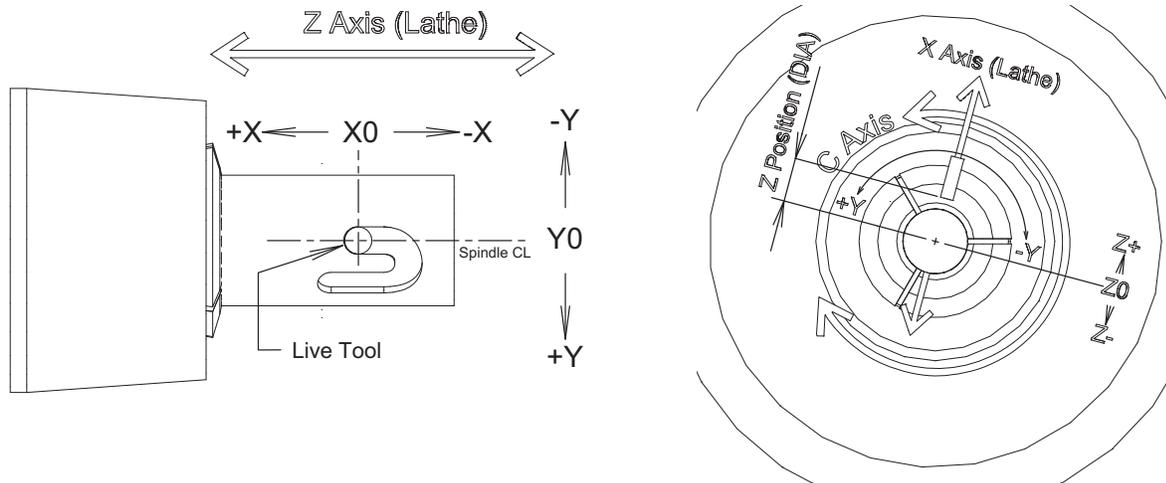
The ZC mode is started with the command (~PLZC).

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

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 ½" square with a .2" diameter circle in the lower left corner.

> g90g72g94f100
> m27
> t11
> x-1z1
> z-.1187 *(x0y0 on virtual mill puts edge of square at Z-0.150)*
> m19 *(enable C axis)*
> (~PLZC) *start ZC mode*
> (f10)
> (z.85) *feed tool into workpiece*
> (d.875) *set diameter for Y axis scaling*
> (x.5625) *move toward spindle face .5+ tool diameter*
> (y.5625) *up .5625*
> (x0) *back to start in X*
> (y0) *then Y*
> (y.0313) *start point for circle*
> (g02x.1313y.1625i.1313j.0313) *do a cw circle in 4 segments centered at x.1313,y.0313*
> (g02x.2625y.0313i.1313j.0313)
> (g02x.1313y-.1i.1313j.0313)
> (g02x0y.0313i.1313j.0313)
> (z1) *withdraw tool*
> (end) *cancel plane-switch mode*
> x-1
> z1
> m30

Part Dimensions →

Tool Path →

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
```
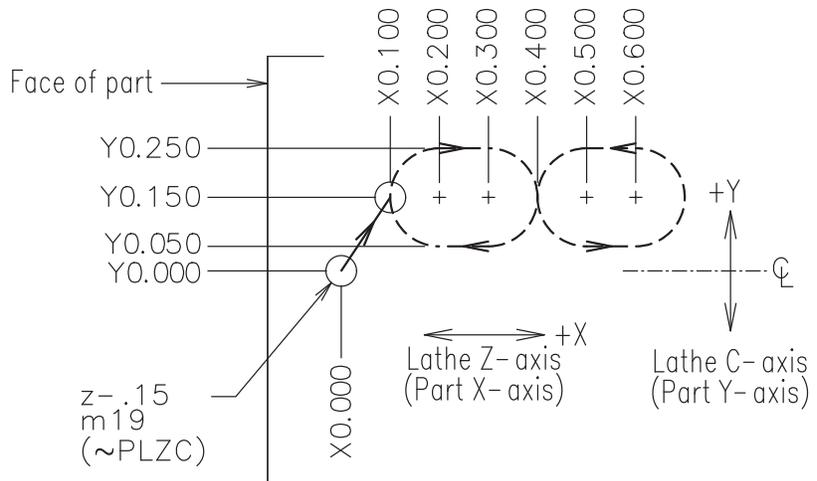
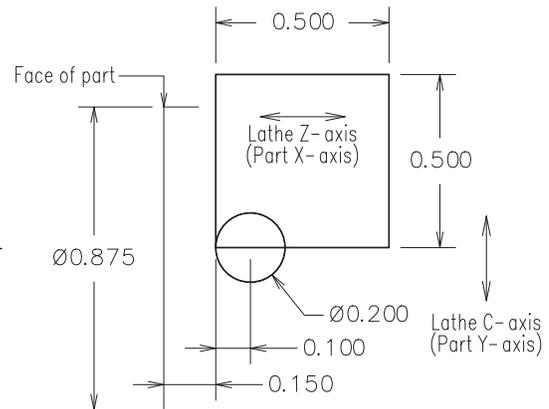## C-Axis ZC Plane Switching: Programming *continued*

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.*
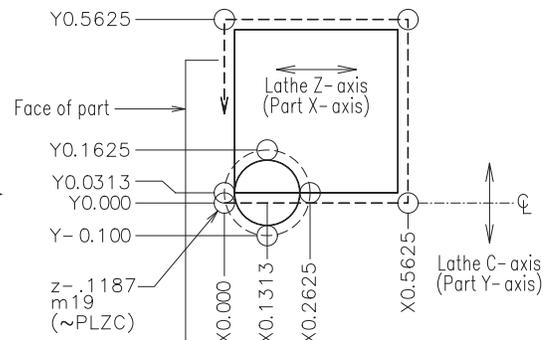

This cuts a J-slot like those found on electrical connector housings.

```
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
```
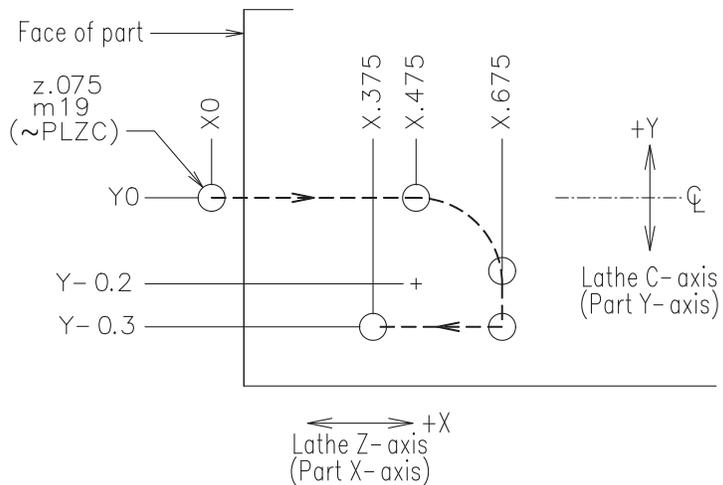
Tool Offsets

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.
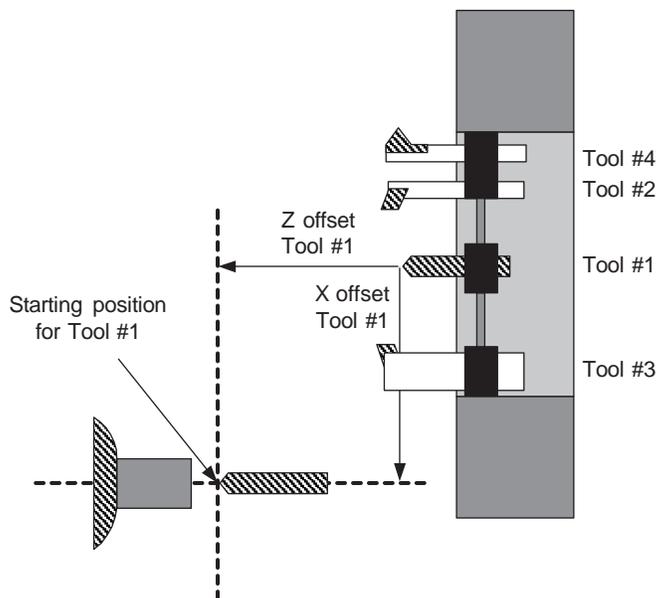
When the control is turned off the offsets are not forgotten, they are remembered on the disk that holds the OmniTurn software. When you turn on the OmniTurn, it automatically loads the tool offsets from the last time the machine was setup. This means that once a job is setup, and the tools have not 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

Illustration of tool offset for Tool #1

Illustration of tool offset for Tool #2
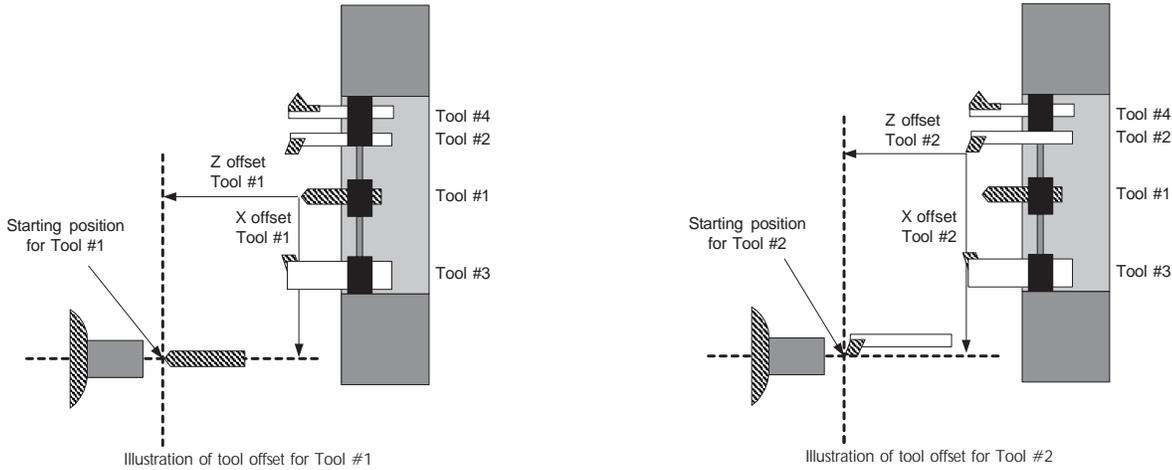
For all of the above examples, the tool is started at the center of the part and some distance from the face. This is being done in the examples only by choice. The tool can be started anywhere. To save moves and time you might have wanted to start tool #3 close to the diameter of the blank and not at 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.
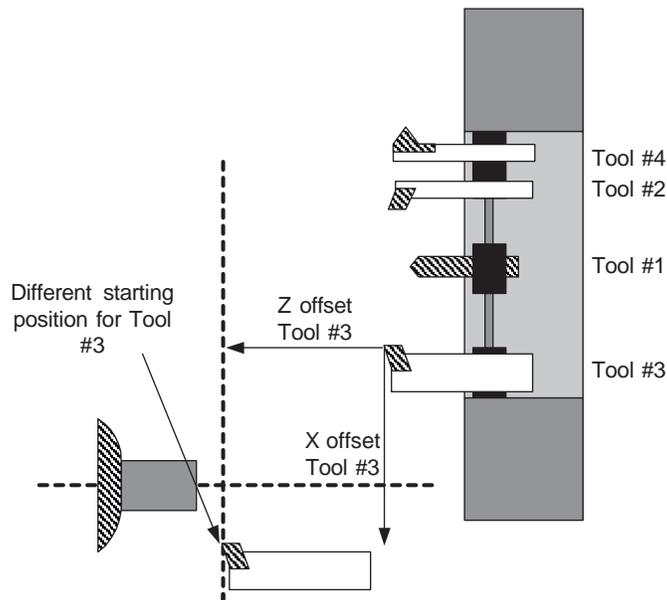
The starting point for the tool is the X and Z values immediately after the Tn command. This is where the tool will move to.

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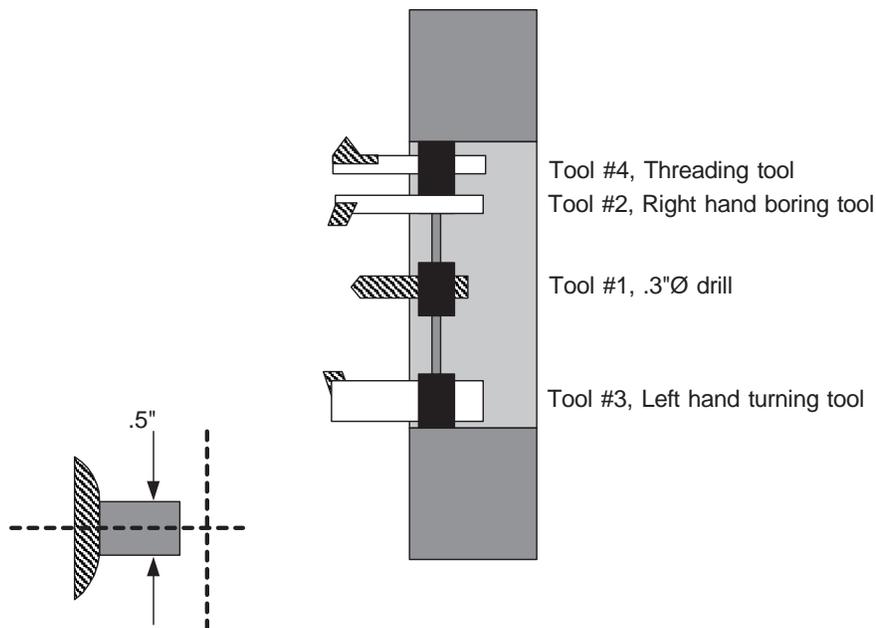
There is no order that the tools must be set up on the slide. It is not important in what order you do the offsets. It is possible to only do one offset and call it #3. The control would not care that there were no values in unused offsets.

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.

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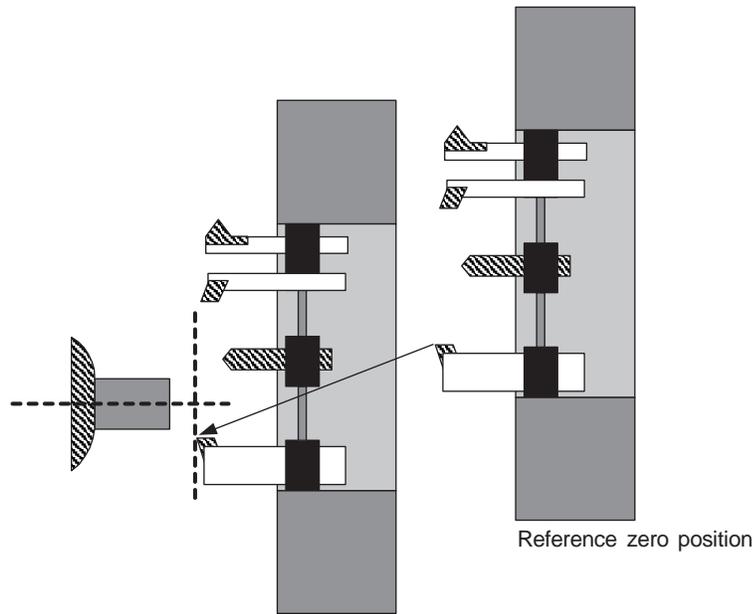
- Turn the spindle on, select the jog speed for slow, and take a skim pass of the material as shown next.

Move slide in and cut material

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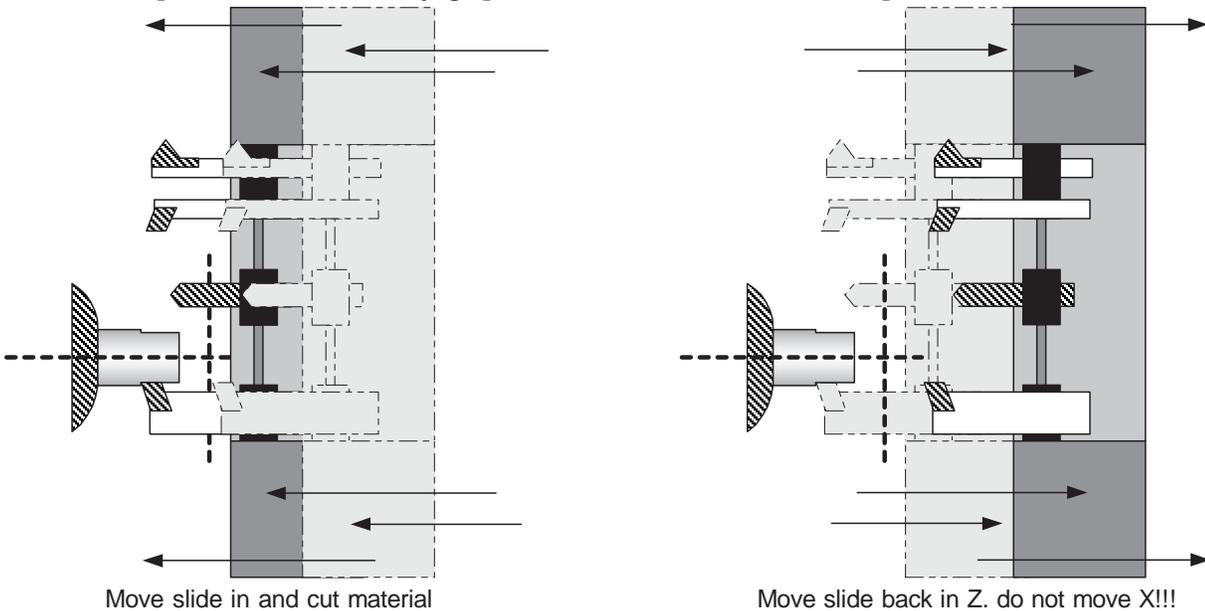
- Then, move the slide back in Z. **Do not move the slide in X.**
  This cut will be used to establish the offset.

Tool Offsets

To Set Left Hand Turning Tools continued

```
1.Slow                    Alt-C will enable C axis        Position
2.Medium                                                  X: +0.00000
3.Fast                                                    Z: +0.00000
4. 0.00005"/.01°
5. 0.00100"/.1°
6. 0.01000"/1°                                            Feed
7. 0.10000"/10°                                             0.0000 IPM
8. 1.00000"/90°                                           100%
9.Establish Home
H.Go Home                                                 Speed
T.Set Tool                                                   0  RPM
M.Manual Data Input                                       100%
A.Automatic                                               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                    Alt-C will enable C axis        Position
2.Medium                                                  X: +0.00000
3.Fast                                                    Z: +0.00000
4. 0.00005"/.01°
5. 0.00100"/.1°
6. 0.01000"/1°                                            Feed
7. 0.10000"/10°                                             0.0000 IPM
8. 1.00000"/90°                                           100%
9.Establish
H.Go Home          Tool Number:3_                     d
T.Set Tool         Press Esc to cancel tool setting   RPM
M.Manual Data I                                       %
A.Automatic                                           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                 Alt-C will enable C axis        Position
2.Medium                                               X: +0.00000
3.Fast                                                 Z: +0.00000
4. 0.00005"/.01°
5. 0.00100"/.1°
6. 0.01000"/1°                                         Feed
7. 0.10000"/10°                                         10.0000 IPM
8. 1.00000"/90°                                        100%
9.Establish
H.Go Home                                              Speed
T.Set Tool                                                0 RPM
M.Manual Data Inp  Press X or Z to set X or Z tool Position   0%
A.Automatic        or press Esc to cancel tool setting  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                 Alt-C will enable C axis        Position
2.Medium                                               X: +0.00000
3.Fast                                                 Z: +0.00000
4. 0.00005"/.01°
5. 0.00100"/.1°
6. 0.01000"/1°                                         Feed
7. 0.10000"/10°                                         10.0000 IPM
8. 1.00000"/90°                                        100%
9.Establish
H.Go Home                                              Speed
T.Set Tool                                                0 RPM
M.Manual Data Inp  Press X or Z to set X or Z tool Position   0%
A.Automatic        or press Esc to cancel tool setting  ol  0
                    Enter current diameter:.4923_
```

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

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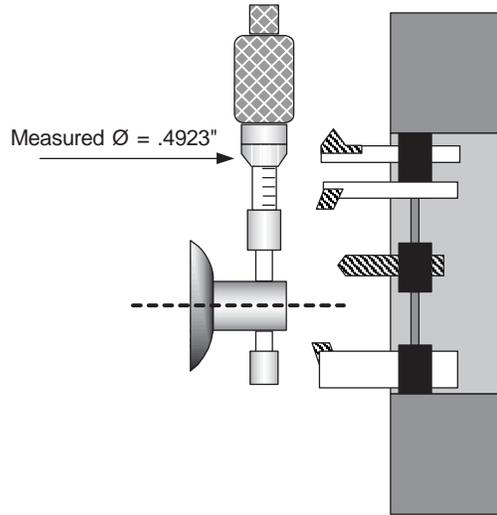
Take some care but do not be overly careful since any error made here can be easily corrected with the tool offset correction later when you are making the first piece. After typing .4923 hit "RETURN".

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

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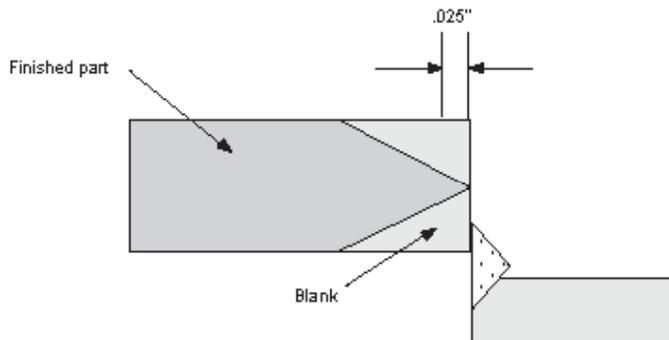
Then when asked what the location is you can enter in the known amount. In the above example you would enter in .025.

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.

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

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.

Postion the drill tip
just touching the face
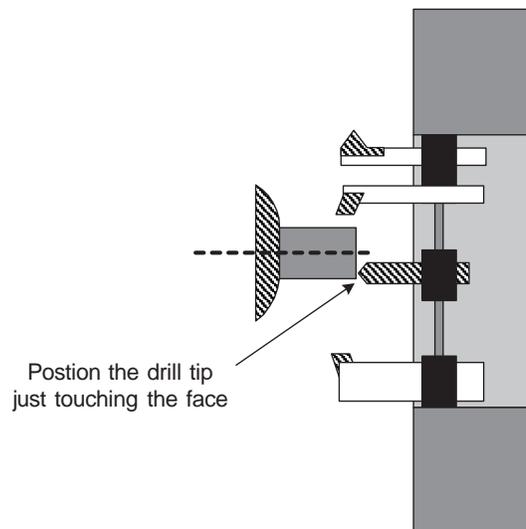
Setting a Drill (Tool #1) -continued

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
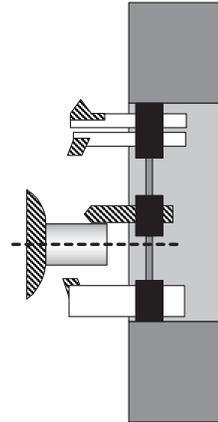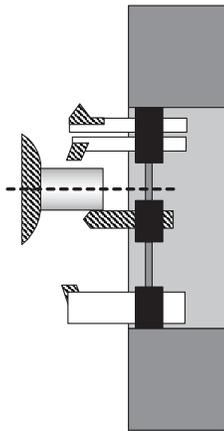
Setting up a drill on the - side of the spindle

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.

Material diameter = .4923" from turning for left hand turning tool

.3" drill

Drill location in X = Ø material + Ø drill
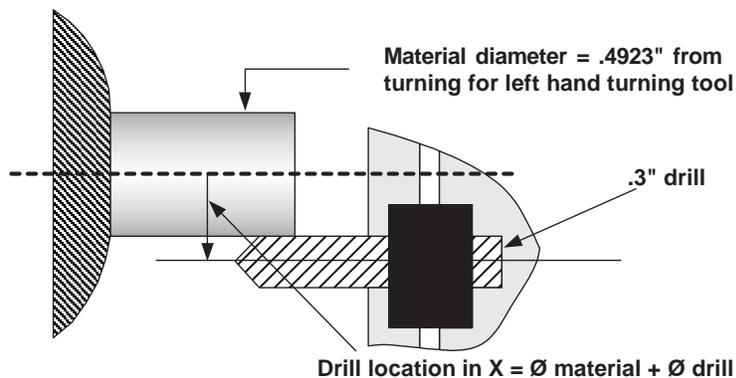
The value is equal to the diameter of the material plus the diameter of the drill. In the above figure you can see that the location of the center of the drill is the sum of these two numbers. .3" drill + .4923" material = .7923" location of drill center in X. For this example we are on the positive 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.
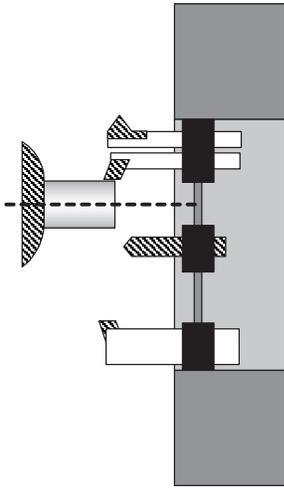
Setting a boring tool on the OD.
The measured Ø will be entered as a
NEGATIVE
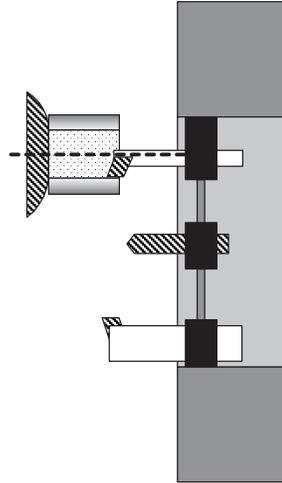
Setting a boring tool on the ID.
The measured Ø will be entered as a
POSITIVE

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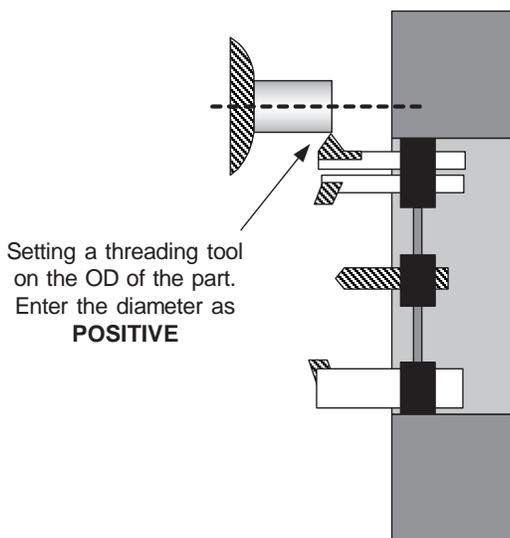
Setting a threading tool
on the OD of the part.
Enter the diameter as
POSITIVE
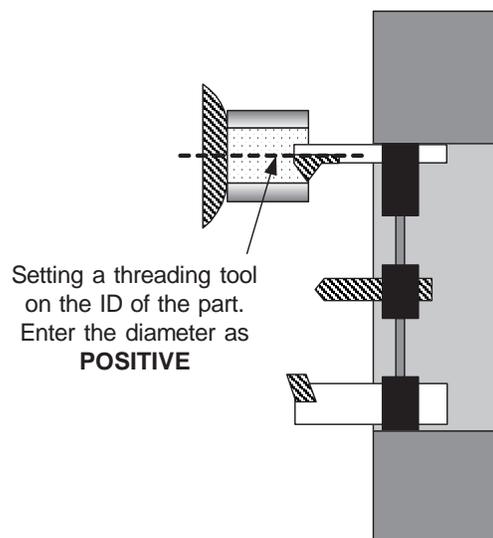
Setting a threading tool
on the ID of the part.
Enter the diameter as
POSITIVE

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.
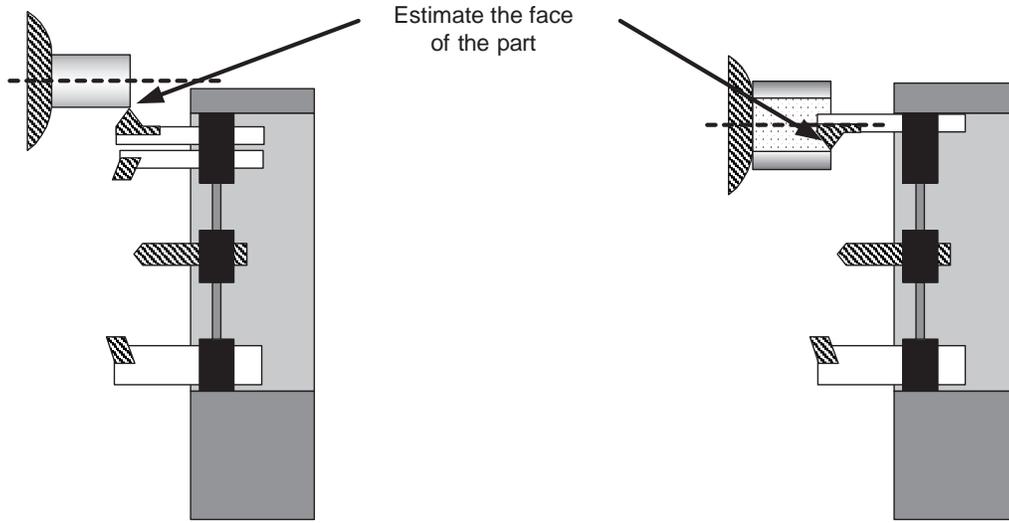
Estimate the face of the part

OmniTurn Sample Part

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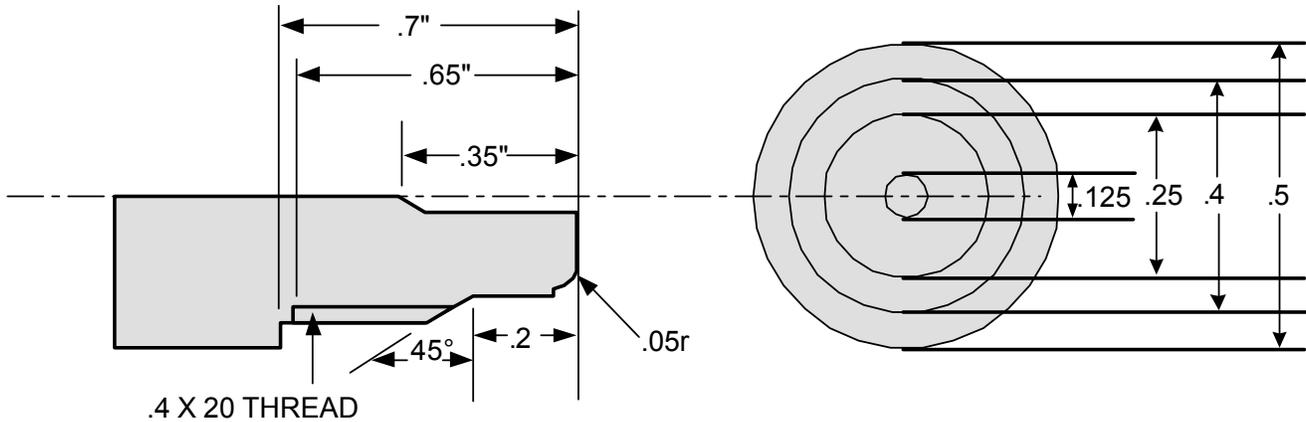
If you are familiar with machining you might want to skip to the next section, "TOOLING"

OPERATIONS

We will start with a solid piece of material 1/2" in diameter, 3" long.
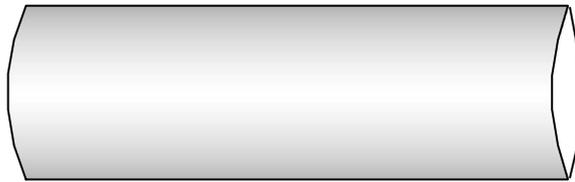
In machining this part we will use the same tool for roughing and finishing the OD. This does not have to be the case. You could setup another tool for the finishing pass. We will be doing the roughing first.
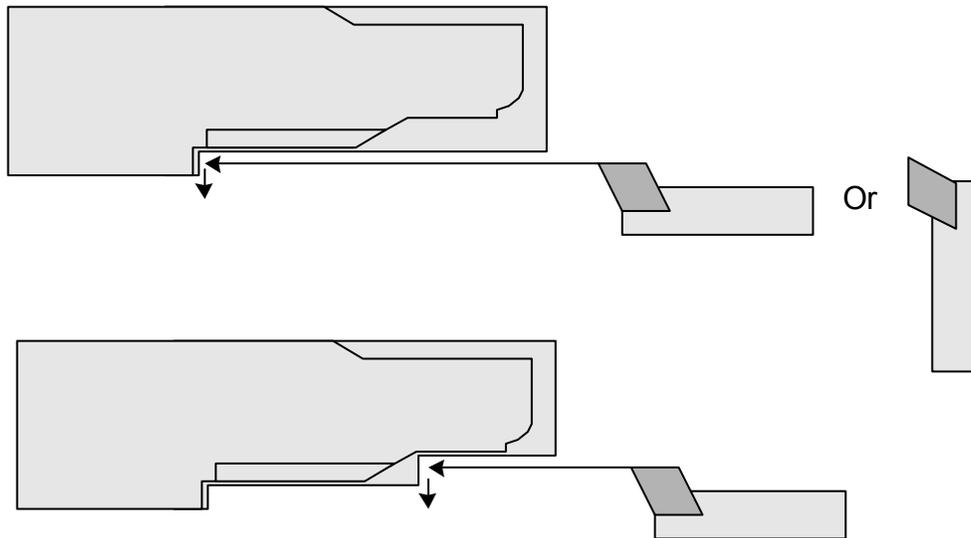
We will take two passes to rough the part. After we rough the OD and remove most of the material we change tools and center (spot) drill. Then we change tools again and peck drill the hole to finish depth.

OmniTurn Startup sample end
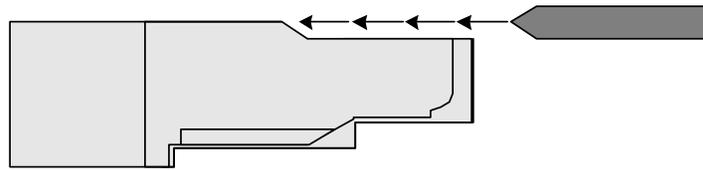
After the drilling we change back to the turning tool to face and finish turn the OD.
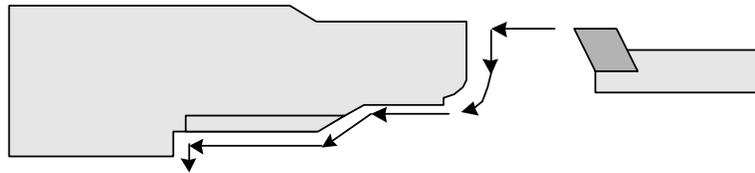
With the OD finished we can thread. (if your attachment does not have an encoder for threading you will have to skip over this operation)
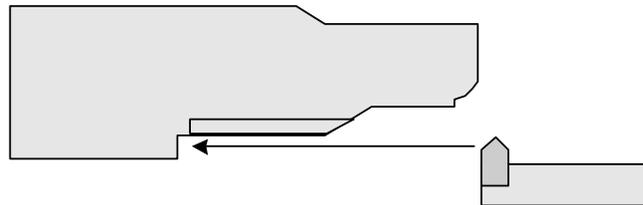
After the threading is complete we will change back to the turning tool to turn the major diameter of the thread to help deburr the OD. Then to make sure that all of the threads are clean and burr free we can take a single pass with the threading tool at it's finished depth. If the appropriate threading 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 to much time. This amount will vary depending on the type of tooling used and preferences of the programmer. As you get used the 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
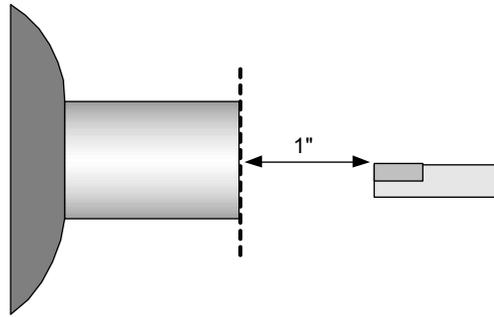
For each of the tools we have to establish a starting location. This location is not that important to worry about much. Just pick a location for each tool that will be safe to rapid to without worrying about collisions. Now we will add some information to the tool table that we started before:

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

OmniTurn Startup sample part

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.
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	• G96 changes the spindle speed to "constant surface feet" mode. We will be cutting at 250 sfm.
G77S3000	• G77 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
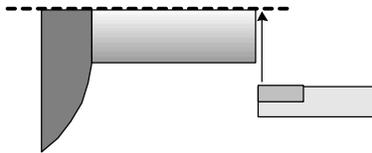
S2500	• Change the spindle rpm to a higher speed for the next smaller diameter pass.
G95F.003Z-.195	• Change to feed mode, and make the second pass
G94F300Z.025	• We move the tool off the face of the part in rapid
G97S2000	• Changes spindle speed to rpm mode, and sets speed to 2000 rpm.
X.205	• Position 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"
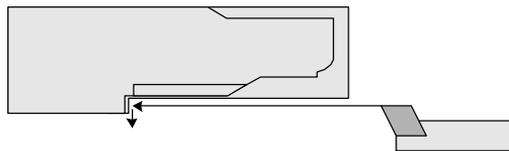
G94F300Z.025	• The tool is moved at rapid to clear the face of the part
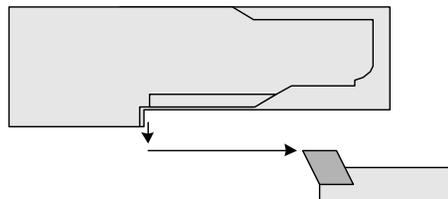
S2500	• Change the spindle rpm to a higher speed for the next smaller diameter pass.
G95F.003Z-.195	• Change to feed mode, and make the second pass

OmniTurn Startup sample part

X.225 • This clears the tool, still in the feed mode
G94F300Z1 • Change to rapid and move the tool to Z 1, this is clearance for a tool change
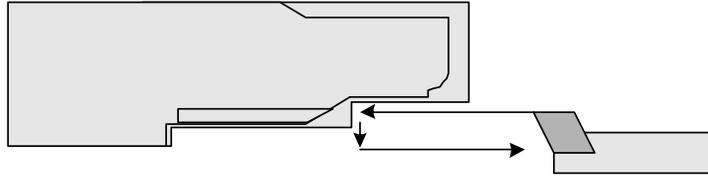
T2S2500(Center drill) • The center drill is called into position and the spindle speed is changed.
X0Z1 • The tool is moved to a safe location
M01 • Optional stop for tool check on the first run through
Z .025 • The tool is rapidly sent to the face of the part with about .02 clearance
G 95 F.002Z-.025 • Feed the center tool into the material at a feed rate of .002"/rev
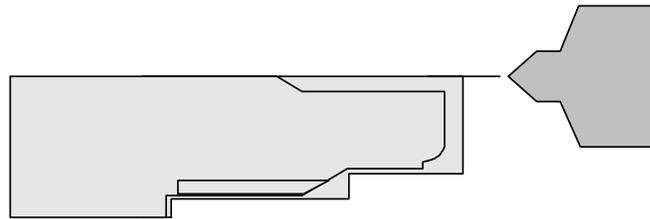
G94F300Z 1 • Rapid the tool clear of the part for a tool change
T3S2800(Drill) • Call the drill into position and change the spindle speed
X0Z1 • Move the drill to a safe location
M01 • Optional stop for the tool check on the first run through
2.025 • Rapid the drill to the face of the part with a little clearance
G95F.003 • Change to the feed mode of IPR
G83Z-.35K.1L300 • Peck drill the hole, .l" per peck, feed rate of .003 IPR, rapid at 300 ipm
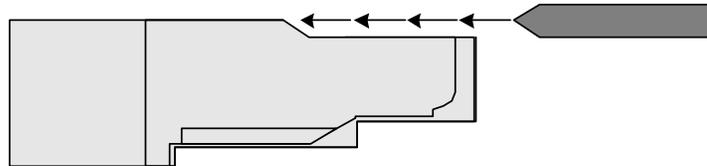
G94F300Z1 S2400 • Move the tool to clear the part for a tool change and change spindle speed.
T1 (LH tool .007 TNR) • The turning tool is brought into position again for the finish pass
X0Z1 • Move the tool to a save location
M01 • Optional stop for the tool check on the first run through
G77S3000 • Set maximum spindle speed to 3000
G96S250 • Change spindle speed mode back to sfm @250
G04F1 • Dwell 1 second to allow the spindle to change RPM
G41 • Turn on tool nose radius compensation
Z.025D1 • The tool is rapidly sent close to the face of the part & turn on comp with D1
G95Z0F.002 • The tool is fed to the face of the part to begin the finish contour
X.075 • The next few lines turn the part
G02X.125Z-.051.075K-.05
Z-.2
X.2Z-.275
Z-.75
X .26
G97S2000 • Spindle back to RPM mode and set speed to 2000
G94F300Z1 • This clears the tool in Z to 1 at rapid feed rate
G40 • Turn tool nose radius compensation off with move off the part

OmniTurn Startup sample part
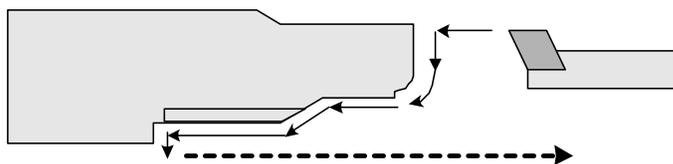
T4(Thread tool)	• Calls the threading tool offset
X0Z1	• Moves the threading tool to a safe location
M0l	• Optional stop for the tool check on the first run through
G04FI	• This is a dwell used to allow the spindle to change speed
X.2	• Moves the tool to the starting diameter of the threading cycle
Z0	• Moves the tool to .2" from the start of the thread in Z
G33X.175Z-.65K.05I .004	• This is the threading command
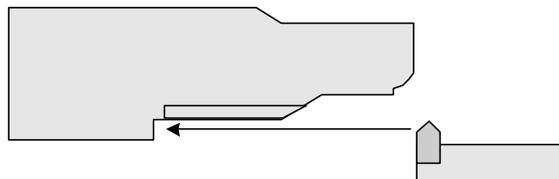
Z1	• This clears the tool in Z to 1 for the tool change
T 1(LH Turn tool)	• Call the turning tool for the deburr pass
X.15Z.1	• First move to location for the tool, more aggressive location than before
	Note: I did not put a MO1 after this tool call since this tool was already called and it was OK. So why stop it again?
G41	• Turn on TNR comp
Z.01D1	• Move tool to a spot so it can start the deburr pass
G95F01 X.125Z-.2	• Move tool to start of deburr (IPR), faster than normal cut pass
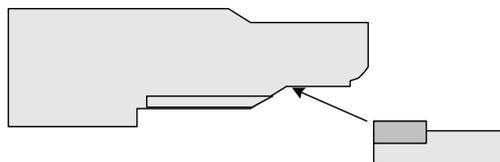
X.2Z-.275F.003	• Start deburr pass
Z-.75	
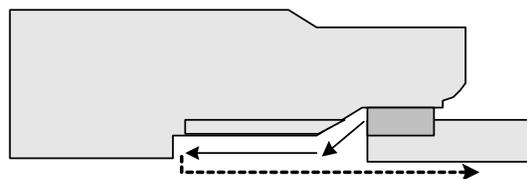
X.26	• Clear tool
G40	
G94F300Z1	
T4S2000(threading tool)	• Call the threading tool for a single pass to cleanup thread
X0Z1	• Establish tool location
G04FI	• Dwell for 1 second to allow the spindle to change speed
X.2Z0	• Moves the tool to same start location as first threading cycle
G33X.175Z-.65K.05I .004O	• This is the threading command for a single pass (note O)
M05	• 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
Z1	• this is a clearance move
M30	• End of the program, the control will now be ready to run another part

OmniTurn Startup sample part

Saving the program to the disk

- Now to save the program press Fl 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 to 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.
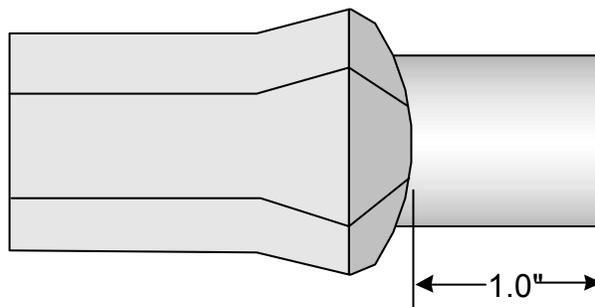
Then put the turning tool on the slide. For this example put it on the near side.

OmniTurn Startup sample part

Setting X axis

Tool #1
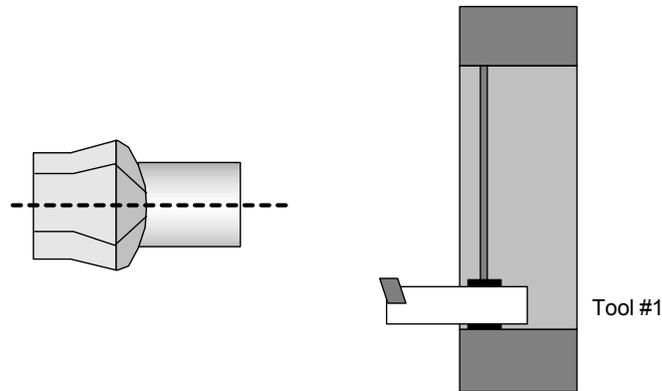
Now go to the jog mode and follow the procedure on setting up tool. Turn the spindle on, (if you have a unit with spindle control do this in the MDI mode first).

Move the slide and take a light cut on the diameter.
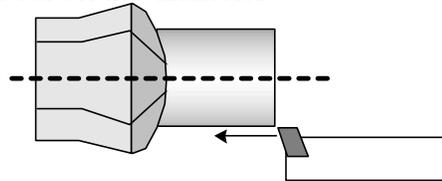
Then move the slide in Z only to clear the material, do not move in X!
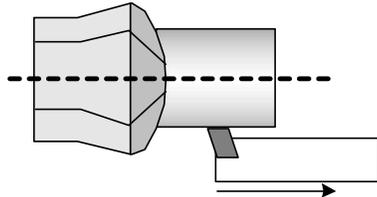
Turn the spindle off. If you have the spindle control use the switch on the OmniTurn panel.

Then measure the diameter with a micrometer

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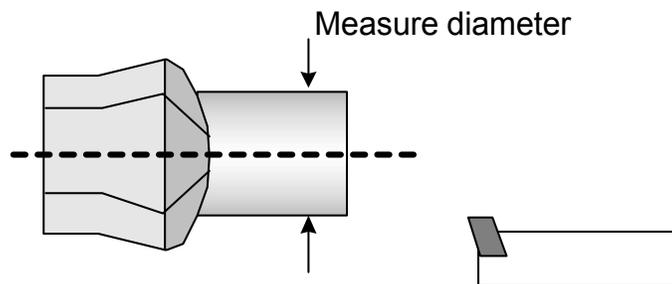
Now press T -"SET TOOL" on the keyboard to start inputting the tool offset.

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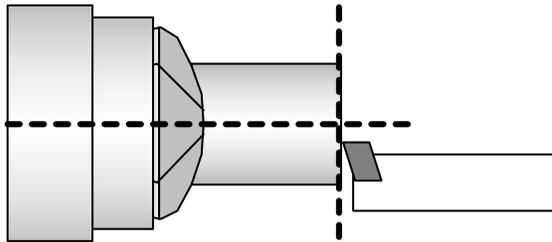
Then press 1 when asked for the tool number, and RETURN

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 to 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, colet, spindle, etc. A simple way to set the tooling is:

Put the control into the jog mode and move the first tool to it's 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 to far apart there is a lot of wasted movement and the cycle time will be to 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

Position drill
so it clears part

Tool #2

Tool #1
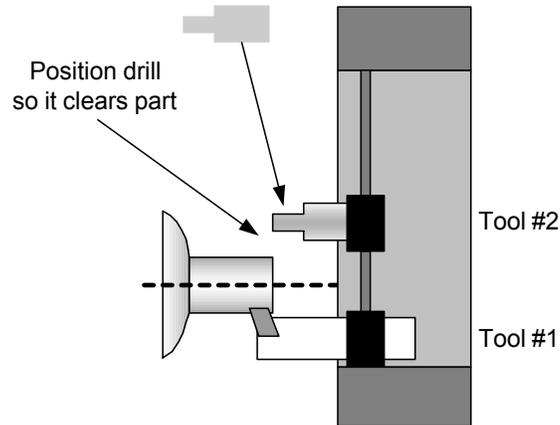
With the slide and tooling set as shown above lightly secure the center drill and holder to the slide.

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.

1/2" pin

1/2" collet

1/2" bushing in holder

Tool #2

Tool #1
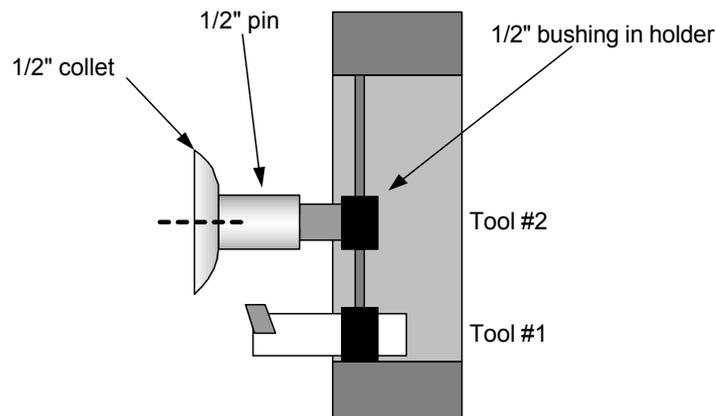
    -In the jog mode select "T"
    -When asked for the tool number, enter "2" and return
    -Press X for the axis you are entering
    -Enter "0" for the diameter measured and return
    -Press "Esc" to return to the jog mode

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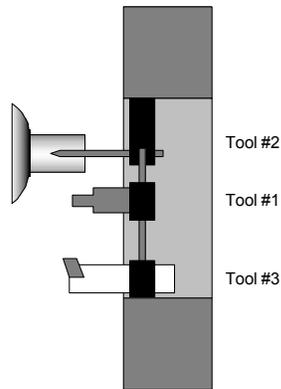
Next is the threading tool. Here we will use another technique. Instead of cutting with the tool we can use the cut surface from setting the first tool to establish the location of this tool.

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.
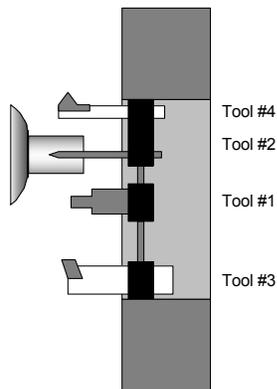
Then jog the slide until the threading tool just clears the turned diameter, it would be best to use a feeler gauge or slide the tool until it just touches the turned diameter. You can do this with the spindle off. If the spindle is on, then you would jog until the tool just touches the material and 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.
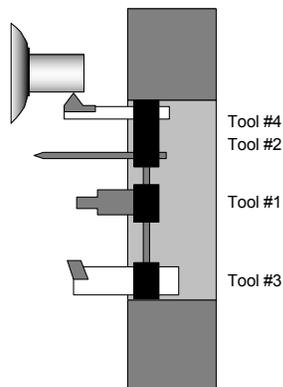
OmniTurn Startup sample part

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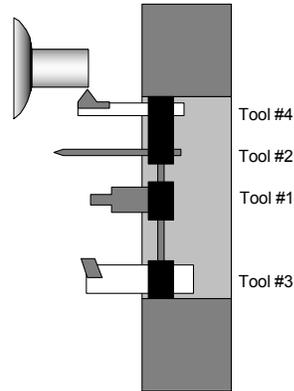
Now that you have the tool where you want it in Z you can establish the tool offset. Press T to start the entry and then select Z. Then enter 0 for the current location and press RETURN. When the control tells you that the offset is entered press Esc to get back to the jog mode.

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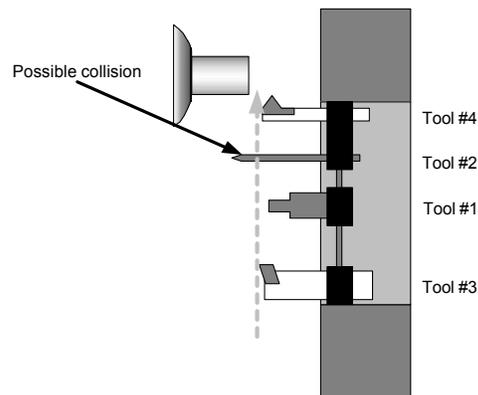
If we move the slide back and over a little there will be no problem.
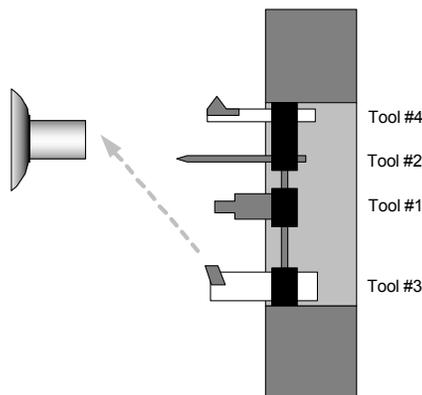
Checking the tool offsets

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 **X0Z1F50,** 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.

```
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
```

After inputting the tool command, the Tool# and relative position are displayed.

After inputting the move, the position will change to X: 0 Z: +1

```
Position
X: -2.00425 φ
Z: +2.20315

Feed
 50.0000 IPM
100%

Speed
   0  RPM
100%
Tool  1
```

```
Command:_
Pending:
Press Cycle start to Execute
or Esc to cancel command or exit MDI
```

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.

```
To Go                c:\programs\SAMPLE1              Position
X: +0.00000                                          X: +0.00000
Z: +0.00000                                          Z: +0.00000

                          File in memory

Switches                                             Feed
C.Cycle Repeat                                       000.000   IPM
/.Block Delete                                       100%
O.Optional Stop
S.Single Block       Press Cycle Start to Begin
                                                     Speed
                                                        0   RPM
Functions                                            100%
F2-Tool Offsets                                      Tool  0
F3-Edit
F4-Verify
F5-New File          Feedrate & Spindle speed
F6-Program Search        override percentages
F8-File Ops
F9-Sec. Offsets
F10-Spec. Functions
J-Jog Mode
M-Manual Data Input
```

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.

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                c:\programs\SAMPLE1              Position
X: +0.00000                                          X: +0.00000
Z: +0.00000                                          Z: +0.00000


Switches                                             Feed
C.Cycle Repeat                                         0.000  IPM
/.Block Skip                                         100%
O.Optional Stop
S.Single Block       Press Cycle Start to Begin
                                                     Speed
                                                        0  RPM
Functions                                            100%
F2-Tool Offsets                                      Tool  0
F3-Edit
F4-Verify
F5-New File
F6-Program Search
F8-File Ops                         Press 'S' key to activate Single Block mode
F9-Sec. Offsets
F10-Spec. Functions
J-Jog Mode
M-Manual Data Input
```

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                c:\programs\SAMPLE1              Position
X: +0.00000                                          X: +0.00000
Z: +0.00000          G95F.003Z-.75                   Z: +0.00000
                     X.26
                     G94F300Z.025
Switches                                             Feed
C.Cycle Repeat                                         0.000  IPM
/.Block Skip                                         100%
O.Optional Stop
S.Single Block       Press Cycle Start to Begin
                                                     Speed
                                                        0  RPM
Functions                                            100%
F2-Tool Offsets                                      Tool  0
F3-Edit
F4-Verify
F5-New File
F6-Program Search
F8-File Ops
F9-Sec. Offsets      Three blocks of the program are displayed:
F10-Spec. Functions  Current block is highlighted with reverse text
J-Jog Mode           Previous block is above it
M-Manual Data Input  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.

Should be .250"
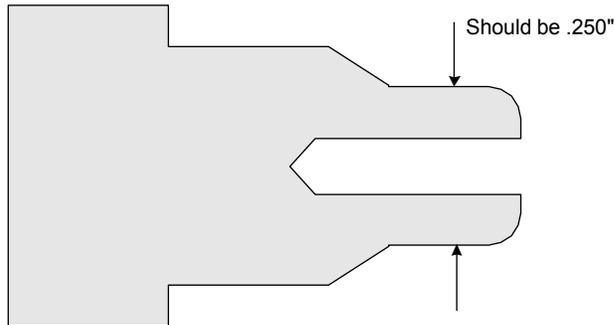
If we now make a part and measure it there will probably be variations from what you program and the size of the finished part. These differences can come from a number of sources, minor errors in establishing tool offsets, tool deflections, material deflection. If the error is small it is possible to 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

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

1/2" x 13 thread
.05r
.3"
.8"
1.0"
.4 .5 .6
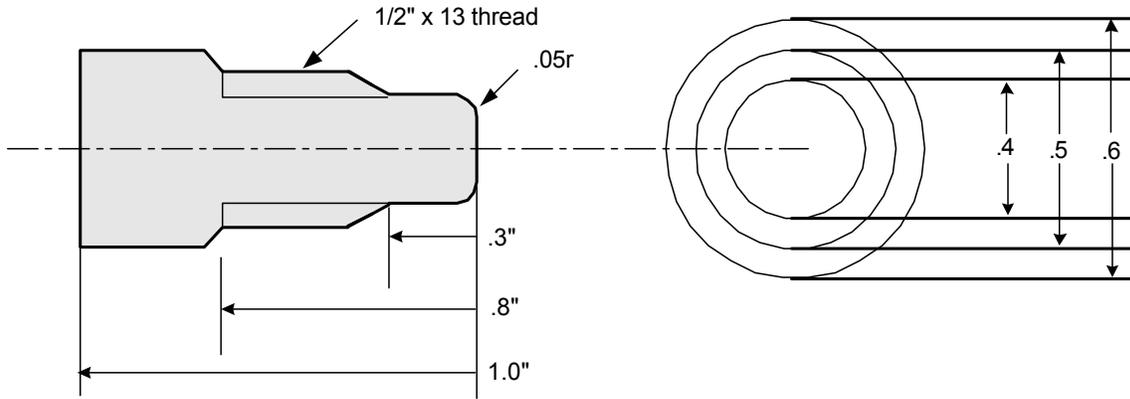
In the above example, the finished part will be made from a cutoff blank .61" diameter by 1.1" long. The first item to take care of is the layout of the job. This will entail what our sequence of operations will be. Then select the tools to accomplish this sequence. Once the tooling is set it is possible 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

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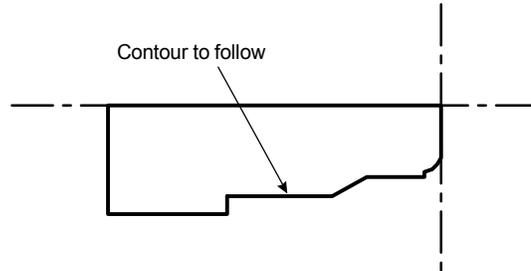
Now, the tools can be selected for the operations listed above. See below the tool selections that will be used.

Tool #2
Threading tool
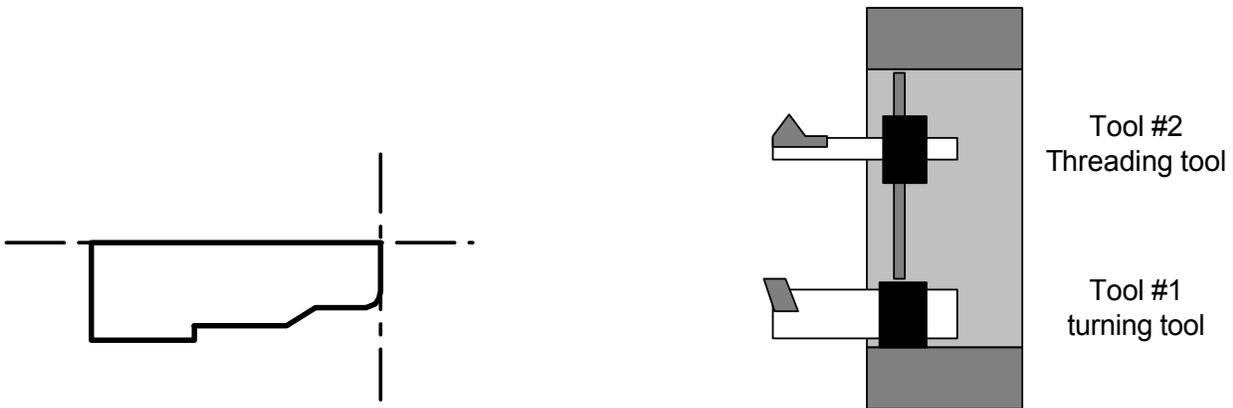
Tool #1
turning tool

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.

Diameter .50" ÷ 2 = .25" 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:

Tool #2
Threading tool

.2"

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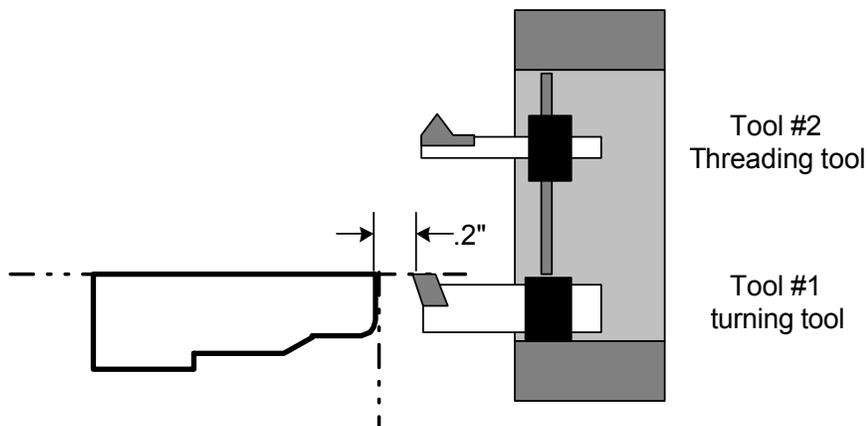
     1. The setting of the tool offsets in the Manual mode.
     2. The writing of the G92 statements after the Tool change in the program

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

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

Tool #2
Threading tool

Tool #1
turning tool
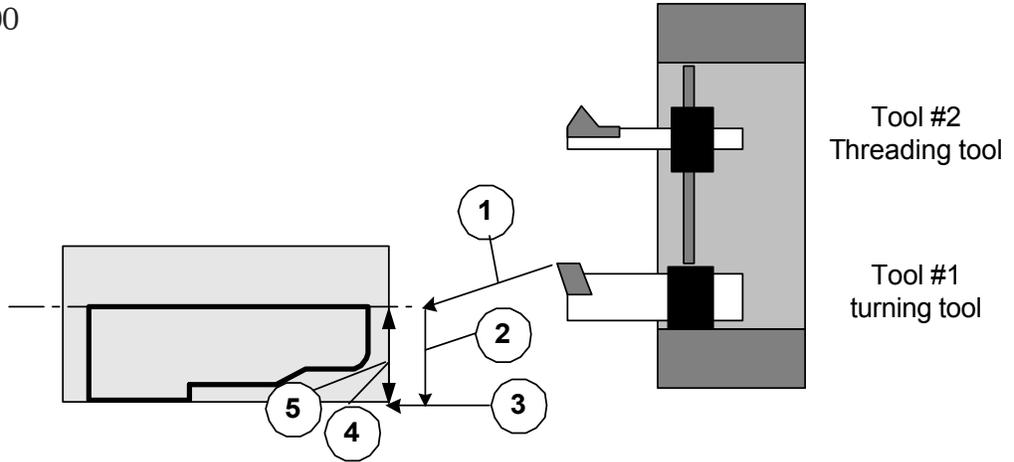
```
(6)     Z-.79F3
(7)     X.27F200
(8)     Z.1
(9)     X.21
```

Tool #2
Threading tool

Tool #1
turning tool
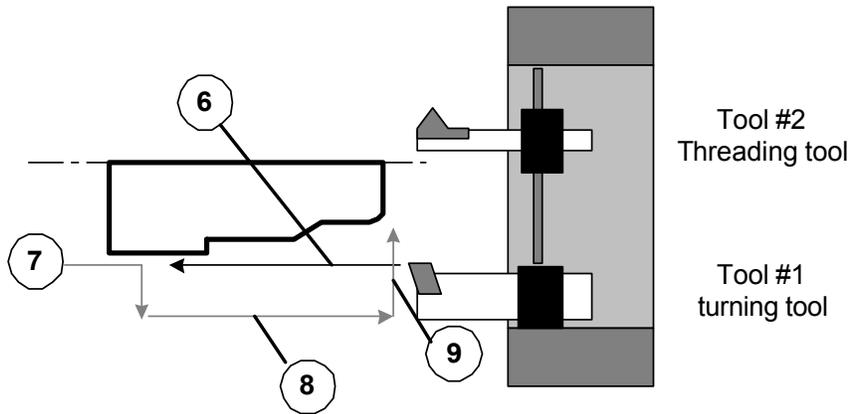
```
(10)    Z-.29F3
(11)    X.22
(12)    Z.1 F200
(13)    X0
(14)    Z0F3
```

Tool #2
Threading tool

Tool #1
turning tool
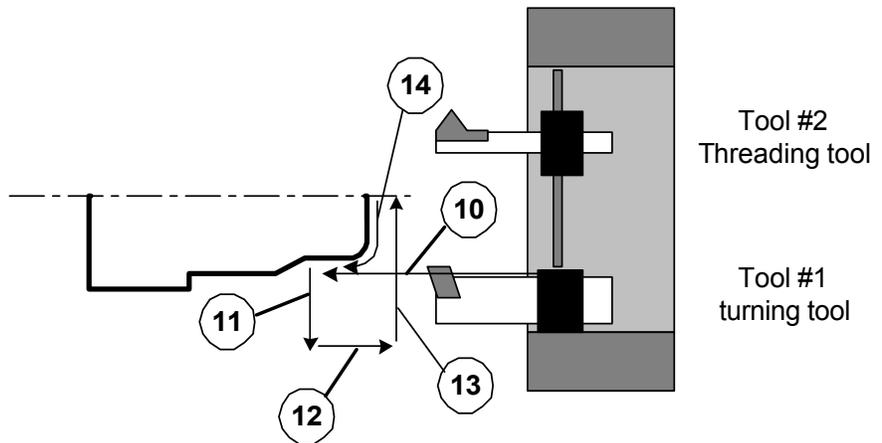
Worked Examples

(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

Tool #2
Threading tool

Tool #1
turning tool

(23) Z.5F200

Tool #2
Threading tool

Tool #1
turning tool
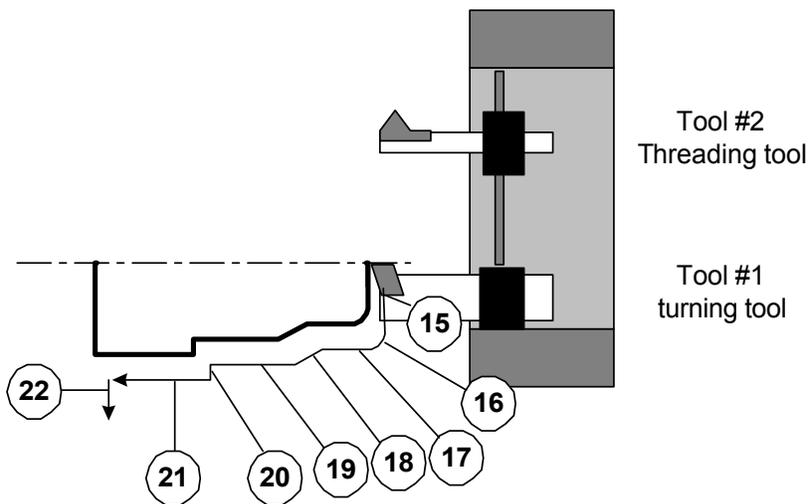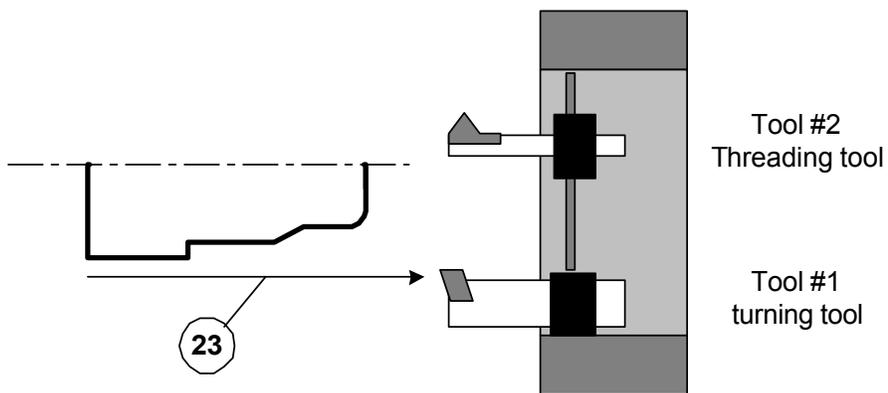
(24)   T2
        G92X.245 Z.2
        M01
(25)   g33x.22z-.8i.005k.0769p

        T0
        M30

Tool #2
Threading tool

Tool #1
turning tool
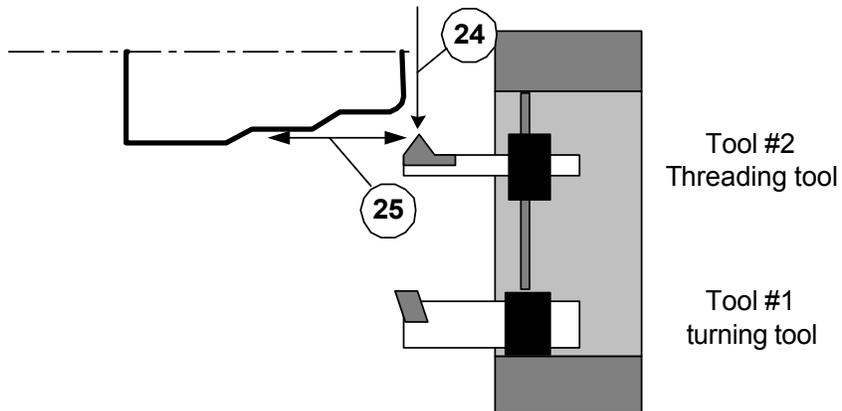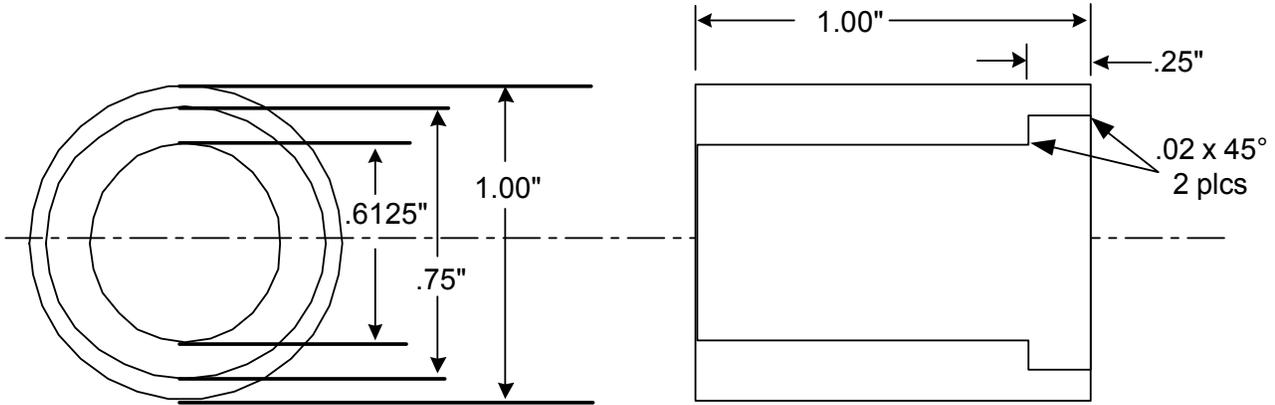
Worked Examples

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

Tool #1
Roughing boring bar

Tool #2
Finishing boring bar
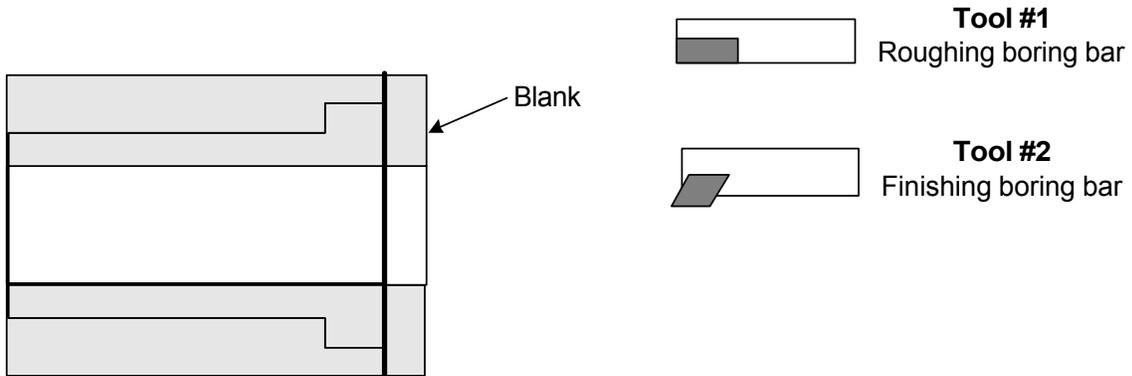
The blank has been predrilled and is a little longer than the finished part.

    G90 G94 F200
    T1
    G92 X0 Z.2
    M03S2000

(1)   X.28 Z.01

Tool in location after T1
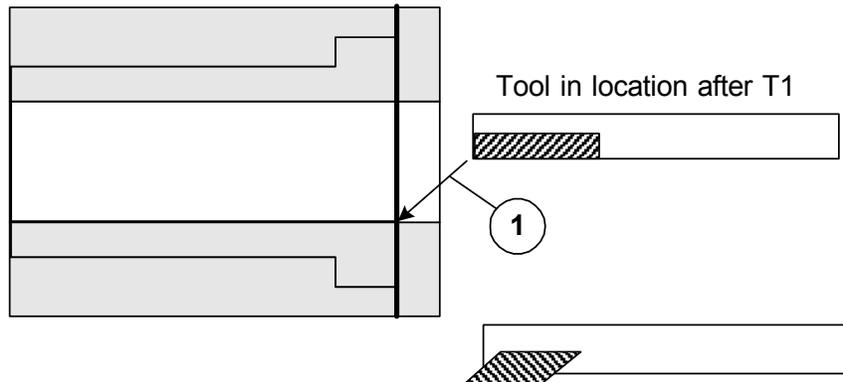
Worked Examples

(2) X.55 G95 F.003
(3) G94 F200 Z.015
(4) X.34

Rough face
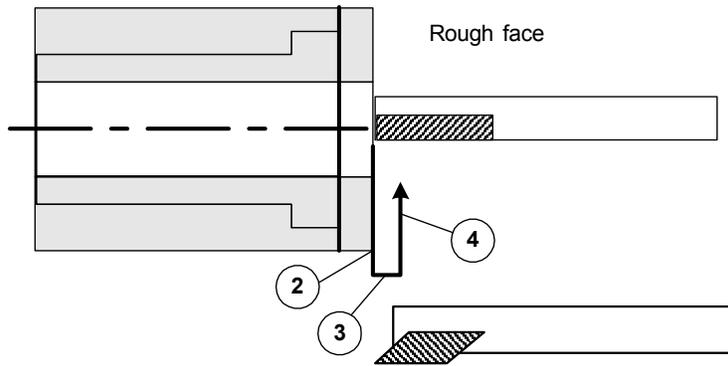
(5) G95 F.002 Z-.235
(6) X.3
(7) Z-1.05
(8) X.29
(9) G94 F200 Z.5
     T2
     G92 X.55 Z.1
     S3000
     Z0

Rough bore
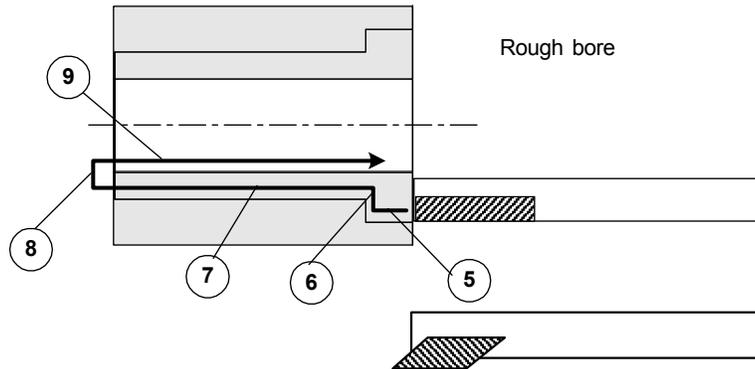
(10) G95 F.002 X.395
(11) X.375 Z-.02 D1   Generate chamfer with correction to effect first bore
(12) Z-.25 D2   Bore first diameter with correction for shoulder length
(13) X.32625
(14) X.30625 Z-.27 D3   Generate chamfer with correction to effect second bore
(15) Z-1.05 D4   Bore second diameter with correction for back of bore
(16) X.29
(17) Z.5 G94 F200
      M05
      T3   Position the tools away from the part for loading, it also
                          Turns off the secondary offsets

      M30

Finish face and bore
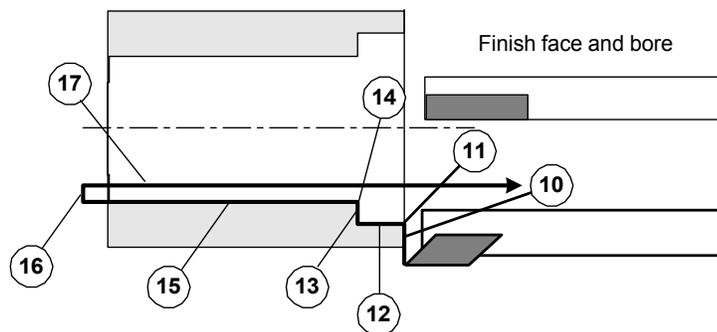
Automatic Mode, Table of Contents

Automatic Mode, Introduction

Loading first program
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, File Handler: Overview for details (Page 1.19).

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.

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 Screen
The Functions and Switches are listed on the next page, then discussed in detail on pages folllowing

Program in memory

Position:
Displays position of active tool.
(Here, Tool 0 is at machine 'home')

To Go:
Displays distance to go

```
To Go                 c:\programs\newprog3        Position
X: +0.00000                                        X: +0.00000
Z: +0.00000                                        Z: +0.00000


Switches                                           Feed
C.Cycle Repeat                                        0.000  IPM
/.Block Skip                                        100%
O.Optional Stop
S.Single Block        Press Cycle Start to Begin


Functions                                          Speed
F2-Tool Offsets                                        0  RPM
F3-Edit                                            100%
F4-Verify                                          Tool  0
F5-New File
F6-Program Search
F8-File Ops
F9-Sec. Offsets
F10-Spec. Functions
J-Jog Mode
M-Manual Data Input   Check and fill CC lube each shift
                      Press Alt-L when done
```

Speed:
Displays spindle speed, spindle override percentage, and active tool.

Feed:
Displays feedrate and feedrate override percentage.

Press 'J' or 'M' key to exit Auto Mode

Functions:
Press Function key to enable function

Switches:
Press key to enable switch; press again to disable. Effective at any time.

Collet closer lube reminder.
NOTE: After ten hours prompt will appear mid-screen and Cycle-start will be inhibited until oil level is verified and Alt-L is pressed (Press and hold "Alt" key, then press "L" key).

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.

Making corrections:

```
                    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           │
           └────────────────────────────────────────┘
```
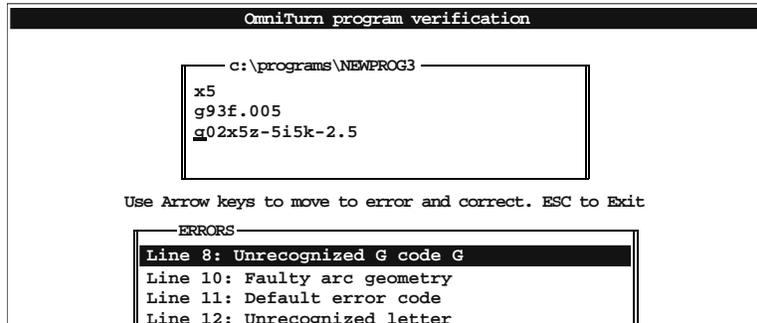
If an error is found in your program it is possible to correct it now. Use the left or right arrow keys to move the cursor to the right of the incorrect character. In the above example the g93 should be g95. Arrow over to the f then Backspace to delete 3, then type a 4. When you are done with the correction 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.

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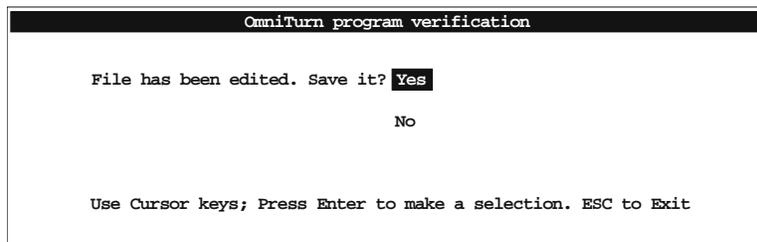
Press Enter to accept the default (Yes). If you choose not to save the program, use the down arrow to select No, then press Enter. Pressing ESC key will save and exit.

If there are no more errors the Automatic Mode screen will appear, showing program in memory, and prompt **Press Cycle Start to Begin**

```
┌──────────────┬────────────────────────┬──────────────┐
│ To Go        │ c:\programs\newprog3   │ Position     │
│ X: +0.00000  │                        │ X: +0.00000  │
│ Z: +0.00000  │                        │ Z: +0.00000  │
│              │                        │              │
├──────────────┤                        ├──────────────┤
│ Switches     │                        │ Feed         │
│ C.Cycle Repeat│                       │   0.000  IPM │
│ /.Block Skip │                        │ 100%         │
│ O.Optional Stop│                      │              │
│ S.Single Block│ Press Cycle Start to Begin│          │
│              │                        ├──────────────┤
│              │                        │ Speed        │
│              │                        │    0  RPM    │
│ Functions    │                        │ 100%         │
│ F2-Tool Offsets│                      │ Tool  0      │
│ 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│                  │              │
└──────────────┴────────────────────────┴──────────────┘
```
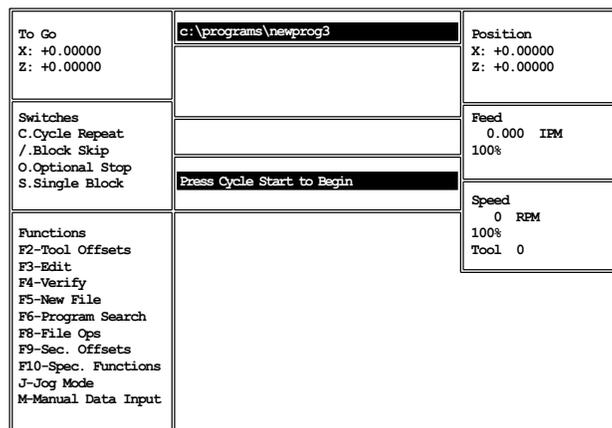
Automatic Mode: Functions & Switches

These Functions and Switches are discussed in detail on pages following

The Function keys have the following functions:
NOTE: These keys are not effective while program is running

F2	Tool Offsets	Adjust tool offsets.
F3	Edit	Edit existing programs or create new programs.
F4	Verify	Verify the program in memory and display the tool paths.
F5	New File	Removes the program from active memory and opens file handler screen.
F6	Prog Search	This enables the program to be started some place other than the start
F8	File Ops	File Operations:

> File handler screen
> Erase programs
> Create new program
> Up- or Download programs from another computer over RS232

F9	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

The Mode keys work as follows:
NOTE: These keys are not effective while program is running

J	Jog Mode	Switch to Jog page
M	MDI	Switch to Manual Data Input page

The Switches have the following funtions:
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.

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

Pg Up		Coolant on/off (M08/09). Press to turn coolant on, press again to turn coolant off.
Ctrl+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 5.18 for keyboard.

Automatic Mode, Switches

Functions and switches available from Automatic Mode

```
To Go                c:\programs\newprog3           Position
X: +0.00000                                         X: +0.00000
Z: +0.00000                                         Z: +0.00000

                                                    Feed
Switches                                              0.000  IPM
C.Cycle Repeat                                       100%
/.Block Skip
O.Optional Stop      Press Cycle Start to Begin
S.Single Block                                      Speed
                                                      0  RPM
Functions                                           100%
F2-Tool Offsets                                     Tool  0
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 'J' or 'M' key to exit Auto Mode

Press Function key to enable function

Press key to enable switch; press again to disable. Effective at any time.

```
To Go                c:\programs\newprog3           Position
X: +0.00000                                         X: +0.00000
Z: +0.00000                                         Z: +0.00000

Switches                                            Feed
C.Cycle Repeat                                        0.000  IPM
/.Block Skip                                        100%
O.Optional Stop
S.Single Block       Press Cycle Start to Begin
                                                    Speed
                                                      0  RPM
Functions                                           100%
F2-Tool Offsets                                     Tool  0
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
```

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.

```
To Go                 c:\programs\newprog3        Position
X: +0.00000                                       X: +0.00000
Z: +0.00000                                       Z: +0.00000

Switches                                          Feed
C.Cycle Repeat                                       0.000  IPM
/.Block Skip                                       100%
O.Optional Stop
S.Single Block        Press Cycle Start to Begin

                                                  Speed
                                                     0  RPM
Functions                                         100%
F2-Tool Offsets                                   Tool  0
F3-Edit
F4-Verify
F5-New File
```

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                 c:\programs\newprog3        Position
X: +0.00000                                       X: +0.00000
Z: +0.00000                                       Z: +0.00000

Switches                                          Feed
C.Cycle Repeat                                       0.000  IPM
/.Block Skip                                       100%
O.Optional Stop
S.Single Block        Press Cycle Start to Begin

                                                  Speed
                                                     0  RPM
Functions                                         100%
F2-Tool Offsets                                   Tool  0
F3-Edit
F4-Verify
F5-New File
```

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 continiue. Press O again to return to normal automatic mode. See Section Two, page 2.61 for more about Optional Stop.

```
To Go                 c:\programs\newprog3        Position
X: +0.00000                                       X: +0.00000
Z: +0.00000                                       Z: +0.00000

Switches                                          Feed
C.Cycle Repeat                                       0.000  IPM
/.Block Skip                                       100%
O.Optional Stop
S.Single Block        Press Cycle Start to Begin

                                                  Speed
                                                     0  RPM
Functions                                         100%
F2-Tool Offsets                                   Tool  0
F3-Edit
F4-Verify
F5-New File
```

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. The following pages describe 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:

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

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. Assume that tool 2 is a turning tool and is cutting a part .001" too large. 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.

After selecting a number and pressing Return the screen will ask "X adjustment".

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

enter the value of change (in this case -.001) and press Enter. The value of X will update and then ask "Z adjustment"

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

Enter the amount of change in Z (leave blank for no change) and then press Enter if you want to adjust another tool, else press Esc to return to the Auto screen

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.

F3 Edit Program

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.

```
 File  Edit  Search  Options  Window  Help                    05:04pm
┌─[■]════════════════ C:\PROGRAMS\SMALTEST ══════════════1=[↕]─┐
│G90                                                            ▲
│G94F300                                                         
│X1                                                              
│Z-1                                                             
│X0                                                              
│Z0                                                              
│X.5                                                             
│G02X.5Z-1I.5K-.5                                                
│g04f.1                                                          
│G02X.5Z0I.5K-.5                                                 
│g04f.1                                                          
│G03X.5Z-1I.5K-.5                                                
│g04f.1                                                          
│G03X.5Z0I.5K-.5                                                 
│g04f.1                                                          
│X0                                                              
│M30                                                             
│                                                               ▼
│════ 1:1 ════◄◘                                             ►▼
Alt+X=Exit  F1=Help  F2=Save  F8=Exec                      ^Lit Edit
```

F5 Load New Program

This function starts the File Handler so you can change to another part program.

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, or press the key that matches the first character of the part program you want to load.

The file handler is described in Section 1, starting on page 1.19.

```
1234A        1235A       2468B       A34        c:\programs\
BUSHING2     COLLAR23    D-0478      DEMO       d:\
FLANGE12     G041414     L12398      SMALTEST   d:\programs\
TESTPROG                                        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 curser

F5 to type filename,F8 to load from RS232
Current machine file; null
Current directory; c:\programs
```

F4 Program Verification

This function calculates the estimated cycle time, and provides a graphic view of the toolpath for each tool in your progarm. This description uses the sample program DEMO illustration.

No spindle speed

In addition to the standard syntax error checking which occurs whenever a program is first loaded into memory, or re-loaded after editing (see *Initial Program Verification*, pg 5.3), the F4 Program Verification also reports error if no spindle speed is called in the program. This is because the estimated cycle time feature requires a spindle speed to calculate g95 moves.

If no spindle speed is found by the verification program, the screen below will appear.
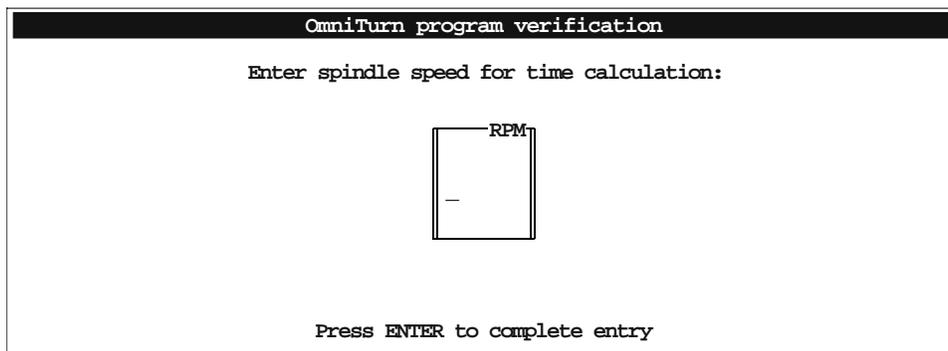
```
                     OmniTurn program verification
                 Enter spindle speed for time calculation:

                            ┌──RPM─┐
                            │      │
                            │ _    │
                            │      │
                            └──────┘


                     Press ENTER to complete entry
```

Enter a value for spindle speed in RPM, then press enter. Note: the speed you enter is NOT written to your program. You must edit your program to include spindle speed and direction.
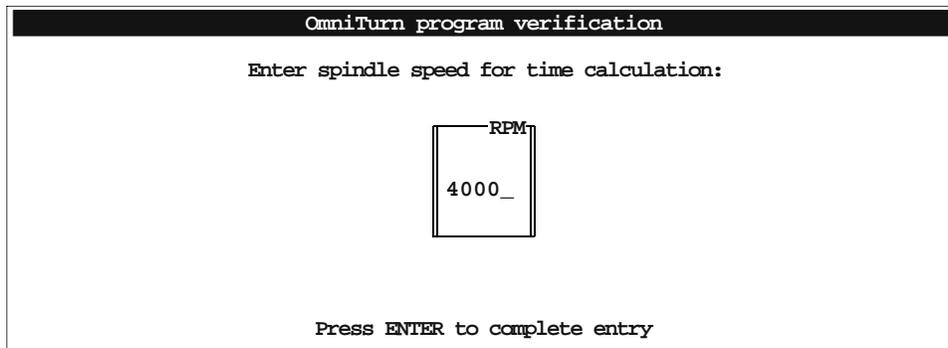
```
                     OmniTurn program verification
                 Enter spindle speed for time calculation:

                            ┌──RPM─┐
                            │      │
                            │ 4000_│
                            │      │
                            └──────┘


                     Press ENTER to complete entry
```

After entering the spindle speed, the Estimated Cycle Time screen will appear. (see next page).

Note that the Estimated Cycle Time screen is normally the first screen because most programs include spindle speed.

Automatic Mode, F4: Program Verification *con't*

Estimated Cycle Time

If there are no syntax errors in your program, 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_
```
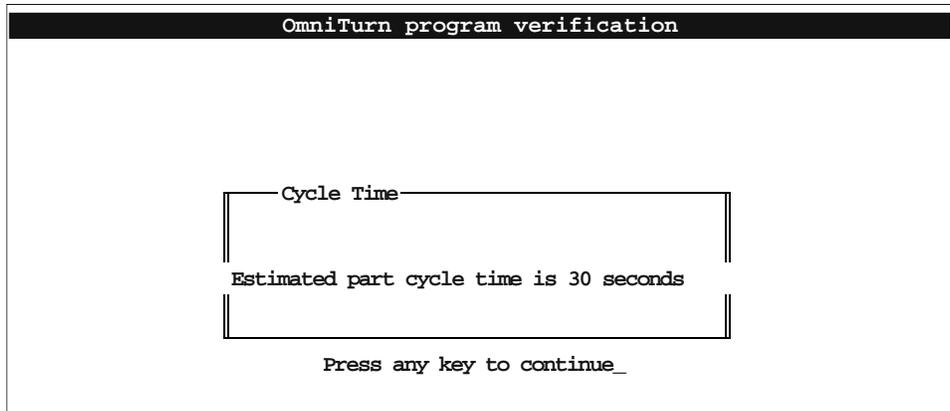
List of Tools

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.

```
                 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
```
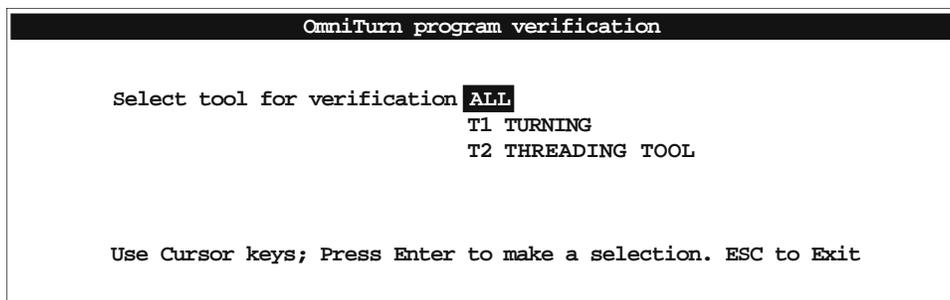
This screen is primarily for choosing which tool to use for the tool path generation. Use the up/dn arrow keys to highlight the desired tool, then press enter to display the tool path.

The default is ALL tools; pressing enter willl display tool pats for all the tools.

Tool path graphics

Select the speed of the graphics display. If you choose **Single Block** you can advance through the program using the Enter to go to the next move. The default is **Full Speed**, which is instantaneous. **Slow Motion** traces the tool path at moderate speed. Press Enter to display the tool paths

```
                    OmniTurn program verification

        Select speed for tool plot:  Single Block
                                      Slow Motion
                                      Full Speed


        Use Cursor keys; Press Enter to make a selection. ESC to Exit
```
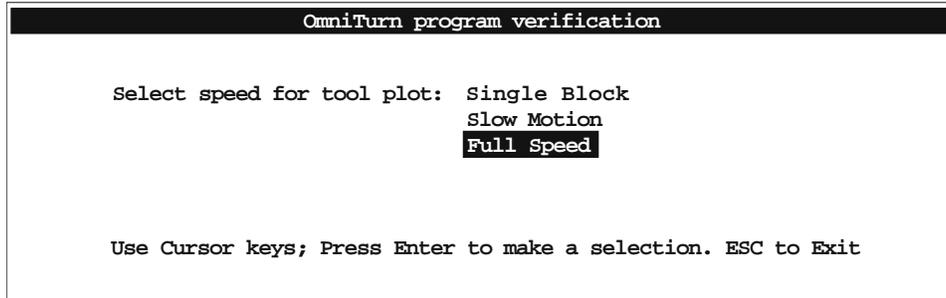
Each tool is shown in a different color. The part contour for each tool is shown as the tool would cut. In this case the turning tool (T1) is a left-hand tool, which cuts on the plus side of the part, and the threading tool (T2) is a right-hand tool, which cuts on the minus side of the part.

```
(Z)oom  (F)ull size  ESC to quit
Window Size: X = .73  Z = 1.21475
Tools:  T1 T2
```
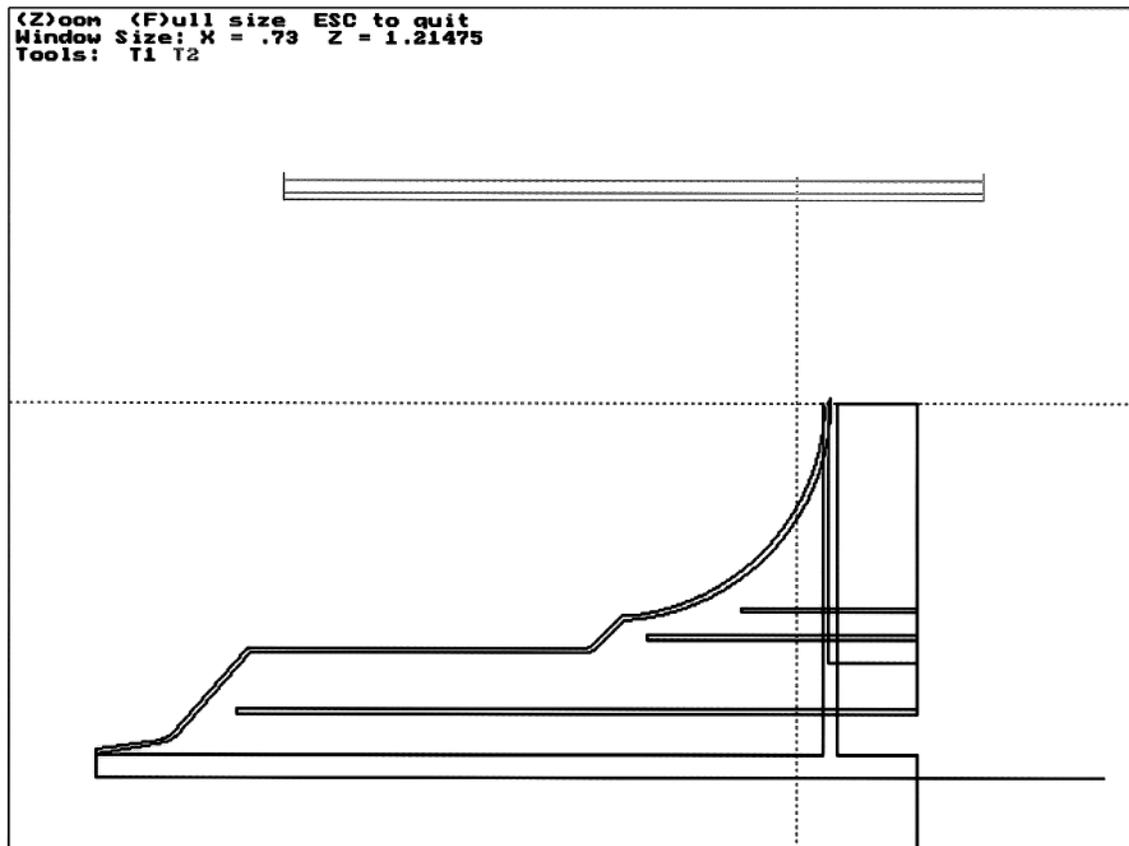
Automatic Mode, F4: Program Verification, con't

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.

(Z)oom (F)ull size ESC to quit
Window Size: X = .73 Z = 1.21475
Tools: T1 T2
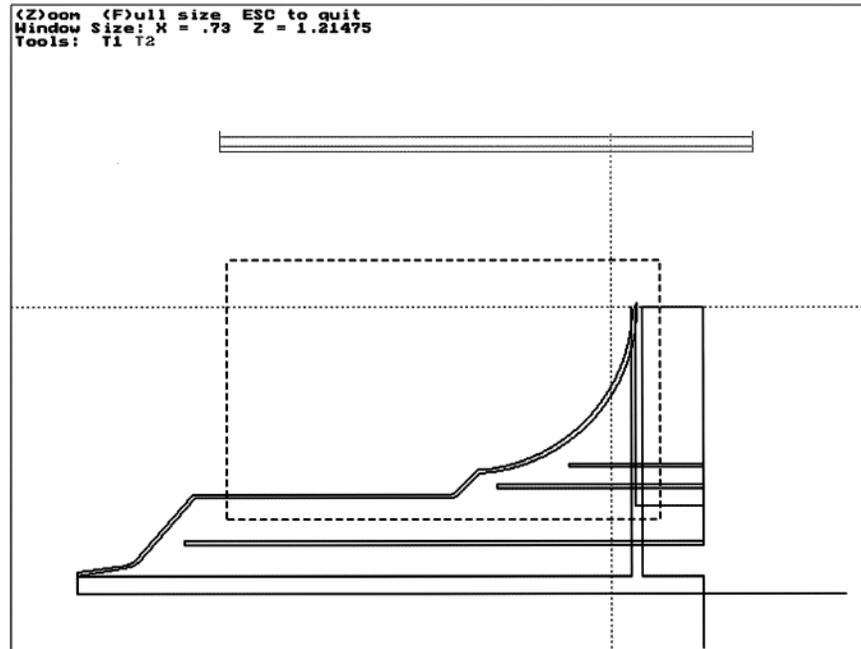
When the window covers the desired area, press Z again (or press Enter) to fill the screen with that section of the tool path, as shown below. If you want to get closer, press Z again to obtain window.

Press Esc to return to Automatic screen

(Z)oom (F)ull size ESC to quit
Window Size: X = .365 Z = .60735
Tools: T1 T2
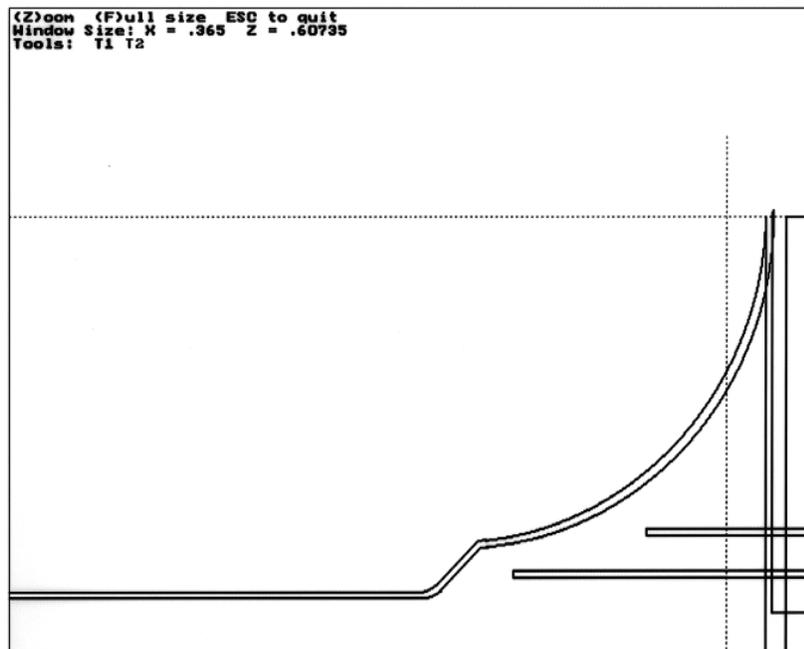
F6 Program Search
This function is used to start the program at a certain tool instead of the beginning.
The program "DEMO" is used in the following example.
This function will allow you to start the program at a certain tool instead of the beginning.
NOTE: If there are less than two tools in your program, 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

```
T 1
T 2
T 0



Use arrow keys to highlight Search Point
Enter to select, or ESC to cancel search
```

arrow keys to select desired tool, then press Enter.
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, Single Block mode, with selected tool highlighted.

```
To Go             c:\programs\DEMO         Position
üüüüü                                      X: +7.86285  ∅
üüüüü             G94F50                    Z: -0.62225
                  T2
                  x-.5Z.2

Switches                                   Feed
C.Cycle Repeat    THREADING TOOL           000.000  IPM
/.Block Skip                               100%
O.Optional Stop
S.Single Block    Press Cycle Start to continue
                  or Esc to terminate program
                                           Speed
                                              0  RPM
Functions                                  100%
F2-Tool Offsets                            Tool  2
F3-Edit
F4-Verify
F5-New File
```

The Progam 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.

NOTE: The File Handler is covered in detail on pages 1.19 - 1.23

The screen is described below:

Directories and disks to search

(a:\ = floppy; d:\ = USB stick)

```
1234A      1235A      2468B      A34          c:\programs\
BUSHING2   COLLAR23   D-0478     DEMO         d:\
FLANGE12   G041414    L12398     SMALTEST     d:\programs\
TESTPROG                                      OTHER
```

Program files in alpbetical order. Use arrow keys to move highlight, or select initial letter or number to highlight first file in series. Press Enter to load selected program into memory.

```
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
```

Options selectable with "Alt" key

```
New path? d:\programs
Current machine file; null
Current directory; c:\programs\
```

Current disk & directory
Program in memory ("null" = no program loaded)
Prompt area

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.

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.

F9 Secondary Offsets

This function calls up the secondary offset table. There are 32 offsets available, including 32 tool nose radius compensation offsets. The Secondary Offset is a powerful feature that allows you to apply an offset to any move with any tool or to adjust the tool path to compensate for tool nose radius.

Note that the numbers 1 - 32 DO NOT refer to tools (as in F2 Tool Offset Table) but refer to Secondary Offsets, which can be applied to any move by preceding the offset number with "D".

Refer to Section 2, Secondary Offsets, and Tool Nose Radius Compensation for full description of the the use of secondary offsets and tool nose compensation.

```
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
```

Entering data:

The X and Z values respond the same as standard tool offsets: X offsets are entered as diameter, but are recorded as radius; Z offsets are recorded as entered. X and Z values are incremental: the offset you enter is added or subtracted from the existing number. The R values, however, are absolute: When you put a value in the R offset table it writes over the existing value and replaces it with the value you enter.

Clearing data:

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 zeros.

Individual offsets can be set to zero by pressing C when asked to enter a correction amount.

F10 Special Functions

This function is used to set the parts counter or to save or load tool offsets.

```
┌────────────────┬──────────────────────────┬────────────────┐
│ To Go          │ c:\programs\newprog3     │ Position       │
│ X: +0.00000    │                          │ X: +0.00000    │
│ Z: +0.00000    │                          │ Z: +0.00000    │
├────────────────┤                          ├────────────────┤
│ Switches       │                          │ Feed           │
│ C.Cycle Repeat │                          │ 000.000   IPM  │
│ /.Block Skip   │  ┌─────────────────────┐ │ 100%           │
│ O.Optional Stop│  │ "L" to load offsets for this program   │
│ S.Single Block │  │ "S" to save offsets for this program   │
│                │  │ "C" to set cycle repeat counter        │
│                │  │ "P" to enable/reset parts counter      │
├────────────────┤  └─────────────────────┘ ├────────────────┤
│ Functions      │                          │ Speed          │
│ F2-Tool Offsets│                          │   0  RPM       │
│ F3-Edit        │                          │ 100%           │
│ F4-Verify      │                          │ Tool  0        │
│ F5-New File    │                          │                │
│ F6-Program Search                                          │
│ F8-File Ops    │                                           │
│ F9-Sec. Offsets│                                           │
│ F10-Spec. Functions                                        │
│ J-Jog Mode     │                                           │
│ M-Manual Data Input                                        │
└────────────────┴───────────────────────────────────────────┘
```

Set Cycle Repeat Counter:

Press **C** at this screen to make a certain number of parts, and then stop.

When C is pressed you will be asked the number of parts you want to make. Enter that number, then press Enter to return to Auto mode. Press C at Auto mode, and Cycles Remaining prompt will appear, ready to count down when you press Cycle Start. The counter will count down each time an M30 or M02 is executed. To clear the counter, press F10, then C.

Set the Parts Counter:

Press **P** at this screen to count the number of parts made.

When P is pressed you will be asked for a number to start counting from. This is an option to continue counting from a previous shift. If starting from zero, press Esc key or Enter key. You will return to Auto mode. Press C in Auto mode to display part count. The counter will count up each time an M30 or M02 is executed. To clear the counter, press F10, then press P.

Load saved offsets for this program:

Press **L** at this screen to *load* the offsets for the current program from the harddrive. If offsets haven't been saved for this program, you will be prompted `Offsets not found-press a key`.

Save the offsets for this program:

Press **S** at this screen to *save* the offsets for the current program to the harddrive. Offsets are saved the C:\PROGRAMS folder (directory) as *filename*.TOF and *filename*.SOF where filename is the name of the program.

TIP: 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.

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
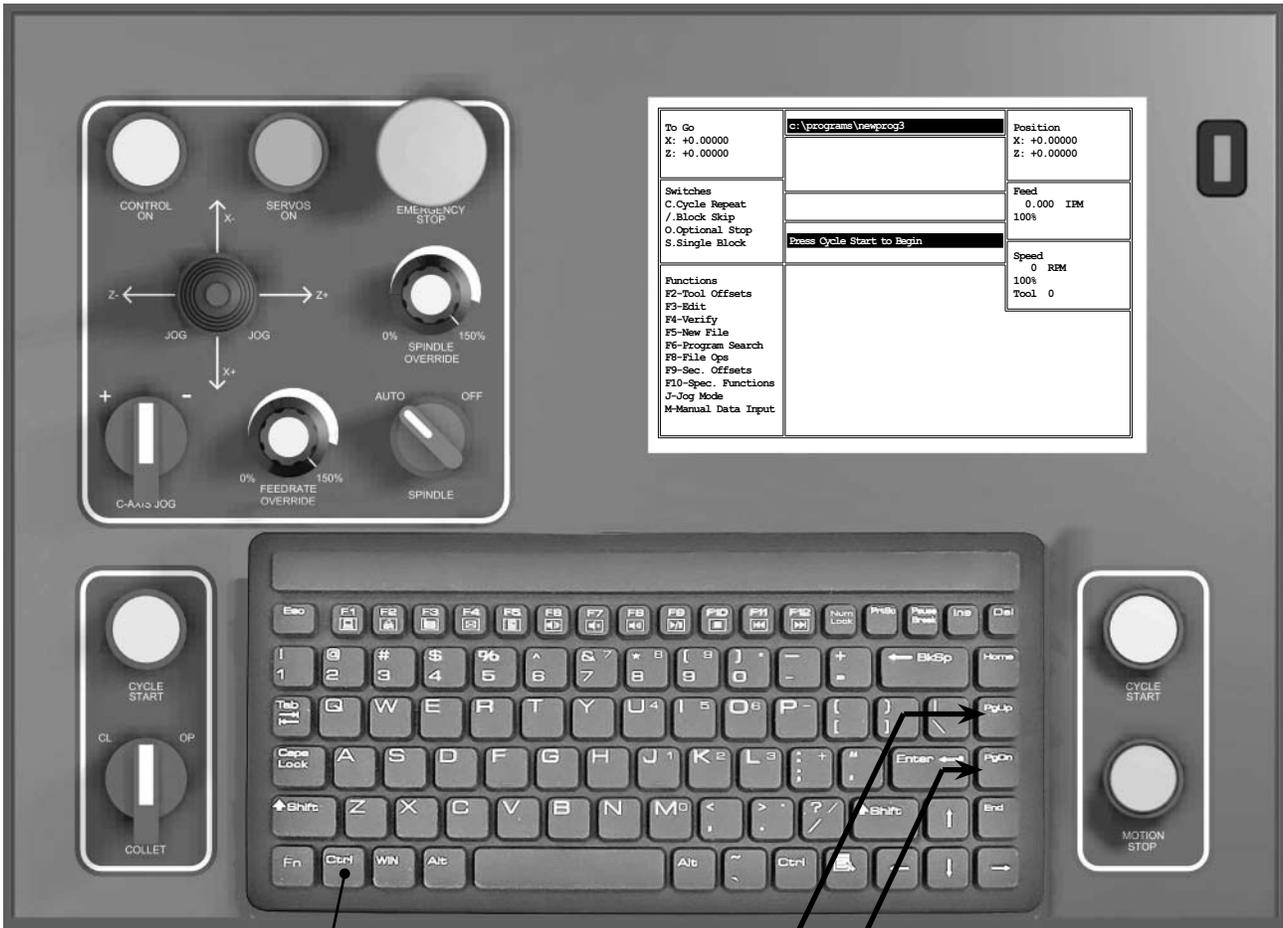
Set M08/M09
(Coolant On/Off)
(PgUp key)

Set M25/M26*
(Parts Catcher Out/In)
(PgDn key)
*Must press Ctrl key to activate

To review the functions of all the front panel controls, refer to pages 1.4 - 1.6

Automatic Mode, F3: Edit Program

F3   **Edit;** used to change existing programs, or create new ones

Note: To create new programs, see page 1.23 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 instead, to invoke File Operations screen, then highlight desired program and press Enter to load it into memory (see 1.17 - 1.20 for details).

To add lines to a program:
• Move the cursor to where you need to add the line.
Note: use Page Up or Page Down to move one screen-full at a time *(see keyboard on following page)*
• Type the new text and press Enter

To change existing text:
• move the cursor to the text to be corrected
• type the new text in
• delete the text being removed

To save changes without exiting editor (do this often while editing)
• Press F2 to save the program

To exit the editor and save corrections made
• Press Alt-X. You will be prompted to press Y to save if you have not saved before exiting. *(Alt-X means press and hold the Alt key, then press X key).*

To exit the editor and NOT save the corrections made
• Press Alt-X; press N to not save.

To save program with different name (useful for creating 'family' of parts)
• Press Shift-F2 to rename program; type new name. *(Shift-F2 means press and hold the Shift key, then press F2 key).*

Note that `C:\PROGRAMS\` before the program name is the disk drive and folder where the program is stored. When you begin typing the new name, the `C:\PROGRAMS\` will disappear, but the new program will be saved to that drive and folder. Press enter to return to editing. Program is saved at this time.
NOTE: If you want to run the new program when you return to automatic mode, you must press F5 (new program) and select the new, renamed program to get it into memeory.

To select text to copy, delete or move:
Press and hold the shift key, then press arrow keys as required to highlight the text. *(To clear the selection, release the shift key, then press arrow key).*

To copy selected text: Press Ctrl-Ins, move cursor to where you want text to begin, then press Shift-Ins.

To delete selected text: Press Del key to delete. *(If you delete the wrong text, press Alt-Backspace to recover.* **Do not move the cursor before Alt-Backspace;** *if you do, you cannot recover.)*

To move selected text: Press Shift-Del, move cursor to where you want text to begin, then press Shift-Ins

Automatic Mode, Editing Keys

Insert Key: use with Ctrl to copy text; or with Shift to paste text; also toggles between 'typeover' and 'insert' text.

Esc Keys: use to cancel certain functions

Function Keys: F1=Help; F2=Save; F7=DOS shell

Delete Key: delete character at cursor

Backspace

Home Key: move to beginning of line

Page Up Key

Page Down Key

End Key: move to end of line

Alt Key: use with Backspace to recover deletions *(before moving cursor)*

Arrow Keys: move cursor left/right/up/dn

Ctrl Key: use with Ins to copy text

Shift Key: use with arrow keys to select text; also with Ins key to paste a copy of text.

CapsLock Key: programs are easier to read if CapsLock is OFF.

Function keys: F1=Help; F2=Save; F7=DOS shell
Arrow keys: Use to move the cursor one character or one line at a time.
Home: molves 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: use with Ctrl and Shift to copy and paste text; use alone to toggle be-
     tween *'typeover'*, which replaces text as you type, and *'insert'*, which
     adds text as you type.
Del: deletes the character at the cursor
Backspace: erases the character to the left of the   cursor; use with Alt
     to recover deletions *(before moving cursor)*
Ctrl: use with Ins to copy text
Shift: use with Ins to past text

Automatic Mode -Notes on the Editor

The text editor can help you manipulate and modify your programs. The editing functions are:
Select text to de
lete, copy or move
This function is used to highlight sections of the program to be erased, copied or moved. To use this function:
• move the cursor to the beginning of the code that you want to select
• press and hold Shift key to start highlighting the text
• move the cursor to the end of the text that you want to select
• if you want to de-select the text release the Shift key and press any arrow key

Now you can copy, delete or move the selected text as follows:

To copy selected text
This function is used to copy the text you've just selected (highlighted).
• press and hold the Ctrl key, then press the Ins key (Ctrl-Ins) to copy the selected text
• move the cursor to where you want the text to be copied (the text will not be highlighted)
• press and hold the Shift key, then press Ins key (Shift-Ins) to insert a copy of the selected text (paste)

To delete selected text
This function is used to copy the text you've just selected.
• press the Del key (if you do this in error, to recover you must press and hold Alt key, the press Backspace *before doing anything else*)

To move selected text
This function is used to move the text you've just selected.
• press and hold the Shift key, then press the Del key (Shift-Del)to cut the selected text
• move the cursor to where you want the text to be moved
• press and hold the Shift key, then press the Ins key (Shift-Ins) to paste the text

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.

To rename program
• press and hold the Shift key, then press the F2 key
• type the new name (note that C:\PROGRAMS\ before the program name is the disk drive and folder where your programs are stored.
• press enter to return to editing (program is saved at this time)

NOTE: If you want to run the new program when you return to automatic mode, you must press F5 (new program) and select the new, renamed program to get it into memeory.
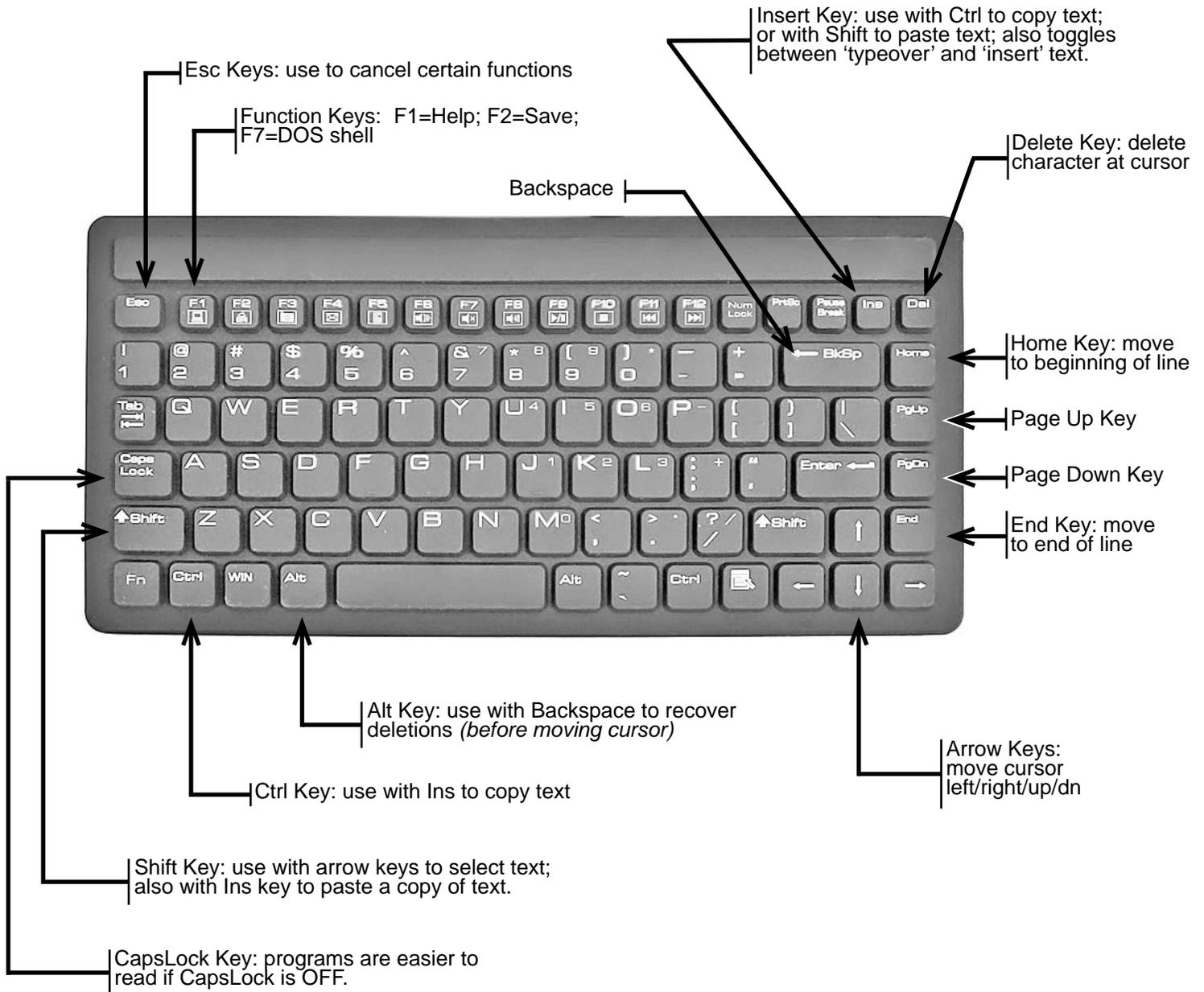
This page intentionally blank

OmniTurn - Trouble shooting guide, G4 CNC

Table of Contents

OmniTurn - Trouble shooting guide, G4 CNC

> *NOTE: More comprehensive troubleshooting details are available online at OmniTurn.com > Support*

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.

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 the 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 6.10). 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 6.13) 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 for loose motor cable; set power off, then disconnect and reconnect.

    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
- 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
- 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

- 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 deacceleration are too short (standard 5hp drive).
      Parameters can be set to change these: see "Inverter Spindle Drive" pg 6.31

Spindle Encoder Counts (Ctrl-S Diagnostic)


```
1.Slow                          Position
2.Medium                        X: +0.00000
3.Fast                          Z: +0.00000
4. 0.00005"/.01°
5. 0.00100"/.1°
6. 0.01000"/1°                  Feed
7. 0.10000"/10°                   0.0000 IPM
8. 1.00000"/90°                 100%
9.Establish Home
H.Go Home
T.Set Tool  M.Manual            Speed
Data Input                      (215) RPM (*)
A.Automatic                     100%
                                Tool 0
```

Encoder counts:
one revolution = 4000,
then count resets to 0

Star indicates Ctrl-S
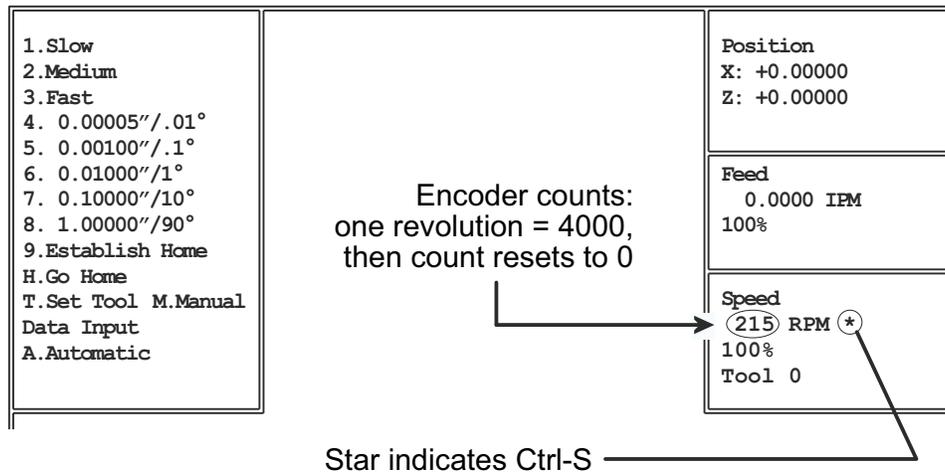
Spindle Encoder Counts (Ctrl-S Diagnostic)

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.

To cancel Ctrl-S press Ctrl-S again.

Input Monitor Function (Alt-I Diagnostic)

```
1.Slow                                      Position
2.Medium                                    X: +0.00000
3.Fast                  X+ Jog:      1      Z: +0.00000
4. 0.00005"/.01°        X- Jog:      1
5. 0.00100"/.1°         Z+ Jog:      1
6. 0.01000"/1°          Z- Jog:      1
7. 0.10000"/10°         C+ Jog:      1      Feed
8. 1.00000"/90°         C- Jog:      1        0.0000 IPM
9.Establish Home        Left CS:     1        100 %
H.Go Home               Right CS:    1
T.Set Tool              Motion Stop: 1
M.Manual Data Input     Collet Open: 1      Speed
A.Automatic             Collet Close:1        0   RPM
                        Spin. Off/Auto: 0   100 %
                        Guards:      0      Tool  0
                        Inhibit:     1
                        M91 input:   1
                        M93 input:   1
                        M95 input:   1
                        X Ref LS :   1
                        Z Ref LS :   1
                        0 indicates active input
```

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.

Also, when door interlock switch (Guards, above) is open (1), the Motion Stop is shown as on (0) regardless of its actual condition.

To cancel Alt-I press Alt-I again.

Temporarily Bypass Guards (Alt-G Diagnostic)

If the slide won't jog, and "Guards" stays at 1 and Motion Stop stays at 0 regardless of whether the door is open or closed, press and hold the Alt key, then press G key (Alt-G), then press Esc. Note that Motion Stop will change to 1, but Guards will remain at 1.

Select a jog speed and attempt to jog: if the slide jogs with Alt-G active, the problem is a bad door switch or open connection in the door interlock string.

Alt-G is cancelled if Auto mode is selected.

Following Error Display (Ctrl-E Diagnostic)

```
1.Slow              Alt-C will enable C axis        Position
2.Medium                                            X: +0.00000
3.Fast                                              Z: +0.00000
4. 0.00005"/.01°              Following Error
5. 0.00100"/.1°     X:  0
6. 0.01000"/1°      Z: -1
7. 0.10000"/10°     C:  0                           Feed
8. 1.00000"/90°     Set X and Z to 70 at Medium jog   0.0000 IPM
9.Establish Home                                    100%
```

Following Error Display (Ctrl-E Diagnostic)

Servo Errors:
If your OmniTurn occasionally throws a servo error, especially between tool calls, one of the servo amplifiers may need to be readjusted.

Pressing Ctrl-E in Jog Mode enables display of the Following Error. At rest, both X and Z should show Following Error between -1 and +1. Press the 2 key (Medium Jog), and jog the slide back and forth in each axis. The Following Error should be between 65 and 75 while jogging.

If the numbers at rest are not 0 ±1, you must readjust the servo amplifier Balance and the motion card Zero. If the numbers while jogging are not 70 ±5, you must readjust the servo amplifier Signal Gain and Tach. See pages 6.30 - 6.32 for servo amplifier adjustment instructions.

_____

Encoder Count Captured (Ctrl-H Diagnostic)

```
1.Slow              Alt-C will enable C axis        Position
2.Medium                                            X: +0.00000
3.Fast                                              Z: +0.00000
4. 0.00005"/.01°    Encoder Count Captured
5. 0.00100"/.1°     X:  0
6. 0.01000"/1°      Z:  0
7. 0.10000"/10°     C:  0                           Feed
8. 1.00000"/90°                                       0.0000 IPM
9.Establish Home                                    100%
                    ctrl-h_screens.dpp
```

Encoder Count Captured Display (Ctrl-H Diagnostic)

Positioning Problems:
Your parts are not maintaining size; measurements are different part to part.

Using Ctrl-H to diagnose repeatability problems
Pressing Ctrl-H in Jog Mode enables display of the Encoder Count 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.

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).

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 repeatablilty or size problem.

Encoder Count Captured (Ctrl-H Diagnostic) *con't*

```
1.Slow              Alt-C will enable C axis        Position
2.Medium                                            X: +0.20535
3.Fast                                              Z: -0.20615
4. 0.00005"/.01°    Encoder Count Captured
5. 0.00100"/.1°     X:   4000
6. 0.01000"/1°      Z: -4000
7. 0.10000"/10°     C:   0                          Feed
8. 1.00000"/90°                                       0.0000 IPM
9.Establish Home                                    100%

                    ctrl-h_screens.dpp
```

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.

Mechanical Positioning Problems:
The slide has mechanical problems with repeating a size. (Both axes read 4000, Ctrl-H test)

- 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, turn servos on and position the slide close to the spindle. Mount an indicator on the tooling plate 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 support bearing, etc)

To check if the slide doesn't repeat, write a program to rapid back and forth in the axis you're testing. Mount an indicator on the tooling plate and zero it at the program end location. Be sure to slow down the feed before the indicator hits. Run cycle repeat for 100 cycles and see if the indicator is zero at the end.

After running the program and studying the way the indicater 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, G4 CNC

To Disassemble the CNC Control
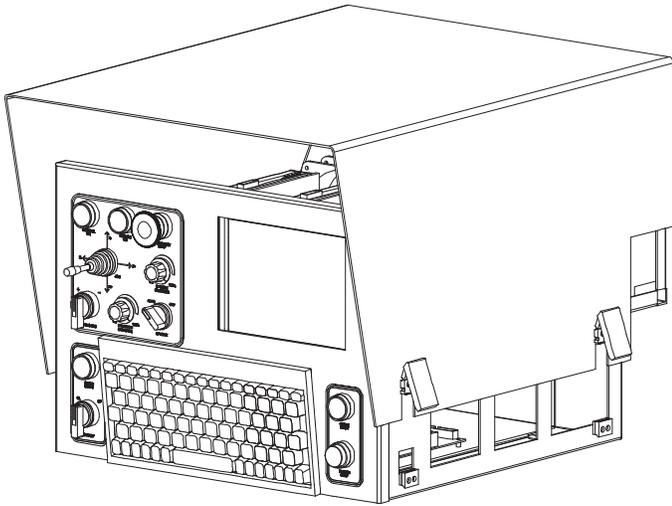
Un-plug power cord, then remove the blue cover.
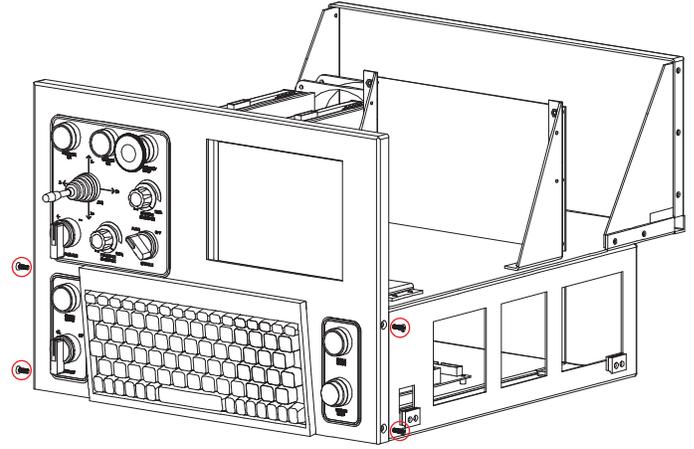
Remove four screws holding front panel.
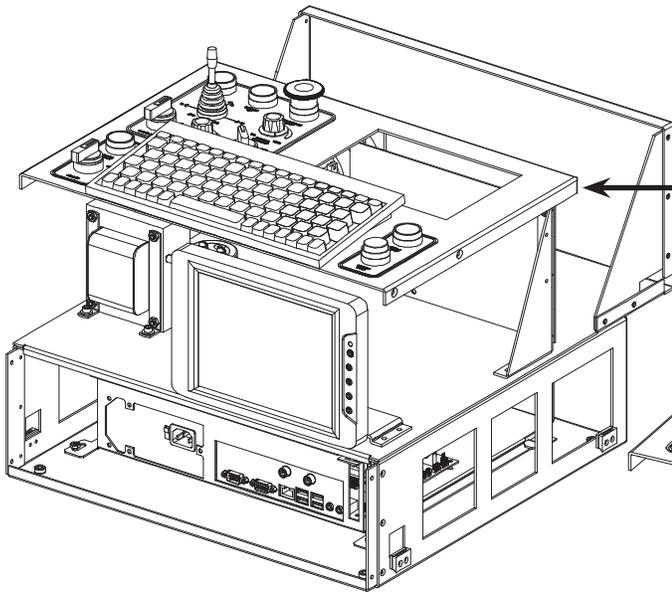
It is not necessary to disconnect cables to front-panel.
Stow panel gently on top of components as shown at left.

The computer is held in with hex-nuts at the front; it slides under tabs at the rear.

NOTE: Three plugs must be disconnected before removing the computer, two on headers indicated at right, and one power supply connector near power supply.
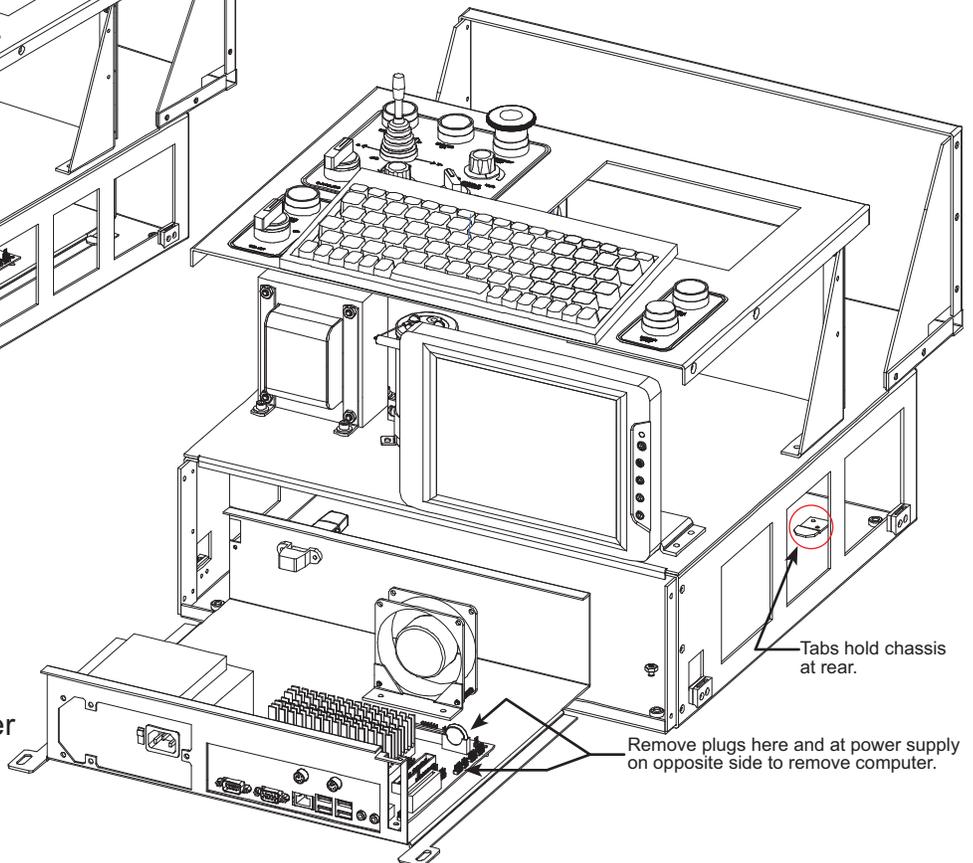
Tabs hold chassis at rear.

Remove plugs here and at power supply on opposite side to remove computer.

OmniTurn - Trouble shooting guide, G4 CNC

CNC Top Chassis
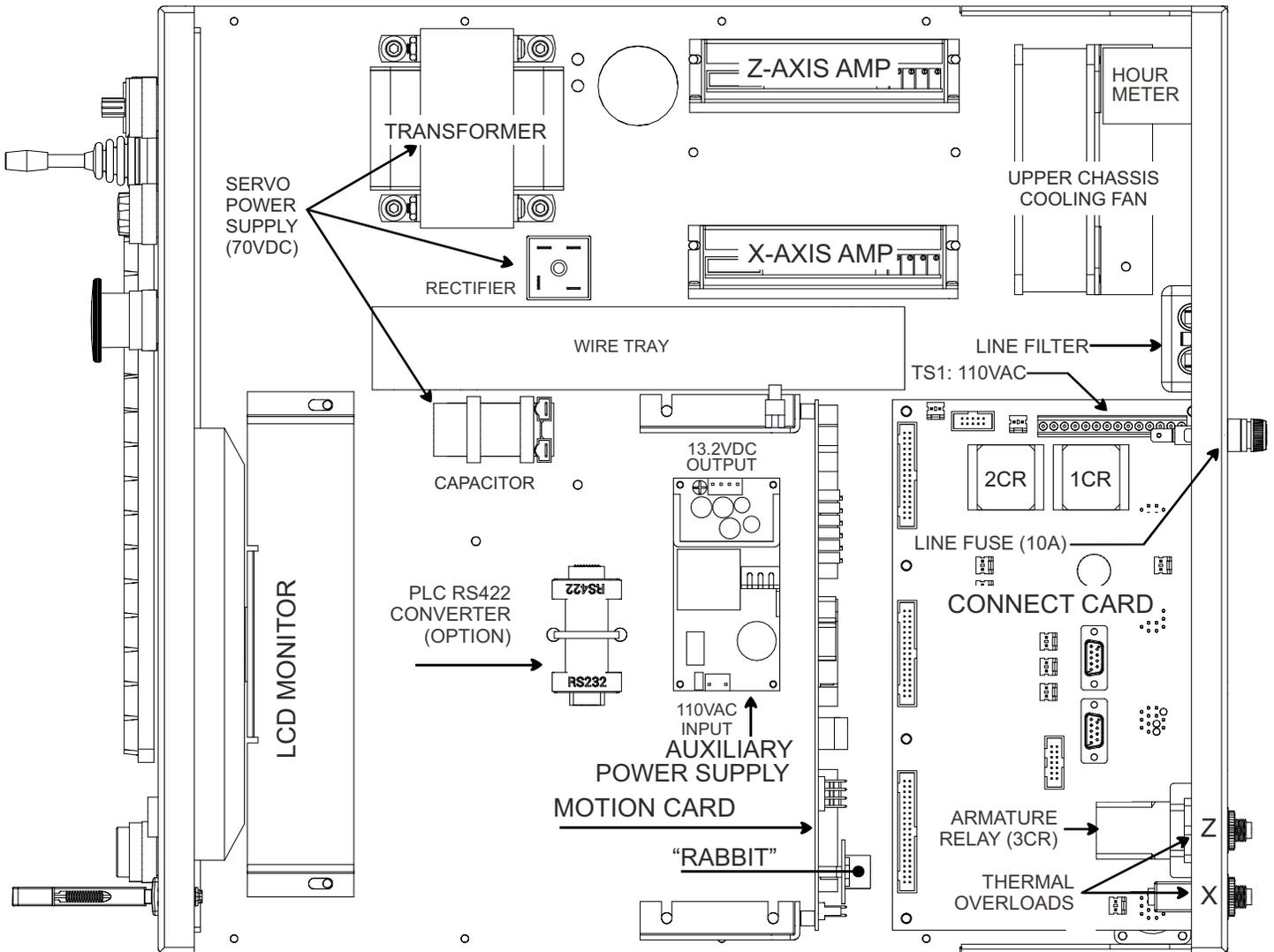
The illustration above shows the location of the main components. Detailed drawings of the Servo Amps, Connect Card and Motion Card will be found on the following pages.

The illustration below shows the location of components on the rear panel of the top chassis.

CNC Rear Panel (front view)

Connect Card

TS1

AC IN
COMP
FAN
SERVOS
AUX PS
PLC

MISC

CN4
ARMATURE
RELAY COIL

BAUMULLER
ENCODER

Z-AXIS MTE

X-AXIS MTE

SPINDLE
or
C-AXIS

2CR
COMPUTER
P0WER SUPPLY

Y-AXIS AMP

Z-AXIS AMP

X-AXIS AMP

CN102
ENCODER 5VDC

13.2VDC
INPUT
CN8

1CR
SERVOS ON

CN5   CN6   CN7

A01   A02   A03
HOUR AUXILIARY
METER OUTPUTS

CN302
ANALOG
SIGNALS

FROM FRONT
PANEL
CONNECT CARD
CN1

CN10 CN11

MON PWR   PLC 12V

JP1*

CONNECT CARD

CN2
VERTICAL
CUTOFF
(VCO)

"CONTROL ON"
SWITCH TO
MOTHER BD
CN9

CRDY (Pin 17)

cnc_disassembly-M830.dpp

CN401
OUTPUTS

*CUT JP1 TO ADD
EXTERNAL E-STOP
SEE *Spindle Drive
Control Logic, CCA*

CN402
INPUTS

CN101
ENCODERS

CNC Front-Panel Connect Card

CNFP11
Y-AXIS JOG

CNFP10
SPIN OFF/AUTO

CNFP7
C-AXIS JOG

CNFP8
JOG STICK

CNFP5
E-STOP

CNFP4
SERVOS ON

CNFP9
CONTROL ON

CNFP1
RT CYCLE START

CNFP3
MOTION STOP

NC ELECTRONICS
FPCON   REV 0

CNFP1
LFT CYCLE START

CNFP6
COL CLSR

CN403

CN1

front_panel_connect_card.dcd

"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 first.

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

NOTE: U414 is spare driver ic

CNC Computer Chassis

COOLING FAN

G4 CNC
COMPUTER
CHASSIS

To CN9 on
Connect Card
(Power Switch
Connector)

MEMORY SLOT

BATTERY

TO MOTION
CARD (CN304)

MOTHER BOARD
EPIA VB7009

PS ON (GRN)
(Short to BLK)

COM3
(optional
PLC)

POWER SUPPLY
CONNECTOR

ATX
POWER
SUPPLY

5V always on (VIO)

RED WIRE
THIS END

PCI
SLOT

ETHERNET
(TO MOTION CARD)

110VAC IN (FROM
CONNECT CARD)

Cable to *USB port
on front panel*

cnc_disassembly-VB7009.ppp

KEYBOARD

MONITOR

COM1 (TO CNC
BACK PANEL)

USB HARD DRIVE

USB KEYBOARD

OmniTurn - Trouble shooting guide, G4 CNC

REAR PANELFUSE
10A
P
N
G

110VAC
LINE N

CN9

COMPUTER
+12V

2CR

TS1

1
2
3
4
5
6
7
8
9
10
11
12
13
14

P
N
G

UPPER FAN

P
N

110VAC
TO SERVOS
P
N

110VAC
FOR PLC
P
N

1CR

12 8
1CR

+12SO

9 8
1CR   FROM "SERVOS ON" PB

6
10

JP1
EXT E-STP
(HDR103 SP CCA)

9
5

E-STOP
STR NG

SERVO
ENABLE

6 7   6 7

TO SERVO AMPS

CONNECT CARD

CRDY
CN401-17

CN1

FRONT
PANEL
CONNECT
CARD

CN1

CRDY

CNFP9
4
3
2
1
CTRL ON

E-STOP

CNFP5
1
2
BLK   BLK
BRN   WHT

CN403

CN403

PALM BOX
E-STOP

CN401

CN402

CN101

MOTION
CARD

CN102   CN102
ENC   ENC
+5V   +5V

CN304

CN8   CN10
COM   AUX +12V   MONITOR PWR

COMP +5V   COMP +12V

AUX PS

MONITOR

LOWER FAN

COMPUTER
PWR SUP
(ATX)

MOTHER BD

PWR SUP
START

ETHERNET

G4 CNC

OmniTurn
CNC BLOCK DIAGRAM
E-STOP STRING

cnc_block-diagram-G4.dcd NOV 2024
Corrected ouptuts, TS1 7-12

OmniTurn - Trouble shooting guide, G4 CNC

SEE "MISC CABLE AND OPERATOR'S STATION" FOR PALM BOX DETAIL

SERVOS ON   CTRL ON   E-STOP   PALM BOX E-STOP   PALM BOX CYC START   LEFT CYC START   RIGHT CYC START   MOT STOP   COLLET OP CL   C-JOG C+ C-   X & Z JOG   SPINDLE OFF AUTO   SPINDLE OVERRIDE   FEEDRATE OVERRIDE

FRONT PANEL CONNECT CARD

CN1   CN403

MOTION CARD

CONNECT CARD

TS1   110VAC L NE N   REAR PANEL FUSE 10A   1CR   2CR   +12SO MISC-9   ARMATURE RELAY   CRDY (PIN 17)   CRDY   CN401   CN403   CN303

SERVOS   ZIPLDR   CN8   CN9   CN10   +12V GND   CN304

UPPER FAN   MONITOR +13V   LOWER FAN

AUXILLIARY PWR SUP   +13V   GND   VGA   MOTHER BD   PWR SUP START   COM2   COMPUTER PWR SUP (ATX)

PLC CNC REAR PANEL

14	1 TxD+
2	2 TxD-
17	3 RxD+
RS422 COVERTER 5	7 RxD-
7	5 5V COM
22	10 +5VDC
	13 13VDC
	14 0V
	8 110VAC N
	4 110VAC P

TO ZIP LOADER PLC (OPTIONAL)

OmniTurn
FRONT PANEL INTERFACE
ZIP LOADER OPTION

G4 CNC w/ZipLoader

Fp_sch-G4.dcd NOV 2024
Changed outputs, TS2 7-12

OmniTurn - Trouble shooting guide, G4 CNC

G4
INVERTER DRIVE

Omni*Turn*
SERVO DRIVE INTERFACE

OmniTurn - Trouble shooting guide, G4 CNC

PWB

COOL PUMP

220VAC LINE IN — RED / WHT / BLK

| M08 | M25 | DISC-1 |

LIGHT IS ON WHEN ACTIVE

AUX OUTPUT

HDR301
- 8/7 → Sp.Prge SldBrk
- 6/5 → ColClsr Lube
- 4/3 → Collet Clamp
- 2/1 → Collet Unclamp

HDR106 2/1 → M08
HDR105 2/1 → M25
HDR308 2/1 → 0V 12V
HDR307 2/1 → 0V 12V
HDR303 2/1 → M17
HDR304 2/1 → M21
HDR305 2/1 → M23
HDR306 2/1 → M27

HDR302 SPINDLE STATUS
- 1 Spin is OFF
- 2 COM
- 3 Spin is ON

HDR310 Door Interlock Bypass Key
HDR309 Door Interlock Switch (NC) (OPENS w/ DOOR)

P302 (Zip Loader Option) (Sp.Cab Door)
HDR108 ZIP LDR
- 1 — ZIP PLC Y7 (FAULT)
- 2 — ZIP PLC Y4 (CLS COL)
- 3 — ZIP PLC Y5 (OPEN COL)
- 15 — ZIP PLC 0V
- 16 — ZIP PLC X14 (INTERLOCK)

HDR104A EXT FLT 2/1
HDR104 EXT FLT 2/1

YASKAWA GA500
WARNING — Risk of electrical shock.
CHARGE

Fuses: F6 2A | F5 2A | F4 2A | F3 25A | F2 25A | F1 25A

F1,F2,F3 = INVERTER
F5 & F6 = COOLANT PUMP

DYNAMIC BRAKE RESISTOR (40Ω 500W)

Parameters w/5hp WEG GT75 & GTJr

To re-initialize: se tA1-03 to 2220
After initialization set as follows:

A1-02=2(Open loop vector mode)
C1-01=1.0(Accel)
C1-02=1.0(Decel)
C6-01=0(Heavy-duty mode)
E1-01=210(Input voltage)
H2-02=1(P1-PC closes at zero speed)
E1-04=134(Max freq)
L3-04=3(Stall prevention w/DBR)
L8-05=0(Disable input phase-loss error)
L8-07=0(Disable output phase-loss error)
E105=230(Max voltage)

G4 CNC
Printed Wiring Board

| M17 | M21 | M23 | M27 |
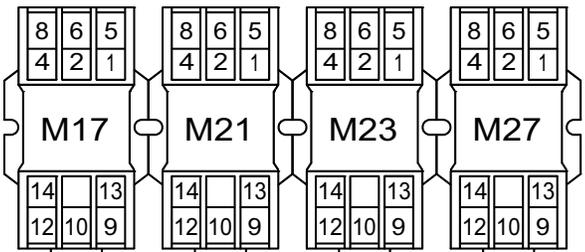
OPTIONAL OUTPUTS (994-39-107)
Pin14 = +12V (from HDR303-1,304-1,305-1,306-1)
Pin13 = Signal (from HDR303-2,304-2,305-2,306-2)
Pins 9, 10, 12 = Common
Pins 9-1, 10-2, 12-4 = Normally Closed (NC)
Pins 9-5, 10-6, 12-8 = Normally Open (NO

Omni*Turn*
INVERTER SPINDLE DRIVE PANEL

SPINDLE DRIVE
ENCLOSURE

RED
WH
T
BLK
220VAC IN

PLC CABLE [OPTIONAL]

PLC
[OPTIONAL]

L1 L2 L3
DISC-1
T1 T2 T3

STEP
DOWN
XFMR

110VAC FOR CNC & WORKLAMP

WORK LAMP

FUSES
(5A)

WH
T
BLK

CIRCUIT
CARD
ASSY
(PWB)

MISCELLANEOUS CABLE (CCA HDR101)

ENCODER CABLE

HDR203

DIRECT MOUNT
ENCODER

X AXIS MTR

CONTROL
TERMINALS

FUSES
(25A)

RED   BLK   L3
RED   RED   L2
RED   RED   L1

BRAKE
RESISTOR

Z AXIS MTR

SPINDLE
DRIVE
(INVERTER)

MOTOR POWER

FUSES
(2A)

6   8 M08

RED   BLK

PALM BOX

COOLANT PUMP

Z AXIS AMP

X AXIS AMP

CONNECT CARD

Z-AXIS   X-AXIS   SPINDLE   MISC   L NE   PLC   RS232
                          or   (OPT)   (COM1)
                        C-AXIS

PLC (OPT)

RSR22
CONV

COM2

AUX PS

MOTION CONTROL CARD (vertical)

OmniTurn BLOCK DIAGRAM
Inverter Spindle Drive G4
(SPINDLE PANEL PWB)

CNC CONTROL

otbdinv_G4_pwb.dcd feb 2015
(re-routed encoder wiring; added detail to spindle panel)

OmniTurn - Trouble shooting guide, G4 CNC

SPINDLE DRIVE CABINET

SPINDLE DRIVE CIRCUIT CARD

HDR103 Ext E-Stp
1
2

HDR101 MISC

Signal	Pin
E-STOP1	12
E-STOP2	14
+12SO	9

HDR309 ByPass
1
2

HDR310 INTLK
1
2

HDR108 ZipLdr
1
5
4
3
2

HDR104 ExtFlt
1
2

INHIBIT	8
GUARDS	13
0V	2

HDR301 AirPak
2 M13
4 M12

HDR106 M08
2
1

HDR105 M25
2
1

12V

CR202 M0
CR201 M01

M13	3
M12	7
M08	5
M25	1
M04	4
M03	6

HDR203 SpinDrv
20 SPIN FWD
30 DRVCOM
22 SPIN REV
1 SP.ANALOG COMMON
2 SPINDLE ANALOG

| SG | 11 |
| VREF | 10 |

MISC CABLE

MISC CONNECTOR AT REAR PANEL OF CNC

CNC CONTROL

CONNECT CARD
CN1
8
10
3
1CR

FRONT PANEL CONNECT CARD
CN1
8
10
3

MOTION CARD
CN403 CN403B CN403

CN402
26
29
30

26
29
30

CN401
1
5
3
2
4
6

1
5
3
2
4
6

CN302
6
5

6
5

E-STOP1 12
E-STOP2 14
+12SO 9
INHIBIT 8
GUARDS 13
0V 2
M13 3
M12 7
M08 5
M25 1
M04 4
M03 6
SG 11
VREF 10

ENCODER

ENC 5V	2
ENC MARK	1
ENC.A	3
ENC. B	4
0V	5

CN101
32
18
13
16
14

32
18
13
16
14

CONNECT CARD   MOTION CARD

ENCODER CABLE

SPINDLE MOTOR

ENCODER CONNECTOR AT REAR PANEL OF CNC

OPERATORS STATION (PALM BOX)

E-STOP
WHT 2
YEL

Clamp
VIO
GRN
Unclamp
BLU

Palm Buttons
BRN
ORG

RED

3	
8	
7	
6	
10	
4	
5	
1	

CNEST
1 TO TOP OF E-STOP
2 SWITCH ON CNC

Pin	Color	
3	BLK	
8	BRN	
7	RED	
6	ORG	
10	YEL	
4	GRN	
5	BLU	
1		

CN403B
21 TO COL CLS SW ON CNC
20 TO COL OPEN SW ON CNC
26 TO COM ON COL CP/CLS SW
2 TO LEFT CY.ST SW ON CNC
4 TO RIGHT CY.ST SW ON CNC

NOTE: THESE WIRES ALSO GO TO CN403 ON MOTION CARD

CONNECTORS AT LEFT-HAND MOTION CARD BRACKET

EXTENSION CABLE

OP STA CONNECTOR AT REAR PANEL OF CNC

INVERTER DRIVE

G4 CNC

OmniTurn
MISCELLANEOUS CABLE AND OPERATORS STATION

misc_cb_inv-G4_pwb.dcd dec 2020
minor cosmetic changes

OLD TB1 DIAGNOSTIC POINTS

TB1-1   Analog Com HDR101-11
TB1-2   Analog Sig. . HDR101-10
TB1-5   M03 . . . . HDR101-6
TB1-7   M04 . . . . HDR101-4
TB1-9   +12V HDR101-9
TB1-10 M08 . . . . HDR101-5
TB1-11 M25 . . . . HDR101-1
TB1-12 M13 . . . . HDR101-3
TB1-15 M12 . . . . HDR101-7
TB1-19 0V HDR101-2
TB1-21 CC Lube . . . HDR301-6
TB1-23 E-Stop HDR104-1
TB1-25 Inhibit. HDR101-8
TB1-30 Guards HDR101-13
revised 0812

FOR SINGLE PHASE
CONNECT LINE TO
L1 AND L2.

2A F6   12 8   BLK   M
10 6   RED   COOLANT PUMP
F5
F4   9   5
M08

BLK   FT1   10A FT3
220VAC   110VAC
WHT   FT2   5A STEP DOWN XFMR
(MOUNTED ON XFMR)   OUTLETS FOR WORKLAMP AND CONTROL

SW1

220VAC LINE

BLK   L1   2T1   RED   25A F3   RED   L1
WHT   L2   4T2   RED   F2   RED   L2
RED   L3   6T3   RED   F1   BLK   L3

SPINDLE MOTOR
T1
T2   M
T3

YASKAWA GA500 (5HP)

To LUBE TIMER   0V

Spin is ON   3
COM   2
Spin is OFF   1

SpinStat HDR302

12V   SPIN   16   1   4   8
CR203

MISC HDR101

M03   6
M04   4

SpinDrv HDR203

M04   22   YEL   S2
DRVCOM   M03   20   RED   S1

CR201
M03
CR202   M04   16   12V

P201 Max Signal
SPINDLE ANALOG   2   WHT   A1
VREF   1 0
SG   11   SP.ANALOG COMMON (SG*)   1   BLK   AC

B1   RED

HDR109 PLC

PLC 0V (BLK)   4
PLC COM (BLK)   3   0V
PLC +12V (BLU)   2   12V
PLC X0 (GRY)   1

0V   12V
/FLT   CR101

RESET   24   BLU   S4
DRVCOM   30   ORG   SN

DBR

B2   RED

DBR = 40 OHMS 500W

SP FLT LAMP

HDR107 SpinRst
1
2   12V
3
4

SPNDL RST PB

HDR113
1
2
OPEN = FLT Ext Inhibit

0V   33   GRN   MC   Opens on Fault
ALM   28   BRN   MB
SG   3

CIRCUIT CARD

ga500_sc-G4_pwb.dcd jan 2021

*Omni*Turn
INVERTER SPINDLE DRIVE w/CCA

OmniTurn - Trouble shooting guide, G4 CNC

5hp Spindle Drive Wiring (Yaskawa GA-500)

MC: GRN   0V to CN203-33

MB: BRN   FLT to CN203-28

SN: ORG   COM from CN203-30

A1: WHT   Analog RPM SIG to CN203-2

AC: BLK   Analog RPM COM to CN203-1

S4: BLU   RESET from CN203-24

S2: YEL   M04 from CN203-22

S1: RED   M03 from CN203-20

T1-T3 Motor Out

L3: BLK   Line In

L2: RED   Line In

L1: RED   Line In

B2: RED   Braking Rsisistor

B1: RED   Braking Resistor

COOL PUMP

220VAC LINE IN — RED WHT BLK

RED — L1
WHT — L2
BLK — L3
L1C
L2C

B1 (+)
B2
B3

Θ1
Θ2

RED — U
WHT — V
BLK — W

MOTOR OUT

YASKAWA SERVO AC K 200V
SGDV-330A01A

CN3
CN7
CN1
CN8
CN2

CHARGE

HDR301
8 / 7 / 4 — Sp.Prge SldBrk
6 / 5 — ColClsr Lube
4 / 1 — Collet Clamp
3 / 2 — Collet Unclamp

HDR106 — 2 / 1 — M08
HDR105 — 2 / 1 — M25
HDR308 — 2 / 1 — 0V / 12V
HDR307 — 2 / 1 — 0V / 12V
HDR303 — 2 / 1 — M17
HDR304 — 2 / 1 — M21
HDR305 — 2 / 1 — M23
HDR306 — 2 / 1 — M27

HDR302 SPINDLE STATUS
1 — Spin is OFF
2 — COM
3 — Spin is ON

HDR310 — 2 / 1 — Door Interlock Bypass Key
HDR309 — 2 / 1 — Door Interlock Switch (NC) (OPENS w/ DOOR)

P302 (Zip Loader Option) (Sp.Cab Door)
HDR108 ZIP LDR
1 — 1 — ZIP PLC Y7 (FAULT)
2 — 2 — ZIP PLC Y4 (CLS COL)
3 — 3 — ZIP PLC Y5 (OPEN COL)
4 — 15 — ZIP PLC 0V
5 — 16 — ZIP PLC X14 (INTERLOCK)

HDR104A EXT FLT — 2 / 1
HDR104 EXT FLT — 2 / 1

M08 — 8 6 5 / 4 2 1 / 14 13 / 12 10 9
M25 — 8 6 5 / 4 2 1 / 14 13 / 12 10 9

LIGHT IS ON WHEN ACTIVE

AUX OUTPUT

DISC-1
L1 L2 L3
T1 T2 T3

F6 — 2A
F5 — 2A
F4 — 2A
F3 — 25A
F2 — 25A
F1 — 25A

F1,F2,F3 = INVERTER
F5 & F6 = COOLANT PUMP

M17 — 8 6 5 / 4 2 1 / 14 13 / 12 10 9
M21 — 8 6 5 / 4 2 1 / 14 13 / 12 10 9
M23 — 8 6 5 / 4 2 1 / 14 13 / 12 10 9
M27 — 8 6 5 / 4 2 1 / 14 13 / 12 10 9

OPTIONAL OUTPUTS (994-39-107)

Pins 9, 10, 12 = Common
Pins 9-1, 10-2, 12-4 = Normally Closed (NC)
Pins 9-5, 10-6, 12-8 = Normally Open (NO
Pin14 = +12V (from HDR303-1,304-1,305-1,306-1)
Pin13 = Signal (from HDR303-2,304-2,305-2,306-2)

G4 CNC
Printed Wiring Board

Omni*Turn*

RED
WHT
BLK

220VAC IN

SPINDLE DRIVE
ENCLOSURE

PLC
[OPTIONAL]

PLC CABLE [OPTIONAL]

L1 L2 L3
DISC-1
T1 T2 T3

FUSES
(5A)

WHT
BLK

STEP
DOWN
XFMR

110VAC FOR CNC & WORKLAMP

WORK LAMP

CIRCUIT
CARD
ASSY
(PWB)

C-AXIS STEP & DIRECTION (CCA HDR201)

C-AXIS ENCODER & ADD'L I/O (CCA HDR102)

MISCELLANEOUS CABLE (CCA HDR101)

HDR203

YASCONN PWB

1CN

FUSES
(25A)

RED   RED
RED   WHT   L1
RED   BLK   L2
              L3

FUSES
(2A)

6   8   M08

RED   BLK

SGDV-330
SERVO
DRIVE

TACHOMETER

MOTOR POWER

C-AXIS
MOTOR

X AXIS MTR

Z AXIS MTR

PALM BOX

COOLANT PUMP

Omni*Turn* BLOCK DIAGRAM
C-Axis Spindle Drive G4
(SPINDLE PANEL PWB)

otbdcxy_G4_pwb.dcd Feb 2015

Z AXIS AMP

X AXIS AMP

MISC

Z-AXIS   X-AXIS   SPINDLE
or
C-AXIS

C-PULSE

L NE

PLC
(OPT)

RS232
(COM1)

CONNECT CARD

PLC (OPT)

RSR22
CONV

COM2

AUX PS

MOTION CONTROL CARD (vertical)

CNC CONTROL

OmniTurn - Trouble shooting guide, G4 CNC
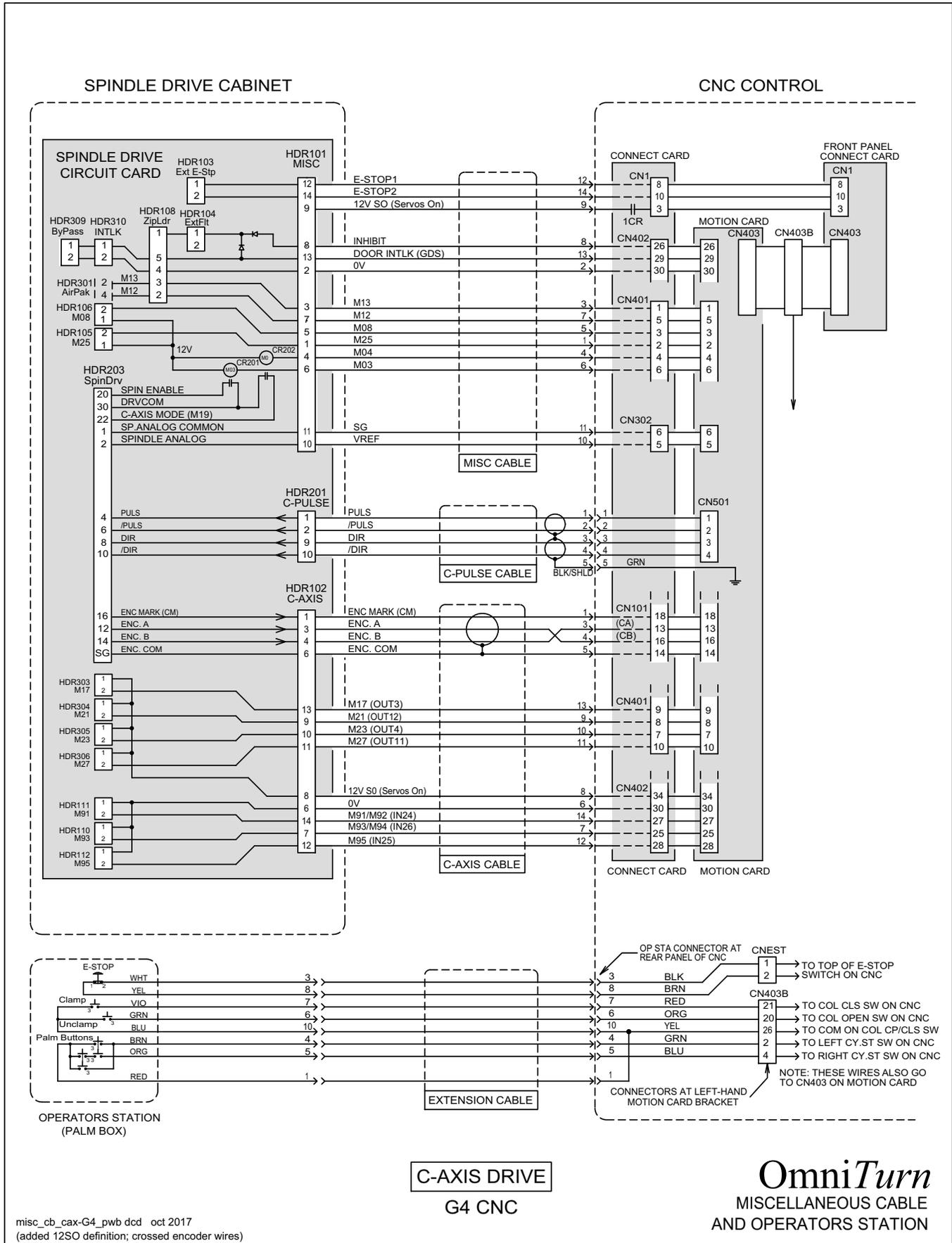
C-AXIS DRIVE

G4 CNC

OmniTurn
MISCELLANEOUS CABLE
AND OPERATORS STATION

misc_cb_cax-G4_pwb dcd oct 2017
(added 12SO definition; crossed encoder wires)
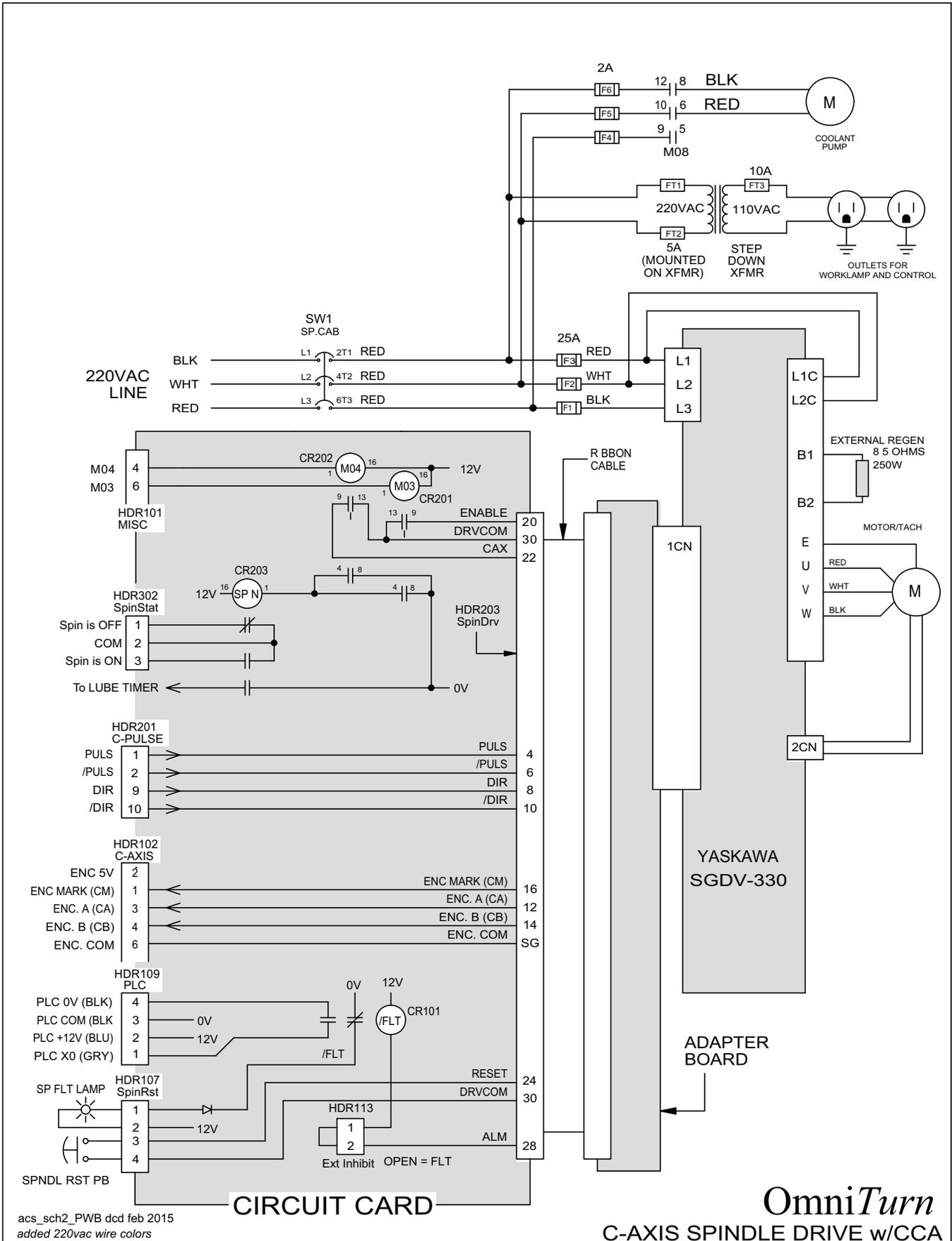
OmniTurn - Trouble shooting guide, G4 CNC

Spindle Drive Circuit Card Assembly

TB1 no longer exists; the following table lists corresponence between the TB1 test points and the current location of signals and voltages.

TB1-1	Analog Com	. . .HDR101-11
TB1-2	Analog Sig.	. . . .HDR101-10
TB1-5	M03	. . . . .HDR101-6
TB1-7	M04	. . . . .HDR101-4
TB1-9	+12V	. . . . .HDR101-9
TB1-10	M08	. . . . .HDR101-5
TB1-11	M25	. . . . .HDR101-1
TB1-12	M13	. . . . .HDR101-3
TB1-15	M12	. . . . .HDR101-7
TB1-19	0V	. . . . .HDR101-2
TB1-21	CC Lube	. . . . . .HDR301-6
TB1-23	E-Stop	. . . . .HDR104-1
TB1-25	Inhibit.	. . . . .HDR101-8
TB1-30	Guards	. . . . . . .HDR101-13

revised 0812

HDR101   **1**   HDR102   **1**

HDR101 MISC

HDR102 CAX

HDR101A   **1**   HDR102A   **1**

HDR101A MISC

HDR102A CAX

1   P201

EXT ESP   BAL   M91   M95   M93

HDR103   **1** **1** **1**

HDR111   HDR112   HDR110   HDR301

AIR PACK   **1**

HDR106   M08   **1**

HDR105   M25   **1**

HDR311   24VDC   HDR308   12V   **1**

PS1   • -
       • 12
       • 0V2   HDR307   12V   **1**

1   **1**

PWB

HDR201

C PULSE   HDR303   M17   **1**

HDR203   HDR202   401CR   HDR304   M21   **1**

SPINDLE DRIVE   SPIN ENC   COL CLSR LUBE   HDR305   M23   **1**

HDR306   M27   **1**

202CR   203CR

M4 / M19   SPIN ON   SPIN STAT

HDR302

M3 / ENAB   201CR

HDR113   DOR INLK

HDR310

EXT INH   NOT FLT   101CR   KEY SW

HDR309   **1**

1

JPR   HDR109   HDR108   HDR107   HDR104

PLC   ZIP LDR   RESET   EXT FLT

1   **1**   **1**   **1**

spindle_panel_pwb.dcd

HDR101 MISC

Connector pins (14 to 1):
E-STOP2, DOOR INTLK (GDS), E-STOP1, SG, VREF, 12V, INHIBIT, M12, M03, M08, M04, M13, 0V, M25

HDR102 C-AXIS

Connector pins (14 to 1):
M91/M92 (IN24), M17 (OUT3), M95 (IN25), M27 (OUT11), M23 (OUT4), M21 (OUT12), 12V, M93/M94 (IN26), SG. COM, ENC. B (CB), ENC. A (CA), ENC 5V, ENC MARK (CM)

HDR103 EXT E-STOP — 2 1

P201 SPINDLE MAX

HDR111 M91/M92 — 1 2
HDR112 M95 — 1 2
HDR110 M93/M94 — 1 2

M91 = Delay program until NOT shorted
M92 = Delay program until shorted
M93 = Delay program until NOT shorted
M94 = Delay program until shorted
M95 = Jump to subroutine 1 if shorted

HDR301

8	+ Sp.Prge
7	4 SldBrk
6	+ ColClsr
5	5 Lube
4	+ Collet
3	1 Clamp
2	+ Collet
1	2 Unclamp

HDR106 — 2 1 — + M08
HDR105 — 2 1 — + M25

HDR311
3 — 24V COM
2 — 0V
1 — +24V

HDR308 — 2 1 — 0V / 12V
HDR307 — 2 1 — 0V / 12V

PWB

HDR201 — C PULSE

HDR203 (SPINDLE DRIVE)

1	SPINDLE.ANALOG COM
2	SPINDLE ANALOG
4	PULS
6	/PULS
8	DIR
10	/DIR
12	ENC. A (CA)
14	ENC. B (CB)
16	ENC MARK (CM)
20	ON or FWD
22	CAX or REV
24	RESET
26	+12V
28	ALM
30	DRVCOM

HDR202 — SPIN ENC

401CR — COL CLSR LUBE
202CR — M4 / M19
201CR — M3 / ENAB
101CR — NOT FLT

203CR — SPIN ON

HDR303 — 2 1 — + M17
HDR304 — 2 1 — + M21
HDR305 — 2 1 — + M23
HDR306 — 2 1 — + M27

HDR302 SPINDLE STATUS
1 — Spin is OFF
2 — COM
3 — Spin is ON

HDR310 — 2 1 — DOR INLK (OPENS w/ DOOR)
HDR309 — 2 1 — KEY SW

HDR113 EXT INHIBIT — 1 2
OPEN = FLT

HDR109 PLC — 1 2 3 4
PLC X0 (GRY)
PLC +12V (BLU)
PLC COM (BLK)
PLC 0V (BLK)

HDR108 ZIP LDR — 1 2 3 4 5

HDR107 RESET — 1 2 3 4

HDR104A — 1 2
HDR104 — 1 2
EXT FLT

P302 (Option) (Sp.Cab Door)
1 — ZIP PLC Y7 (FAULT)
2 — ZIP PLC Y4 (CLS COL)
3 — ZIP PLC Y5 (OPEN COL)
15 — ZIP PLC 0V
16 — ZIP PLC X14 (INTERLOCK)

SPNDL RST PB
SPNDL FLT LAMP

Omni*Turn*
SPINDLE DRIVE PRINTED WIRING BOARD

spindle_cab_pwb_lbl.dcd jan 2023
(added HDR104A)

HDR103 *MUST CUT TRACE ON CONNECT CARD AT JP1 TO ENABLE
Ext E-Stp SEE PAGE 6.10 FOR LOCATION

OPEN = FLT*

KEY SW: BYPASS
(SPINDLE DRV DOOR)
HDR309
ByPass

HDR310
INTLK
Door Intlk

HDR101
MISC

12 E-STOP1
14 E-STOP2

HDR108
ZipLdr
ExtFlt 1

HDR104A
ExtFlt
/Flt
0V

HDR104
ExtFlt
/Flt
0V

8 INHIBIT

Door Intlk 5
0V 4

13 DOOR INTLK (GDS)
2 0V

Clmp Col 2
UnClmp Col 3

HDR308 HDR307
12V
0V

9 12V SO (Servos On)

HDR301
AirPak

HDR105
M25
12V

1 M25

Collet
Unclamp

3 M13
7 M12

Collet
Clamp

Sp.Prge
SldBrk
0V
12V

HDR106
M08
12V

5 M08

ColClsr
Lube

12V
CR202
M04

4 M04
6 M03

M03
CR201

HDR203
SpinDrv

ON or FWD 20
DRVCOM 30
CAX or REV 22
SPINDLE ANALOG 2
(INVERTER ONLY)

10 VREF
11 SG

SP.ANALOG COMMON (SG*) 1

*SG = ODD#s HDR201 1-17

P201
Max Signal

LUBE
TIMER
0V

12V
SPIN
CR203

0V

See table page
6.25 for old TB1
diagnostic points

HDR302
SpinStat

Spin is OFF 1
COM 2
Spin is ON 3

HDR201
C-PULSE

PULS 4
/PULS 6
DIR 8
/DIR 10

TO CAX
SERVO DRIVE
(CAX ONLY)

1 PULS
2 /PULS
9 DIR
10 /DIR

FROM MOTION CARD
(CAX ONLY)

HDR202
SP.ENC

SIG GND SG
+12V 26
12V

6 ENC. COM
4 0V
1 ENC. B
2 ENC.A
3 ENC MARK
5 ENC 5V

FROM INVERTER
MOTOR ENCODER
(INVERTER ONLY)

HDR311
24V
+24V 1
0V 2
24V COM 3

PS1
+12V
0V
-12V

+12V
0V

HDR102
ENCDR

2 ENC 5V
1 ENC MARK (CM)
3 ENC. A (CA)
4 ENC. B (CB)
6 ENC. COM

TO MOTION CARD
(BOTH TYPES)

FROM CAX
SERVO DRIVE
(CAX ONLY)

ENC MARK (CM) 16
ENC. A (CA) 12
ENC. B (CB) 14
ENC. COM SG

HDR109
PLC

CR101
/FLT
/FLT

12V 0V

4 PLC 0V (BLK)
3 PLC COM (BLK)
2 PLC +12V (BLU)
1 PLC X0 (GRY)

0V
12V

RESET 24
DRVCOM 30
ALM 28

HDR107
SpinRst

12V

SPNDL FLT LAMP

SPNDL RST PB

HDR113
Ext Inhibit

OPEN = FLT*

*MUST CUT TRACE AT HEADER

HDR303
M17
+12V

13 M17 (OUT3)

HDR304
M21

9 M21 (OUT12)

HDR305
M23

10 M23 (OUT4)

HDR306
M27

11 M27 (OUT11)

HDR111
M91

8 12V SO (Servos On)
6 0V

HDR110
M93

14 M91/M92 (IN24)
7 M93/M94 (IN26)

HDR112
M95
0V

12 M95 (IN25)

CIRCUIT CARD

OmniTurn
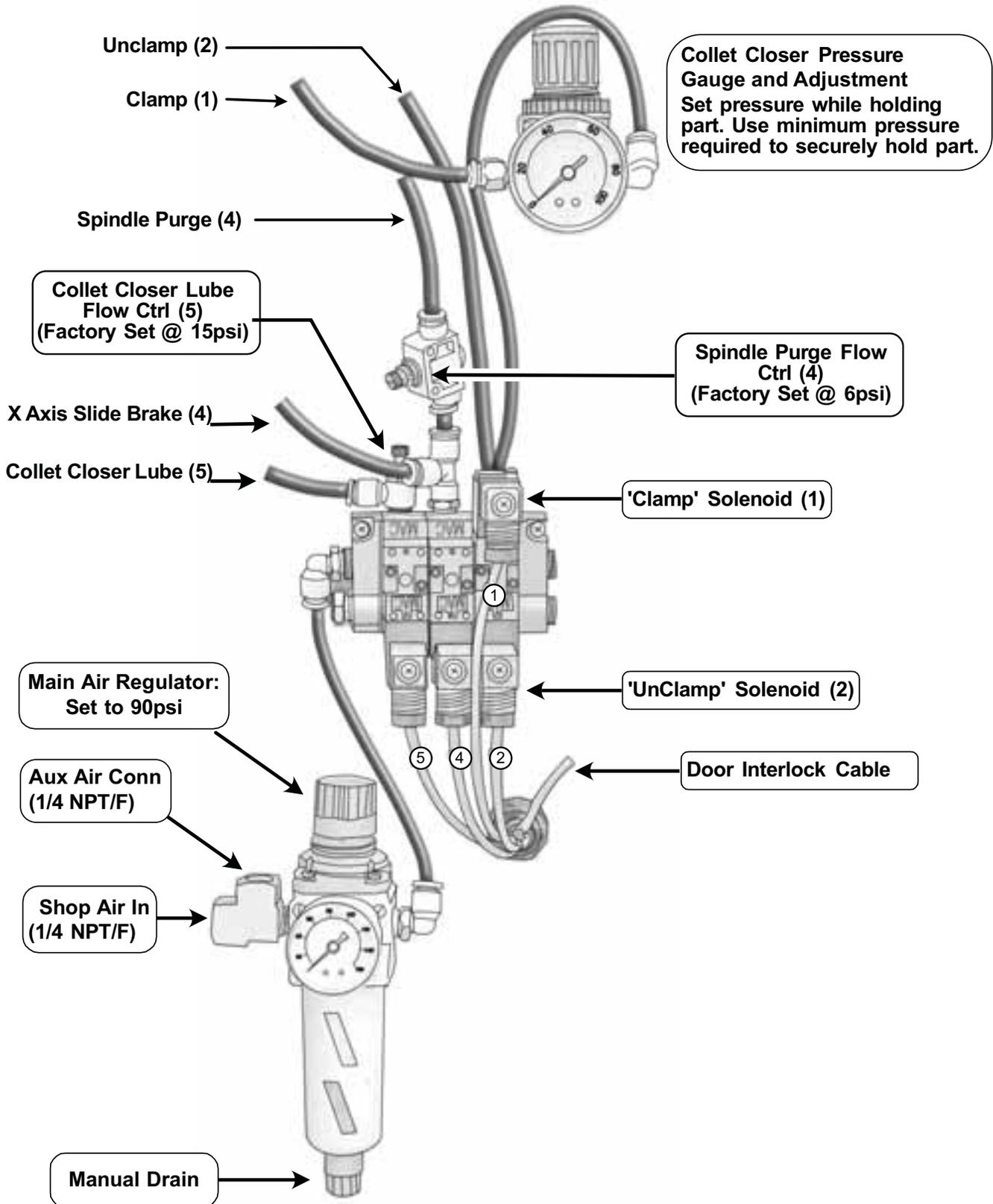
spindle_cab_pwb_sch.dcd feb 2024
(corrected pg# for HDR103; clarified trace-cut HDR113))

SPINDLE DRIVE CONTROL LOGIC (CCA)

GT75 Air-Pack

Unclamp (2)

Clamp (1)

Collet Closer Pressure Gauge and Adjustment Set pressure while holding part. Use minimum pressure required to securely hold part.

Spindle Purge (4)

Collet Closer Lube Flow Ctrl (5) (Factory Set @ 15psi)

Spindle Purge Flow Ctrl (4) (Factory Set @ 6psi)

X Axis Slide Brake (4)

Collet Closer Lube (5)

'Clamp' Solenoid (1)

Main Air Regulator: Set to 90psi

'UnClamp' Solenoid (2)

Aux Air Conn (1/4 NPT/F)

Door Interlock Cable

Shop Air In (1/4 NPT/F)

Manual Drain

OmniTurn - Trouble shooting guide, G4 CNC

Four Way
Dual Solenoid ①

Ex

In ②

To Collet Clamp

R

Regulator
(Adj)

IN

FR

Filter/Regulator

To Collet UnClamp

③

Four Way
Single Solenoid

Ex

In

To Parts Catcher (Optional)

④

Three Way

Ex

In

To Spindle Bearings

Flow Control
(6psi back pressure)

⑤

To Slide Brake

Ex

In

To Collet Closer Lube

Flow Control
(15psi back pressure)

*Omni*Turn
GT AIR PACK

gt_air5.dcd Sep 2010

OmniTurn - Trouble shooting guide, G4 CNC

Setting OMNITURN Servo Amplifiers

> **NOTE:** *The amplifiers are pre-set at the factory and should only rarely require slight adjust-mets to following error. Step 2 (Zero Adjust) and Step 3 (Following Error) 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
Jewlers common screwdriver or trim-pot adjustment tool

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

Adjust BALANCE pot for Zero Volts between pins 4 & 5 with Servos On.

Put DVM probes on 4 & 5
Brown & Black wires on both axes

amp adjustments1-G4.dpp

Setting OMNITURN Servo Amplifiers, con't

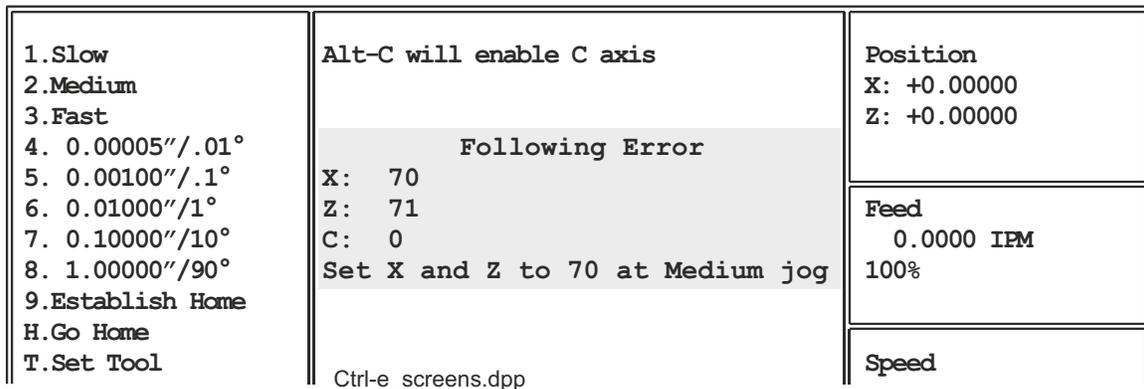### *2. Motion Card Zero Adjustment (Ctrl-E)*
In Jog Mode, open Following Error display box. (Press and hold Ctrl key, then press 'E' key).
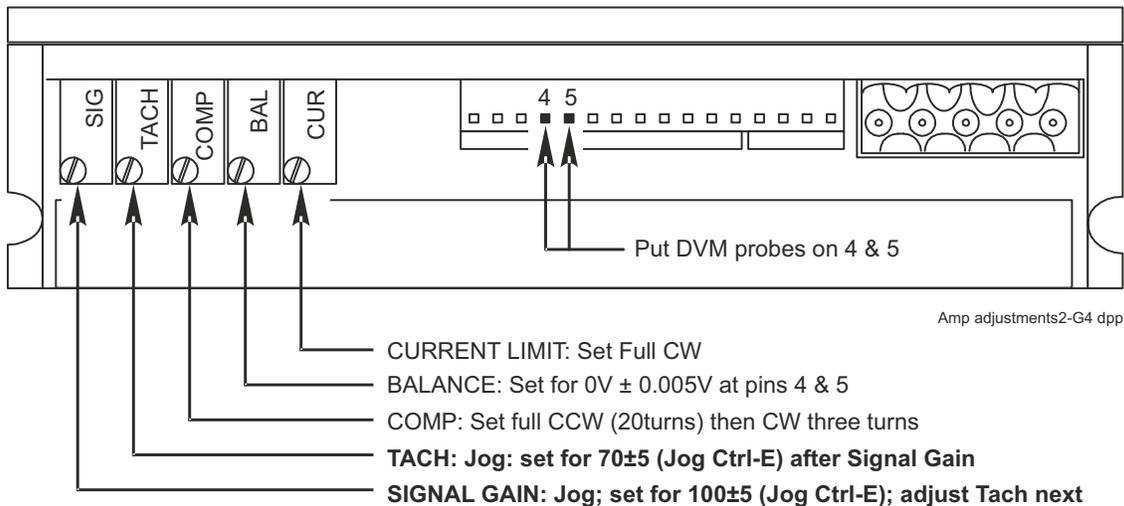On the Motion Card, adjust the X and Z Axis Zero pots for zeros at the X: and Z: locations.
Note that +/- 1 is acceptable; the numbers may alternate between 0 and 1.

```
1.Slow              Alt-C will enable C axis        Position
2.Medium                                            X: +0.00000
3.Fast                                              Z: +0.00000
4. 0.00005"/.01°            Following Error
5. 0.00100"/.1°     X:  0
6. 0.01000"/1°      Z: -1                           Feed
7. 0.10000"/10°     C:  0                             0.0000 IPM
8. 1.00000"/90°     Set X and Z to 70 at Medium jog  100%
9.Establish Home
H.Go Home
T.Set Tool            Ctrl-e_screens.dpp            Speed
```

MOTION CARD ZERO ADJUSTMENT

X-Axis }
Z-Axis } Adjust for Zero Following Error (Ctrl-E in Jog)

adjust_pots2.dpp
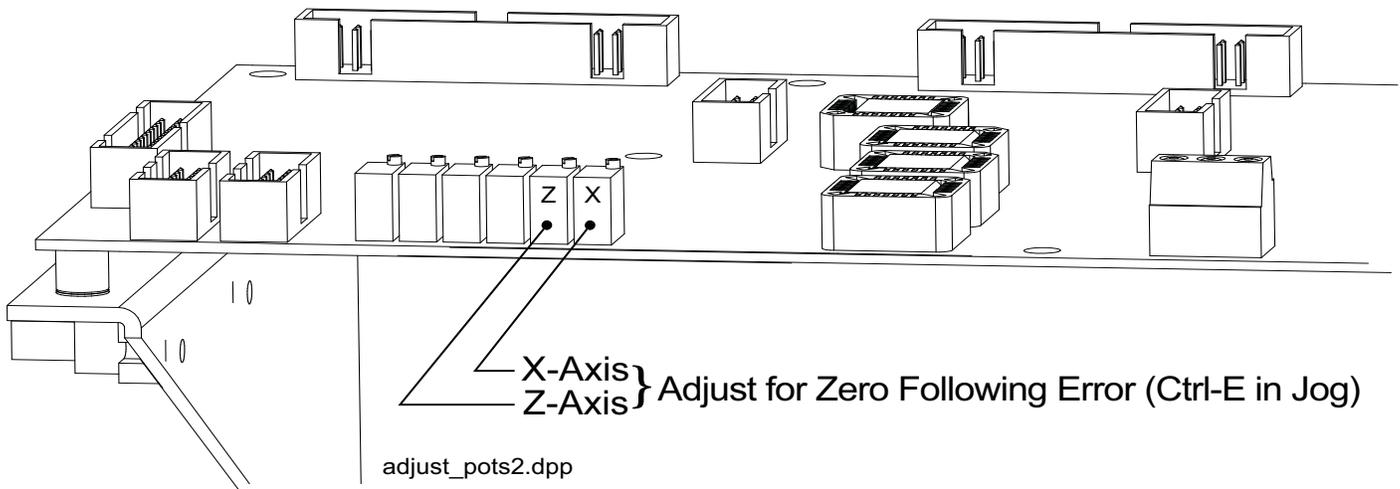
Setting OMNITURN Servo Amplifiers, con't

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 ± 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              Alt-C will enable C axis        Position
2.Medium                                            X: +0.00000
3.Fast                                              Z: +0.00000
4. 0.00005"/.01°            Following Error
5. 0.00100"/.1°     X:   70
6. 0.01000"/1°      Z:   71
7. 0.10000"/10°     C:   0                          Feed
8. 1.00000"/90°     Set X and Z to 70 at Medium jog   0.0000 IPM
9.Establish Home                                    100%
H.Go Home
T.Set Tool
                    Ctrl-e_screens.dpp              Speed
```

SERVO AMPLIFIER FOLLOWING ERROR ADJUSTMENT

Amp adjustments2-G4 dpp

CURRENT LIMIT: Set Full CW
BALANCE: Set for 0V ± 0.005V at pins 4 & 5
COMP: Set full CCW (20turns) then CW three turns
TACH: Jog: set for 70±5 (Jog Ctrl-E) after Signal Gain
SIGNAL GAIN: Jog; set for 100±5 (Jog Ctrl-E); adjust Tach next

Put DVM probes on 4 & 5

4. Re-check Servo Amp balance and re-adjust as required
Verify that the voltage between pins 4 & 5 is still 0V ± 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.

SERVO MOTOR REPLACEMENT INSTRUCTIONS
Following pages illustrate servo motor, support unit and motor coupler replacement for both axes

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 3/8-16 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: ιf you are set up on a job you must re-set all offsets.

Servo Motor, Support Unit (Thrust Assy), and/or Coupler Replacement)

X-Axis Support Unit
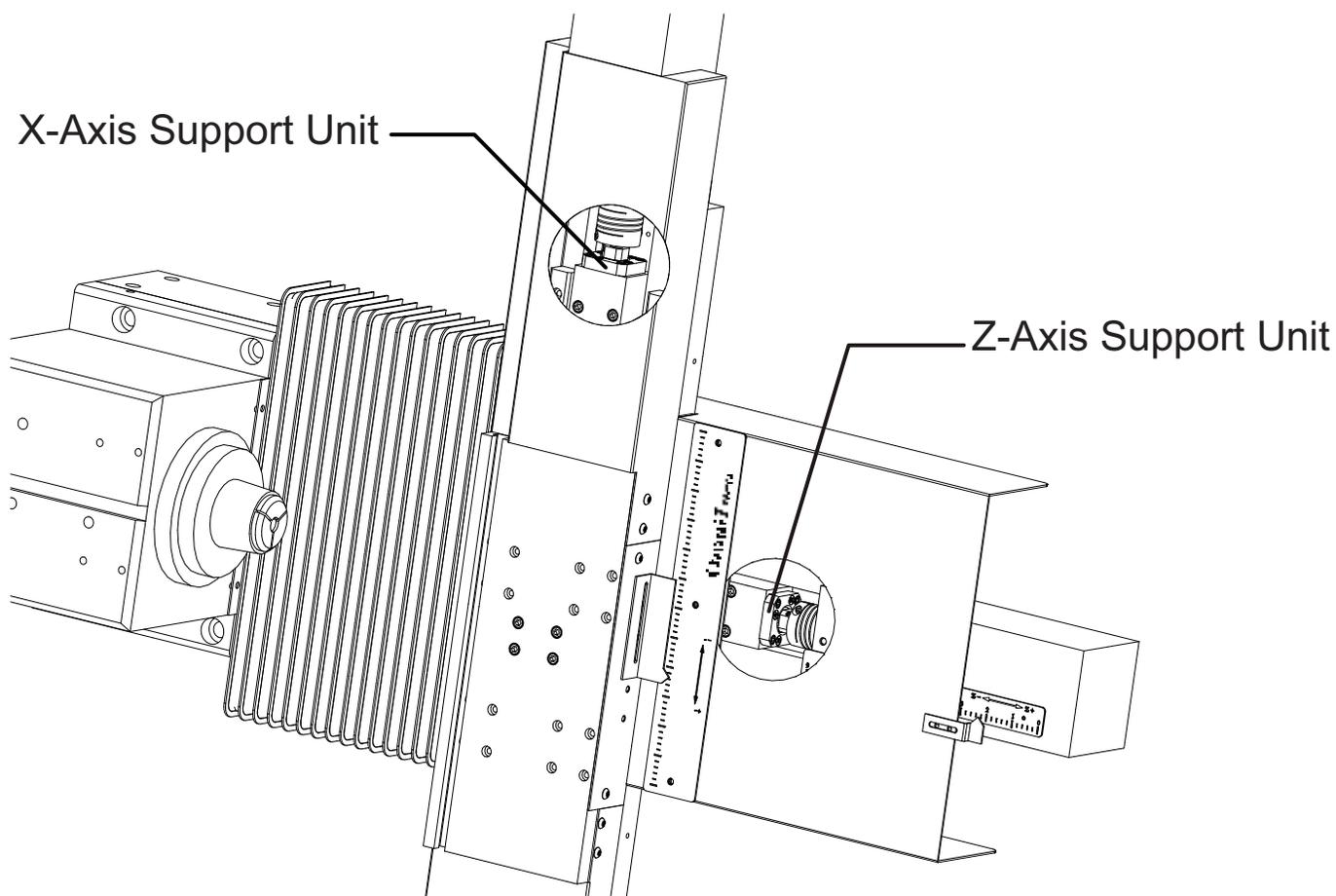
Z-Axis Support Unit

This document describes removal and replacement of the servo motors, flexible couplers and support units (thrust bearings) on the OmniTurn slides.

The illustration above shows the location of the components.

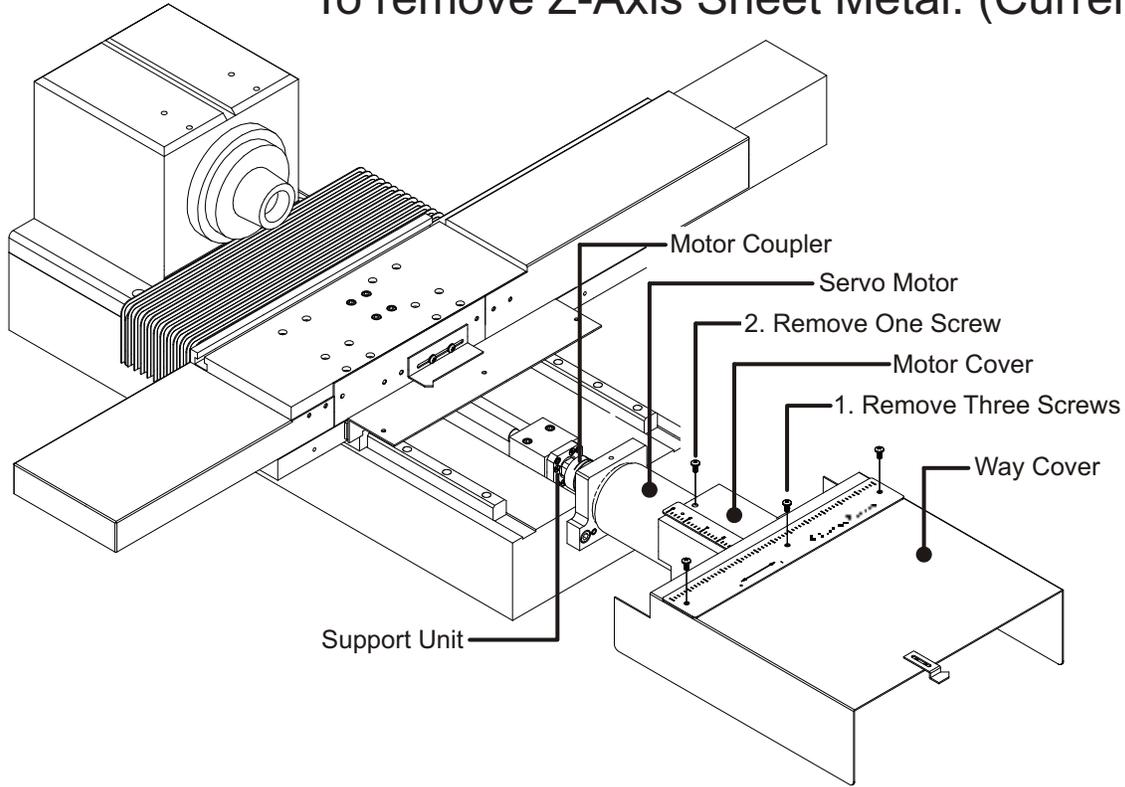Sheet metal removal is illustrated on pages 2 and 3
Coupler removal and replacement: pages 3 and 5
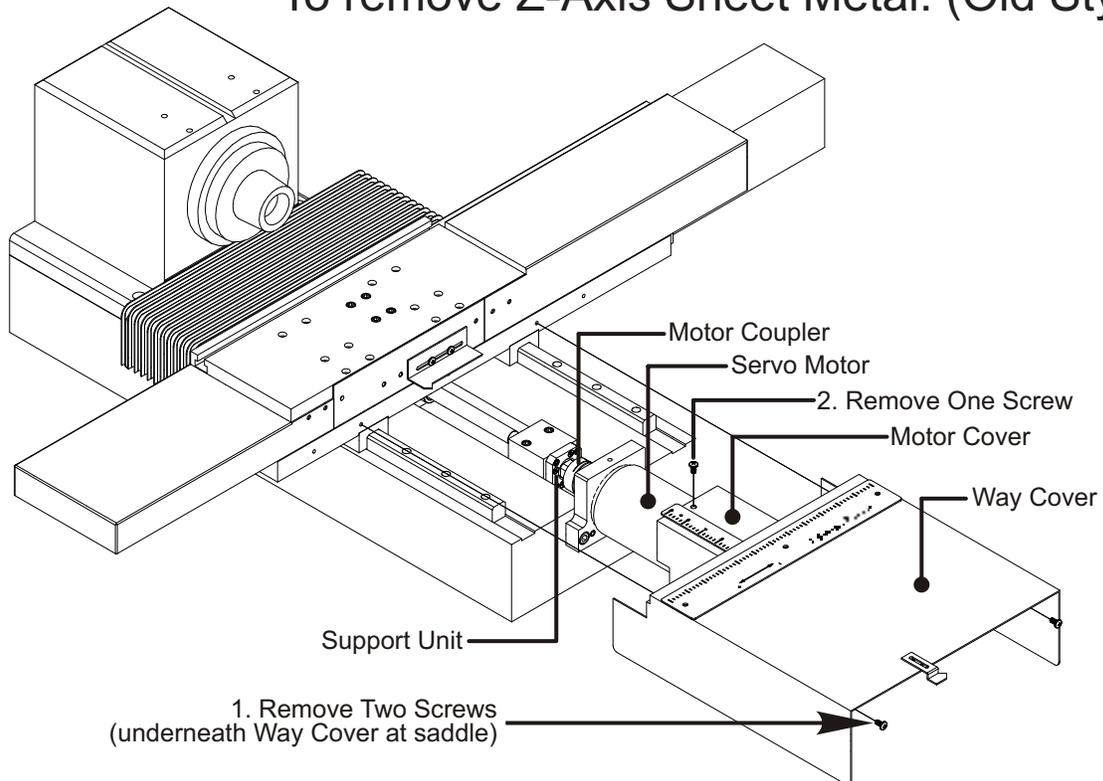Support unit removal and replacement: pages 4 and 5
Servo motor removal and replacement: pages 3 and 5

To remove Z-Axis Sheet Metal: (Current)

Motor Coupler

Servo Motor

2. Remove One Screw

Motor Cover

1. Remove Three Screws

Way Cover

Support Unit

- -

To remove Z-Axis Sheet Metal: (Old Style)

Motor Coupler

Servo Motor

2. Remove One Screw

Motor Cover

Way Cover

Support Unit

1. Remove Two Screws
(underneath Way Cover at saddle)

To remove X-Axis Sheet Metal :

Way Cover

1. Remove two screws

Motor Cover

Servo Motor

Motor Mount

Motor Coupler

2. Remove Slide Brake

For X-Axis, the slide brake assembly must be removed. Two 6-32 cap screws secure the cylinder to the saddle.
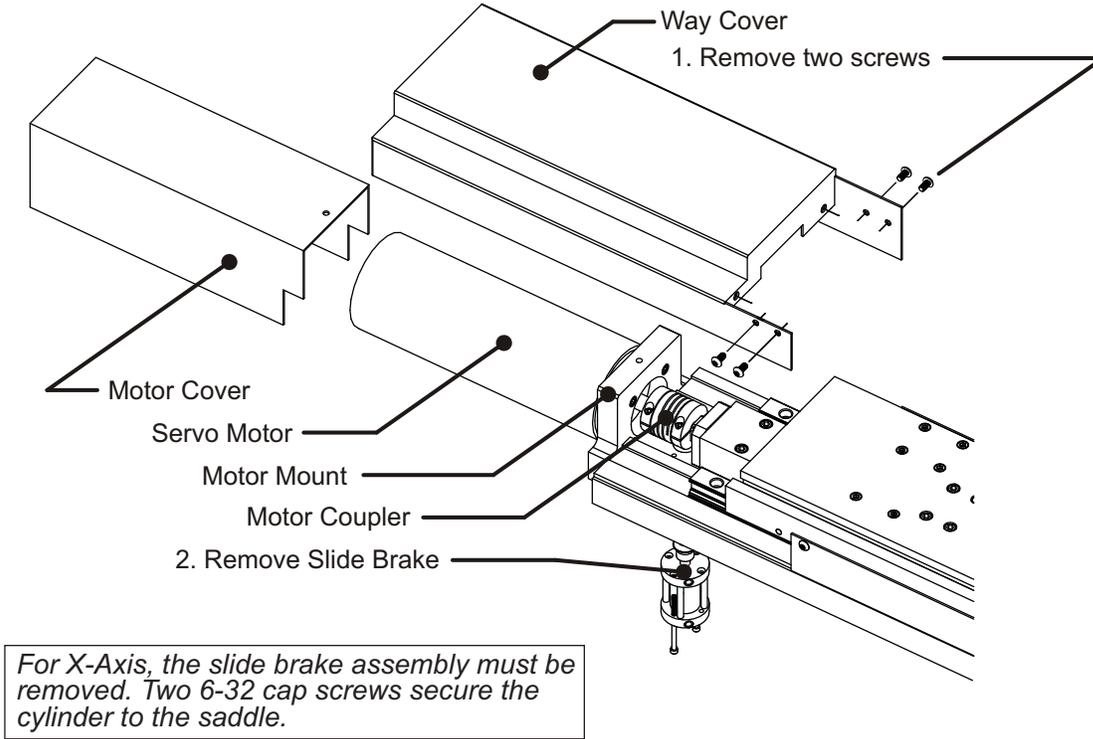
To remove Servo Motor*:*

If only replacing coupler, install new coupler, then skip to Page 6.13 for motor re-installation instructions.

Coupler clamps (Tighten to 40lb-in)

1. Loosen clamp on coupler at ballscrew

2. Remove two 3/8-16 bolts
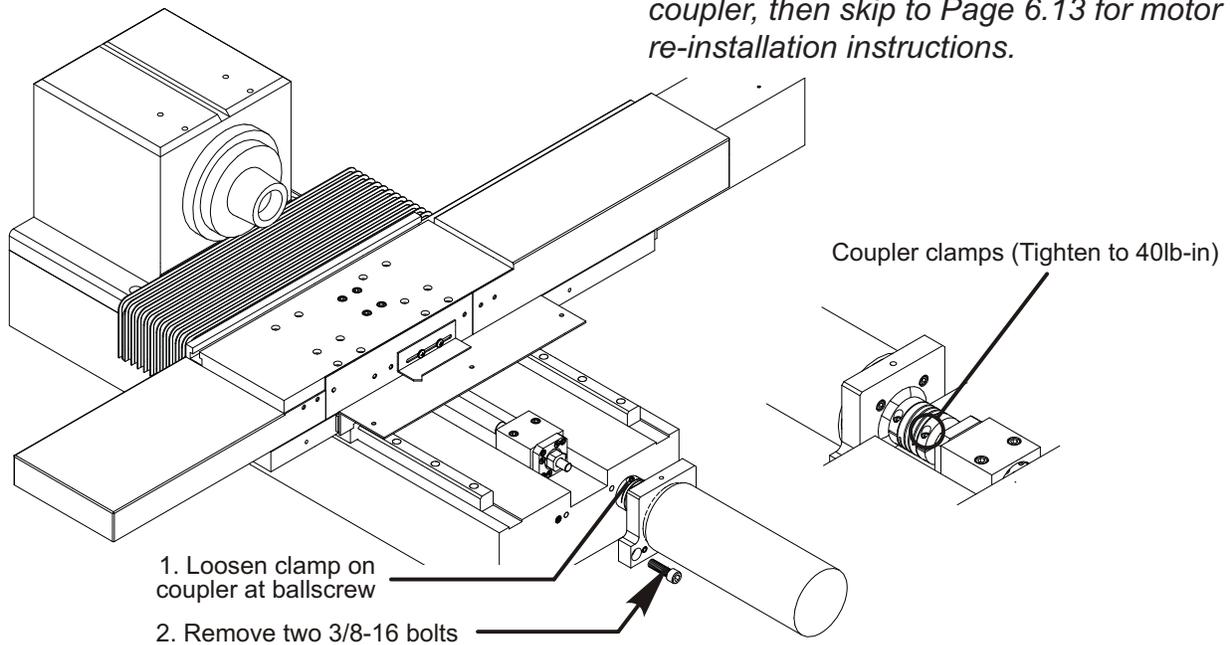
OmniTurn - Trouble shooting guide, G4 CNC

To remove support unit from ballscrew:

19mm flat on ballscrew

Support unit mtg screws (4)
Support unit spacer Support
unit nut & set-screw
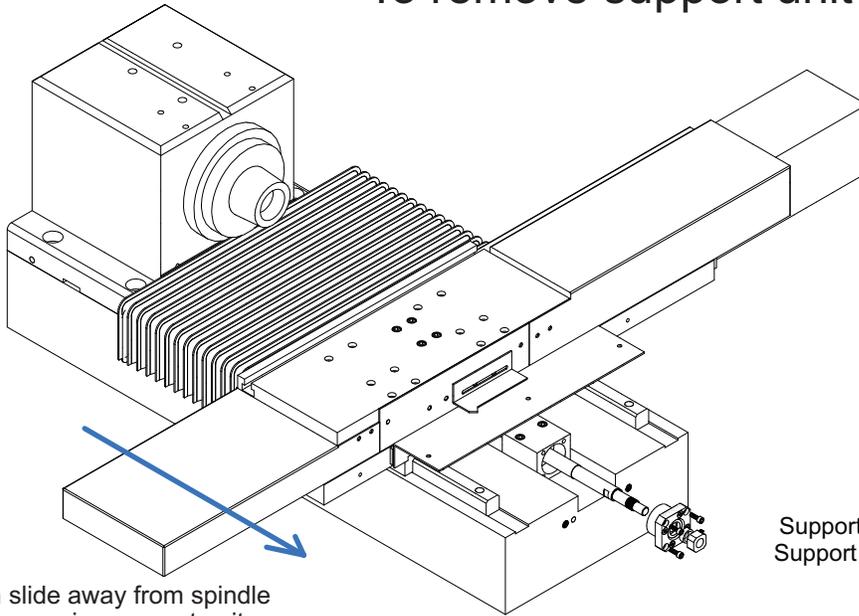
Push slide away from spindle
after removing support unit
mounting screws.
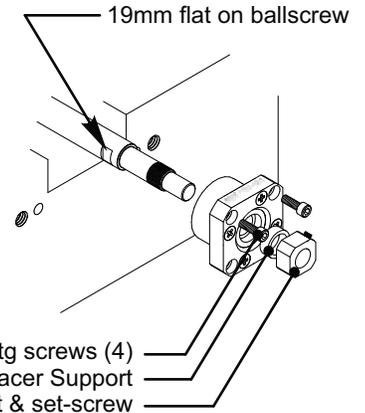
1. Remove four socket screws holding support unit to block.
2. Push slide in z-axis to expose flat on ballscrew.
3. Loosen support unit nut set-screw
4. Loosen support unit nut using1/2"and 19mm wrenches.

- -

To attach new support unit to ballscrew:

Small brass
cushion
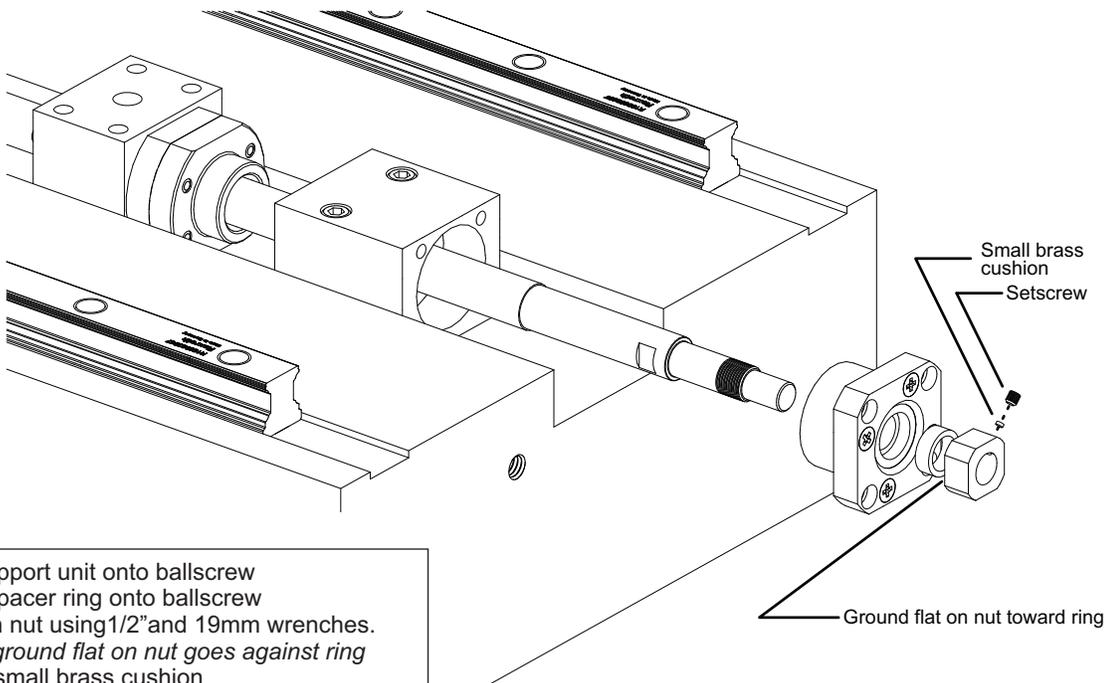
Setscrew

1. Slip support unit onto ballscrew
2. Slide spacer ring onto ballscrew
3. Tighten nut using1/2"and 19mm wrenches.
   NOTE: ground flat on nut goes against ring
4. Install small brass cushion
5. Secure nut with setscrew (not too tight)

Ground flat on nut toward ring

To install New Support Unit:
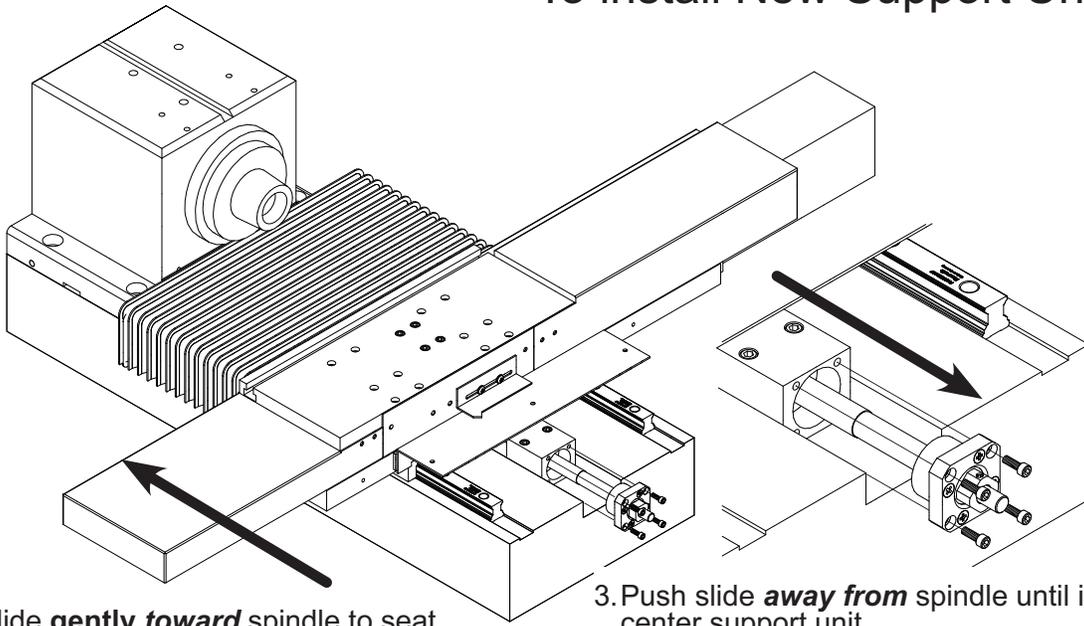
1. Push slide **gently *toward*** spindle to seat support unit in block.
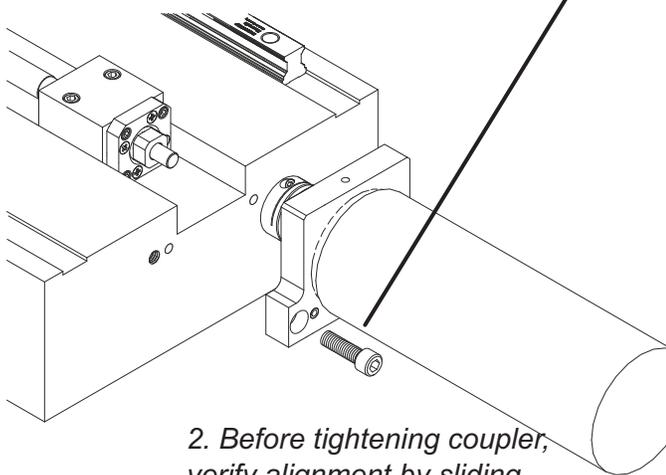2. Install four mounting screws, but don't tighten yet.

3. Push slide ***away from*** spindle until it stops, to center support unit.
4. Tighten four mounting screws to 30lb/in.

Replace Servo Motor

"Home" marks at 180° apart
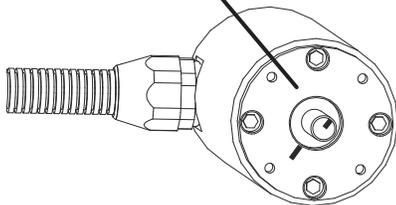
1. Tighten two 3/8-16 bolts (30lb/ft)

2. Before tightening coupler, verify alignment by sliding coupler on both shafts. Tighten coupler clamps to 40lb/in.

1. Re-install Servo Motor:
1a. On *Z-Axis,* pull the slide all the way away from the spindle, against the Belleville stack; on *X-Axis,* push slide all the way down, against Bellevilles.
1b. 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.
1c. Attach the motor to motor mount, then attach mount to the base, slipping the shaft into the coupler. Don't let the motor shaft turn much off its 180° setting.
1d. Tighten the motor mount to the machine (30lb/ft).
1e. Check alignment by sliding coupler back and forth on motor and ballscrew shafts.
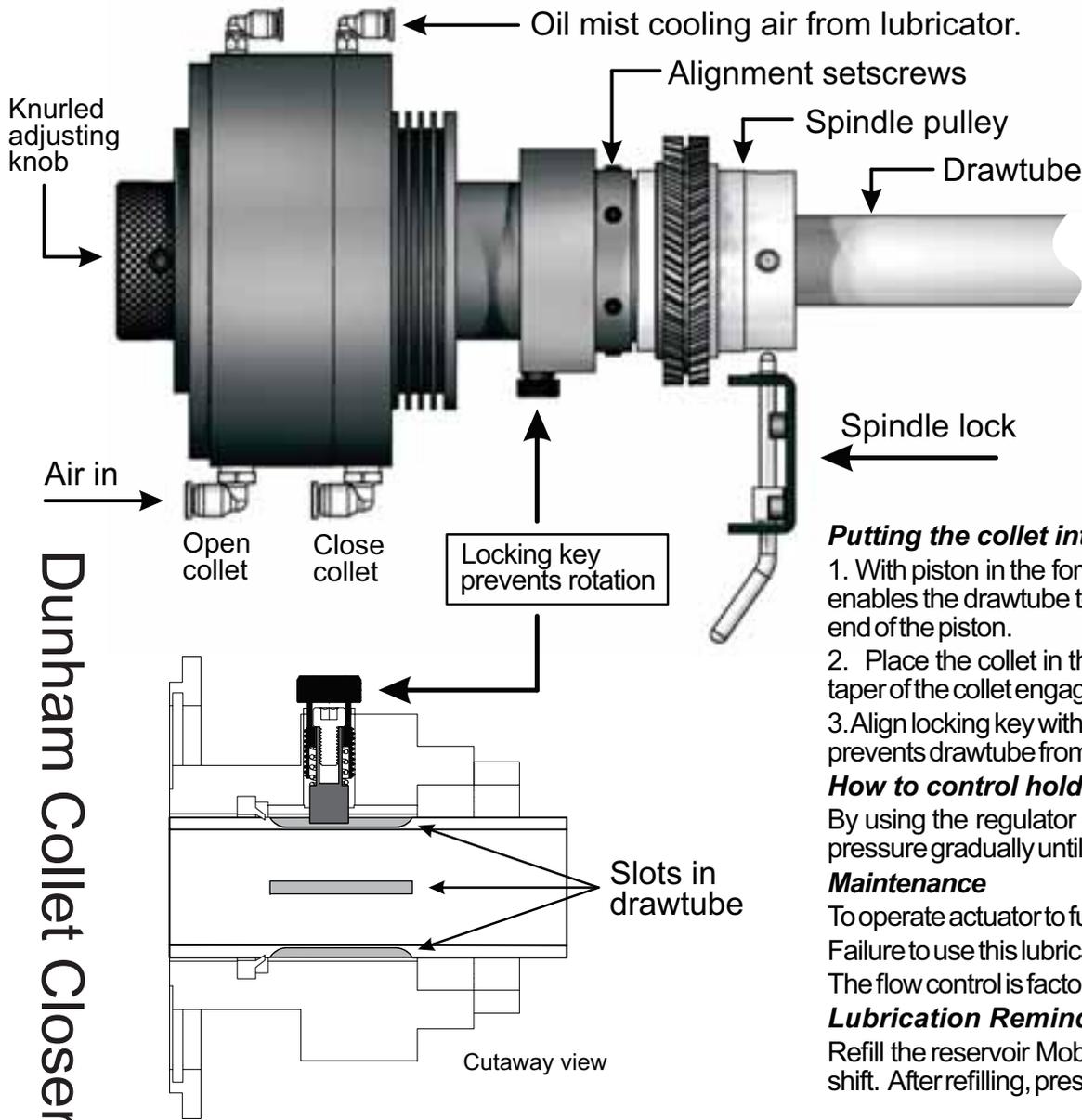**1f.** Tighten the coupler clamps to 40lb/inch.
1g. On X-Axis, replace slide brake.

2. Install all sheet metal
2a. After re-assembly, jog the axis to both ends and verify that the scale pointer will go just slightly past "0"; jog back inside of "0", then establish Home as usual. If the pointer is not at "0", loosen it and move it as required.

3. Re-set all tools before running!

Dunham Collet Closer

OmniTurn - Trouble shooting guide, G4 CNC

Oil mist cooling air from lubricator.

Alignment setscrews

Spindle pulley

Drawtube

Knurled adjusting knob

Air in

Open collet

Close collet

Locking key prevents rotation

Spindle lock

NOTE: *To maximize bearing life, and to minimize operating temperature,* ***always use the least air pressure required to hold part securely.***

Max air pressure:	*90psi*
Max RPM:	*4000 rpm*
Max Pull force:	*1790lbs @ 90psi*
Max Push force:	*1458bs @ 90psi*
Lubrication:	*Mobil "DTE" Light (or Eq)*
Lube Reminder:	*Refill lube; press Alt-L each shift*
Cooling air pressure:	*15psi clean, lubricated*

Slots in drawtube

Cutaway view

Putting the collet into the spindle:

1. With piston in the forward (collet open) position, back off locking key. This enables the drawtube to be turned by the knurled collet adjusting knob at the end of the piston.

2. Place the collet in the spindle. Turn the knurled knob clockwise until the taper of the collet engages with taper of the spindle.

3. Align locking key with adjusting slot on draw tube. Engage locking key. This prevents drawtube from turning and becoming loose during use.

How to control holding force:

By using the regulator and gauge, start with air pressure at 30 lbs. Increase pressure gradually until part is held sufficiently for machining operation.

Maintenance

To operate actuator to full potential, oil-mist lubrication is required.

Failure to use this lubrication may result in unit failure.

The flow control is factory set and locked at 15psi back-pressure.

Lubrication Reminder

Refill the reservoir Mobil "DTE" light (or equivalent) at the beginning of each shift. After refilling, press "Alt-L" on keyboard to reset Lubrication Reminder.

Collet Usage and Collet Closer Maintenance

(Refer to illustration on following page)

Inserting Collet:

1) With piston (item 2) in forward or released position, back off locking key (item 17) , enabling draw-tube (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 Ibs.; 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 serated collet; serrations increase gripping force by10% 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) 15psi cooling air with oil mist must be applied when using actuator.

2) Oil (Mobile "DTE light" or equivalent) must be available for the oil-mist system.

NOTE:
A lubrication reminder "*Check and fill collet closer lube!*" will appear every ten hours. Cycle-start is inhibited while reminder is active. Refill reservoir and press Alt-L *(Press and hold 'Alt' key, then press 'L' key)* at beginning of each shift to avoid interruption.

3) Verify lubricator provides four drops of oil per minute.

THE DUNHAM
TOOL COMPANY
Div. of Preferred Technologies, Inc.

8 Parklawn Drive· Bethel· CT· 06801 • USA· 203-739-0415· FAX: 203-739-0461
www.dunhamtool.com preferred@dunhamtool.com

POSITION INDICATOR HERE

0.31 MAX STROKE

6.19 DIA

OmniTurn - Trouble shooting guide, G4 CNC
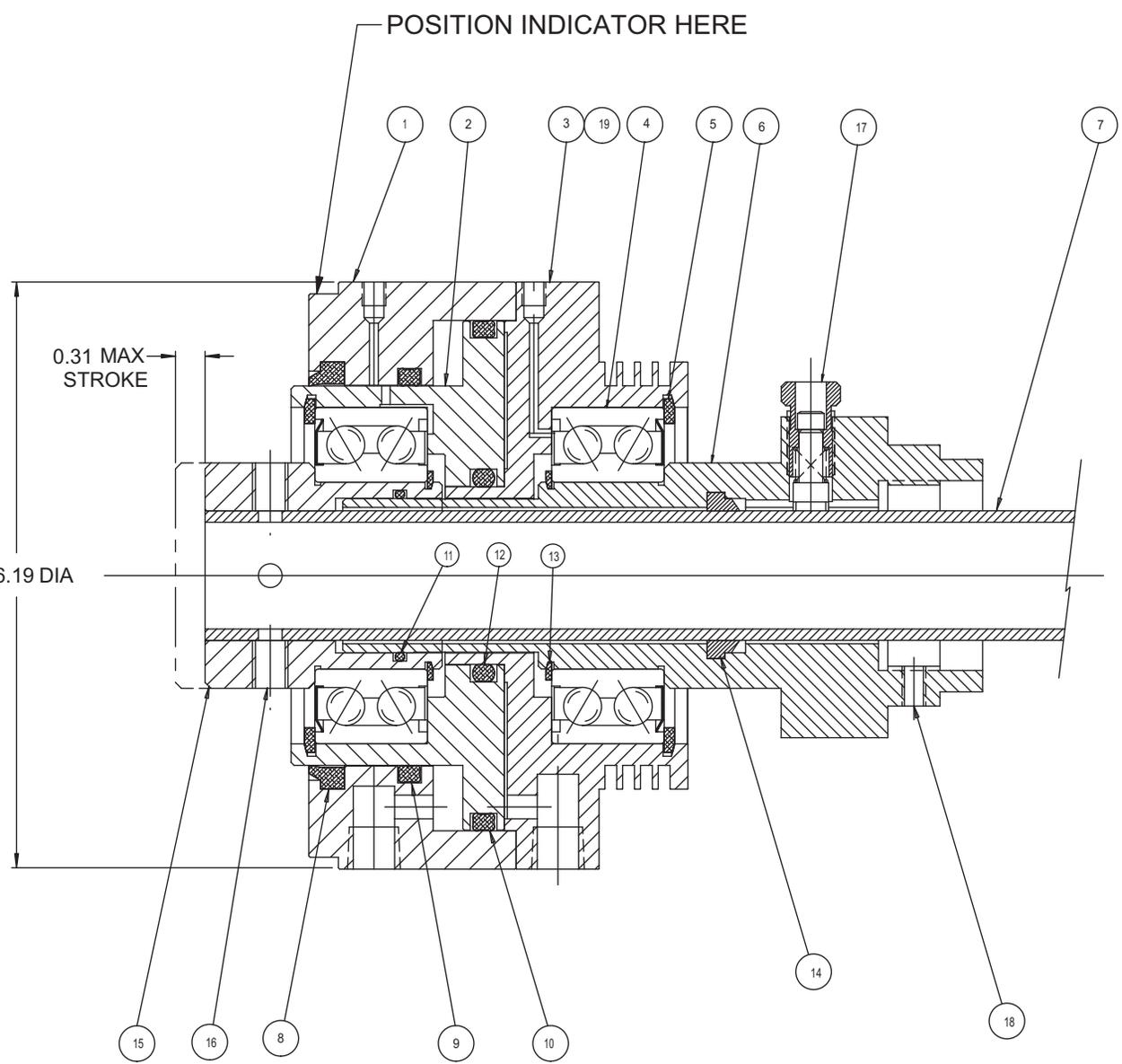
ITEM	PART NO.	DESCRIPTION	QTY.
19	#10-24 X 1" LG.	SOCKET HEAD CAP SCREW	8
18	1/4-20 X 3/8 LG	BRASS PO NT SOC SET SCREW	3/ 6
17	MA299-H	LOCKING KEY ASSEMBLY	1
16	3/8-16 X 5/8 LG	HALF DOG POINT SET SCR	4
15	MC544	KNURLED ADJUSTING KNOB	1
14	00602100A	DISOGR N SEAL	1
13	5102-196	TRU-ARC SNAP R NG	2
12	D-328	"O" - RING	1
11	D-130	"O" - RING	1
10	Q4353	QUAD RING	1
9	Q4345	QUAD RING	1
8	006-044	DISOGR N SEAL	1
7	MB244-H6	DRAWTUBE	1
6	MC543-A / MC543-F	MAIN BRG. MOUNT	1
5	5002-354	TRU-ARC SNAP RING	2
4	D-55510ND	DACC BEAR NG (ONE SHIELD)	2
3	MC542	MAIN BRG. HOUSING	1
2	MC541	PISTON	1
1	MC540	CYLINDER	1

UNLESS OTHERWISE SPECIFIED
ANGLES MAY VARY
DECIMAL DIM. MAY VARY
.xx ±.015   .xxx ±.005
FRACTIONAL DIM. MAY VARY   ±1/64
ROUND EDGES   .005-.015
CORNERS TO HAVE FILLETS .005-.020
SURFACE FINISH

THE DUNHAM TOOL COMPANY
BETHEL, CONNECTICUT
AIR COLLET ACTUATOR - OIL MIST
NC ELECTRONICS

DWN. MG	DATE 11/97	MATL	FIN
CHK.	DATE	NEXT ASSM.	
SCALE 1:1	DWG NO. MC586		REV A

Collet Closer Specifications and Maintenance

Maximum pull force: 1791lbs. @ 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 : 4000 (continuous)
6000 (intermittent)
Largest diameter : 6.18"
Length : 7.06" open (without spindle adapter)
6.7S" closed
Weight : 14 lbs 11 oz. (without spindle adapter)
Through hole : 1.06"

Compressed air requirements:
Actuation: Variable 15 to 90 psi; clean, dry air required.
USE MINIMUM PRESSURE NECESSARY FOR HOLDING PART
Cooling :15 psi, clean, lubricated (oil mist)

Adjusting Actuator Concentricity:

Adjust six brass point set screws (item 18) to general tightness. Reduce air pressure supplied to forward port of actuator to 15 psi. Using a dial indicator reach to indicator point identified on illustration, previous page. Rotate spindle slowly to measure the rise and fall of this portion of the actuator. Adjust concentricity with the use of the six set screws (item 18) lightly. If actuator is in good alignment with the spindle, the indicator should measure .0005"-.0010" concentricity. Tighten six set screws to spindle securely.
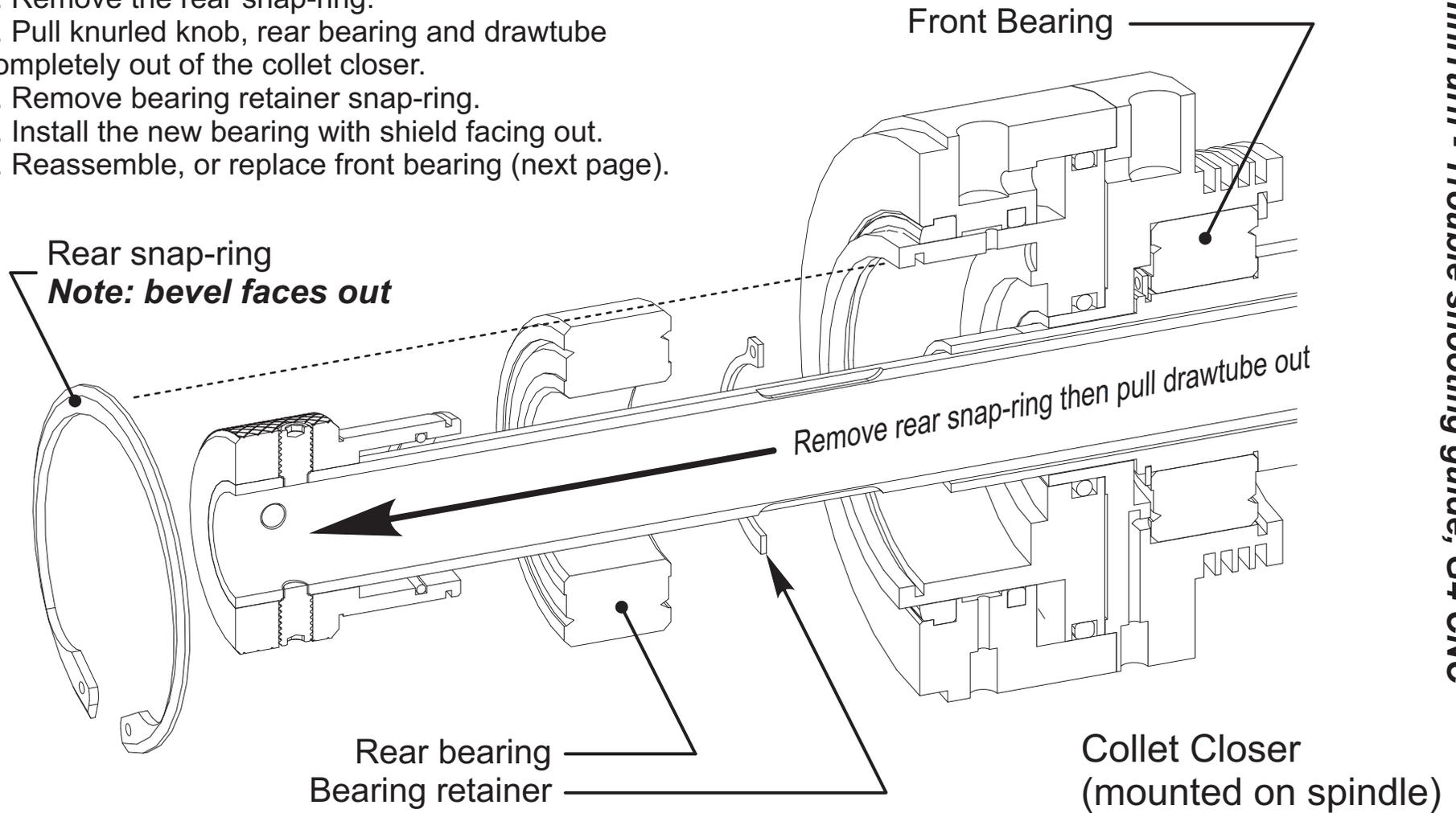
Oil-Mist Lubrication:

The air regulator should be adjusted to 15 psi. The Parker 06L12BE Lubricator should be adjusted to supply 1 drop every 15 seconds. The reservoir should be replenished with Mobil "DTE" light (or equivalent) as required.

FAILURE TO USE THIS LUBRICATION MAY RESULT IN UNIT FAILURE

To replace the bearings:
*(Do not remove collet closer from spindle)*

Rear Bearing

1. Remove the collet.
2. Remove the rear snap-ring.
3. Pull knurled knob, rear bearing and drawtube completely out of the collet closer.
4. Remove bearing retainer snap-ring.
5. Install the new bearing with shield facing out.
6. Reassemble, or replace front bearing (next page).

Rear snap-ring
Note: bevel faces out

Front Bearing

Remove rear snap-ring then pull drawtube out

Rear bearing
Bearing retainer

Collet Closer
(mounted on spindle)

OmniTurn - Trouble shooting guide, G4 CNC

Front bearing:

7. Remove anti-rotation bracket
8. Remove four air hoses (push-lock fittings)
9. Remove front snap-ring.
10. Slip collet closer body off main bearing mount.
11. Remove bearing retainer snap-ring.
12. Install the new bearing with shield facing out.
13. Reassemble and verify concentricity.

Main bearing mount
(mounted on spindle)

Remove collet closer body

Front snap-ring
Note: bevel faces out

Bearing retainer
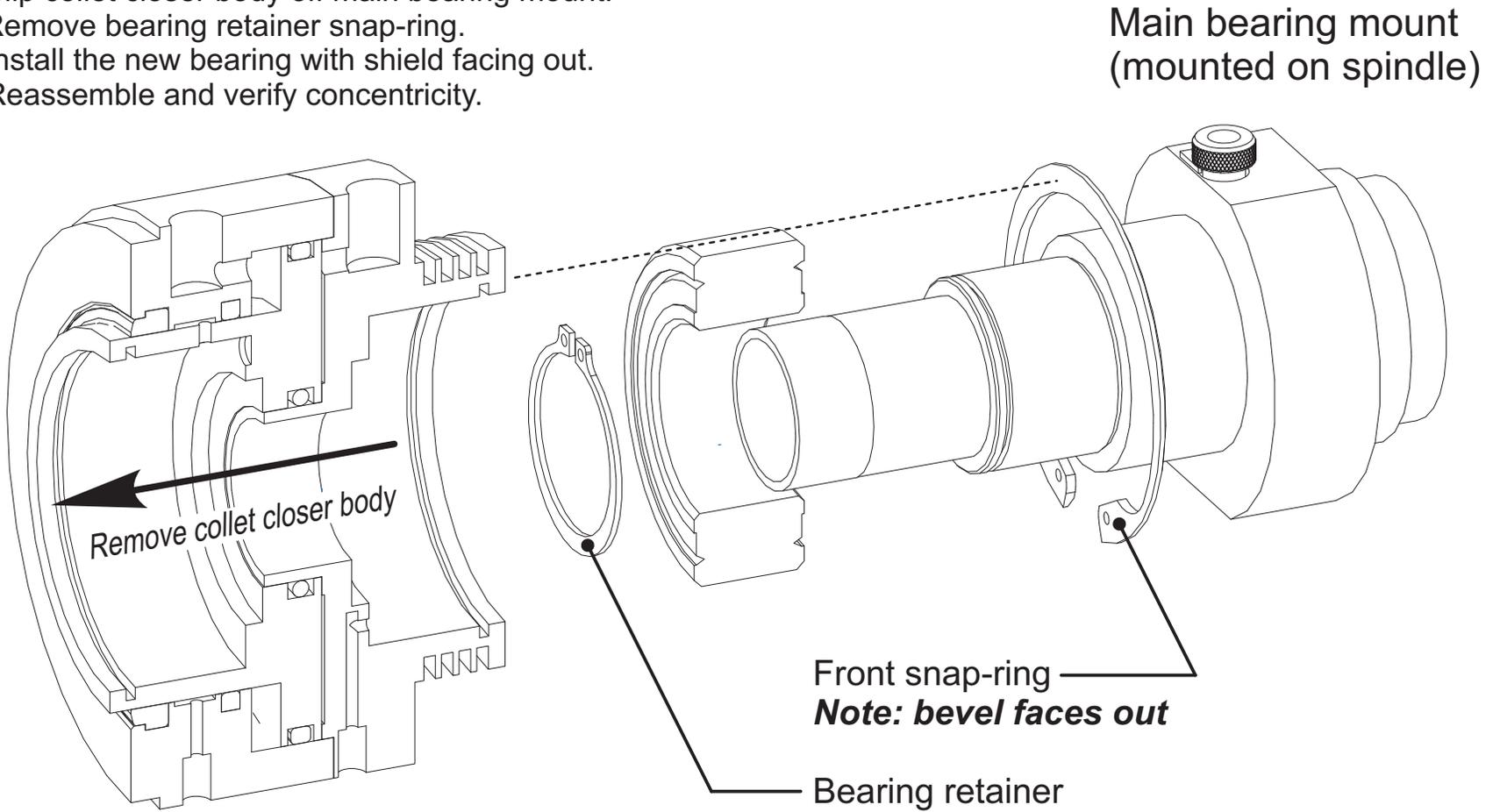
MAINTENANCE & LUBRICATION SCHEDULE

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. Use Mobile Light DTE oil or equivalent. Refill reservoir, then press Alt-L on keyboard to reset lubrication reminder. *(Reminder will appear every ten hours; cycle-start is inhibited when reminder is present.)* With spindle running observe the sight glass for oil flow. If necessary, adjust oil flow to one drop per 15 seconds.
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)

MONTHLY
1. Examine main regulator element; replace if it shows serious contamination.

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 attached to the tooling plate (table) with four 8-32 phillips-head screws. See page 6.35 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 can be lubricated with a drizzle of Slick-50, E900, white lithium grease, or any suitable ball screw lubricant over the entire exposed length.

This page intentionally blank