Safety Warnings

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

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

DO NOT OPERATE WITH DOOR or SPLASH SHIELD OPEN!

DO NOT OPEN DOOR DURING OPERATION!

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

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

DO NOT wear loose clothing while operating machine.

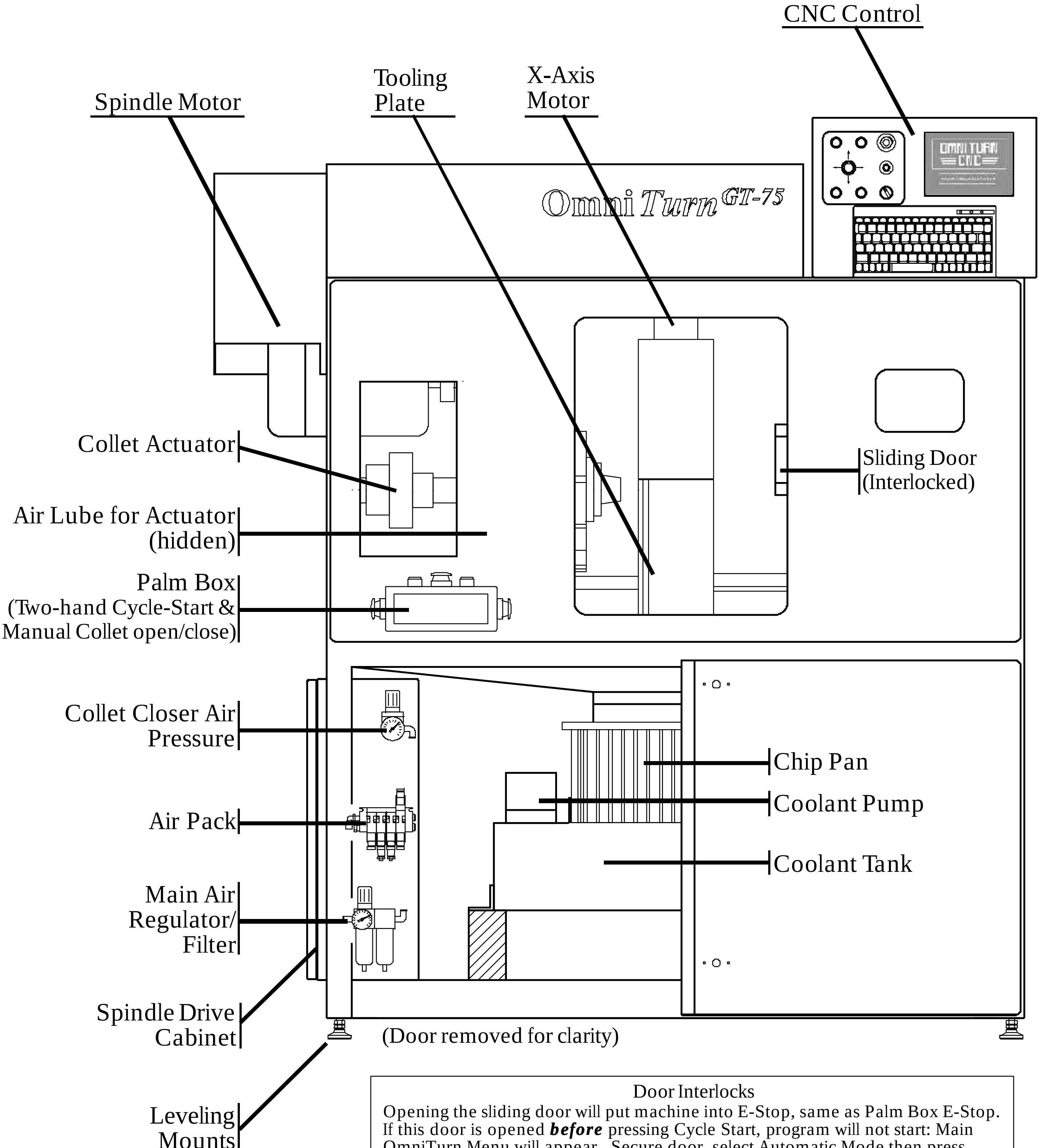
DO NOT wear jewelry (necklaces, bracelets, watches, rings) while operating machine.

Keep long hair tied up and out of the way.

Wear eye and face protection.

Report any malfunction to your supervisor.

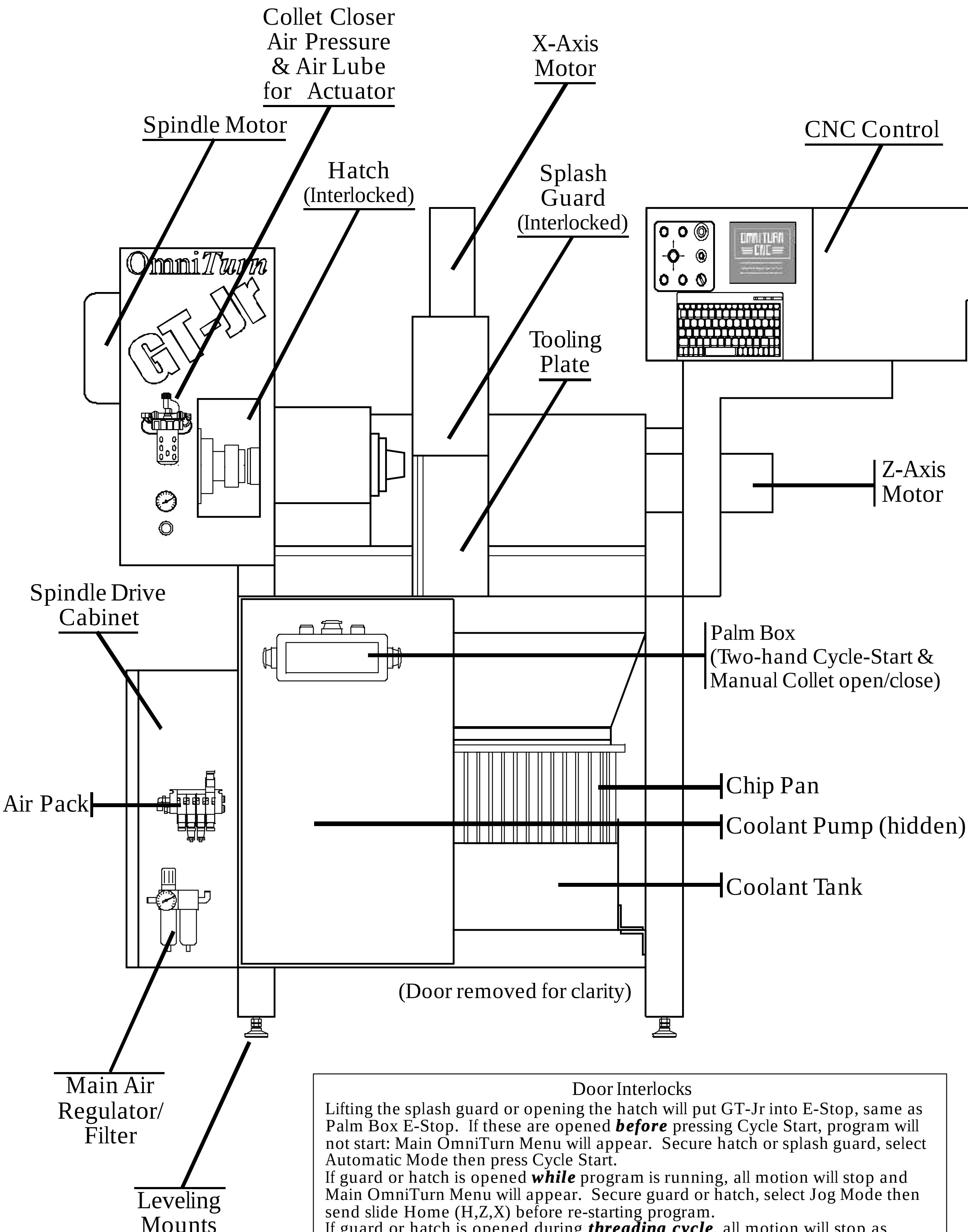
OmniTurn GT-75 Major Components



Door Interlocks

Opening the sliding door will put machine into E-Stop, same as Palm Box E-Stop. If this door is opened ***before*** pressing Cycle Start, program will not start: Main OmniTurn Menu will appear. Secure door, select Automatic Mode then press Cycle Start.
If door is opened ***while*** program is running, all motion will stop and Main OmniTurn Menu will appear. Secure door, select Jog Mode then send slide Home (H,Z,X) before re-starting program.
If door is opened during ***threading cycle***, all motion will stop as above, but CNC Control must be shut off, then on to re-boot.

OmniTurn GT-Jr Major Components



Door Interlocks

Lifting the splash guard or opening the hatch will put GT-Jr into E-Stop, same as Palm Box E-Stop. If these are opened ***before*** pressing Cycle Start, program will not start: Main OmniTurn Menu will appear. Secure hatch or splash guard, select Automatic Mode then press Cycle Start.

If guard or hatch is opened ***while*** program is running, all motion will stop and Main OmniTurn Menu will appear. Secure guard or hatch, select Jog Mode then send slide Home (H,Z,X) before re-starting program.

If guard or hatch is opened during ***threading cycle***, all motion will stop as above, but CNC Control must be shut off, then on to re-boot.

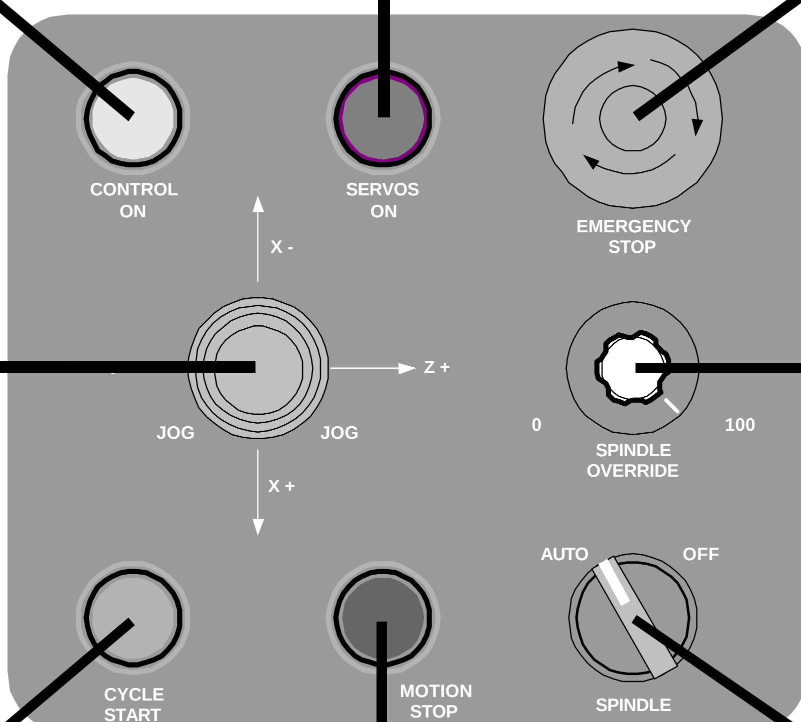
OmniTurn Front Panel: Knobs & Switches

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

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

Halts all machine movement and 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. (Not effective for C-Axis machines). Spindle will not run when this is full CCW.

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

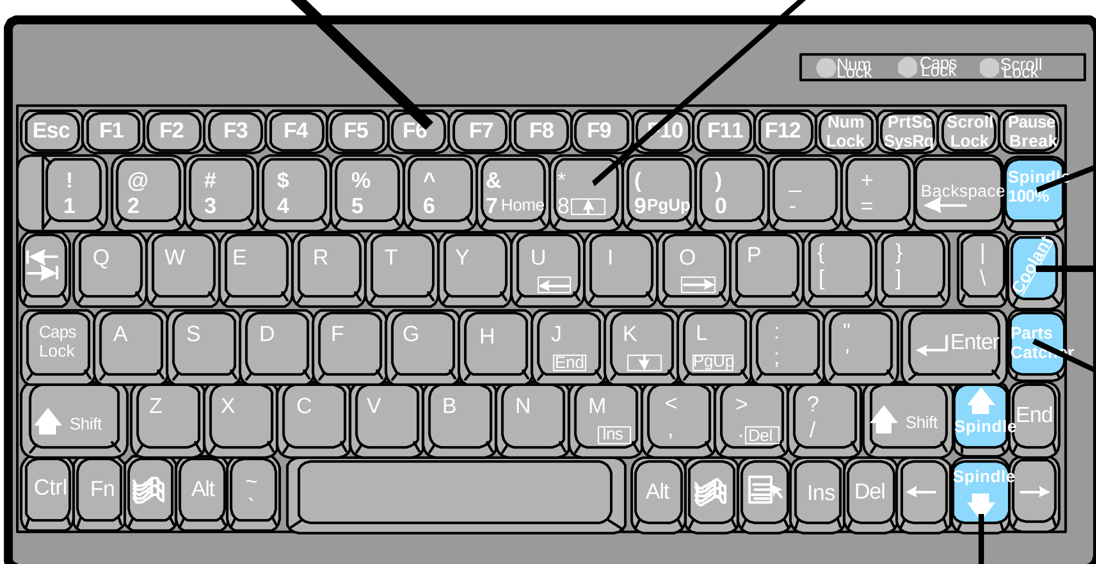
Stops slide movement during program. Illuminates when active. Press Cycle Start to continue or Esc Key to stop program.

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

OmniTurn Front Panel: Keyboard

Function Keys set Feed Rate Override while running program. Special functions are described on page 5.3.

Number Keys set Jog speed and Jog increment in Jog Mode. Functions are described starting on page 1.9.



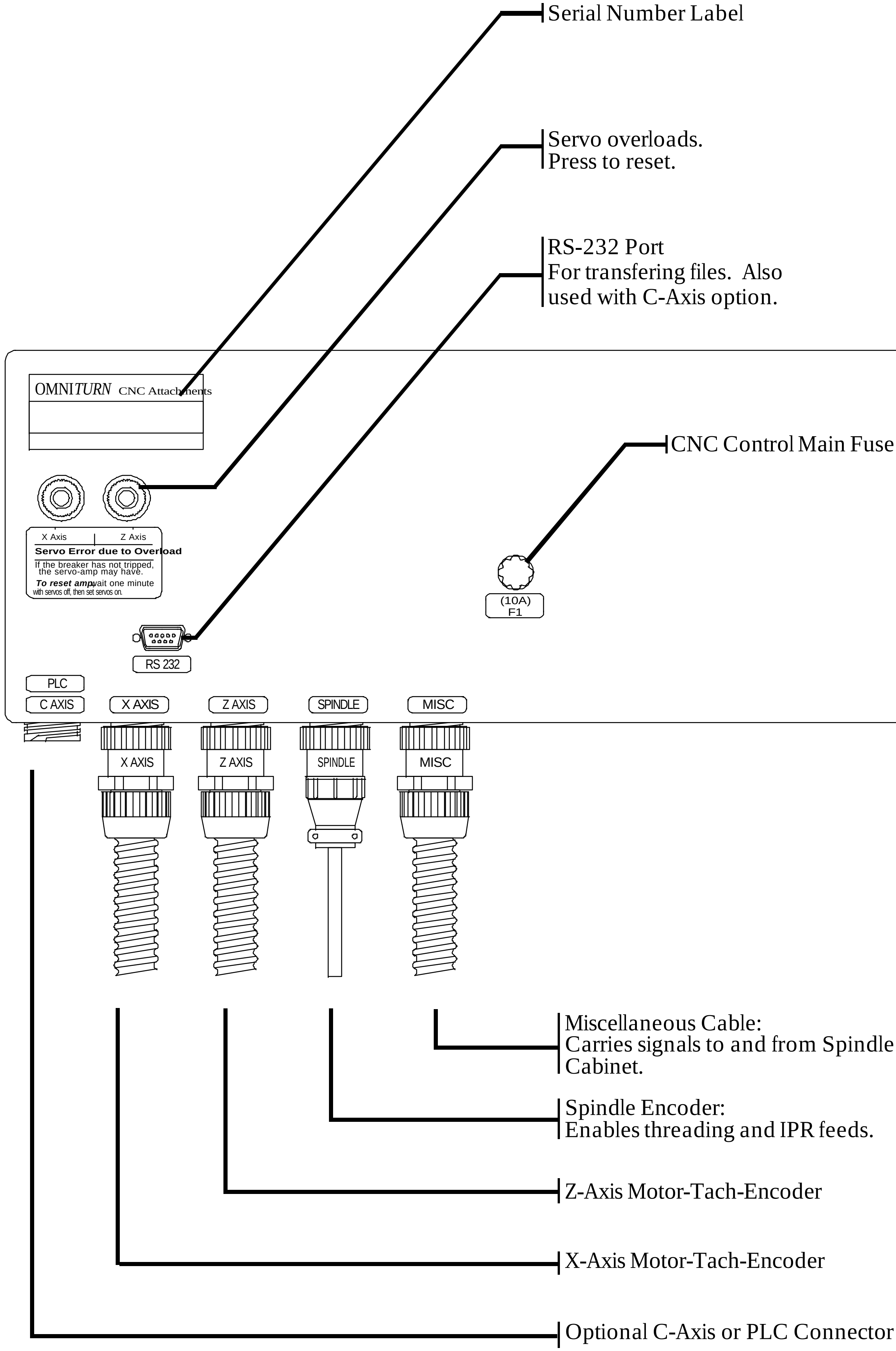
Set spindle speed to 100%. (PgUp)

Set coolant ON/OFF (PgDn)

Set M25/M26 or Parts Catcher out/in

Increase or Decrease spindle speed. Effective for C-Axis and standard spindles.

OmniTurn Rear Panel

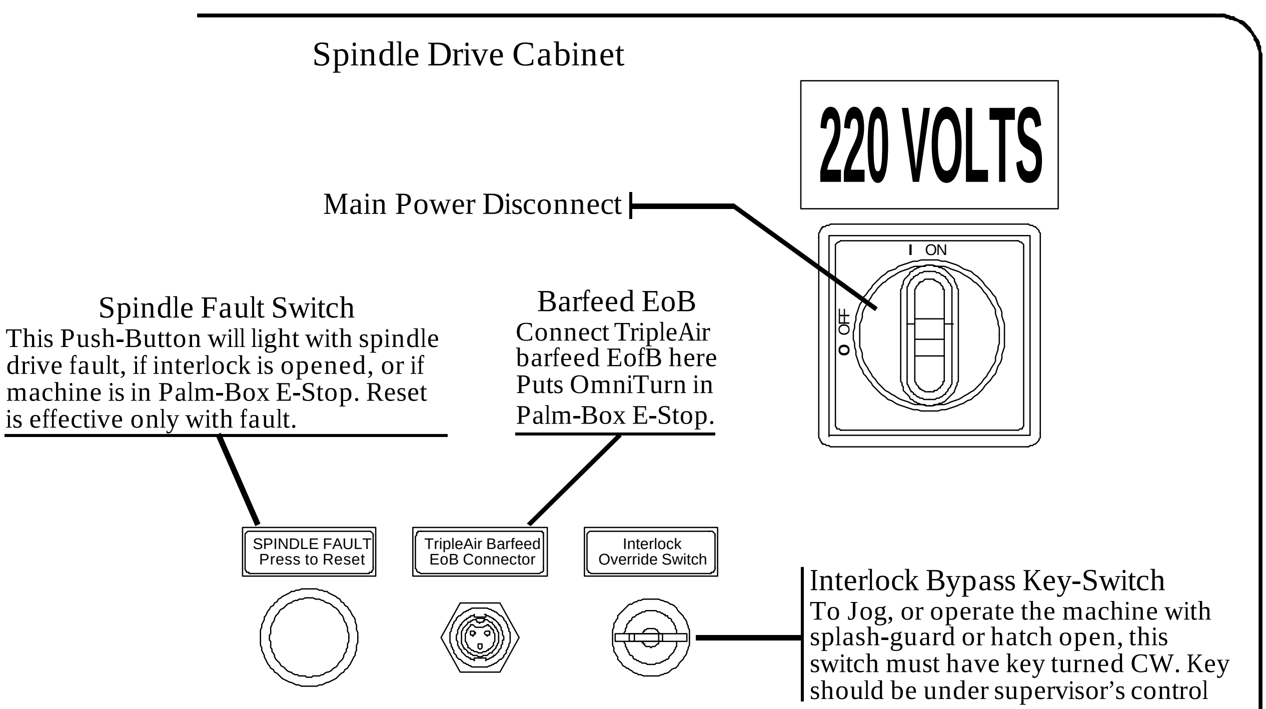


Start - Up: Apply Power

After the OmniTum has been installed and all of the cables are connected set 220vac disconnect ON. The 220vac disconnect is located on the Spindle Drive Cabinet at left-hand side of the GT-75 or GT-Jr, as illustrated below.

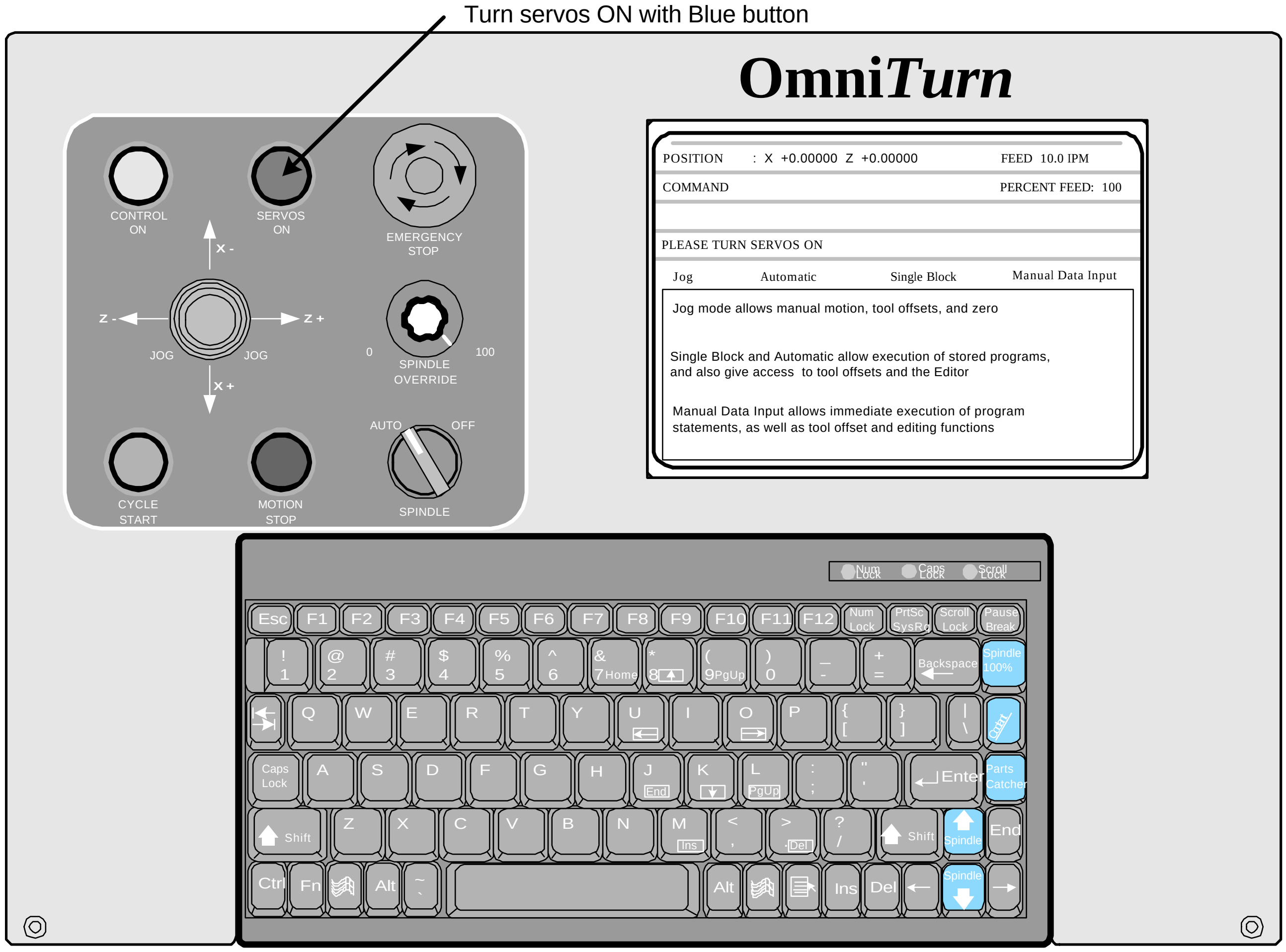
OmniTurn front panel Keyboard

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 floppy disk at rear of machine



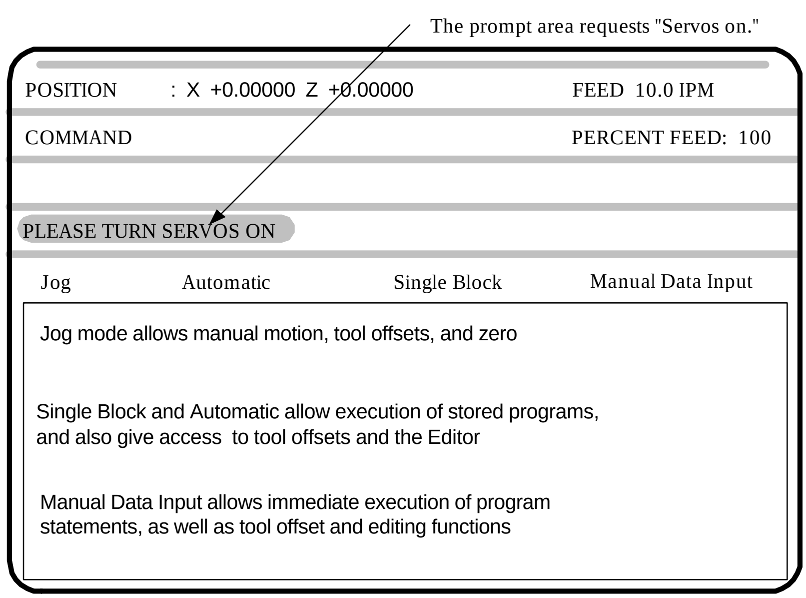
Start - Up: Turn on Servos

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

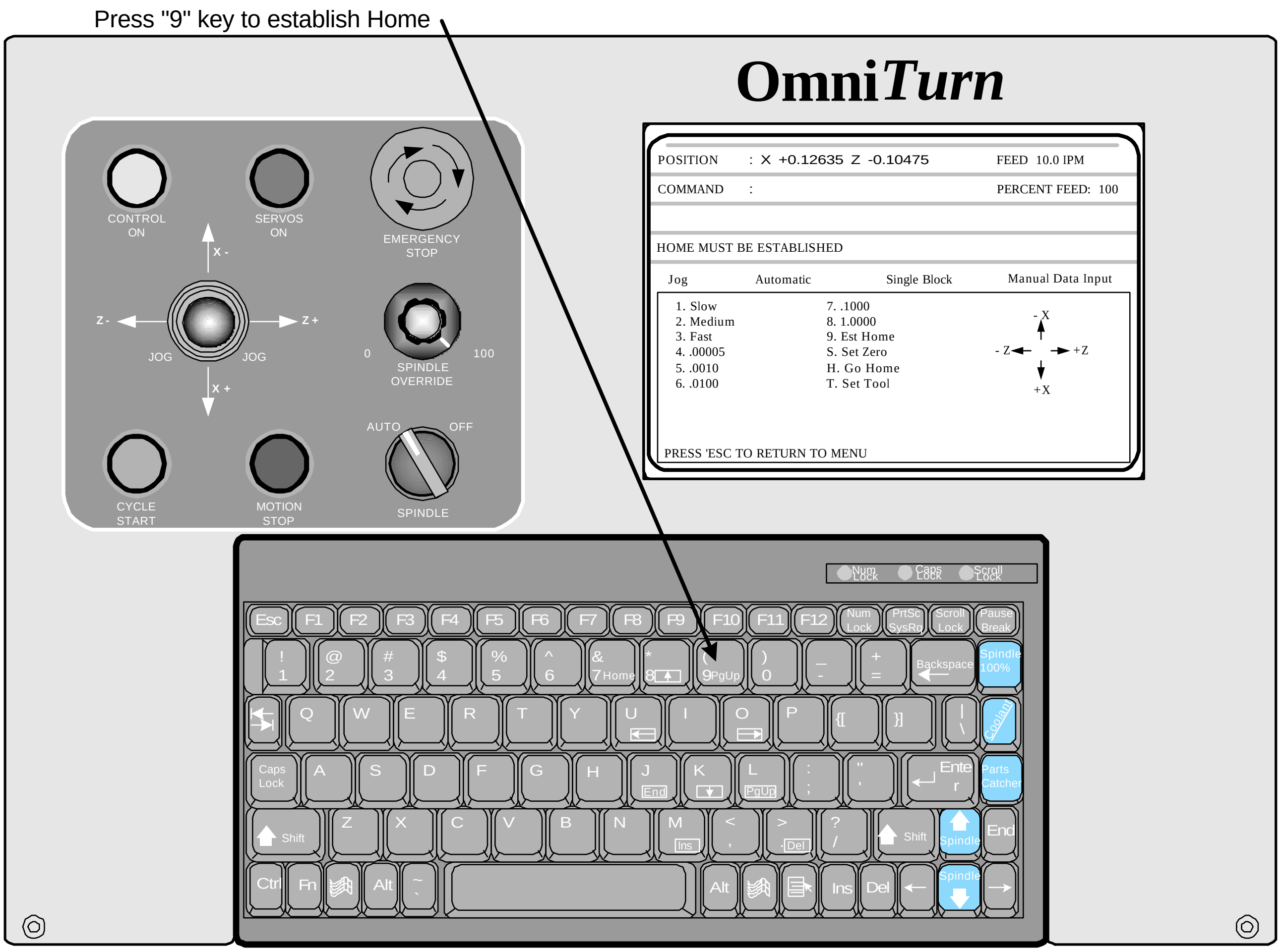


OmniTurn front panel Keyboard

Press the blue “Servos On” pushbutton. It should illuminate and you should hear a slight hum as the axes motors engage. This part of the screen is the prompt area, and different information will be presented here according to the current mode or page.



Start - Up: Establish Home



OmniTurn front panel Keyboard

Establish "Home"

After the servos are turned on, the control automatically goes to the JOG mode. This is done so that machine "HOME" can be established. The control will not allow you to leave the JOG mode until the homing procedure has been completed.

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 as illustrated on the next page. Press "9" 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.

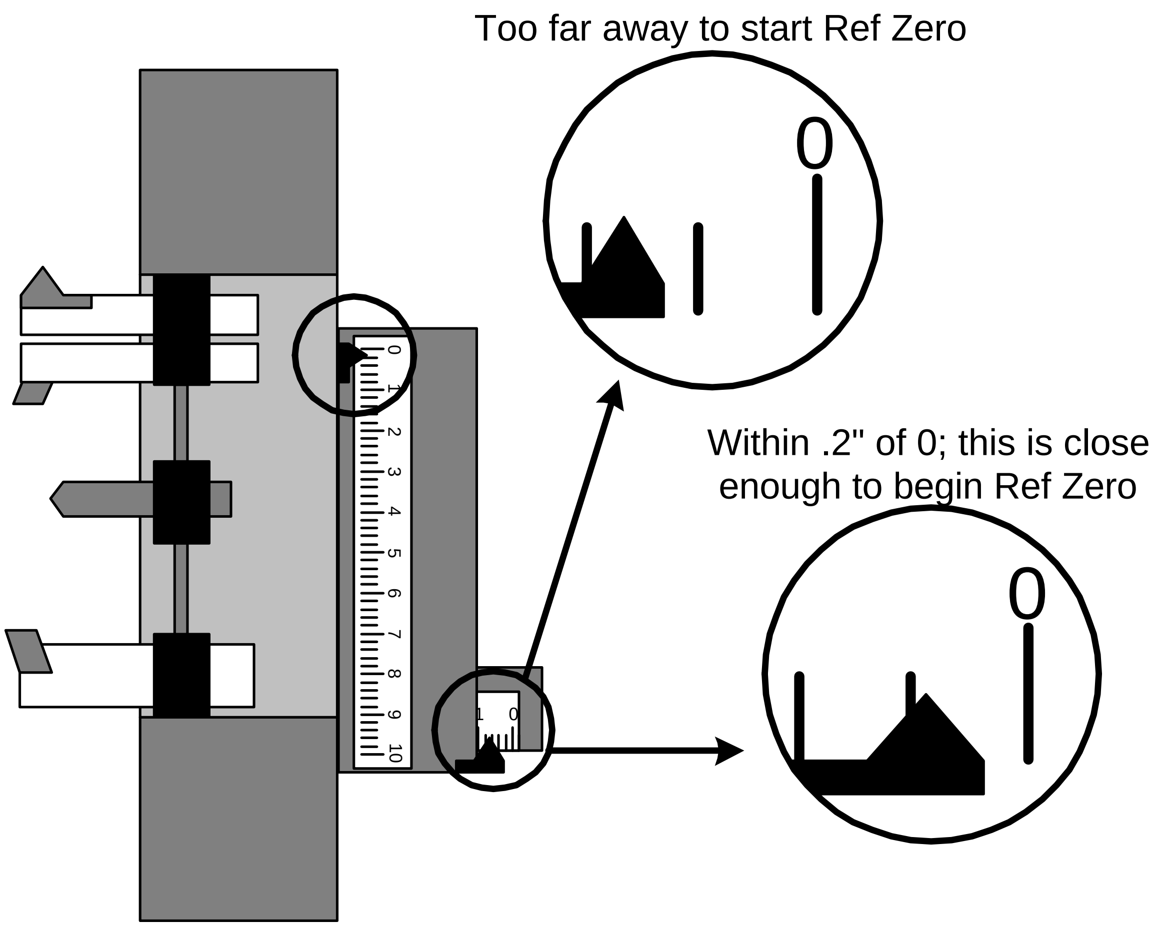
Power Down Procedure

It is a good practice, but not necessary, to press the E-Stop before turning the control off. To turn the control off, press the amber Control ON button. Then set the spindle drive disconnect to OFF.

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 as illustrated below. The X axis should be on the plus side of zero, and the Z axis should be on the minus side.

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



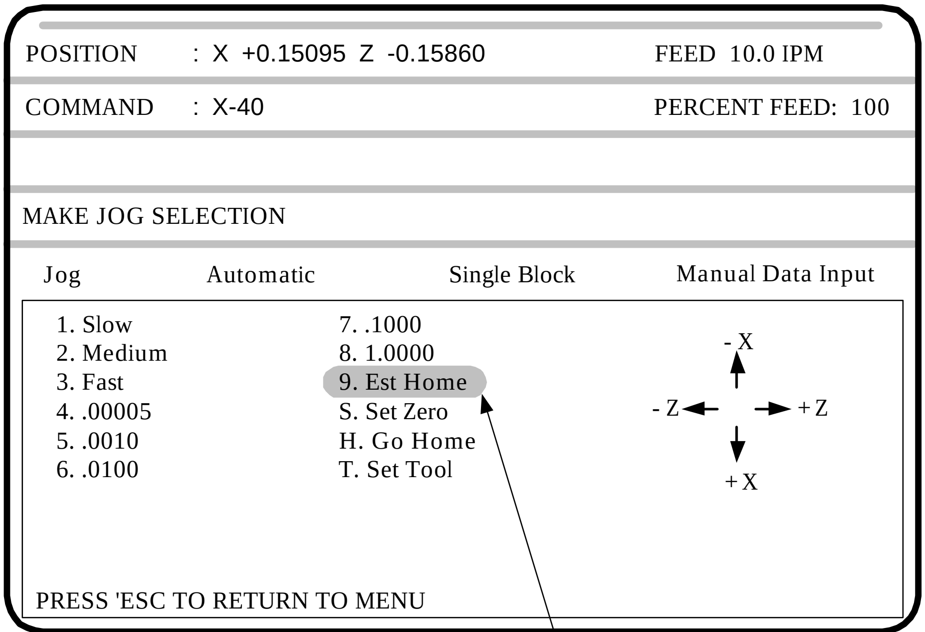
When the slide has been jogged to within 0.200” of zero for ***both*** axes, press the ”9” key on the keyboard, then press Cycle Start pushbutton to complete the homing sequence.

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

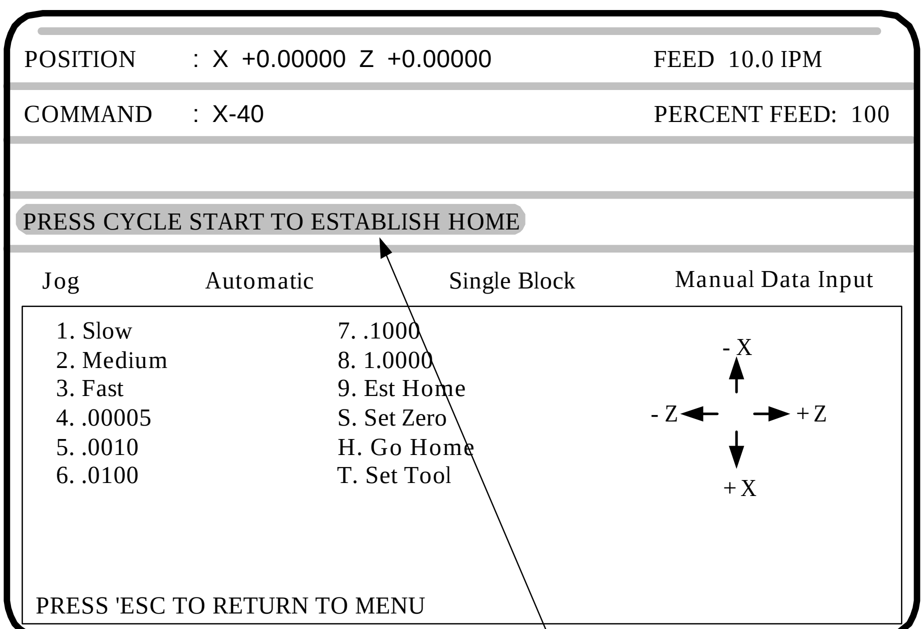
After the procedure, check that the location for zero is where you want it. If you have missed the zero, press the “2” key (Med Jog) then jog the slide to the correct location, then press “9” then Cycle Start again.
Verify that the position display at the top of the screen now reads zero for both axes.

See following page for the screens which display the process.

Establish Home: Screens



Press 9 when BOTH axes are within 0.200" of Home.

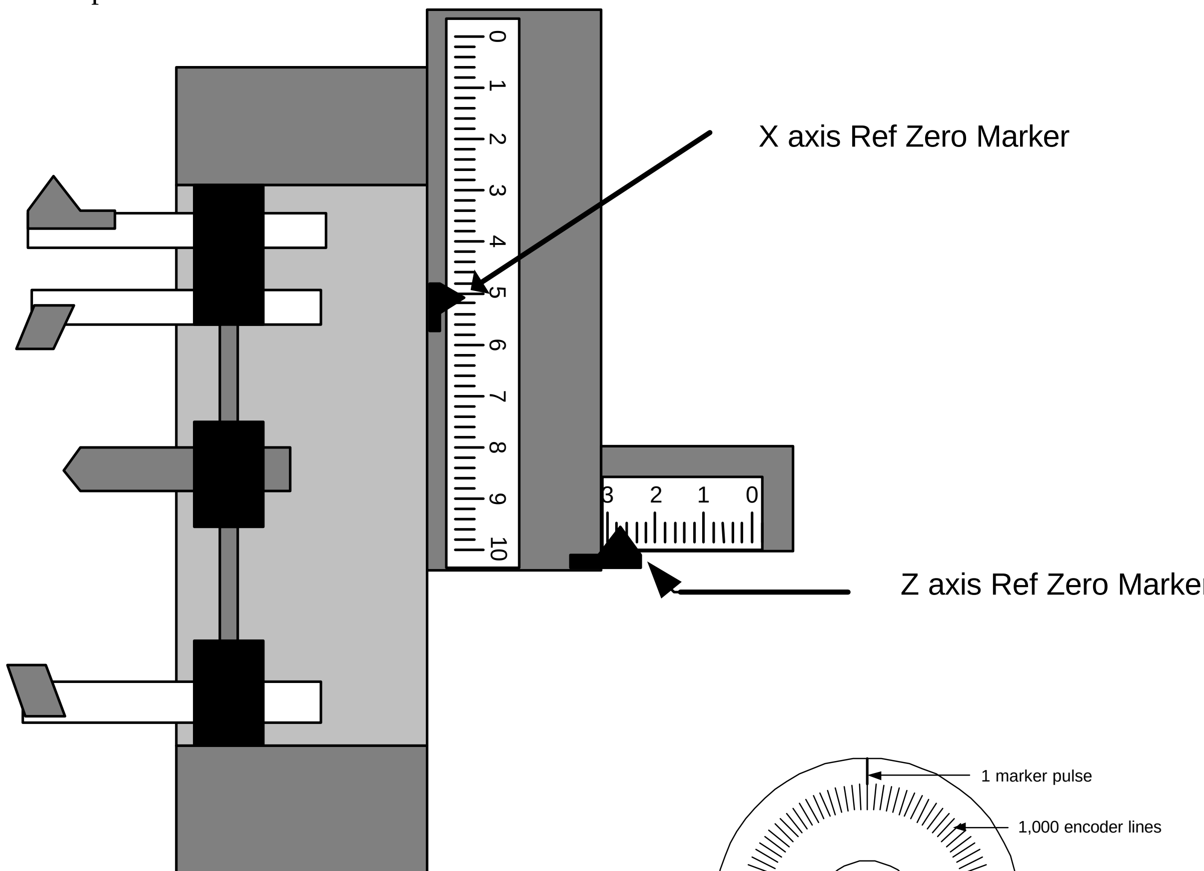


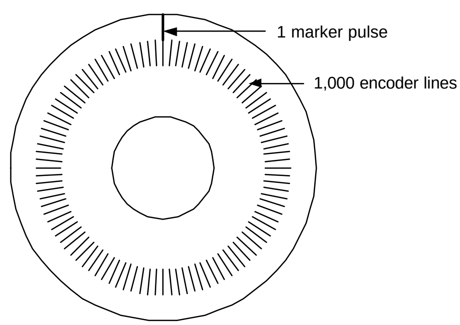
After selecting "Jog 9" (Est Home), press Cycle Start to complete the homing sequence.

After Home has been established, you can leave the Jog Mode by pressing "Esc"key on the keyboard to go to the Main Menu, or you can stay in the Jog Mode and establish the tool offsets. On the following pages the Jog Mode is discussed in detail.

Establish Home: Explanation

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





How HOME is established

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

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

The drawing above is a sketch of this type of rotary encoder. The inside ring of lines is counted to maintain location. The outside line is used to find a definite location. The encoder will ”see” this line once every revolution of the ball screw used to position the slide. The ball screw used on the OmniTurn moves the slide .2” / revolution. So if we position the slide within .2” 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.

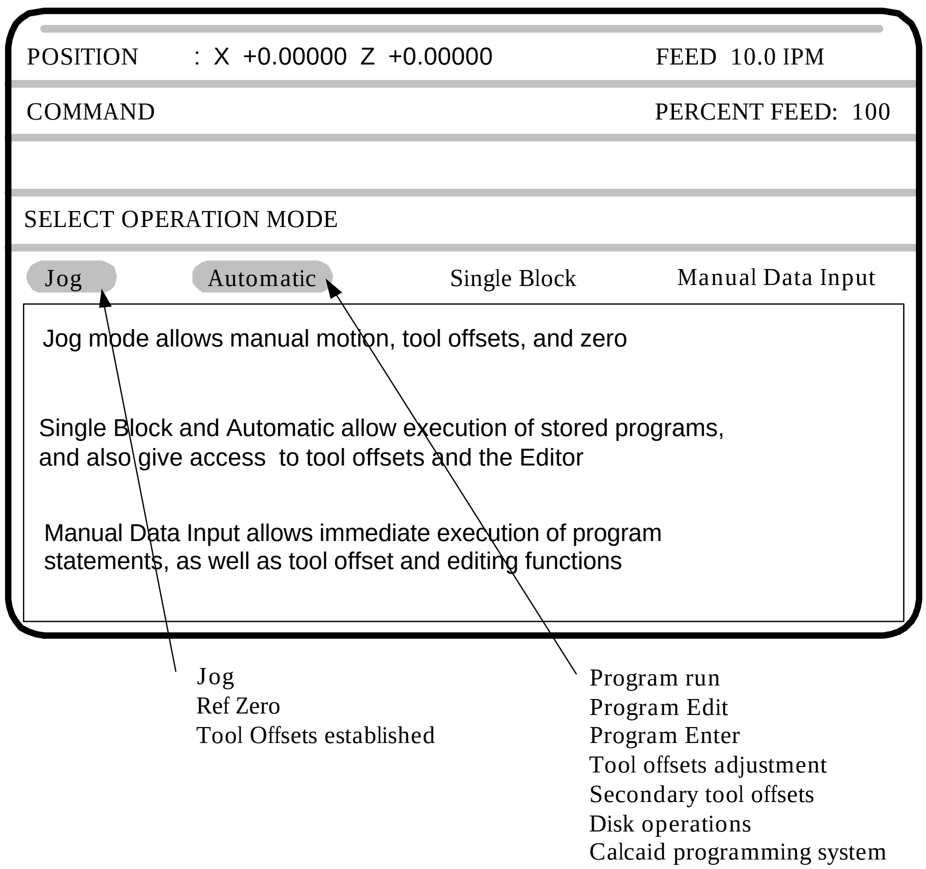
Jog

The purpose of the jog mode is to:

- Establish your ”HOME” position. This is always done after starting up the control.
- Set tool offsets
- Move the slide manually
- Bore soft jaws or collets

To enter the jog mode:

If you are just powering up the control it will automatically take you to the jog screen after the servos are turned on.
If you are at the main menu as shown below, type ”J” .

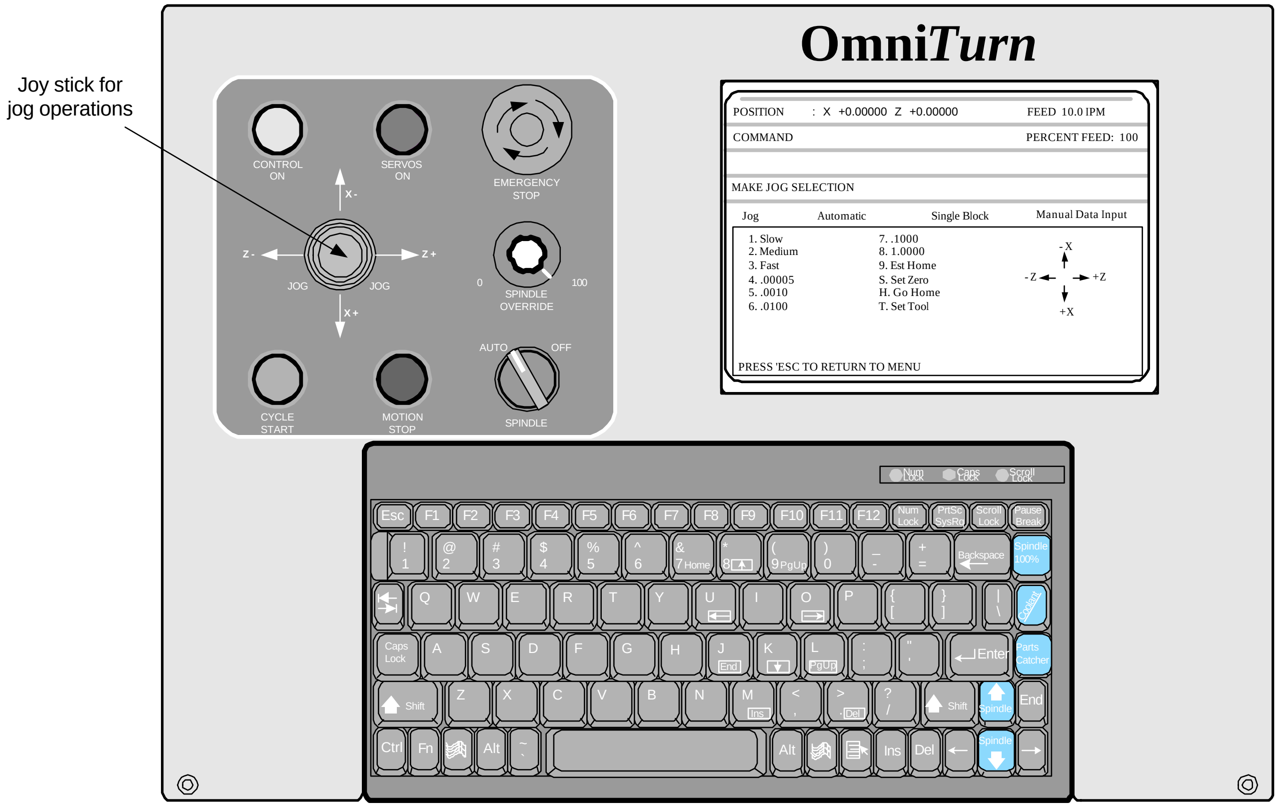


Jogging with the Joy Stick

The method for moving the slide is to use the ”Joy stick” actuator on the operators panel. Deflecting this actuator in the four directions available produces the following motions:

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

Jog



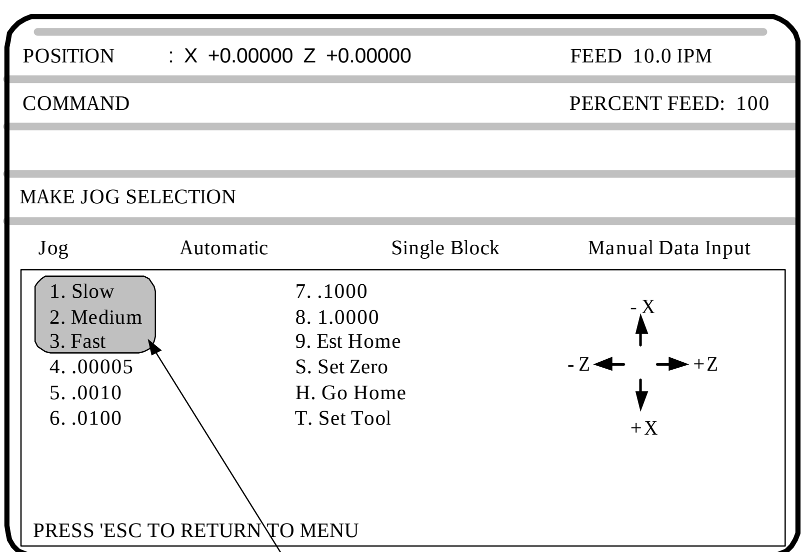
OmniTurn front panel Keyboard

The speed or increment of the jog mode can be selected. This is done by pressing a number between 1 -8 on the keyboard. The jog speed or increment selected zzzwill change shade. To select a new speed just select a new number.

Continuous Jog Mode

Deflecting the joystick after selecting 1 -3 will move the slide at a continuous rate:

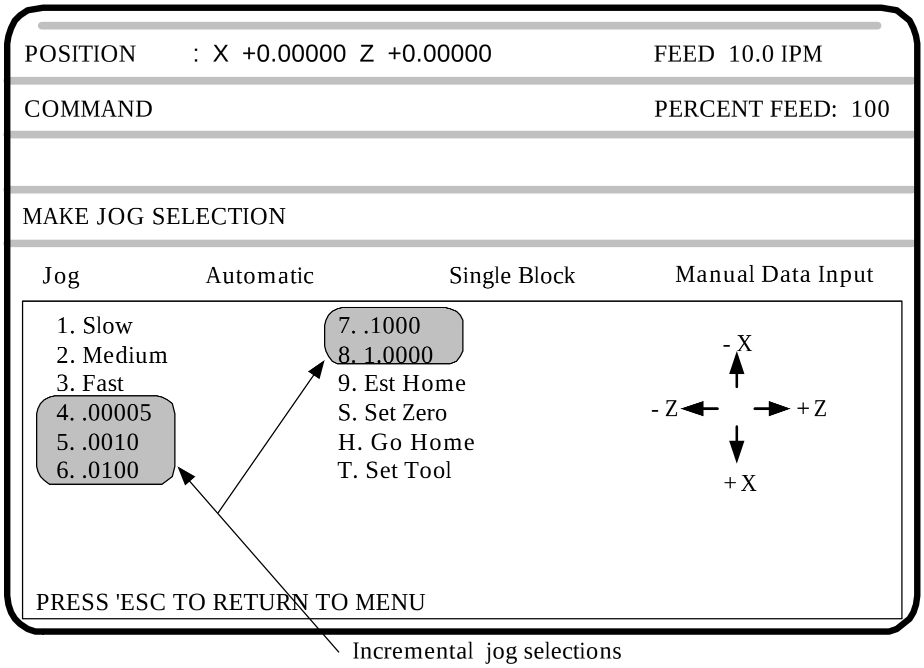
1.	SLOW	1 inch per minute
2.	MEDIUM	10 inches per minute
3.	FAST	100 inches per minute



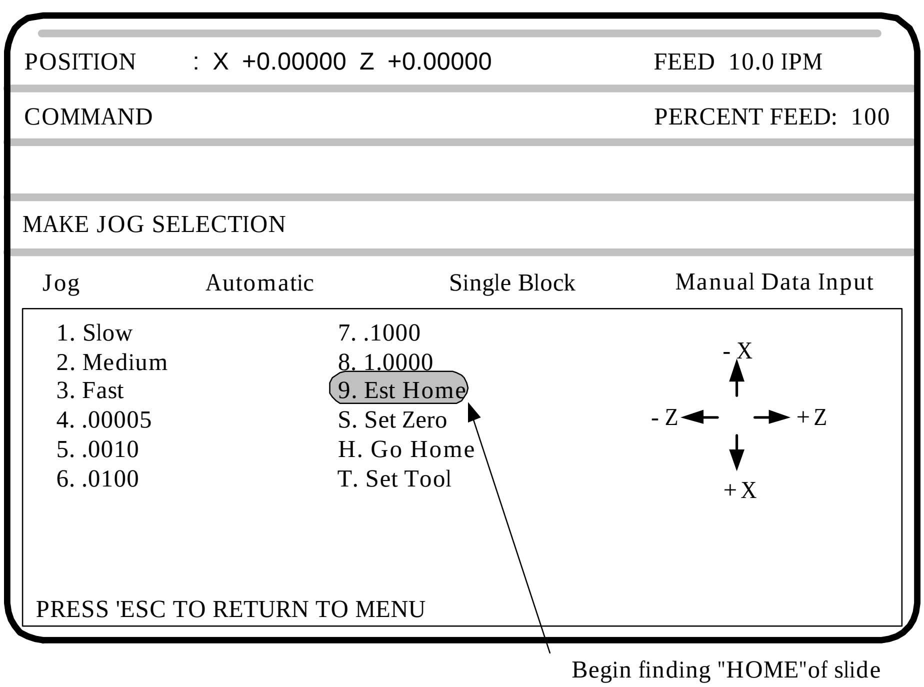
Continuous jog selections

Incremental Jog Mode

Deflecting the joystick after selecting 4 -8 will move the slide the indicated increment in the direction deflected. Each deflection produces one increment. Holding the joystick in the deflected position will not move the slide more than one increment. The cycle start light flashes to indicate recognition and then execution of the increment.

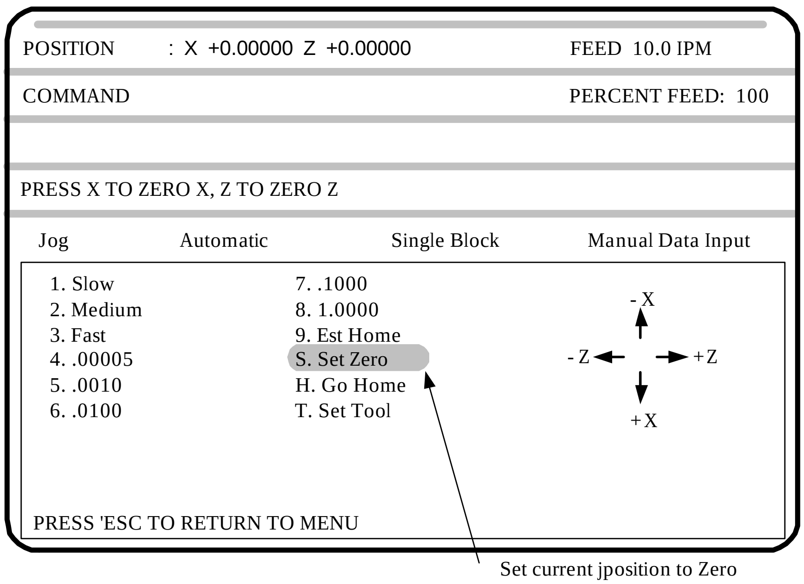


The #9 selection is covered in the Establish Home section



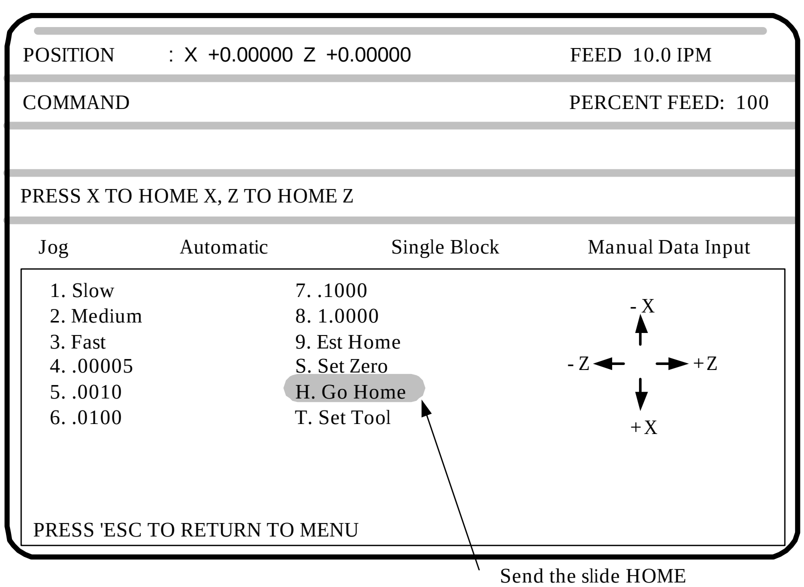
To set a floating zero

Depressing ”S” will cause the present position of the axis to become the zero point. This setting of zero does not effect the location of the Home. This is a local zero that is used in the manual mode if the operator cares to use it. When this option is used, the control will prompt you to ”PRESS X TO ZERO X, Z TO ZERO Z”. You can terminate this procedure at any time by pressing the ”ESC” key.



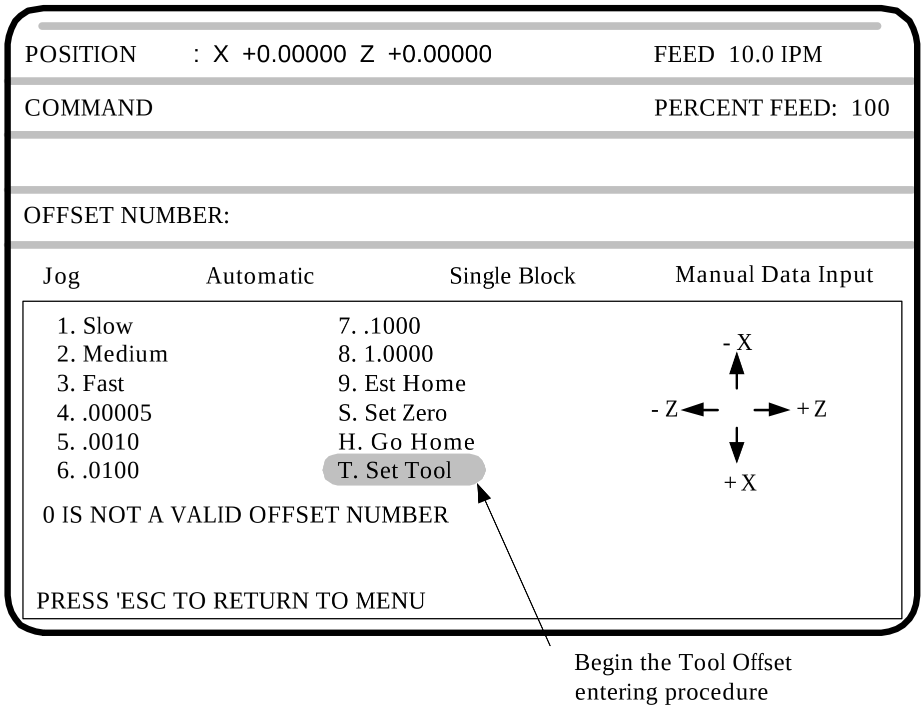
To send the slide HOME

Depressing ”H” will bring up the message ”PRESS X TO HOME X, Z TO HOME Z”. It is suggested to depress Z first, as this will home Z axis and get the tools back out of the way. Then depress X and the slide will home in the X axis. This function is used to check that the Home you have set is still where you think it should be. Never use this function until you have first gone through the ”REFERENCE ZERO” procedure after start-up.



To set Tool Offsets

The ”T” selection is used to begin the Tool Offset procedure.
This is covered in Section 3, ”TOOL OFFSETS”.



Hard Disk Features

Backing Up Programs

OmniTurn CNC Controls are shipped with a solid-state hard drive **(C:)** for system and programs, and one 3.5 floppy drive, **(A:)** for part program backup.

The user is given an opportunity to backup his program files when machine is booted up. There is no convenient, automatic way to backup before shutting down, so we back-up at turn-on instead.

The solid-state hard-drives are very reliable, and unlikely to 'crash' but like any piece of machinery, they *will* crash someday. Back up your programs when you boot up.

The message "**Do you want to back up your program files?**" will be displayed.

Answering yes by pressing 'Y'will backup the files to drive A:.

Only new programs or those which have changed since last backup are copied.

Copying Programs

User can **copy** programs between hard drive and floppy using the file handler from Auto Mode by selecting F8 (DiskOps), then choosing 1 (Copy Files to/from Hard Drive).

See next page for full description of file handling.

Programs are listed in alphabetical order, and can be searched by the first letter of the name. The first ten lines of each program are displayed, but the entire program can be reviewed if desired: tab to the preview window below the programs window, then use arrow keys or page up/page down keys to look at every block. Editing in these windows will not change program.

Individual programs can be **deleted** from the hard drive by hi-lighting them in the hard drive window, then pressing Ctrl-D. **To copy an entire diskette of programs to hard drive, hi-light any file in floppy window, then press Ctrl-C.**

To Restore Hard Drive

The hard-drive is high-reliability 'solid-state, with no moving parts. A hard drive restore disk is shipped in a pouch inside the rear panel of the control for use in the rare event that thehard drive fails for any reason. After the new hard drive is installed, follow the instructionson the restore disk label.

Copying Programs to and from the Hard Drive

The OmniTurn allows true "background editing" because programs copied to the Users Disk may be edited at a desk-top computer while the program is running on the OmniTurn. The newly edited program can be copied back to the hard drive from the Auto Mode screen with almost no interruption in the machining process.

In Auto Mode select F8.
Disk Options Menu will appear.

Press 1 to copy programs TO or FROM the hard drive.
OmniTurn File Copy Screen will appear.

There are four windows, a help line and an exit button.
1. Upper left window lists the programs on the User Disk
2. Lower left window displays the program lines of the program high-lighted in the upper left window.
3. Upper right window lists the programs on the Hard Drive
4. Lower right windows displays the program lines of the program high-lighted in the upper right window.
5. The text below the top two windows displays the action as the highlight is moved from window to window.

Use the arrow keys, or the initial letter of the program name to highlight a program to view or copy.

Use the Tab key to move from window to window, including the Exit box. Holding the Shift key, then pressing Tab will move the cursor to the previous window.

To copy a high-lighted program to or from the hard drive, press the Rtrn key.

To **delete** a high-lighted program **from the hard drive** press and hold Ctrl key, then press d key (Ctrl-D). There is **no prompt:** the program is deleted. You cannot delete programs from the floppy.

To **copy all programs from floppy to hard drive**, high-light any program in floppy window, then press Ctrl-C.

Changes made in the preview window WILL NOT change the program.

To exit the file handler press and hold the Alt key, then press X, or use the Tab key to move the cursor to highlight the Exit button, then press Rtrn.

```
OmniTurn FILE COPY MENU

Programs on Floppy                                   Programs on Hard disk
  DEMO                                                 DEMO
  PROG1                      Exit                      PROG1
  PROG2                                                PROG2
  PROG3                                                PROG3
```

Top prompt line area (see Notes on Use, below)

```
/------------------------   PREVIEW PROGRAMS ------------------------\
Program on Floppy                         Program on Hard Drive

Use Tab or Shft-Tab Between Boxes, arrows Within Boxes, Rtrn to Select
```

Using the File Handler: Notes and Prompts

NOTE: You must have floppy in drive a:

Bottom Prompt Line: (moving around)
Use Tab or Shft-Tab Between Boxes, Arrows Within Boxes, Rtrn to Select
Use tab key to move counter-clockwise between windows; shift key with tab key to move clockwise. When in any window, use up and down arrow keys to move from line to line.

Top Prompt Line, Initial condition:
Press Arrow or Initial Letter to Select Progarm. Press Tab Key for Next Window
The cursor is in the Programs on Floppy window. If user presses the initial letter of any program on disk, the cursor will jump to first program which starts with that letter, the program will be selected, and the program lines will be displayed in the Floppy Preview Programs window. If the user presses the down arrow, the first program will be selected, and the program lines displayed.

When program is selected, Top Prompt Line changes:
Copy filename to Hard Drive? (Rtrn = Copy this File; Ctrl-C = Copy ALL Files)
If the user presses Enter, the program is copied and the Top Prompt changes as follows:
Copying filename to Hard Drive appears while program is being copied, then filename Copied to Hard Drive appears after program has been copied. If the user presses and holds Ctrl key, then presses ‘C’ key (Ctrl-C), all programs will be copied to the hard drive.

Tab to program view window with program selected:
User may view entire program by using arrow keys. ***Changes made in preview window will not affect program.***

Tab to Hard Drive window to copy programs to floppy, or to delete programs:
Copy FILENAME to Floppy? (Rtrn = Copy: Ctrl-D to Delete !NO PROMPT!)

To Exit, Tab to exit window:
Press Rtrn (or Space) to Exit, else Tab to Next Window

Codes Honored by the OmniTurn control

(Sort by Code)

Code	Usage	Description	Pages
G00	G00	Rapid move	11,12
G01	G01Fn	Feed move	12,13
G02	G02XnZnInKnFn	Arc -Clockwise	6,15,17-24
G02	G02XnZnRn	Arc -Clockwise	6,17-24
G03	G03XnZnInKnFn	Arc -Counter Clockwise	6,17-24
G03	G03XnZnRn	Arc -Counter Clockwise	6,17-24
G04	G04Fn	Dwell	6,25,62
G10	G10XnZn	Work Shift	6,26-28,73
G33	G33XnZnInKnCnPO	Threading cycle	6,29-36
G35	G35n	Extra Course feeds in IPR	6,29,36,74
G36	G36	Cancels G35	6,36,74
G40	G40	Cancels Tool Nose Radius Compensation	16,37-43
G41	G41	Left hand Tool Nose Radius Compensation	16,37-43
G42	G42	Right hand Tool Nose Radius Compensation	37-43
G70	G70	Inch mode	6,44
G71	G71	Metric mode	6,44
G72	G72	Diameter programming mode	6,10,14,16,21,22,29,38,44,46,49,59
G73	G73	Radius programming mode	6,10,14,16,21,20,22,29,38,44,46,49,59
G74	G74XnZnInUnFn	Box Roughing cycle	45-47
G75	G75InUnFnPn	Box Contour Roughing cycle	48-52,54
G76	G76Sn	Minimum spindle speed for constant surface feet	6,60
G77	G77Sn	Maximum spindle speed for constant surface feet	6,60
G78	G78UnFnPn	Rough Contour Cycle	51-55
G81	G81ZnFn	Drill cycle	6,56
G83	G83ZnKnFn RnLnCn	Peck drill cycle	6,57,58
G89	G89	Stop spindle and lock (C-Axis only)	
G90	G90	Absolute mode selection	5,6,10,12,17,19,21,20,56,57,59
G91	G91	Incremental mode selection	5,6,10,17,56,57,59
G92	G92XnZn	Preset axis position	36,59,74
G94	G94Fn	Inches per minute mode	6,7,11,12,20,45,49,56,59
G95	G95Fn	Inches per revolution mode	6,7,11,12,45,49,56,59
G96	G96Sn	Spindle speed set as surface feet	6,60,62,65
G97	G97	Spindle speed set as RPM	6,60
M00	M00	Program stop - does not cancel active "M" functions	61
M0l	M0l	Optional stop	61
M02	M02	End program - does not cancel active"M" functions	26,62,65
M03	M03Sn	Spindle on, CW	16,62,65,74
M04	M04Sn	Spindle on, CCW	62,65,74

Codes Honored by the OmniTurn control

(Sort by Code)

Code	Usage	Description	Pages
M05	M05	Spindle off, stop	62,65,74
M08	M08	Coolant on	16,62,65
M09	M09	Coolant off	62
M12	M12	Collet clamp	62
M13	M13	Collet unclamp	62
M19	M19	Spindle Positioning (optional C-Axis only)	62,74
M25	M25	User assigned on	62
M26	M26	User assigned off	62
M30	M30	End of program - cancels all active ”M” functions	26,62,65
M31	M31	Cancels Cycle Repeat mode	62
M89	M89	Stop the spindle and lock it (optional: C-Axis only)	63
M91	M91	Wait for TB2-5 to be open circuit (optional: C-Axis only)	63
M92	M92	Wait for TB2-5 to be short to 0VDC (optional: C-Axis only)	63
M93	M93	Wait for TB2-7 to be open circuit (optional: C-Axis only)	63
M94	M94	Wait for TB2-7 + to be short to 0VDC (optional: C-Axis only)	63
M95	M95	Jump to subroutine 1 if TB2-9 is short to 0VDC (opt: C-Axis only)	63
M97	M97InCnPn	Jump to subroutine, conditional (optional: PLC option only)	63
M98	M98Pn	Jump to subroutine	63
M99	M99	End subroutine	63
CI	CInnn.nn	Incremental spindle angle (optional: C-Axis only)	74
CA	CAnnn.nn	Absolute spindle angle (optional: C--Axis only)	74
C	XnZnCn	Automatic chamfer at intersection	15,16,67
D	Dn	Secondary offsets, axis correction or TNR comp value	68-71
F	Fn	Feedrates, dwell	48,56
LS	LSn	Loop start	72
LF	LF	Loop finish	73
R	XnZnRn	Automatic radius at intersection	15,16
S	Sn	Spindle speed selection, SFM or RPM	60,65,66,74
T	Tn	Tool offset call command	9
/	/	Block delete	*Section 5.3*
}	}n	Begin subroutine	63

Codes Honored by the OmniTurn control

(Sort by Description)

Code	Usage	Description	Pages
G90	G90	Absolute mode selection	5,6,10,12,17,19-21,56,59
CA	CAnnn.nn	Absolute spindle angle (optional: C—Axis only)	74
G02	G02XnZnInKnFn	Arc -Clockwise	6,1517-24
G02	G02XnZnRn	Arc -Clockwise	6,17-24
G03	G03XnZnInKnFn	Arc -Counter Clockwise	6,17-24
G03	G03XnZnRn	Arc -Counter Clockwise	6,17-24
C	XnZnCn	Automatic chamfer at intersection	15,16,67
R	XnZnRn	Automatic radius at intersection	15,16
}	}n	Begin subroutine	63
/	/	Block delete	*Section 5.3*
G75	G75InUnFnPn	Box Contour Roughing cycle	48-52,54
G74	G74XnZnInUnFn	Box Roughing cycle	45-47
M31	M31	Cancels Cycle Repeat mode	62
G36	G36	Cancels G35	6,36,74
G40	G40	Cancels Tool Nose Radius Compensation	16,37-43
M12	M12	Collet clamp	62
M13	M13	Collet unclamp	62
M09	M09	Coolant off	62
M08	M08	Coolant on	16,62,65
G72	G72	Diameter programming mode	6,10,14,16,21,22,29,38,44,46,49,59
G81	G81ZnFn	Drill cycle	6,56
G04	G04Fn	Dwell	6,25,62
M30	M30	End of program - cancels all active M functions	26,62,65
M02	M02	End program - does not cancel active M functions	26,62,65
M99	M99	End subroutine	62
G35	G35n	Extra Course feeds in IPR	6,29,36,74
G01	G01Fn	Feed move	12,13
F	Fn	Feedrates, dwell	48,56
G70	G70	Inch mode	6,44
G94	G94Fn	Inches per minute mode	6,7,11,12,20,45,49,56,57,59
G95	G95Fn	Inches per revolution mode	6,7,11,12,45,49,56,57,59
G91	G91	Incremental mode selection	5,6,10,17,56,57,59,70
CI	CInnn.nn	Incremental spindle angle (optional: C-Axis only)	74
M98	M98Pn	Jump to subroutine	63
M95	M95	Jump to subroutine 1 if TB2-9 is short to 0VDC (opt: C-Axis only)	63
M97	M97InCnPn	Jump to subroutine, conditional (optional: PLC option only)	63

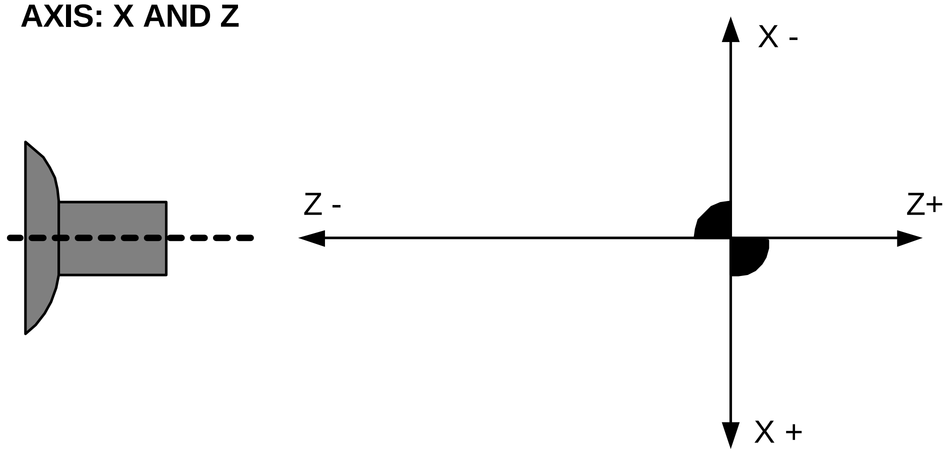
Codes Honored by the OmniTurn control

(Sort by Description)

Code	Usage	Description	Pages
G41	G41	Left hand Tool Nose Radius Compensation	16,37-43
LF	LF	Loop finish	73
LS	LSn	Loop start	72
G77	G77Sn	Maximum spindle speed for constant surface feet	6,60
G71	G71	Metric mode	6,44
G76	G76Sn	Minimum spindle speed for constant surface feet	6,60
M0l	M0l	Optional stop	61,62
G83	G83ZnKnFnRnLnCn	Peck drill cycle	6,57,58
G92	G92XnZn	Preset axis position	36,59,74
M00	M00	Program stop - does not cancel active M functions	61,62
G73	G73	Radius programming mode	6,10,17,18,20,29,31,32,38,44,46,49,59
G00	G00	Rapid move	11,12
G42	G42	Right hand Tool Nose Radius Compensation	37-43
G78	G78UnFnPn	Rough Contour Cycle	51-55
D	Dn	Secondary offsets, axis correction or TNR comp value	68-71
M05	M05	Spindle off, stop	62,65,74
M04	M04Sn	Spindle on, CCW	62,65,74
M03	M03Sn	Spindle on, CW	16,62,65,74
M19	M19	Spindle Positioning (optional C-Axis only)	62,74
S	Sn	Spindle speed selection, SFM or RPM	60,65,66,74
G97	G97	Spindle speed set as RPM	6,60
G96	G96Sn	Spindle speed set as surface feet	6,60,62,65
G89	G89	Stop spindle and lock (C-Axis only)	
M89	M89	Stop the spindle and lock it (optional: C-Axis only)	63
G33	G33XnZnInKnCnPO	Threading cycle	6,29-36
T	Tn	Tool offset call command	9
M26	M26	User assigned off	62
M25	M25	User assigned on	62
M91	M91	Wait for TB2-5 to be open circuit (optional: C-Axis only)	63
M92	M92	Wait for TB2-5 to be short to 0VDC (optional: C-Axis only)	63
M93	M93	Wait for TB2-7 to be open circuit (optional: C-Axis only)	63
M94	M94	Wait for TB2-7 + to be short to 0VDC (optional: C-Axis only)	63
G10	G10XnZn	Work Shift	6,26-28,73

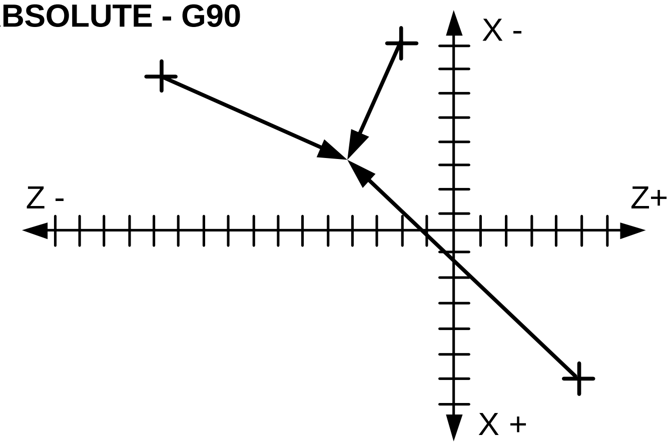
Nomenclature

AXIS: X AND Z



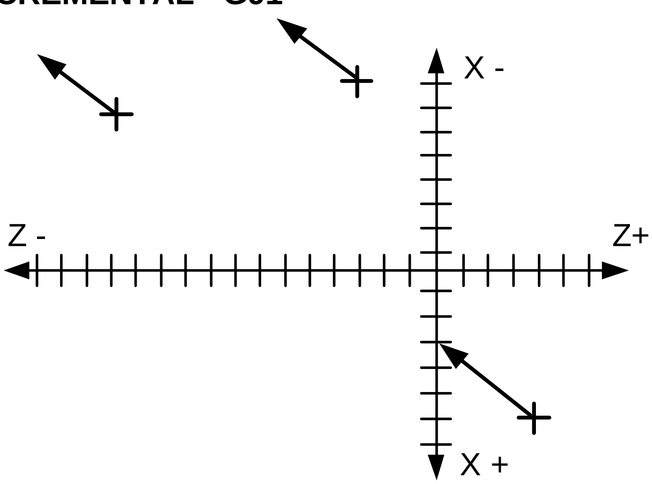
The slide has two axis's of travel.
X: Towards and away from you. Travel away from you is (-) minus. Towards you is plus (+).
Z: The slide that travels along the axis of the spindle. Going towards the spindle is (-) minus. Away from the spindle is (+) plus.

ABSOLUTE - G90



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

INCREMENTAL - G91



IN INCREMENETAL (G91) THE FOLLOWING MOVES YOU THE SAME AMOUNT FROM EVERY START.
X-3 Z-4

Programming Format

- **The default** mode for X moves is G73 - radius moves, to program in diameters you must use G72 in the beginning of the program.

- **The first command** of a program must be G90 or G91 to define if the program is in absolute or incremental.

- **No blank** lines are allowed in a program, blank spaces are OK.

- **Comments** are any text or data enclosed in parentheses“( )”.
  Their purpose is to convey to the operator any information that the programmer might think is useful. Comments are displayed in the lower left corner of the screen. They stay on the screen till the comment is changed. As an example, you may want to use the comment to tell the operator what action to take when the spindle stops. For an example, the slide is told to go ”HOME” and then the comment is displayed on the screen. Then the slide stops with the message on the screen.

  - Do not put text on lines by itself. Comments must be on a line with a command!
  - Keep the amount of text to a minimum, to much text can cause problems.
  - A good place to put comments is on a line with a tool call ie: T1(LH turn tool)
  -Use only text, do not use periods or commas or any other symbol such as i.e.:
  ! @ # $ % ^ & * “ ‘ ? > < / \| = -.

- **Do not put** any text in a loop.

- **Commas are not allowed anywhere in the program**

- **Dimensional** data is interpreted with a resolution of .00005”. The fifth digit to the right of a decimal point must be a 0 or a 5. NOTE: when programming in diameter mode the X axis resolution is .0001 ”, not .00005”.

- **Decimal** point programming is used. Leading and trailing zeros need not be entered.
  For example ”X1” is interpreted as 1 inch. X1 = X1.00000

- **G and M codes** must be programmed as two digit codes. ”G2” is not a legal code and it will be ignored. Also be sure to use the zero and not the letter O as part of the G and M codes.

- **Model commands**: These are commands that remain active until canceled:
  G90, G91 -G94, G95 -G70,G71 -G76, G77, G96,G97 -G72, G73
  All ”M” codes, G35, G36 (GT-75 only)-G10

- **One shot commands:** These act only on the statement they are programmed in:
  G02, G03 -G04 -G33, G34 -G81, G83 -G92

- **Conflicting commands:**
  There can be only one ”M” command per line of code
  There can be only one “one shot” G code per line of code
  There can be more than one nonconflicting modal G code per line
  The S and F commands can be with any other command

- **N sequence** are not allowed. They can cause intermittent problems.

Programming Format

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

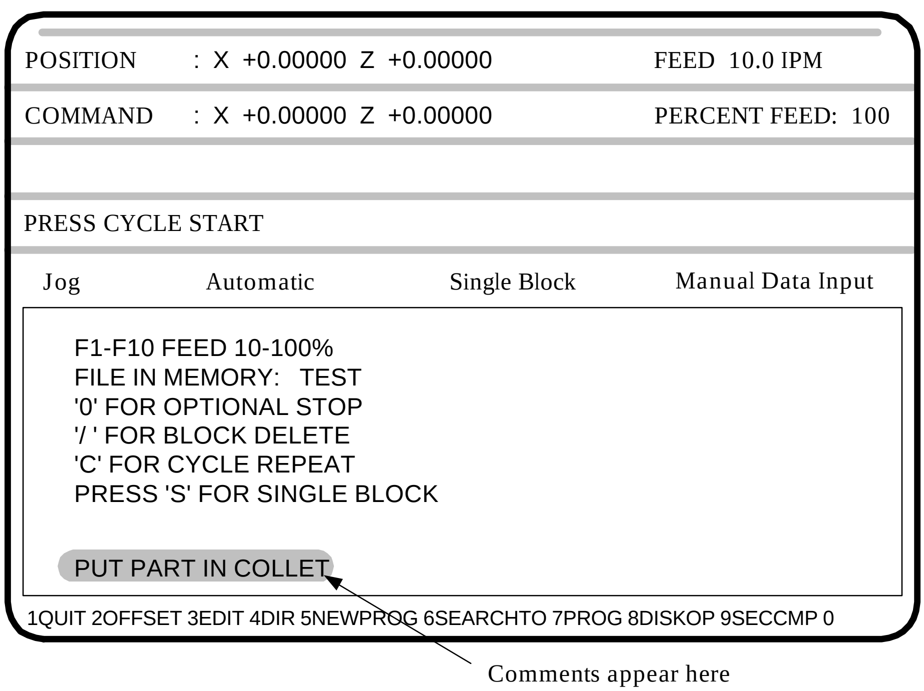
  X.1Z-.4F7 This line of code has a feedrate of 7 IPM when in G94 (F300 max)

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

- **Tool Selection:**
  This command is used to make a new tool offset active. The letter ”T” identifies a tool command.

  T2 Makes the tool offset for T2 active. After this command is executed the absolute position display will show the distance from the tip of tool two to the absolute zero of the work piece.

  T1
  X0Z2
  M00 (PUT PART IN COLLET)



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

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

Creating a new program

There are a number of ways to create a new program. Here are a few:

-Use the text editor in OmniTurn. This is found in the Automatic section, use F3. (for additional information on the editor and this function please refer to section 5).

-Go to the Automatic mode menu. First a new program name has to be created. This is done by going into the Automatic mode and typing in the new name when the control asks "FILE TO BE PROCESSED". After the RETURN key is hit the control will answer "FILE NOT FOUND, PRESS ANY KEY TO CONTINUE". By doing this you have accomplished two things.

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

2. If there was no other program that had the new name then there was one created and loaded into the text editor. Once the new name is entered into the text editor, press F3 to enter the editor. The text editor will ask "PRESS F1 TO CREATE A NEW FILE, ESC TO ABORT". After pushing F1 the editor will provide a new blank screen to enter your program.

-CAM system off line. Transfer a file via floppy or RS-232. Once they are on the OmniTurn program disk they can be run like any other existing program. Please refer to the section in DOS notes on the format.

-RS-232 or Disk transfer. Manually enter a program in a text editor on another computer, transfer as above. Once they are on the OmniTurn program disk, they can be run like any other existing program. Please refer to the section in DOS notes on the format.

-**Use Calcaid in OmniTurn**. See the section on using Calcaid.

Tool call statements

The "**tool call statement**" is done in two lines:

Tn
XnZn

The T command must have a tool number in the range of I through 32. When this command is executed the slide will not move. The display showing the absolute location will change to show the distance the tool being used is from the absolute zero of the part. In order to use this command the tool must have its tool offset established first.

The following line must be a X and Z move. You must have both values given! This is the location that the tool will move to. In the beginning you should have the tool come to a location that is clear of the part in Z, and then move the tool into the work. This will help eliminate some the tool interference problems new users encounter.

T1...Tool call for #1
X.25Z1 ....move the tool to a location of X=.25 and Z= I
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).

X moves default to RADIUS mode (G73), not diameters. If you want your moves to be in diameter you must put a G72 in the beginning of your program. See the example at the end of-this chapter

Feed and Rapid moves are accomplished as follows:

Movement in X:

Xn	Single axis (X) move
example: X.25	This will move to a diameter of .5

The programming in X can be done as radius movements (G73) or diameter (G72).

Movement in Z:

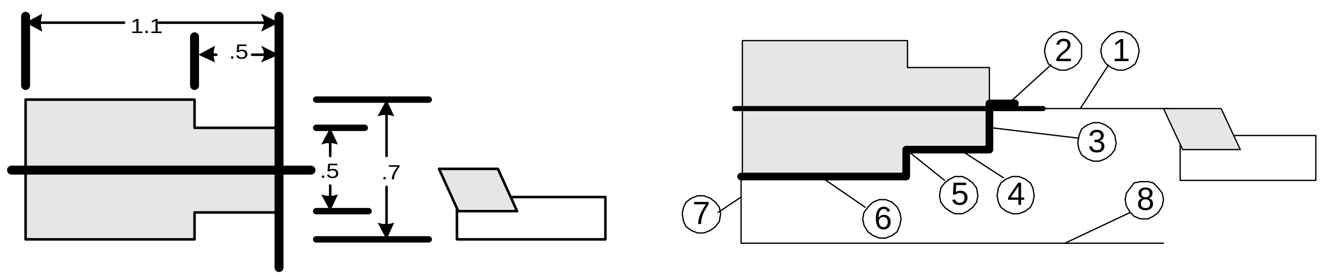
Zn	Single axis (Z) move
example: Z-.1	This will move the tool -.1 into the material

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)	Z1 G00	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”/minute. F.001 would be .001”/minute, Very Slow!!! This is a good way to rapid around, but it is not a good way to program feed (cutting) moves. If the spindle is not turned on the slide will still have the tool make the cut.

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.

Example formats are:

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

Rapid travel (G00):
Rapid travel is accomplished by either:

Using G94 for rapid travels:
Setting the feedrate in the Inches per minute mode (G94) with a value of F300 (check your system to see what the fastest IPM rate is, systems shipped before 7/93 are usually 200IPM. This will move the slide at its fastest rate. When the rapid moves are completed the feedrate is reset to the desired feedrate. An example format is:

G94F300	Set the feed to IPM @ 300”/minute
Xn	The move in X
Zn	Move in Z

Note:
The advantage of using this way to rapid verses the G00 command is that you can adjust the speed of the rapid travel by changing the Fnnn number. With G00 you are fixed to the rapid travel feedrate set in the first line of code. With the G94Fnnn method you can adjust the rapid travel for each move if you need to.

Note good tip:
When you are running a program for the first time and proving it out you could make all of your rapid moves F100. Then when the program is known to work you could change all the F100 to F300 by using the editor with ‘find and replace ”. See the notes in chapter 5, F3 on using this feature.

Linear moves X n Z n XnZn

Using G00 for rapid travels

The rapid travel mode can be established by using the G00 command. This will set the mode to IPM at a feedrate specified in the first line of the program. If you do not set the feedrate in the first line of the program it will use the last rapid move it did in either the manual mode (100”/min) or from the last program that was run. So it is best to establish a rapid rate in your program. In the following example we set the speed to 300”/min in the first line of the program. The next time you use a G00 command the rapid travel will be 300”/min.

G90G94F300

In the following example the G00 is used to change to the rapid mode:

G00	Set the feed to what is set in the first line of the program.
Xn	The move in X
Zn	Move in Z

Feedrate mode (G01):

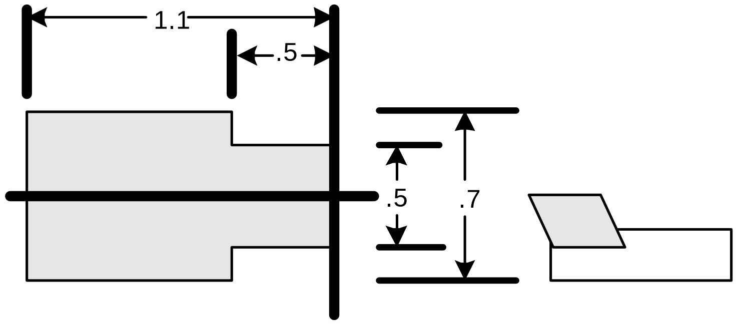
You can go from rapid travel mode to feedrate mode by either specifying a new feedrate and mode, IE: G94F300 for rapid travel, and G95F.n for feedrate. Or using G01 to revert to the last described feedrate mode in the program. This eliminates having to use the G95F.n over and over. The other advantage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

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

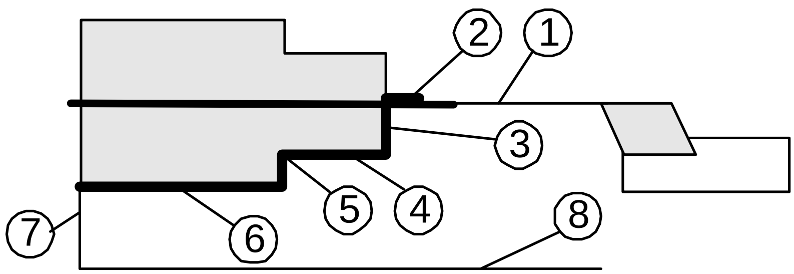
```
G94F300X0Z1
Z.1
G95F.003Z.005
X.5 G00Z.05
X0
G01Z0
X.5
G00Z1
```

Linear moves X n Z n XnZn

In the example below we will be finish turning a sample part with a left hand tool. The part dimensions are given in diameters. When the part is programmed these will have to be converted to radius moves.
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.



The same example in radius mode

	G90 G 73 G94 F300	Establish Abs. positioning, Feed IPM @ 300"/minute, Radius mode
	Tl (turn tool)	Call Tool #1 offset
	XO Z1	moves the tool to a safe start location
(1)	Z.1	Rapid to .1" from the face
(2)	G95 F.003 Z0	Set feedrate to IPR @ .003"/ rev, Feed to the face
(3)	X.25	Feed out to .25" radius (.5" diameter)
(4)	Z-.5	Feed to Z -.5
(5)	X . 3 5	Feed to .7" diameter
(6)	Z-1.1	Feed to -1.1 in Z
(7)	G94 F300 X.5	Set feedrate to IPM @ 300"/minute, Pull the tool away in X
(8)	Z 1	Withdraw the tool in Z
	M30	End of program command

The same example with G00 and G01

	G90 G94F300 G73	Establish Abs. positioning, rapid IPM @ 300"/minute, diameter mode
	T 1	Call Tool #1 offset
	X0 Z1	moves the tool to a safe start location
(1)	Z.1	Rapid to .1 "from the face
(2)	G95 F.003 Z0	Set feedrate to IPR @ .003"/ rev, Feed to the face
(3)	X.25	Feed out to .25" radius (.5" diameter) '
(4)	Z-.5	Feed to Z -.5
(5)	X .35	Feed to .7" diameter
(6)	Z-1.1	Feed to -1.1 in Z

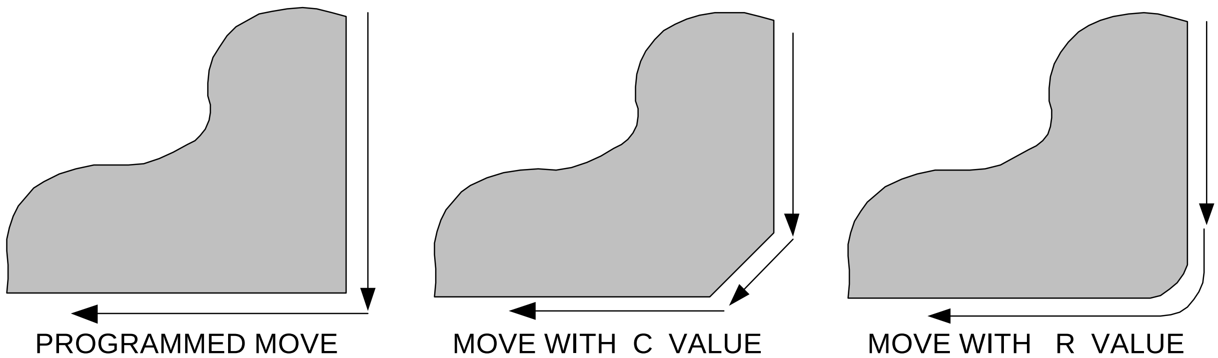
(7)	G00 X.5	Rapid feedrate, Pull the tool away in X
(8)	Z 1	Withdraw the tool in Z
	M30	End of program command

The same example written in diameter mode (G72)

	G90 G72 G94 F300	Establish Abs. positioning, Feed IPM @ 300"/minute, Radius mode
	T1 (turn tool)	Call Tool #1 offset
	X0 Z1	moves the tool to a safe start location
(1)	Z.1	Rapid to .1" from the face
(2)	G95 F.003 Z0	Set feedrate to IPR @ .003"/ rev, Feed to the face
(3)	X.5	Feed out to .25" radius (.5" diameter)
(4)	Z-.5	Feed to Z -.5
(5)	X.7	Feed to .7" diameter
(6)	Z-1.1	The moves to generate this tool path could be as shown below.
(7)	G94 F300 X1	Set feedrate to IPM @ 300"/minute, Pull the tool away in X
(8)	Z 1	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 move and put a C or R with the absolute amount of the radius or chamfer needed.



Format

X nZnRn -ZnRn -X nRn

X nZnCn -ZnCn -X nCn

XnZn The linear move leading to the intersection point of two lines
Rn The n is the absolute value of the radius used to blend the two lines
Cn The n is the absolute value of the chamfer used to blend the two lines

RULES

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

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

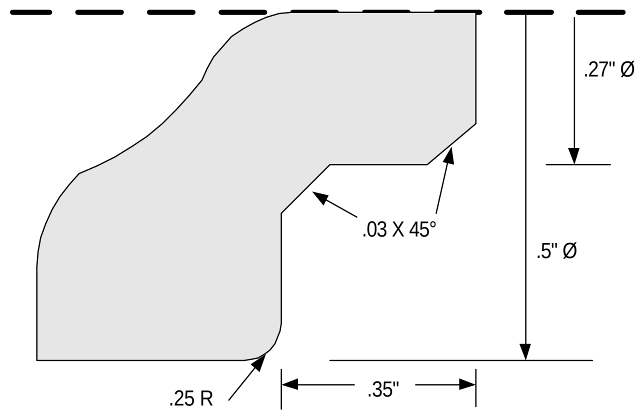
Running programs using C or R

When you use the automatic corner radius or chamfer commands the OmniTurn creates a number of moves to generate what you want. If you look at the command line while you run a program you will notice lines of code that you did 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

The arcs G02 and G03 are one shot commands. They are used one time and then turned off.
G02 is used to generate a clockwise arc.
G03 is used to generate a counterclockwise arc

G02 Xn Zn In Kn **or** **G03 Xn Zn In Kn**

G02 Xn Zn Rn **G03 Xn Zn Rn**

The programming of an arc is much different when written in diameter or radius modes.

Radius mode G73

Before you execute this command position the tool at the start of the arc.

The values given to the variables will effect the travel of the slide differently depending on whether the program is in absolute (G90) or incremental (G91).

Absolute Mode (G90) with G73 -radius programming active:

X the position of the end of the arc from absolute zero in X
Z the position of the end of the arc from absolute zero in Z
I the position of the center of the arc from absolute zero in X
K the position of the center of the arc from absolute zero in Z
R The length of the radius to be used to connect the start and end points

Incremental Mode (G91) -**not commonly used!:**

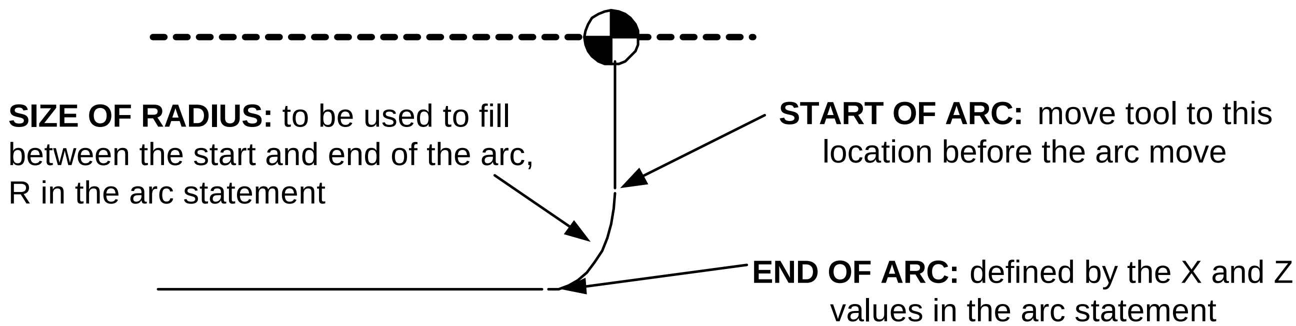
X the distance from the start of the arc to the end in X
Z the distance from the start of the arc to the end in Z
I the distance from the start of the arc to the center in X
K the distance from the start of the arc to the center in Z
R The length of the radius to be used to connect the start and end points

There are two versions of the arc statement. One uses I and K to define the location of the center of the are. When using this version it is important to calculate the values of all three locations exactly. If any of the values are off by .00005 the arc statement will not work. If the version with R is used there is a little flexibility built in. The arc used will be the one that best fits the end points and arc length.

Arc statements G02 and G03

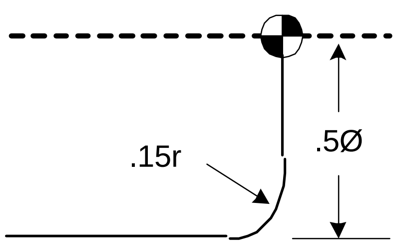
Using R version:

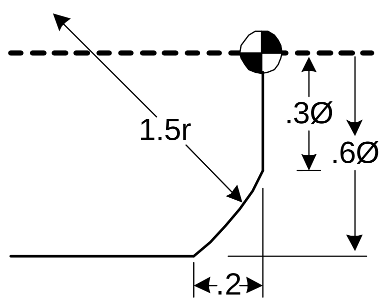
Before the arc statement is used the tool must be moved to the start location of the arc. Then the arc statement follows with the end of the arc location (X and Z) and the length of the connecting arc's radius.

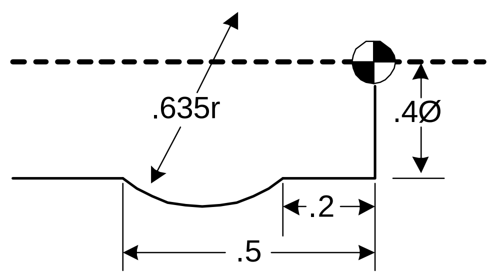


Examples of arc statements using R

Following are three examples of arc statements using the arc statement with R. These sample programs are showing only a finish pass. The are done in the radius mode -G73







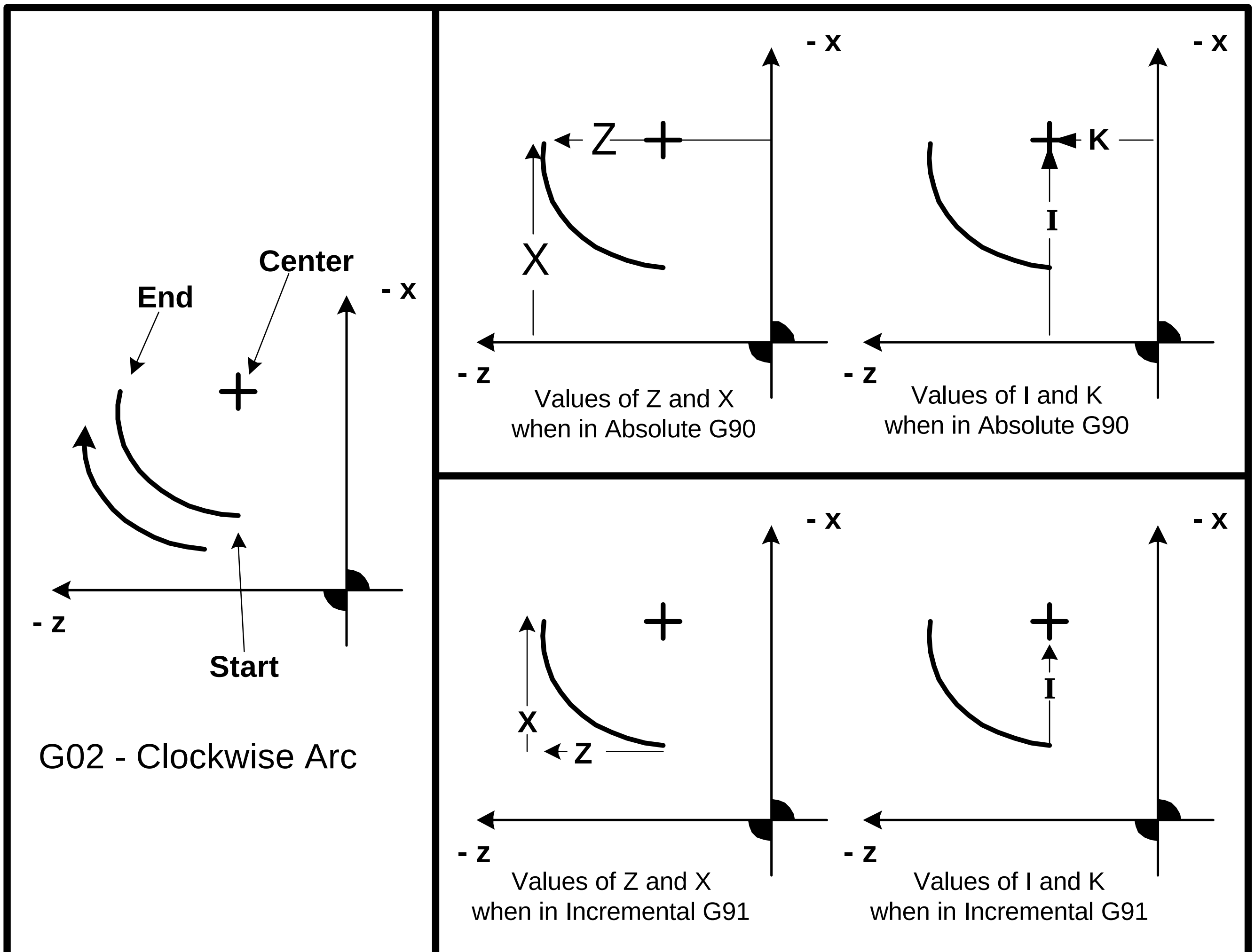

```
G90G94F300G73
M03S2500
M08
TI
X0Z1
Z.1
G95F.003Z0
X.1
G02X.25Z-.15R.15
Z-.3
X.275
G00Z1
M30
```

```
G90G94F300G73
M03S2500
M08
T1
X0Z1
Z.1
G95F.003Z0
X.15
G02X.3Z-.2R1.5
Z-.3
X.325
G00Z1
M30
```

```
G90G94F300G73
M03S2500
M08
T1
X0Z1
Z.1
G95F.003Z0
X.2
Z-.2
G02X.2Z-.5R.635
Z-.7
G00X.3
Z1
M30
```

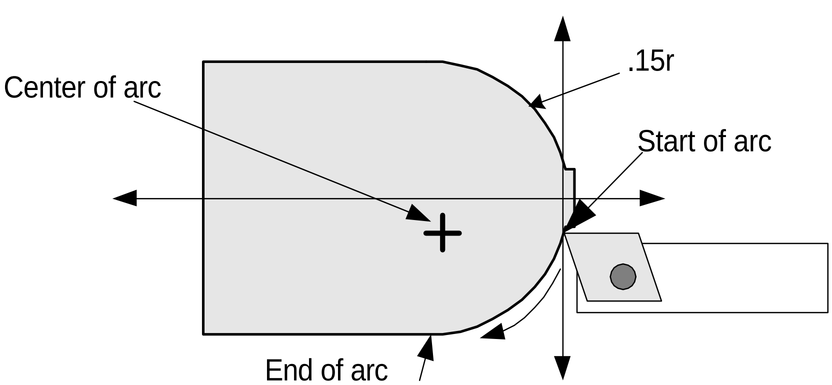
Arc statements G02 and G03

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



G02 - Clockwise Arc

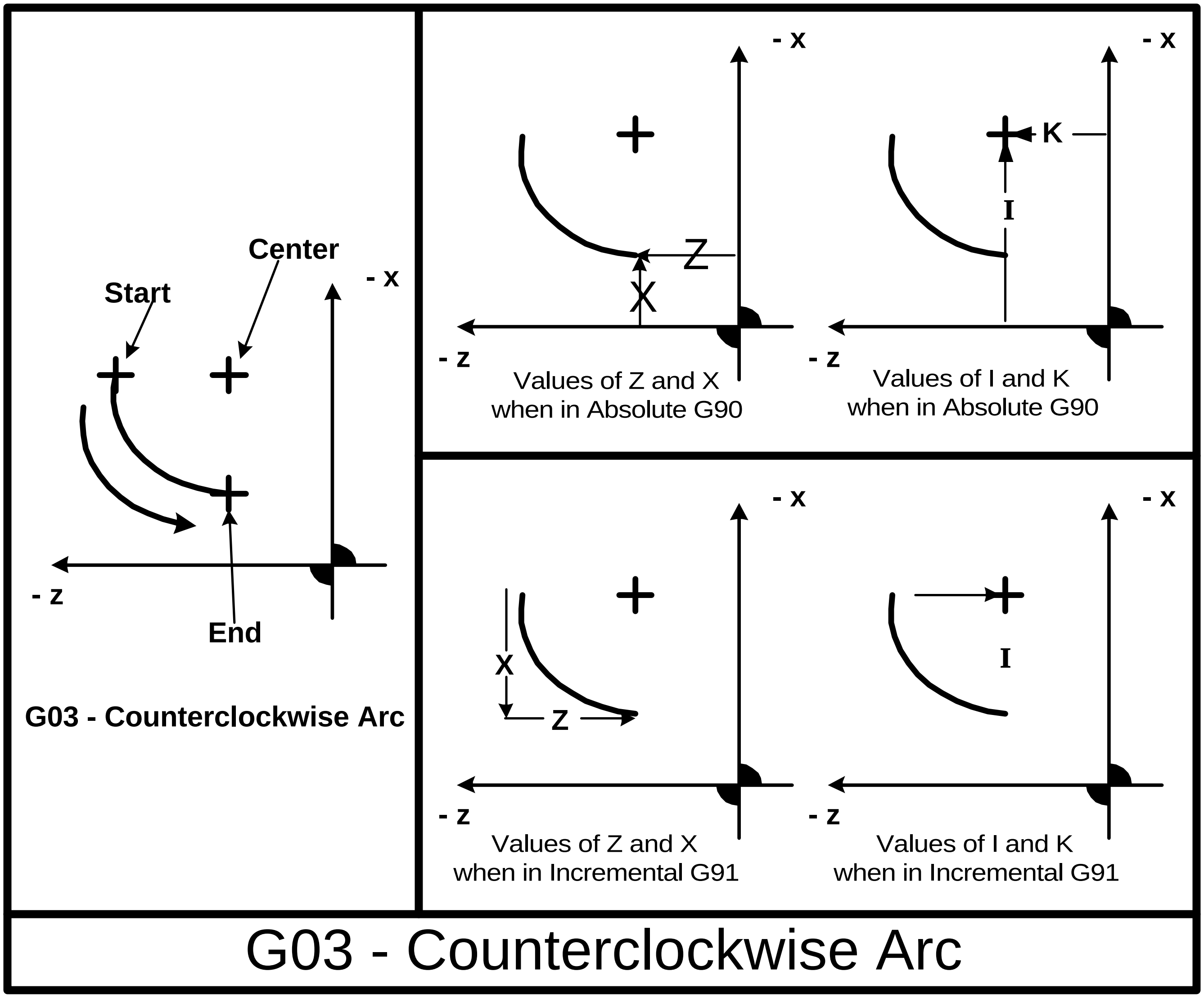
Using G02 - Clockwise Arc



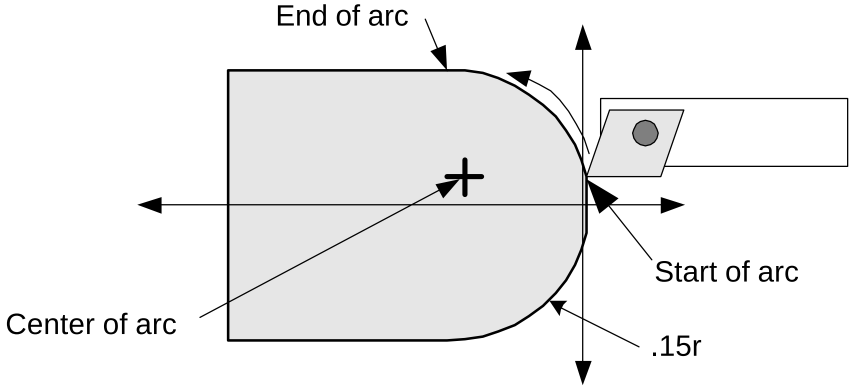
G90 G94 G73
T1F300
X0 Z.1
X.1 Z0
G02 X.25 Z-.15 I.1 K-.15
Z-.5
or the arc statment could be:
G02 X.25 Z-.15 R.15

Arc statements G02 and G03

G73 - Radius Mode



Using G03 - Counterclockwise Arc while in G73 - radius mode




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

Arc statements G02 and G03

Diameter mode

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

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

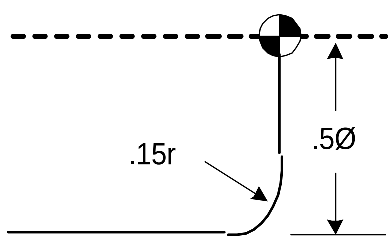
Note: -Using arc statements in the diameter mode (G72) be sure you are in absolute (G90)
-Position tool at start point before using arc move
-This format follows the Fanuc format more-closely than previously

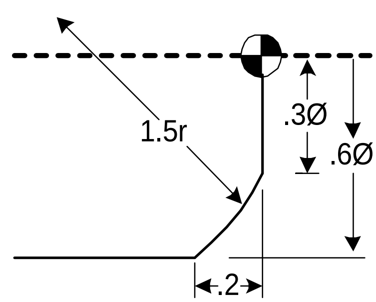
G02XnZnInKn • G03XnZnInKn

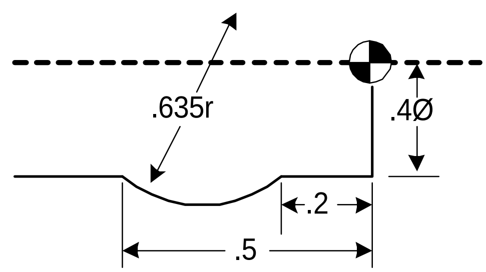
G02	Clockwise arcs
G03	Counter Clockwise arcs
Xn	Diameter value at the end of the arc
Zn	Location of end of the arc in Z from the part zero
In	Incremental distance from arc start to the arc center in X
Zn	Incremental distance from arc start to the arc center in Z
R	The length of the radius to be used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

Examples of arc statements using R in diameter mode:

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







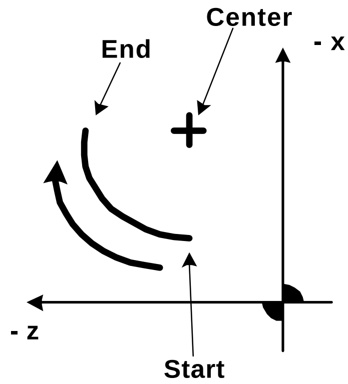

```
G90G94F300G72
M03S2500
M08
T1
X0Z1
Z.1
G95F.003Z0
X.2
G02X.5Z-.15R.15
Z-.3
X.55
G00ZI
M30
```

```
G90G94F300G72
M03S2500
M08
T1
X0ZI
Z. I
G95F.003Z0
X.3
G02X.6Z-.2R1.5
Z-.3
X.65
G00Z1
M30
```

```
G90G94F300G72
M03S2500
M08
T1
X0Z1
Z.1
G95F.003Z0
X.4
Z-.2
G02X.4Z-.5R.635
Z-.7
G00X.6
Z1
M30
```

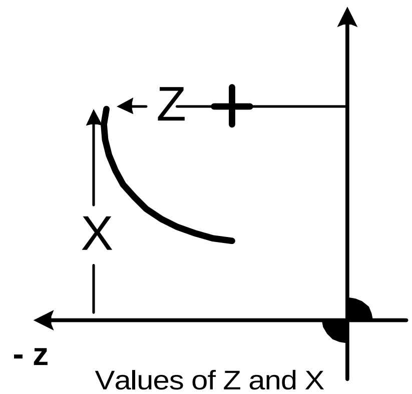
Arc statements G02 and G03

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



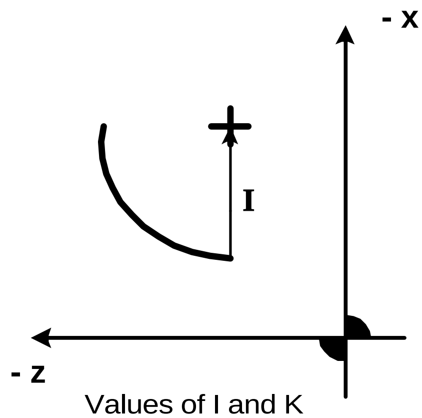
G02 - Clockwise Arc

End of arc (Xn Zn): This is the same. This is the location of the end of the arc.



Values of Z and X
when in Absolute G90

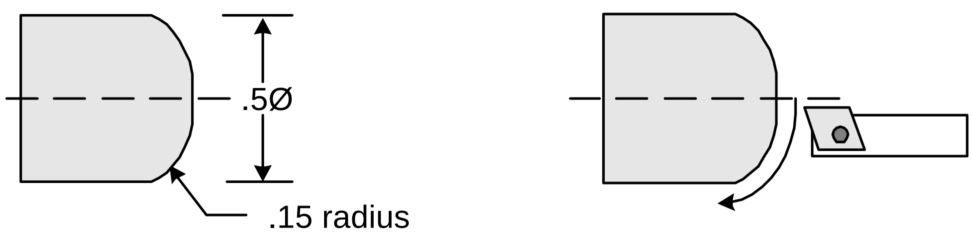
Arc center (In Kn): This is different. With diameter programming the arc center is defined as the incremental distance from the arc start to the center.



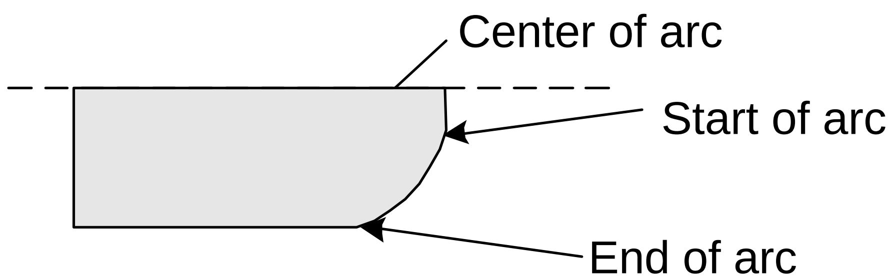
Values of I and K
Incrimental from start

Arc statements G02 and G03

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

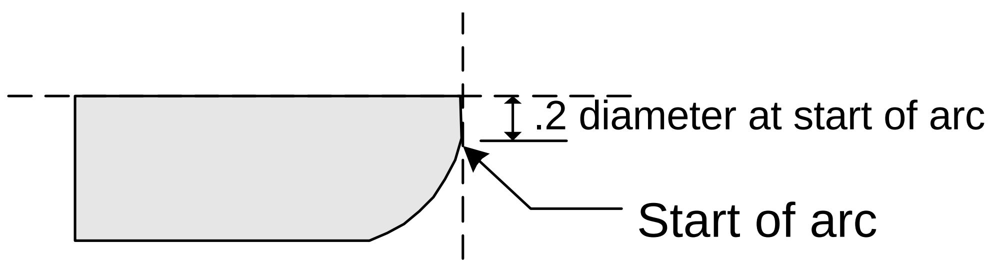


For this example first we show the three important locations that must be defined to write the arc statement:

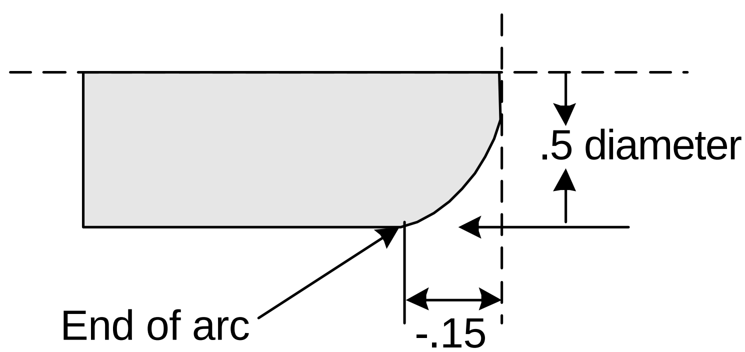


The starting point of the arc is where X=.2 (diameter value) and Z=0. So these values will be used to write the position move before the arc statement:

X.2Z0

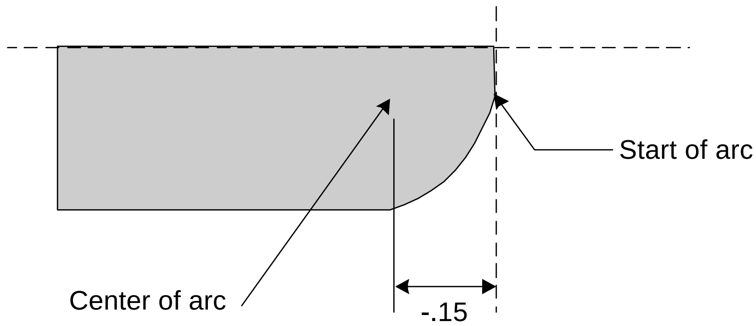


The end of the arc is defined from the absolute zero for the part. As shown on the following figure X= .5 and Z= -.15. These are used for the X and Z values in the G02



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

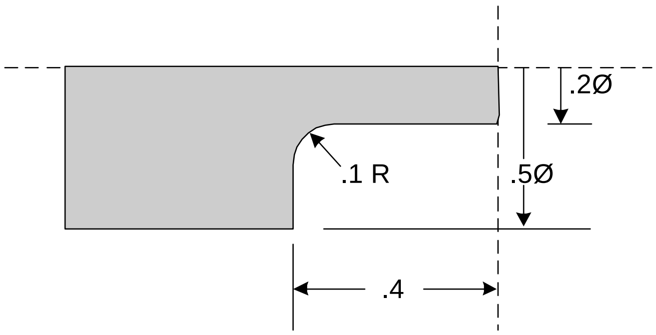
Arc statements G02 and G03

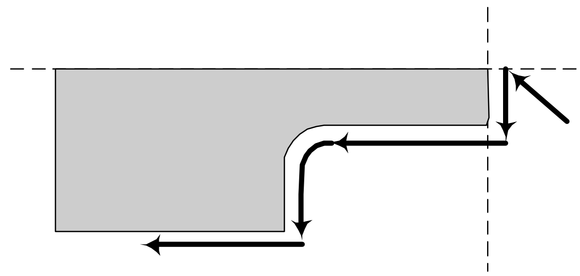


So the program lines could be:

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

For the next example we will show a G03:





X0Z0
X.2
Z-.3
G03X.4Z-.4I.1K0
X.5
Z-.5

Dwell - G04

The dwell statement format is:

G04Fn

Note:

• The 0 in the G04 must be the number 0, not the letter o.

• The "n" after the F is the number of seconds needed to dwell.

• The shortest dwell is .1 seconds

• There is no limit to the length of the dwell. However we suggest you only use it for machining dwells. DO NOT use it for loading and unloading operations in the program. This would be unsafe!

Example

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

G10 Work shift

Work shift is used to offset a program from the original starting point. Typical applications are:
- -Machining multiple parts off a single shootout of a bar.
- -Shifting a program away from the spindle the first time it is run

G10XnZn

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

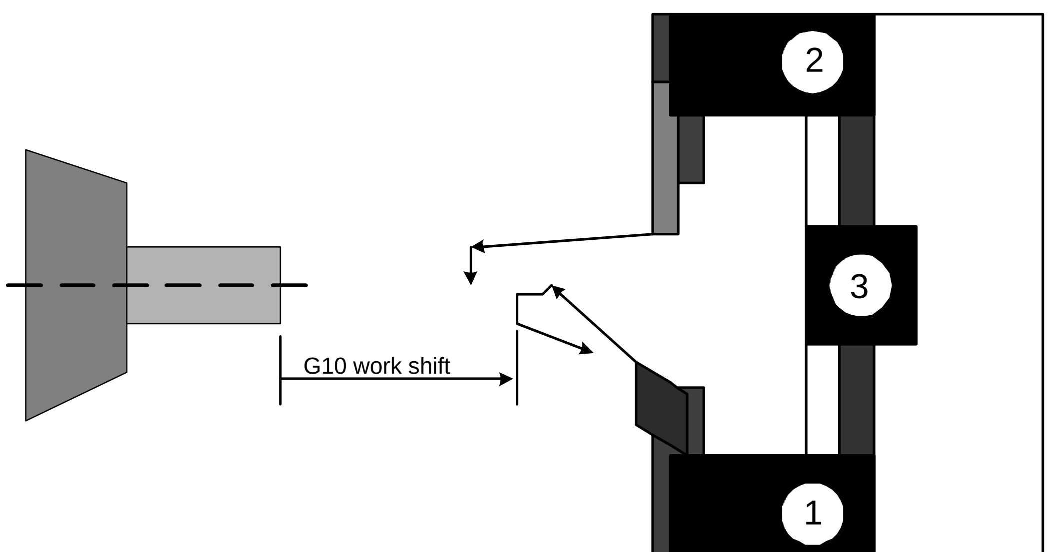
Note: The shift is executed on tool calls! Use the GIO command before tool calls otherwise there will be no effect.

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

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

Example of shifting a program for test running

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



G90G72G94F300
M03S2000
G04F2
G10XOZ3 **Shifts program 3" to the right**
T 1(LH TURN TOOL)
XOZ.2
ZOG95F.01
X.1 Z-.05F.002
Z-.15
X.2
G94F300Z1
T2(PART OFF TOOL)
X0Z1
X-.4
Z-.35

G10 Work shift

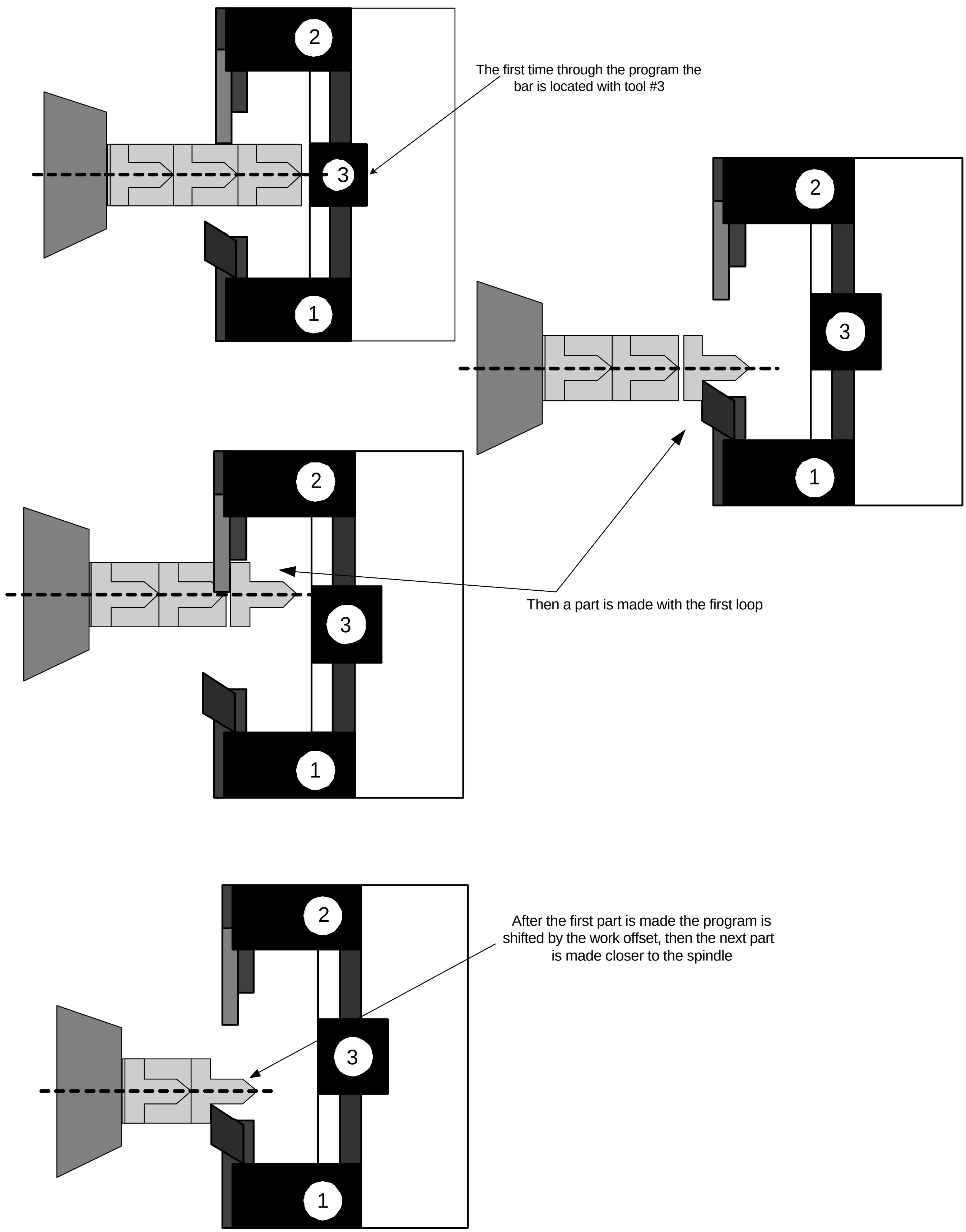
```
G96S150
G76S500
G95F.001 X.005
G94F300Z1
G97
M30
```

Example of shifting a program for bar work

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

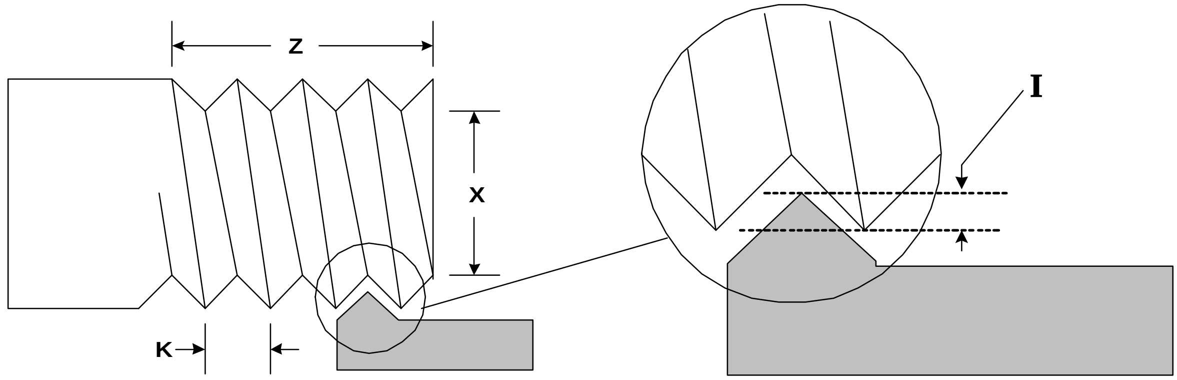
```
G90G72G94F300
M03S2000
T3(WORK STOP)
XOZ1
Z-1
M13
Z.025F50
M12
Z 1 G94F300
LS3                     The loop starts here and will repeat 3 times. Notice it is
T 1(LH TURN TOOL)       before a tool call.
X0Z.2
S2000
Z0G95F.01
X.1 Z-.05F.002
Z-.15
X.2
G94F300Z1
T2(PART OFF TOOL)
X0Z 1
X-.4
Z-.35
G96SI50
G76S500
G95F.001 X.005
G94F300Z1
G97
G10XOZ-.35              Here is the work shift, shifting the next loop
LF                      End of loop
M30
```

G10 Work shift



G33 Threading

The format is: G33XnZnInKnAnCnPO



X The X axis location (as a radius) of the final pass of the cycle in G72 mode this is the final pass as a diameter.
Z The Z axis location of the end of the thread
I The starting incremental amount of material to be removed after the first pass.
This is to be defined as the diameter removal in diameter mode
K The lead of the thread, amount per revolution, .2" max. For larger see G35
A Used for tapered threading, it specifies the amount the X axis will move over the length of a tapered thread
C Causes the infeed to be at an angle, the default is 29°
P Used when you want the tool to keep traveling forward while it pulls out of the work. This will leave no undercut
O Including the letter O makes a single pass at the finished depth

Notes:

Diameter or radius mode

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

Starting position in Z

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

Starting position in X

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

Depth of each pass: I

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

G33 Threading

Retraction position between passes:

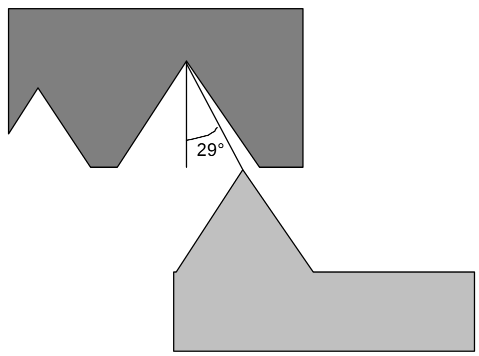
The tool will back away from the starting position plus 3 times the amount of I. Even as the tool gets deeper into the material it will always retract to the same point.

Pullout position in Z when using P option:

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

Angle infeed C option

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



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

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

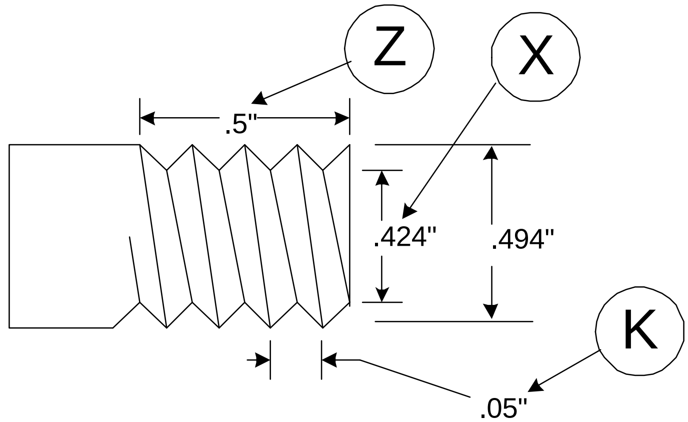
End of cycle position:

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

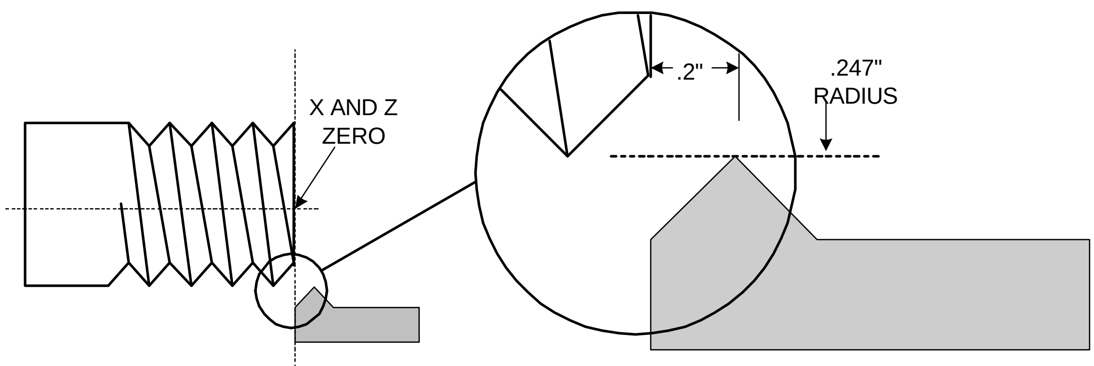
G33 Threading

Threading example

Example: A straight thread, Male, 20 pitch, Minor Ø = .424”, Major Ø = .494”, length of thread = .5”, and there is no undercut.



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



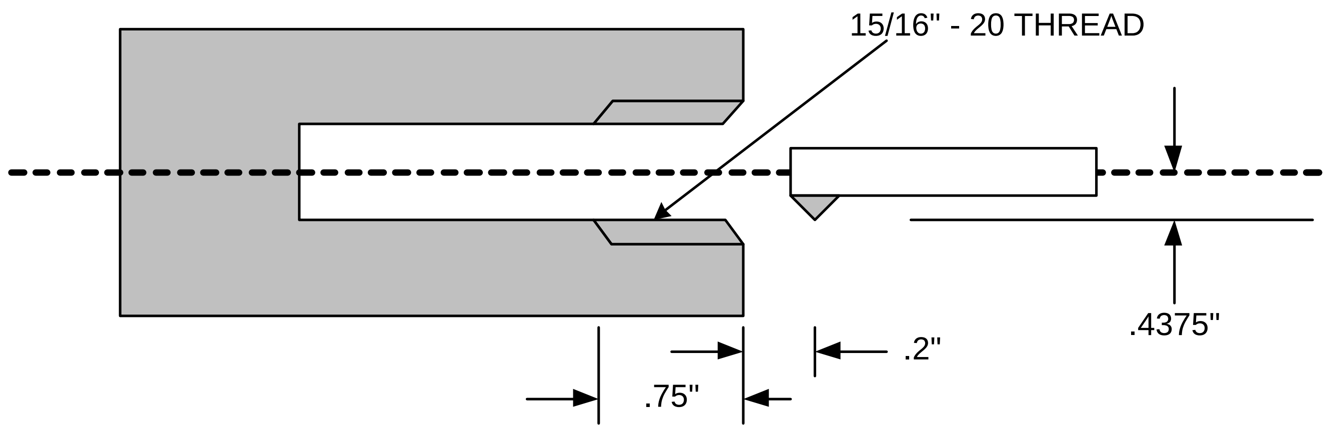
The locations shown above are given in radiuses. The starting location in X is the major diameter, less the amount of the first pass. In this case the major diameter is .494”, half of this is .247”. Less .O1 to be taken for the first pass: .237

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

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

G33 Threading

Internal threading example



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

T8 (Internal threading tool)	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.

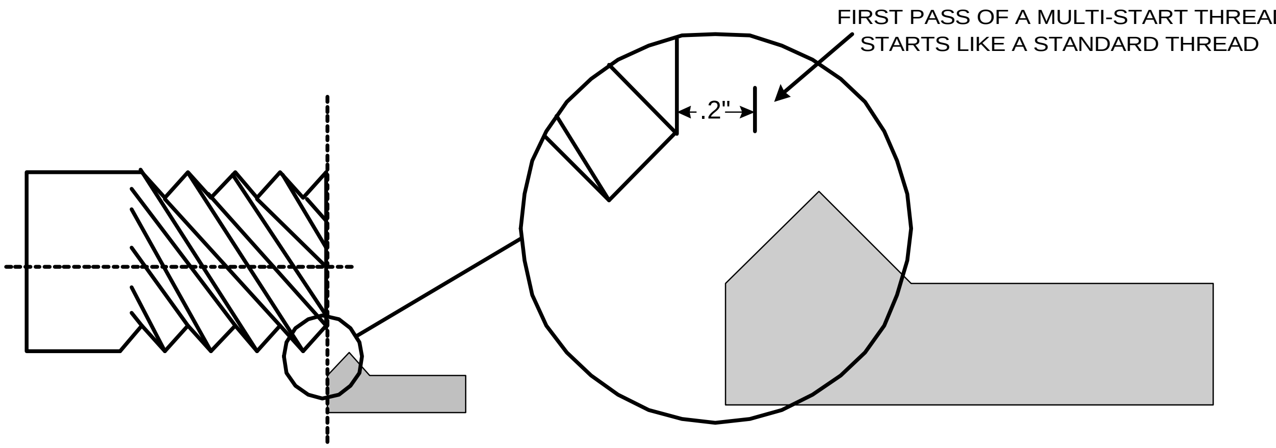
T4F300	Call the threading tool and position it at tool offset
X0Z.2	The offset puts the tool at the part center, .2” from the face in Z
X.237	Positions the tool at the starting radius to cut the thread
G33X.212**Z**-.5**I**.008**K**.05**PCO**	Cut a single pass thread

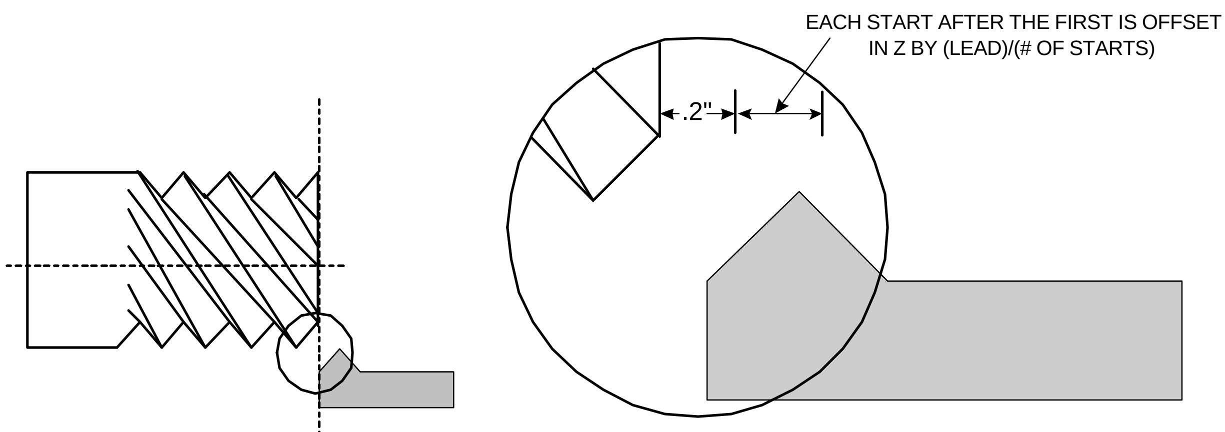
G33 Threading

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, I start value for K is .05". With the three start the value is three times that, .150".





Value of Z:
The starting position of Z is offset for each start by (pitch)/(number of starts). IE. If we have a three stag 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:

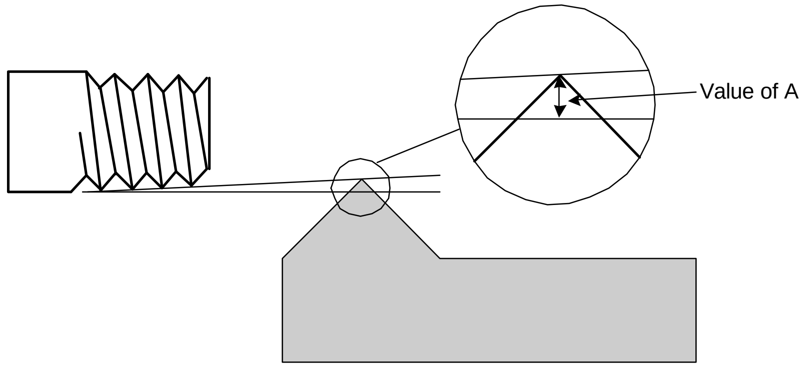
```
G90G94 F300
T4
X0Z.2
X.247
G33X.212Z-.5I.01K.15P
Z.25
G33X.212Z-.5I.01K.15P
Z.3
G33X.212Z-.5I.01K.15P
```

G33 Threading

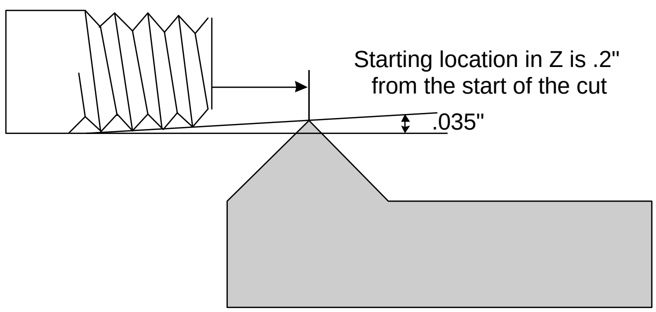
Threading, Tapered

G33XnZnInKnAnP

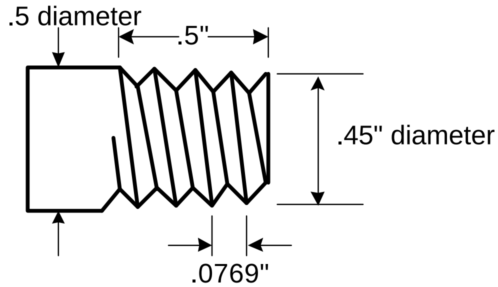
Tapered threading is done with the G33 command that includes an ”A”. This is the amount traveled in X over the distance traveled in Z.



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

G33 Threading

Extra Coarse Feeds (IPR)

G35 -IPR feeds up to 1”

G35F2 -IPR feed up to 2”

Format:
Start Mode -G35 or G35F2 must be on a line by itself. The following line must be a G92XnZn, where XnZn are the current working coordinates.

Cancel coarse mode -G36. The tool must first be returned to the location where the G35 was first used. Then put G36 on a line by itself. The following line must again have a G92XnZn, where XnZn are the current work coordinates.

The normal maximum feedrate in IPR is .2” per revolution of the spindle. The code would be G95F.2. At this feedrate the spindle rpm is limited to 1500 rpm. because the maximum feedrate of the slide is 300 ipm (1500 x .2 = 300).

Feedrates higher than .2” are available in the G35 mode. Multi-start threads can often require long leads. an 8 pitch 4 start acme thread requires a .5” lead for each of the four starts.

G35 alone enables feeds up to 1.000” per revolution
G35F2 enables feeds up to 2.000” per revolution

Resolution limits are imposed with the use of these codes:
- .00005” resolution in normal feed mode
- .00025” resolution in G35 mode
- .0005” resolution in G35F2 mode

Remember also that the 300ipm limit imposes rpm limits on the spindle speed; a 1” lead requires a spindle speed of 300 rpm or less.

Example
In the following example we will show a four start thread with a course pitch:

```
T4(Threading tool)
x0z1
x.5z.2
g35
g92x. 5z.2
g33x.45z-.75i.01k.4
x.5z.3
g33x.45z-.75i01k.4
x.5z.4
g33x.45z-.75i01k.4
x.5z.S
g33x.45z-.75i01k.4
x.5z.2
g36
g92x.5z.2
g00z2
```

TOOL NOSE RADIUS COMPENSATION G41,G42, G40

Tool nose radius compensation

Notes on use:

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

Format

Right Compensation	G42
Left Compensation	G41
Cancel Compensation	G40

Compensation Value Location Xn.nnnnZn.nnnnDn

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

Sequence

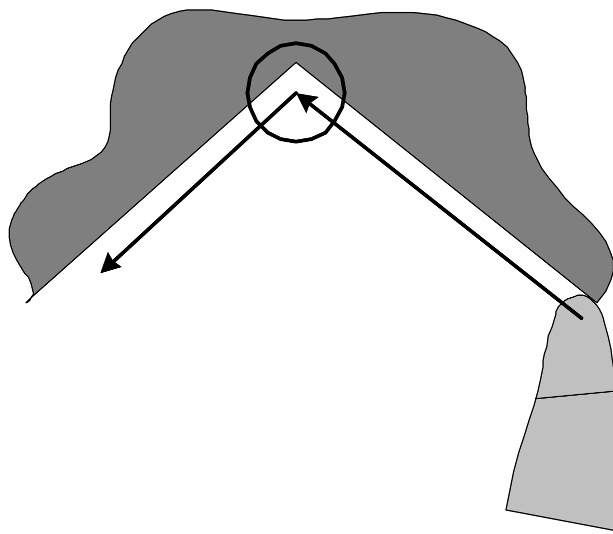
G42	Turn compensation on
XnZnDn	Move with secondary offset radius value used to turn on comp
XnZn	Move used to turn off compensation
G40	Turn compensation off

Rules

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

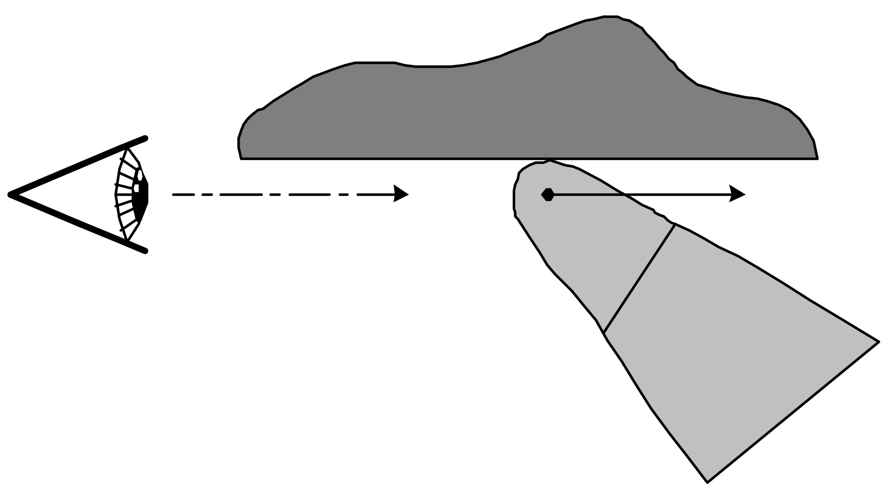
TOOL NOSE RADIUS COMPENSATION G41,G42, G40

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



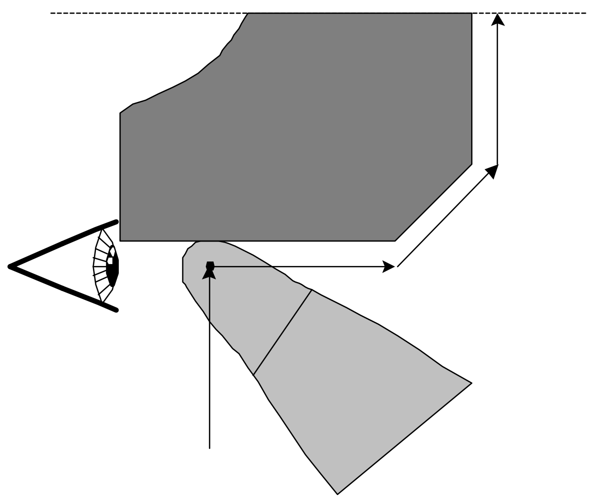
When using the TNR compensation the tool path gets shifted off the finished size. This does not matter if the tool being used to take a finish pass is different than the roughing tool. The tool is shifted in the setup to give a correct finished size. If the same tool is used to do the rough and finish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

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

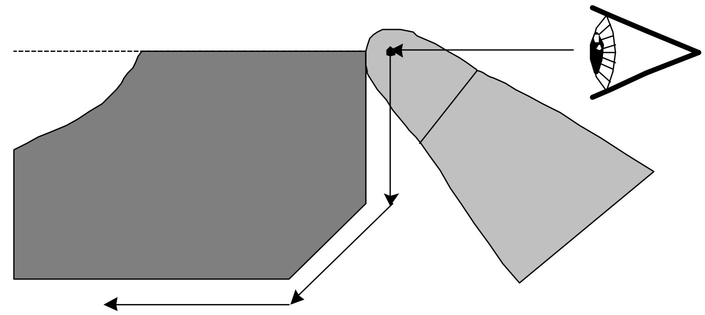


TOOL NOSE RADIUS COMPENSATION G41,G42, G40

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



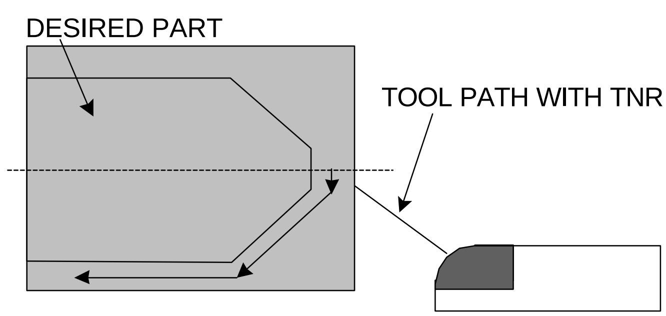
RIGHT HANDED COMPENSATION
G42

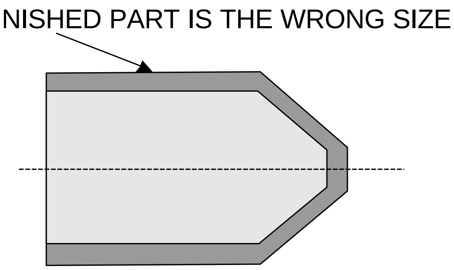


LEFT HANDED COMPENSATION
G41

Shifting the TNR compensation

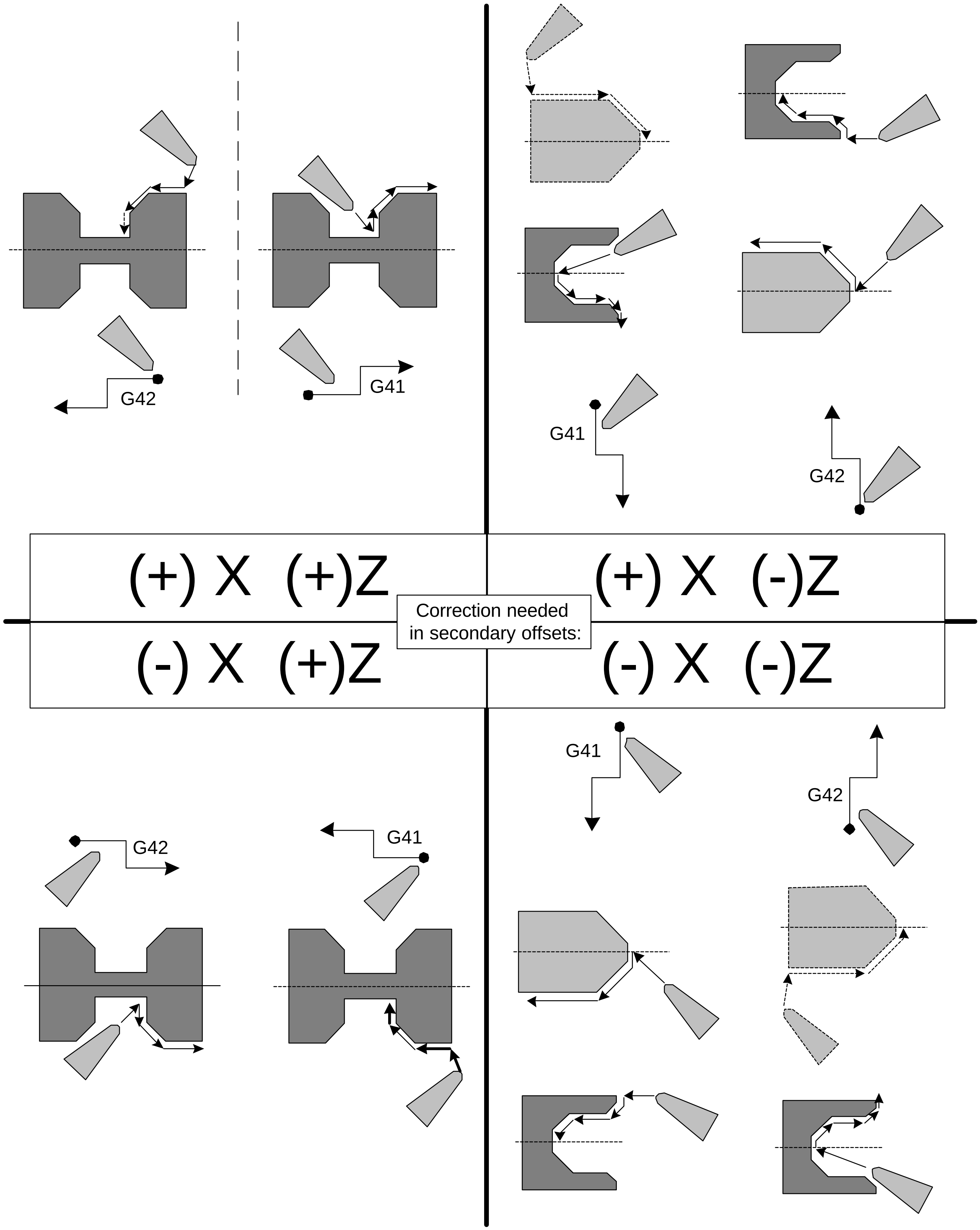
The direction of the correction will depend on the direction of the tool path and desired TNR compensation.





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

TOOL NOSE RADIUS COMPENSATION G41,G42, G40



TOOL NOSE RADIUS COMPENSATION G41,G42, G40

Setting the TNR value:

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

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

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

First: Select a secondary offset number

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

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

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

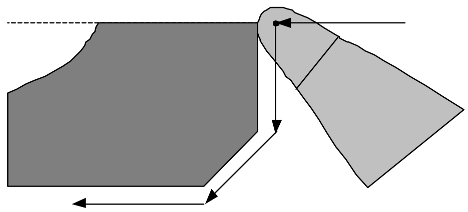
Then: Enter the value of tool nose compensation, IE .007 and then press ESC

Changing a compensation value:

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

Worked examples

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



G90G94F300
M03S2000
T1 (LH turn tool with .015 tnr)
X0Z 1
Z.05
G95F.003

G41	Turn on left hand tool nose radius compensation
X0Z0**DI**	Use the radius value found in secondary offset # 1

X.22
X.25Z-.03
Z-.3
X.27
G94F300Z2

G40	Turn off the TNR compensation on the Z2 move

M30

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

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

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

TOOL NOSE RADIUS COMPENSATION G41,G42, G40

Running a program that used Tool Nose Radius Compensation

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

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

G70 - G71 Inch - Metric Modes

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

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

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

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

G72 - G73 Diameter and Radius Programming

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

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

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

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

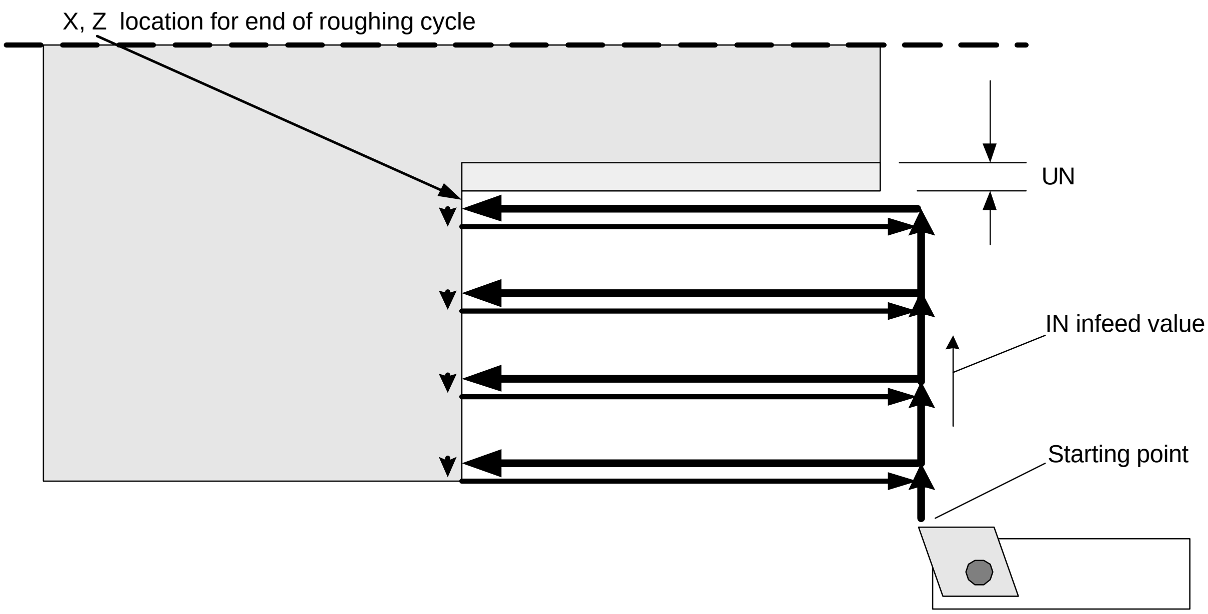
```
T1
G72X.5Z.1
```

G74 - Box Roughing Cycle

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

G74XnZnlnUnFn

X and **Z** is the corner of the box area to be cleared out
In is the maximum amount to be roughed per pass, defined as the depth of cut per side
Un amount of material to be left by the cycle for a finish pass in X only.
(depth of cut, as a radius)
Fn is the feedrate



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

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

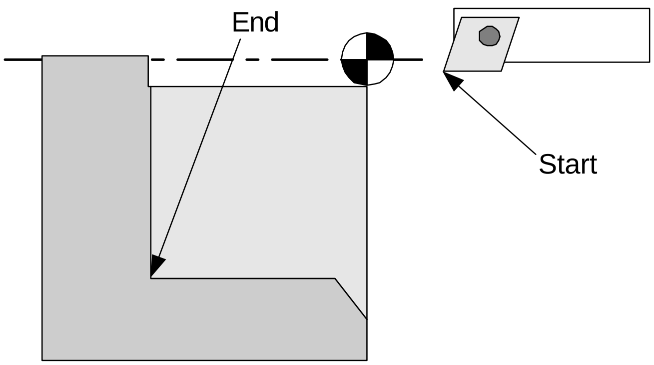
The feedrate is IPM (G94) or IPR (G95), depending on the mode when the cycle is started.

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

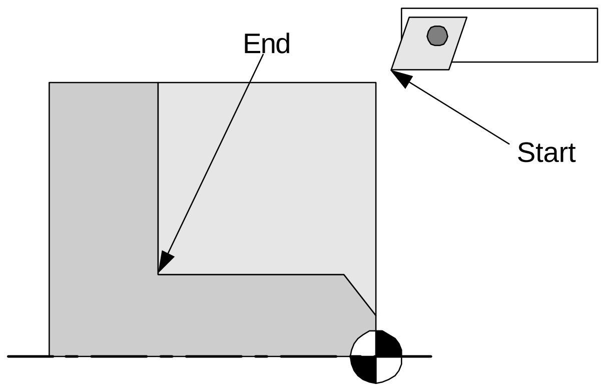
The return passes are at a fixed clearance distance (.02") from the last cutting pass.

G74 - Box Roughing Cycle continued

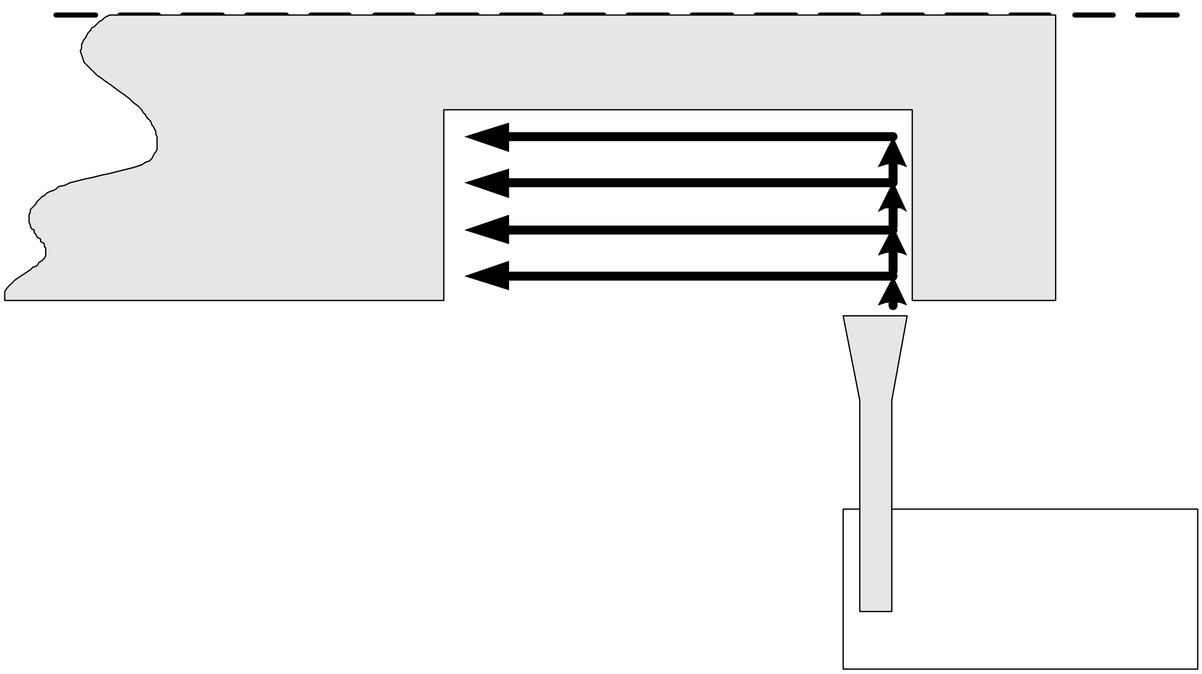
The G74 cycle can be used for either internal or external removal. It can also be used from the front (x+) or back (x-) of the part.



Internal example



External example from the back



Example behind a shoulder

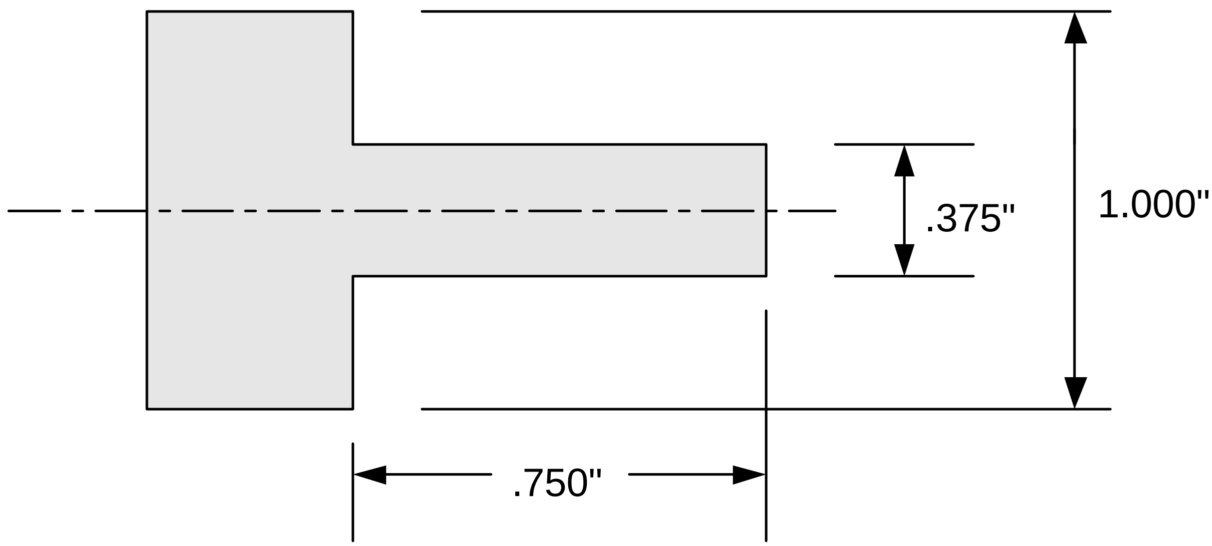
The G74 cycle can be used in radius (G73) or diameter (G72) mode. In both cases the In value is the amount to be taken off on a side. An In value of .05 would take .1 off the diameter of the part with each pass.

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

G74 - Box Roughing Cycle continued

Worked example for G74

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



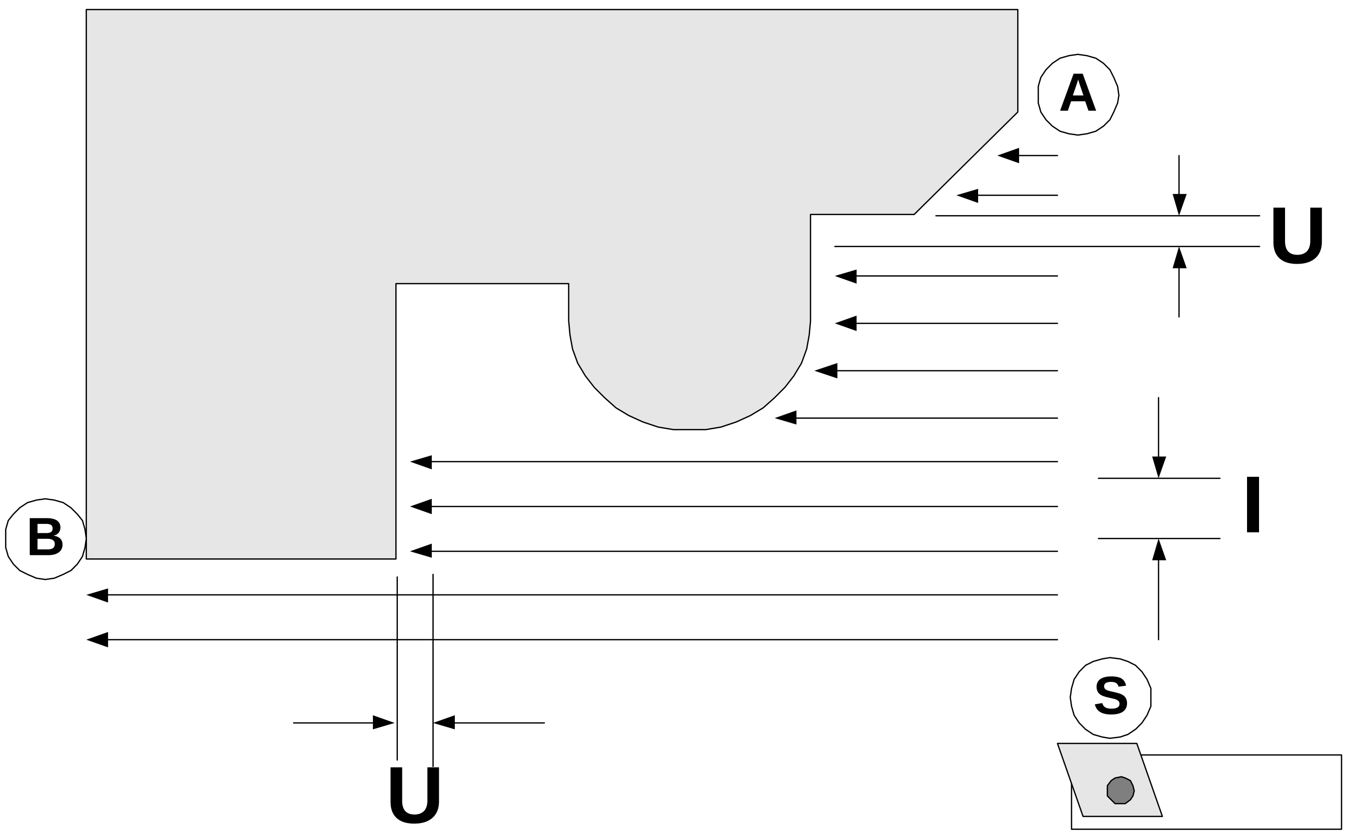
G90G94F30OG72
M03S2000
T1(LH Turn tool for roughing)
X1.05Z.5
Z.1
G95
G74X.375Z-.747I.075U.005F.003 programmed to leave .005 on X and Z for finish cut
M05
G00Z 1
X-1
M30

G75 - Box Contour Roughing Cycle continued

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

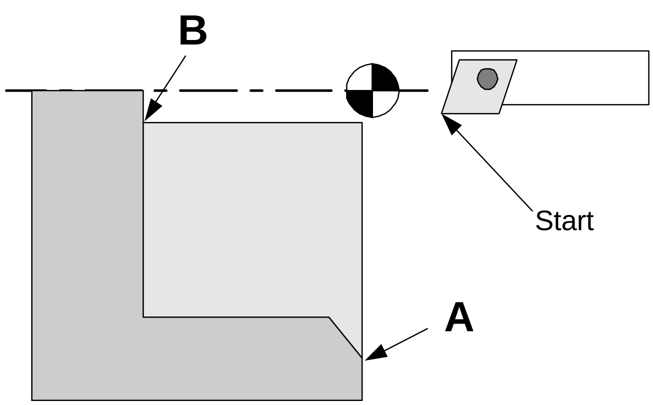
G75 - Box Contour Roughing Cycle continued

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

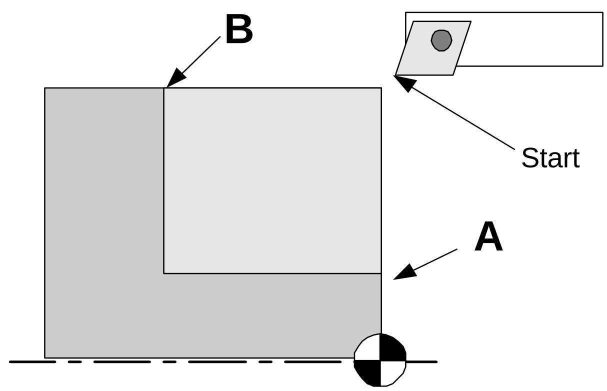
The RF code must be on a line by itself.

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.



Internal example



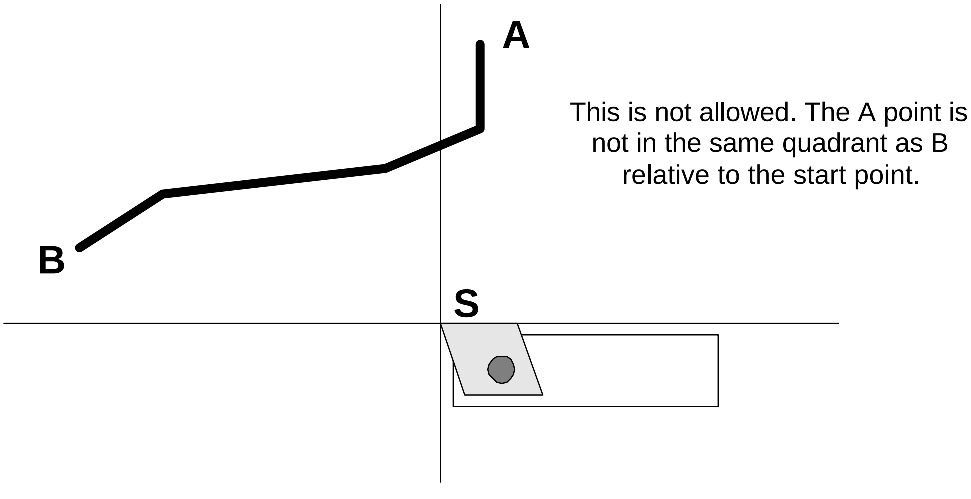
External example from the back

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.

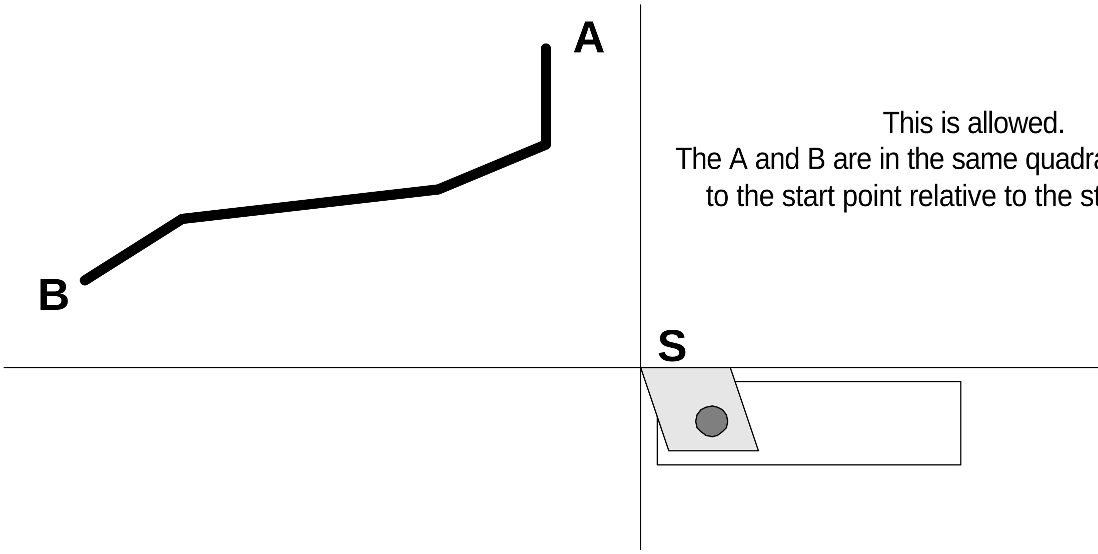
In calculation of the contour, the control honors those codes which affect the geometry of the contour (presets, absolute/incremental, etc.) but ignores any other information which might be present (T & D codes, M codes, feedrates, etc.)

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)

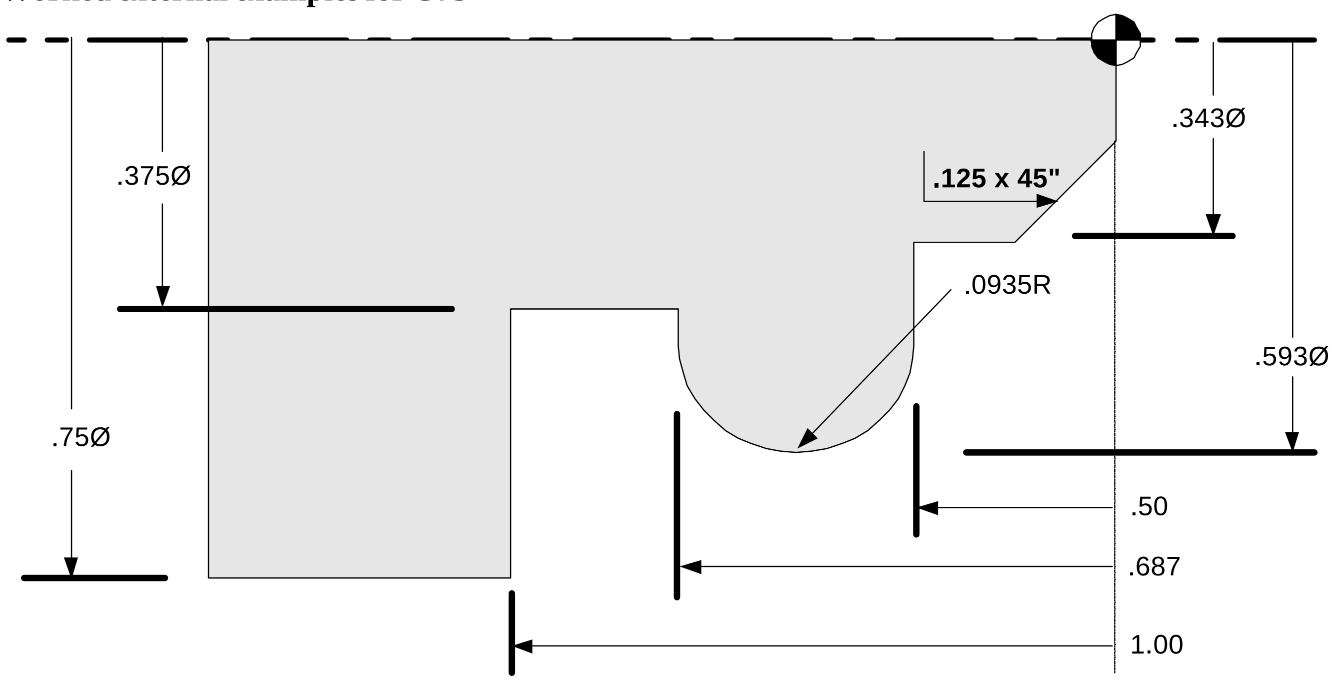


If the start point is moved over to include both A and B the cycle will work.

G75 - Box Contour Roughing Cycle continued



Worked external examples for G75



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

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

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

G75 - Box Contour Roughing Cycle continued

Written in Diameter mode

```
G90G94F300G72
M03S2500
T 1(LH TURN TOOL)
X0Z1
X.8Z.1
G95F.003
G751.05U.02F.003
X0Z0
X.343C.125
Z-.5
X.406
G02X.406Z-.687R.0935
X.375
Z-1
X.75
RF
M05
G00Z 1
X-1
M30
```

Written in Radius mode

```
G90G94F300G73
M03S2500
T 1(LH TURN TOOL)
X0Z1
X.4Z.1
G95F.003
G751.05U.02F.003
X0Z0
X.1723C.125
Z-.5
X.203
G02X.203Z-.687R.0935
X.1875
Z-1
X.375
RF
M05
G00Z 1
X-. 5
M30
```

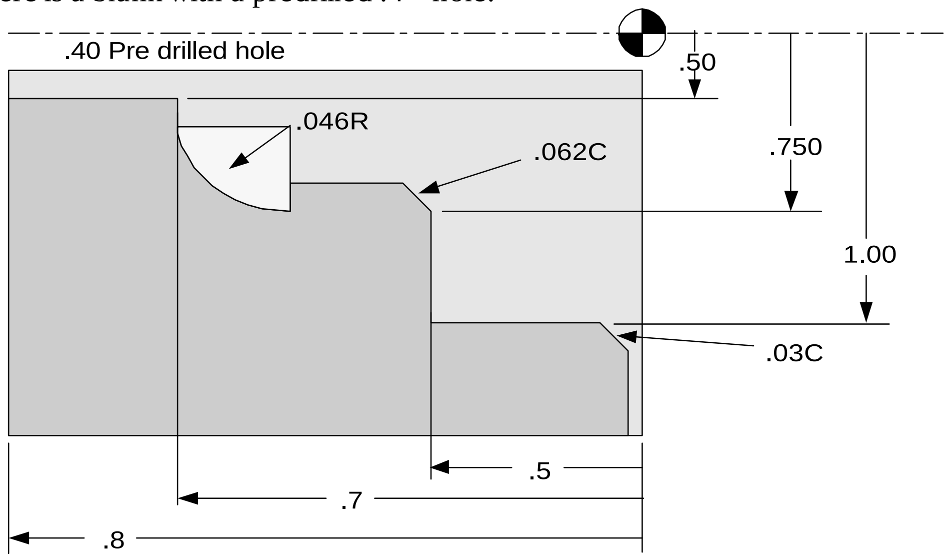
In this example the contour code is given as a subroutine. You can use the same subroutine for the G75 and G78 roughing passes. If you have to make a correction to the contour, it can be done in one place instead of for each cycle. You can also use it for the finishing pass if you are not going to use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

```
G90G94F300G72
M03S2500
T 1(LH TURN TOOL)
X0Z 1
X.4Z.1
G95F.003
G751.05U.02F.003P1
M05
G00Z1
X-1
M30
}1
X0Z0
X.343C.125
Z-.5
X.406
G02X.406Z-.687R.0935
X.375
Z-1
X.75
M99
```

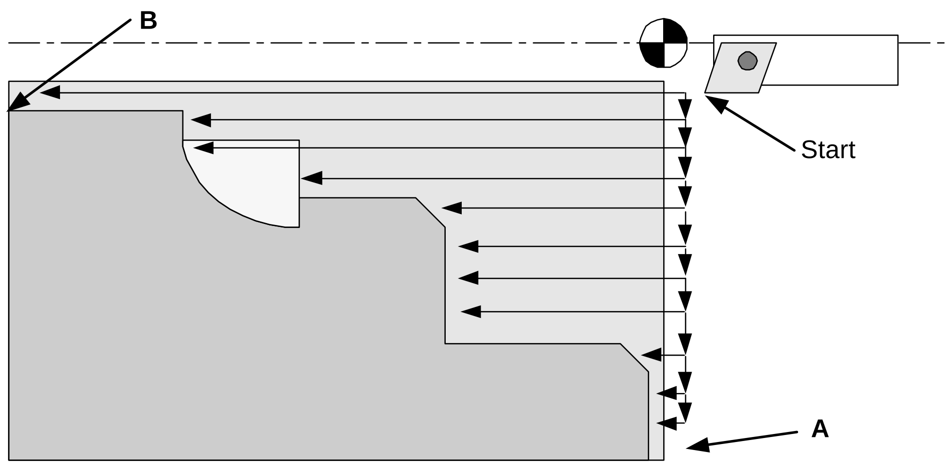
G75 - Box Contour Roughing Cycle continued

Worked internal example for G75

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



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



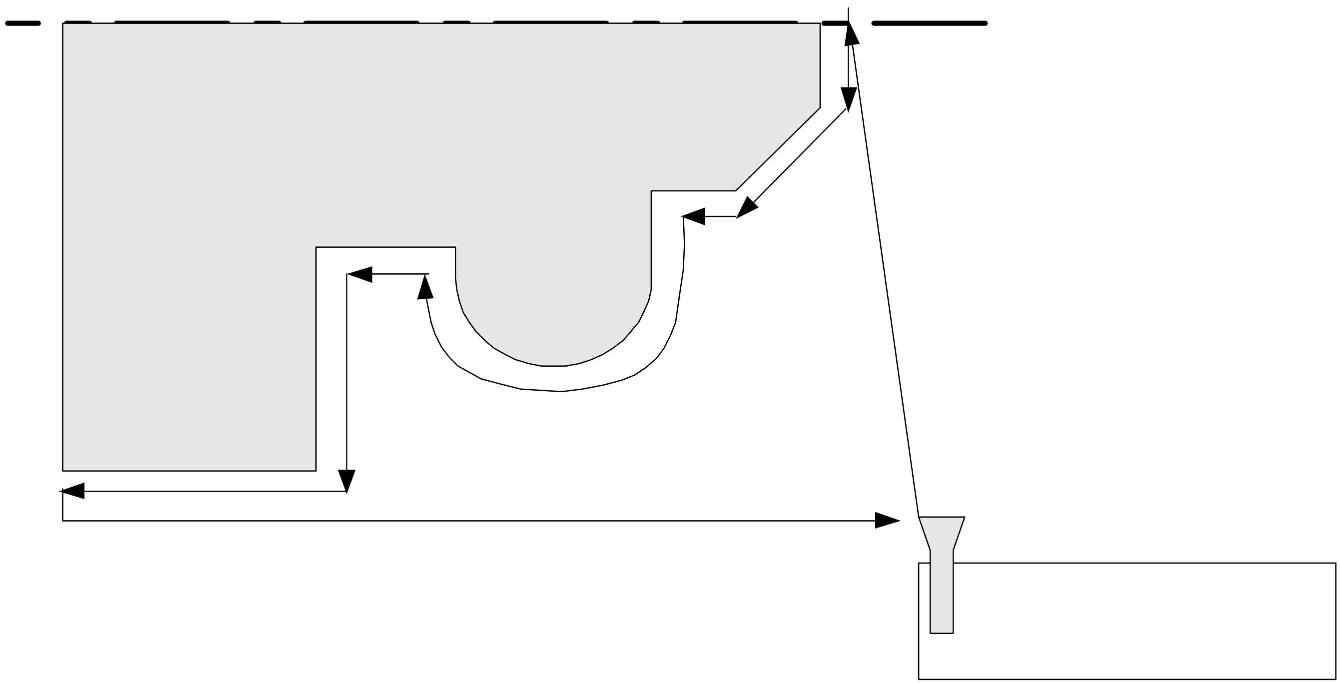

```
G90G94F300G72
M03S2000
T1
X.4Z.1
G95F.003
G751.05U.02F.003
X 1.2Z0
X1.06
X 1 Z-.03
Z-.5
X.75C.062
Z-.704
G03X.658Z-.75R.046
X.5
Z-1.
RF
G00Z 1
M30
```

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

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

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

Worked example for G78

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

G90G94F300G73
M03S2500
T2(LH Finish TURN TOOL)
X.8Z.1
G95F.003
G78U.01F.003
X0Z0
X.343C.125
Z-.5
X.406
G02X.406Z-.687R.0935
X.375
Z-1
X.75
RF
G00Z 1
M30

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

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

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

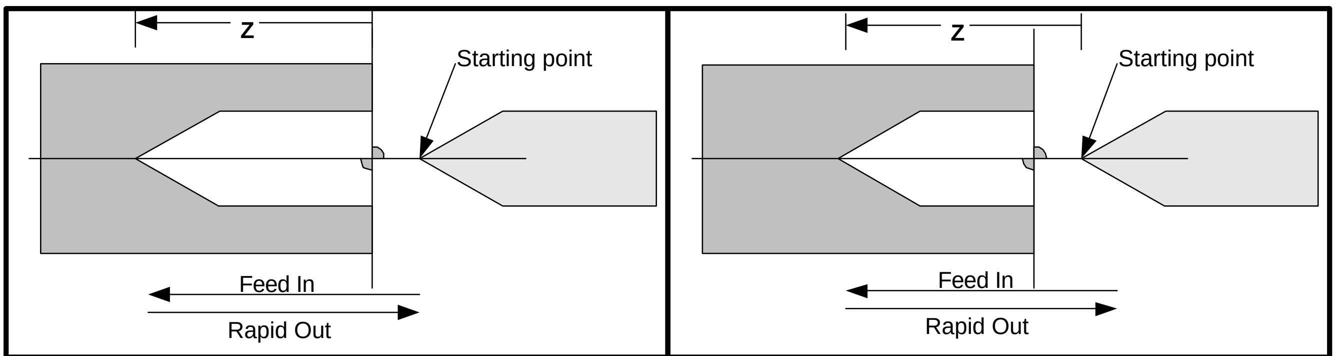
.....continued on next page .....

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

G81 Drill Cycle

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

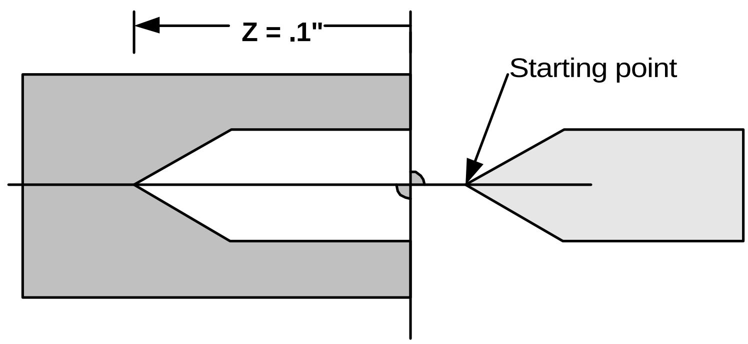
G81 Zn Fn



G81 used in absolute mode, G90	G81 used in incrimental mode, G91

In G90, absolute mode: **Z** specifies the end of the hole from absolute Zero.
In G91, incremental mode: **Z** specifies the distance the tool will travel from the starting point.
F is the drilling feedrate in inches per rev or minute depending on if you are in G94 or G95.

G81 Example



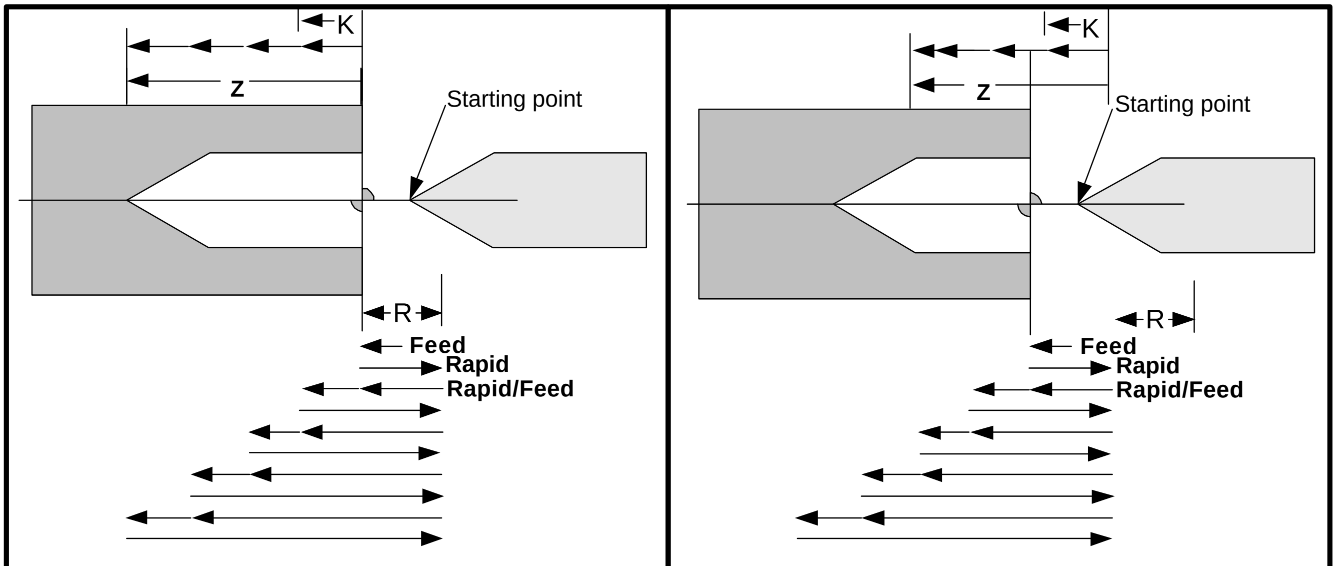
To drill a part .l” deep at a feed of .005” per revolution. The program would be:

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

G83 Peck Drill Cycle

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

G83 Zn Kn Fn Rn Ln Cn



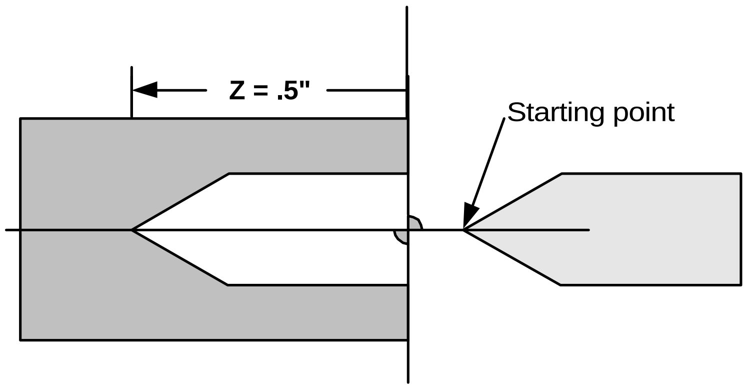
G83 used in absolute mode, G90	G83 used in incrimental mode, G91

In G90, absolute mode: **Z** specifies the end of the point of the hole from the part Zero.
In G91, incremental mode: **Z** specifies the distance the tool will travel from the starting point.

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

K specifies the depth of cut per peck.
F is the drilling feedrate in inches per rev or minute depending on whether you are in G94 or G95.
R is the retraction plane, the tool will rapid back to this location at the end of each peck. **Default is the starting point of the cycle**
L is the rapid travel feedrate for the retraction move, noted in **IPM. Default is 200ipm**
C is the clearance distance left when the drill returns to the cut. **Default is .02"**

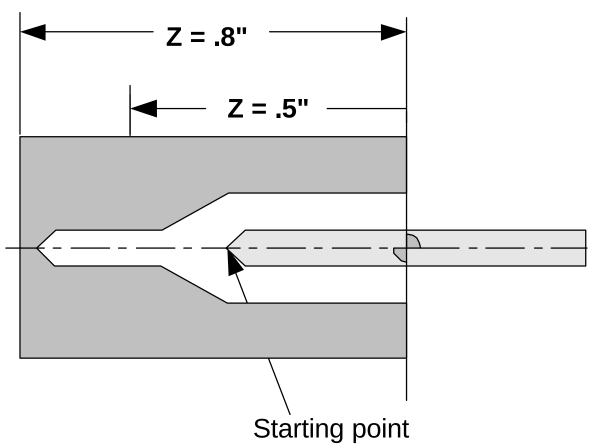
G83 Peck Drill Cycle



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

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

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



G90 G94F300	
T1	First drill (large one)
X0Z.01	
G95	
G83Z-.5K.2F.003R.5C.05L300	
G00Z1	
T2	Second drill (small one)
X0Z 1	
Z-.4	
G83Z-.8K. 1 F.002R.5C.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: distances given will move the tool relative to an absolute zero.

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

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

G92 is a position preset.

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

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

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

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

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

G94 and G95 Feedrate selections

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

G94 -feed set at inches per minute
Slowest: .00005” per minute
Fastest: 300” per minute

G95 -feed set at inches per revolution
Slowest: .00005” per revolution
Fastest: depends on spindle speed, the limit has to be calculated to 300 ipm
As an example, the fastest feedrate at 3000 rpm is. l ”/rev

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

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

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

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

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

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

Maximum spindle speed -G77: Sets the Maximum spindle speed, G77Sn
Notes: The default spindle speed mode is G97, RPM mode.
• M03, M04, and M05 operate the same for both modes of spindle control

Important Note

Notes on use:
The constant surface speed control is not intended to be turned on at the beginning of the program and then left on. If you do this the spindle speeds will vary greatly every time the machine moves! This will create excessive ware 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:

$$SFM = \frac{(RPM)(2)(3.14)(\text{distance from center})}{12} \qquad RPM = \frac{SFM \times 12}{2(3.14)(\text{distance from center})}$$

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 Model. That is, once turned on they stay active until turned off.

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

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

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

M00 Program Stop

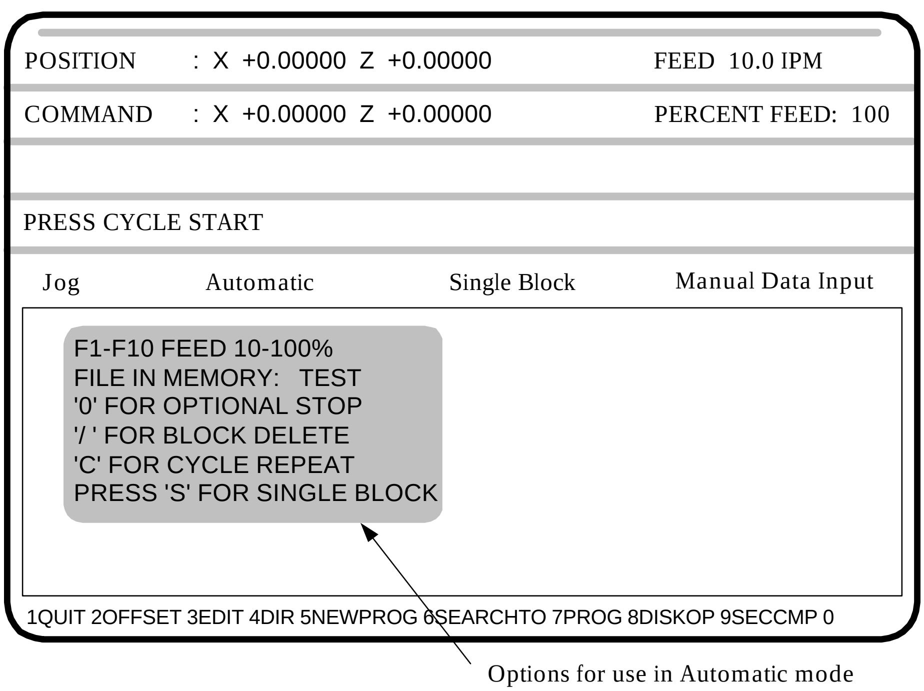
This program stop is not optional. When the control encounters the M00 it will stop and wait for the operator to press "CYCLE START". As an example: This command can be used to stop the program when the slide has a stop in position to allow an operator to push a piece of bar stock to the stop. Once the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

M01 Program Stop - Optional

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

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

-Have a 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.



M03, M04, M05 -Spindle control
These commands will control the direction of the spindle. These commands require optional equipment. There are a number of different types of spindle drives, the action of M03, M04, & M05 commands will vary depending on the type of drive installed.

In general these commands are:

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

OmniTurn with no spindle encoder: The M03 and M04 will stop the program and instruct the operator "WAITING FOR SPINDLE". If the control has an encoder for threading it will know that the spindle is turning. So once the spindle is started the control will start. The control will not know if the spindle is turning the correct direction. If the control does not have an encoder then the control will stop at an M03 or M04 command and instruct the operator "WAITING FOR SPINDLE". To continue past this line press "ESC". Once this has been done the control will pass all M03 and M04 commands without stopping.

OmniTurn with AC spindle control (standard on GT-75)
M03 and M04 will turn the spindle on (in the appropriate direction). With a speed given by the S command. The default mode is RPM. For CSF see G96. The correct format is:

M03S2000 This turns the spindle on forward, at 2000 RPM

M08, M09 -Coolant control
These commands require optional equipment. The M08 will turn the coolant on, M09 turns it off. Additional information about wiring in this command to your machine can be found in the "TECHNICAL" section.

M12, M13 -Collet clamp control
These commands require optional equipment. Additional information about wiring in this command to your machine can be found in the "TECHNICAL" section.

M12 clamps the collet
M13 unclamps the collet

There are many different styles of collet closer. OmniTurn has included a number of different ways to control the closers. Please contact the factory to find out how to select the correct sequence of operations for your type of collet closer. (see section on setting PRM.SER)

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

M31 - Cancel Cycle Repeat
If you want a subroutine to stop the program, as in an end of bar subroutine, for example, you must cancel the Cycle Repeat mode before the M30 command.

Program commands -"M-codes" continued

M89 - Stop the spindle and lock it (optional: C-Axis only)
This code is used to quickly stop the spindle to put a hole or a slot in an arbitrary C-Axis location. It is quicker because the spindle does not go through it's "homing" routine before locking, as it does with M19.

M91, M92, M93, M94 - Wait for input (optional: C-Axis only)

These M-codes stop the program until an input ins "on" or "off". This is useful for coordinating activity for an auto-loader primarily. 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 TB2 in the spindle cabinet. (see page 6-22 for spindle panel layout).
The commands are as follows:

M91 Wait for TB2-5 to be open circuit

M92 Wait for TB2-5 to be short to 0VDC

M93 Wait for TB2-7 to be open circuit

M94 Wait for TB2-7 + to be short to 0VDC

M95 - Conditional jump to subroutine (optional: C-Axis only)
This command will cause the program to jump to subroutine 1 if input 7 is "on" (shorted to 0VDC).
Input 7 is located at TB2-9 in spindle cabinet (see page 6-22 for spindle panel layout). **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

M97 - Conditional jump to subroutine (optional: 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.
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.

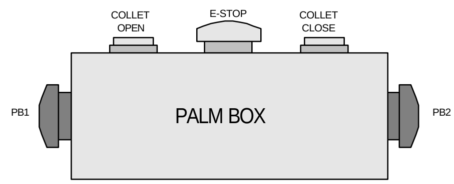
}n - Begin subroutine n
The first line in any subroutine must be the brace } followed by the subroutine number. No other text on that line.

M99 - End subroutine
The last line in any subroutine. The ***next*** line which will execute will be the line i***mmediately after*** the line that called the subroutine.

Spindle Control Option

Cycle Start

The cycle start PB on the face of the OmniTurn control is deactivated when a spindle drive control is installed. In its place an Operator Station is supplied:



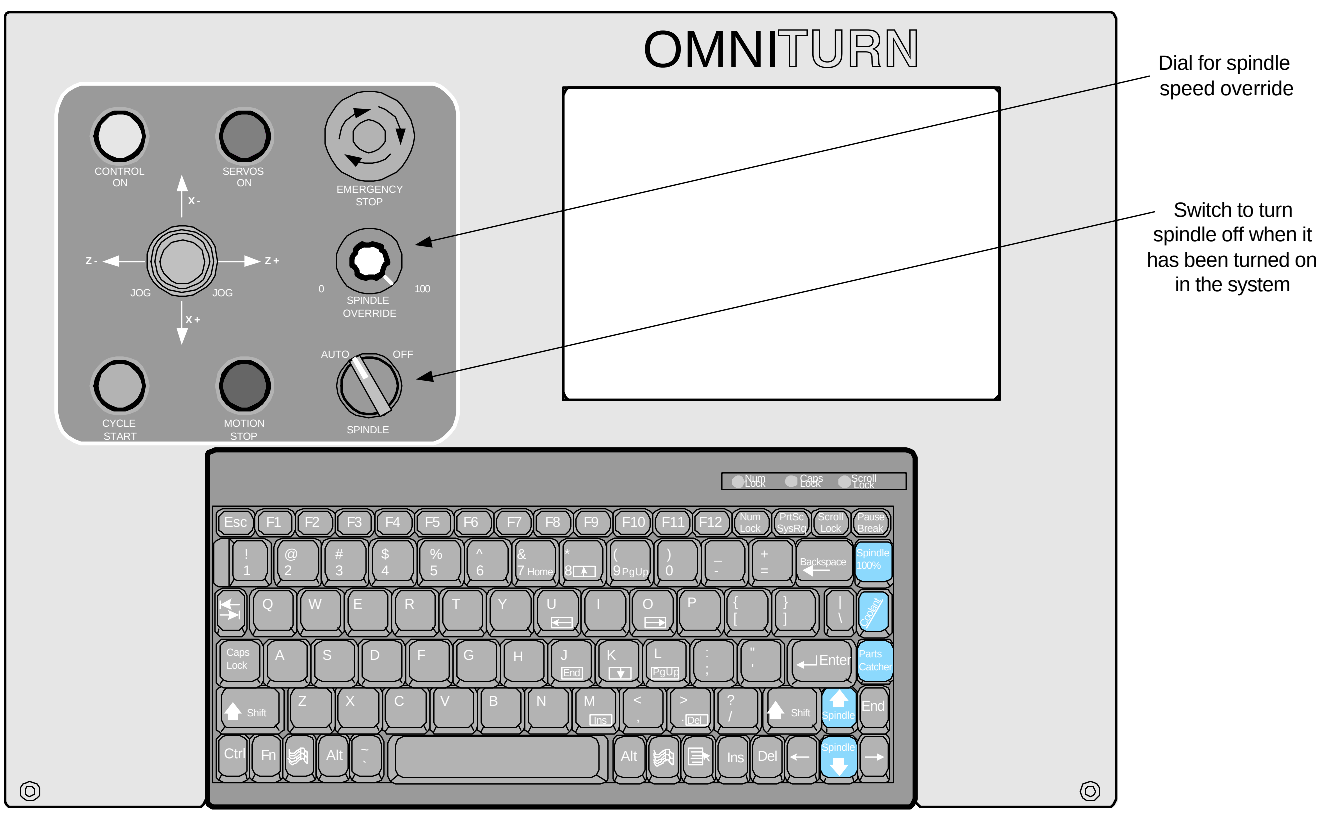
This station has two palm buttons on the sides of the box -PB 1 and PB2. Push both buttons together for cycle start.

Controlling the spindle (not C axis machines):

Manual control of the spindle

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

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



OmniTurn front panel Keyboard

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 M02 and M08.

Setting a spindle speed:

S is used to set a spindle speed. This can be either RPM or SFM. Please refer to chapter 2 -G96 for instructions on using SFM.

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

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

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

Tapping with the spindle control:

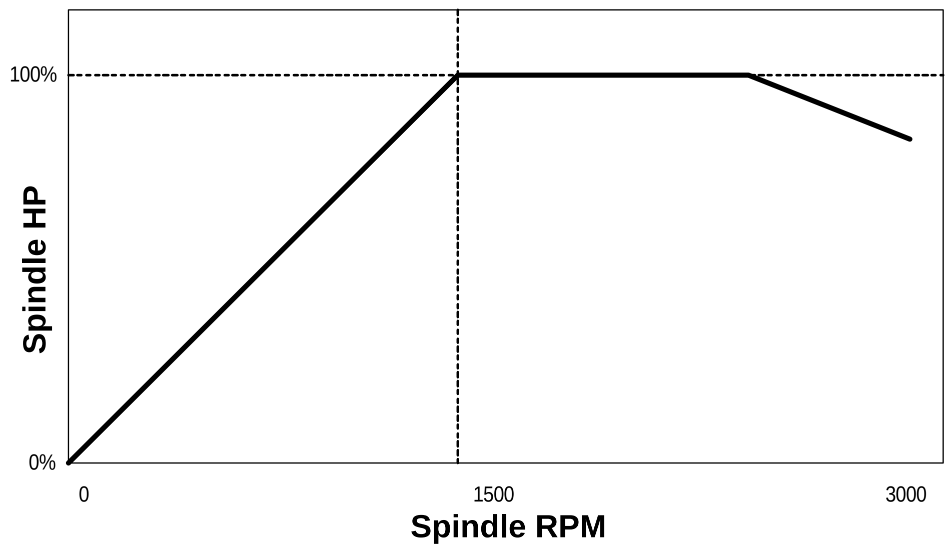
It is possible to tap with the spindle control option. You will need either a self releasing or pitch compensating tap holder. The following is a short program that will tap at 20 pitch approx .5" deep:

```
G90G94F300
M03S1000          TURN SPINDLE ON AT 1000 RPM
T4(1/4-20 TAP)    CALL TAP INTO POSITION
G92X0Z1
Z.1               RAPID TO FACE OF PART
G95F.05Z-.4       FEED IN AT PITCH, STOP SHORT OF FULL DEPTH
M04               REVERSE THE SPINDLE
G04F.5                 **OPTIONAL DWELL TO ALLOW SPINDLE REVERSAL
Z.1               FEED TAP OUT OF THE PART AT CORRECT PITCH
G94F300Z1
M30
```

** This dwell is used to give the spindle some time to reverse before starting the tap back out. If the spindle reversal takes longer than you allow then the tap will start to pull out before the spindle is going reverse and not good things will happen, *snap*. 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.

Setting the spindle speed range (for attachments only, GT-75 is done by the factory):

With the spindle drive it is possible to setup different maximum speeds to make available more HP at lower spindle speeds. The inverter drives that we use to vary the speed are constant torque at all RPMs. However the HP is lower at the lower RPMs. See chart below:



So if you need to take a heavy cut a low RPM you can set the machine to a lower range. When you are done with the low rpm work you can then easily set it back to the high speed range.

Changing the maximum speed:

HSL: With this machine there is the three speed pully:
Use the middle speed set for the High speed range (3400 rpm max)
Use the lower speed setup for 1750 rpm max
Note: the high speed set is not used!

DSM or DV:

The variable speed cone drive belt system on the Hardinge can be used to make an easily set high and low range. Set the spindle speed control on the machine so that when you use the speed variation controls to go up it will stop when it reaches 1500 rpm. This can be done by adjusting the stop on the threaded rod on the vari-shiv. The speed shown on the Hardinge control will show half of what the inverter drive will put out. ie if you set the speed for 1500, you will get 3000 rpm at the spindle. If you set 500, you will get 1000 rpm. And set the low speed to about 500 (or whatever you want as the low range).

Setting the control:

For one time use go to the Automatic mode and use F10 -special functions. This will allow you to set the Max speed. As an example: if you want to drill a large hole in stainless steel and you want to be at 350 rpm for the drilling operations. You could set the spindle drive on the Hardinge to read 500rpm. This would give you 1000 rpm max. Go to special functions -F10, follow the instructions for setting max spindle speed and enter 1000. This will now enable you to enter a speed in the program (350) and get that speed. The Max speed will revert back to 3000 when you restart the Omnitum

Setting the Max speed permanently:

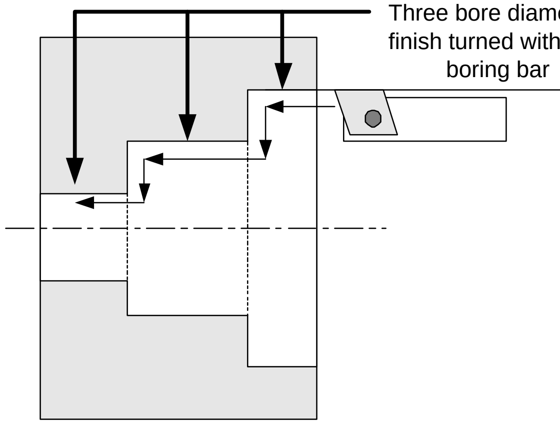
This is done by changing the PRM.SER file. To do this look in chapter 7 -DOS notes for a procedure.

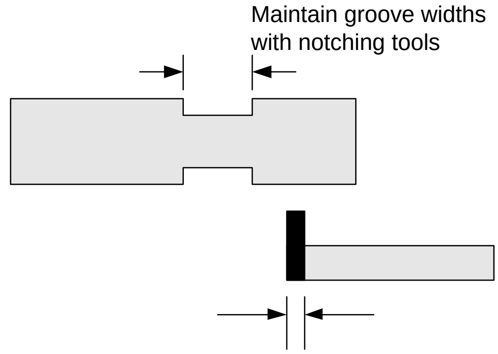
Secondary Offsets

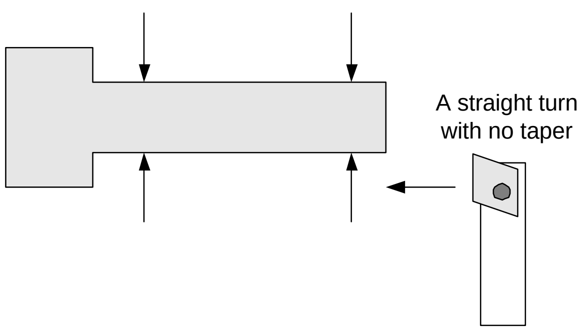
What are secondary offsets?

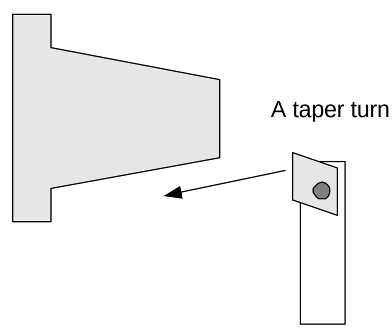
Secondary offsets are corrections that you can put into your program that the operator can adjust when running the program without having to go into the program to edit it. Once the program has been written with the secondary offsets incorporated, these corrections are made by pressing F9 while in the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

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









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

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

NOTE: Clear secondary offsets before using them!

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

Secondary Offsets

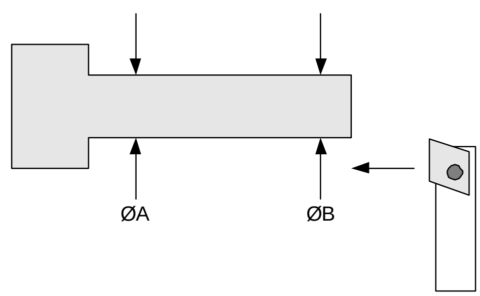
Using Secondary Offsets

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

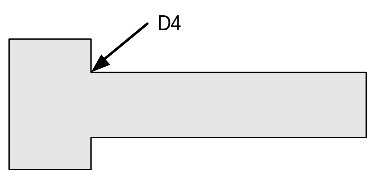
1	X: +0.00000	Z: +0.00000	R: 0.00000	17	X: +0.00000	Z: +0.00000	R: 0.00000
2	X: +0.00000	Z: +0.00000	R: 0.00000	18	X: +0.00000	Z: +0.00000	R: 0.00000
3	X: +0.00000	Z: +0.00000	R: 0.00000	19	X: +0.00000	Z: +0.00000	R: 0.00000
4	X: +0.00000	Z: +0.00000	R: 0.00000	20	X: +0.00000	Z: +0.00000	R: 0.00000
5	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:



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



The code to face and turn the shaft is:

Without correction	With correction	
X0Z0	X0Z0	move to the center and face

Secondary Offsets continued

X. 125	X. 125	move to the diameter
Z-.75	Z-.75**D4**	turn the diameter, corrected
X.2	X.2**D0**	move to the major diameter, turn offset off

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

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

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

X.1245Z.75

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

Canceling a secondary offset

The secondary offset correction stays in effect until:
- There is a tool change. This cancels an offset

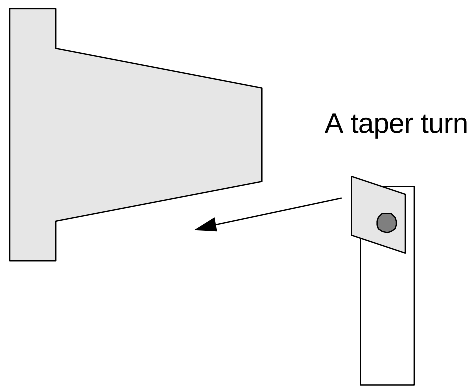
- A secondary offset DO will turn off the offset.

- Calling up another offset will cancel the original offset and enact the new offset

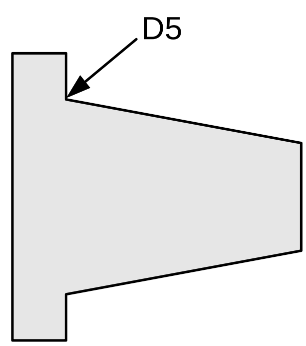
Secondary Offsets continued

Secondary offset examples:

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



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



The commands might look like this:

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

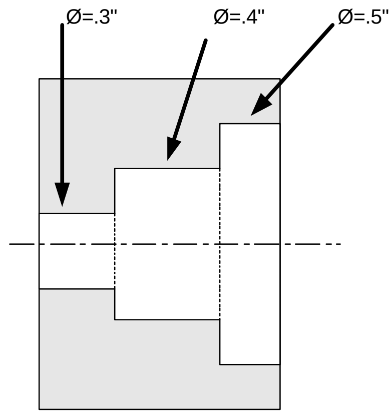
In this case D5 can be used to adjust two features:

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

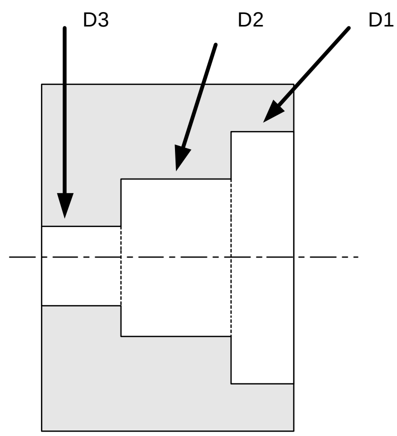
Neither, either, or both values can be entered. Each will effect the angle generated.

Secondary Offsets continued

Here is another example:



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



The coding for this is:

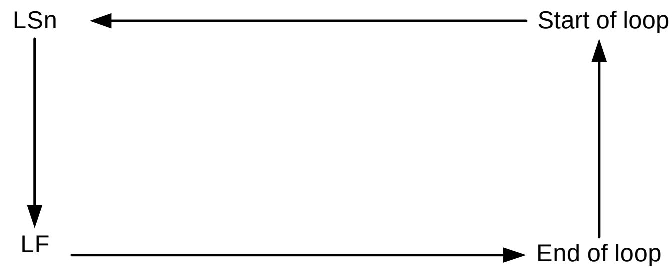
X-.35Z0	Move to the face of the part at -.7 diameter
X.25**D1**	Move to .5” diameter, corrected
Z-.2	Turn the first bore
X.2**D2**	Move to the second bore diameter, corrected
Z-.5	Bore the second diameter
X.15**D3**	Move to the last bore diameter, corrected
Z-.7	Bore the last diameter

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

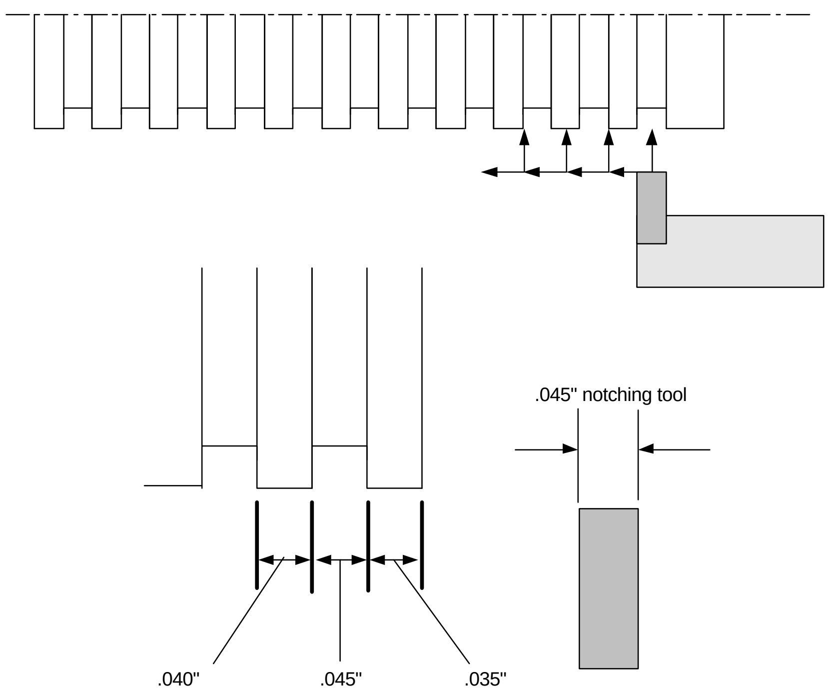
Looping

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

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



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

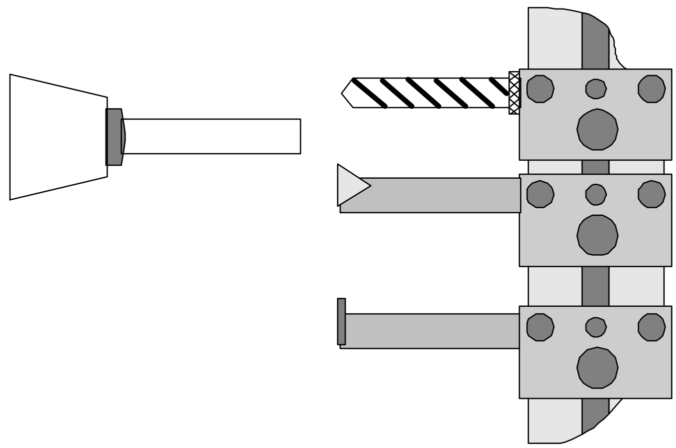


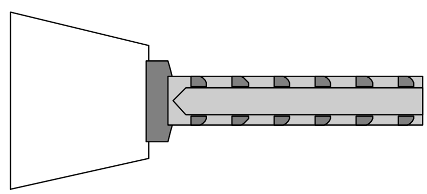
For this example the could be:
G90G94F300
T 1 (notch tool 045 wide)
X.25Z.2
Z-.035
G91-----------------------NOTE THIS LOOP IS DONE IN INCRIMENTAL
LS16
G95F.001 X-.1
G04F.05
G00X.1
Z-.085
LF
G90-----------------------BACK TO ABSOLUTE MODE
Zl
M30

Looping

Looping with Work Shift (G10)

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





```
G90G94F300G72
M03S2000
M08
T 1 (DRILL)
X0Z.5
Z. 1
G95F.003
G83Z-1 K.3L300C.200
G00Z1
LS6
T2 (TURN TOOL)
X.3Z.5
Z. 1
G95F.003Z0
X.35Z-.025
Z-.15
G00X.36
Z1
T3 (PART OFF TOOL)
X.4Z.5
Z-.15
G95F.001 X.25
G00Z 1
G10X0Z-.15
LF
M30
```

Spindle Positioning

Spindle positioning system specifications -Option on GT-75 only.

Spindle power:	3HP
Voltage:	200 -230V 3 phase or single phase (contact the factory for wiring)
Resolution:	.02 °
Max Speed:	3500 rpm
Min Speed:	.004 rpm

M19 Programmed by itself causes the spindle to position via the shortest route to 0°. After the command is executed the spindle is locked in position. To release the spindle use **M05**. This is a one shot command, it's modal.

CI(-)nnn.nn This makes the spindle move an incremental amount of degrees.

CA(-)nnn.nn This makes the spindle move to an absolute location of degrees.

Snnn.nn The "S" number if programmed along with a M19 indicates the spindle speed in RPM. With no sign the spindle will rotate in the M03 direction. The "-" sign will cause the spindle to rotate in the M04 direction.

G35/G36 - (see notes in G33 section on use and formats) **Extra course long-lead ipr feeds.** The G35 allows long lead ipr feeds. G35 sets Max feedrates to 1 ipr. G36 cancels G35. When G35 is active the system resolu tion drops to .00025". G35 may be activated any time. There is also a G35F2 mode for 2"/rev feeds. Please refer to the threading section for details on format and use.

After G35 and G36 there must be a G92 command

NOTE: Both axis's must be returned to the position they were in when the G35 was invoked before G36 is programmed. G35 must be canceled before a tool change!

Notes on use:

• Before a spindle positioning in absolute command can be executed there must be a M19 command to orient the spindle.

• Be sure that you calculate the amount of C needed for a coordinated C and Z move. In the following example there is not enough C given to complete the Z move, the slide will then hang up. A solution would be to increase C to 432° to complete the Z move.

Formula to find number of degrees needed = the distance travel IPR x 360

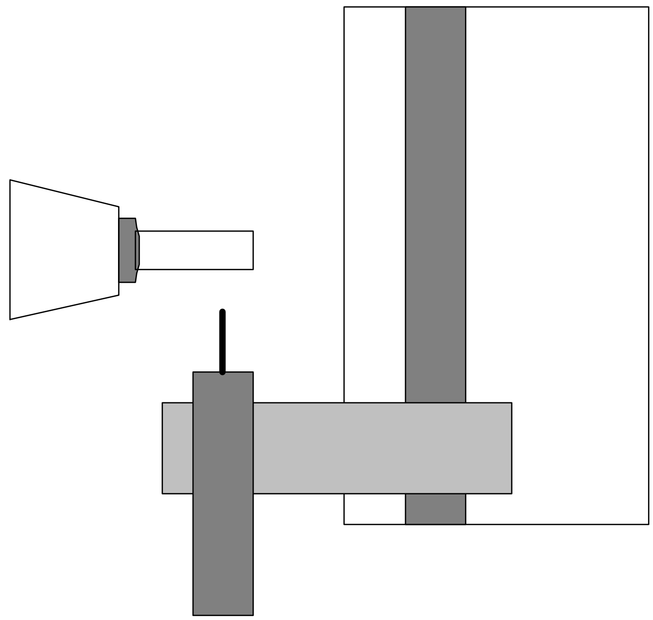
```
Z0
G35
G92X0Z0
G95F.25
C360Z-.3S5
G94F50Z0
G36
G92X0Z0
```

• Currently there is no feedback from the spindle drive that a move to a location has been completed. When you rotate the spindle into position you will have to put a dwell after a rotation command to allow it time to complete the move.

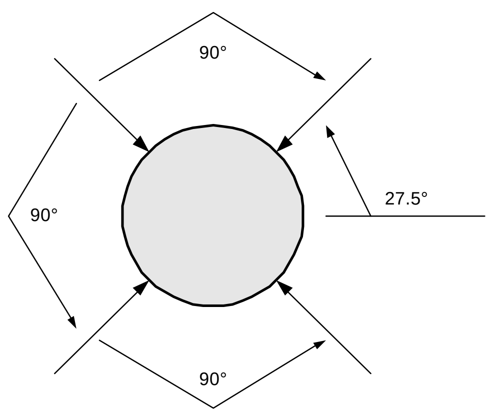
Spindle Positioning

Example showing positioning and cross drilling

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



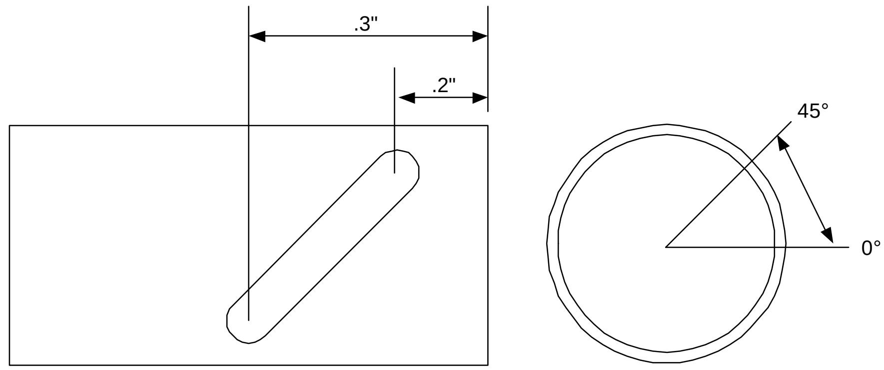
The slide will be used to drill the holes. We will drill (4) holes 90° apart, the first hole is located at 27.5° from a reference 0°.




```
G90G94F30OG73
T5 (LIVE DRILL FROM THE SIDE)
X.75Z 1
Z-.3
M15 (TURN DRILL ON)
M19                 ORIENT THE SPINDLE TO 0 °
CA27.5              GOES TO 27.5° ABSOLUTE
LS4                 START OF LOOP
G94F1X.5            DRILL A HOLE
X.75F300
CI90                ROTATE 90 ° INCREMENTALLY
G04F1               DWELL TO ALLOW SPINDLE TO ROTATE
LF                  END OF LOOP
Z3
M30
```

Spindle Positioning

Two examples showing rotational milling, g95 & g94



Using g95 mode to cut the slot as a lead where z-length of slot is percentage of one revolution:

g90g94f300g73
t2 .... (Live Mill from the side, below part)
x.75z 1
z-.2
g35 .... (Coarse Resolution Mode ON)
g92x.75z-.2 .... (Tool Preset at starting location: necessary for Coarse Resolution Mode)
m15 .... (User defined m-function to turn on mill)
m19 .... (Positioning Mode: locate spindle at 0°)
x.5f1 .... (Feed mill into part)
g95f.8 .... (Set feed rate to .800” per revolution)
ca-45.1z-.3s3.5 . (Rotate spindle m04 direction just over 1/8 turn; z will move -.100” See Note*)
g94f100x.75 ...... (Clear tool, ipm mode)
m16 .... (User defined m-function to turn off mill)
z-.2 .... (Move to starting location *before* g36)
g36 .... (Coarse Resolution Mode OFF)
g92x.75z-.2 .... (Tool Preset again, at starting location: necessary for Coarse Resolution Mode)
g00z2 .... (Clear tool)
m30

Note*: The C-Axis Encoder has much higher resolution than the normal Spindle Encoder. To insure that the Z-axis move completes, add tiny addition C-Axis rotation, as shown

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

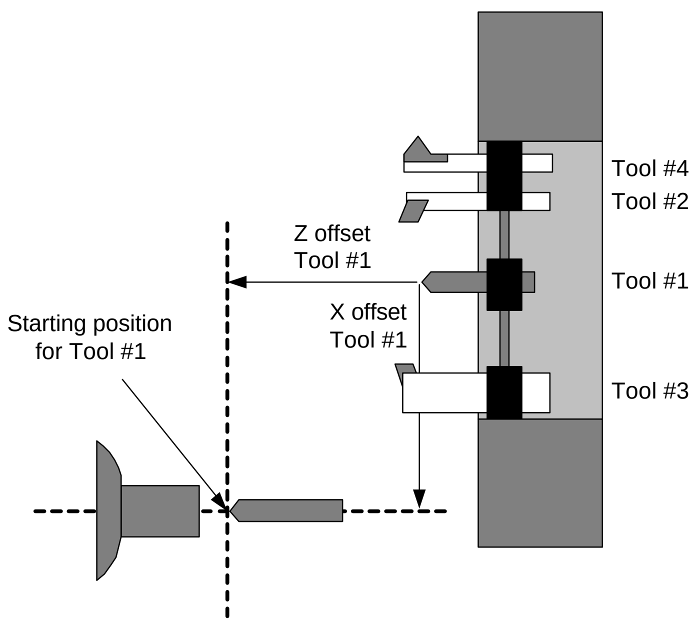
2. **Call the tool offset and establish what the location is:** This is done in the program you have written and used when the program is run

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

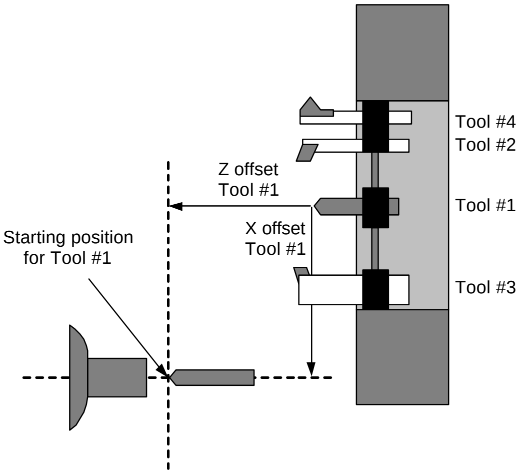


Illustration of tool offset for Tool #1

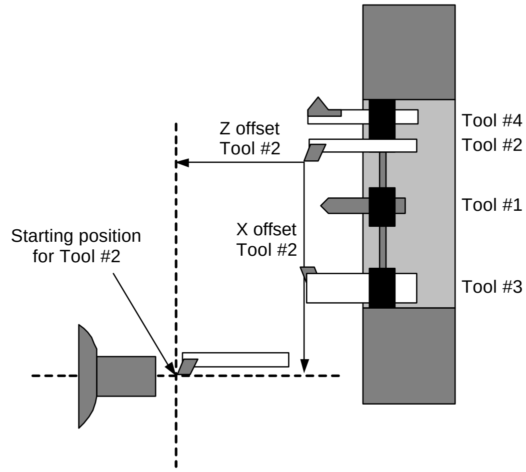
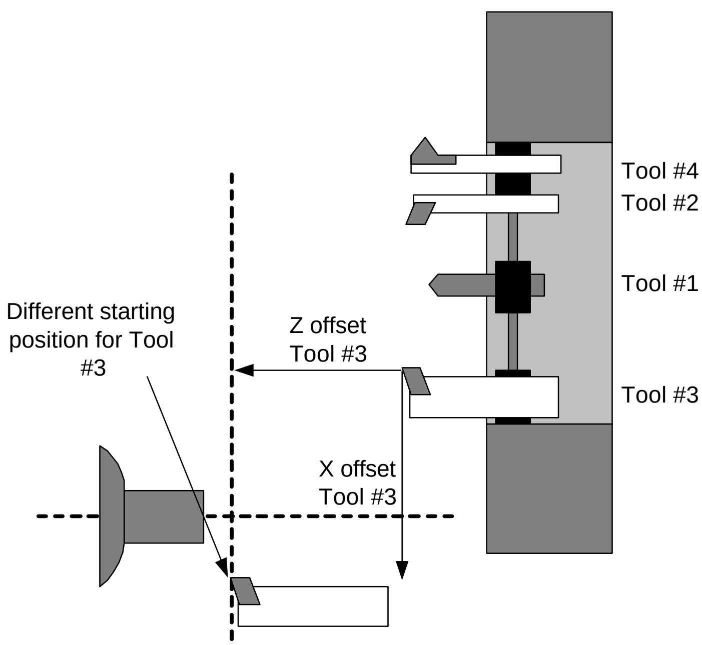


Illustration of tool offset for Tool #2

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
```

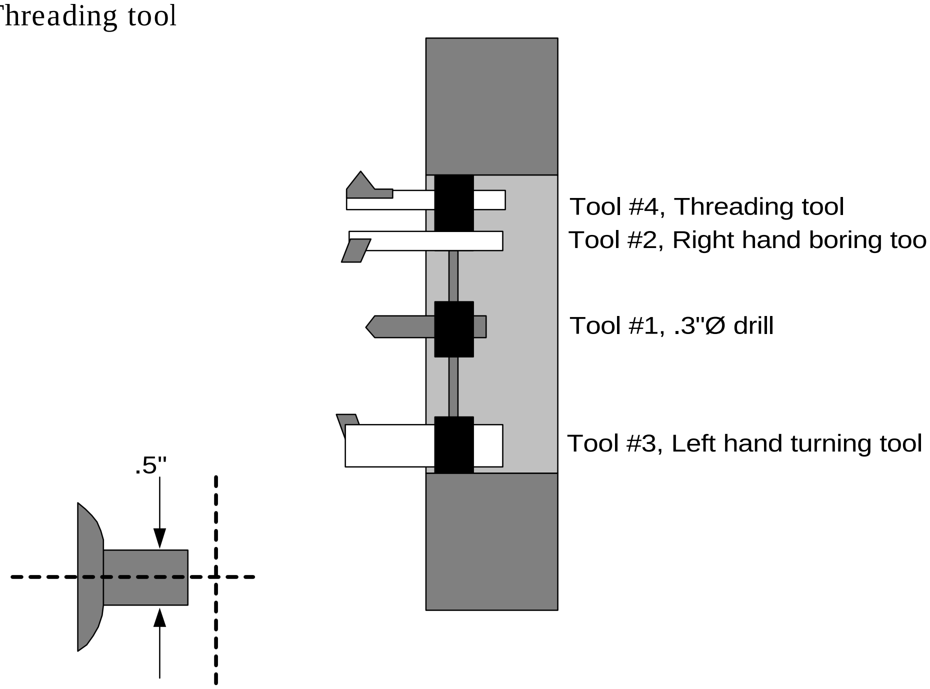
This will move the tool to 1/2" or 1" diameter depending on whether the program is in the diameter or radius mode.

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



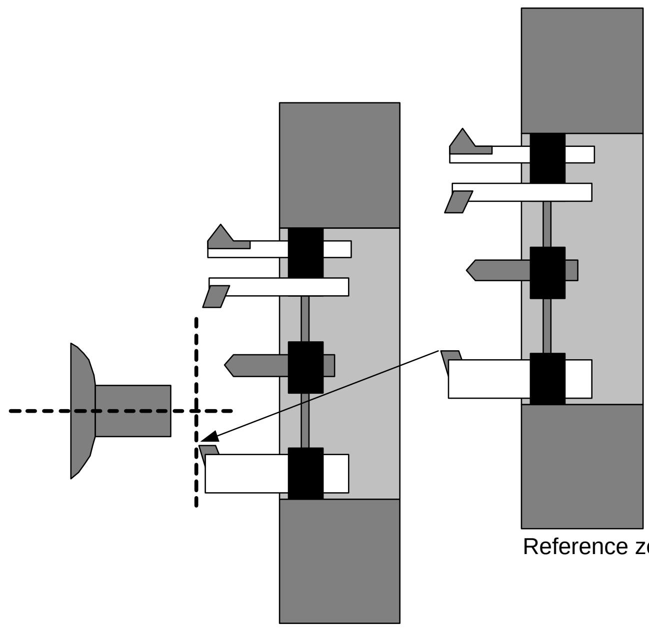
There is no order that the tools must be set up on the slide. It is not important in what order you do the offsets. It is possible to only do one offset and call it #3. The control would not care that there were no values in unused offsets.

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!

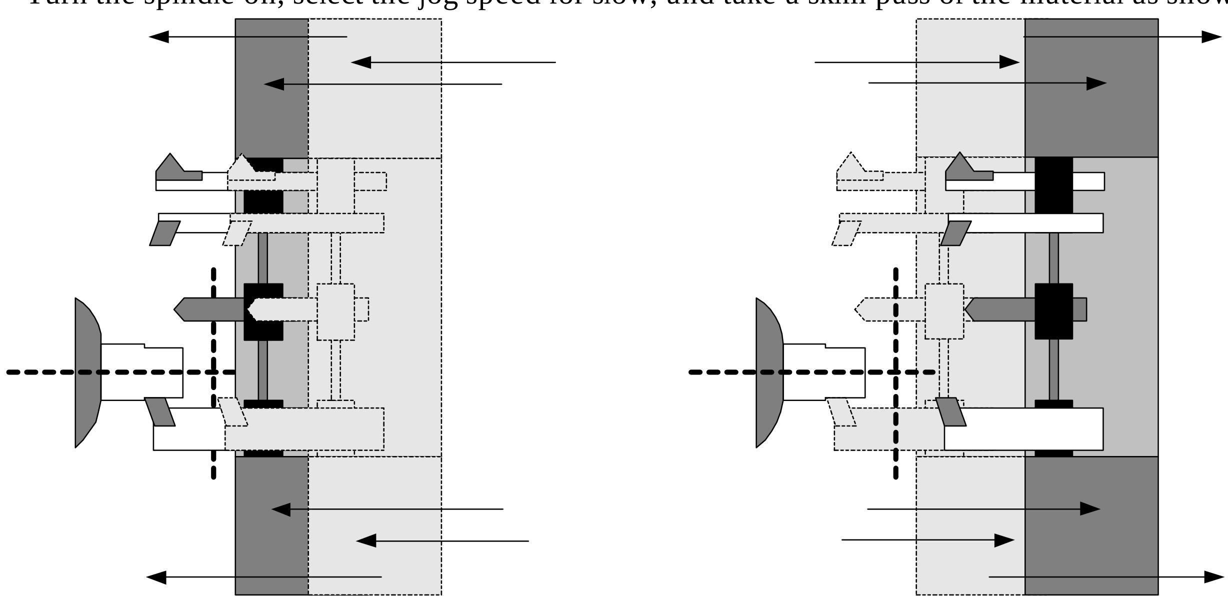
To Set Left Hand Turning Tools (Tool #3)

• 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

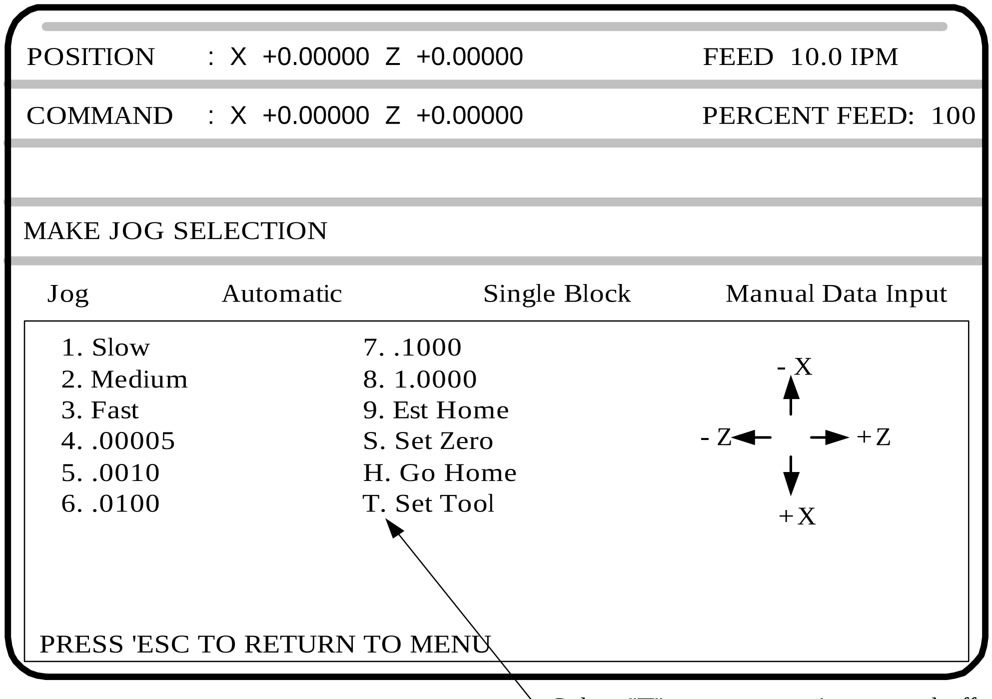
• Turn the spindle on, select the jog speed for slow, and take a skim pass of the material as shown next.



Move slide in and cut material   Move slide back in Z. do not move X!!!

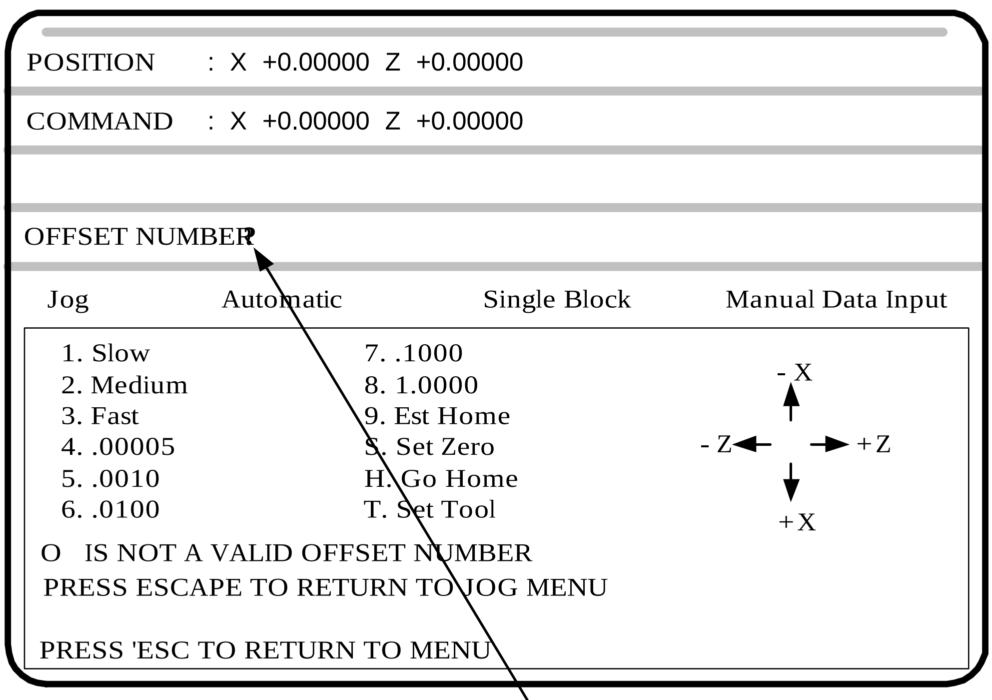
• Then, move the slide back in Z. **Do not move the slide in X.**
This cut will be used to establish the offset.

To Set Left Hand Turning Tools continued



Select "T" to start entering to tool offset

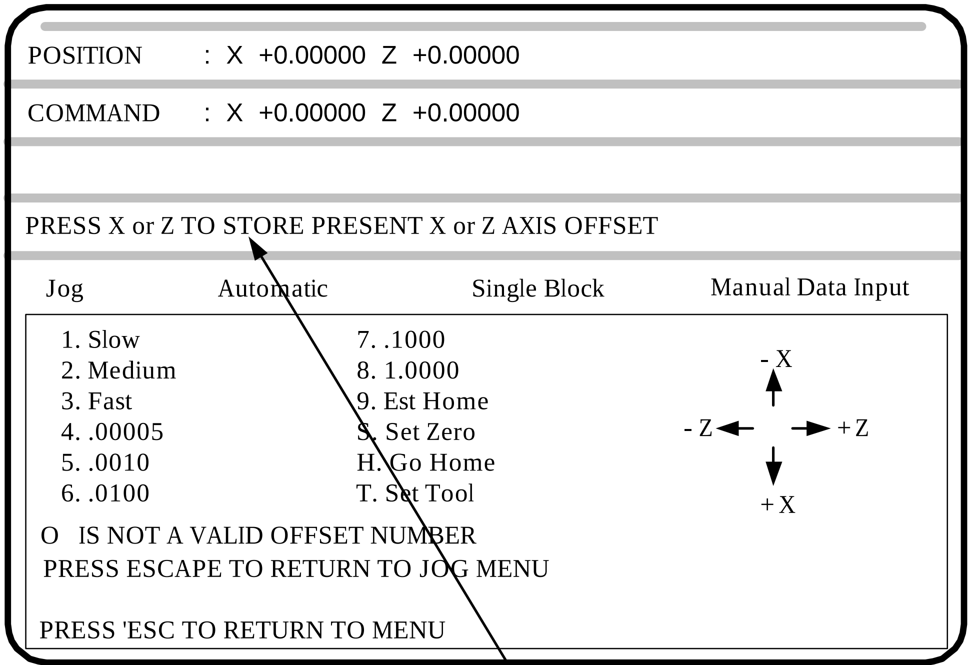
At this point, you are still in the jog mode. Instead of selecting a new jog speed now, select ”T”. The control will now begin the sequence for entering a tool offset. See the next screen.



Enter the tool number you are setting

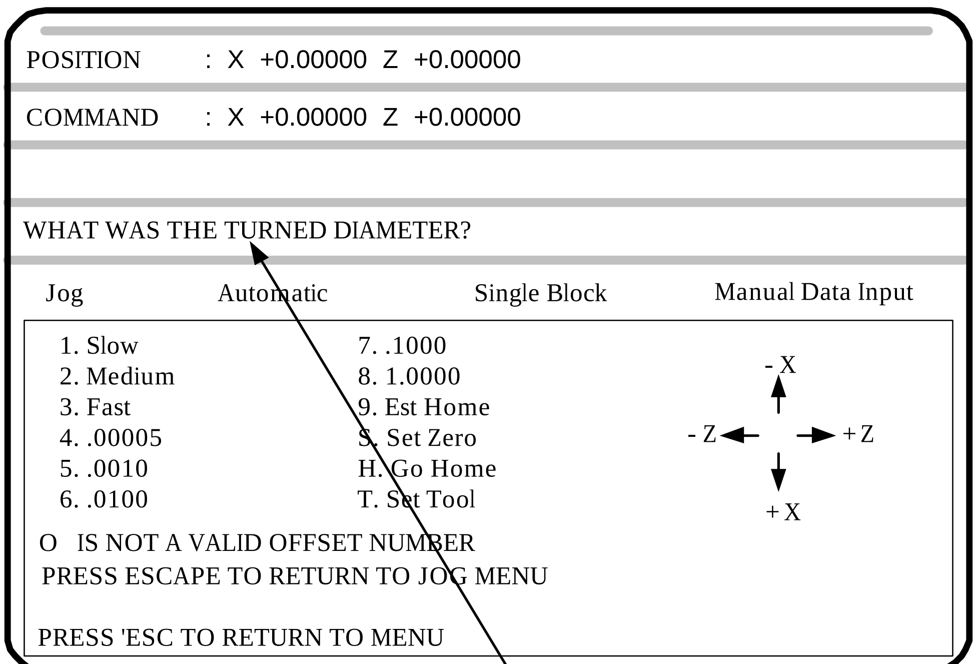
After you have selected T, the control will ask what tool it is that you are about to enter. Type the number tool, in this case 3. Then hit the ”RETURN” key.

To Set Left Hand Turning Tools -continued



Select the axis 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.

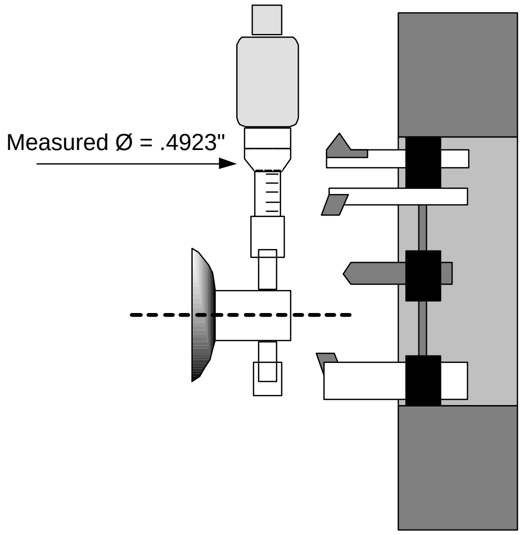


Enter the diameter of the material just cut

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.

To Set Left Hand Turning Tools -continued

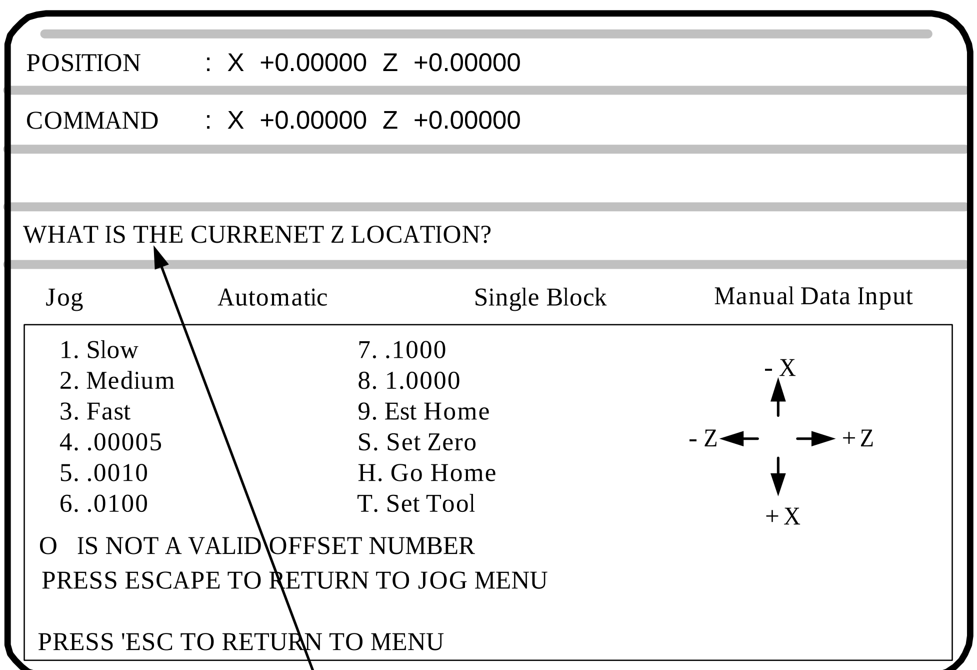


Take some care but do not be overly careful since any error made here can be easily corrected with the tool offset correction later when you are making the first piece. After typing .4923 hit ”RETURN”.

Now establish the Z offset, for tool #3

The setting of the Z offset is a little different.

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

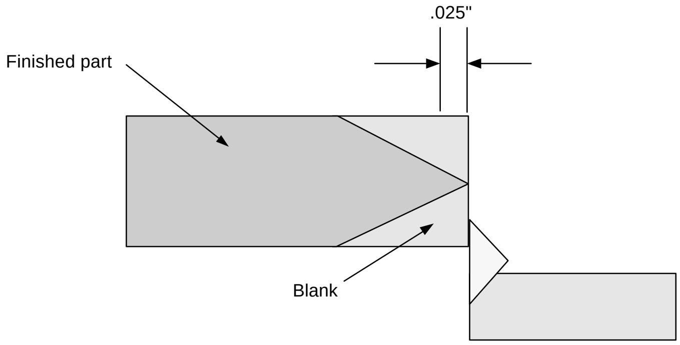


Enter where the tool is from absulute zero on the part

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.

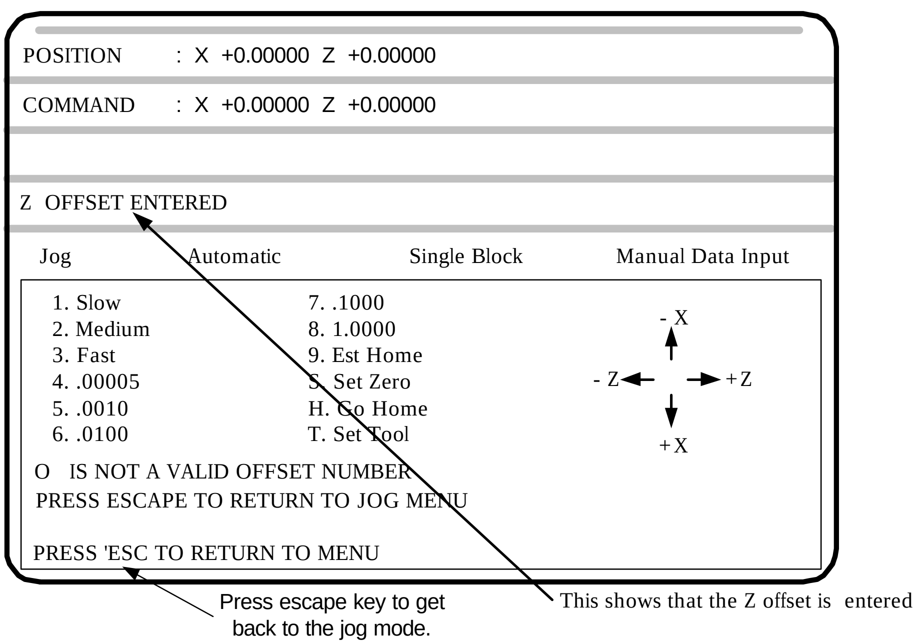
Working with bar stock:
If you are working with bar stock then the location of the Z start is only important in a relative way. Set the tool starting point far enough from the collet or chuck so there will not be a collision.

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.

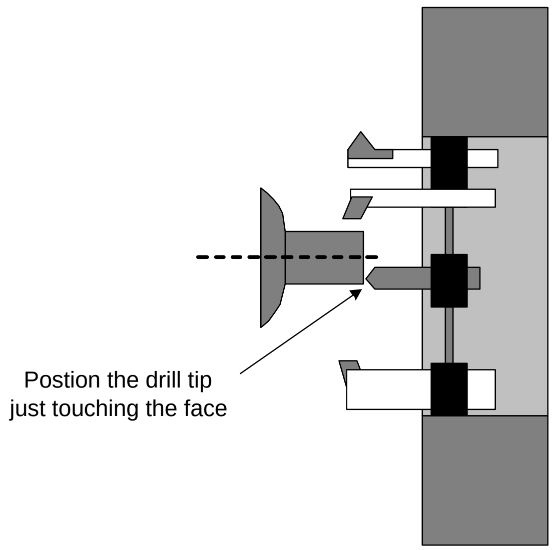
To do this follow the procedure to get the tools into location, then press T and set the X tool, then before pressing ESC, press Z. This will set the Z tool and then exit back to Jog.



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

Setting a Drill (Tool #1)

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



Setting a Drill (Tool #1) -continued

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

• Touch the drill tip to the face of the part

• Select "T" from the jog menu

• Enter the tool number, in this case it is still # 1, then "RETURN"

• Select the axis being set, "Z".

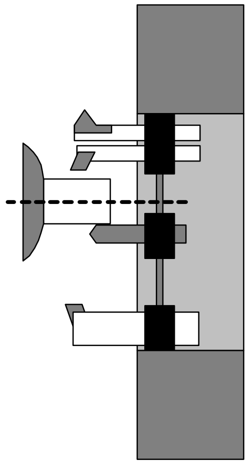
• Now enter the location of the tool from zero. This could be 0 when touching off on a finished part. Then

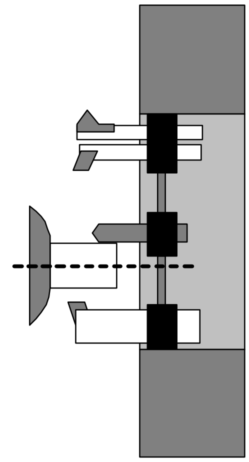
-Press "ESC" to get back to the jog mode.

An alternative method of setting drill:

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

Setting up a drill on the + side of the spindle





Setting up a drill on the - side of the spindle

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

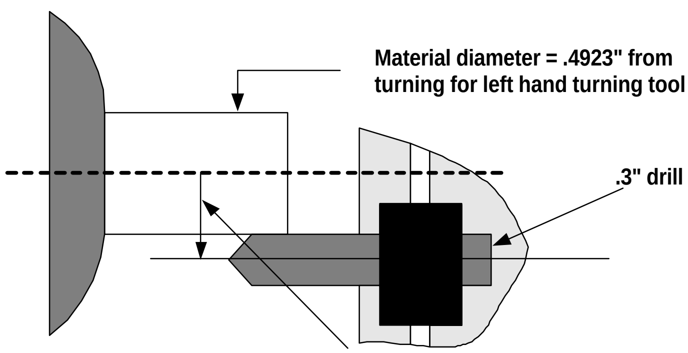
(+) side of spindle: enter diameter measured as a positive number
(-) side of spindle: enter diameter measured as a negative number

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

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

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

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

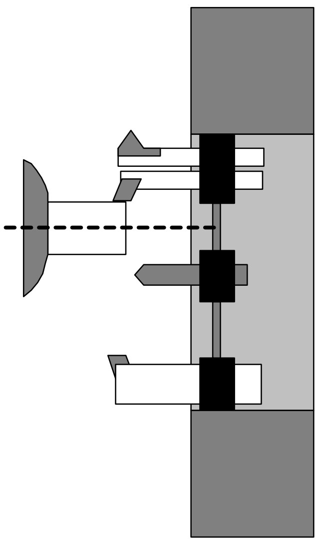


The value is equal to the diameter of the material plus the diameter of the drill. In the above figure you can see that the location of the center of the drill is the sum of these two numbers. .3” drill + .4923” material = .7923” location of drill center in X. For this example we are on the positive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

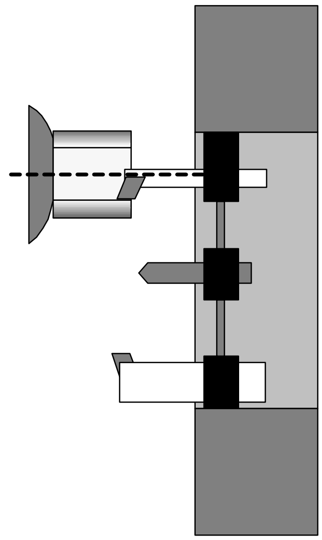
• Then set the Z as you have been shown before.

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

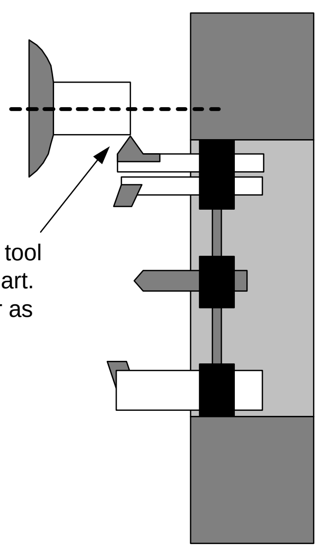


Setting a boring tool on the ID.
The measured Ø will be entered as a
POSITIVE

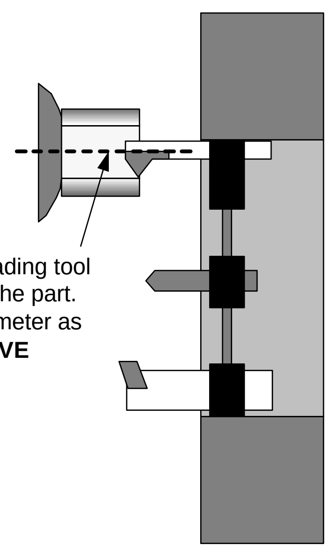
Setting Threading tools

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.



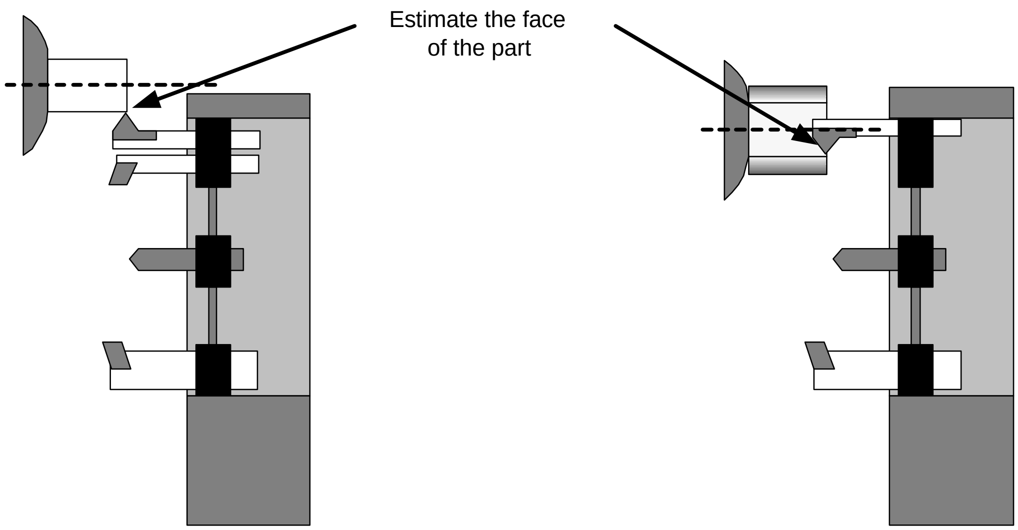
Setting a threading tool
on the OD of the part.
Enter the diameter as
POSITIVE



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

Setting Threading tools, continued

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



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 -(these can be purchased from OmniTum as ”Start-up tooling kit”)
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

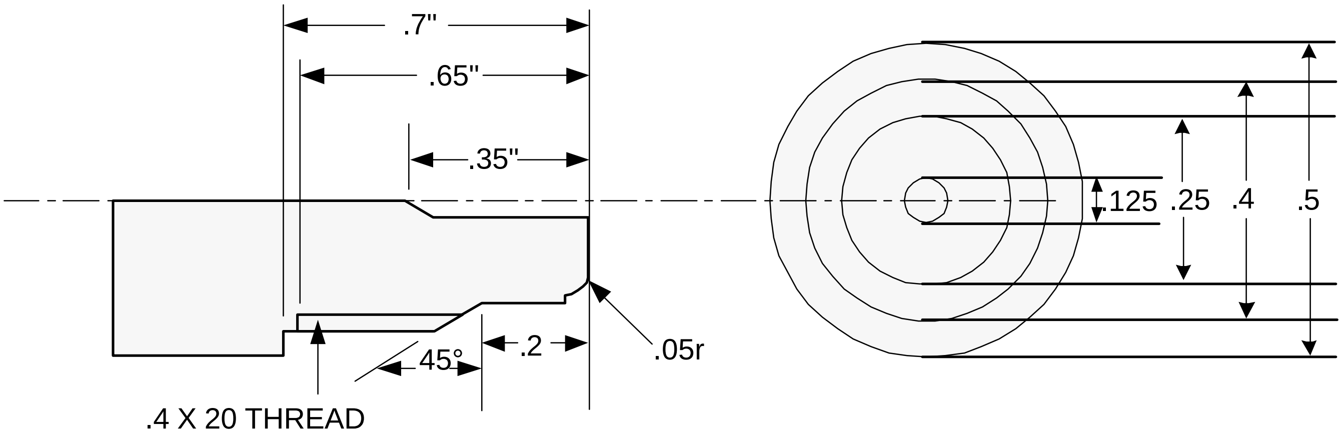
The OmniTurn has a number of options. Such as:
Spindle control - On / off
Spindle speed control - infinitely
Threading encoder
I/O additional ”M” functions

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

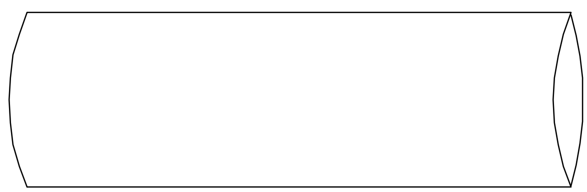
OmniTurn Startup sample part



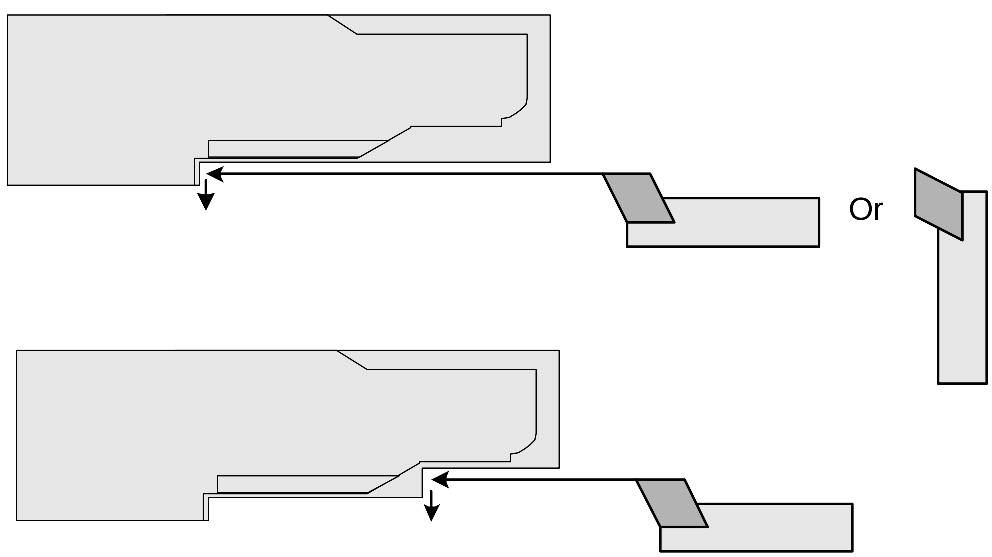
If you are familiar with machining you might want to skip to the next section, "TOOLING"

OPERATIONS

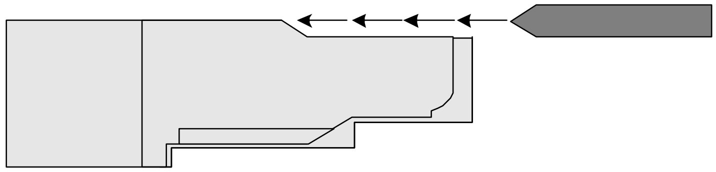
We will start with a solid piece of material 1/2" in diameter, 3" long.



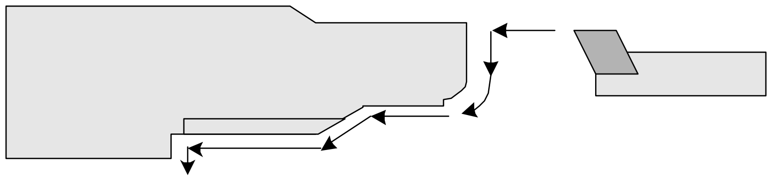
In machining this part we will use the same tool for roughing and finishing the OD. This does not have to be the case. You could setup another tool for the finishing pass. We will be doing the roughing first.



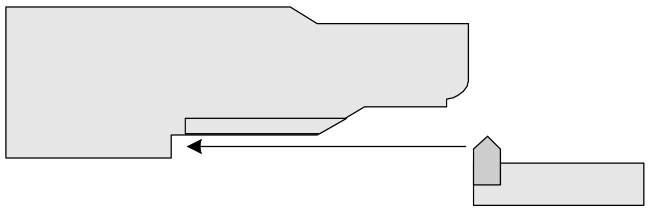
We will take two passes to rough the part. After we rough the OD and remove most of the material we change tools and center (spot) drill. Then we change tools again and peck drill the hole to finish depth.



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

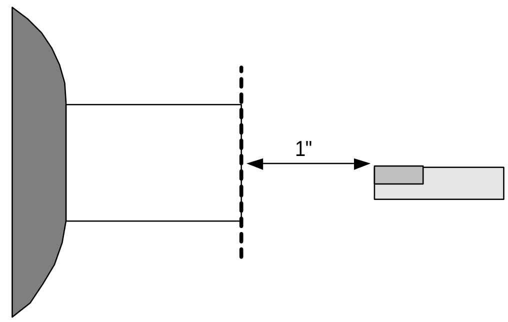
Before we start to program there are a number of decisions that must be made. Tooling is one. First you have to select the tools to be used. This has been gone over in the previous section but they have not been formally listed yet. Now we can assign tool numbers:

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's is not to 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:

-Turn the OmniTurn on
-HOME the slide (see the section on homing in the Jog section)
-ESC back to the main menu
-Go to the Automatic menu by pressing A
-When asked ”FILE TO BE PROCESSED” type the name of the new program. For this example we will use SAMPLE
-The control will say ”FILE NOT FOUND, PRESS ESC TO CONTINUE”.
-If you do not get the above message, that means there is already a program with this name and you should either delete it or select a new name.
-Then press F3 to enter the editor
-Press Fl to create a new file

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 the 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.

X.1 Z.025 • We will use the first tool to act as a material length stop. Here we are positioning the tool a little off center and in front of zero of Z. There are two reasons we stop the tool a little on the plus side of zero: when the collet is closed the part will pull back some, the other reason is we want to leave a little material to be faced off.

M00(close the collet) • Stop the program to allow the operator time to pull the material out to the stop and close the collet. The comment will alert the operator to perform this.

M03S2000 • Turn the spindle on at 2000rpm.
This command operates an optional function. If your machine does not have spindle control please do not enter this command. What you might want to do instead is to add a comment on the previous line ”and turn the spindle on”.

X.27 • This moves the tool beyond the major diameter so it can move in Z for a rough facing cut. This move is still in rapid

G96S250 • This changes the spindle speed to ”constant surface feet” mode. We will be cutting at 250 sfm. If you have a spindle drive option you can choose to cut in this mode or just leave it in RPM.

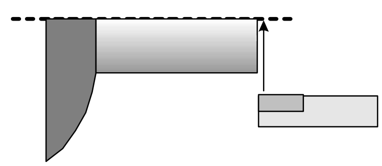
G77S3000 • Sets maximum spindle rpm to 3000 rpm so the spindle won’t go to fast while facing off the part. If you had the part in the chuck you would probably make the G77 speed much lower.

G04F1 • This is a dwell to allow the spindle to come up to speed before cutting

Z.015 • This positions the tool in Z for the rough facing move. (rapid feed rate)

G95F.003X-.015 • Here we change to IPR at .003” per rev. We move the tool past zero in X to face the part. We move past zero since there is a radius on the tip of the tool. This move assumes that the radius is .015”. Please adjust this move to coincide with the tool you are using.

X.13 • Move the tool for the second roughing pass, still in rapid



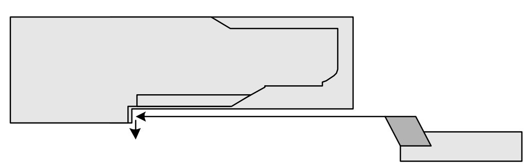
G94F300Z.025 • We move the tool off the face of the part in rapid

G97S2000 • Changes spindle speed to rpm mode, and sets speed to 2000 rpm.

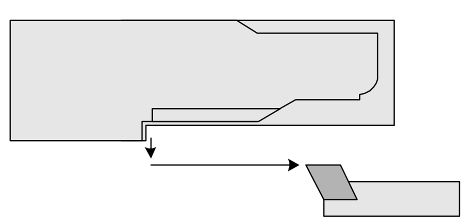
X.205 • Position the tool to take the first roughing pass, this is still in rapid

G95F.003Z-.75 • Change the feed rate to IPR and move the tool to take the first rough pass

X.26 • This moves the tool out in X at the same feed rate of .003”



G94F300Z.025 • The tool is moved at rapid to clear the face of the part

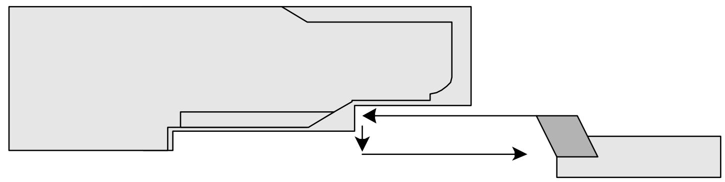


S2500 • Change the spindle rpm to a higher speed for the next smaller diameter pass.

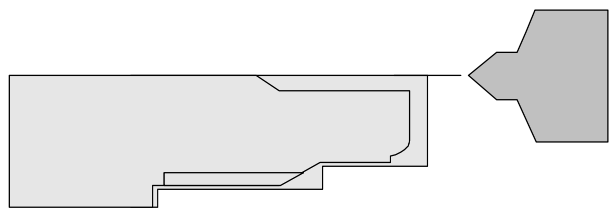
G95F.003Z-.195 • Change to feed mode, and make the second pass

OmniTurn Startup sample part

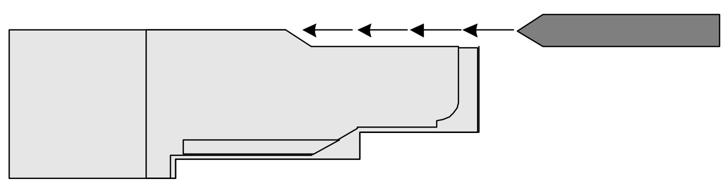
X.225	• This clears the tool, still in the feed mode
G94F300Z1	• Change to rapid and move the tool to Z 1, this is clearance for a tool change



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

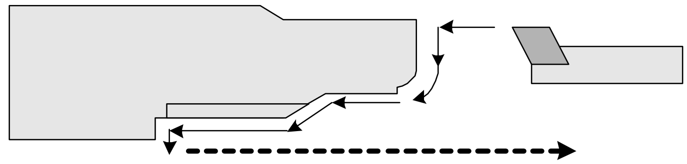


G94F300Z 1	• Rapid the tool clear of the part for a tool change
T3S2800(Drill)	• Call the drill into position and change the spindle speed
X0Z1	• Move the drill to a safe location
M01	• Optional stop for the tool check on the first run through
2.025	• Rapid the drill to the face of the part with a little 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

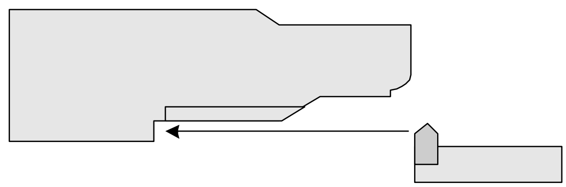


G94F300Z1 S2400	• Move the tool to clear the part for a tool change and change spindle speed.
T1 (LH tool .007 TNR)	• The turning tool is brought into position again for the finish pass
X0Z1	• Move the tool to a save location
M01	• Optional stop for the tool check on the first run through
G77S3000	• 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

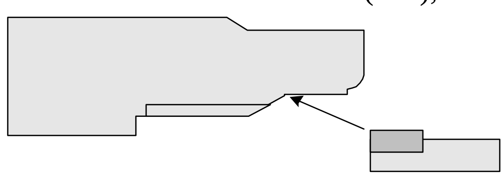
OmniTurn Startup sample part



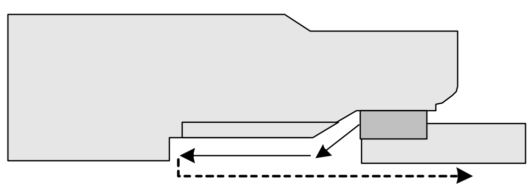
T4(Thread tool) • Calls the threading tool offset
X0Z1 • Moves the threading tool to a safe location
M0l • Optional stop for the tool check on the first run through
G04FI • This is a dwell used to allow the spindle to change speed
X.2 • Moves the tool to the starting diameter of the threading cycle
Z0 • 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 .0040 • 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
T0 • This moves the slide to it's home location **(This is not needed, you do not have to go home at the end of every part!)**
M30 • End of the program, the control will now be ready to run another 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”

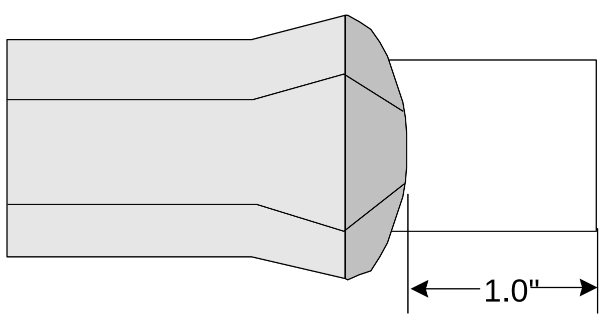
Then press F4 - Please see the notes on using the verification software in chapter 5

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:

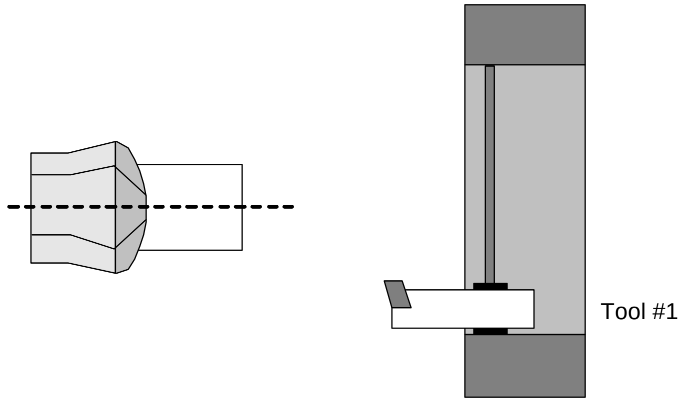
• Set the tools so that they will not interfere with one another during tool change or during a cutting cycle. Be sure to consider both the length and width clearances.
• Keep the tools as close together as reasonable so there is not to much time wasted during tool changes.

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.



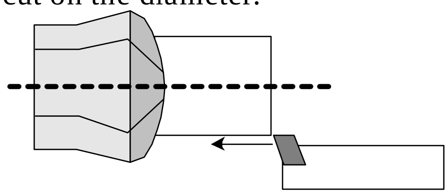
Then put the turning tool on the slide. For this example put it on the near side.

Setting X axis

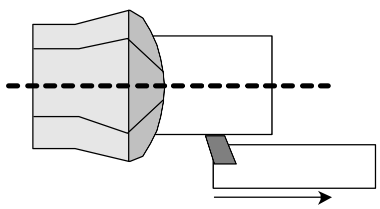


Now go to the jog mode and follow the procedure on setting up tool. Turn the spindle on, (if you have a unit with spindle control do this in the MDI mode first).

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

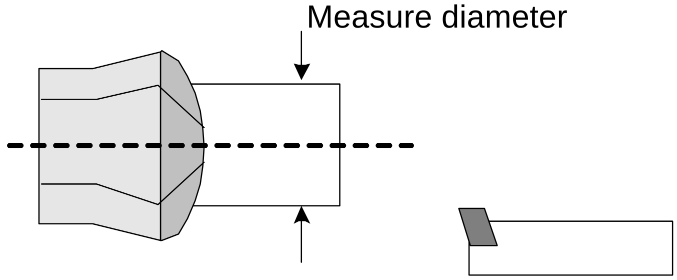


Then move the slide in Z only to clear the material, do not move in X!



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

Then measure the diameter with a micrometer



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

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

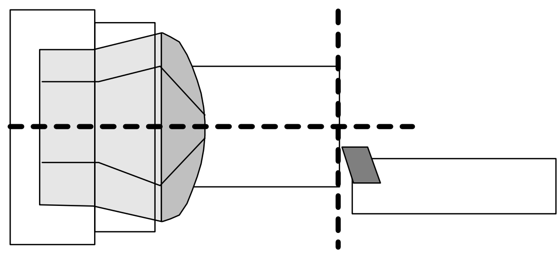
Next you have to tell the control that you are setting the X axis for the tool offsets, press X.

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

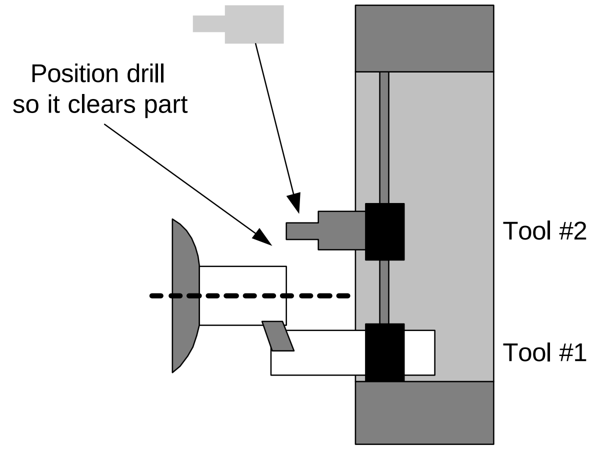
Press ESC to continue in the jog mode.

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:

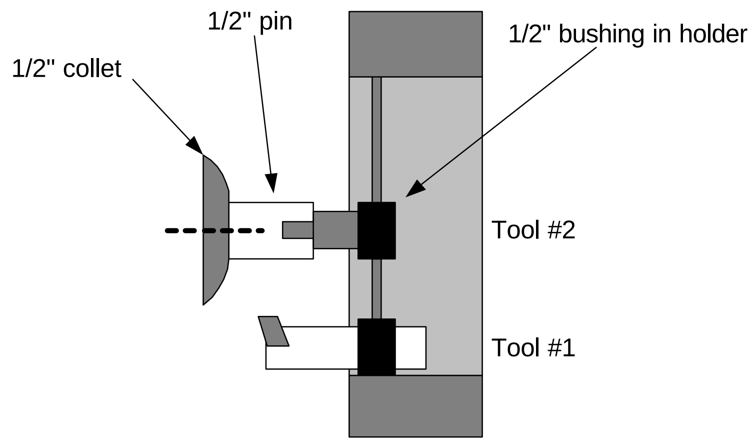
Put the control into the jog mode and move the first tool to it's worst case position and then put the next tool on the slide so there is enough clearance with all obstacles. This does not have to be very exact with most examples since it is only for clearance. When you get into high production jobs 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.

Setting X for a Drill



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

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



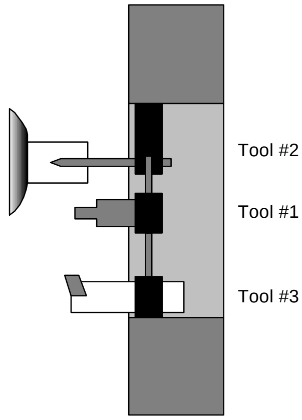
-In the jog mode select ”T”
-When asked for the tool number, enter ”2” and return
-Press X for the axis you are entering
-Enter ”0” for the diameter measured and return
-Press ”Esc” to return to the jog mode

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

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

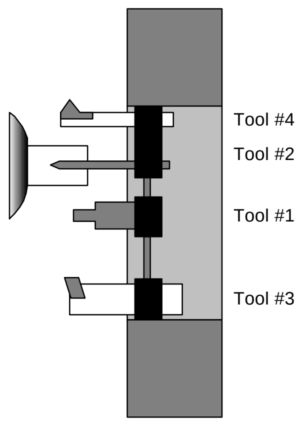
OmniTurn Startup sample part

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

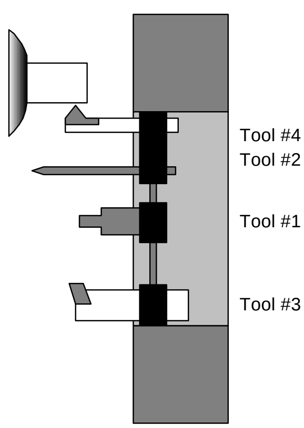


Next is the threading tool. Here we will use another technique. Instead of cutting with the tool we can use the cut surface from setting the first tool to establish the location of this tool.

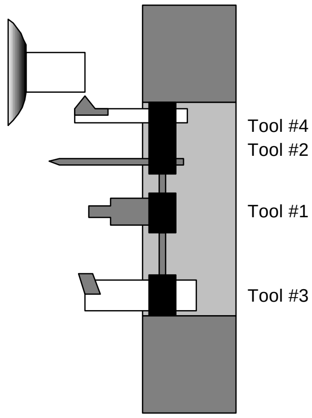
First, while the slide is still on center with the drill, mount the threading tool so it does not collide with the spindle or the work piece.



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

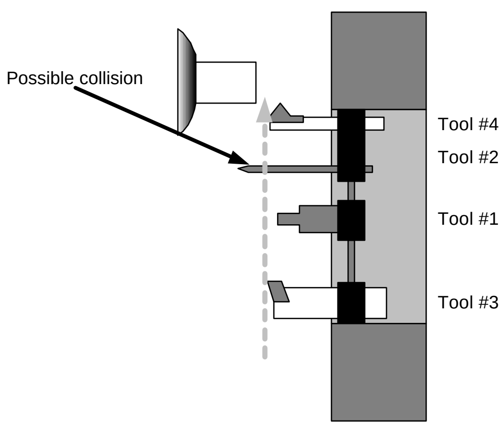


Then press ESC and jog the tool until it is just even with the end of the material. This location is generally not that critical so don't waste to much time.

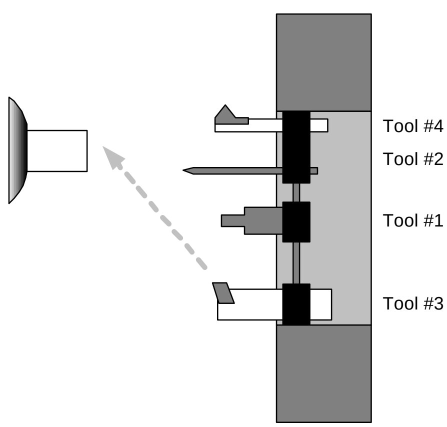


Now that you have the tool where you want it in Z you can establish the tool offset. Press T to start the entry and then select Z. Then enter 0 for the current location and press RETURN. When the control tells you that the offset is entered press Esc to get back to the jog mode.

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.

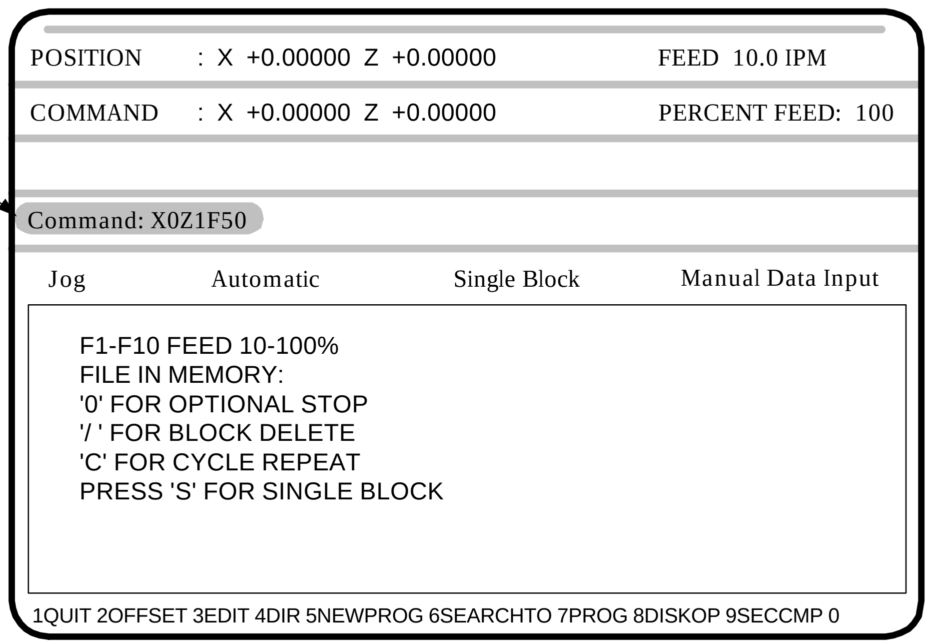


Checking the tool offsets

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 exit the jog mode back to the Main menu. From the main menu press M. In the MDI mode you can enter one command at a time and execute it. To call the tool into position enter the command that calls the tool:

T1, return, and cycle start. The position display will show the distance from the face and center of the work piece to the tool location. If this distance does not look correct you might want to reset the tool. Then type the command **X0Z1F50 and RETURN.** This command will move the tool to center and 1" from the face of the work piece at a feed rate of 50 ipm. After you press cycle start the slide will move. If you want to stop the motion press the MOTION STOP button on the face of the control. If you want the motion to continue press the cycle start again. To cancel the command press the ESC key while in motion stop.

Call the tool with
a slow feedrate



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 F1, this will bring you back to the main menu.

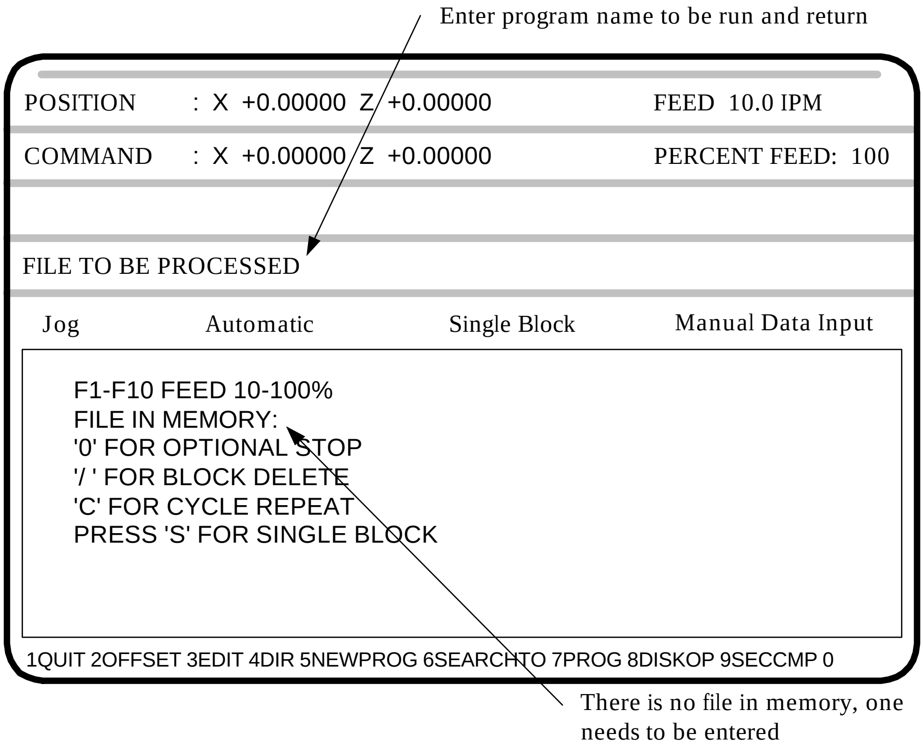
Inputting secondary offsets for TNR Compensation

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. SECCMP

- 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.,

Testing the program

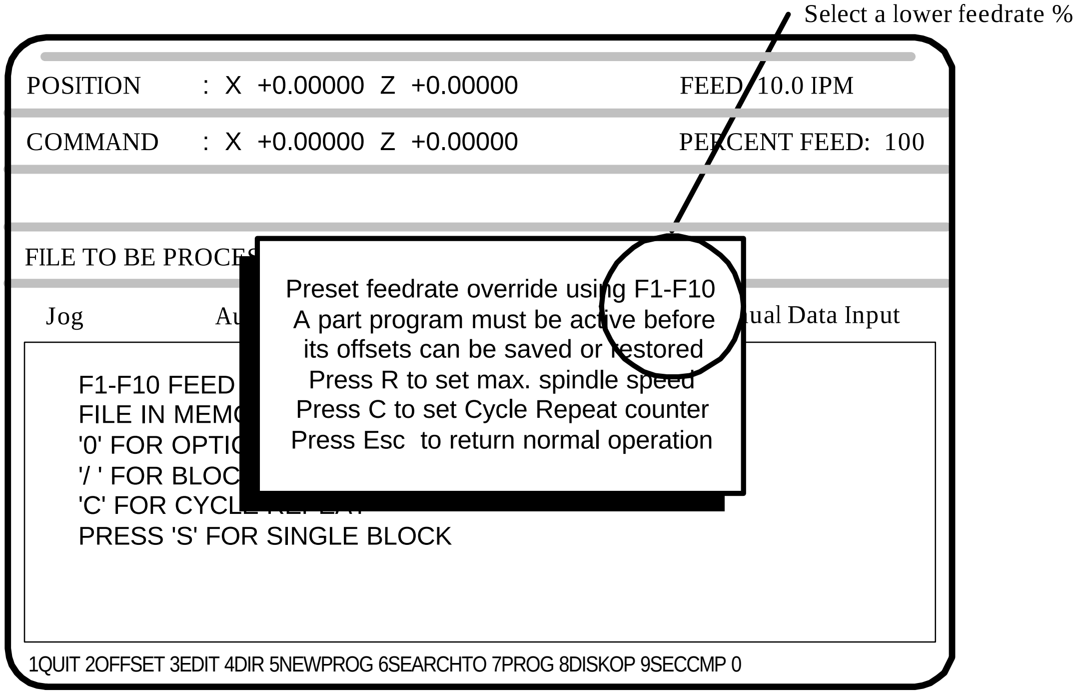
Now that you have made sure that the tools go to where you think they should it is time to test the program you have written. Go to the Automatic mode screen. If you were still in the MDI mode you could press F1 to quit MDI, and then press A from the main menu screen. Then at the Automatic mode screen you have to enter the name of the program to be run. If it is already listed as **the FILE IN MEMORY** and the control asks you to press cycle start, then you are ready to go on. If there is no file in memory then enter the name now. If there is another program active press F5 to allow you to enter the new name now.



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

Feed rate over rides:

The F" keys will lower the feedrate as a percentage. If you press F1 while the slide is running a program the slide will slow down to 10% of it's normal speed. F2 will give you 20%, all the way up to F10 at 100%. 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. You can preload a percentage by going into F10 from the Automatic menu. With this page you can press an F key before you start. When you press a new feed rate over ride you will notice the PERCENT FEED will update and confirm your selection. Then press Esc to go back to the Automatic mode.



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 put the control in the main menu.

OPTIONAL STOP - M01

In the example program we put MO I, 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.

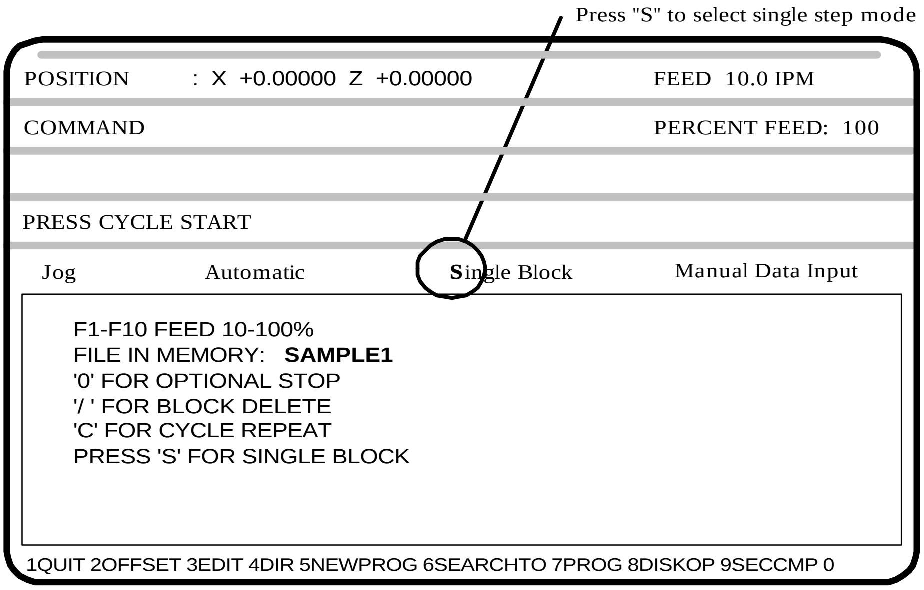
To activate the Optional stop mode press the letter O when in the automatic mode before you start the program.

Single Block mode

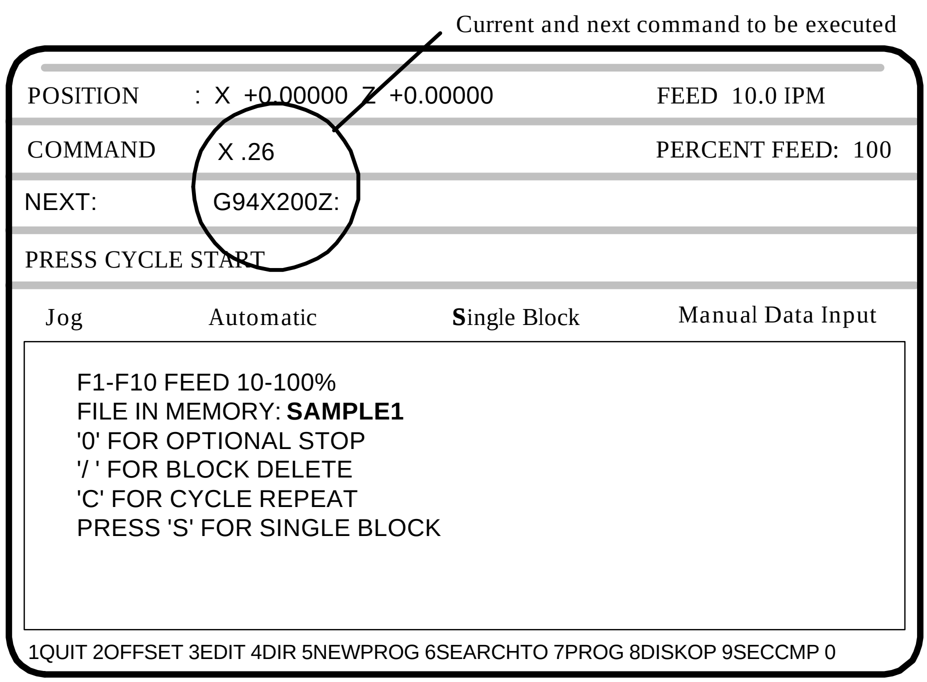
This mode will execute one line of the program and wait for you to press cycle start again to execute the next line. This way you can see what will happen with your program before it happens. This is really good to do if you are not sure what the program is going to do. After you gain some experience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

OmniTurn Startup sample part



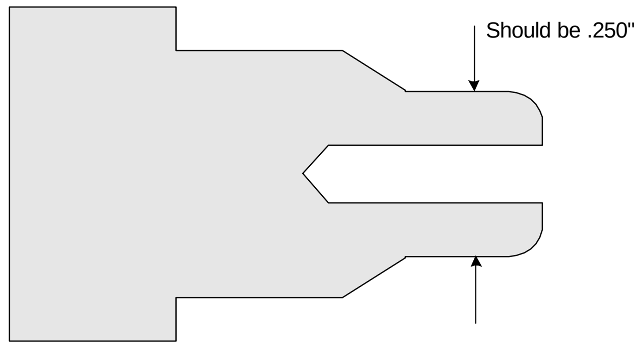
This mode will show you the line of code that you are currently executing and the next line that will be run. This way you can make sure that the code you are running will go where you think it should. If you find and error and want to edit the line before you run it you can press ESC and then go back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



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.

OmniTurn Startup sample part



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.

OmniTurn Startup sample part

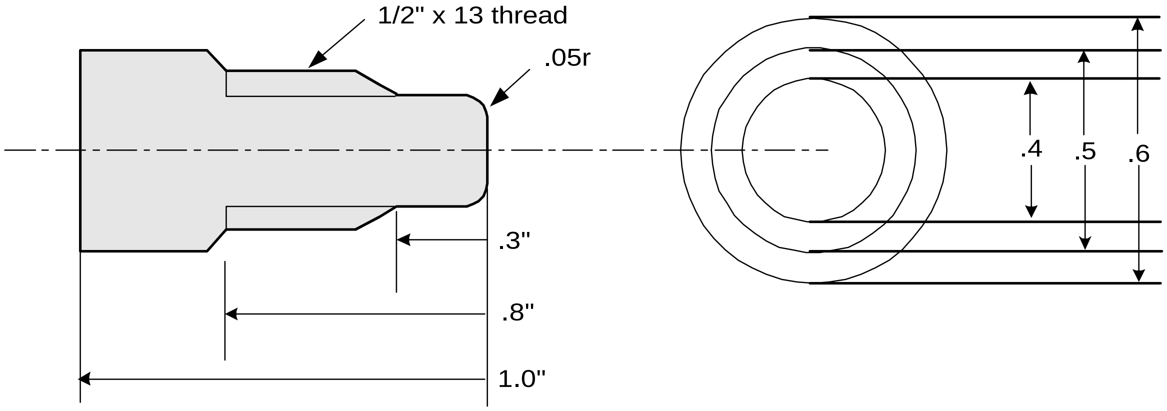
1	X: +0.86480 Z: -1.25340	17	X: +0.00000 Z: +0.00000
2	X: +1.65025 Z: -1.99200	18	X: +0.00000 Z: +0.00000
3	X: +2.91130 Z: -0.93885	19	X: +0.00000 Z: +0.00000
4	X: +0.00000 Z: +0.00000	20	X: +0.00000 Z: +0.00000
5	X: +0.00000 Z: +0.00000	21	X: +0.00000 Z: +0.00000
6	X: +0.00000 Z: +0.00000	22	X: +0.00000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

X DIAMETER ADJUSTMENT:
Press Esc to exit offset adjustment screen

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

Now you are ready to run parts.

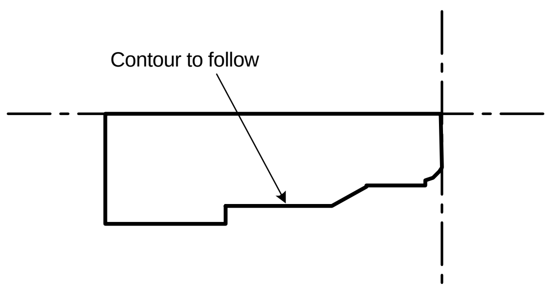
Worked Examples



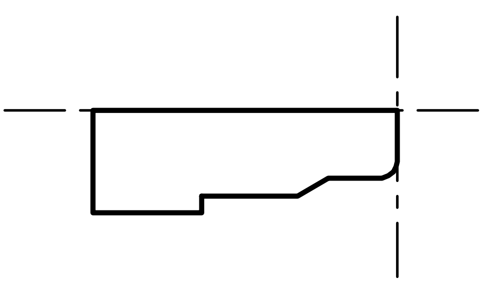
In the above example, the finished part will be made part from a cutoff blank .61” diameter by 1.1” long. The first item to take care of is the layout of the job. This will entail what our sequence of operations will be. Then select the tools to accomplish this sequence. Once the tooling is set it is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

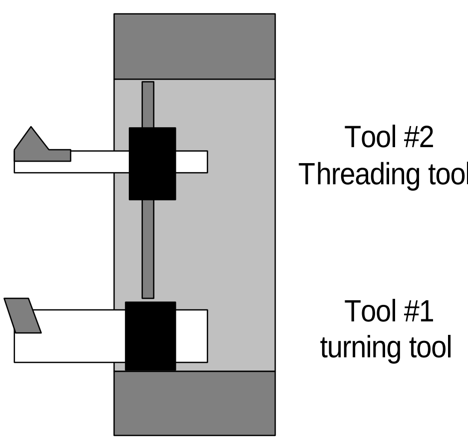
The machining operations that we will be performing are:

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



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





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.

Ø.5” ÷ 2 = .25” radius

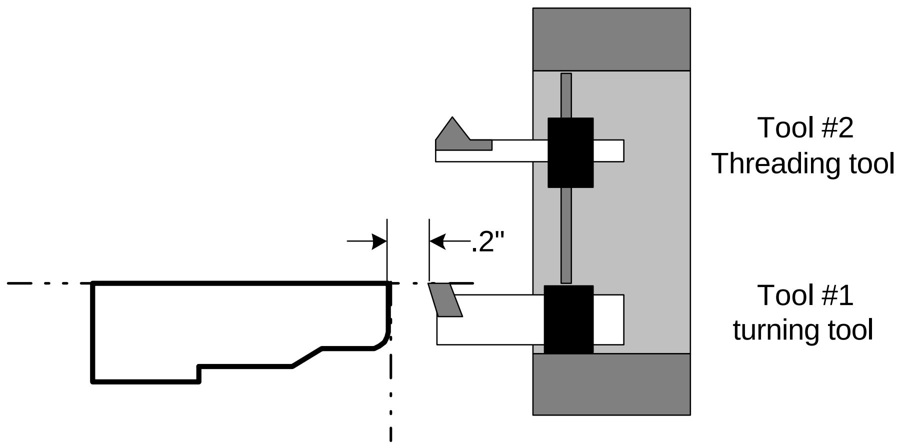
This radius less the first cut of .005” is .25” -.005” = **.245”** starting location in X

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:

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.

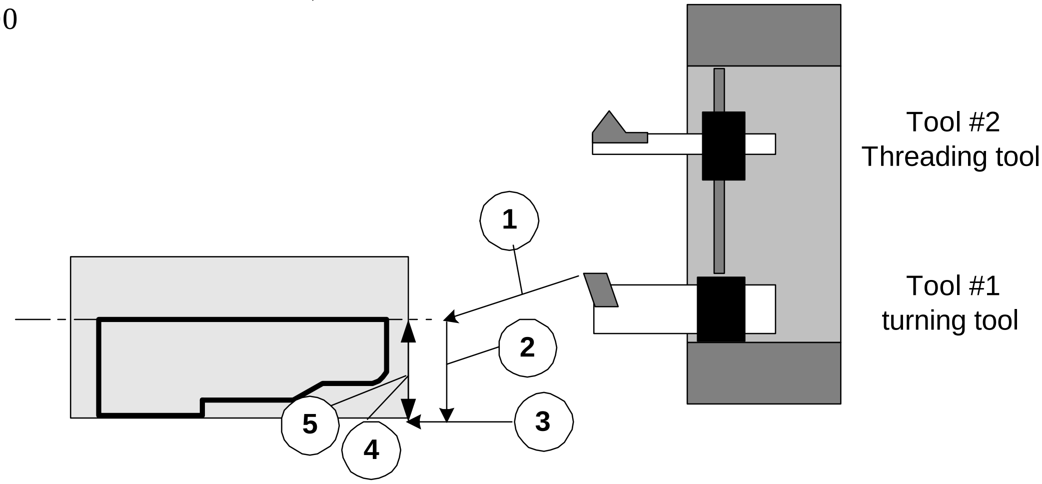
Before we start to program there are a few more comments to be made:

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.

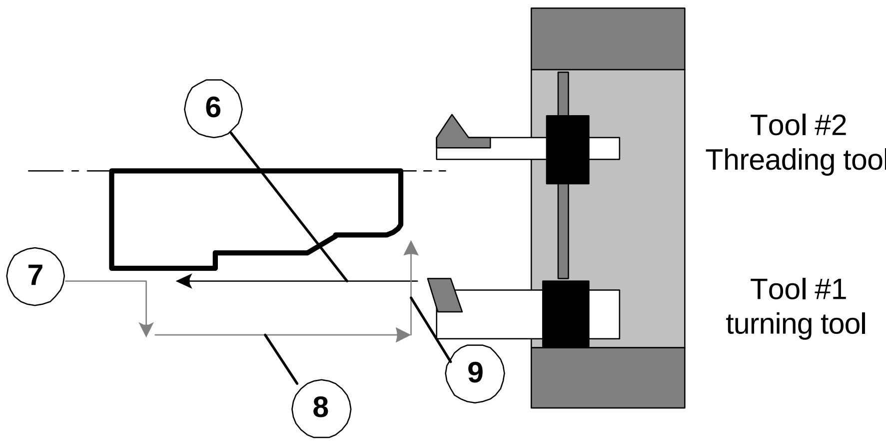
G94 or G95: This part will be programmed in G94, Inches per Minute.

Worked Examples

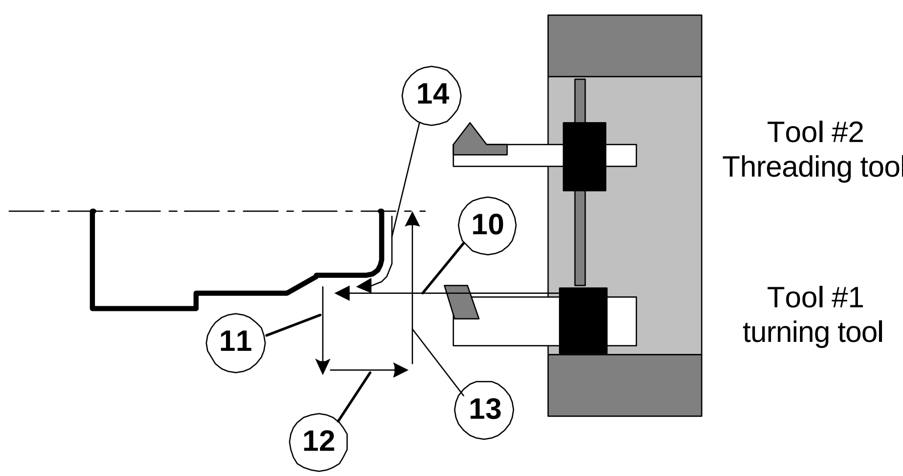
```
(THIS IS A SAMPLE PART FOR OMNITURN)
        G90 G94 F200
(1)     T1
        G92 X0 Z.2
        M01
(2)     X.35
(3)     Z.01
(4)     X0F3
(5)     X.26F200
```



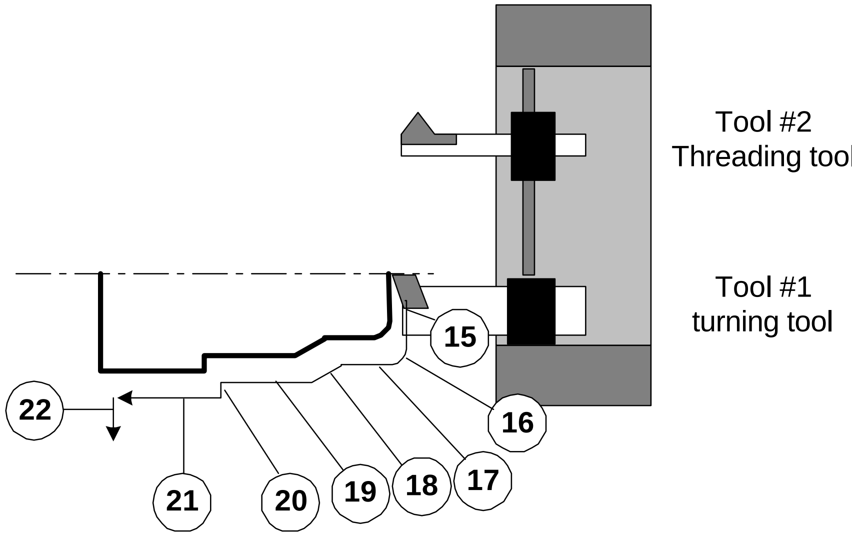

```
(6)     Z-.79F3
(7)     X.27F200
(8)     Z.1
(9)     X.21
```



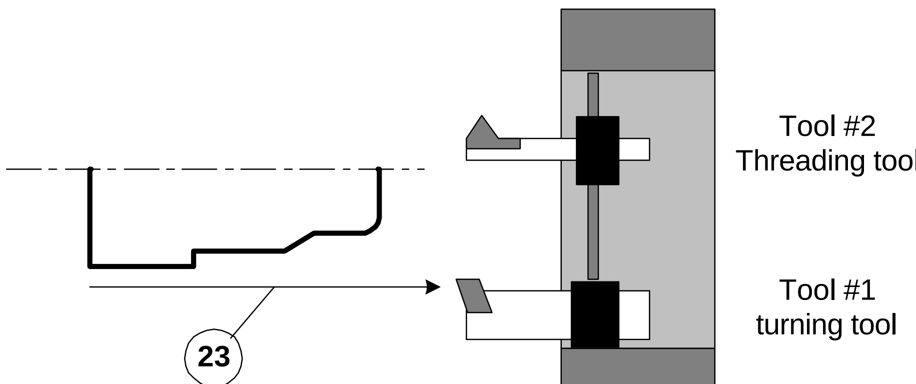

```
(10)    Z-.29F3
(11)    X.22
(12)    Z.1 F200
(13)    X0
(14)    Z0F3
```



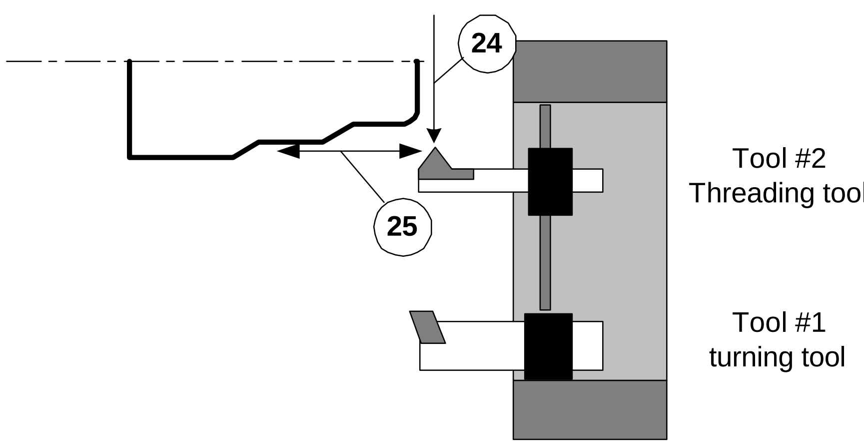
(15) X.15
(16) G02X.2Z-.051.15K-.05
(17) Z-.3
(18) X.25Z-.35
(19) Z-.8
(20) X.3Z-.85
(21) Z-1
(22) X.32



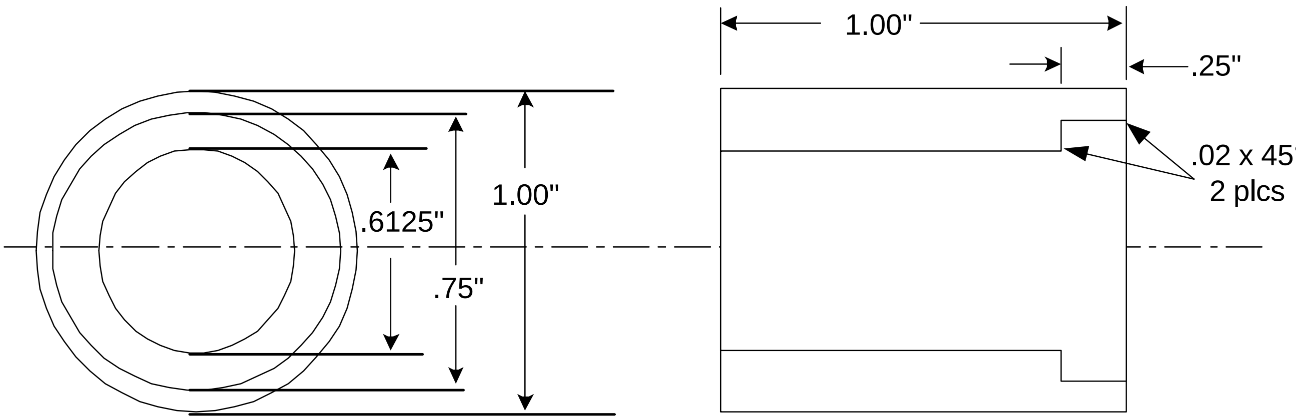
(23) Z.5F200



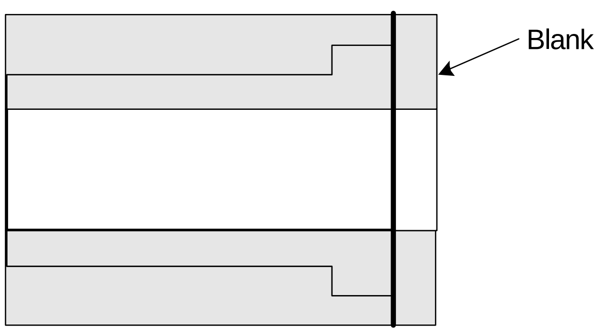
(24) T2
G92X.245 Z.2
M01
(25) G33X.22Z-.81.005K.0769P
T0
M30

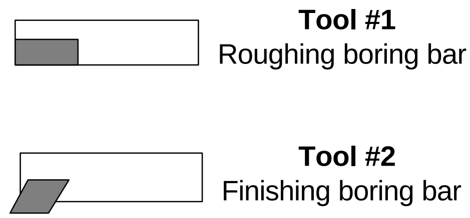


Worked Examples



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

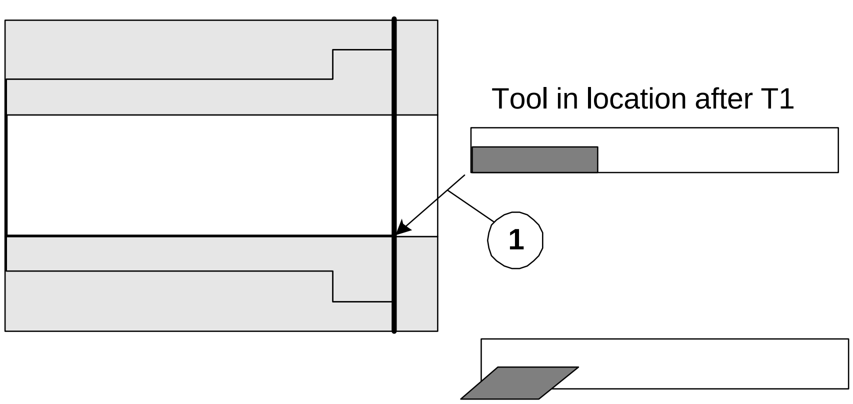




The blank has been predrilled and is a little longer than the finished part.

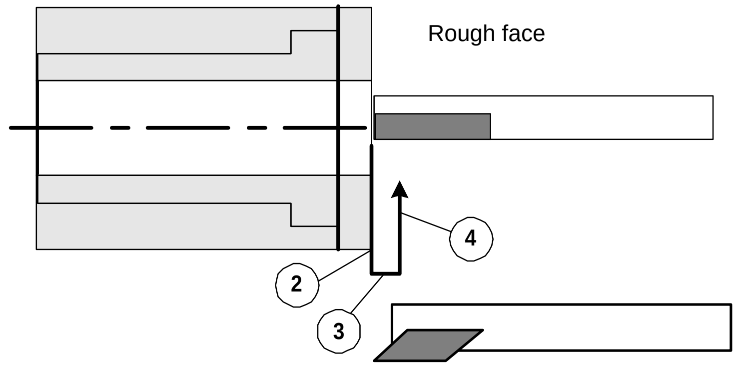
G90 G94 F200
T1
G92 X0 Z.2
M03S2000

(1) X.28 Z.01

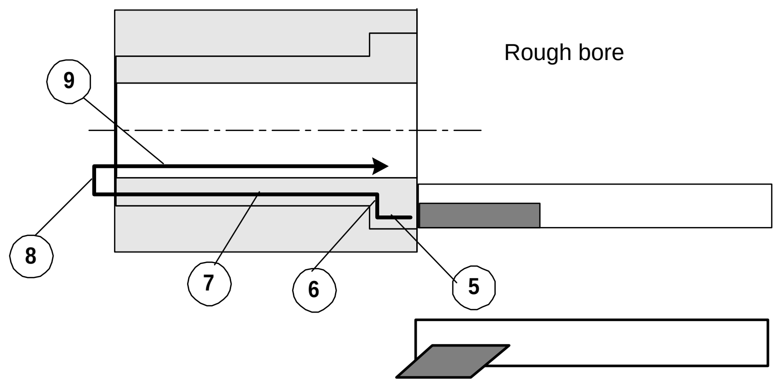


Worked Examples

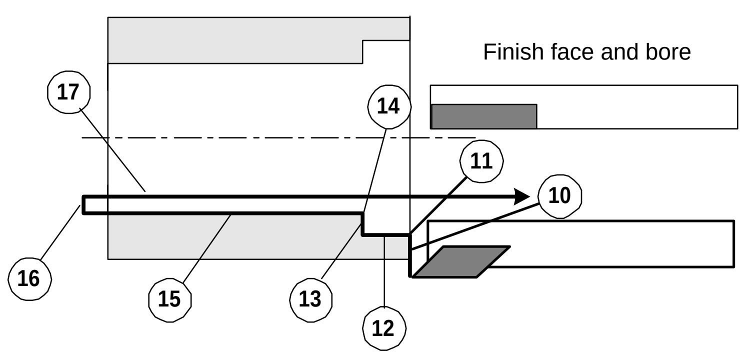
(2) X.55 G95 F.003
(3) G94 F200 Z.015
(4) X.34



(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



(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	

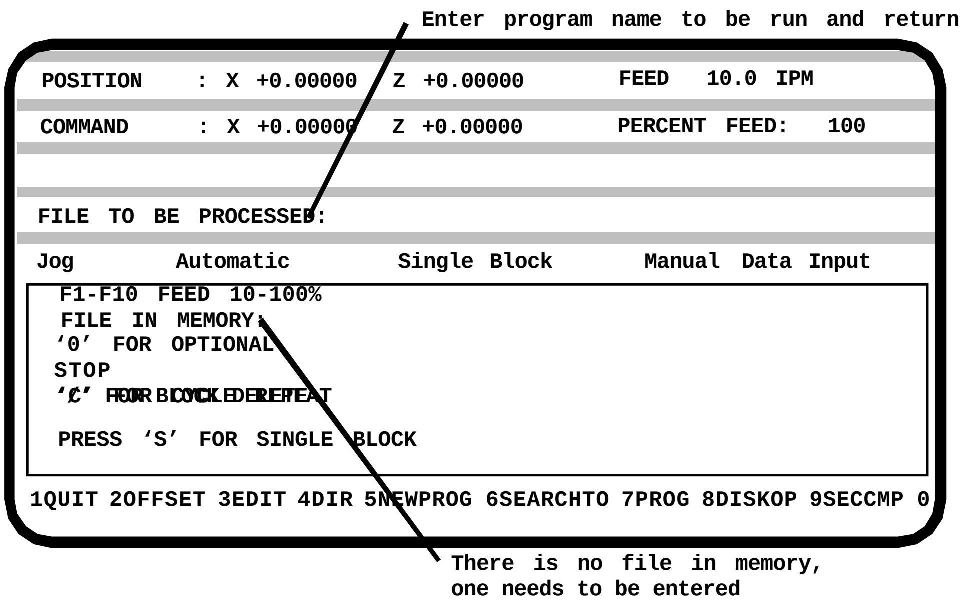


Automatic Mode

Running programs

In the Automatic mode the control displays the program that it is currently running. When the control is turned on there is no program selected to run and this space is blank.

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 recall tool offsets from memory refer to F10 in this section.



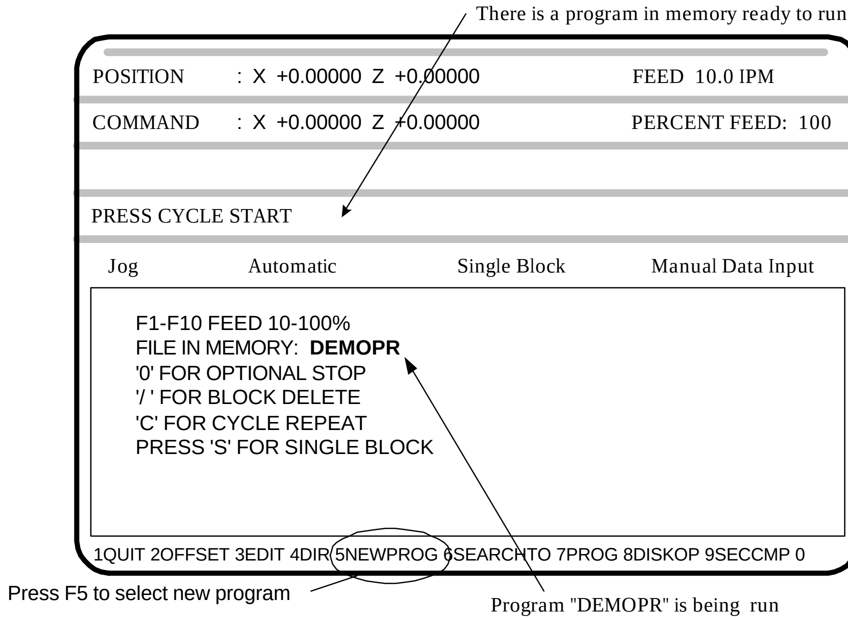
1. Running an existing program

If you have a program saved on the program disk it's name can be entered now and the screen will show that this is the file now in memory. When the cycle start button is pressed the program will be executed.

2. Running a different existing program (F5)

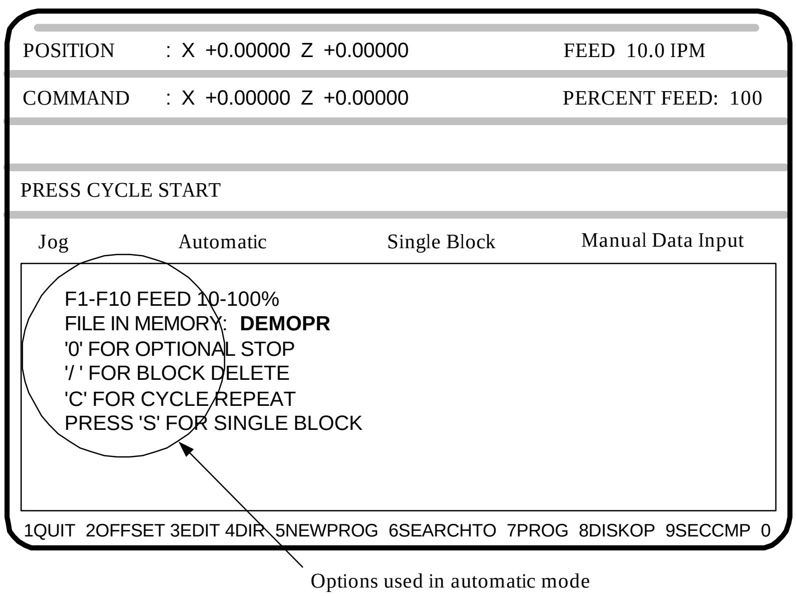
Once a program has been selected it stays in memory until it is changed. In the Automatic mode, F5 will delete the currently running program and ask for a new program name. You will notice that after F5 has been pressed the file in memory is blank. If you forget the exact name of the program that you want to run you can press F4 (directory) after F5 has been pressed. This F4 command will list all of your programs on the A: drive, 5-1/4", this is where your programs are stored.

Automatic Mode



Program Run - Single step -”S”

It is possible to run the program one line at a time. This is useful when running a new program for the first time. The control displays the next command to be executed before it is run. You can look to see what is going to happen before a mistake is made. To accomplish this get the control into the Automatic mode & input the file to be run. At this point do not push the cycle start yet. Press ”S” to activate the single block mode.



To toggle back to the regular automatic cycle press ”A” to turn off the single block mode.

Automatic Mode

The "F" keys have the following functions:

F 1	Quit	Go back to the Main menus
F 2	Offset	Adjust tool offsets, correct part size
F 3	Edit	Input and correct programs
F 4	DIR	When no file is in memory this will list all the programs on the user disk
	VER	With a program in memory it will verify and plot the program
F 5	Newprog	This will remove the program from active memory and allow a new one to be entered
F 6	Searchto	This enables the program to be started some place other than the start
F 7	Prog	Runs Calcaid programming system
F 8	Diskop	Disk Operations
		Erase programs
		Make a new System disk
		Make a copy of the user program disk
		Down and Up load programs from another computer over RS232
		Set communication parameters
F 9	Seccmp	Adjust values of secondary tool offsets and TNR compensation values
F10	Sp.fun	Special functions
		Parts Counter
		Set value of countdown counter for Continuous cycle
		Preset feed rate override before starting a program
		Store tool offset values with program in memory
		Load tool offset values from memory
		Set max spindle speed of machine

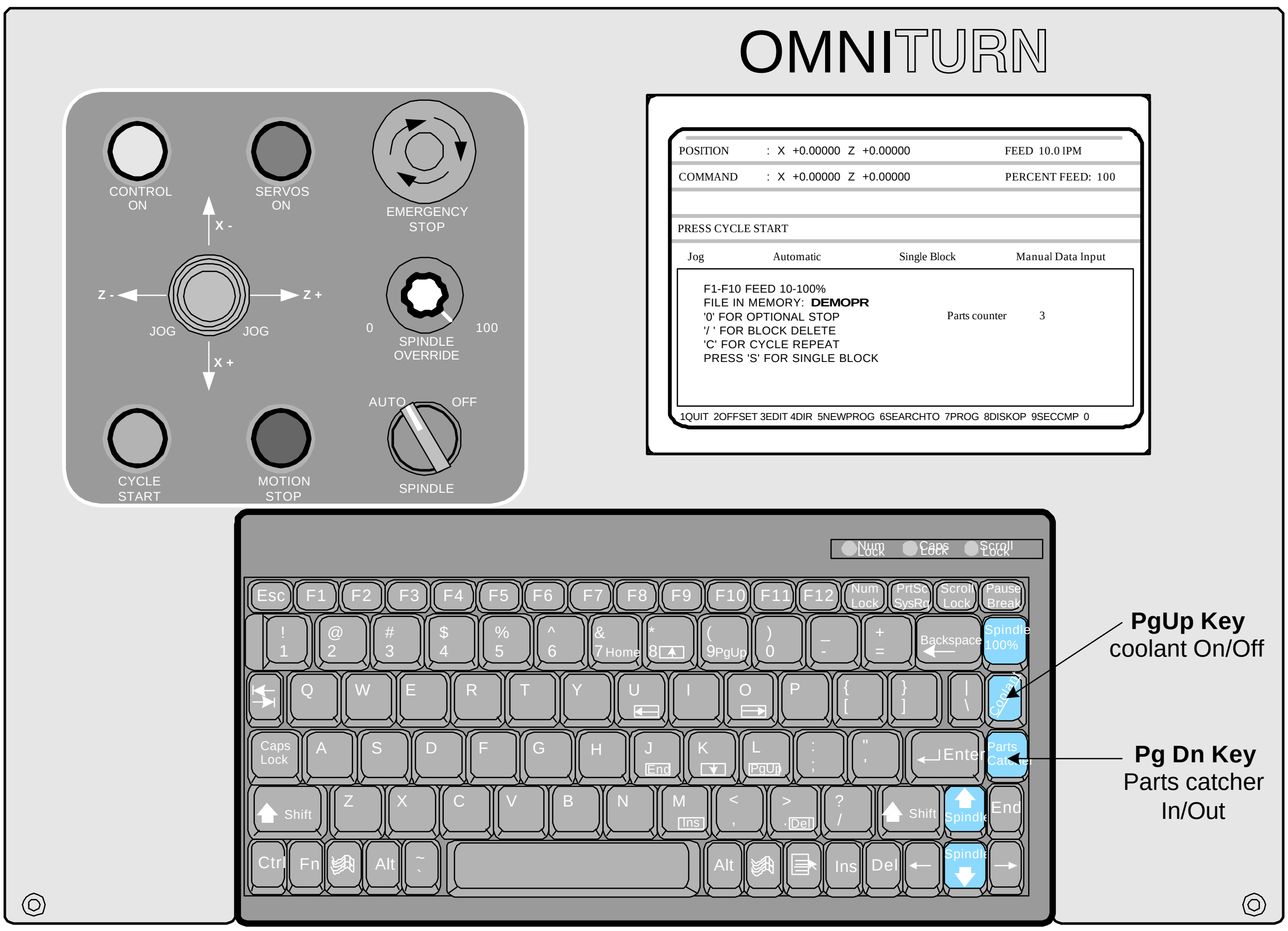
HOT keys on the keyboard while in the automatic mode

C	Continuous	The program will run continuously -This toggles on / off
O	Optional stop	This makes the M0l act as a stop program command -This toggles on / off
/	Block delete	With this active the control will skip over program lines starting with "/"
A	Automatic	The program will run from start to finish with one cycle start
S	Single block	The program will run one line at a time with each press of the cycle start
F1 -F10		Feed rate overrides. The function keys will adjust feed rates (only while program is in motion)
Pg Up		Coolant on/off (M08/09)
Pg Dn		Parts catcher Out/In (M25/26)

M function keyboard controls

Toggle M functions on and off with keyboard controls.

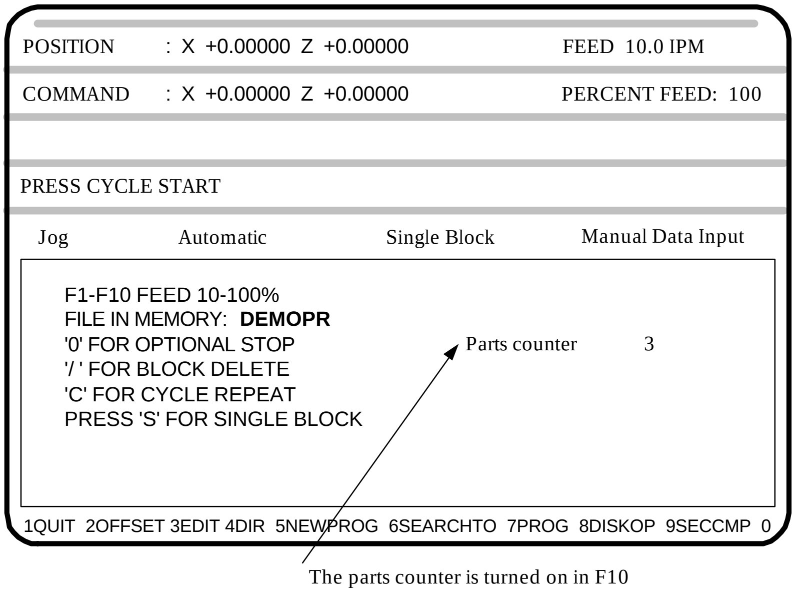
- -Press the key once to turn the function on, Press again to turn it off
- -Works only in Jog or Automatic mode



OmniTurn front panel Keyboard

Automatic Mode

Parts counter - P



To turn on a PARTS COUNTER first press F10, then Press P while you have this screen open. There will be no effect here. When you go to the Automatic mode main screen there will be a counter on the screen. The counter will count up each time a M30 or M02 is executed. To clear the counter come back into the F10 screen and press P again. See notes at the end of this chapter on Function keys, F10.

Program run -Optional stop activation - M01 -”O”

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

Insert an MO1 after a G92 statement for a new tool. This will help when running a new program to be sure that the tool offsets have been entered correctly. Once the program is tested you can turn off the stop and let the program run automatically.

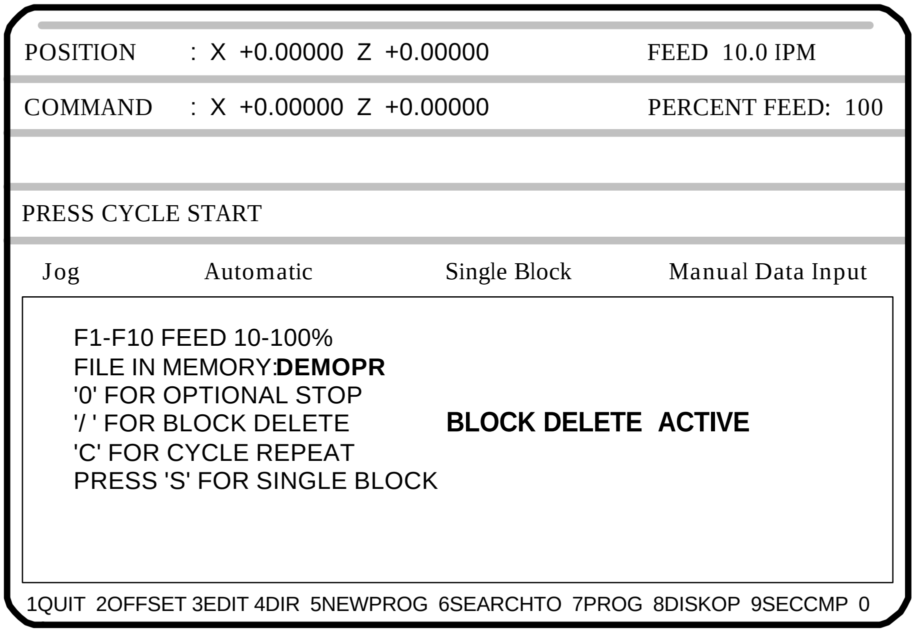
Have an MOl at the beginning of a program that is going to use an automatic bar feed or parts loader. This way, you can have the optional stop activated when you are setting up the machine. Once the cycle and program are proven correct, you can turn off the stop and let the machine run automatically.

Program run - Cycle repeat -”C”

This is useful for automatic bar feeder or automatic loader operations. When this is activated the program will automatically go back to the beginning of the program after a M30 is encountered and run the program again. The program will continue to run until it is stopped or the continuous counter is set. (see F10 in automatic mode). To turn it off, press ”C” again.

Automatic Mode

/ - Block Delete



The block delete is used to by pass lines of the program. Put the forward slash 'P' at the beginning of a line. When you want to skip the line just press the "P"' key in the automatic mode. Then the **Block Delete Active** text will show up on the screen. This is commonly put on the line with the coolant on command. This way you can turn the coolant off (skip over the coolant on line) by activating the Block Delete.

Creating a new program

There are a number of ways to create a new program. Here are a few:

-Use the text edit in OmniTurn. First a new program name has to be created. This is done by going into the Automatic mode and typing in the new name when the control asks "FILE TO BE PROCESSED". After the RETURN key is hit the control will answer "FILE NOT FOUND, PRESS ANY KEY TO CONTINUE". By doing this you have accomplished two things.

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

2. If there was no other program that had the new name then there was one created and loaded into the text editor. Once the new name is entered into the text editor, press F3 to enter the editor. The text editor will ask "PRESS F1 TO CREATE A NEW FILE, ESC TO ABORT". After pushing F1 the editor will provide a new blank screen to enter your program.

-**CAM system off line,** transfer a file via floppy or RS-232. Once they are on the OmniTurn program disk they can be run like any other existing program. Please refer to the section in DOS notes on the format.

-**RS-232 or Disk transfer.** Manually enter a program in a text editor on another computer, transfer as above. Once they are on the OmniTurn program disk, they can be run like any other existing program. Please refer to the section in DOS notes on the format. -Use **Calcaid in OmniTurn.** See the section on using Calcaid.

Automatic Mode

Function Keys

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

Function Keys - Automatic Mode - Program not in process

Following are the definitions of the function keys when the control is in the automatic mode and the program is not in motion.

F1 Exit Automatic mode, go to main screen

Pressing the F1 key will exit you from the Automatic mode. This is necessary to get to any of the other modes, ie. Jog or MDI.

F1 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. As an example, we will assume that tool 2 in the above example is a turning tool and is cutting apart .001” too large. So enter the offset change of -.001” for the X Diameter offset. When this value is entered you will notice that the total value of X has changed. This addition does not have to be done by the operator.

Automatic Mode

After selecting a number and pressing Return the screen will ask

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

X DIAMETER ADJUSTMENT:
Press Esc to exit offset adjustment screen

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

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

Z ADJUSTMENT:
Press Esc to exit offset adjustment screen

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

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

Automatic Mode

Notes:

1. The control will allow you to clear 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:
   .00005” in Z
   .0001” in X, (this is equal to .00005” on the radius)

3. For Tool Offset changes of more than .02” the control will ask the operator if this is correct. If not it will ask you to re-enter the correction This is a safety feature to ensure that you do not put in a large correction in error, ie 1” instead of .001”

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.

Program Editing -F3

F3 Edit, On screen text editor, used to change existing programs, or enter new ones

The editor is a full function text editor. In the OmniTurn you will be using only a small part of the capability of the editor. In the following description the most basic functions. If you want to learn more follow the instructions given in the HELP screens. (F1 while in the editor is active)

Starting the editor:

The editor is accessed from the Automatic mode by pressing F3 at any time. The program listed as ”FILE IN MEMORY” will be activated. If you want to work on a program you have to make it the active program. When you enter the Automatic mode it asks ”FILE TO BE PROCESSED”, type the file name that you want to edit and press RETURN. When the file name appears as the file in memory press F3. If you have to correct the program that is currently running just press F3.

• It will ask if you want to make a Backup copy. Making these backup copies is not required. If you are new to PC’s and DOS it is suggested that you make the backup copy so that if you loose the work that you have created it can be brought back.

• Either press ”ESC” for no backup or F1 to create the backup file

The editor can also be used to enter new programs.

• Get to the Automatic mode
• If there is a program in file memory press F5
• Enter the new program name when prompted: ”FILE TO BE PROCESSED”
• If the control does not tell the program is not found then you are using a name that already exists. Either pick a new name or plan on erasing the program that already exists with that name.
• Press any key to continue
• Press F3 to enter the editor
• The editor will ask if you really intend to create a new program, press F1 if you do, if not press ”ESC”

Exiting the editor and saving corrections made

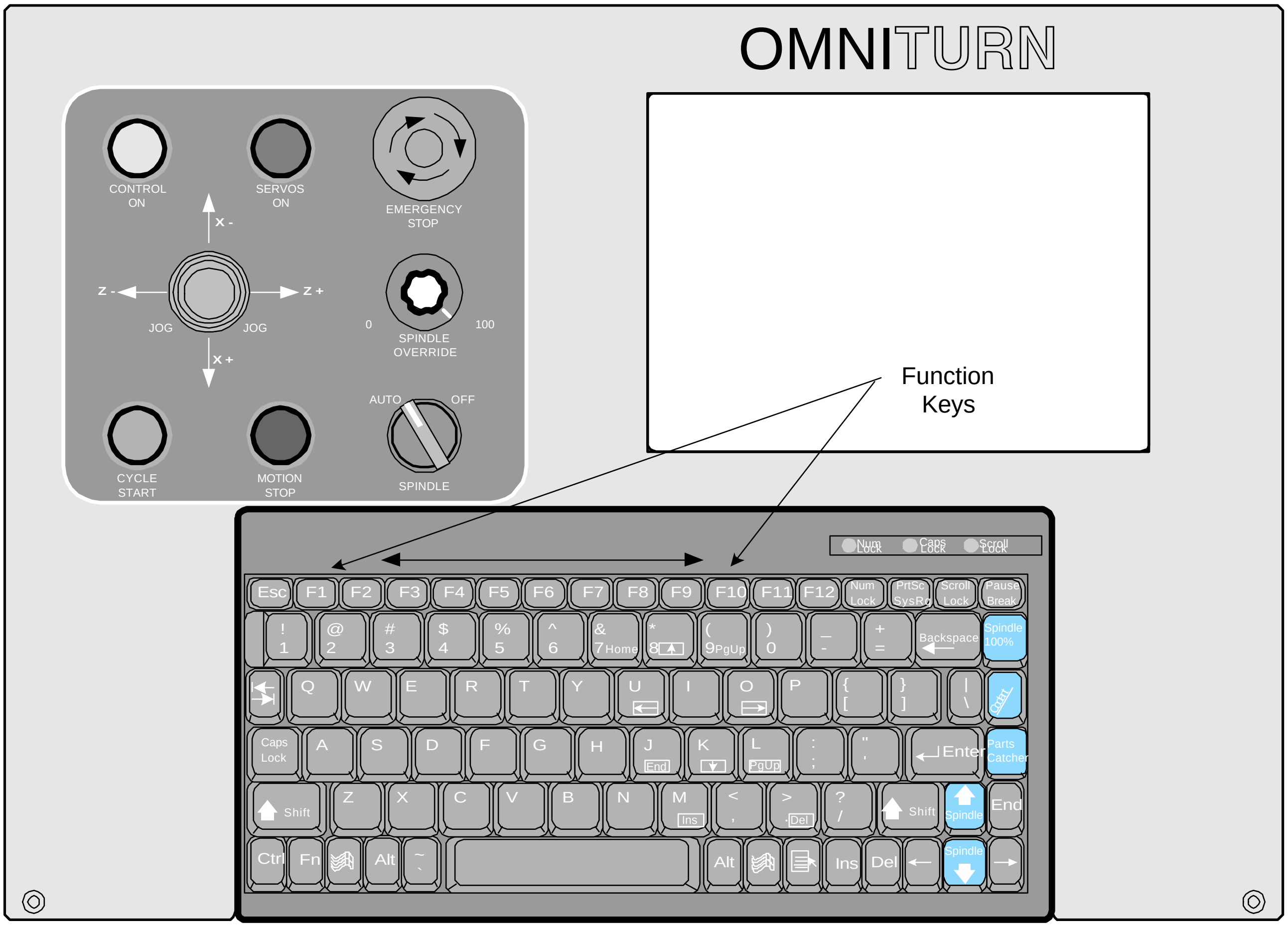
• Press F1 -this is for HELP
• Press F2 -this will exit the editor and save the corrections that have been made

Exiting the editor and NOT saving the corrections made

• Press F1 -this is for HELP
• Press ”ESC” -this will halt the automatic saving function of the exit routine
• Press F2 -this will exit the editor without saving the corrections

Automatic Mode

The Keyboard



OmniTurn front panel Keyboard

Number keys: Use the number keys across the top of the keyboard. The number keys on the right side are not active as numbers unless set with the number lock key. (located above the #7 key)

Cursor keys: These are the number keys on the right side of the keyboard. They will move the cursor around the screen as required.

Pgup: will move the cursor one screen back per hit
Pgdn: will move the cursor one screen forward per hit
End: will move the cursor to the end of the line it is on
Ins: will add spaces in front of the cursor location
Del: will delete the character at the cursor
Back Space: will erase the character to the left of the cursor

If you want to add lines to a program:

- Move the cursor to where you need to add the line.
- Type the new text and enter a RETURN if needed

Correcting existing text:

- move the cursor to the text to be corrected
- type the new text in
- delete the text being removed

Automatic Mode -Notes on the Editor

The text editor can help you manipulate and modify your programs. The editing functions are:

F4 -Block delete

This function is used to erase sections of the program. To use this function:

• move the cursor to the beginning of the code that you want to erase
• press F4 to start the text marking
• move the cursor to the end of the text that you want to erase. The text that you are going to erase is now being marked.
• if you want to
the code and cancel the delete command press F5
• to complete the erase press F4 again

F3 -Block copy

This function is used to copy sections of the program.

• move the cursor to the beginning of the code that you want to copy
• press F3 to start the text marking
• move the cursor to the end of the text that you want to copy. The text that you are going to copy is now being marked.
• if you want to unmark the code and cancel the copy command press F5
• to end the marking of the code to be copied press F3 again.
• then move the cursor to the location that you want the code copied to, then press F3
• you can copy the code more than once, move the cursor again and press F3 again
• to end the marking press F5

F9 -Find, F10 -Replace

This function is used to find text in the program and then replace it with new text. An example for using this is to change the rapid travel feed rates. Sometimes it is advantageous to lower the rapid feed rates when you are working with a new program and expensive tooling. Before running the program for the first time you could change the F300 moves to F100. Run the program and prove it out, then change the F100's back to F300 for production

• go to the text editing screen
• move the cursor to the beginning of the program
• press F9, at the top of the screen you will see FIND ”.” type the text that you want to replace. In the case of the example you could type F300
• press F10, at the top of the screen you will see REPLACE ”. ” type the text that you want to put in place of the found text. In the case of the example you could type F100
• press RETURN get back to the edit mode
• press F10 to find and replace, repeat pressing the F10 key until you have gone through the entire program

Renaming programs

In the editor you can duplicate a program and give it a new name. This is very useful! IE: if you want to make changes to an existing program and still have the original to fall back to if the changes don't work the way you want, or if you have a family of parts that use the same basic program with only minor changes. With renaming you could make a custom program for each part without having to type the entire program again.

• make the program active in the automatic mode
• press F3 to go to the editor
• press ESC to bring the text to the screen, then press F1 to get to the HELP screen
• at the HELP screen press F5 for RENAME. You will see the original file name at the top of the screen with B: in front of the name. An example of this would be ”B: FILENAME”. Use the arrow keys to move the cursor to the right of the B:. Then type the new name of the file and delate the old one. The new name could be ”B:NEWFILE”.
• after the name has been typed press RETURN. This will take you back to your program.
• when you are done working with the program, save and exit the editor as you normally would, press Fl, then F2. Now you will still have the old program, and a new one.

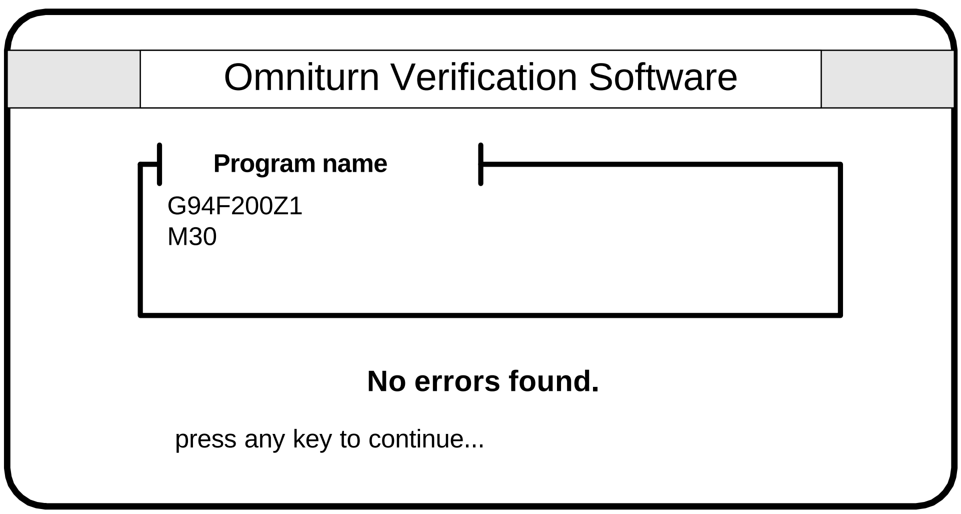
Verification (F4)

This function will check the current program. The verification does a few things automatically:

1. Check syntax and look for possible errors, then allow corrections
2. List all the tools used
   List all secondary offsets used and their current values
3. Show estimated cycle time
4. Show graphically the tool path created by the G code program

Running Verification
To use the verification software you must first make your program active in the Automatic mode. Once the program is active you will notice that the F4 indication on the bottom of the screen changes to VER. Press F4 and the verification software will be called up.

1. Syntax and program check.



The first operation performed by this section is the checking of syntax (is the program grammatically correct) and a screen for possible gross errors (like trying to move Z -50). If there are no errors or comments then the screen will appear as above.

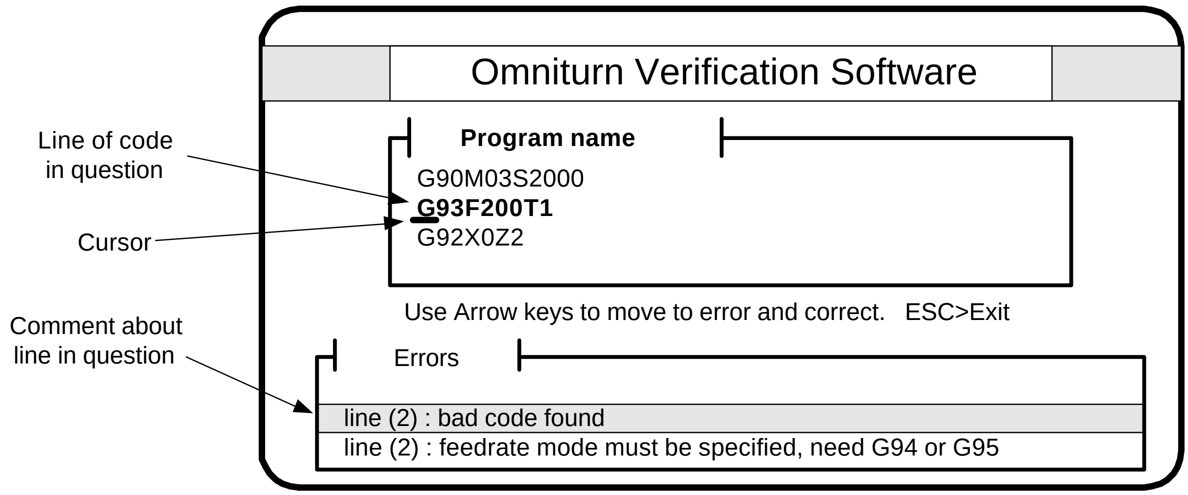
Error comments and corrections

The error comments are just that, comments.

There are times that the "errors" it finds are moves that you intended to do. Use the verification software as a guide, not a rule. When the software was developed it was intended to try and screen out all sorts of possible errors, so this causes it to catch more good lines. Make corrections to your program with care.

If the verification finds lines of your program that it does not like it will show the line of code in the upper box, highlighted. In the lower box is a description of the possible problem.

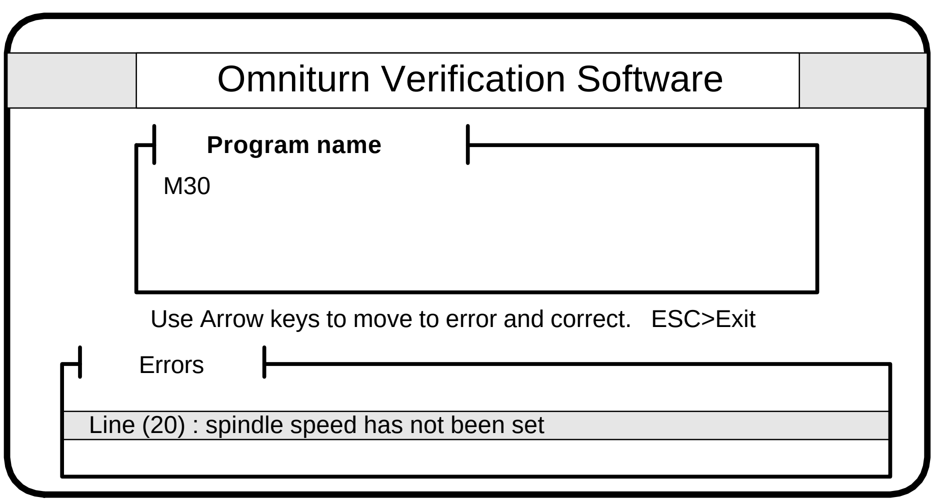
Verification -F4



Making corrections:
If you find an error in your program it is possible to correct it now. Use the left / right arrow keys to move the cursor to the point in question. In the above example the G93 should be G94. Arrow over to the 3 and then type a 4. When you are done with the correction you can go on to the next comment. When you are done with all the corrections the checker will go back and run the check one more time.

Going to the next comment:
If there are more than a few comments about the program you can use the up and down arrows to move to new comments.

Missing spindle speeds:
If the program does not have a spindle speed the OmniTurn will comment. This is because a spindle speed is needed to estimate the cycle time. If you do not want to add a spindle speed to the program you can force a cycle time estimate in the verification when it asks for a spindle speed.

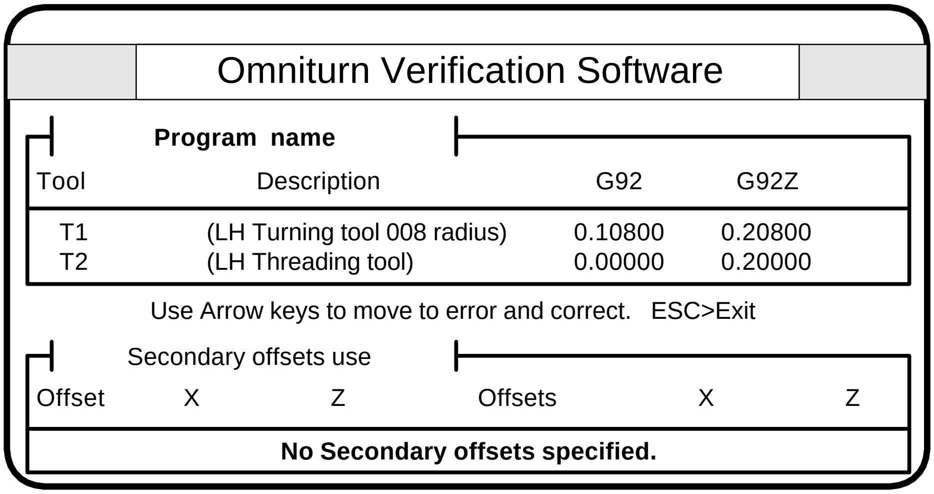


Exiting Syntax checking: Press ESC.

2. Listing tools and offsets

The next screen is a list of the tools used and some information about them:

Verification - F4



This screen shows a lot of information:

• It lists the tools to be used with their G92 starting points. This makes setting up a program easier. No longer do you have to look through long programs to find the tools and their starting values.

• There is also a description of the tool. The verification software will show any comment that appears on the same line with the tool call in the program. In the above program the are comments on the line:

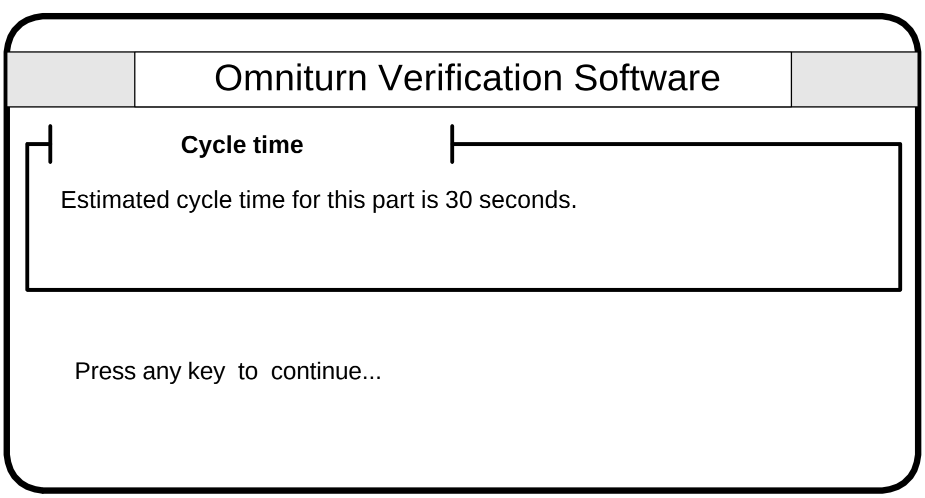
T1 (LH Turning tool 008 radius)
G92X.108Z.208

It is good practice to show the type of tool, the size of the radius used, and which direction it points

• The secondary offsets are also shown. The current value of these offsets are listed. To make changes to these offsets you have to go back to the automatic mode and then F9.

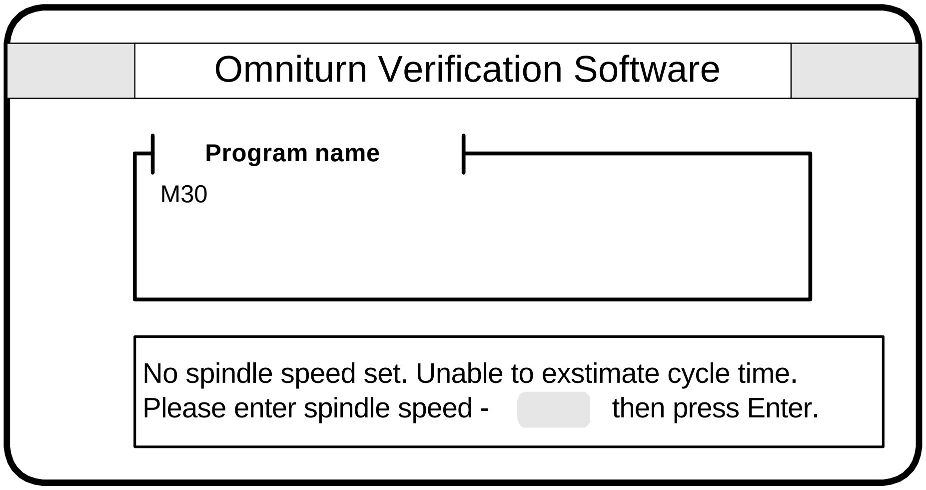
3. Estimating cycle time

Now the cycle time is estimated.



Verification -F4

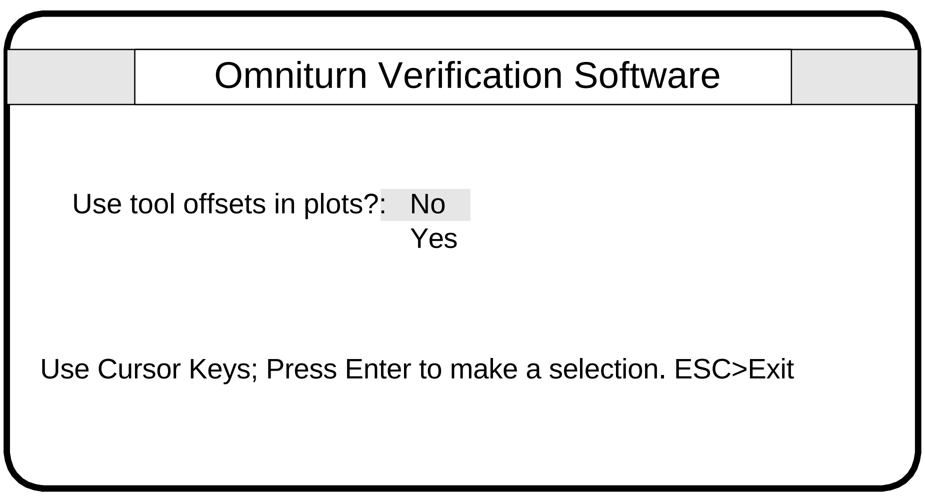
If the program does not have a spindle speed you are now given a chance to input a single speed for estimating the cycle time:



4. Tool path graphics

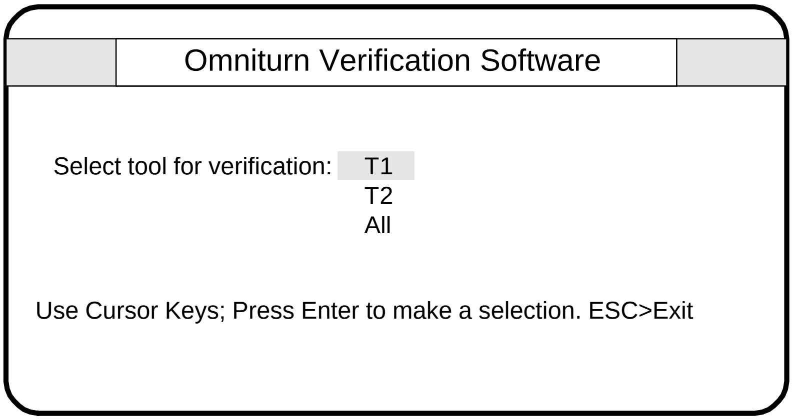
Before you show the graphics you have to chose to show them with or without the tool offset locations. If you are running this software on your OmniTurn and the tools have all been set, then enter yes. If they have not been set or you are running the software off-line, then say no. Use the Up / Down arrow keys to highlight your selection. Then press Enter.

Running the graphics with tool offset locations helps pickup possible coalition conditions between tool paths and the work piece.

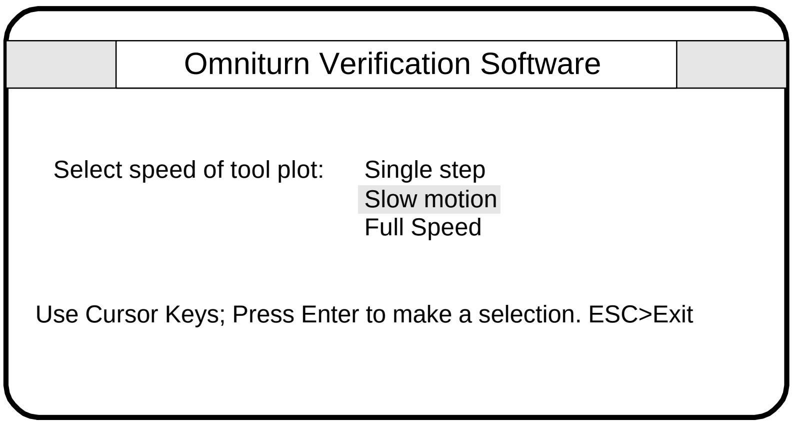


Verification -F4

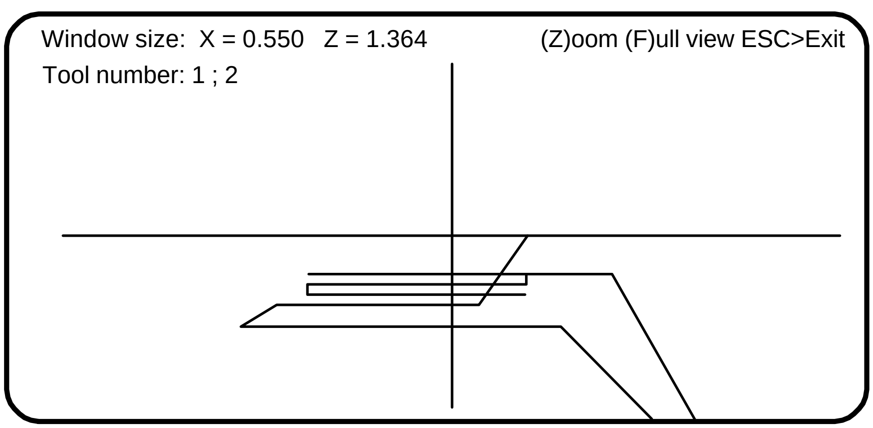
Then select the tool that you want to see. Or all. Use the Up / Down arrow keys again. Then press Enter.



Select the speed of the graphics display. If you choose single step you can advance through the program using the Enter to go to the next move.



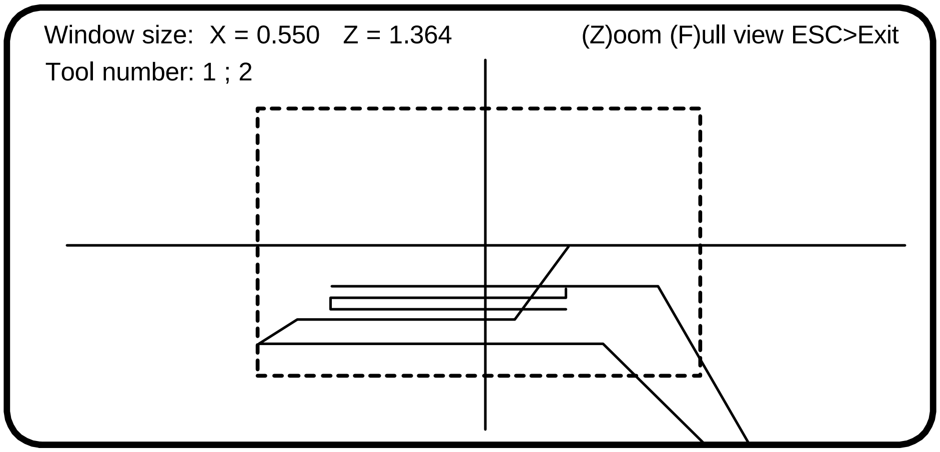
Then the graphics are shown:

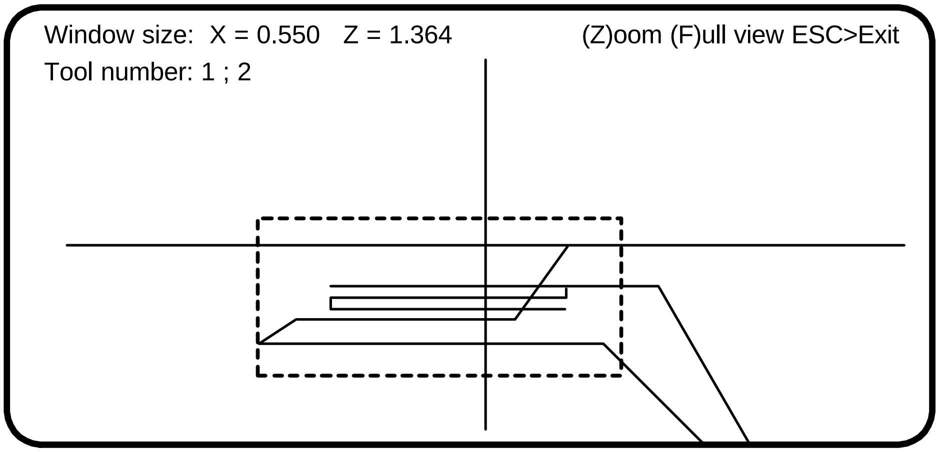


Zooming You can look closer at a section of the graphics by zooming in. Press the Z key. A window locator will appear. If you want to get much closer than one zoom, press it more than once, the window will get smaller and smaller.

Move the window over the area of the screen you want to zoom. Use the arrow keys to move it around.

Verification -F4





Notes about the graphics:

The graphics will show the path created by the G code in the program. If you have tool nose compensations created in Calcaid the graphics will be showing the path of the center of the tool nose radius. So the location of paths from tool to tool may overlap incorrectly since the tool nose radiuses will be different for each tool.

Automatic Mode

F4 Directory, list all of the programs on the disk -in use when a program is not active

This function key is not always active. Once you have selected a program to run and there is an active file in memory this function is deactivated. In you have yet to select a current program this key will bring up a list of programs available on the program disk. This will list all of the programs you have stored on the A: disk, (the 3 1/2” floppy). Once you have reviewed the programs available press ”Esc” to return to the Automatic mode screen.

F 5 Different file, used to change the running program to a new one

This function key is used to change the program that is in the current file and allow the operator to enter in a new program name to be run. After F5 is pressed the control will ask ”FILE TO BE PROCESSED”. Type the new name in and Return. Then press the cycle start and the new program will run. Be sure that the tool offsets are set before running the new program. This key is not always active. If there is no active file in memory F5 does not appear.

F 6 Search to, used to start the program at another location other than the beginning

This function key will allow you to start the program at a point other than the beginning. This is very useful for running new programs and skipping over sections of program that you do not have to check. It is intended for skipping to a tool change, this is an easily noted beginning section of code. F6 can also be used with programs that have line numbers.

After the F6 command is pushed (while in the Automatic mode) the screen will ask ”SEARCH TO?”

```
POSITION   : X  +0.00000  Z  +0.00000          FEED  10.0 IPM
COMMAND    : X  +0.00000  Z  +0.00000          PERCENT FEED:  100

SEARCH TO ?
 Jog          Automatic          Single Block          Manual Data Input

     F1-F10 FEED 10-100%
     FILE IN MEMORY:DEMOPR
     '0' FOR OPTIONAL STOP
     '/ ' FOR BLOCK DELETE
     'C' FOR CYCLE REPEAT
     PRESS 'S' FOR SINGLE BLOCK

1QUIT 2OFFSET 3EDIT 4DIR 5NEWPROG 6SEARCHTO 7PROG 8DISKOP 9SECCMP 0
```

After typing in the text to search to press the Return key. Then press the cycle start button to start the slide.

As an example, if you are want to skip to Tool #2 type in T2. The control will skip the code before this line and start the program with the T2 command. If you are using an OmniTurn with spindle control be sure that after your tool changes you have a spindle on (M03 or M04) and an S command. The F6 - Search to does not read the previous commands and you will have to be sure that the spindle is running. If you use line numbers in your program, it is possible to skip to these instead of the tool changes.

F7 Programming system, Calls up the Calcaid programming system See the section on Calcaid. Chapters 9 and 10 in this manual.

F 8 Diskop, Calls up a list of Disk operations

The F8 key will call up the following screen:

File Handler

Would you like to:
1. Make a new boot/program disk
2. Erase a file from the boot/program disk
3. See a directory of files on the boot/program disk
4. Download a program from another computer
5. Upload a program to another computer
6. Make a backup copy of the boot/program disk
7. Set communications parameters
8. Exit file handler

By entering a number you will be prompted through the requested task. Listed below is a description of each option.

1. This will help make a new boot/program disk, the larger of the two disks. It stores all of your programs.

2. This will prompt you through erasing programs that you no longer need.

3. This will list all of your programs on the larger disk.

4. This will prompt you through downloading programs from another computer. This is used if you write programs on another computer and want to transfer it to the OmniTum. You will have to run a cable between the RS-232 ports on both computers. (See Section 7 for cable configuration) The communications parameters are set by option 7 on this screen.

5. This will send programs from the OmniTum to another computer via the RS-232 port. (See Section 7 for cable configuration)

6. This will copy the boot/program disk completely. All of your programs will be duplicated. Take this disk away from the machine and store it in a safe place that is dry, dark (no direct sunlight), cool (room temp.), and no magnets near by. This should be done after you have written and tested a program. If there is a problem with the disk in the OmniTum you can replace it with your backup and continue uninterrupted operation.

7. This will display the communications parameters and allow you to change them.
   The communications parameter options are:
   Baud rate (2400, 1200, 300)
   Parity (o,e,n)
   Data bits (7,8)
   Stop bits (1,2)

8 . This will exit the file handling screen and bring you back to the Automatic mode screen.

F 9 Secondary tool offset screen, used to modify secondary tool offsets

This function will call up the secondary offset table. There are 32 offsets available and 32 tool nose radius compensation offsets. Please refer to the section on secondary offsets for their use. Notice that this differs from the offset table screen in that almost all of the offset values are set to 0.00000. Secondary offsets are corrected like offsets.

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

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

First: Select a secondary offset number
Next: Use the return key to enter past the X and Z inputs.
Then: Enter the value of tool nose compensation, IE .007 and then press ESC

Changing a TNR compensation value:

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

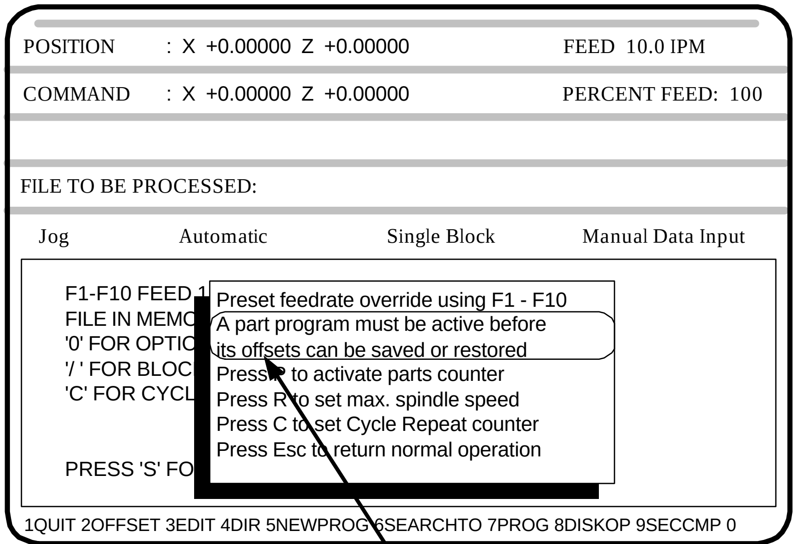
Clearing secondary offsets to Zero.

It is possible to clear all of the secondary offsets by pressing C when asked for a secondary offset number. This will set the entire table to zeros.

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

Automatic Mode

F 10 Special Function, used to call up a list of special functions.



If a program is active the promt will be:
PRESS L TO LOAD OFFSETS
PRESS S TO SAVE OFFSETS

This screen will allow you to:

• Save and recall tool offset tables.
If you are using a tooling system that allows you to remove and replace tooling exactly this function is useful to you. Also if you use the same tool set for a number of different programs. When you save your program it is possible to save the tool offsets as well. The saved offsets are put on the A: disk with the programs.

The screen above is shown if you do not have a part program active. Offsets can not be saved or re-called. If you have a part program active the control will allow you to save and load the offsets from memory:

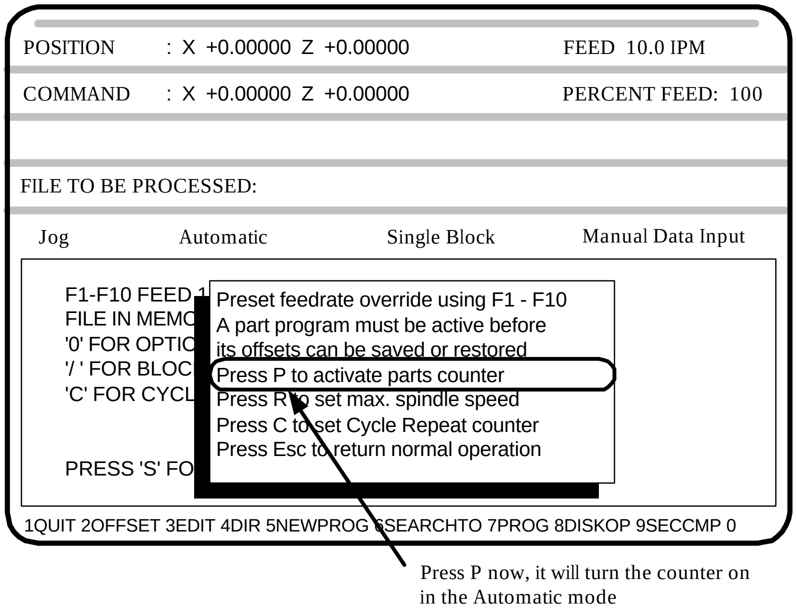
L -press L to load the offsets from memory
S -press S to save the offsets to the disk

• Modify the feedrate :
To be used in a program before you start running it. This eliminates the need to race to the function keys after you press the cycle start. When you are running a program for the first time you might want to lower the feedrate to only 20% by pressing F2 so that you can watch the motion of the tool before you cut material.

• Set the number of:
Cycles the Automatic mode will run before stopping when you set the cycle repeat to ”C”. This is good for use with a barfeed. As an example you could set the machine up and tell it you need 20 pieces. The OmniTurn will make the required amount and then stop.

• If you have the infinitely variable spindle speed control this will let you tell the control what spindle speed the machine is set at. That way the control will output the speed requested in your program without having to figure any ratios out. This is covered in greater detail with the documentation on the option.

Automatic Mode

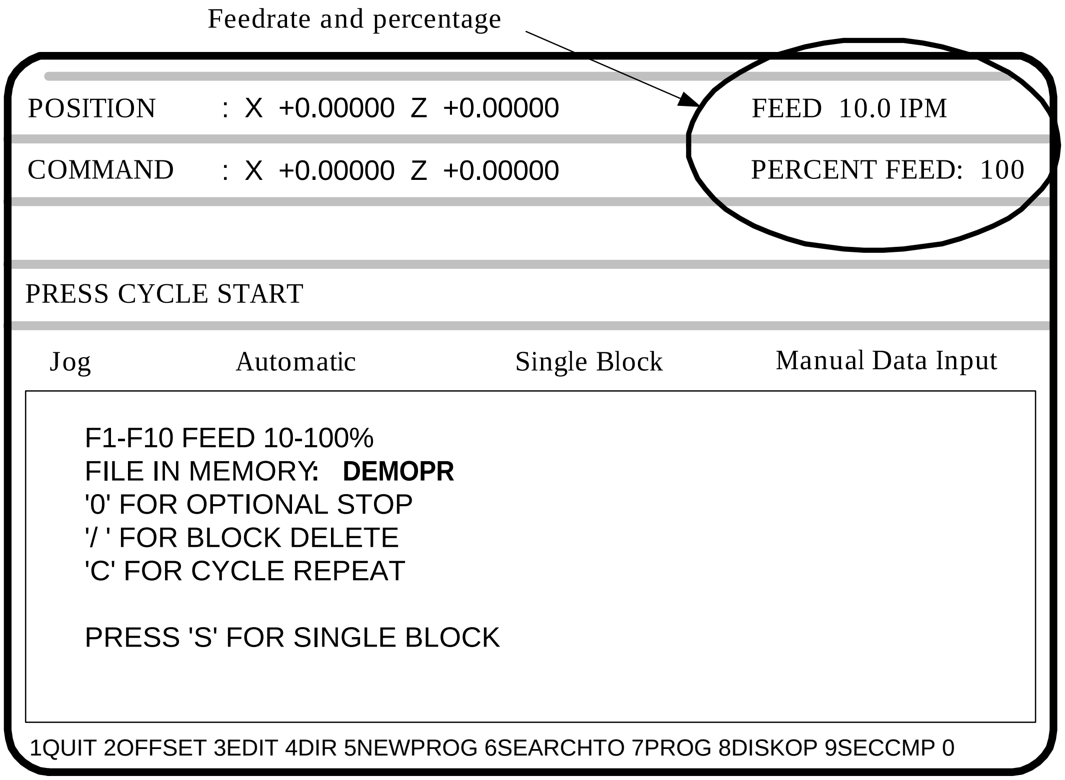


• **Turn on a PARTS COUNTER**: Press P while you have this screen open. There will be no effect here. When you go to the Automatic mode main screen there will be a counter on the screen. The counter will count up each time a M30 or M02 is executed. To clear the counter come back into the F10 screen and press P again.

Feedrate override

Function Keys - Automatic Mode - Program in process

When the program is running it is possible to change the feed rates. The function keys will select a percentage of the original feedrate. F1 = 10%, F2 = 20%, ... F10 = 100%. IE if you push F1 while the program is running the feedrate will drop to 10% of what ever you have set in the program. The feedrate and what percentage of that feedrate is being run is displayed on the automatic mode screen on the upper right corner of the screen.



If you want to change the feedrate before you press cycle start select F10. This will allow you to preload a percentage before the program is started.

OmniTurn - Trouble shooting guide

The Yellow power light won’t come on

The yellow power light indicates that the 12V power supply is working. The 12V should come on when the system is turned on and the computer is working.

- 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 control box.
- 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 there is a problem with the drives or their control system.

- Check that the Emergency stop is released
- Check the relay on the connect card in the control, be sure it is plugged in

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 is a problem with the monitor; check the cable on the back of the monitor. Make sure 12V cable from CRT, is attached to J7 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 Emergency Stop button on CNC; be sure it’s reset
- Check the two thermal overloads inthe control. These are located on the rear panel of the CNC. It can take a few minutes for the overload to cool enough to be reset. Press “Esc” and try to jog again.
- Turn the control off, wait 15 seconds, turn the control on again. This will reset the Servo driver card. If the control shuts down again when you try to jog, go to the next step.
- Check for loose motor cable; set power off, then disconnect and reconnect.

- 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)

Positioning Problems:

The slide has problems with repeating a size

Using Ctrl-C & Ctrl-H to diagnose repeatability problems

These two tests will determine if the problem is in the control or the axis motor without needing any additional tooling or indicators.
NOTE: before starting this check to see that all connections on the MC-2 and servo cables are clean and have no obvious defects. Check to see all pins on the axis cables are level and not pushed back.

Ctrl-C and Ctrl-H are diagnostic tools to help determine the cause of repeatability problems.

If Ctrl-C does not indicate zero, the problem is internal to the MC2.
If Ctrl-H does not indicate zero, the problem could be the motor encoder or encoder circuitry on the MC2.

- Establish Home as normal (press 9, then Cycle Start), then move slightly away from Home and establish Home again (press 9, then Cycle Start) to clear all registers.

- Load “smaltest”, which is a test program which makes a one inch circle inside a one inch square centered one half inch from machine home. If “smaltest” is not available just run the following program:

```
g90g94f300
x1
z-1
x0z0
m30
```

- Set Cycle Repeat and let it run twenty times. Go to Jog Mode when program is done.

- Press H key, then X, then Z. Now press and hold Ctrl Key, then press C Key. X and Z readings should be zero. If the axis you’re having trouble with is not zero, the problem is probably a bad motion control (MC2) card.

- Jog the slide a little ways from home, then Establish Home (press 9, then Cycle Start). Press and hold Ctrl Key, then press H Key. X and Z readings should be zero. If the axis you’re having trouble with is not zero, the problem could be a bad MC2 card or a bad motor encoder.

- If the either Ctrl-C or Ctrl-H reports non-zero, press E-Stop to shut servos off, then swap motor connections at back of CNC, and test again.

- If same axis reports non-zero, the problem is a bad MC2. If opposite axis reports non-zero, the problem is bad motor encoder.

A repeatability or size problem which persists despite good readings with Ctrl-C and Ctrl-H indicates a mechanical problem: loose coupler, excess backlash, entire slide loose on ways, bad spindle, tooling, etc.

Review some of the suggestions on the next page for help in determining the mechanical cause of the repeatablilty or size problem.

Positioning Problems:
The slide has problems with repeating a size

- The part is moving, check your work holding fixture
- Be sure the tooling is held tightly
- If you have an **attachment**, see if the slide is loose on the lathe
- The ball screw nut has come loose in it's housing
- Loose encoder at the end of the axis motor
- Loose coupling between the Servo motor and the ball screw for axis movement
- Bad motor-tach-encoder; bad cable; bad motion control card

To see if it is a mechanical problem position the slide close to the spindle, mount an indicator on the slide so it touches the headstock, zero out the indicator and then push and pull the slide with servos on. The indicator should show some movement as you push and pull on the slide. However it should return to Zero when you let go. If the slide does not come back to Zero then there is something loose, (Slide, Ballscrew nut, Ballscrew taper roller bearing, etc).

To check if the slide for repetition try running a simple program that will show the type of error that you are getting clearly. Run the program in "cycle repeat":

```
g90g94f300(TEST PROGRAM FOR Z AXIS)
t30(Move the slide so indicator is 1 inch away from headstock)
x0z1
z.1
f20z0 (Creep up on the indicator)
g04f1(Read the indicator during the dwell)
f300z1
m30

g90g94f300(TEST PROGRAM FOR X AXIS)
t31(Move the slide so indicator is 1 inch away from headstock)
xlz0
x.1
x0f20
g04f1(Read the indicator during the dwell)
f300x1
M30
```

After running the program and studying the way the 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.

Computer won't complete start-up

- You get a message that the OmniTurn is "Initializing" and on the next line there is a number: ie. 255.

- This indicates that the OmniTum Motion card is not found. This card has either come loose and needs to be resettled or replaced. See notes on replacing system cards.

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 Emergency stops are reset, on control face and operator station
- Turn the "spindle override" pot on the control face full CW
- Turn the spindle switch on the control face to "AUTO"
- Check that the pin on the head of the hardinge to lock the spindle is pulled out
- Push the red button on the spindle drive box.

This is a reset for the inverter used to vary the spindle speed. This could be tripped for a number of reasons:

- Low or high voltage
- The duty cycle is too much; cycle time is too short and too often.
- Noise from coolant pump or other contacts
- Acceleration and Deacceleration are to short. *Parameters can be set to change these: call the factory (541.332.7004)*
- Spindle drive box must be turned on (attachments only)
- Cables to the spindle drive box must be plugged in
- Check that MISC cable from the OmniTum CNC to Spindle Cabinet is connected
- Check that Operator Station Cable is connected to Spindle Cabinet

OMNITURN MOTOR REPLACEMENT INSTRUCTIONS

Removal:

1. You need to have the room to access the motor coupling on X or Z axis in order to change the motor, so first move the slide away from the motor.

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.

3. Loosen the cap screw holding the clamp on the coupling on the motor side. Turn the coupling as required to access the allen screw.

4. Remove two 3/8-18 cap screws that hold the motor mount and motor to the base. You may have to lightly tap the motor mount to remove it from the machine, as it is pinned with dowel pins.

5. 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.

7. 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 1/2 turn apart; that is 1/2 turn, or 0.100” from “home”. If X-Axis, push the slide all the way down, against the stop; if Z-Axis, push the slide all the way to the right, against the stop.

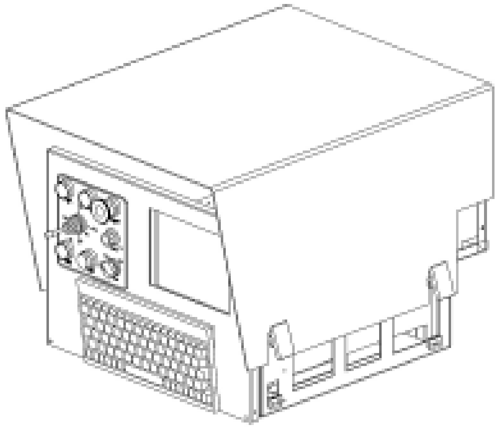
8. Attach the motor mount to the base, slipping the shaft into the coupler. Don’t let the shaft turn much. Tighten the motor mount to the machine before tightening the clamp on the coupler.

9. Replace the sheet metal as required.

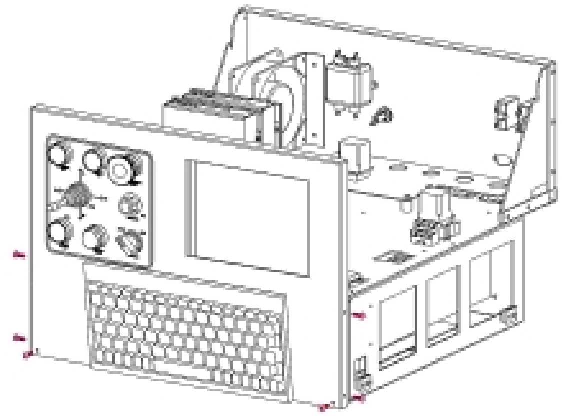
10. After re-assembly, jog the axis to both ends and verify that the pointer will go just slightly past “12” and “0”, then establish Home as usual. If the pointer does not indicate ”0”, loosen it and move it to zero.

CAUTION - YOUR TOOL OFFSETS HAVE CHANGED. The slide will not home *exactly* where used to: If you are set up on a job you should re-set all offsets.

To Disassemble the CNC Control

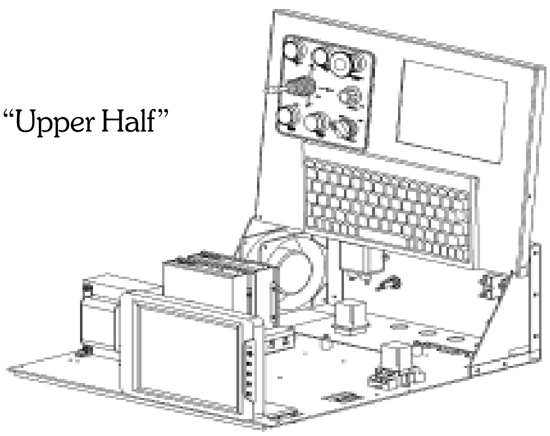


Un-plug power cord, then remove the blue cover.

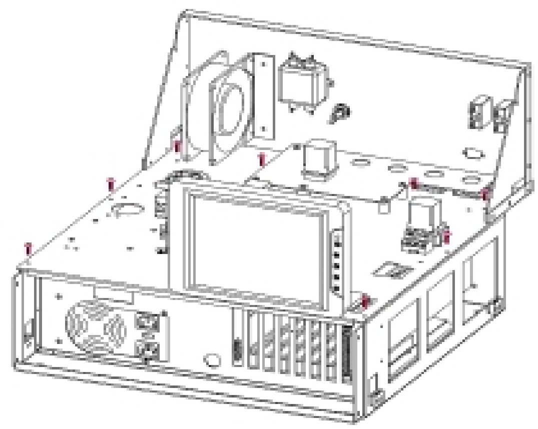


Remove six screws holding front panel.

"Upper Half"

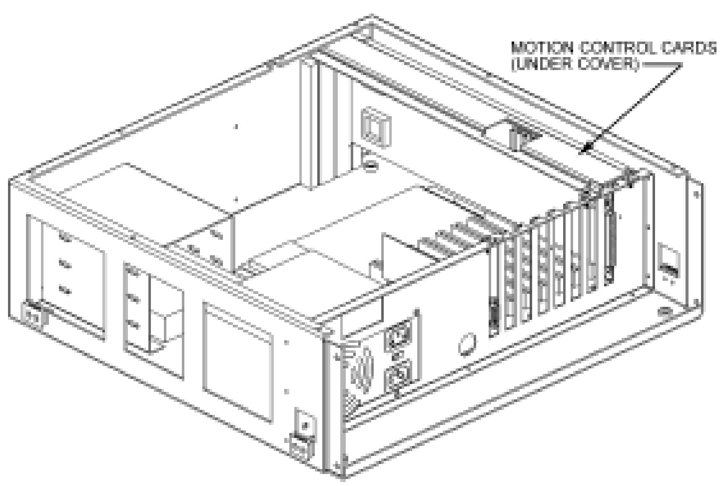


It is not necessary to disconnect cables to front-panel. Stow panel in notches at rear of top chassis

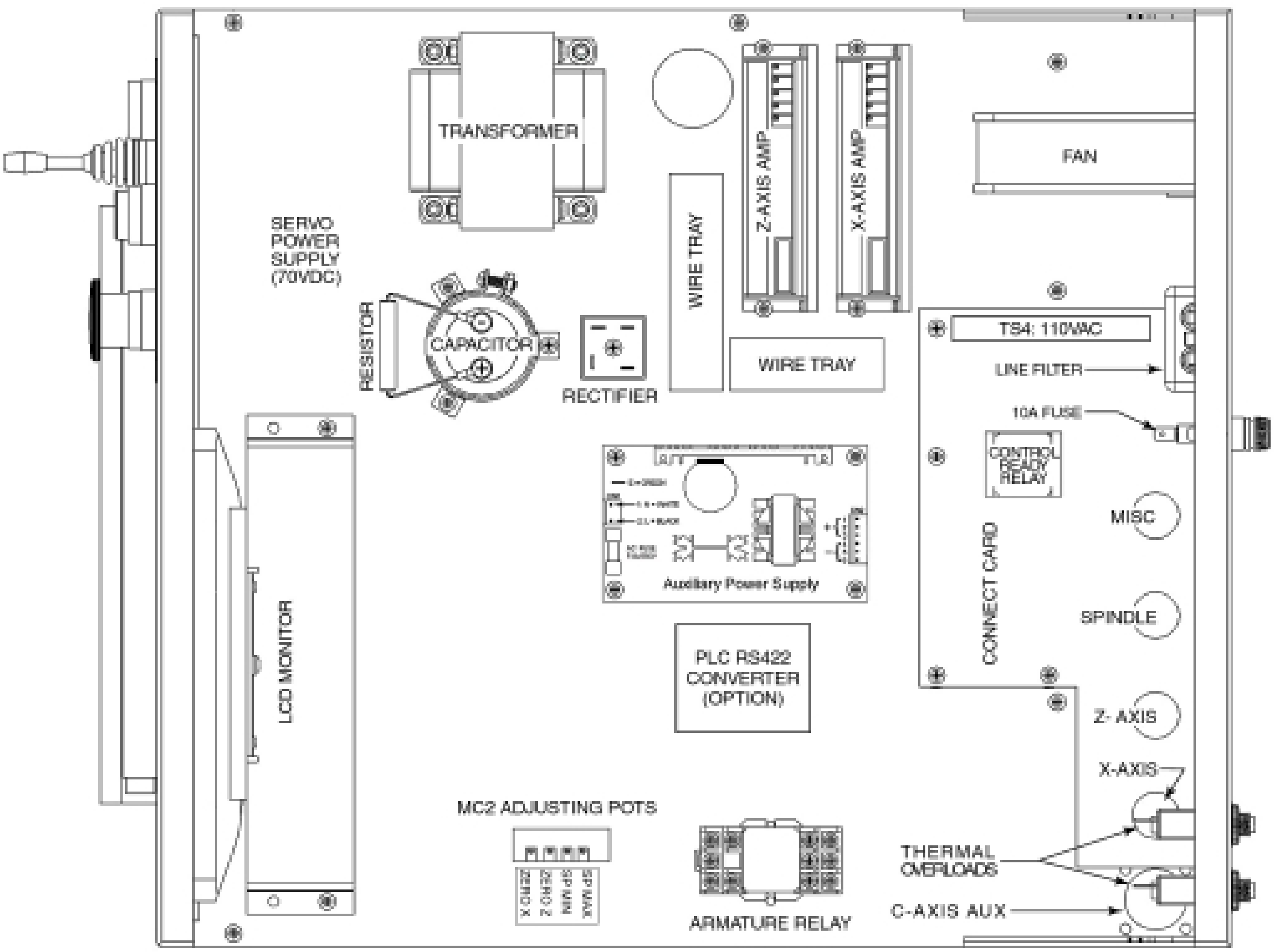


To gain access to the computer, disconnect all cables at rear of CNC, remove eight screws and set chassis aside.

"Lower Half"

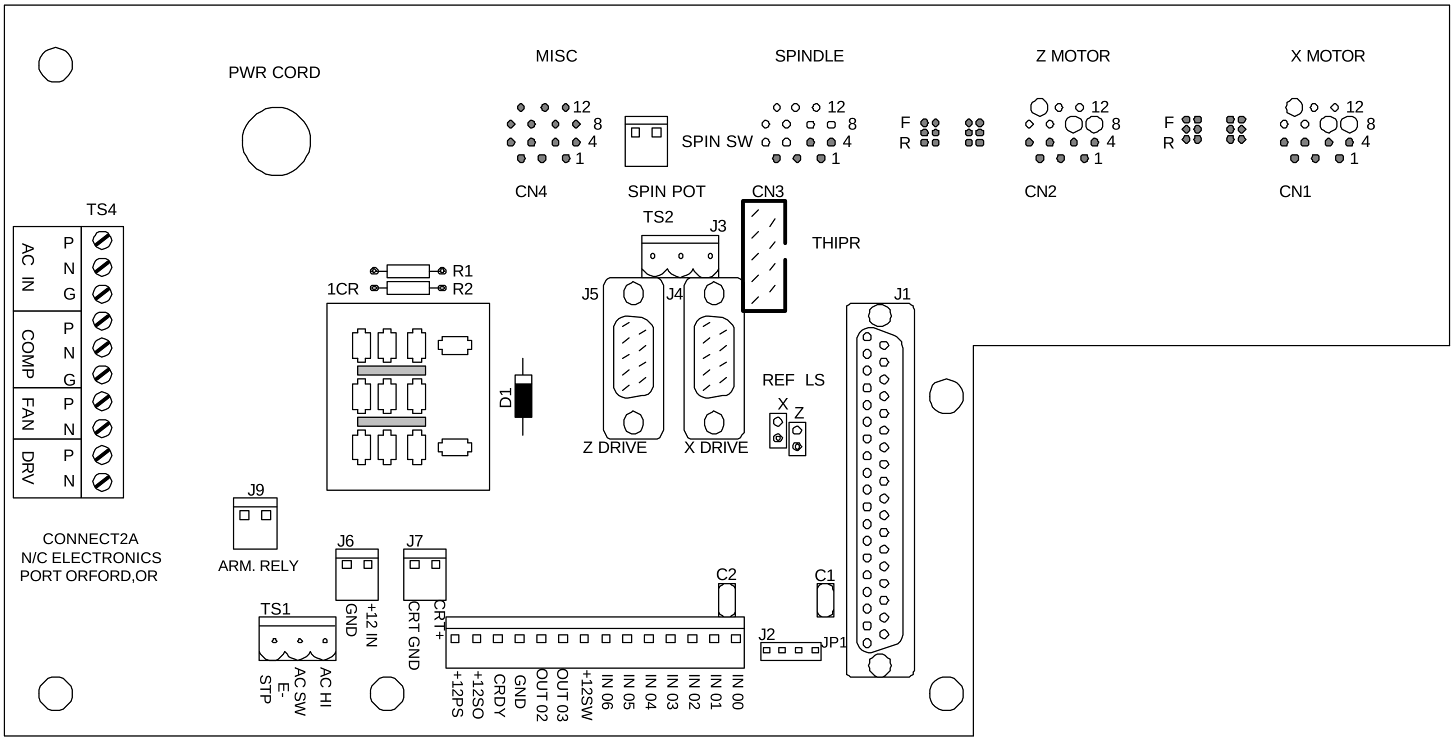


OmniTurn CNC "Top Half"

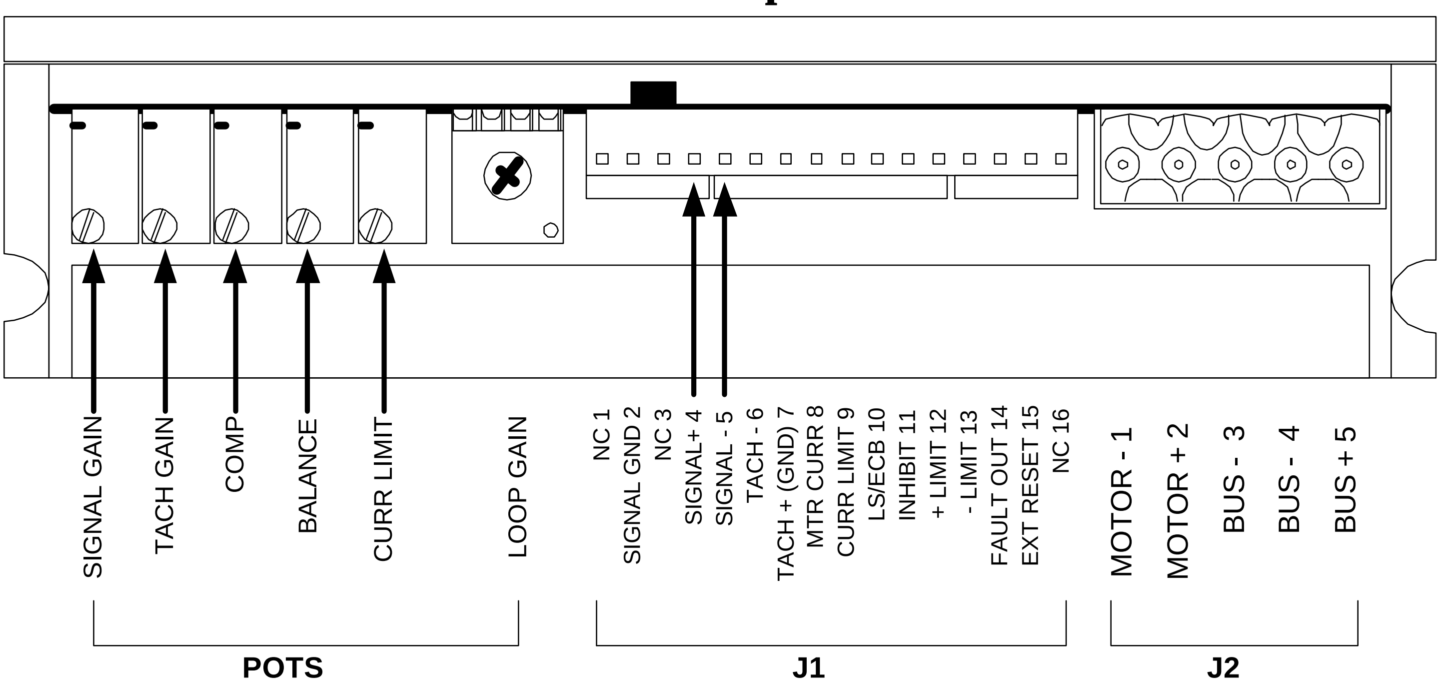


To adjust the spindle speed with a tach:
1. Issue M03 S0 from MDI; adjust SP MIN for no rotation.
2. Issue M03 S4000 (S3000 on attachments); adjust SP MAX for correct rpm.
3. Re-check M03 S0 and adjust as required.

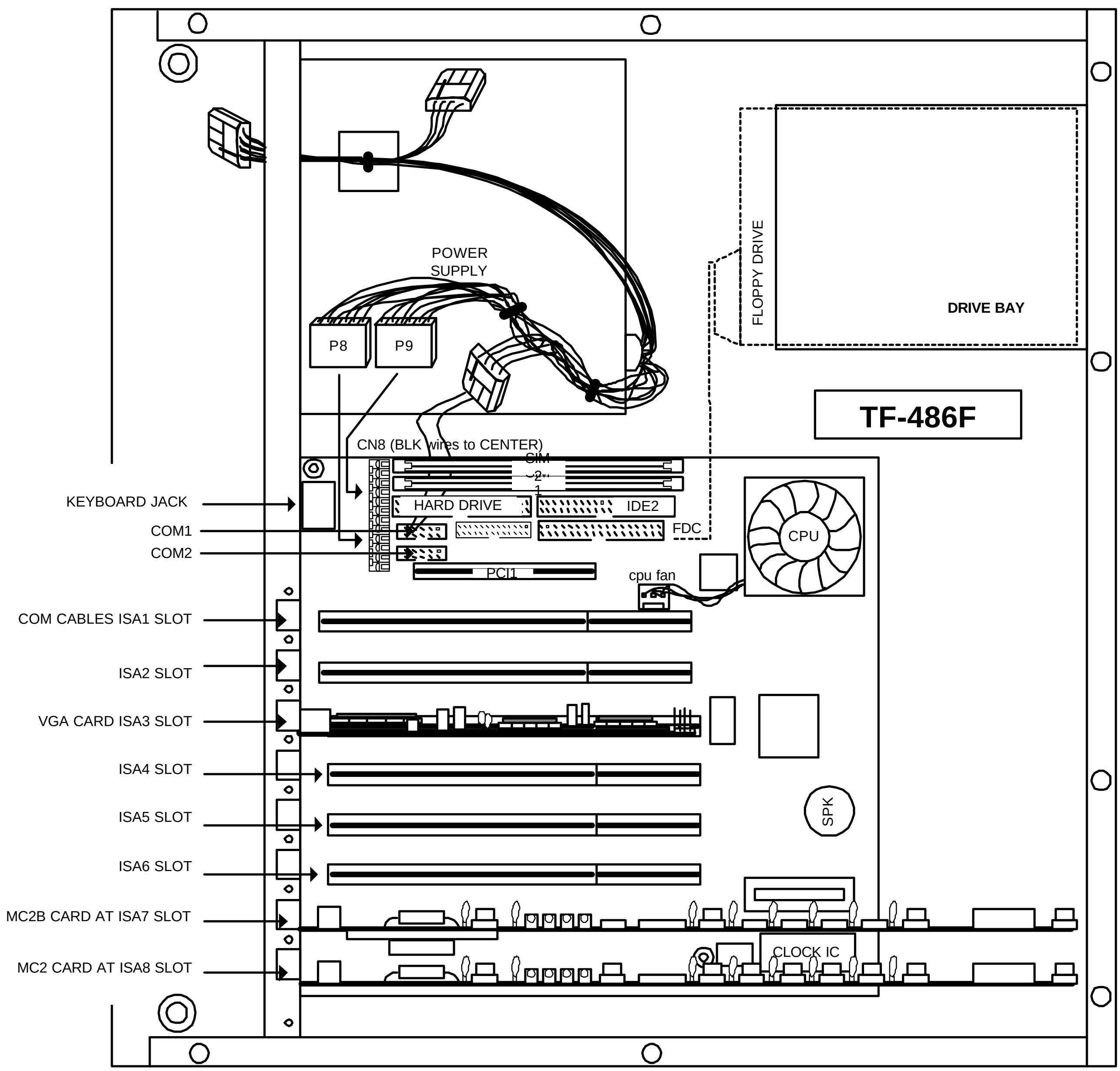
OmniTurn CNC "Connect Card"



OmniTurn CNC Servo Amplifier



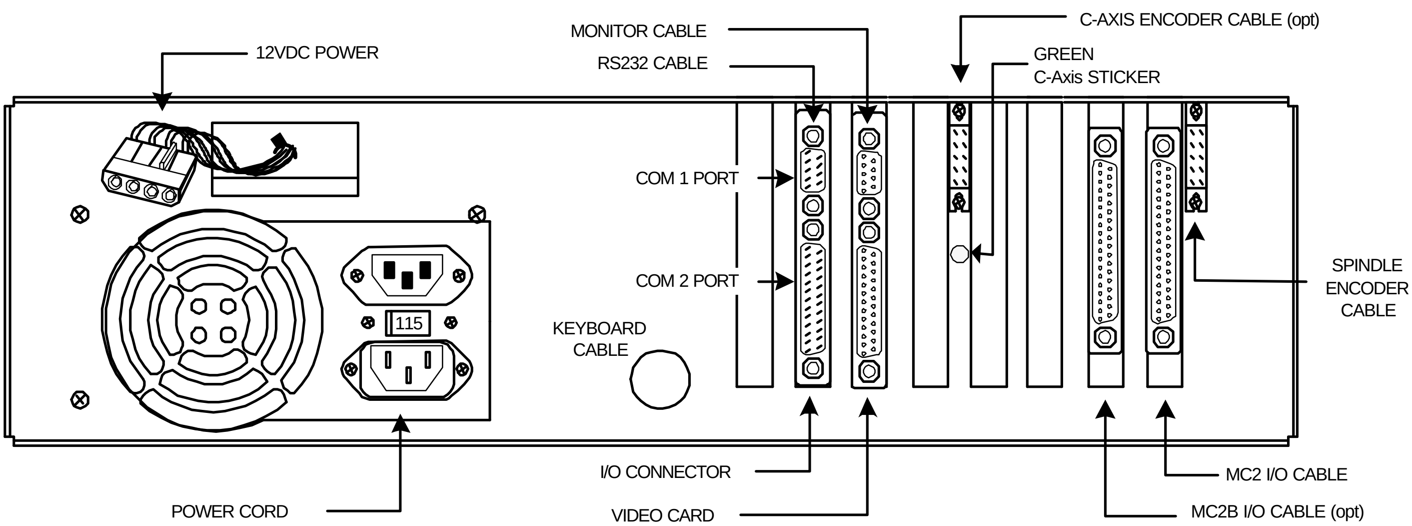
OmniTurn CNC "Bottom Half"



Replacing the MC2 (Motion Control or C-AXIS cards)

To access these cards the entire "top half" of the control must be removed:

- Unplug the power cord.
- Disconnect the cables at the back (two axis motors, MISC, Encoder, PLC (optional)
- Remove the blue cover
- Remove front panel. (Two screws on the face and two on each side). Lift and set on top of control. Disconnect the keyboard cable from lower half of control.
- Unplug all cables going to control "lower half", being careful not to damage cables on removal (12VDC power, Power Cord, RS232 Cable, Monitor Cable, C-Axis Encoder Cable (opt), MC2B I/O Cable (opt), Spindle Encoder Cable, MC2 I/O Cable)
- To remove upper half of control, unscrew (8 screws).
- Carefully lift controls upper half of control and set aside.



With computer facing as shown, you will see one or two motion control cards (depending on model), mounted on the right side. Remove top brace.

FOR MC2 CARD:
Remove 5" ribbon cable that is attached to this motion card and pull cable out of slot away from card.
Pull card straight up out of motherboard.
Place new card in same space, pushing straight down gently into motherboard.
Carefully work 5" cable back through slot and attach to MC2 card.

FOR MC2B (C AXIS) CARD:
Remove 5" ribbon cable that's attached to this motion card.
Pull card straight up out of motherboard.
Place new card in same space, pushing straight down gently into motherboard.
Attach the MC2B's 5" cable to card.

Replace top brace. Screw down tightly.

Setting OMNITURN Servo Amps

This needs to be done if you ever have to replace a servo AMP

Tools required: A digital voltmeter (DMM or DVM)
Fine tipped probes, or paper clips
Jewlers common screwdriver or "tweaker"

1. With the OmniTurn control completely powered down, depress "CONTROL ON" and allow the control to boot up to the point where the message "PLEASE TURN ON SERVOS" is displayed.

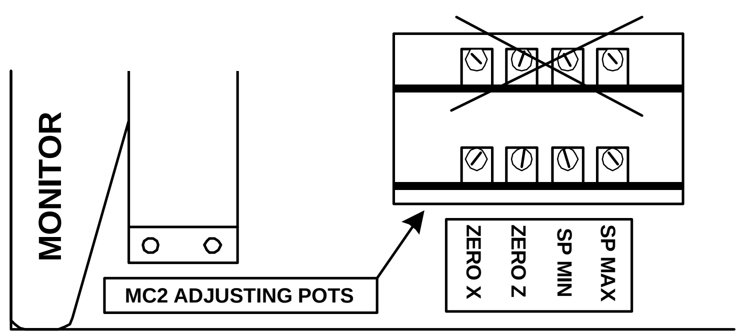
DO NOT TURN SERVOS ON AT THIS TIME

2. With the computer turned on, but the servos still turned off set the balance pot on the MC2 card

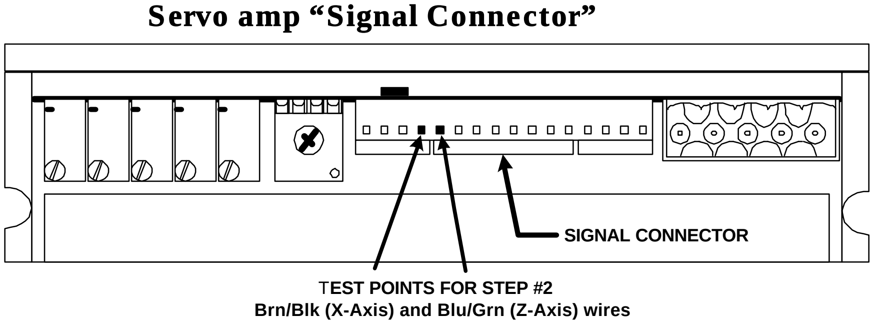
- To do this set your meter to measure DC voltage on lowest range, or Auto Range.
- Put your meter on pins 4 and 5 of the Signal plug. This is a brown connector on the top of the card. Verify that your meter is making contact. If your meter tips are too large to make contact with the wires in the plug gently insert a small wire (a paper clip will work) into the top of the plug and test off of it.
- With meter on Brown and Black wires for X axis, adjust "Zero X" pot through hole in monitor chassis for 0.000 ±.005V on your meter. Refer to monitor drawing on page 6-12 to locate acces hole in monitor chassis.
- Put the meter on Green and Blue wires for Z axis, and adjust "Zero Z" pot through hole in monitor chassis for 0.000 ± .005V on your meter.

This completes setting the balance pots on the MC2 card. Proceed to next page.

The Adjustment Pots for balancing X and Z axis with servos off are accessable through hole in chassis:



Servo amp "Signal Connector"



Setting OMNITURN Servo Amps, con't

3. On both servo amps
   adjust CURRENT LIMIT full CW (10 turns)
   adjust COMP full CCW (10 turns)
   adjust LOOP GAIN full CW (1 turn)

TURN SERVOS ON NOW

4. Turn the servos on now. Go to the Jog mode. The MC2 card has some built in diagnostics that help balance the card. To set the offset for the drive press and hold the Ctr and then E. This will pop up a window on the screen:

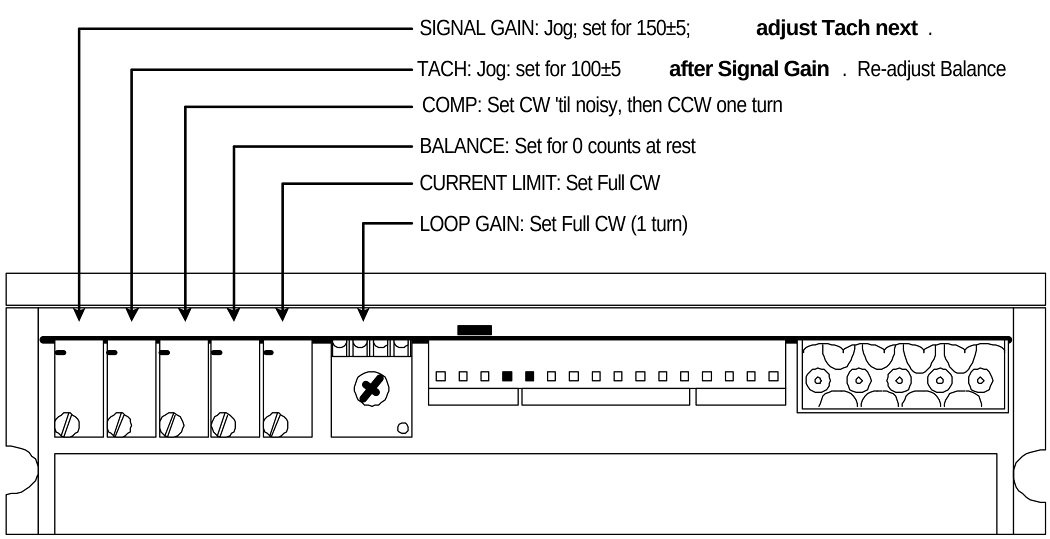
```
Set Test/Offset for Ø when slide is stopped
Set Ref In Gain for 100 in Med. Jog
Following Error
X:   Ø
Z:   Ø
```

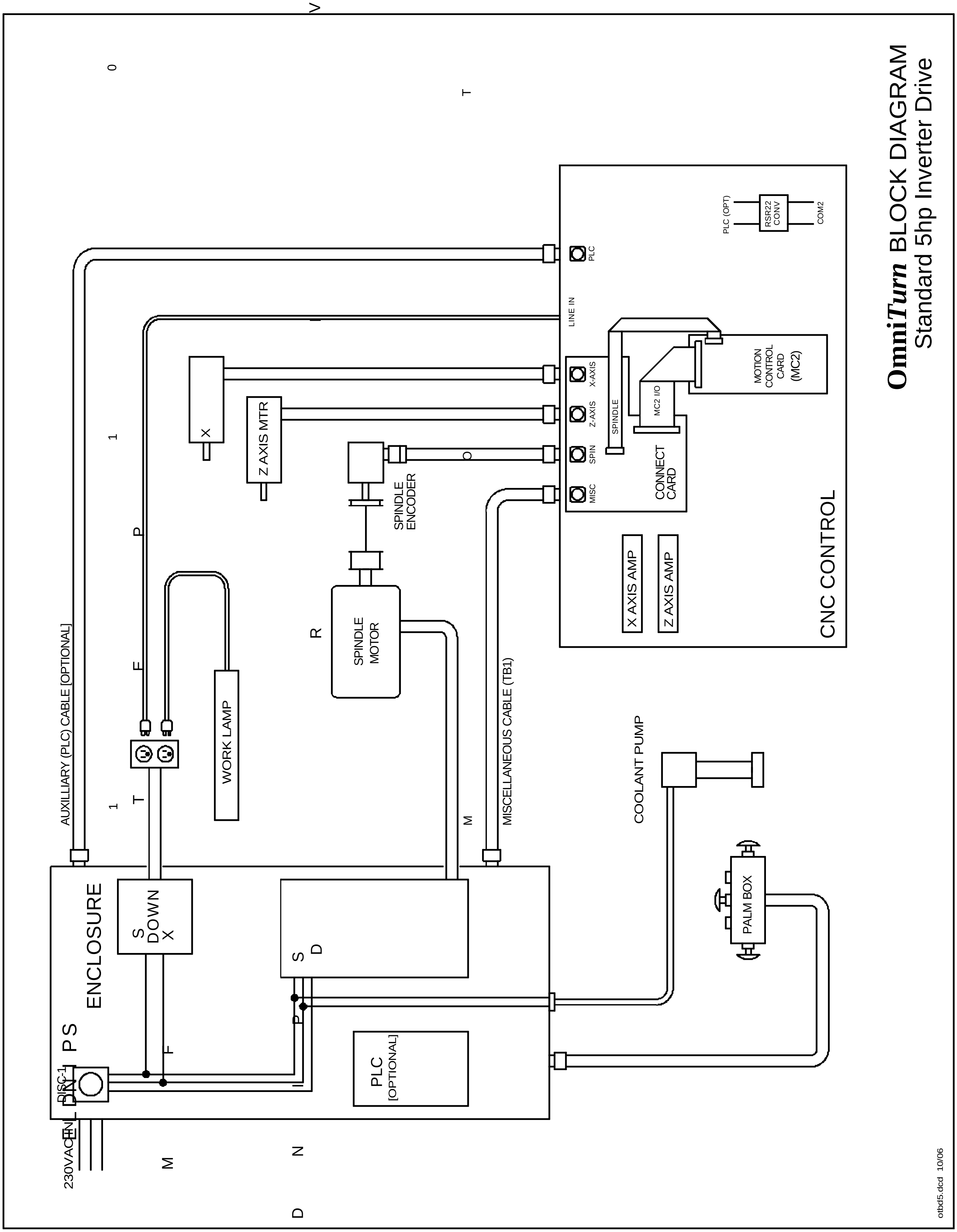
5. Use a small screw driver to adjust the "BALANCE" pot on the OMNITURN AMP. Turn the pot until the value on the screen is 0. Do this procedure with the slide at rest, no motion. Do this for both the X and Z axes.

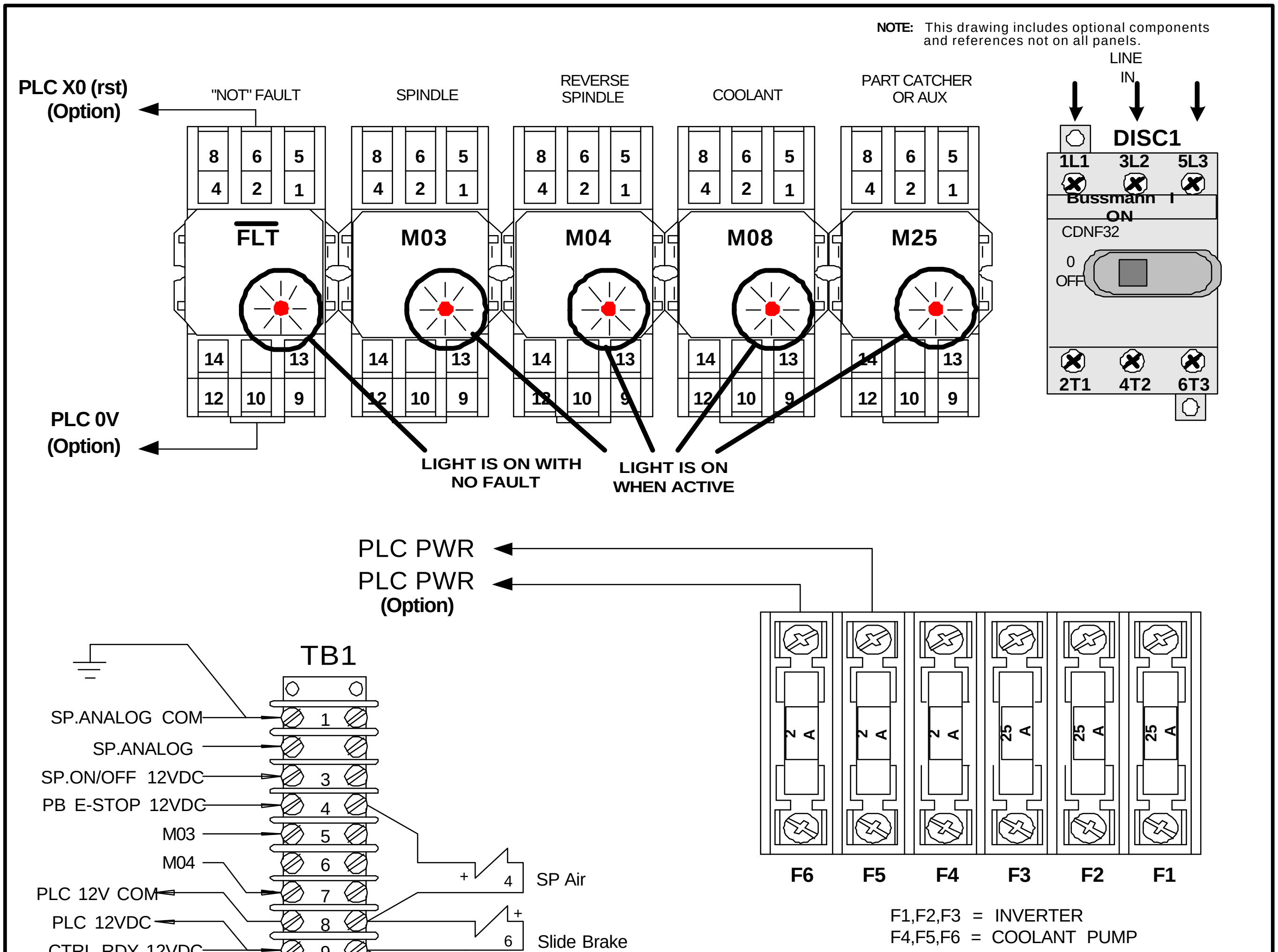
6. Now use the screw driver to adjust the "SIGNAL GAIN". To do this move the slide at medium jog speed (2 on Jog Menu). Adjust for 150 while jogging. Do this for both Axes. Re-adjust the BALANCE for zero at rest (not jogging).

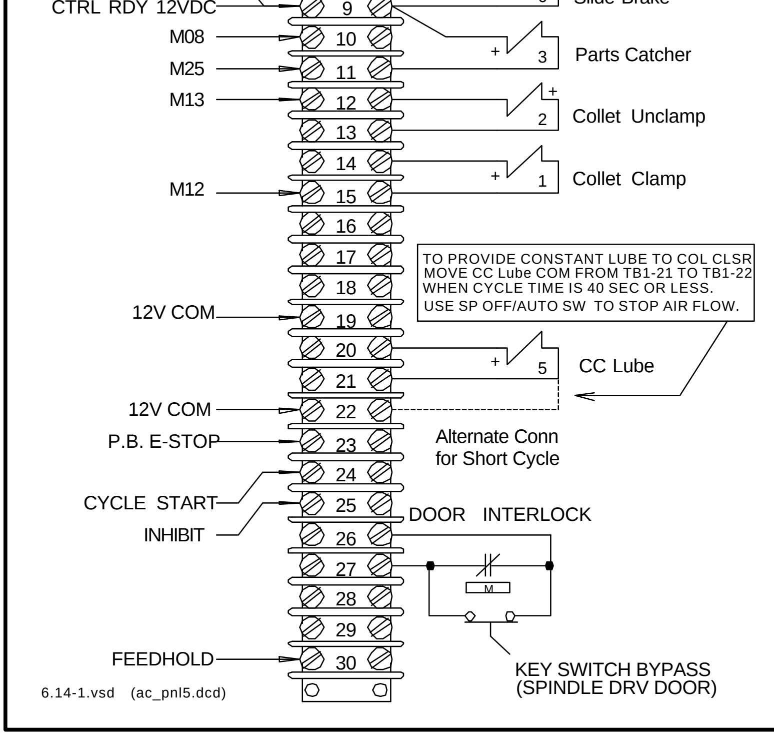
6. Next, adjust the TACH pot for 100 while jogging at medium speed (2 on Jog Menu). Do this for both Axes. Re-adjust the BALANCE for zero at rest (not jogging).

7. Adjust the COMP pot CW 'til some noise is heard in the servo motor, then turn CCW one turn. If no noise is heard at rest, run the slide in Jog 3 and see if smoothness and response is improved by adjustment. Best results are obtained by repeating a simple program which runs the slides back and forth at 300 ipm and adjusting for best performance.




OmniTurn BLOCK DIAGRAM
Standard 5hp Inverter Drive
CNC CONTROL
PLC (OPT)
RSR22 CONV
COM2
PLC
LINE IN
X-AXIS
Z-AXIS
SPIN
MISC
SPINDLE
MC2 I/O
CONNECT CARD
MOTION CONTROL CARD (MC2)
X AXIS AMP
Z AXIS AMP
Z AXIS MTR
SPINDLE ENCODER
SPINDLE MOTOR
WORK LAMP
AUXILLIARY (PLC) CABLE [OPTIONAL]
MISCELLANEOUS CABLE (TB1)
COOLANT PUMP
PALM BOX
ENCLOSURE
DISC-1
230VAC
PLC [OPTIONAL]
otbd5.dcd 10/06





PARAMETERS w/5hp WEG
ACCEL=019: Set to 2.0 for Attachments with Vari-Speed pulleys
DECEL=020: Set to 3.0 for Attachments with Vari-Speed pulleys

---

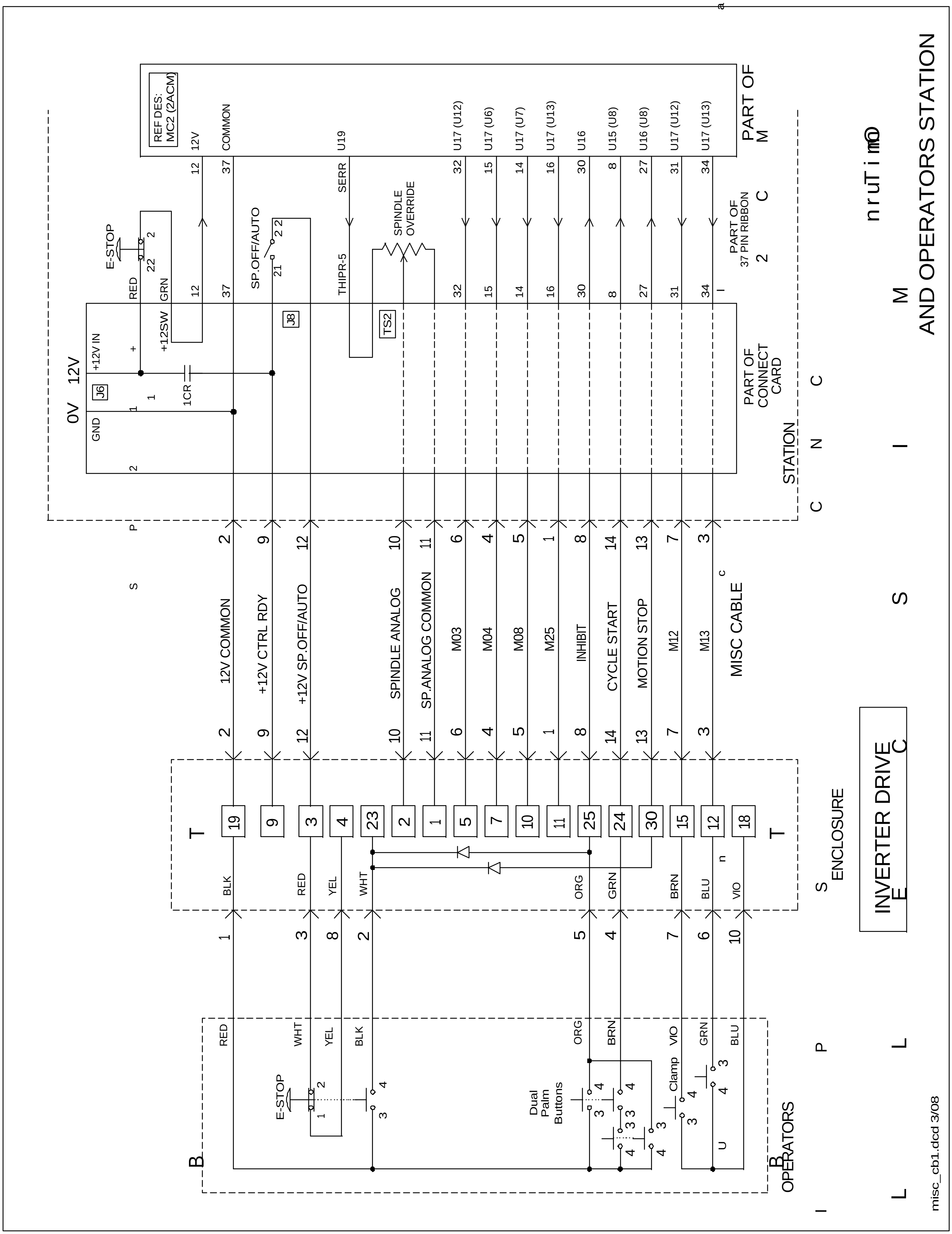
TO RE-INITIALIZE: SET NO 1 TO 8.
AFTER INITIALIZATION SET AS FOLLOWS:

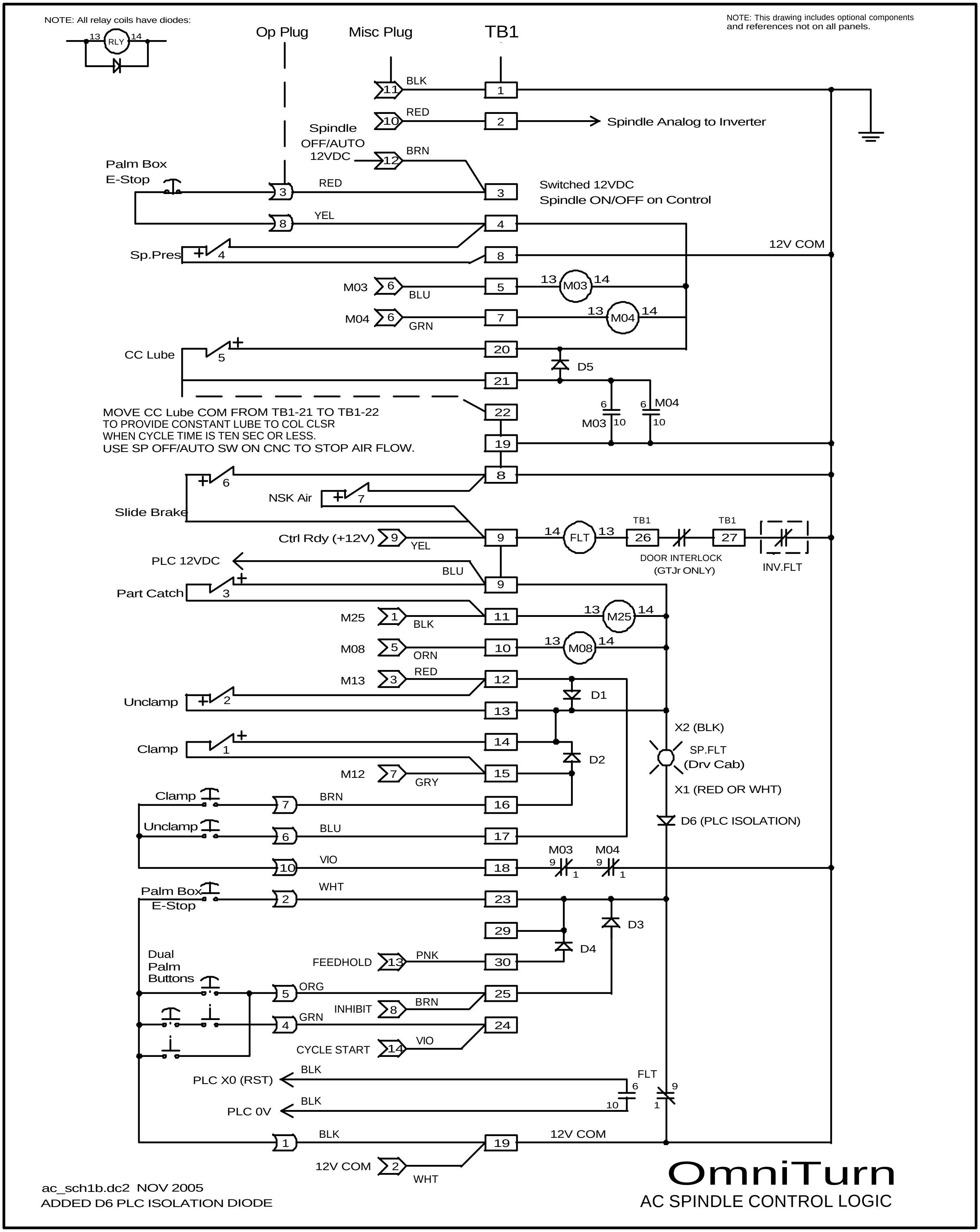
001=3	011=135	058=3
002=1	012=230	106=1.7
003=1	019=1.0	109=250
004=2	020=1.0	110=23
007=1	036=12.9	001=1 (LOCK)

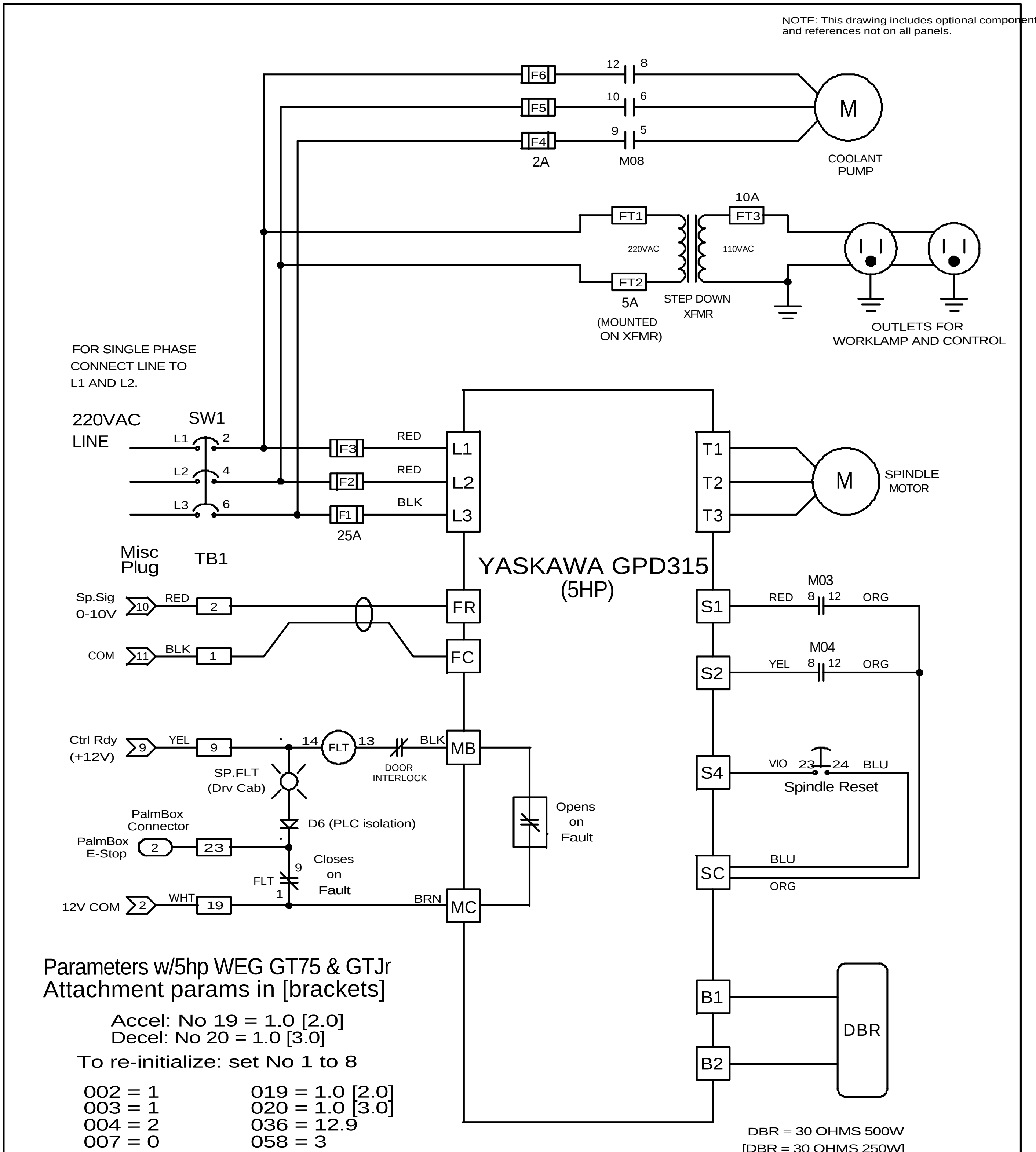
NOTE: If you are NOT using WEG 5hp motor, 036,106 & 110 must be set: refer to GPD315 manual,
page 2.3 thru 2.5

AC SPINDLE DRIVE PANEL JUN2007

Omni *Turn*


REF DES:
MC2 (2ACM)
PART OF
AND OPERATORS STATION
PART OF
37 PIN RIBBON
PART OF
CONNECT
CARD
STATION
E-STOP
SP.OFF/AUTO
SPINDLE
OVERRIDE
THIPR-5
SERR
12V
COMMON
U19
0V
12V
GND
+12V IN
+12SW
1CR
J6
J8
TS2
12V COMMON
+12V CTRL RDY
+12V SP.OFF/AUTO
SPINDLE ANALOG
SP.ANALOG COMMON
M03
M04
M08
M25
INHIBIT
CYCLE START
MOTION STOP
M12
M13
MISC CABLE
INVERTER DRIVE
ENCLOSURE
OPERATORS
Dual
Palm
Buttons
Clamp
misc_cb1.dcd 3/08


NOTE: All relay coils have diodes:
13 RLY 14
NOTE: This drawing includes optional components and references not on all panels.
Op Plug
Misc Plug
TB1
BLK
RED
Spindle Analog to Inverter
Spindle OFF/AUTO 12VDC
BRN
Palm Box E-Stop
RED
Switched 12VDC Spindle ON/OFF on Control
YEL
Sp.Pres
12V COM
M03
BLU
M04
GRN
CC Lube
D5
MOVE CC Lube COM FROM TB1-21 TO TB1-22
TO PROVIDE CONSTANT LUBE TO COL CLSR
WHEN CYCLE TIME IS TEN SEC OR LESS.
USE SP OFF/AUTO SW ON CNC TO STOP AIR FLOW.
Slide Brake
NSK Air
Ctrl Rdy (+12V)
YEL
FLT
DOOR INTERLOCK (GTJr ONLY)
INV.FLT
PLC 12VDC
BLU
Part Catch
M25
BLK
M08
ORN
M13
RED
Unclamp
D1
Clamp
D2
M12
GRY
X2 (BLK)
SP.FLT (Drv Cab)
X1 (RED OR WHT)
D6 (PLC ISOLATION)
Clamp
BRN
Unclamp
BLU
VIO
Palm Box E-Stop
WHT
D3
D4
Dual Palm Buttons
FEEDHOLD
PNK
ORG
INHIBIT
BRN
GRN
CYCLE START
VIO
PLC X0 (RST)
BLK
FLT
PLC 0V
BLK
BLK
12V COM
12V COM
WHT
ac_sch1b.dc2 NOV 2005
ADDED D6 PLC ISOLATION DIODE
OmniTurn
AC SPINDLE CONTROL LOGIC



Parameters w/5hp WEG GT75 & GTJr
Attachment params in [brackets]

Accel: No 19 = 1.0 [2.0]
Decel: No 20 = 1.0 [3.0]

To re-initialize: set No 1 to 8

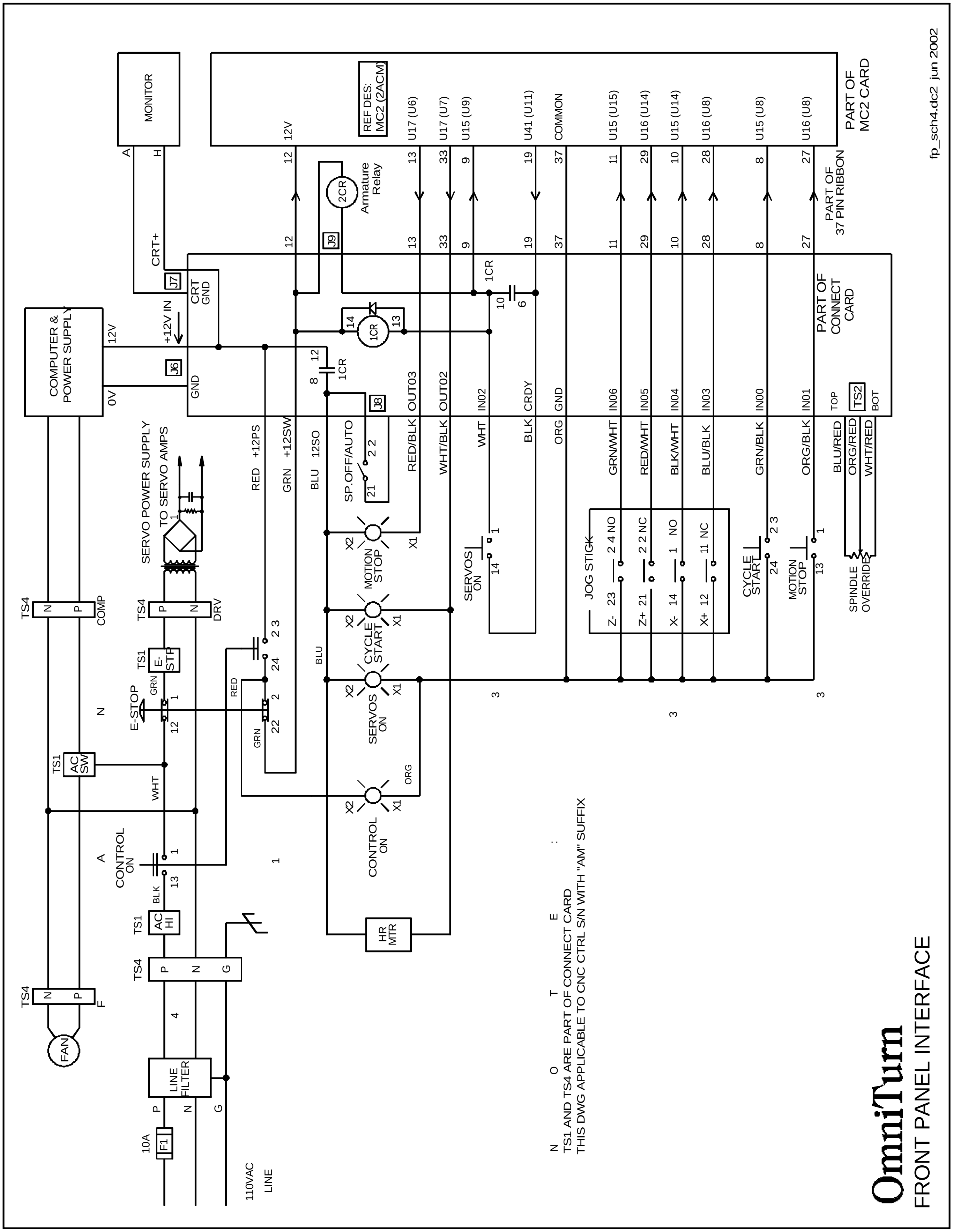
002 = 1	019 = 1.0 [2.0]
003 = 1	020 = 1.0 [3.0]
004 = 2	036 = 12.9
007 = 0	058 = 3
011 = 135 [120]	106 = 1.8
012 = 230	109 = 250
	110 = 23

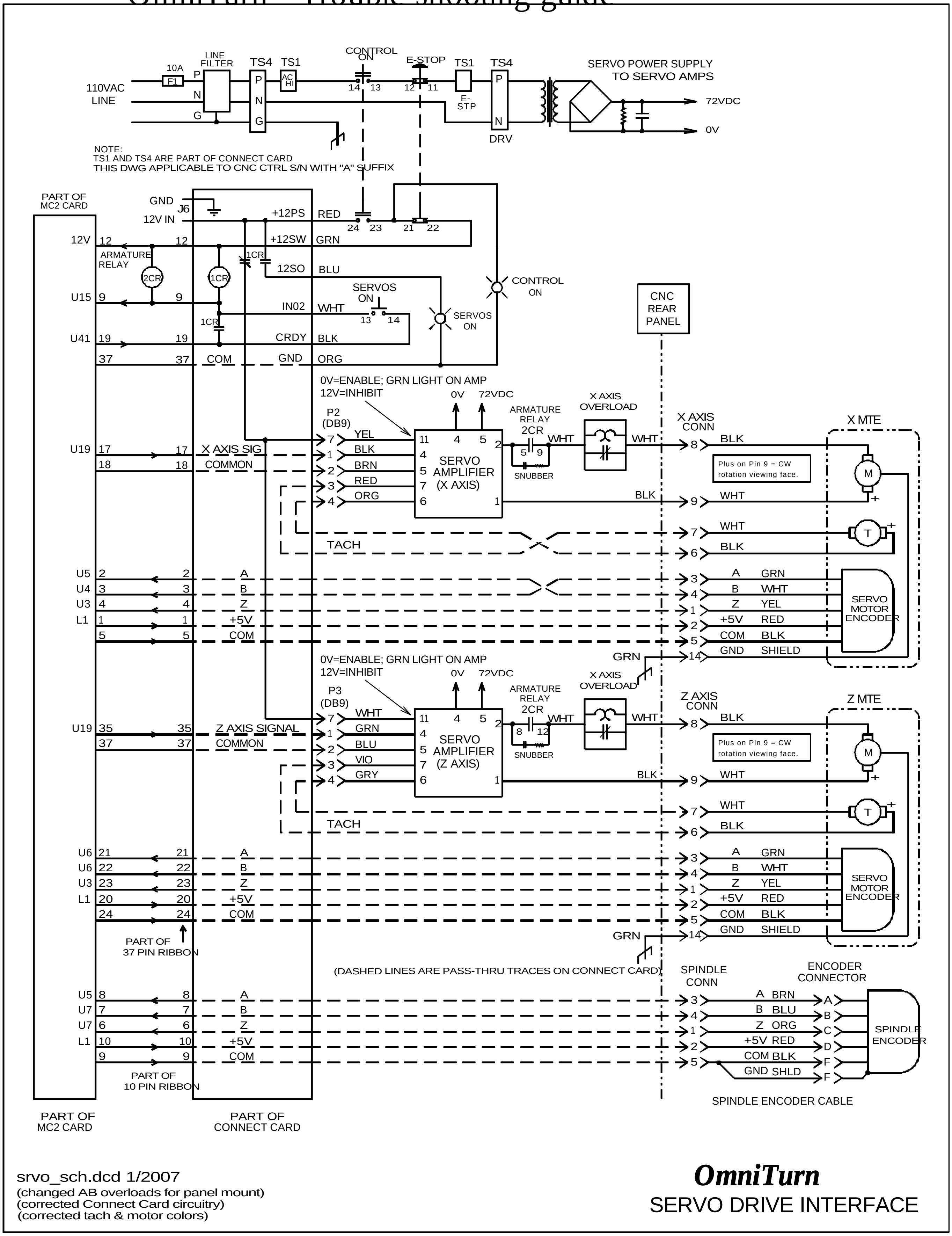
NOTE: If you are NOT using WEG 5hp motor, 036, 106 & 110 must be set: refer to GPD315 manual, page 2-3 thru 2-5.

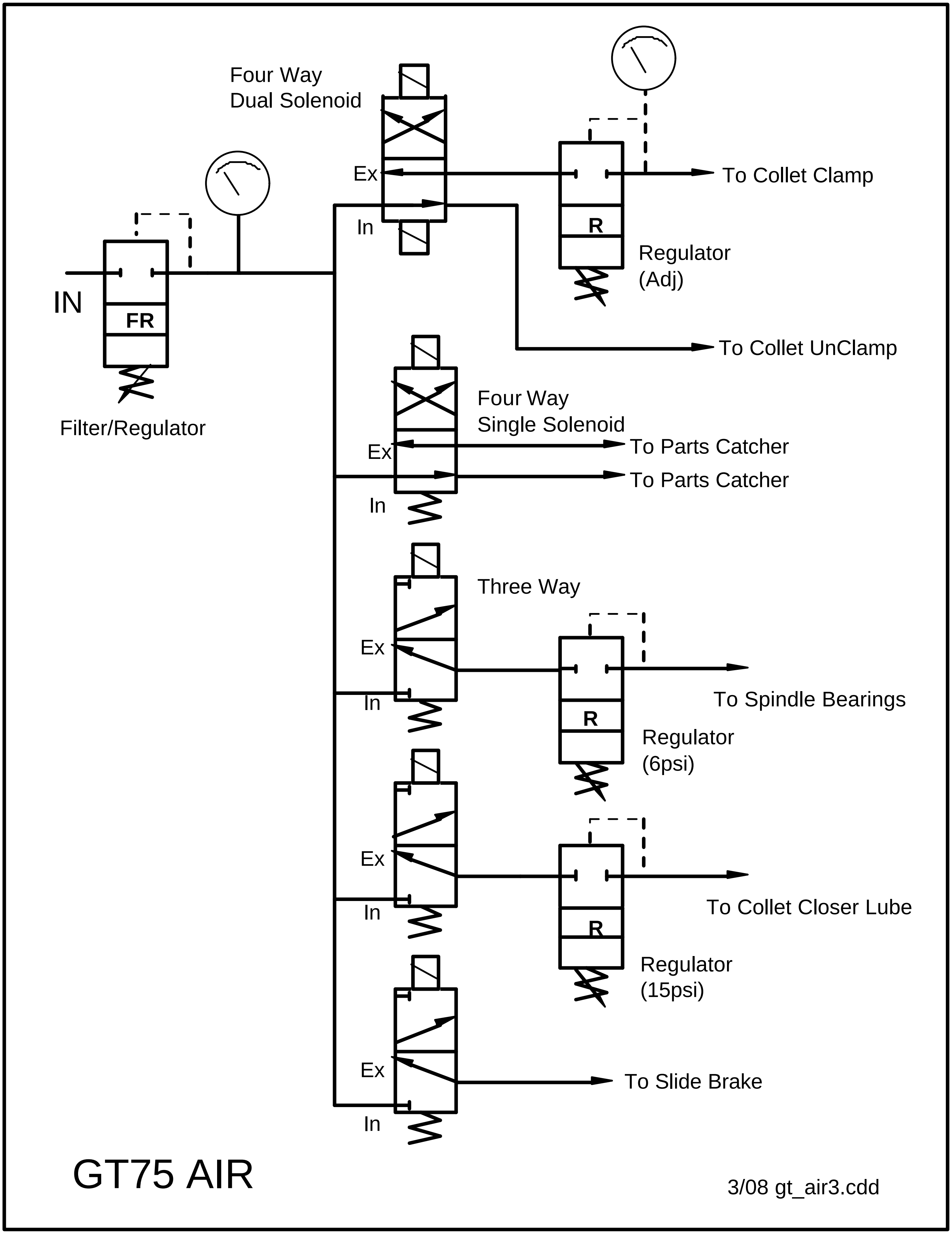
OmniTurn OCT 2006

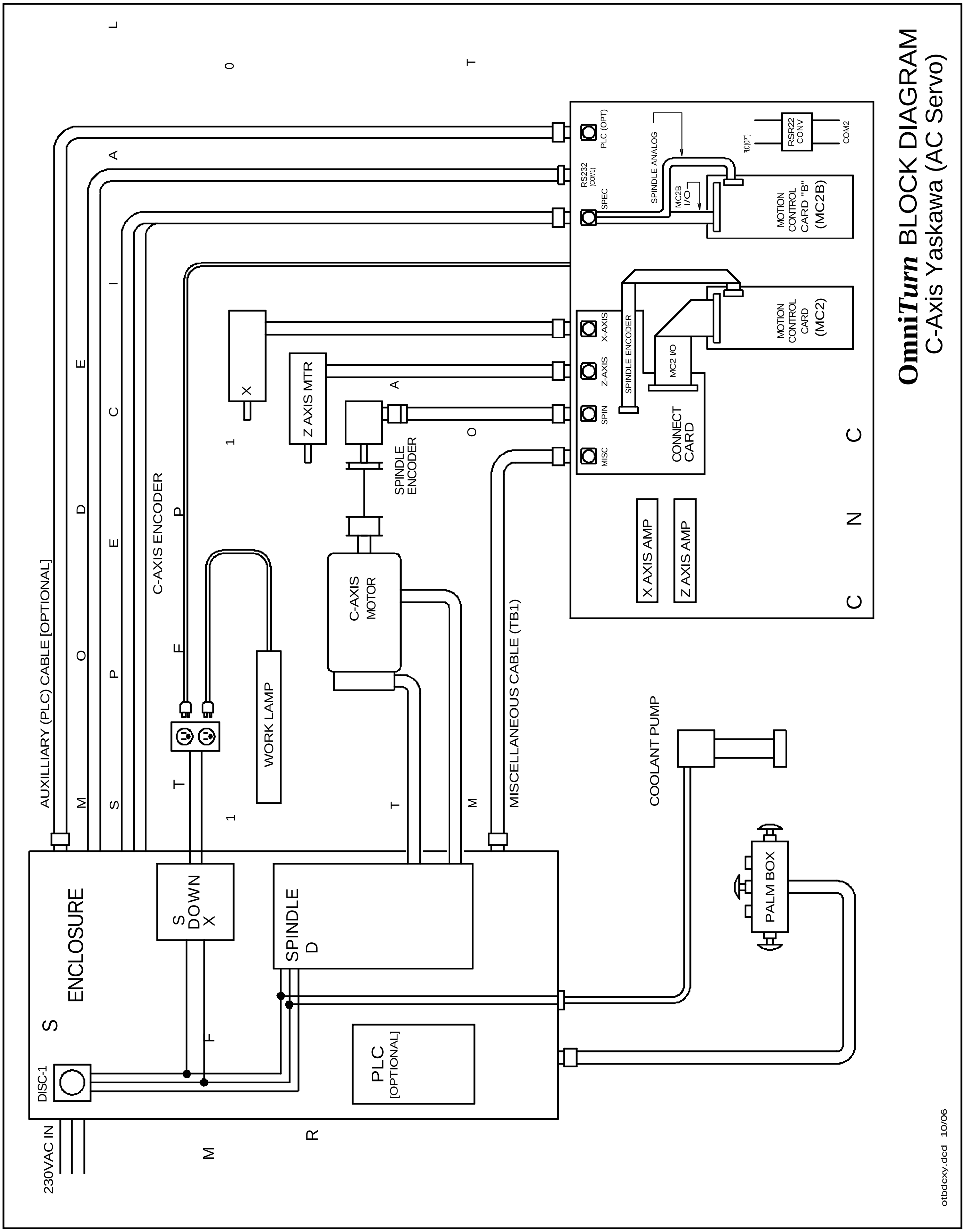
AC SPINDLE DRIVE (GPD 315 5HP)

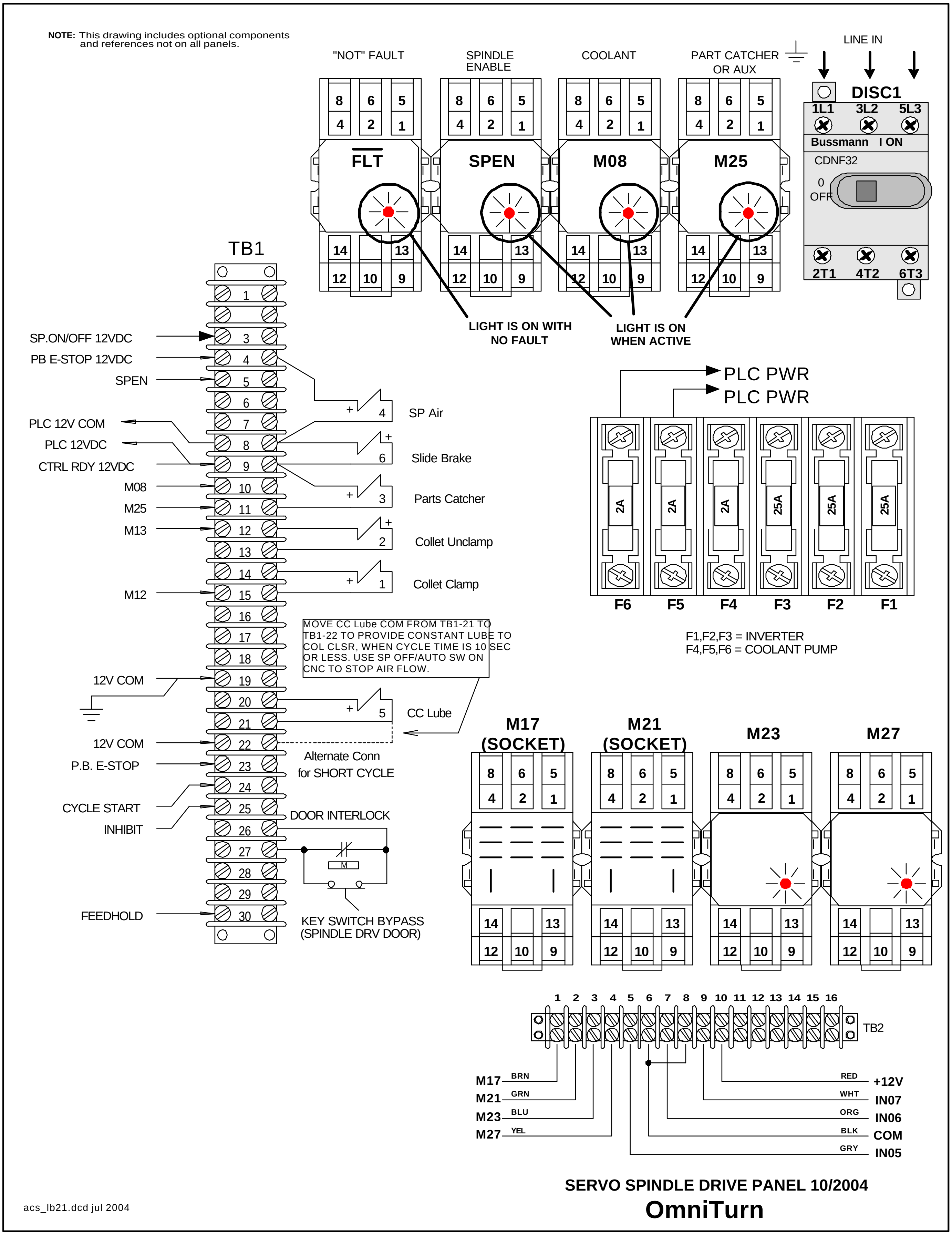
gpd5_sc.dc2

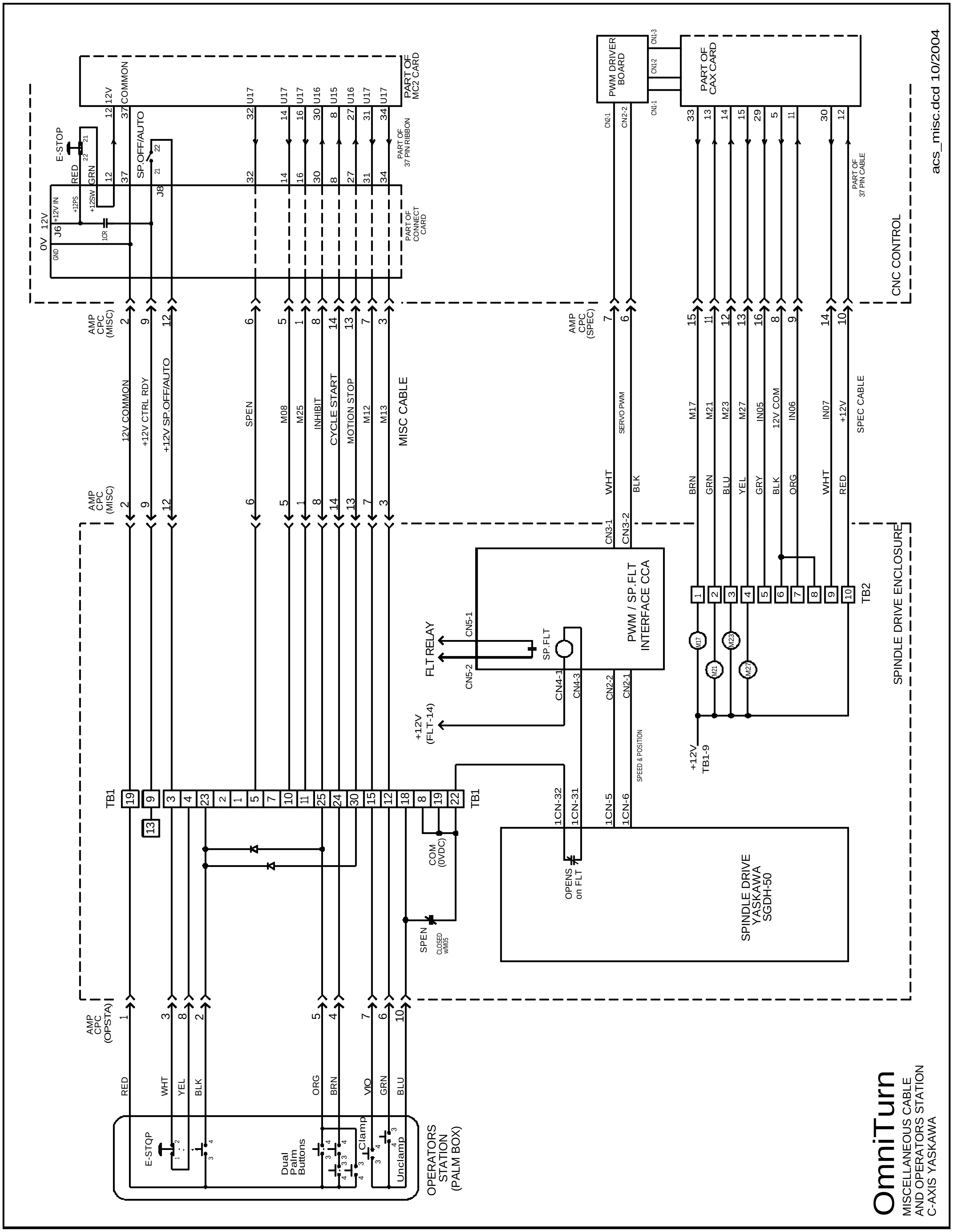

OmniTurn
FRONT PANEL INTERFACE
fp_sch4.dc2 jun 2002
TS1 AND TS4 ARE PART OF CONNECT CARD
THIS DWG APPLICABLE TO CNC CTRL S/N WITH "AM" SUFFIX
PART OF MC2 CARD
REF DES: MC2 (2ACM)
PART OF 37 PIN RIBBON
PART OF CONNECT CARD
COMPUTER & POWER SUPPLY
MONITOR
SERVO POWER SUPPLY TO SERVO AMPS
Armature Relay
JOG STICK
CYCLE START
MOTION STOP
SERVOS ON
CONTROL ON
SPINDLE OVERRIDE
E-STOP
LINE FILTER
FAN
HR MTR
110VAC LINE
SP.OFF/AUTO

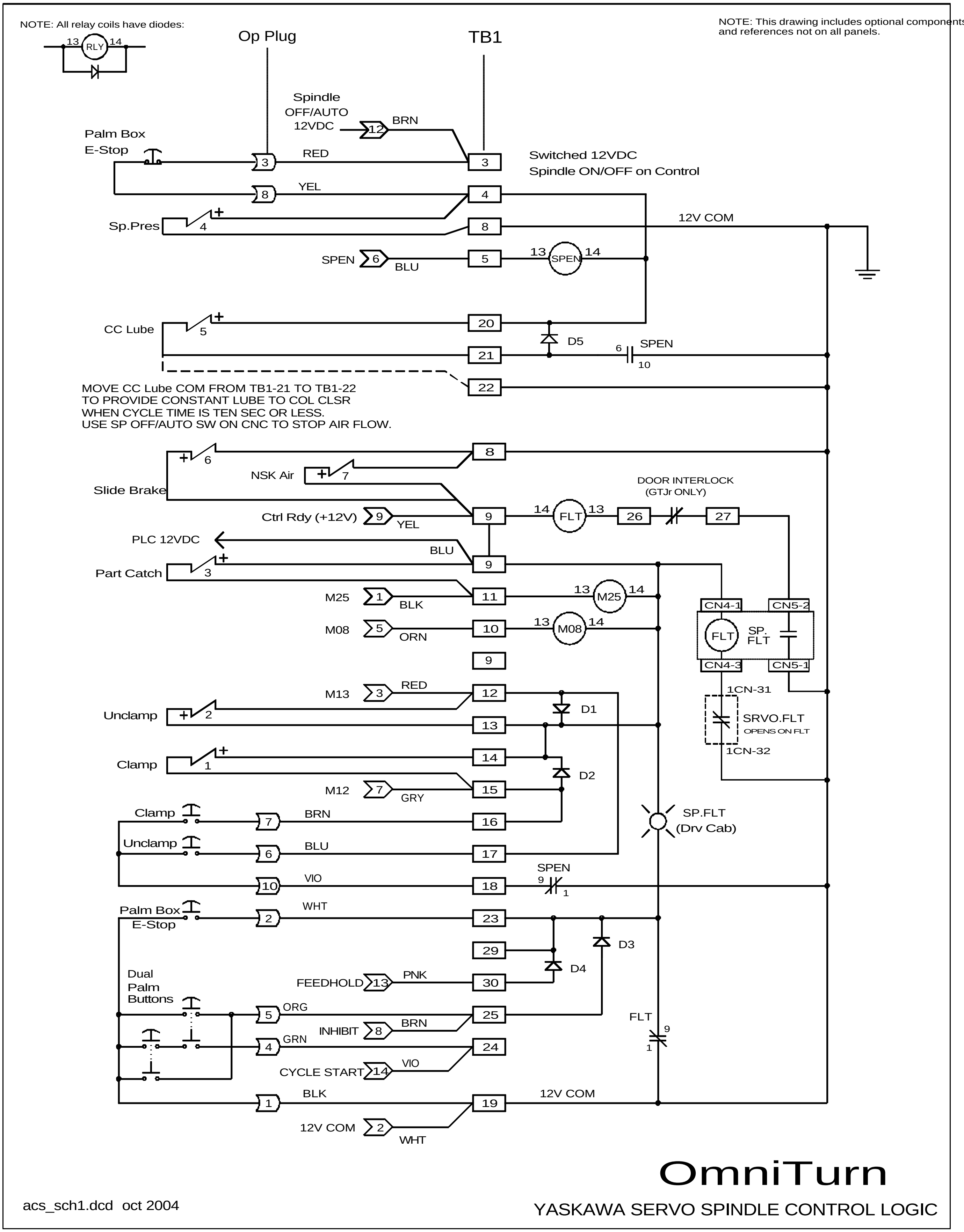

110VAC LINE
10A F1
LINE FILTER
TS4 TS1
CONTROL ON
E-STOP
TS1 TS4
SERVO POWER SUPPLY TO SERVO AMPS
72VDC
0V
DRV
NOTE:
TS1 AND TS4 ARE PART OF CONNECT CARD
THIS DWG APPLICABLE TO CNC CTRL S/N WITH "A" SUFFIX
PART OF MC2 CARD
GND
J6
12V IN
+12PS RED
+12SW GRN
12SO BLU
IN02 WHT
CRDY BLK
COM GND ORG
ARMATURE RELAY
SERVOS ON
CONTROL ON
CNC REAR PANEL
0V=ENABLE; GRN LIGHT ON AMP
12V=INHIBIT
P2 (DB9)
X AXIS SIG
COMMON
SERVO AMPLIFIER (X AXIS)
ARMATURE RELAY 2CR
SNUBBER
X AXIS OVERLOAD
X AXIS CONN
X MTE
Plus on Pin 9 = CW rotation viewing face.
TACH
SERVO MOTOR ENCODER
P3 (DB9)
Z AXIS SIGNAL
SERVO AMPLIFIER (Z AXIS)
Z AXIS CONN
Z MTE
PART OF 37 PIN RIBBON
(DASHED LINES ARE PASS-THRU TRACES ON CONNECT CARD)
SPINDLE CONN
ENCODER CONNECTOR
SPINDLE ENCODER
PART OF 10 PIN RIBBON
SPINDLE ENCODER CABLE
PART OF MC2 CARD
PART OF CONNECT CARD
srvo_sch.dcd 1/2007
(changed AB overloads for panel mount)
(corrected Connect Card circuitry)
(corrected tach & motor colors)
OmniTurn
SERVO DRIVE INTERFACE

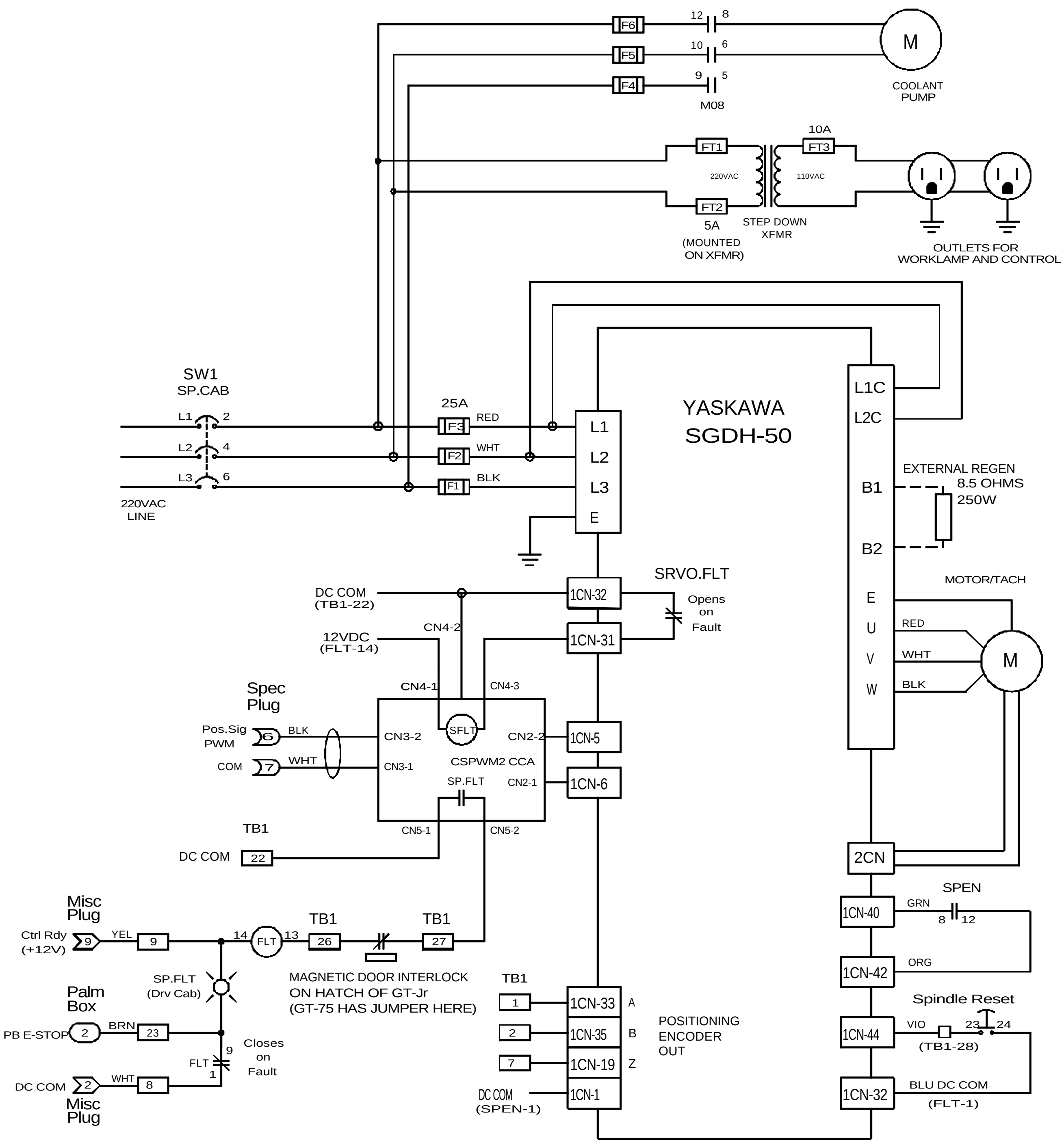

Four Way
Dual Solenoid
Ex
In
To Collet Clamp
R
Regulator
(Adj)
IN
FR
Filter/Regulator
To Collet UnClamp
Four Way
Single Solenoid
Ex
To Parts Catcher
To Parts Catcher
In
Three Way
Ex
In
To Spindle Bearings
R
Regulator
(6psi)
Ex
In
To Collet Closer Lube
R
Regulator
(15psi)
Ex
To Slide Brake
In
GT75 AIR
3/08 gt_air3.cdd


OmniTurn BLOCK DIAGRAM
C-Axis Yaskawa (AC Servo)
PLC (OPT)
RS232 (COM1)
SPEC
SPINDLE ANALOG
MC2B I/O
RSR22 CONV
COM2
PLC (OPT)
MOTION CONTROL CARD "B" (MC2B)
MOTION CONTROL CARD (MC2)
X-AXIS
Z-AXIS
SPIN
MISC
SPINDLE ENCODER
MC2 I/O
CONNECT CARD
X AXIS AMP
Z AXIS AMP
C N C
X
Z AXIS MTR
SPINDLE ENCODER
C-AXIS MOTOR
C-AXIS ENCODER
WORK LAMP
AUXILLIARY (PLC) CABLE [OPTIONAL]
MISCELLANEOUS CABLE (TB1)
COOLANT PUMP
PALM BOX
ENCLOSURE
S DOWN X
SPINDLE D
PLC [OPTIONAL]
DISC-1
230VAC IN
otbdcxy.dcd 10/06


NOTE: This drawing includes optional components and references not on all panels.
"NOT" FAULT
SPINDLE ENABLE
COOLANT
PART CATCHER OR AUX
LINE IN
DISC1
1L1 3L2 5L3
Bussmann I ON
CDNF32
0 OFF
2T1 4T2 6T3
FLT
SPEN
M08
M25
LIGHT IS ON WITH NO FAULT
LIGHT IS ON WHEN ACTIVE
TB1
SP.ON/OFF 12VDC
PB E-STOP 12VDC
SPEN
PLC 12V COM
PLC 12VDC
CTRL RDY 12VDC
M08
M25
M13
M12
12V COM
12V COM
P.B. E-STOP
CYCLE START
INHIBIT
FEEDHOLD
SP Air
Slide Brake
Parts Catcher
Collet Unclamp
Collet Clamp
CC Lube
MOVE CC Lube COM FROM TB1-21 TO TB1-22 TO PROVIDE CONSTANT LUBE TO COL CLSR, WHEN CYCLE TIME IS 10 SEC OR LESS. USE SP OFF/AUTO SW ON CNC TO STOP AIR FLOW.
Alternate Conn for SHORT CYCLE
DOOR INTERLOCK
KEY SWITCH BYPASS (SPINDLE DRV DOOR)
PLC PWR
PLC PWR
2A 2A 2A 25A 25A 25A
F6 F5 F4 F3 F2 F1
F1,F2,F3 = INVERTER
F4,F5,F6 = COOLANT PUMP
M17 (SOCKET)
M21 (SOCKET)
M23
M27
TB2
M17 BRN
M21 GRN
M23 BLU
M27 YEL
RED +12V
WHT IN07
ORG IN06
BLK COM
GRY IN05
SERVO SPINDLE DRIVE PANEL 10/2004
OmniTurn
acs_lb21.dcd jul 2004


OmniTurn
MISCELLANEOUS CABLE
AND OPERATORS STATION
C-AXIS YASKAWA
acs_misc.dcd 10/2004
CNC CONTROL
PART OF MC2 CARD
PART OF CONNECT CARD
PART OF 37 PIN RIBBON
PWM DRIVER BOARD
PART OF CAX CARD
PART OF 37 PIN CABLE
E-STOP
SP.OFF/AUTO
AMP CPC (MISC)
AMP CPC (SPEC)
AMP CPC (OPSTA)
12V COMMON
+12V CTRL RDY
+12V SP.OFF/AUTO
SPEN
M08
M25
INHIBIT
CYCLE START
MOTION STOP
M12
M13
MISC CABLE
SERVO PWM
M17
M21
M23
M27
IN05
12V COM
IN06
IN07
+12V
SPEC CABLE
PWM / SP.FLT INTERFACE CCA
FLT RELAY
SP.FLT
+12V (FLT-14)
SPEED & POSITION
SPINDLE DRIVE ENCLOSURE
TB1
TB2
+12V TB1-9
COM (0VDC)
OPENS on FLT
SPEN CLOSED w/M05
SPINDLE DRIVE YASKAWA SGDH-50
E-STOP
Dual Palm Buttons
Clamp
Unclamp
OPERATORS STATION (PALM BOX)


NOTE: All relay coils have diodes:
13 RLY 14
Op Plug
TB1
NOTE: This drawing includes optional components and references not on all panels.
Spindle OFF/AUTO 12VDC
BRN
Palm Box E-Stop
RED
Switched 12VDC Spindle ON/OFF on Control
YEL
Sp.Pres
12V COM
SPEN
BLU
13 SPEN 14
CC Lube
D5
6 SPEN 10
MOVE CC Lube COM FROM TB1-21 TO TB1-22
TO PROVIDE CONSTANT LUBE TO COL CLSR
WHEN CYCLE TIME IS TEN SEC OR LESS.
USE SP OFF/AUTO SW ON CNC TO STOP AIR FLOW.
NSK Air
Slide Brake
DOOR INTERLOCK (GTJr ONLY)
Ctrl Rdy (+12V)
YEL
14 FLT 13
PLC 12VDC
BLU
Part Catch
M25
BLK
13 M25 14
M08
ORN
13 M08 14
CN4-1 CN5-2
FLT SP. FLT
CN4-3 CN5-1
M13
RED
Unclamp
D1
1CN-31
SRVO.FLT
OPENS ON FLT
1CN-32
Clamp
D2
M12
GRY
Clamp
BRN
Unclamp
BLU
SP.FLT (Drv Cab)
VIO
SPEN
Palm Box E-Stop
WHT
D3
D4
Dual Palm Buttons
FEEDHOLD
PNK
ORG
INHIBIT
BRN
GRN
CYCLE START
VIO
FLT
BLK
12V COM
12V COM
WHT
OmniTurn
acs_sch1.dcd oct 2004
YASKAWA SERVO SPINDLE CONTROL LOGIC



AC SERVO SPINDLE DRIVE

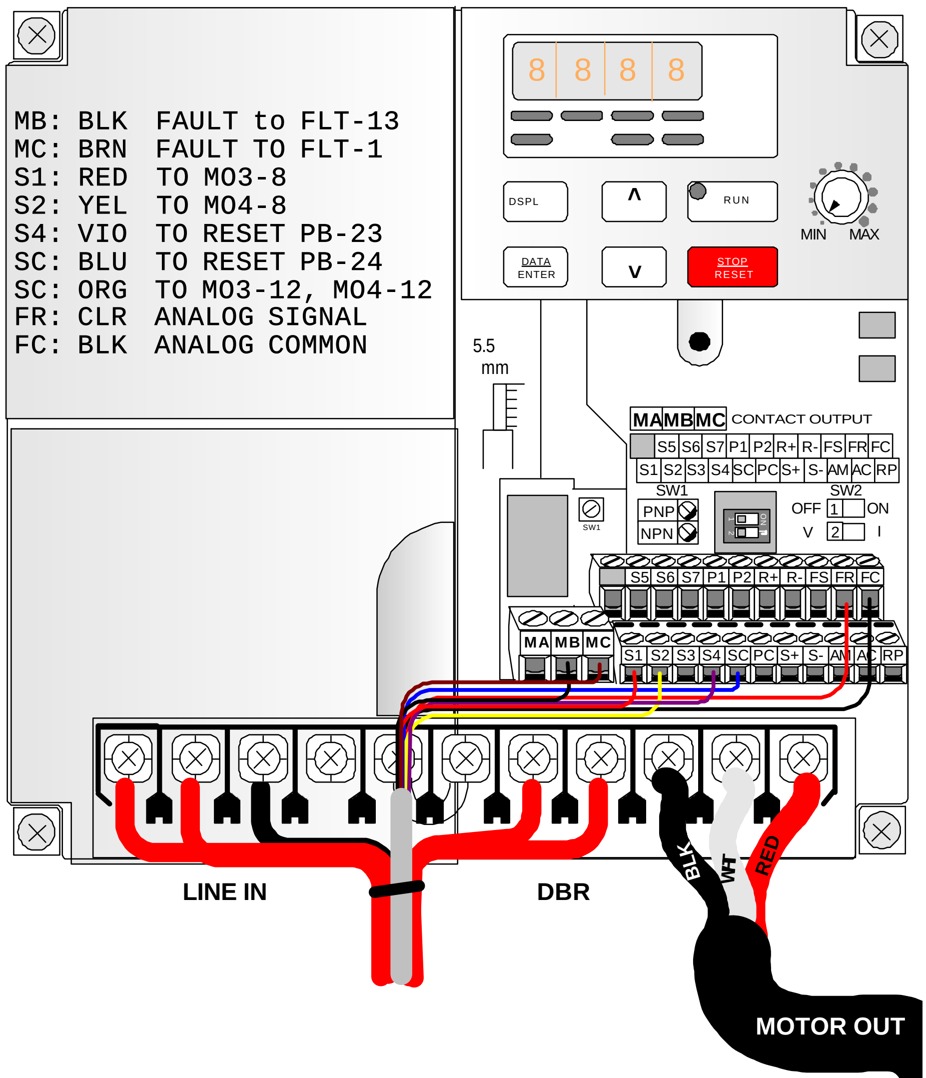
YASKAWA

OmniTurn

acs_sch2.dcd oct2004

5HP SPINDLE DRIVE

YASKAWA GPD315



PARAMETERS w/5hp WEG
ACCEL=019
DECEL=020

TO RE-INITIALIZE: SET NO 1 TO 8.
AFTER INITIALIZATION SET AS FOLLOWS:

001=3	011=120	036=12.9
002=1	012=230	106=1.8
003=1	019=1.0	109=250
004=2	020=1.0	110=23
007=0		

NOTE: If you are NOT using WEG 5hp motor, 036,106 & 110 must be set: refer to GPD315 manual, page 2.3 thru 2.5

PRM.SER -System Parameters

This is the file that holds some of the system parameters. Modifications to this file will effect the way in which the OmniTurn will function.

• This file is stored on the system disk on the A: drive.

• Exit the OmniTurn software and go to DOS. This is done by going to the main menu and pressing the left shift and then while holding it press the ESC key. If this does not work you can get to DOS through the word processor. Go to the Automatic mode page, Then make a program active, then press F3 for the editor. Then ESC to make the editor active. Then press F1 to get to the help menu, then press F4 for SHELL. Now you should be in DOS This will get you to C>\or A>\OMNISLID depending on your system. Then type A: return, then type cd\omnislid.

• From A:\OMNISLID type EW PRM.SER and return. This will start the word processor. Work with the file and modify it as if it were a program. Then exit(F1) and save(F2).

• Get back to the OmniTurn restart the control

"+##.#####",20000,20000,300,1,3,15,0,0,1,4000,c:\programs,c:,0,STD,12,150,50

"+##.#####" FOR 2 DIGITS BEFORE AND 5 AFTERTHE DECIMAL POINT

NUMBERS ARE ENCODER COUNTS PER INCH (X & Z)

300 IS MAX FEEDRATE

1 - IS FOR CONSTANT VOLUME THREADING
0 - FOR "OLD" STYLE

3,15,0 - ARE FOREGROUND AND BACKGROUND SCREEN COLORS

0 - MEANS NO PASSWORD, 1 MEANS PASSWORD

NEXT IS COLLET VARIABLE; 1 FOR PULSED, 0 FOR MAINTAINED (DO NOT CHANGE!)

NEXT IS DEFAULT TOP SPINDLE SPEED

NEXT IS DRIVE WHERE PART PROGRAMS ARE STORED

NEXT IS DRIVE WHERE TOOL OFFSETS ARE STORED

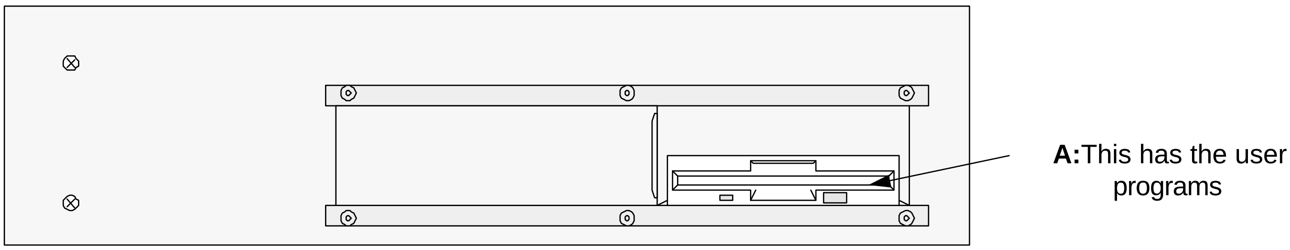
NEXT IS INCH/METRIC DEFAULT (0=INCH, 1=METRIC)

The OmniTum is based on a small computer that runs on DOS. This is the same as most computers that people have in their office. Because of this there are a number of procedures that you can perform that make this system very user friendly.

• **DOS Version.** The OmniTum is shipped with OPENDOS 7.0 We do not include all available DOS commands. There are only a few that are needed for system operations.

• **System specification (for OmniTurn)*:*** 32meg of RAM, 16MB solid state HD, 1.44 MB FLOPPY. (specifications may change without notice)

The computer floppy drive located at the back of the OmniTurn control



• Backing up your disks:

It is regarded good practice to make copies of the disks that are in the computer to insure that if you have a problem with your system that you do not loose all the work you have.

At Screen Prompt: “Do you want to back up your program files (Y/N)” Press Y to back up only programs that have changed since last back up.

TO UPDATE SOFTWARE ON OMNITURN CNC HARD DRIVE:

1. On website OMNITURN.COM go to download software
2. Create SYSTEM DISK for your system per instructions on that page.
3. Power up your Omniturn, then drop to DOS by holding left “Shift” key and pressing “Esc” key.
4. Put new system disk in lower drive. At C:\OMNITURN prompt, type **A:\UPD** then press Rtrn.
5. Several files will copy from floppy to hard drive, then “system files copied” message will appear.
6. Set power OFF, wait a few seconds, then set power ON to reboot.

This method insures that you have latest System Disk on hand as well.
Questions?
Call OmniTurn: 541-332-7004
or email **service@omniturn.com**

Once you have the new system disk in and running:

1. Go to the Automatic mode
2. Make the program you are using active
3. Press F10 for special functions
4. Press L to copy the offsets to the A drive in the TCMP file.

It is good practice to save your offsets once you have them established. This would enable you to run your program right away after a system failure. You would not have to reset the offsets.

• **New software updates.** Nobody is perfect, we are constantly making improvements to our software. In order to take advantage of this all you have to do as a user is load a new disk in the back of the control. When updates occur they will be sent to you with instructions and comments on the additions and changes included in the update.

• **BIOS SETUP** - On some of the older systems you will have to adjust the BIOS of your system. You will know that you have to do this when you try and start the system with a new system disk and they won't work. If you are replacing a 5-1/4" disk drive with a 3-1/2" you will have to perform these changes! This is done when the computer is first turned on. Then it asks if you want to make changes. Normally you don't do anything and the system goes past this and it starts the OmniTurn software. Follow the instructions on the screen to make changes.

Change the following:

Disable -shadow All shadow RAM
Enable -memory relocation
Set A: and B: Floppy disks to 1.44 Meg

If you get an error "Memory Size Mismatch" after changing the Bios, please store the BIOS setup again and the problem should go away.

See the BIOS setup descriptions at the end of this section

• **Disk storage:**
The lower DRIVE is where the A: disk is inserted. This has the system software needed to run the OmniTurn. Below find a list of the files found on this disk

AUTOEXEC.BAT

```
path=c:;a:\dos;a\omni-chk;a:\omnislid;a:\calcaid
prompt $p$g
copy command.com c:
cd \omnislid
dir omni* >bdir.txt
copy *.* c:
cd \omni-chk
copy *.* c:
cd\calcaid
copy *.* c:
cd \omnislid
c:
set comspec=c:\command.com
omni2
```

CONFIG.SYS

```
DEVICE=\DOS\HIMEM.SYS
DEVICE=\DOS\ramdrive.SYS 1344/E
```

/DOS -This directory has the DOS information needed by the computer to run. We use 5.0

/OMNI_CHK

```
OMNI_CHK.EXE
CHKHERE
```

/CALCAID -

CALCAID.EXE -This is the CAM program used for programming the OmniTurn
PRM.SER
/OMNISLID -This has the files for running the OmniTurn
OMNISLID.EXE -OmniTurn software
EW.EXE -Word processing software
HELPE.DEF -Help pages for the word processor
RULER.DEF -Info needed for word processor
PRM.SER -parameter file for setting system functions (text file, edit with EW)
TCMP - the table of numbers used for tool offsets, it is a text file and can be edited with the word processor. It should have 32 pairs of numbers.
SECTCMP - the table of secondary tool offsets, also has 32 pairs of numbers, it is a text file and can be edited with EW
IOCFG.DAT - A text file used to define additional M function format for the I/O card
Mn..USR - the definition of the M function, a text file

B: USER PROGRAMS -The upper drive is where the B: disk is inserted. This has all of the user programs. Older systems have no directories on this disk. All files are stored on the root directory.
Since we found that there is a limitation to the number of files stored in the root we have changed to using a directory B:/NCFILES. To change your system software to have it save programs to this directory you have to change the PRM.SER file. Please refer to the page on this file at the end of this chapter.
All programs are ASCII files with no extension
file.GEO -are geometry files stored from CALCAID
file.& -are backup copies of programs stored with the word processor
file.TOF -are tool offsets that have been stored for the program
file.SOF -are secondary offsets stored for the program

C: The OmniTurn software constructs a RAM disk C: drive. This is not a hard disk, it is only temporary. All information stored on this drive is lost when the system is shut down. This ”disk” is used to speed up the operation of the word processor. Whenever the program is modified or tool offsets are corrected the information is stored to the A: drive. If you ever want to go to DOS and work on any of the files remember to save all work to the correct permanent drive.

When the OmniTum is turned on the system creates the C drive and copies everything that is in the A:\OMNISLID directory to the new C:\. The OMNISLID.EXE or OMNI2.EXE is executed from here.

Getting to DOS

Depending on your system there might be two ways to get to DOS.

First Method
- Get to the main menu
- Press and hold the left shift, while holding the shift key press ESC

Second Method (shell)
- In order to get to DOS first go to the ”MD” screen.
- Then type the command EXIT, press enter, then cycle start
- The control will eventually stop and show:

  C:>
- You are now in DOS on the RAM disk, C:

• **Off-line work - programming or word processing** - System requirements: (1) 1.44 meg floppy drive, 286 system or later, DOS 3.2 or later.

• **Off-line Programming:**

If you do have a DOS base computer you can use the software in the OmniTurn for programming in the office. If your computer has a Hard Drive you can copy the Calcaid (CALCAID.EXE) and Word processor (EW.EXE) and the Verification software (OMNI-CHK.EXE) from the system disk in the OmniTurn. To do this take the A: (3-1/2”disk) disk and place it in the disk drive of your computer.

* Make a new directory name OMNI.
    First go back to the root directory by typing CD\return
    Then type MD OMNI and return
    Go to that directory by typing CD\OMNI
* Place the system disk from the OmniTurn in the A: or B: drive.
    Make that drive active by typing A: return
    Then change-to the OMNISLID directory by typing CD\OMNISLID
* Now copy the files from this directory to the new OMNI directory on your hard drive. Type:

    COPY A:PRM.SER C:\OMNI and return
        Before using any of the following programs rest this file to save to the correct drive! See a page about setting this file on one of the following pages in this chapter.
    COPY A:EW.EXE C:\OMNI and return

Change to CALCAID directory by typing CD\CALCAID
Then type
    COPY A:*.* C: and return

Change to the verification directory CD\OMNI-CHK
Then type
    COPY A: *. * C:\OMNI

• **Word processing off line:** you can input and edit your programs on a computer in your office. You can use the word processor that you are used to like Word Perfect. (Save the file as an ASCII file) Then bring the disk out to the OmniTurn, put it into the B: drive and run it. The rules that you must follow are:

-Save the program in ASCII format
-The name must have no extension

TEST.	valid
TEST.DNE	illegal

-The program must have no commas ”,”
-After the M30 or M02 command there must be a carriage return

If you want to use the word processor that is in the OmniTurn you can copy it into your hard disk. This can be done by following the same instructions as in the previous paragraph.

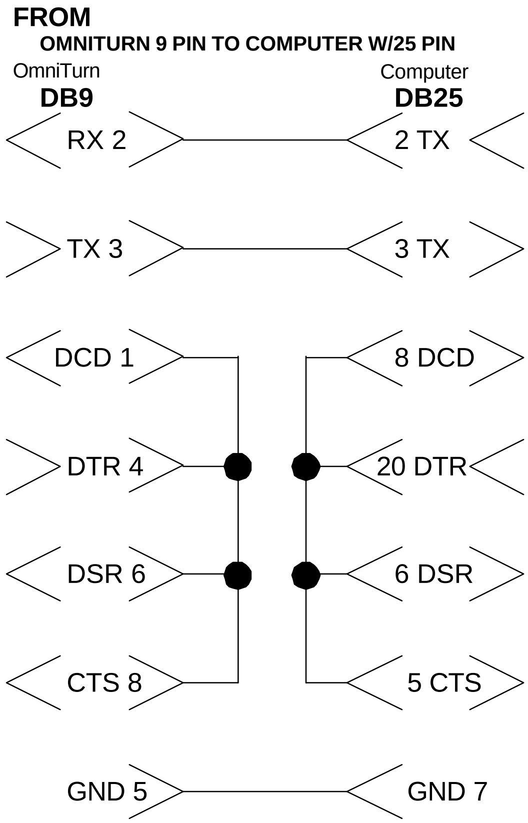
Then you can write programs at your desk and put them onto the program disk in the OmniTum.

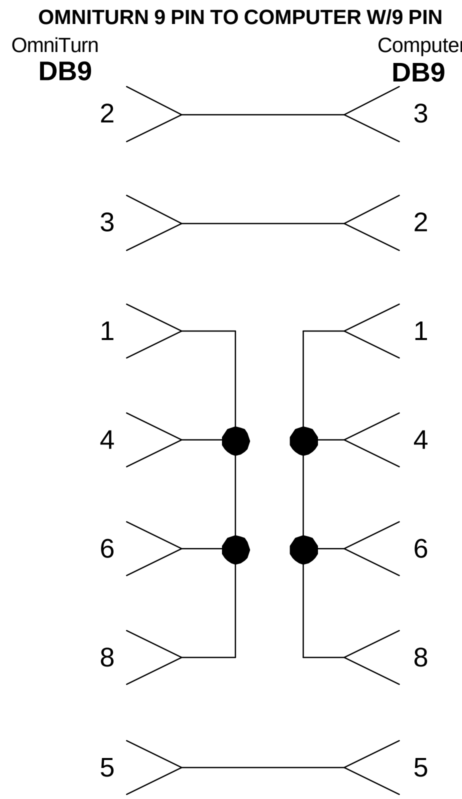
• **Printing a program off line:**

To print a program from your computer you could use the command:

    TYPE FILENAME > PRN

RS-232 Cable Default settings -2400, 8, 1, N





OmniSupport - the modem is optional

The software to run the OmniSupport is supplied in all OmniTurns. When the control is shipped the OmniSupport software is not turned on since it takes the control longer to start up if it is activated. If you need to modem into the factory turn the OmniSupport on by following the procedure below:

Turning on OmniSupport on and off

- Install a modem in any of the open slots in the computer
- The OmniSupport is turned on from DOS. To get to DOS first go to the ”main” screen.
- Then hold down the left shift and press ”ESC”
- The control will eventually stop and show: C:>
- Type: A: and return
- The screen will show A:>
- To turn OmniSupport on type: **PHONE** and return
  To turn OmniSupport off type: **NOPHONE** and return
- Now turn the control off and restart. When the control is restarted it will have the OmniSupport switched how you want it.
- To activate the OmniSupport hold the right shift down and press the ”ESC”. Then follow the instructions on the screen.

RAM BIOS Setup For a 286 Computer

Date:	
Time:	
Floppy *A*:	1.44 Mb 3 1/2”
Floppy B:	1.44 Mb 3 1/2”
Hard Disk C:	
Hard Disk D:	
Primary Display:	Monochrome
Keyboard:	Installed
Video BIOS Shadow:	**Disabled**
Scratch RAM Option:	1
Main BIOS Shadow:	**Disabled**
Turbo Speed:	Enabled
EMS Function:	Enabled
AT Bus Clock Mode:	Asynchronous

Amibios Bios Setup for 386SX and Up

Standard Setup

Time	
Date	
Hard Disk C:	Not Installed
Hard Disk D:	Not Installed
Floppy Drive A:	1.44 MB 3.5
Floppy Drive B:	1.44 MB 3.5
Primary Display:	Monochrome

Advanced Setup

Typematic Rate Program:	Disabled
Typematic Rate chars/sec:	15
Above 1 MB Memory Test:	Disabled
Memory Test Tick Sound:	Enabled
Memory Parity Error Check:	Disabled
Hit < Del > Message Display:	Enabled
Hard Disk Type 47 Ram Area:	0:300
Wait For < F1 > If Any Error:	Enabled
Numeric Processor Test:	Disabled
Floppy Drive Seek at Boot:	Enabled
System Boot Sequence:	A:, C:
Fast Gate A20 Option:	Disabled
Password:	Setup
Video RAM Shadow C000, 32K:	Disabled
System RAM Shadow F000, 64K:	Disabled
Cyrix cache:	Disabled

Networking the OmniTurn Controls

Why Network?

Getting programs to and from a control has always been a problem. There are a number of ways to get a program into a control: - Manually enter the program at the control keyboard - Run a RS-232 cable and transfer the file - Use floppy disk storage and transfer the disk - Network the control with computers in the office The ability to connect the OmniTurn control to other computers has a number advantages over the other methods.

• When networked the OmniTurn will automatically look to the network server for its programs. You do not have to ”transfer” programs to the control. When the networking is setup the OmniTurn control is configured so that it looks to its own directory on the server. The operator of the OmniTurn will not have to do anything. This is better than the RS-232 method.
• When transferring files over a RS-232 connection a number of functions must be done by the operator: Set the control to receive, go to the other computer and send the file, go to the OmniTurn and terminate the session. This is not difficult but if you have to get files all the time it can be tedious.
• Another advantage is that if the program is corrected by the operator the changes are automatically saved to the network.
• The person in charge of the OmniTurns can put only what files that are to be run in the directory for each OmniTurn. When the network is setup it is easy to have each control go to its own directory. This lessens the chance of an operator running the wrong program.

Which Network to choose?

The OmniTurn control is based on a 386 PC. This makes networking them with other computers very easy. There are a number of different network operating systems (NOS) available on the market. We have had experience with Novell Netware v3.12 and find it works very well and is easy to setup. If you have a network up and running in your office and need assistance in getting the OmniTurn plugged in, have your systems personnel give us a call and we will be happy to offer whatever help we can.

Configuring the OmniTurn

Since the OmniTurn is a PC in most cases all you will have to do is add a network card to the control and some software to the system disk. With the OmniTurn software you will have to adjust the PRM.SER file to look to the server for its programs. There are instructions in the OmniTurn manual in chapter 7 DOS notes on what to do.

Setting up new networks

There are a number of customers who have asked us to setup networks for them. We strongly feel that it is not in the best interest of our customers to have us setup networks. New computer systems and software should have someone close at hand to answer questions and help with hardware problems. Because of this we suggest that users needing a new computer system and network purchase it from a local vendor and have it installed by them. We would be very happy to help with specifying what you need and make sure that what you are getting is what is called for. The actual connection of the OmniTurn to the network is very simple and can be done in minutes.

Basic system configuration for networking

Computer: Here we list a simple system that will work in supporting a network for the OmniTurn. With the changing computer market it is possible to get a lot more computer for just a little bit more money.

- 16MB RAM memory
- 256KB Cache
- 512 MB Hard disk
- 14” monitor -
- CD-ROM (this makes loading the NOS much easier)
- 3.5” disk drive
- Keyboard and mouse

10Base-T Concentrator: SMC ELITE 3512TP 10Base-T Concentrator. This stackable hub has 12 plus 2 port repeaters with retiming on all ports. The Concentrator is easy to install.

Network Interface Cards (NIC): SMC EtherEZ l0Base-T ISA Adapter card

Novell Netware ver 3.12: 5 or 25 user on CD-ROM