

## MODES OF OPERATION

JOG: Move the table; Establish home; set tools

MDI (Manual Data Input): Execute any valid program statement

AUTOMATIC: Load, edit, run programs

### • MOVING THE TABLE (JOG MODE)

From Main Menu press J to enter Jog mode, then press one of the following:

- 1 = Jog at 1 ipm (slow)
- 2 = Jog at 10 ipm (med)
- 3 = Jog at 100 ipm (fast)
- 4 = Jog exactly .00005 (50 millionths)
- 5 = Jog exactly .001 at 1 ipm
- 6 = Jog exactly .010 at 1 ipm
- 7 = Jog exactly .100 at 10 ipm
- 8 = Jog exactly 1.000 at 20 ipm

Press Esc to return to Main Menu

### • TO ESTABLISH HOME (JOG MODE)

Like any CNC machine, home must be established once each day, when the machine is first turned on, before running any program.

Jog the table until the Z pointer is less than one division to left of zero on the Z scale, and the X pointer is less than one division in front of zero on the X scale.

Press 9, then press Cycle Start.

Table will automatically move to precise home. Insure that pointers are at zero.

### • SETTING TOOLS (JOG MODE)

Home the machine first.

•Diameter:

Take a light cut on the diameter, either OD or ID, as required. You may also touch the part with the tool tip without cutting.

Measure the diameter, either OD or ID.

Press T, press X, type the number of the tool, then press Retrn.

Type the diameter you measured, then press Retrn, then press Esc.

NOTE: If the tool is cutting on the farside of centerline of part (or "above" center line on GT75 or GTJr), either OD or ID, type a minus before the diameter (-.362).

•To set a drill:

Touch one flute of the drill against the known diameter of part.

Press T, press X, type the number of the tool, then press Retrn.

Type a number equal to the diameter of the part plus the diameter of the drill, then press Retrn, then press Esc.

NOTE: As above, if the drill is touching on the farside of centerline of part (or "above" center line on GT75/GTJr), type a minus before the sum of the diameters.

•Length:

Using a finished part in collet stop, touch tool to face of part, press T, type number of tool then press Retrn. To use an unfinished part, determine material to be removed from rough face of part, touch tool to rough face; when asked for location enter the distance from the finished face. With barstock, locate the starting point far enough from the spindle so the part can be cut off.

### • STARTING THE SPINDLE (MDI MODE)

Verify Spindle Off/Auto Switch is Auto; verify spindle override is full CW.

Verify that mechanical spindle lock is disengaged (spindle turns freely).

From Main Menu press M to enter Manual Data Input Mode

Type *Snnnn* where *nnnn* is spindle speed in RPM (S1500).

Press Retrn, then press Cycle Start button.

Type M03 (M ZERO 3) for spindle forward (ccw), M04 for spindle reverse.

Press Retrn, then press Cycle Start button.

Spindle will start.

### • TO STOP SPINDLE

Type M05, press Retrn, then press Cycle Start button;

or set spindle Off/Auto Switch to Off;

or set Spindle Override full CCW

## AUTOMATIC MODE

**FUNCTION KEYS** (Before pressing Cycle Start; after pressing Cycle Start the function keys are percent of feed rate override: 10% - 100%):

F1	QUIT	Go back to the Main menu
F2	OFFSET	Adjust tool offsets to correct the part size
F3	EDIT	Create or modify programs
F4	DIR	With no program loaded (or after F5), lists programs on disk
	VER	When program is loaded verifies program and draws tool path
F5	NEWPROG	Allows a different program to be loaded and run
F6	SEARCHTO	Allows program be started somewhere other than the start
F7	PROG	Runs Calcaid programming system
F8	DISKOP	Disk operations: <ol style="list-style-type: none"> <li>1. Copy files between hard drive and floppy</li> <li>2. Erase a program file</li> <li>3. See a listing of program files</li> <li>4. Receive a program from another computer by RS232 cable</li> <li>5. Send a program to another computer by RS232 cable</li> <li>6. Make a backup copy of the user program disk</li> <li>7. Update the system file on a hard drive</li> <li>8. Set RS232 communications parameters</li> <li>9. Return to Automatic menu</li> </ol>
F9	SECCMP	Adjust secondary tool offsets; set tool nose radius values
F10	SPECFUN	Special functions: <ul style="list-style-type: none"> <li>Preset feed rate override before pressing Cycle Start</li> <li>Set maximum spindle speed (pulley scaling factor)</li> <li>Save tool offsets for active program</li> <li>Load tool offsets for active program (must be previously saved)</li> <li>Activate Parts Counter (number of parts made)</li> <li>Activate Cycle Repeat Counter (number of parts to make)</li> </ul>

OTHER KEYS	S	Single Block .....Step through program one block at a time
	C	Cycle Repeat .....Run the program without stopping
	O	Optional Stop.....Stop the program at a block with "M01"
	/	Block Delete .....Skip block which starts with "/"

## PROGRAMMING

### • MACHINE MODES

G90	Absolute position ( <i>default</i> )
G91	Incremental position
G70	Inch mode ( <i>default</i> )
G71	Metric mode
G73	Radius programming mode ( <i>default</i> )
G72	Diameter programming mode
G76Sn	Minimum RPM limit in G96 ( <i>n</i> =RPM)
G77Sn	Maximum RPM limit in G96 ( <i>n</i> =RPM)
G94	Inches per minute mode ( <i>default</i> )
G95	Inches per revolution mode
G97	Spindle speeds set as RPM ( <i>default</i> )
G96	Spindle speeds set as surface feet per minute (SFM)

### • MOVING

<i>Feeds</i>	
G00	Rapid move
G01	Move at last feed rate
G94	Feed rate in inches per minute
G95	Feed rate in inches per revolution
F <i>n</i>	Feed rate <i>n</i> IPM or IPR

## PROGRAMMING con't

### • MOVING con't

<i>Position</i>		
G10	G10XnZ <i>n</i>	Work Shift
G92	G92XnZ <i>n</i>	Preset axis position
<i>Corners</i>		
C	XnZnC <i>n</i>	Automatic chamfer at intersection
R	XnZnR <i>n</i>	Automatic radius at intersection
<i>Arcs</i>		
G02	G02XnZnInK <i>n</i>	Clockwise, End/Center
	G02XnZnR <i>n</i>	Clockwise, End/Radius
G03	G03XnZnInK <i>n</i>	Counter Clockwise, End/Center
	G03XnZnR <i>n</i>	Counter Clockwise, End/Radius
		(I=absolute center in X; K=absolute center in Z)

### • THREADING

G33	G33XnZnInK <i>n</i> A <i>n</i> C <i>n</i> P <i>O</i>	Threading cycle; to 5tpi (0.2" lead)
G35	G35F1	Modify threading cycle for long lead; to 1tpi (1" lead)
	G35F2	Modify threading cycle for long lead; to 0.5tpi (2" lead)
G34	G34ZnInK <i>n</i> P	Threading cycle, special single pass for long lead
G36	G36	Cancels G35 (I=depth of first pass; K=lead; A=taper; C=infeed; P=vanish thread; O=single pass at finished depth)

### • CANNED CYCLES

G74	G74XnZnInUnF <i>n</i>	Box Roughing cycle
G75	G75InUnFnP <i>n</i>	Box Contour Roughing cycle
G78	G78UnFnP <i>n</i>	Rough Contour Cycle (I=amt per pass; U=amt to leave; F=feed rate)
G81	G81ZnF <i>n</i>	Drill cycle
G83	G83ZnKnFnRnLnC <i>n</i>	Peck drill cycle (K=depth per peck; F=feed rate; R=retract to; L=retract feed rate; C=clearance to cut)
C	XnZnC <i>n</i>	Automatic chamfer at intersection
R	XnZnR <i>n</i>	Automatic radius at intersection

### • TOOL COMPENSATION

G41		Left hand tool nose radius compensation
G42		Right hand tool nose radius compensation
G40		Cancel tool nose radius compensation
D	D <i>n</i>	TNR comp value or secondary offsets

### • PROGRAM CONTROL

M00		Program stop; cancels active "M" functions
M01		Optional program stop
M02		Program end; does not cancel "M" functions
M30		Program end; cancels all active "M" functions
M98	M98P <i>n</i>	Jump to SubRoutine
M99		End of SubRoutine
G04	G04F <i>n.n</i>	Dwell; seconds
T	T <i>n</i>	Tool offset call command
LS	LS <i>n</i>	Loop start
LF		Loop finish
/		Optional block delete. Press / key at Auto Menu to activate.
}	} <i>n</i>	Start of subroutine number <i>n</i> (on line by itself)

*Optional with C-Axis or PLC*

M97	M97InC <i>n</i> P <i>n</i>	Jump to SubRoutine, Conditional (PLC)
M95		Jump to SubRoutine on Input #7 (C-axis)
M94		Dwell until Input #6 goes ON (C-axis)
M93		Dwell until Input #6 goes OFF (C-axis)

### • MACHINE CONTROL

<i>Spindle</i>		
M03		Spindle CW
M04		Spindle CCW
M05		Spindle stop
S	S <i>n</i>	Spindle speed, RPM or SFM
G97		Spindle speed set as RPM
G96		Spindle speed set as surface feet
G76	G76Sn	Minimum spindle speed while in CSF
G77	G77Sn	Maximum spindle speed while in CSF
M19		Home spindle (opt w/C-axis)
CA	CA <i>n</i>	Absolute angle (opt w/C-axis)
CI	CI <i>n</i>	Incremental angle (opt w/C-axis)
<i>Other</i>		
M08		Coolant on
M09		Coolant off
M12		Collet clamp
M13		Collet unclamp
M25/M26		User assigned on/off

**EDITOR COMMANDS**

• **LOAD PROGRAM**

First program of the day:  
Go to Auto Mode; type name of program, press Retrn.  
Subsequent programs:  
Go to Auto Mode; press F5 (new program); type name of program, press Retrn.

• **EDIT PROGRAM**

Load program, press F3, then press Esc

• **CREATE PROGRAM**

First program of the day:  
Go to Auto Mode; type name of program, press Retrn.  
At \*filename NOT FOUND\* prompt, press F3 twice, then press F1.  
Subsequent programs:  
Go to Auto Mode; press F5 (new program); type name of program, press Retrn.  
At \*filename NOT FOUND\* prompt, press F3 twice, then press F1.

• **USING THE EDITOR**

Moving Around:  
Arrows to move left/right up/down one line/character  
End Key to go to end of line  
Home Key to go to beginning of line.  
Alt Key and + to go to beginning of program.  
Alt Key and - to go to end of program.  
Shift Key and PgUp (Coolant) to go up one screen  
Shift Key and PgDn (Parts Catch) to go down one screen

New Line: Press Retrn Key.

To Erase at left of cursor: Backspace Key  
To erase at cursor: Del Key

To copy text: Press F3 at beginning of text, arrow to end of text then press F3 again; arrow to new location, then press F3 to make copy of text.

To copy text to disk: Press Ctrl key and F6 at beginning of text, arrow to end of text then press Ctrl key and F6 again; type name for file then press Retrn. Useful for saving part of a program for use in another program.

To insert text from disk: Press Ctrl key then F3 key; type name of file then press Retrn. Useful for inserting special routines, like NPT threads, which are saved on disk.

To move text: Press F6 at beginning of text, arrow to end of text then press F6 again; arrow to new location, then press F6 to move text.

To delete text: Press F4 at beginning of text, move to end of text then press F4 again. (Press Esc to recover deleted text).

To clear text marks: Press F5.

To Search: Press F9, type text to search for, then press + Key on keypad to search forward, - Key on keypad to search backward.

To Replace: Press F9, type text to search for; Press F10, type text to replace with, then press + Key on keypad to search forward; then press F10 to replace each occurrence.  
To search and replace all, press Alt Key and F10 key instead of + Key .

• **SAVING YOUR WORK**

To Save and Continue Editing (**do this often**): Press F1, then F3.

To Save program under another name: Press F1, then F5

To Save and Exit Editor: Press F1, then F2

**Threading Rules of Thumb:**

**Unified Threads**

Depth (per side) OD threads =  $.61343 / \text{TPI}$   
Depth (diameter) OD threads =  $1.22685 / \text{TPI}$   
Depth (per side) ID threads =  $.54127 / \text{TPI}$   
Depth (diameter) ID threads =  $1.08254 / \text{TPI}$

**Pipe Threads**

Depth (per side) OD/ID threads =  $8 \times \text{Lead}$   
Depth (diameter) OD/ID threads =  $1.6 \times \text{Lead}$   
Taper (per side) =  $.031135 \times \text{length of threading pass (include start-up)}$   
Taper (diameter) =  $.06227 \times \text{length of threading pass (include start-up)}$

Maximum RPM =  $250 \times \text{TPI}$

**Number of Passes:**

$(72 \times \text{Lead}) + 4$  if lead is inch  
 $(2.8 \times \text{Lead}) + 4$  if lead is mm

**First Pass Depth** ("i" in G33 statement):

Depth (per side) / square root of Number of Passes

**Starting X Position, Diameter Mode (G72):**

Major Diameter minus First Pass Depth (diameter)

**Starting X Position, Radius Mode (G71):**

Major Radius minus First Pass Depth (per side)

**Starting Z Position:** .2" minimum from start of thread

**Threading Definitions:**

TPI = Threads per Inch  
Lead =  $1 / \text{TPI}$  (inches per revolution)

**OmniTurn G-Code fragment for 1/4 - 18 NPT thread (OD):**

First turn contour per NPT specification, then call threading tool.  
Offset for tool is at point of tool in X, center of tool in Z.

```
⋮  
g00 (rapid)  
t3 (call threading tool)  
x.497z.2 (locate in X at depth of first pass; start-up in Z)  
g33x.4205z-.402i.02k.0556a.0312p (cut taper thread to NPT specs)  
g00z.5 (rapid back to safe)  
⋮
```

Code fragments for making OD & ID NPT for 1/16" to 3/4" are included with each OmniTurn in two files called NPT-OD and NPT-ID .

West Coast Sales & Service:  
(541) 332-7004 ph (541) 332-1018 fax

website: [omniturn.com](http://omniturn.com)  
email: [sales@omniturn.com](mailto:sales@omniturn.com)

East Coast Sales & Service:  
(516) 694-9400 ph (516) 694-9415 fax



OmniTurn  
CNC Lathes

Ready Reference Card

