OmniTurn Training

Jeff Richlin
631 694 9400
jrichlin@gmail.com

Codes Honored by the OmniTurn control

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 (G3 and G2 control only)	6,29,36,74
G36	G36	Cancels G35 (G3 and G2 controls only)	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
G88	G88	Precision C axis orientation	
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
M01	M01	Optional stop	61
M02	M02	End program - does not cancel active"M" functions	26,62,65
M03	M03Sn	Spindle on, CW (spindle top coming)	16,62,65,74
M04	M04Sn	Spindle on, CCW (spindle top going)	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
M88	M88	C axis spindle orientation - Precision	
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	63
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

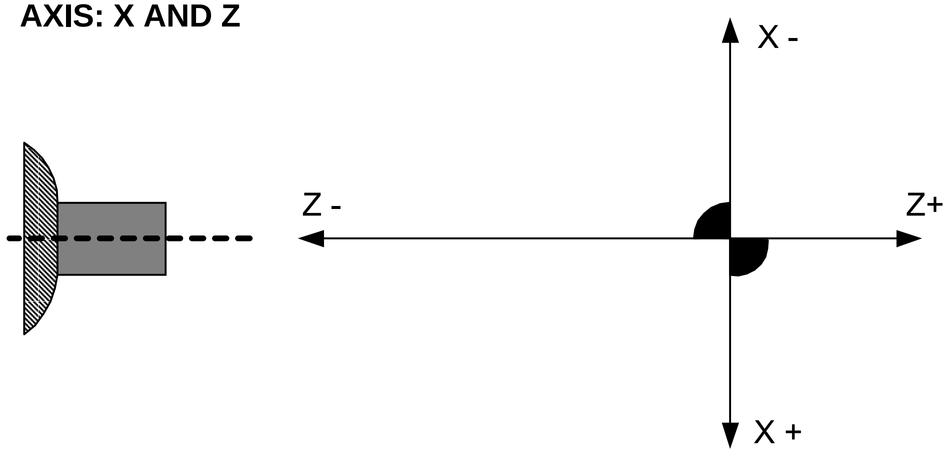
2.3

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
M01	M01	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

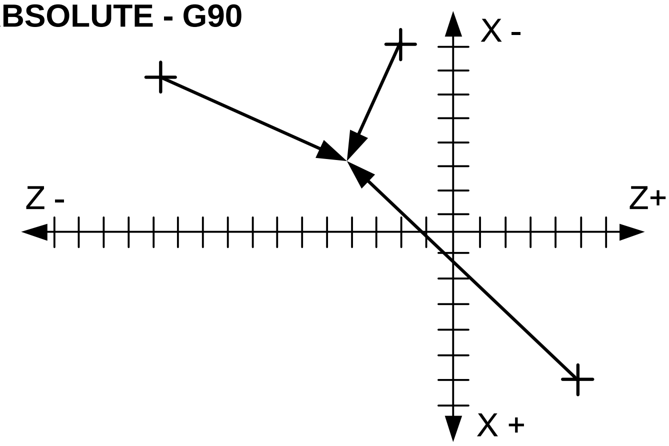
Nomenclature

AXIS: X AND Z



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

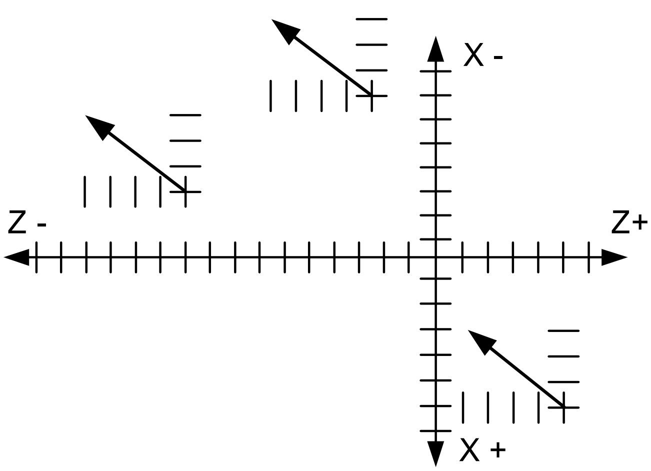
ABSOLUTE - G90



IN ABSOLUTE (G90) THE FOLLOWING MOVE BRINGS YOU TO THE SAME POINT NO MATTER WHERE YOU START

X-3 Z-4

INCREMENTAL - G91



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

X-3 Z-4

Programming Format

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

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

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

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

  - Do not put text on lines by itself. Comments must be on a line with a command!
  - Keep the amount of text to a minimum, 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.

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 incrimentally. 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.

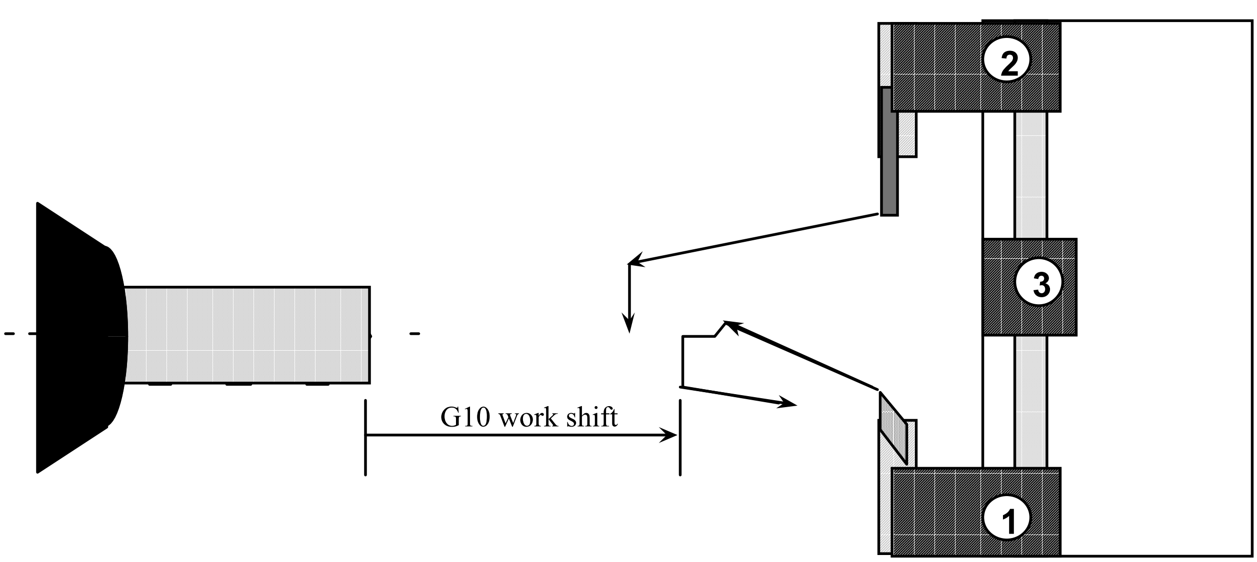
Note: ***The shift is executed on <u>tool calls</u>! Use the G10 command before tool calls otherwise there will be no effect.***

The shift will be cancled with these commands: T0 - M30 - M02

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

Example of shifting a program for test running

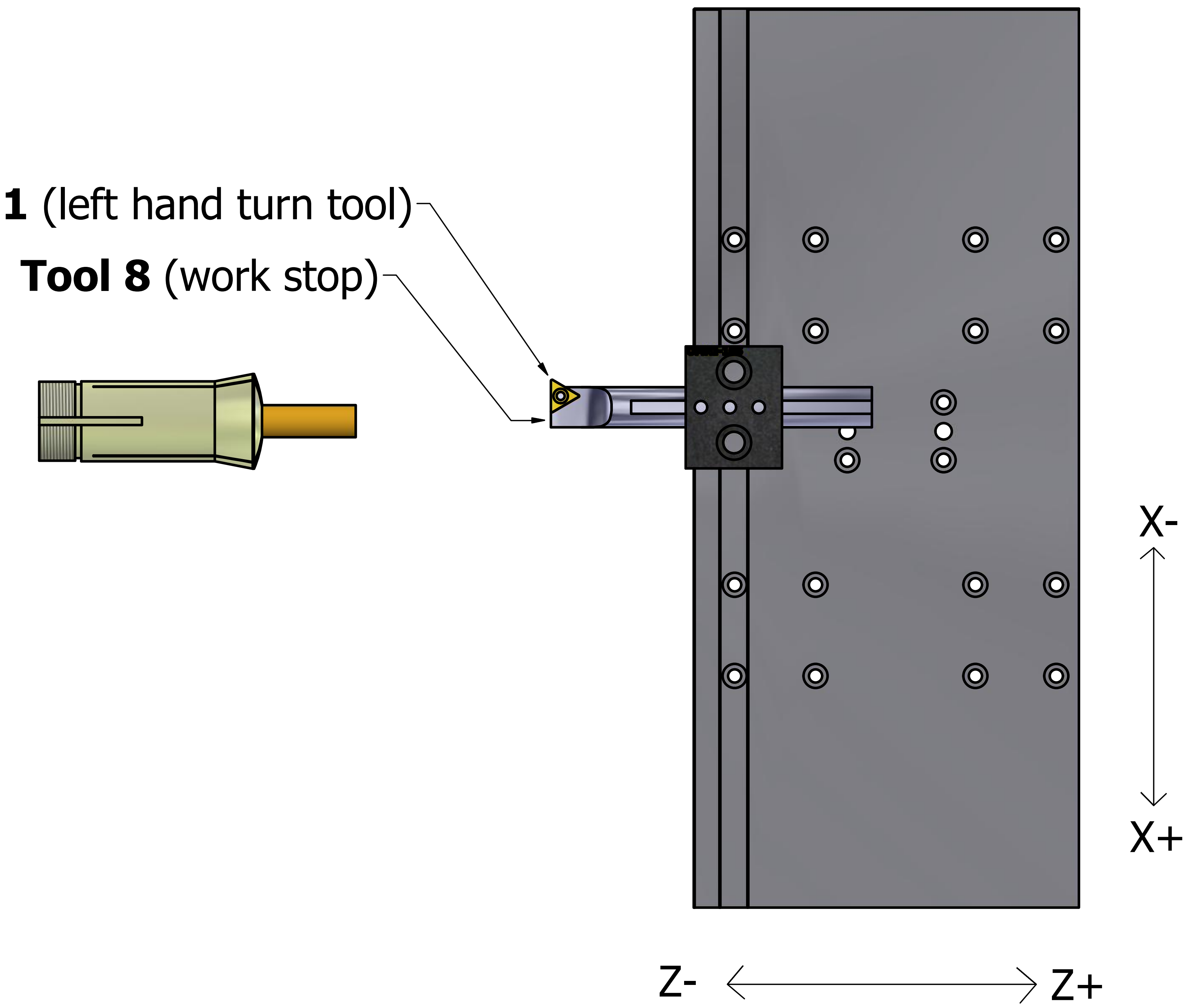
In the following example we show using the G10 work shift for running a program the first time way from the work to make sure that the program looks like it will run OK. In this example you would set the tools to make the part. After the program is run a few inches away with the work shift the G10 command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

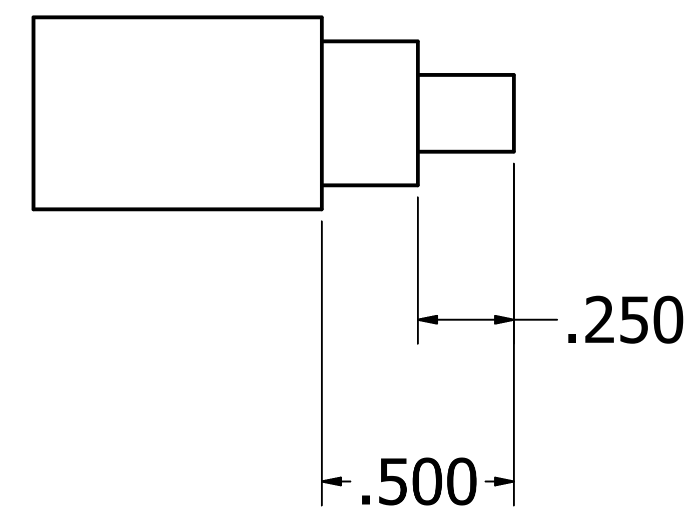


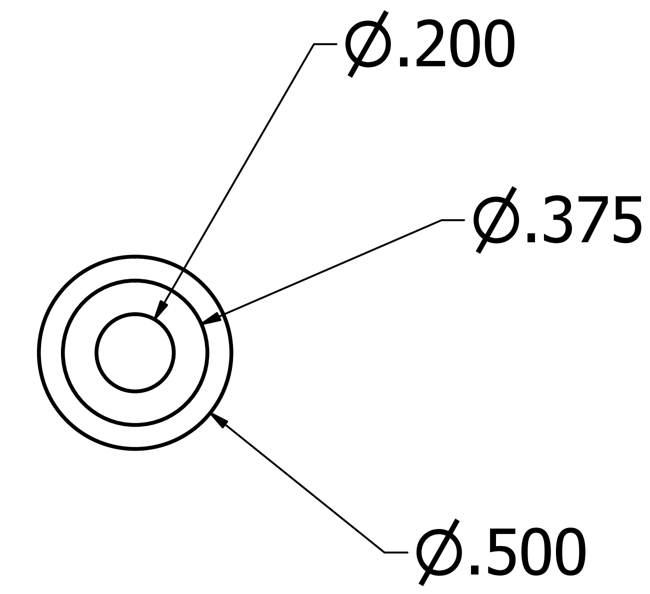
G90G72G94F300
M03S2000
G04F2
G10X0Z3 **Shifts program 3" to the right**
T1(LH TURN TOOL)
X0Z.2
Z0G95F.01
X.1Z-.05F.002
Z-.15
X.2
G94F300Z1
T2(PART OFF TOOL)

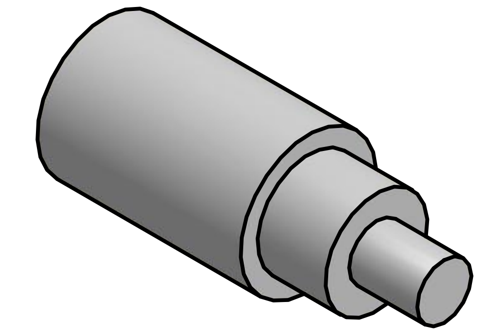
Notes

Topic: Basic program format and structure









Sample 1

Process:

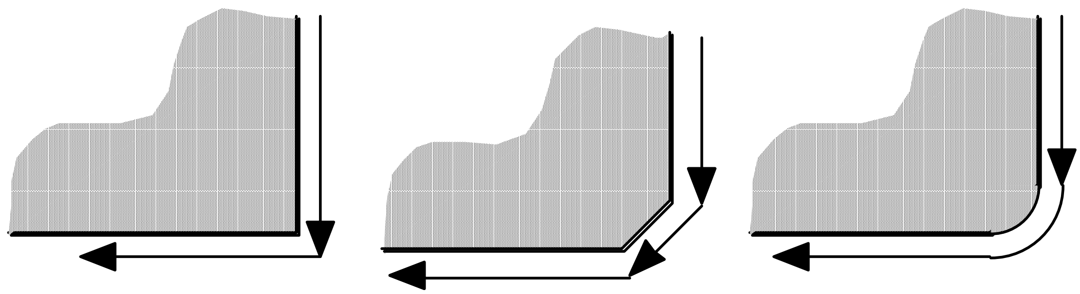
Use the front of the tool as a work stop
T8, stop motion, pull part to stop, close collet
Use T1 (LH turn tool) to rough the part as shown

DRAWN		
JRichlin	3/17/2010	Richlin Machinery Inc **OmniTurn Training 631 694-9400**
		TITLE **Part #1**
Material:		SIZE C — DWG NO **OmniTurn Training** — REV
		SHEET 1 OF 11

Part: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.iam
Drawing: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.idw

```
G90G72G94F300 (PART-1)
G10X0Z2
T8(WORK STOP)
X0Z2
Z.2
F50Z.01
M00 (PULL PART TO STOP)
M12
Z2F300
M03S1500
M08
T1(LH TURN TOOL)
X.6Z2
Z0
G95F.003X-.015
X.2F.01
Z-.25F.003
X.375
Z-.5
X.6
G94F300Z2
M30
```

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.



Programmed move Move with C value Move with R value

Format

X*n*Z*n***R*n*** - Z*n***R*n*** - X*n***R*n***

X*n*Z*n***C*n*** - Z*n***C*n*** - X*n***C*n***

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.

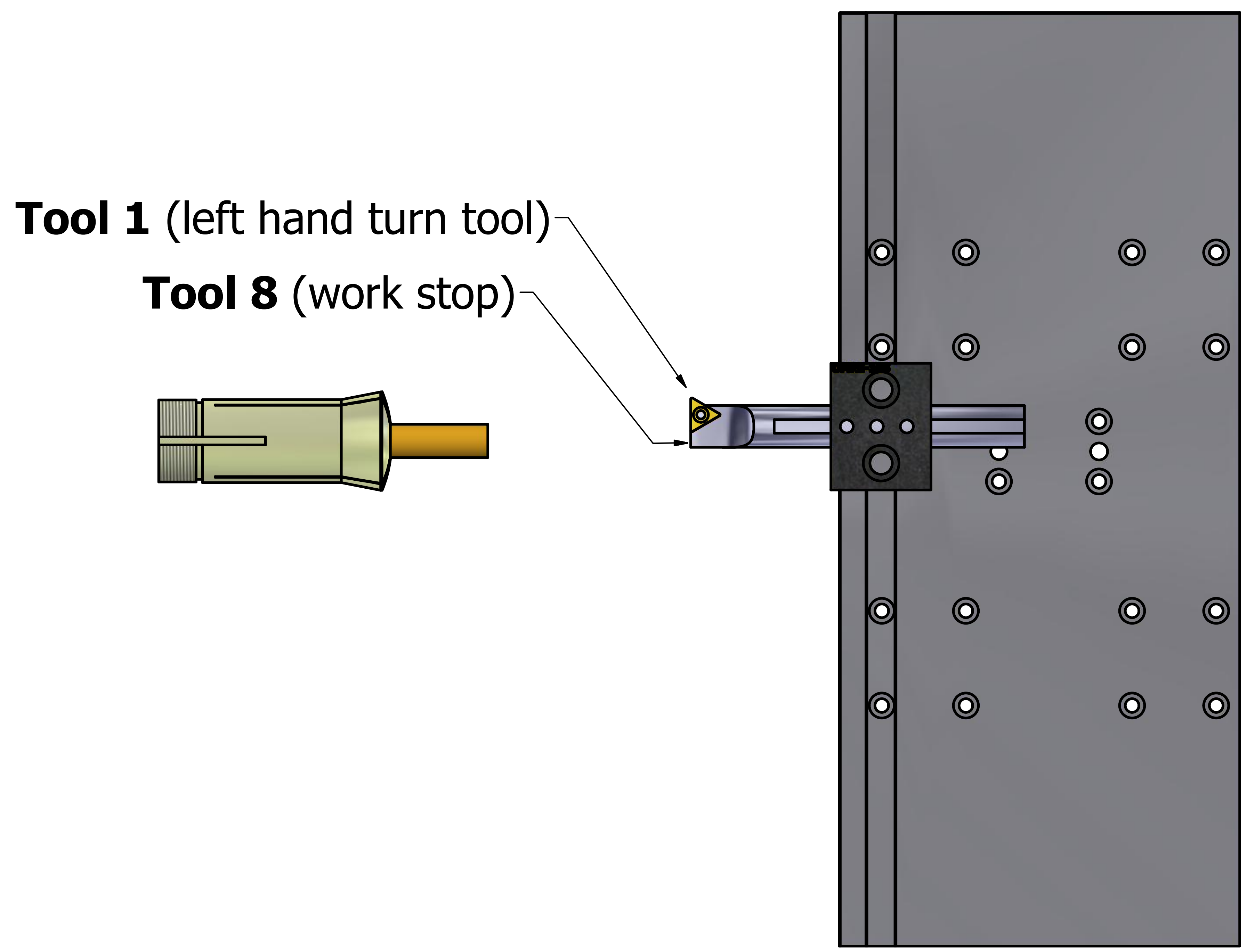
A radius created is made tangential to the two intersecting lines. The direction (CW or CCW) of the radius is determined automatically by the OmniTurn. It looks ahead to the next move.

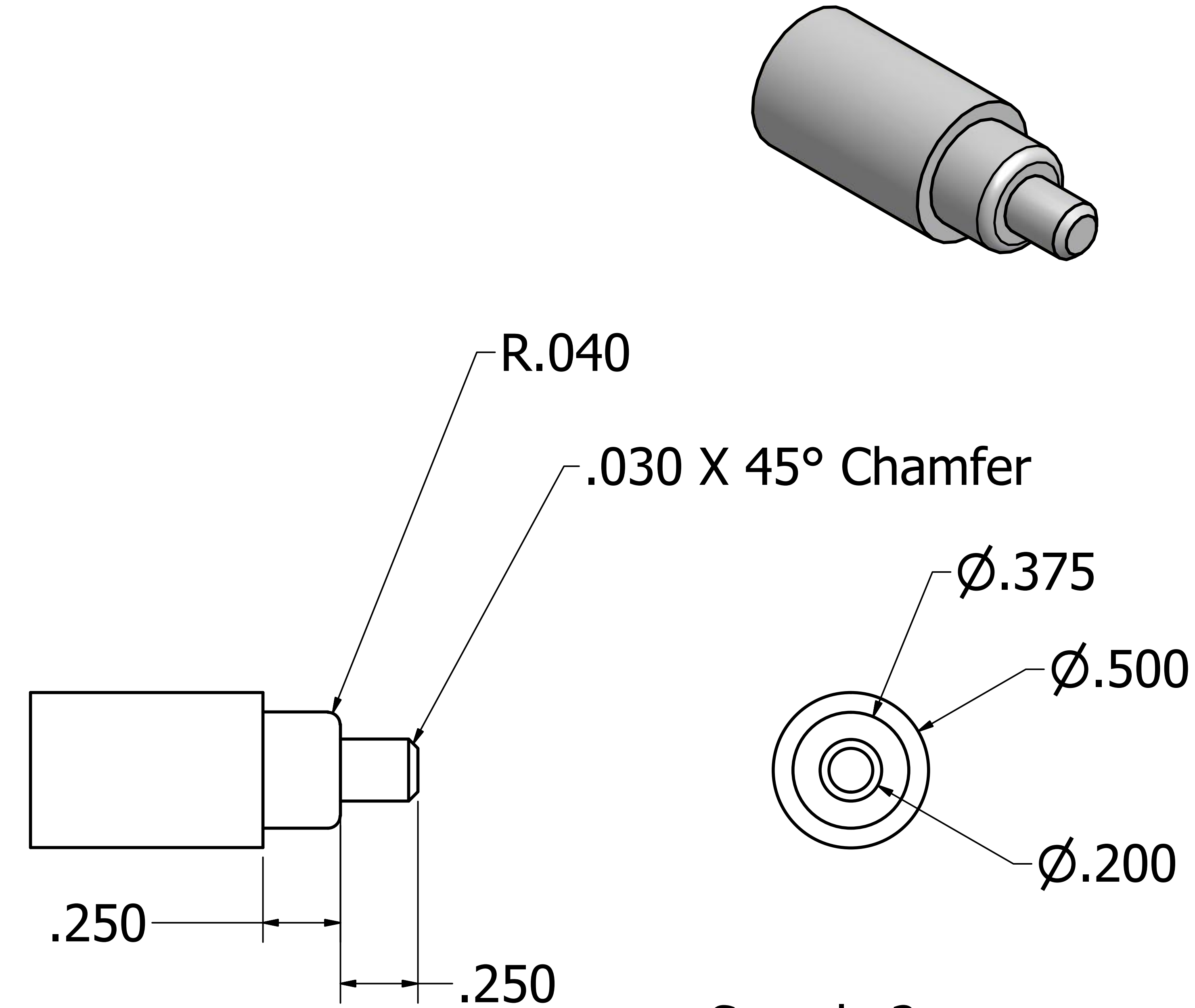
The n value must be the absolute (+) value

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

Notes

Topic: Corner chamfer and corner radius





Sample 2

Process:

Use the front of the tool as a work stop
T8, stop motion, pull part to stop, close collet
Use T1 (LH turn tool) to rough the part as shown
adding the chamfer and radius moves

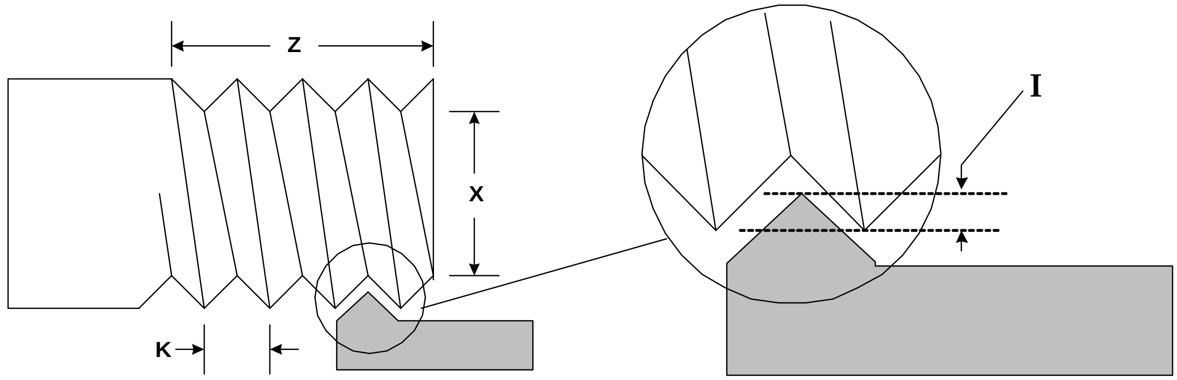
DRAWN JRichlin	3/17/2010	Richlin Machinery Inc **OmniTurn Training 631 694-9400**
		TITLE **Part #2**
Material:		SIZE C · DWG NO **OmniTurn Training** · REV
		SHEET 2 OF 11

Part: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.iam
Drawing: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.idw

```
G90G72G94F300 (PART-2)
G10X0Z2
T8(WORK STOP)
X0Z2
Z.2
F50Z.01
M00 (PULL PART TO STOP)
M12
Z2F300
M03S1500
M08
T1(LH TURN TOOL)
X.6Z2
Z0
G95F.003X-.015
X.2C.03F.01
Z-.25F.003
X.375R.04
Z-.5
X.6
G94F300Z2
M30
```

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:

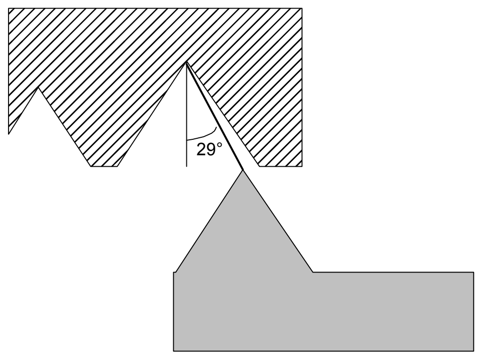
The tool will back away from the starting position plus 3 times the amount of I. Even as the tool gets deeper into the material it will always retract to the same point.

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:

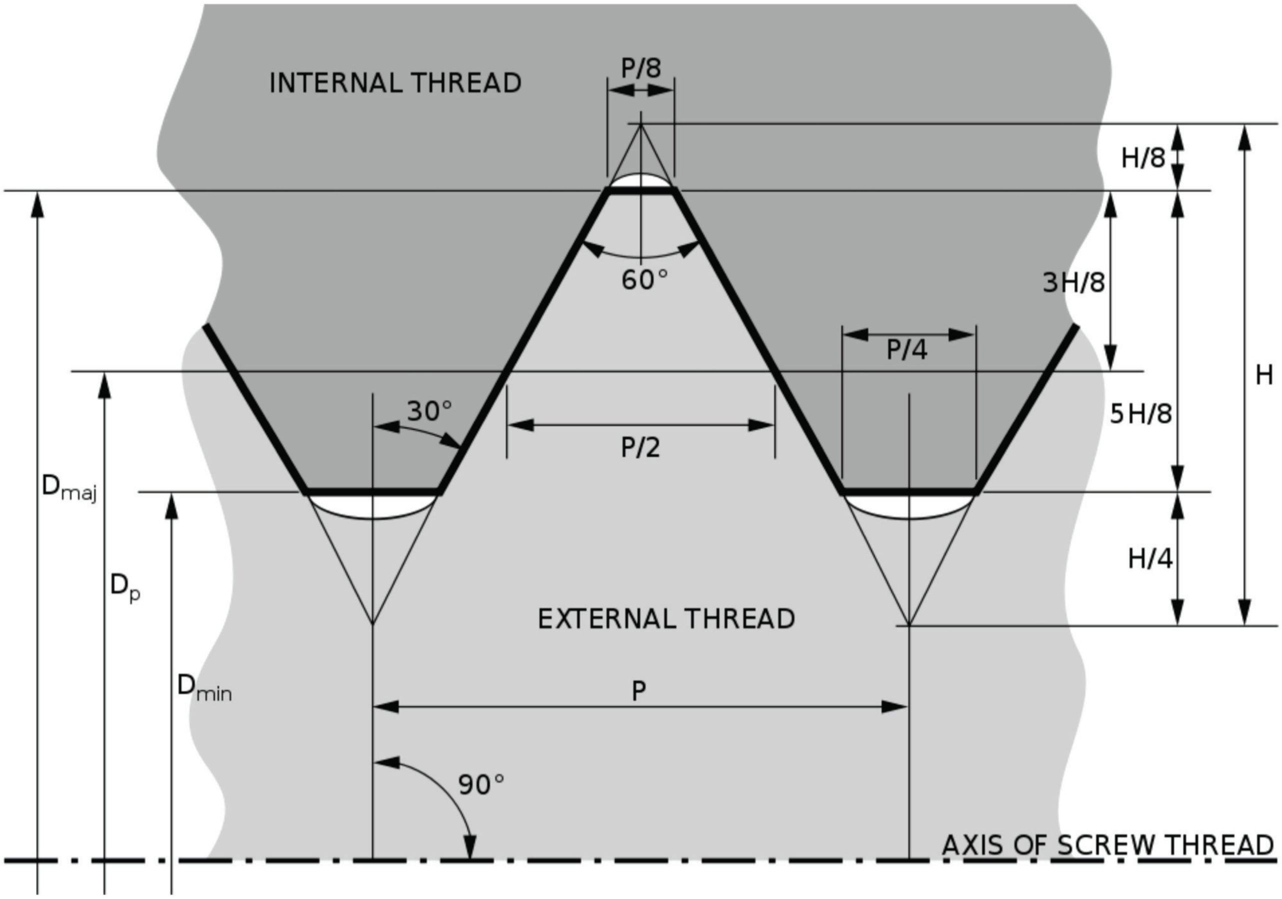
When a single pass is needed write the same threading pass as used for cutting the thread. Just add a O to the command. Be sure to start the thread at the same point and at the same spindle speed. This option can be used with all variations of the threading command.

End of cycle position:

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

THREAD/SCREW - Drill & Tap Chart

Machine Screw Size		Threads Per Inch	Minor Dia	Tap Drills				Clearance Hole Drills			
				Alum, Brass, & Plasitcs 75% Thread		Stainless Steel, Steels & Iron 50% Thread		All Materials			
								Close Fit		Free Fit	
# or Dia	Major Dia			Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.
0	.0600	80	.0447	3/64	.0469	55	.0520	52	.0635	50	.0700
1	.0730	64	.0538	53	.0595	1/16	.0625	48	.0760	46	.0810
		72	.0560	53	.0595	52	.0635				
2	.0860	56	.0641	50	.0700	49	.0730	43	.0890	41	.0960
		64	.0668	50	.0700	48	.0760				
3	.0990	48	.0734	47	.0785	44	.0860	37	.1040	35	.1100
		56	.0771	45	.0820	43	.0890				
4	.1120	40	.0813	43	.0890	41	.0960	32	.1160	30	.1285
		48	.0864	42	.0935	40	.0980				
5	.125	40	.0943	38	.1015	7/64	.1094	30	.1285	29	.1360
		44	.0971	37	.1040	35	.1100				
6	.138	32	.0997	36	.1065	32	.1160	27	.1440	25	.1495
		40	.1073	33	.1130	31	.1200				
8	.1640	32	.1257	29	.1360	27	.1440	18	.1695	16	.1770
		36	.1299	29	.1360	26	.1470				
10	.1900	24	.1389	25	.1495	20	.1610	9	.1960	7	.2010
		32	.1517	21	.1590	18	.1695				
12	.2160	24	.1649	16	.1770	12	.1890	2	.2210	1	.2280
		28	.1722	14	.1820	10	.1935				
		32	.1777	13	.1850	9	.1960				
1/4	.2500	20	.1887	7	.2010	7/32	.2188	F	.2570	H	.2660
		28	.2062	3	.2130	1	.2280				
		32	.2117	7/32	.2188	1	.2280				
5/16	.3125	18	.2443	F	.2570	J	.2770	P	.3230	Q	.3320
		24	.2614	I	.2720	9/32	.2812				
		32	.2742	9/32	.2812	L	.2900				
3/8	.3750	16	.2983	5/16	.3125	Q	.3320	W	.3860	X	.3970
		24	.3239	Q	.3320	S	.3480				
		32	.3367	11/32	.3438	T	.3580				
7/16	.4375	14	.3499	U	.3680	25/64	.3906	29/64	.4531	15/32	.4687
		20	.3762	25/64	.3906	13/32	.4062				
		28	.3937	Y	.4040	Z	.4130				
1/2	.5000	13	.4056	27/64	.4219	29/64	.4531	33/64	.5156	17/32	.5312
		20	.4387	29/64	.4531	15/32	.4688				
		28	.4562	15/32	.4688	15/32	.4688				
9/16	.5625	12	.4603	31/64	.4844	33/64	.5156	37/64	.5781	19/32	.5938
		18	.4943	33/64	.5156	17/32	.5312				
		24	.5114	33/64	.5156	17/32	.5312				
5/8	.6250	11	.5135	17/32	.5312	9/16	.5625	41/64	.6406	21/32	.6562
		18	.5568	37/64	.5781	19/32	.5938				
		24	.5739	37/64	.5781	19/32	.5938				
11/16	.6875	24	.6364	41/64	.6406	21/32	.6562	45/64	.7031	23/32	.6562
3/4	.7500	10	.6273	21/32	.6562	11/16	.6875	49/64	.7656	25/32	.7812
		16	.6733	11/16	.6875	45/64	.7031				
		20	.6887	45/64	.7031	23/32	.7188				
13/16	.8125	20	.7512	49/64	.7656	25/32	.7812	53/64	.8281	27/32	.8438
7/8	.8750	9	.7387	49/64	.7656	51/64	.7969	57/64	.8906	29/32	.9062
		14	.7874	13/16	.8125	53/64	.8281				
		20	.8137	53/64	.8281	27/32	.8438				
15/16	.9375	20	.8762	57/64	.8906	29/32	.9062	61/64	.9531	31/32	.9688
1	1.000	8	.8466	7/8	.8750	59/64	.9219	1-1/64	1.0156	1-1/32	1.0313
		12	.8978	15/16	.9375	61/64	.9531				
		20	.9387	61/64	.9531	31/32	.9688				



The pitch P is the distance between thread peaks. For UTS threads, which are single-start threads, it is equal to the lead, the axial distance that the screw advances during a 360° rotation. UTS threads do not usually use the pitch parameter; instead a parameter known as threads per inch (TPI) is used, which is the reciprocal of the pitch.

P = 1 / TPI .05 = 1 / 20 .03125 = 1 / 32

The outermost 0.125 and the innermost 0.25 of the height H of the V-shape are cut off from the profile.

H = .866 x P

Major diameter = Screw # × 0.013" + 0.060".
For example, a number 10 calculates as: #10 × 0.013" + 0.060" = 0.190" major diameter.

The formula for number sizes smaller than size #0 is given by Major diameter = 0.060" - Zero size × 0.013", with the zero size being the number of zeroes after the first.
So a #00 screw is .047" dia, #000 is .034" dia, etc.

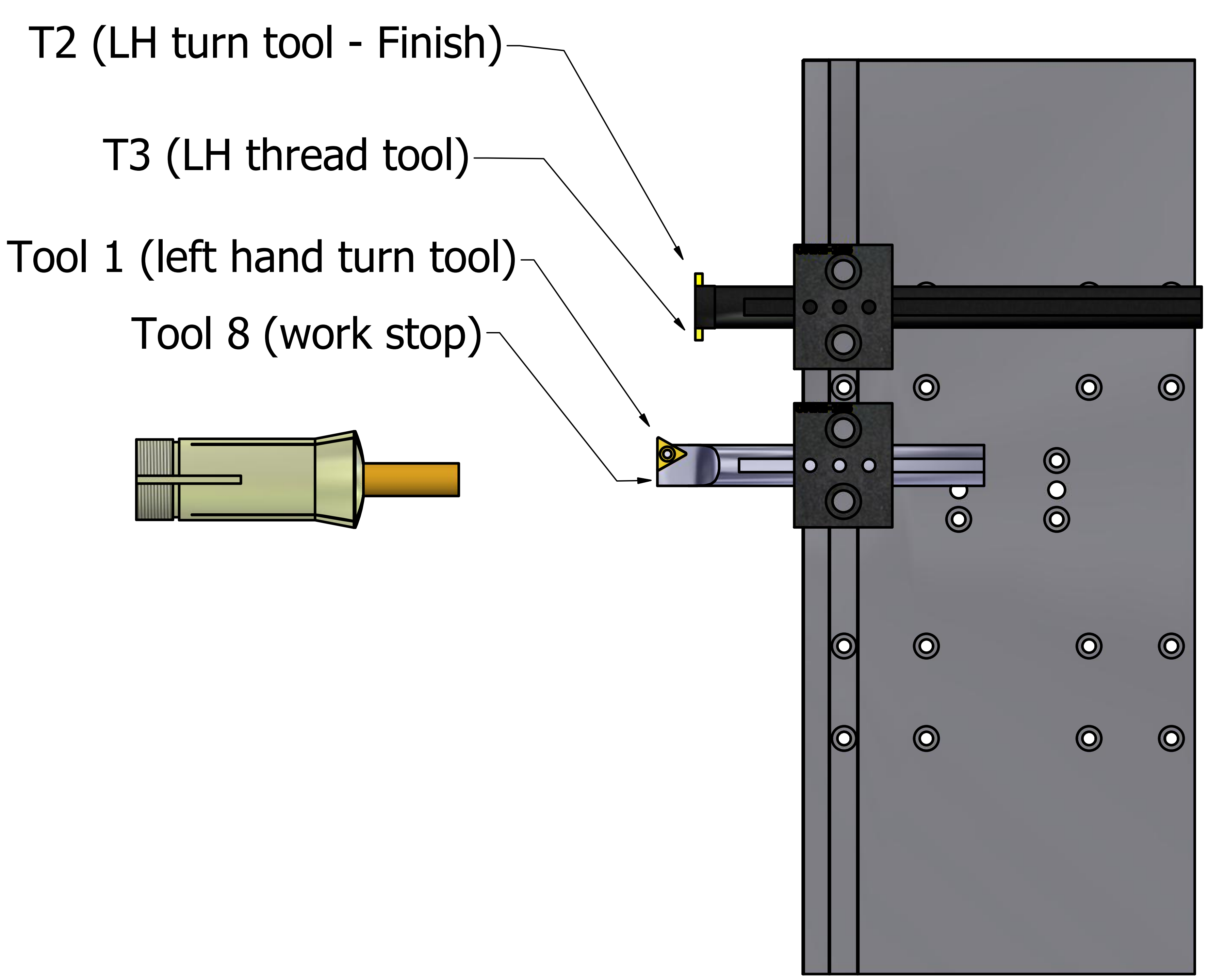
Classes 1A, 2A, 3A apply to external threads; Classes 1B, 2B, 3B apply to internal threads.

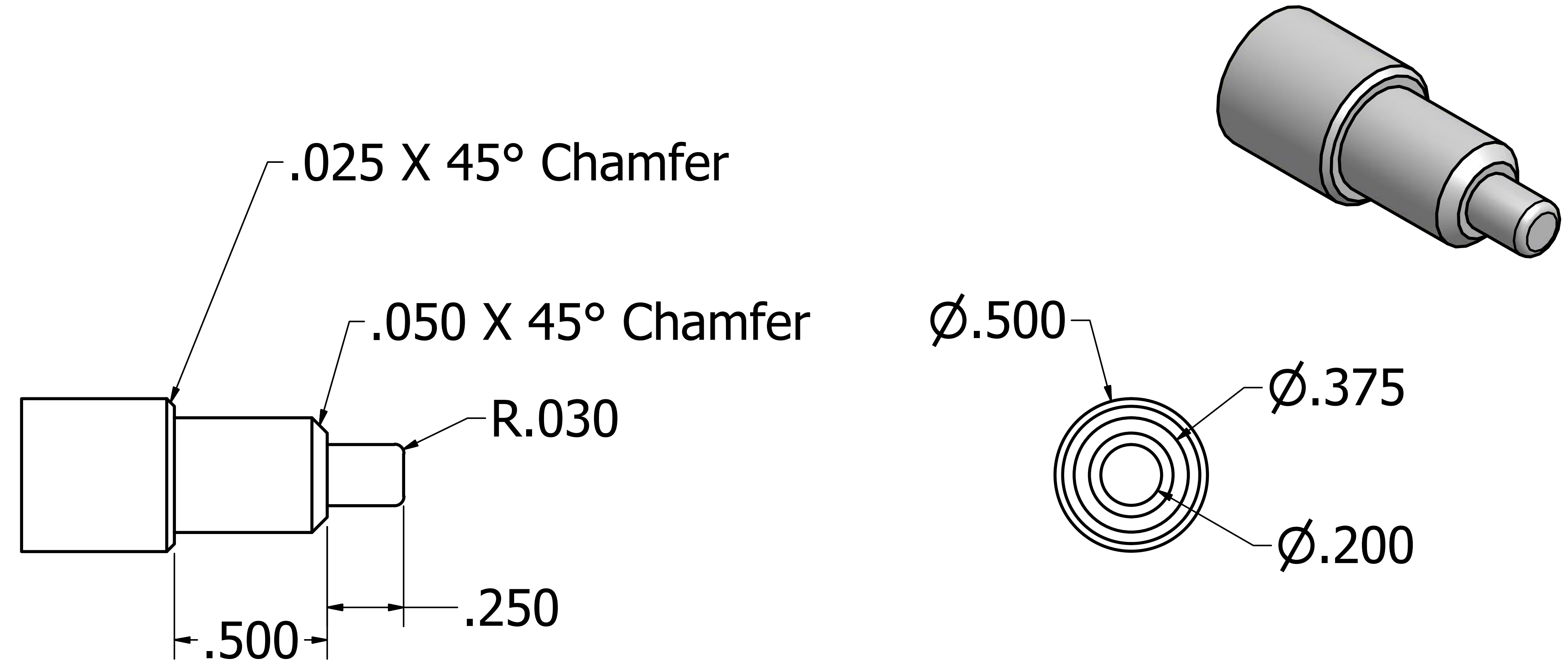
Class 1 threads are loosely fitting threads intended for ease of assembly or use in a dirty environment.

Class 2 threads are the most common. They are designed to maximize strength considering typical machine shop capability and machine practice.

Class 3 threads are used for closer tolerances.

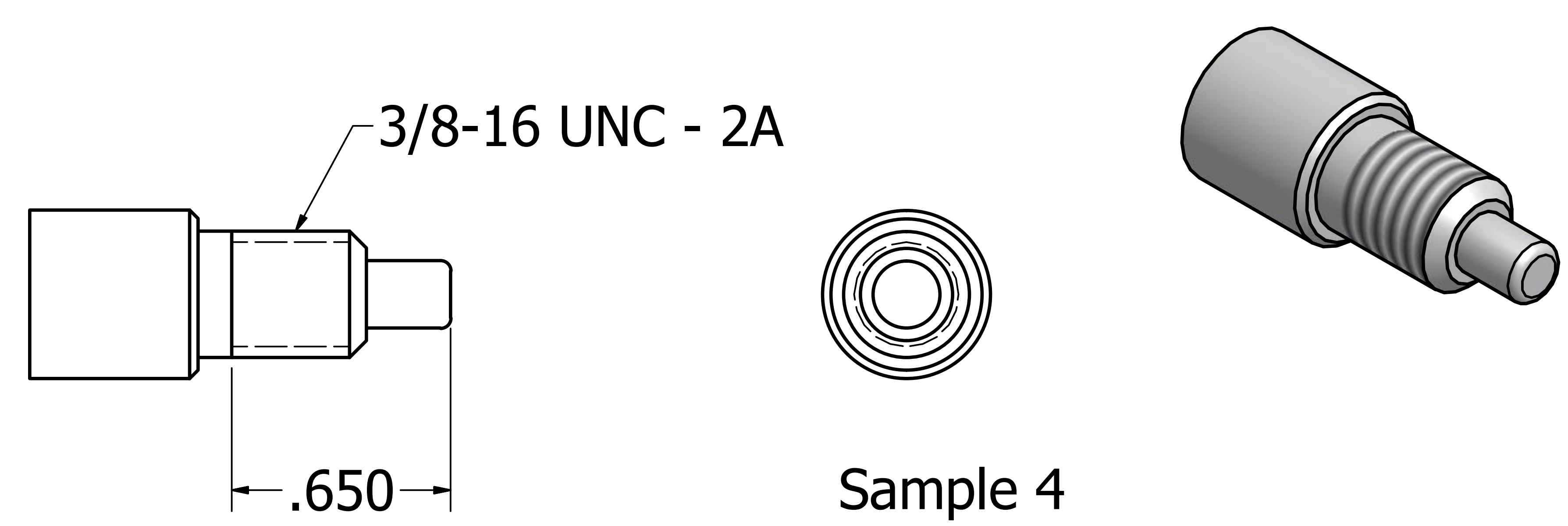
Topic: Repeat basic programming example
Topic: Single point threading
Topic: Tools on the (-) side of the spindle





Sample 3

Sample 3 Process:
Use the front of the tool as a work stop
T8, stop motion, pull part to stop, close collet
Use T1 (LH turn tool) to rough the part as shown
Leave .01 to clean up on a finish pass
Use T2 (LH turn tool - finish)
perform a cleanup pass with the chamfers and radius



Sample 4

Sample 4 Process:
Use the front of the tool as a work stop
T8, stop motion, pull part to stop, close collet
Use T1 (LH turn tool) to rough the part as shown
Leave .01 to clean up on a finish pass
Use T2 (LH turn tool - finish)
Perform a cleanup pass with the chamfers and radius
Uset T3 (LH thread tool) to thread OD
Major diameter .375, minor diameter .298

Part: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.iam
Drawing: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.idw

DRAWN JRichlin	3/17/2010	Richlin Machinery Inc OmniTurn Training 631 694-9400
		TITLE **Part #3 and Part #4**
Material:		SIZE C · DWG NO OmniTurn Training · REV
		SHEET 3 OF 11

```
G90G72G94F300 (PART-3)
G10X0Z2
T8(WORK STOP)
X0Z2
Z.2
F50Z.01
M00 (PULL PART TO STOP)
M12
Z2F300
M03S1500
M08
T1(LH TURN TOOL - rough)
X.6Z2
Z0
G95F.003X-.015
X.22F.01
Z-.24F.003
X.395
Z-.74
X.6
G94F300Z2
T2(LH TURN TOOL - finish)
X.2Z2
Z.2
G95F.003
Z-.25F.003
X.375C.05
Z-.75
X.6C.075
Z-.95
G94F300Z2
M30
```

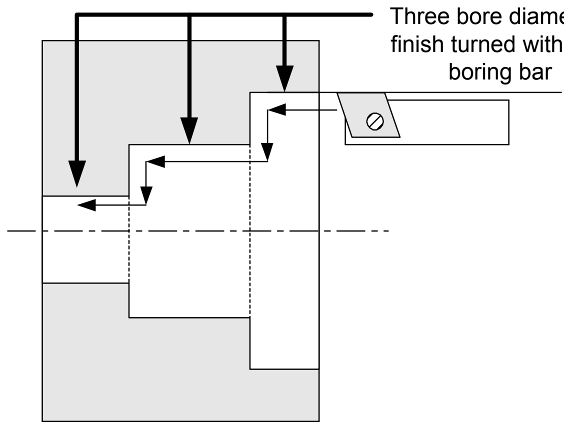
```
G90G72G94F300 (PART-4)
G10X0Z2
T8(WORK STOP)
X0Z2
Z.2
F50Z.01
M00 (PULL PART TO STOP)
M12
Z2F300
M03S1500
M08
T1(LH TURN TOOL - rough)
X.6Z2
Z0
G95F.003X-.015
X.22F.01
Z-.24F.003
X.395
Z-.74
X.6
G94F300Z2
T2(LH TURN TOOL - finish)
X.2Z2
Z.2
G95F.003
Z-.25F.003
X.375C.05
Z-.75
X.6C.075
Z-.95
G94F300Z2
T3 (THREADING TOOL)
X-.365Z2
Z.2
G95
G33X-.298Z-.65K.0625I.03C
M30
```

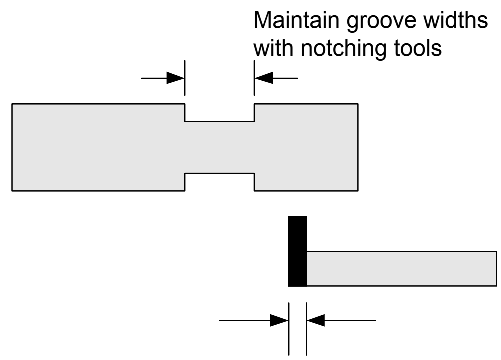
Secondary Offsets

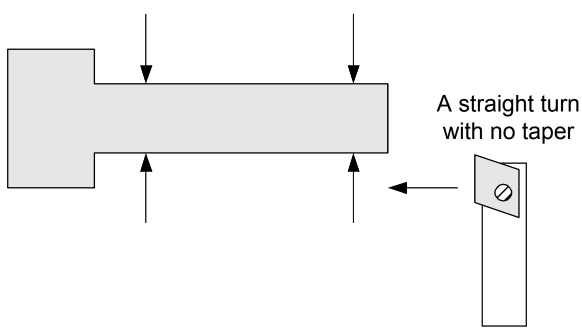
What are secondary offsets?

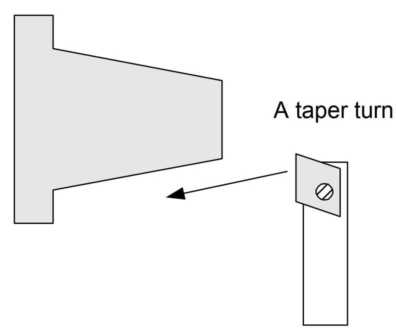
Secondary offsets are corrections that you can put into your program that the operator can adjust when running the program without having to go into the program to edit it. Once the program has been written with the secondary offsets incorporated, these corrections are made by pressing F9 while in the 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.

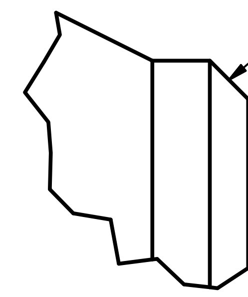
If, however, you had a taper in the long thin part (lower left sample) and had to correct it to get the part straight, offset changes would not help. The secondary offset allows you to add or subtract a little, to any move, at any point in the program. So the correction of the taper can easily be taken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

NOTE: Clear secondary offsets before using them!

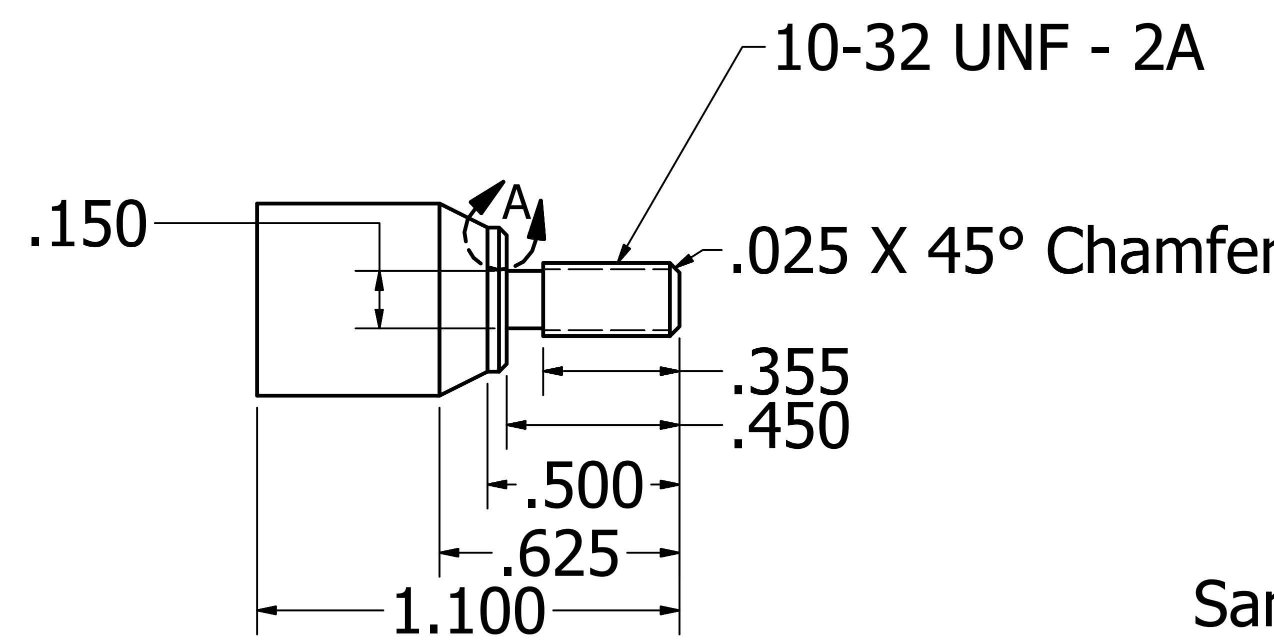
Before you run a program that uses secondary offsets be sure that you have reset the secondary offsets that you are using to zero! This can be done by pressing C when asked to make a correction to the offset table (See F9 in the Automatic section)

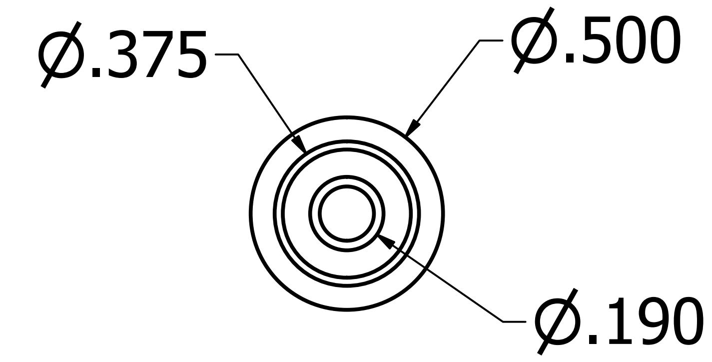
Notes

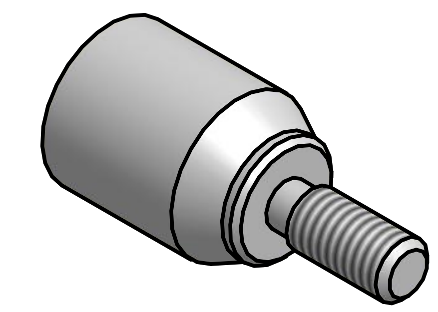
Topic: Repeat Single point threading
Topic: Grooving with secondary offsets



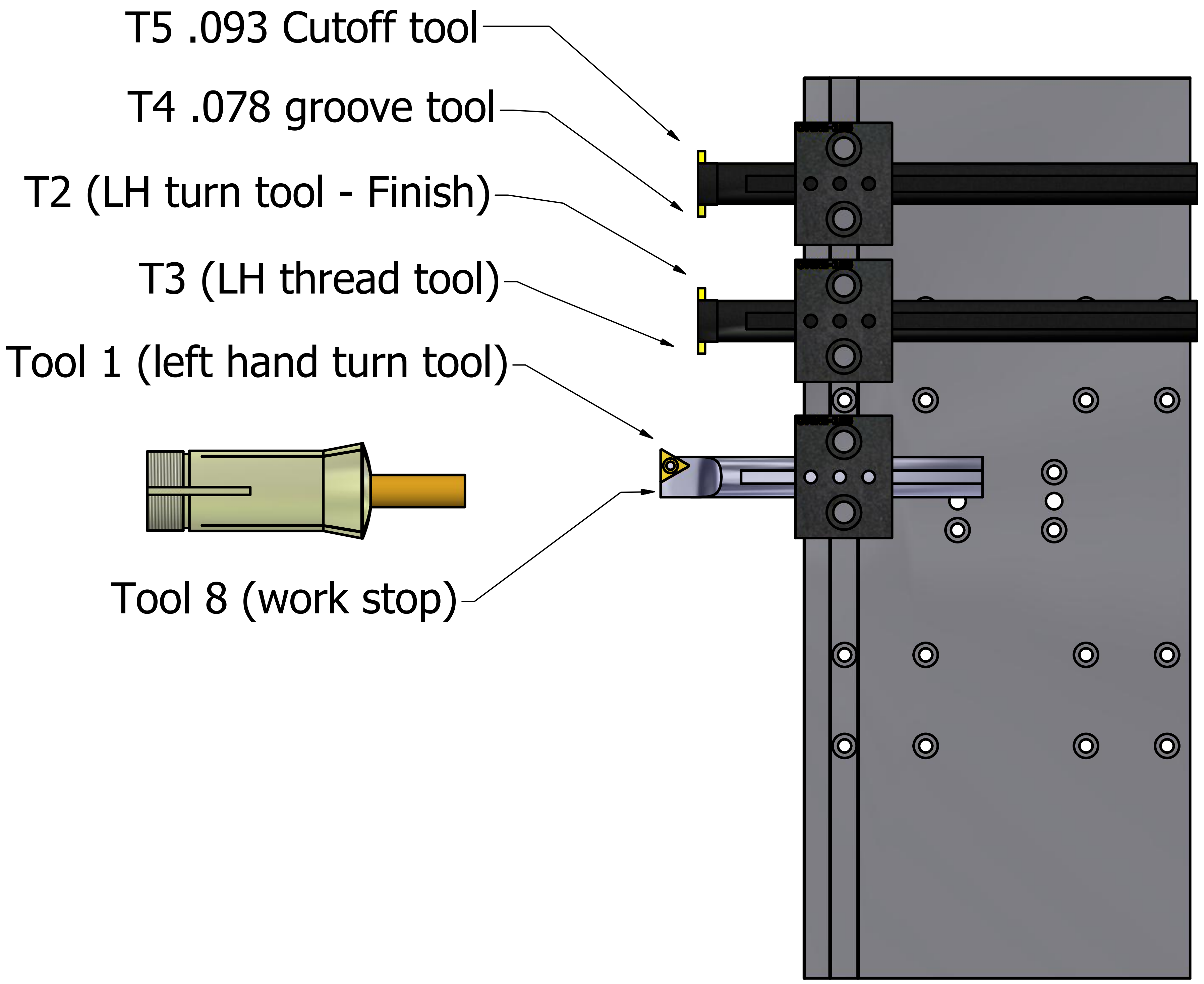
DETAIL A
SCALE 10 : 1







Sample 5



<u>Process:</u>
Use the front of the tool as a work stop
    T8, stop motion, pull part to stop, close collet
Use T1 (LH turn tool) to rough the part as shown
    Leave .01 to clean up on a finish pass
Use T2 (LH turn tool - finish)
    perform a cleanup pass with the chamfers
Use T4 (.078 wide groove tool)
    Undercut behind the thread.
    Use secondary offsets to adjust size.
Use T3 (LH thread tool) to thread OD
    Major diameter .190, minor diameter .151
Use T5 to cut off

Part: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.iam
Drawing: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.idw

DRAWN JRichlin	3/17/2010	Richlin Machinery Inc **OmniTurn Training 631 694-9400**	
		TITLE **Part #5**	
Material:		SIZE C	DWG NO **OmniTurn Training**	REV
			SHEET 4 OF 11

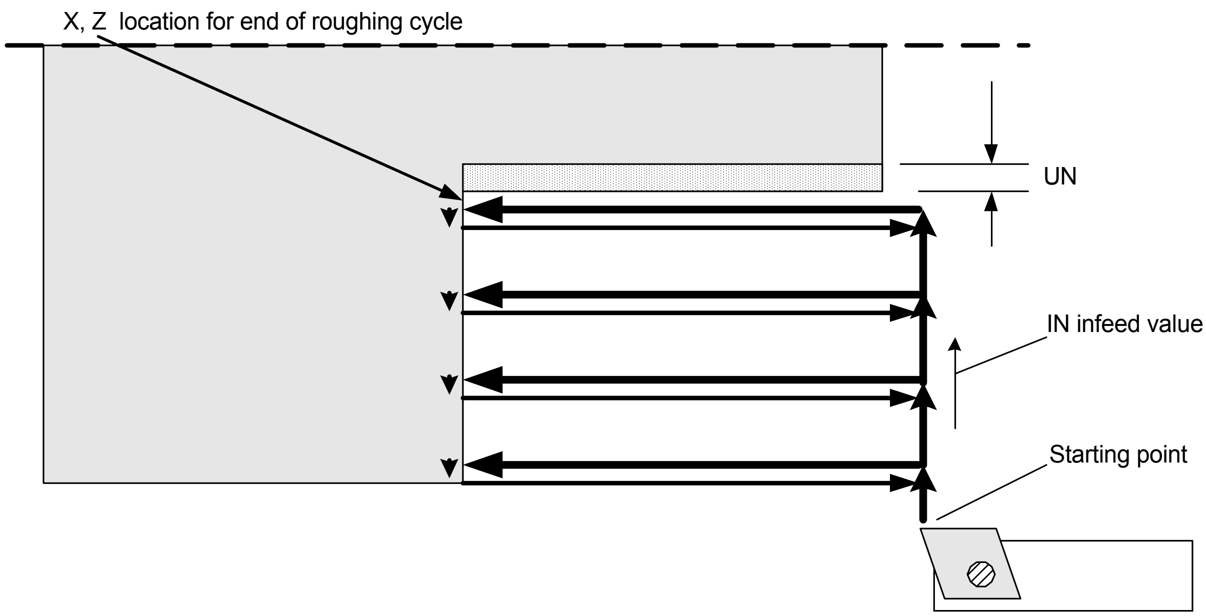
```
G90G72G94F300 (PART-5)
G10X0Z2
T8(WORK STOP)
X0Z2
Z.2
F50Z.01
M00 (PULL PART TO STOP)
M12
Z2F300
M03S1500
M08
T1(LH TURN TOOL - rough)
X.6Z2
Z0
G95f.003X-.015
X.3F.01
Z-.44F.003
X.6F94F300
Z.01
X.1
Z0G95F.003
X.21
Z-.44
X.395
Z-.49
X.52Z-.615
X.6
G94F300Z2
T2 (LH FINISH TOOL)
X.3Z2
Z0
G95F.003X.15
X.19C.025
Z-.45
X.375C.02
Z-.5
X.5Z-.625
X.6
Z2G94F200
T4 (078 GROOVE TOOL)
X-.5Z2
Z-.45
G95F.002X-.15
G04F.3
X-.25
Z-355 D1
X-.15
G04F.3
X-.25
G94F300Z2
T3 (THREADING TOOL)
X-.18Z2
Z.2
G95
G33X-.15Z-.38K.03125I.025C
G94F300Z2
T5 (CUT OFF TOOL)
X-.6Z2
Z-1.1
G95F.003X.1
G94F300X-.6
M05
M08
Z2
M30
```

G74 - Box Roughing Cycle

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

G74XnZnInUnFn

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.

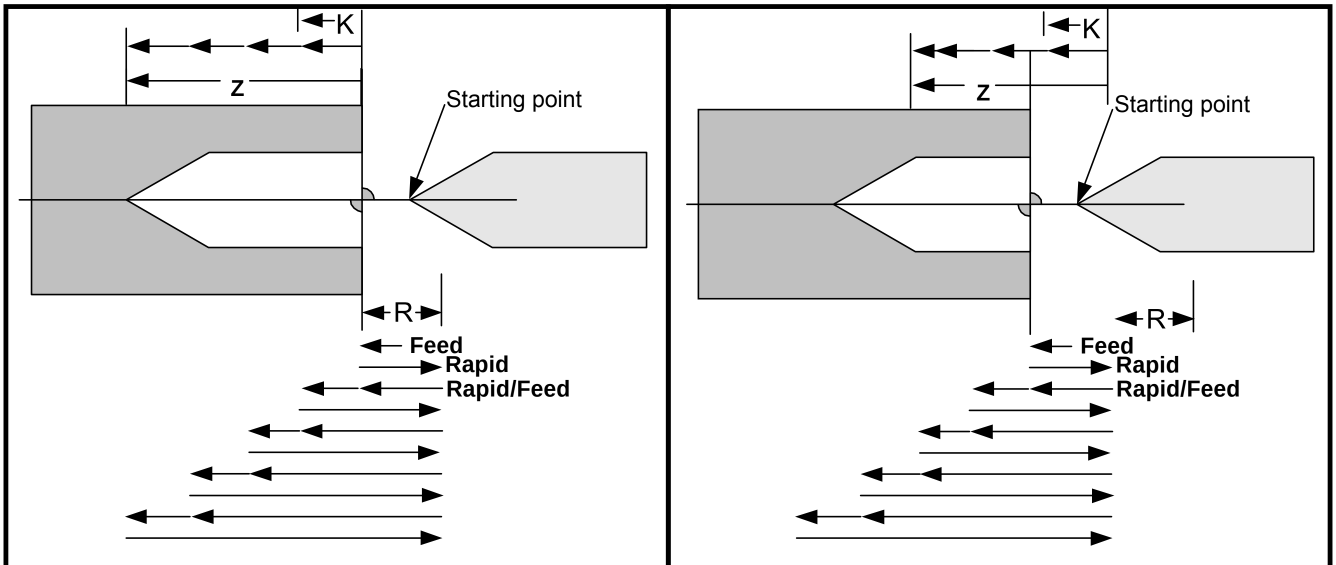
The X, Z coordinate may be absolute or incremental, based on the current mode of the control.

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

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



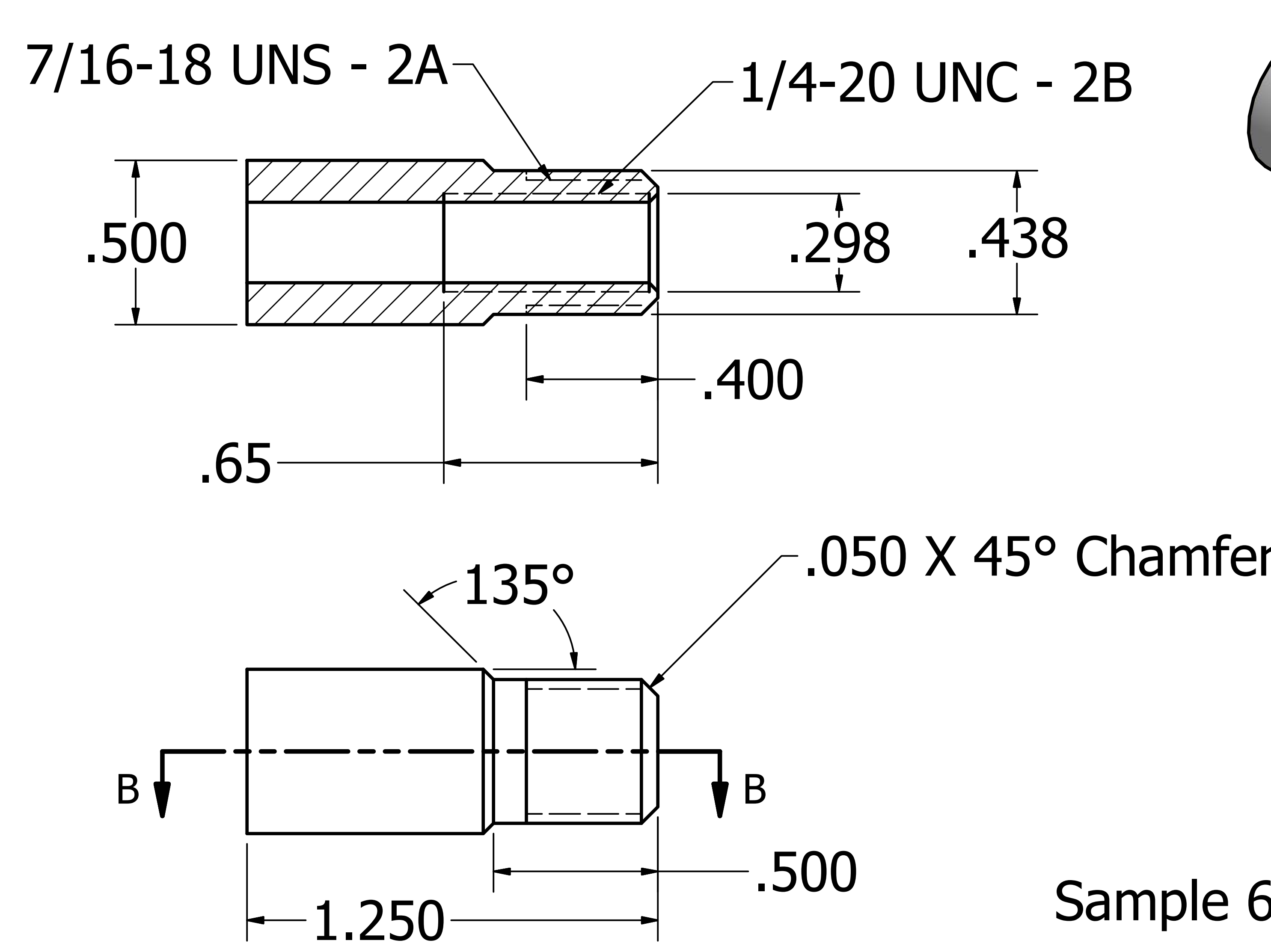
G83 used in absolute mode, G90	G83 used in incrimental mode, G91

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.

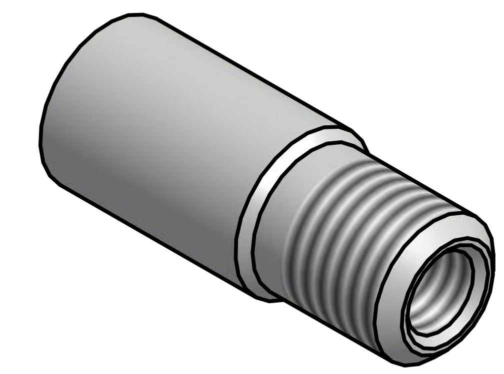
Start location: Position the drill where you want the first drill peck to start. After the first peck the drill will rapid out to the **R** location, and then back to where it started less the **C** value.

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"**

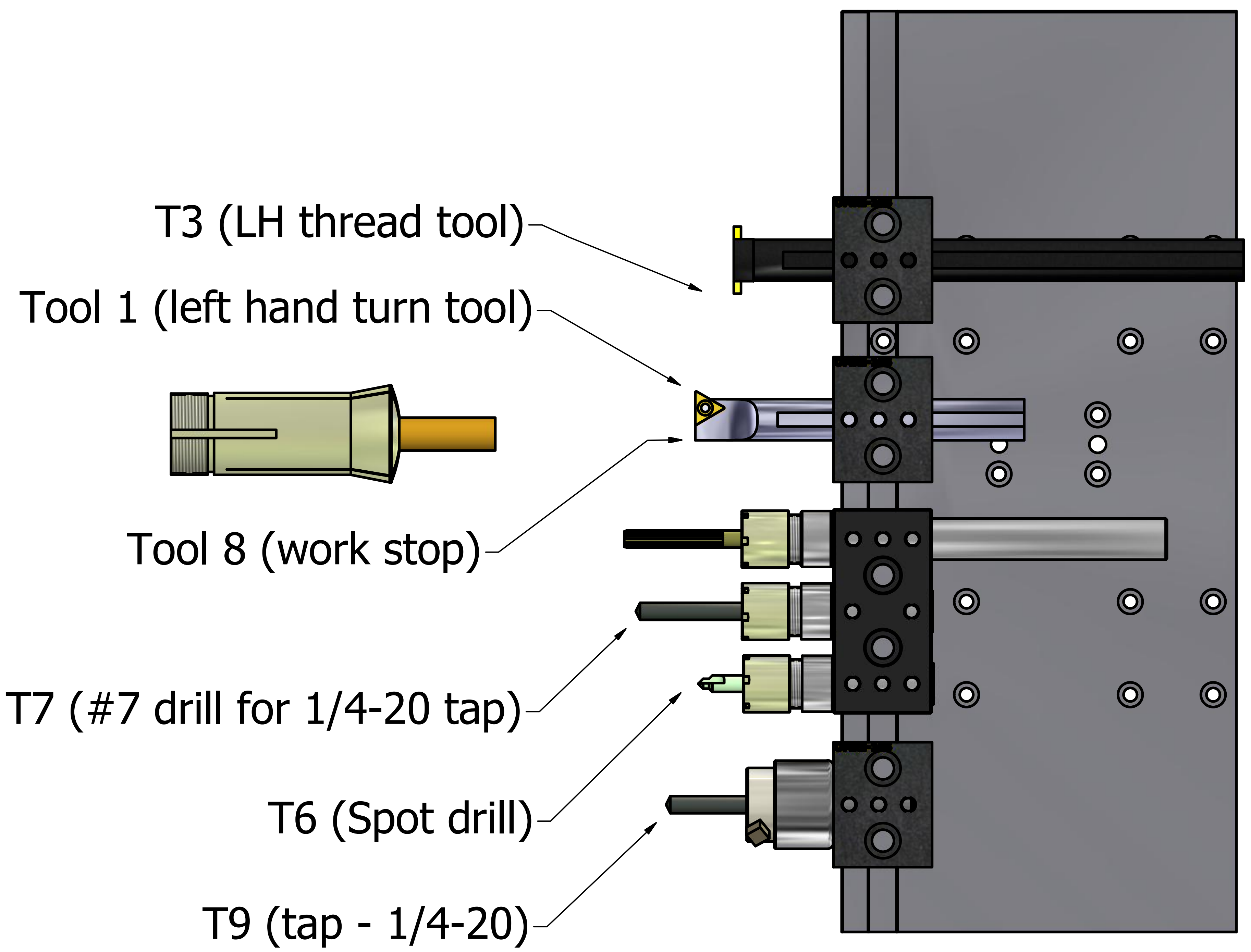
Topic: Drilling, peck drilling
Topic: Tapping
Topic: Multi-Start threading



SECTION B-B
SCALE 2 : 1



Sample 6



Process:

Use the front of the tool as a work stop
T8, stop motion, pull part to stop, close collet
Use T1 (LH turn tool) turn the part as shown
Finish to size with chamfers
Use T3 (LH thread tool) to thread OD
Major diameter .4375, minor diameter .348
Use T1 to clean tops of threads
Use T3 (LH thread tool) to thread OD, one pass to clean up.
Use T6 (spot drill) to spot and chamfer face
Use T7 (#7 drill) use G83 peck drill to drill through part
Use T9 (1/4-20 tap) to tap at least 5/8" deep
Use T5 to cut off

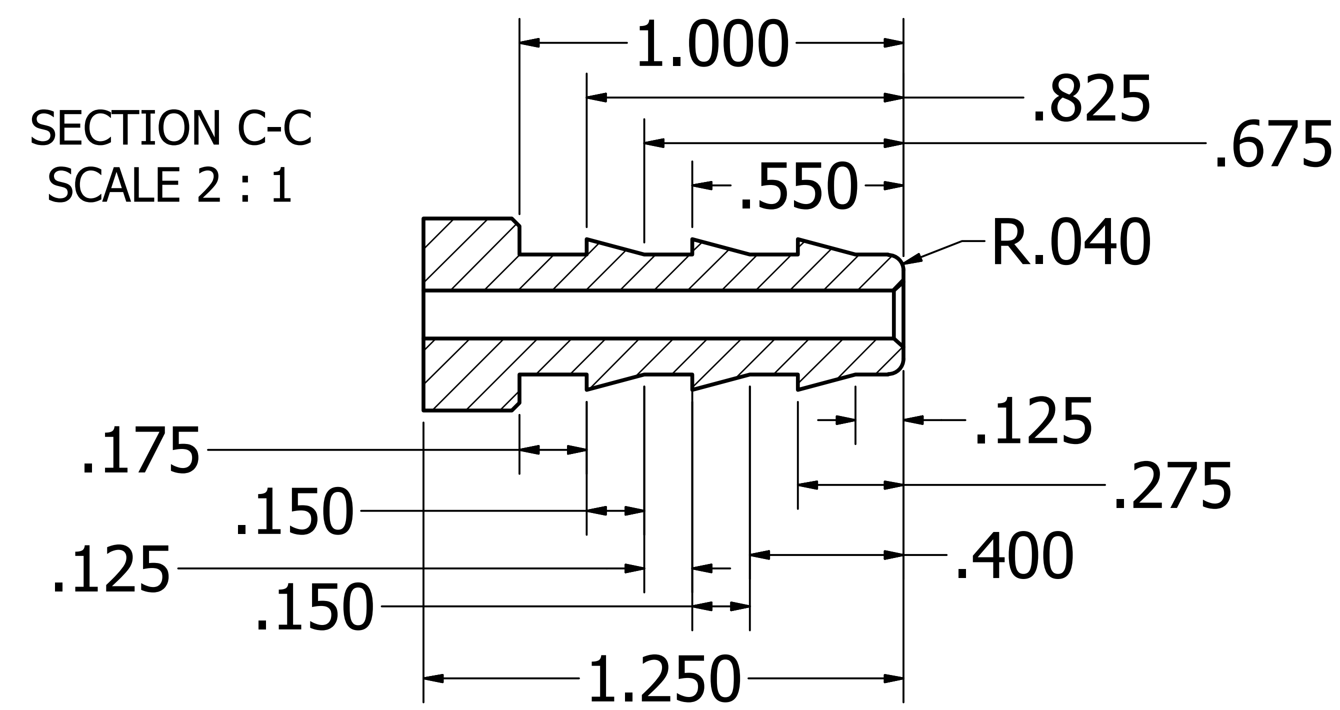
Variation
Cut the 7/16 - 20 thread with three starts instead of 1 start

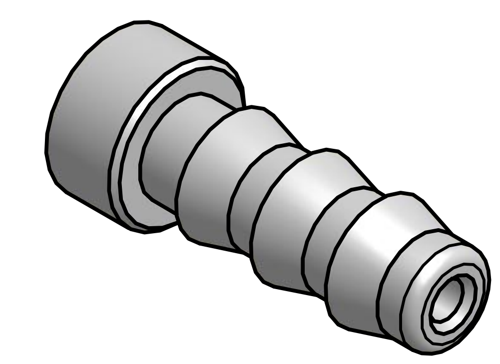
DRAWN JRichlin	3/17/2010	Richlin Machinery Inc **OmniTurn Training 631 694-9400**
		TITLE **Part #6**
Material:		SIZE C · DWG NO **OmniTurn Training** · REV
		SHEET 5 OF 11

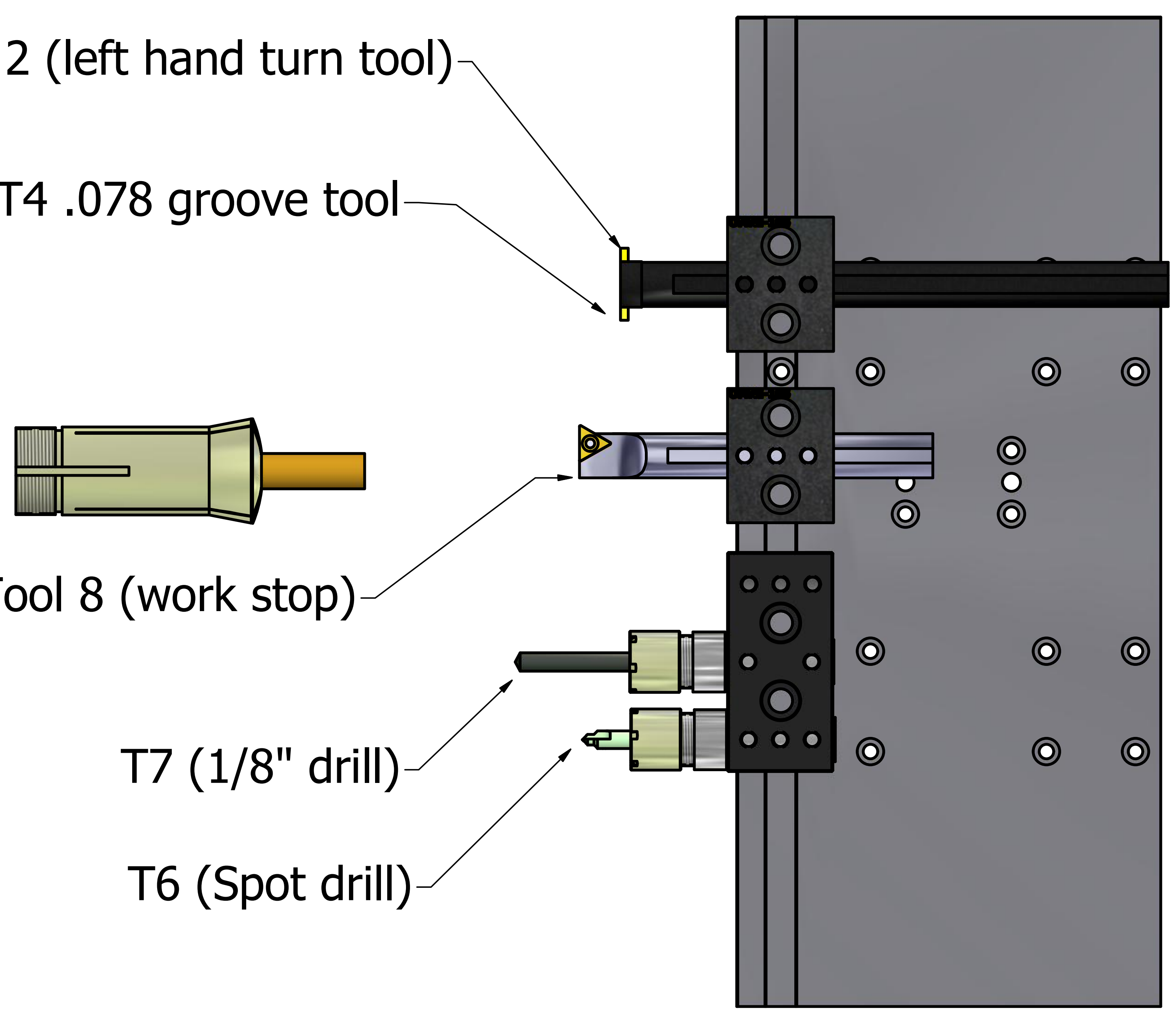
Part: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.iam
Drawing: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.idw

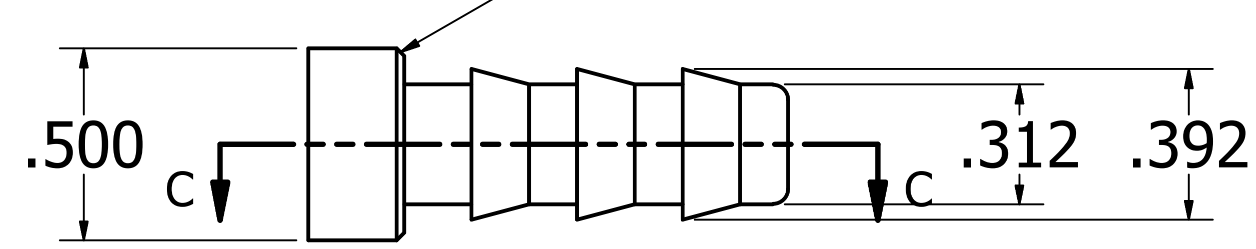
```
G90G72G94F300 (PART-6)
G10X0Z2
T8(WORK STOP)
X0Z2
Z.2
F50Z.01
M00 (PULL PART TO STOP)
M12
Z2F300
M03S1500
M08
T1(LH TURN TOOL - rough)
X.6Z2
Z0
G95f.003X-.015
X.4375C.05
Z-.500
X.6Z-.5825
G94F300Z2
T3 (THREAD TOOL)
X.427Z2
Z.2
G33X.336 Z-.4I.03K.05C
G94F300Z2
T1
X.4375Z2
Z.2
G95F.004Z-.5
X.6
G94F300Z2
T3 (THREAD TOOL)
X.427Z2
Z.2
G33X.336 Z-.4I.03K.05CO
G94F300Z2
T6 (SPOT DRILL)
X0Z2
Z.2
G95F.005Z-.2
G94F300Z2
T7 (NUMBER 7 DRILL)
X0Z2
Z.1
G95
G83Z-1.35K.3R.5L300
G94F300Z2S500
T9 (1-4 20 TAP)
X0Z2
Z.2
G95F.049Z-.7
M04
Z.2
M03S1500
G94F300Z2
T5 (CUT OFF TOOL)
X-.6Z2
Z-1.25
G95F.003X.1
G94F300X-.6
M05
M08
Z2
M30
```

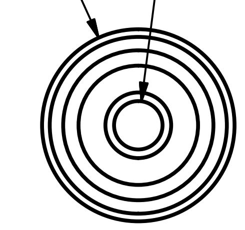
Topic: Repeat peck drilling
Topic: Repeat grooving with secondary offsets











Sample 7

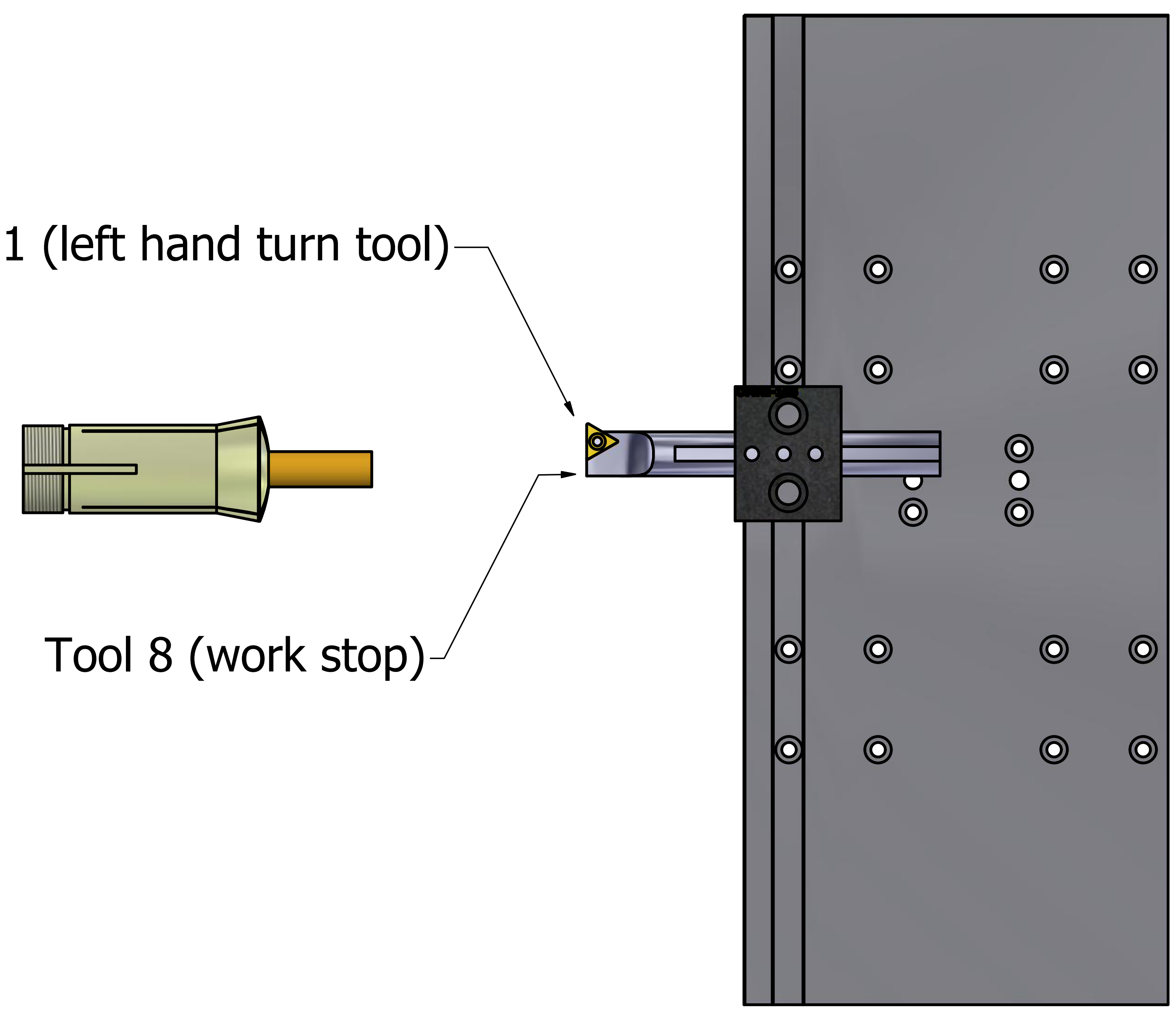
Process:
Use the front of the tool as a work stop
T8, stop motion, pull part to stop, close collet
Use T2 (LH turn tool) to turn the part as shown
Finish to size all that can be reached
Use T6 (spot drill) to spot and chamfer face
Use T7 (1/8" drill) use G83 peck drill to drill through part
Use T4 to finish grooves behind barb (use secondary offsets)
and infront of end wall.
Use T5 to cut off

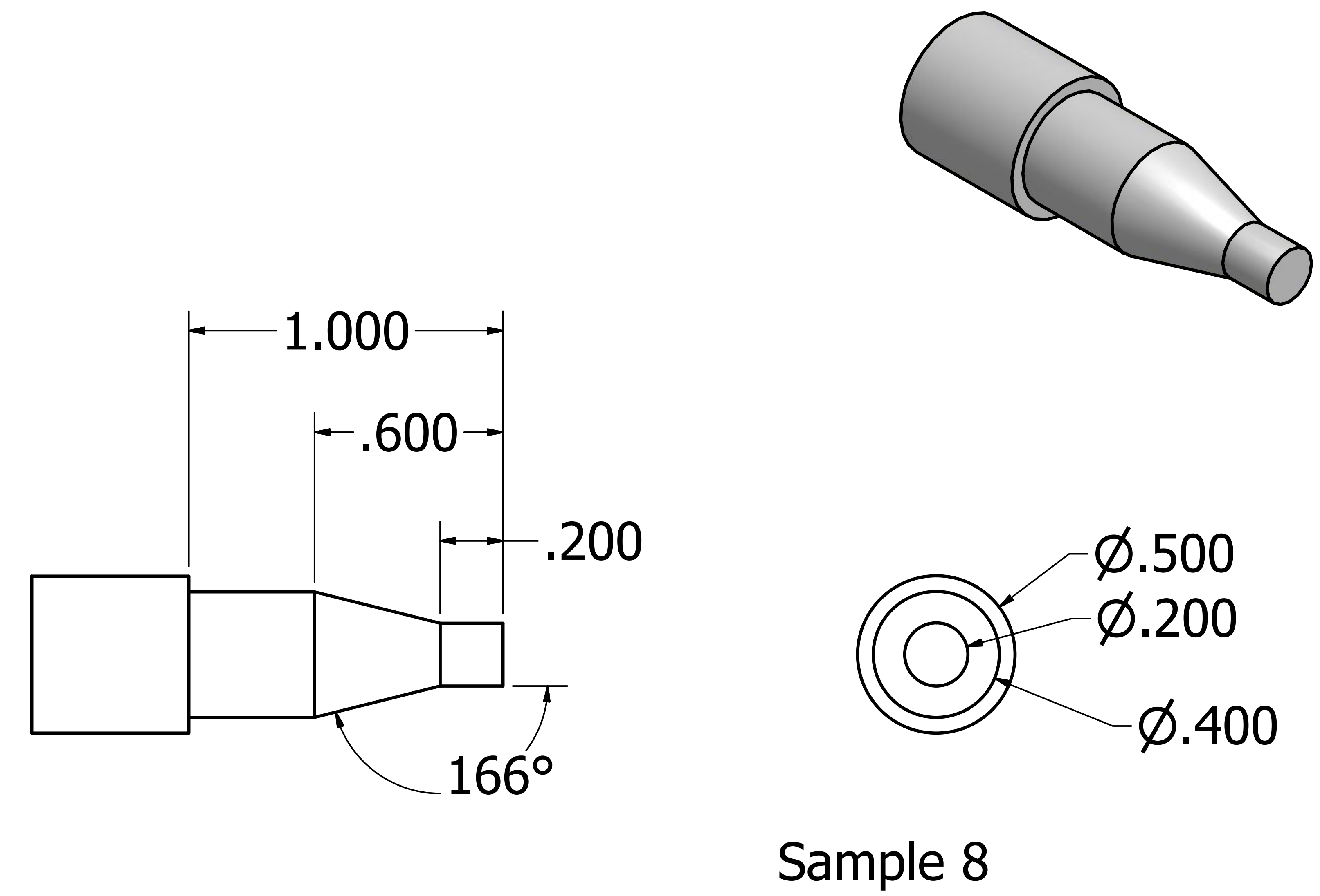
Part: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\part 7.ipt
Drawing: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.idw

DRAWN JRichlin	3/17/2010	Richlin Machinery Inc OmniTurn Training 631 694-9400
		TITLE **Part #7**
Material:		SIZE C / DWG NO **OmniTurn Training** / REV
		SHEET 6 OF 11

```
G90G72G94F300 (PART-7)
G10X0Z2
T8(WORK STOP)
X0Z2
Z.2
F50Z.01
M00 (PULL PART TO STOP)
M12
Z2F300
M03S1500
M08
T2(LH TURN TOOL )
X.6Z2
Z0
G95f.003X-.015
X.312R.04
Z-.125
X.392Z-.275
Z-.4
X.312
G04F.3
X.392Z-.55
Z-.675
X.312
G04F.3
X.392Z-.825
Z-1
X.312
G04F.3
X.51C.025
Z-1.1
G94F300Z2
T6 (SPOT DRILL)
X0Z2
Z.2
G95F.005Z-.2
G94F300Z2
T7 (125 DRILL)
X0Z2
Z.1
G95
G83Z-1.35K.2R.5L300
G94F300Z2
T4 (078 WIDE GROOVE TOOL)
X.4Z2
Z-.275 D1 (SET TO -078)
G95F.003X.312
G04F.3
G94F300X.4
Z-.55
G95F.003X.312
G04F.3
G94F300X.4
Z-.825
G95F.003X.312
G04F.3
G94F300X.4
Z2
T5 (CUT OFF TOOL)
X-.6Z2
Z-1.25
G95F.003X.1
G94F300X-.6
M05
M08
Z2
M30
```

Topic: Secondary offsets for correcting diameters and tapers





Process:

Use the front of the tool as a work stop
T8, stop motion, pull part to stop, close collet

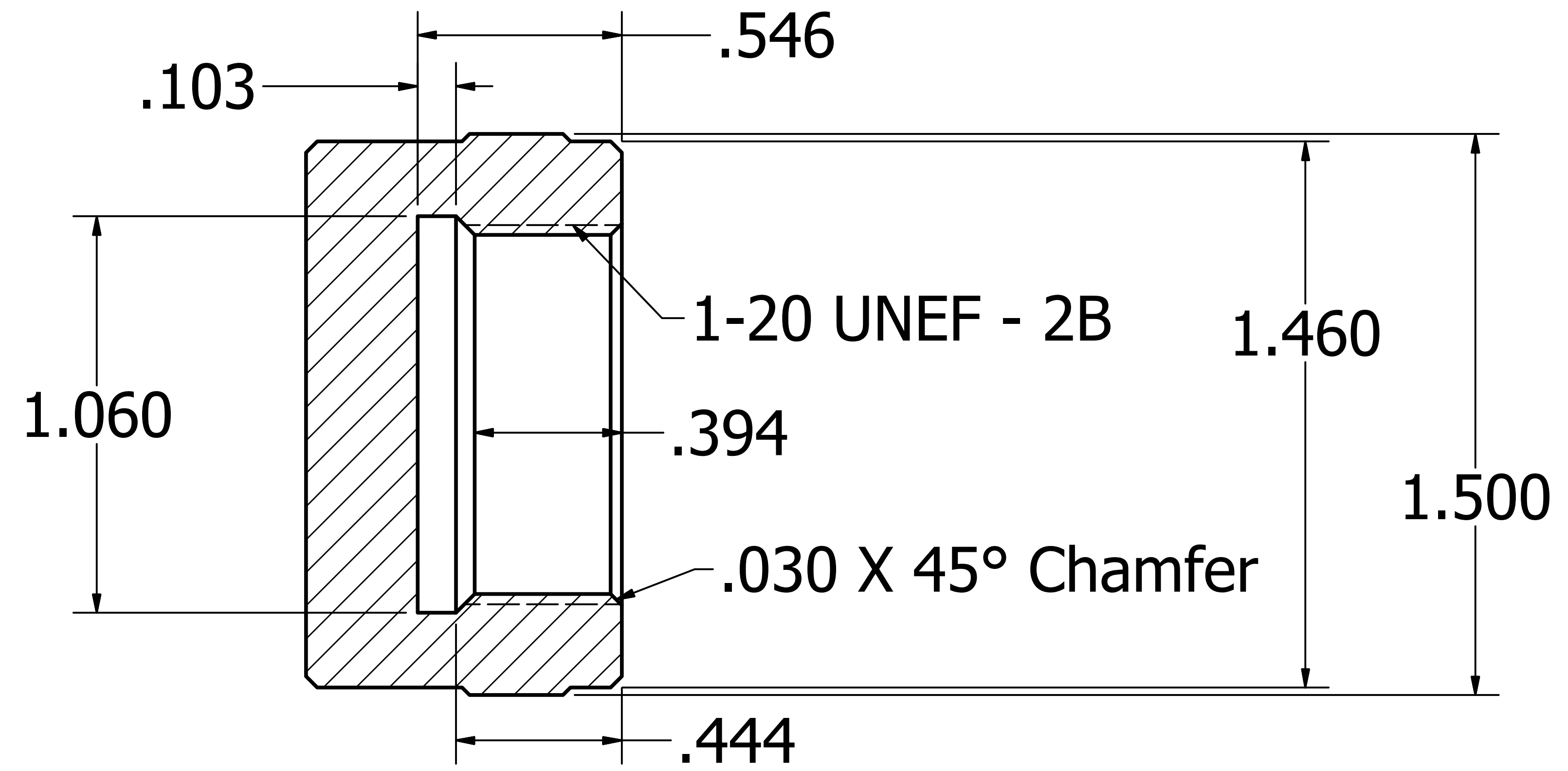
Use T2 (LH turn tool) to turn the part as shown
Use secondary offsets to adjust for exact sizes and taper.

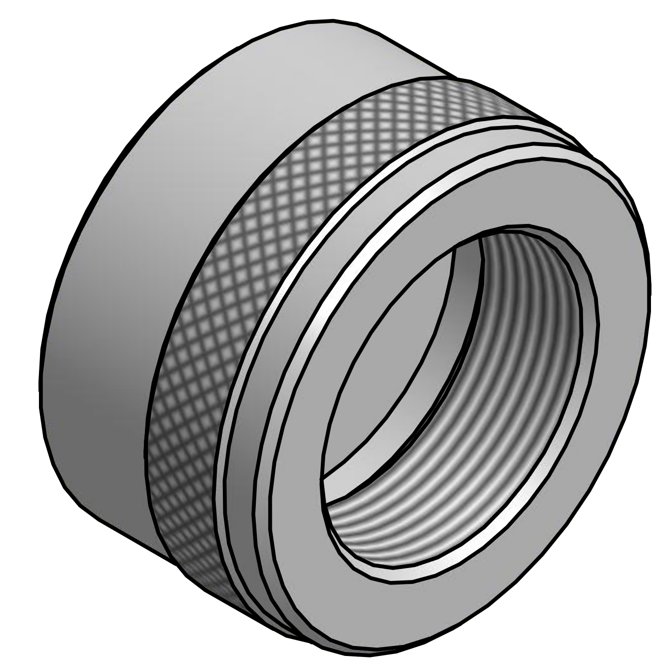
Part: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.iam
Drawing: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.idw

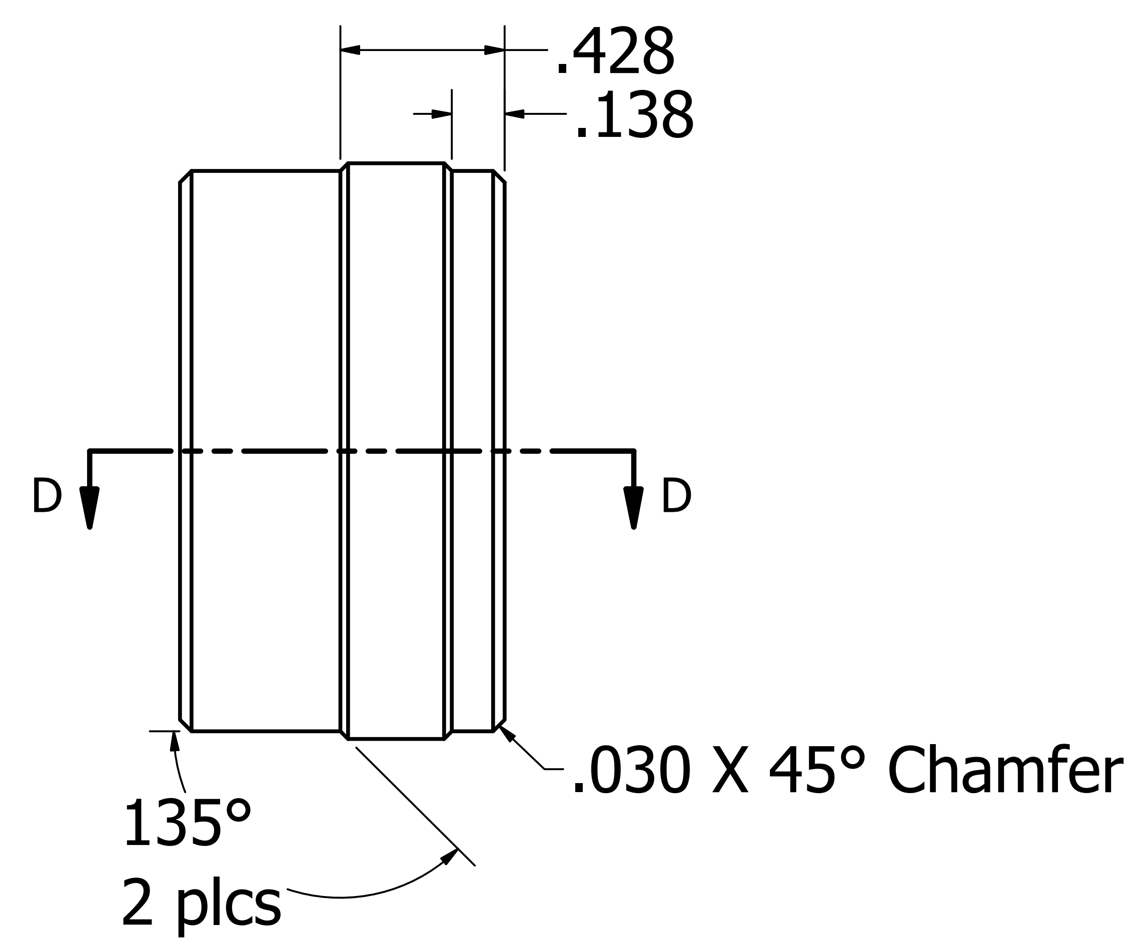
DRAWN JRichlin	3/17/2010	Richlin Machinery Inc **OmniTurn Training 631 694-9400**
		TITLE **Part #8**
Material:		SIZE C	DWG NO **OmniTurn Training**	REV
		SHEET 7 OF 11

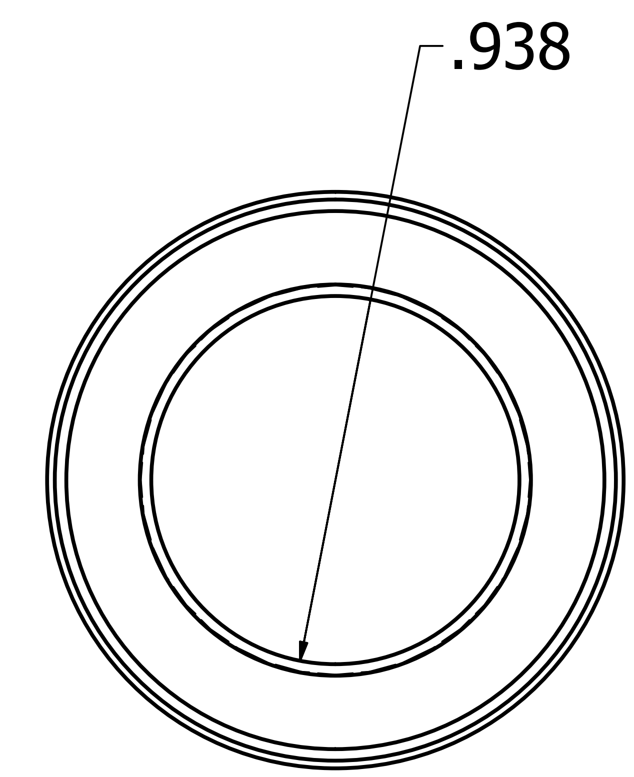
```
G90G72G94F300 (PART-8)
G10X0Z2
T8(WORK STOP)
X0Z2
Z.2
F50Z.01
M00 (PULL PART TO STOP)
M12
Z2G300
M03S1500
M08
T1(LH TURN TOOL )
X.6Z2
Z0
G95f.003X-.015
X.2
Z-.2D1
X.4Z-.6D2
Z-1
X.6
G94F300Z2
M30
```

Topic: Boring
Topic: Thread ID
Topic: Knurl

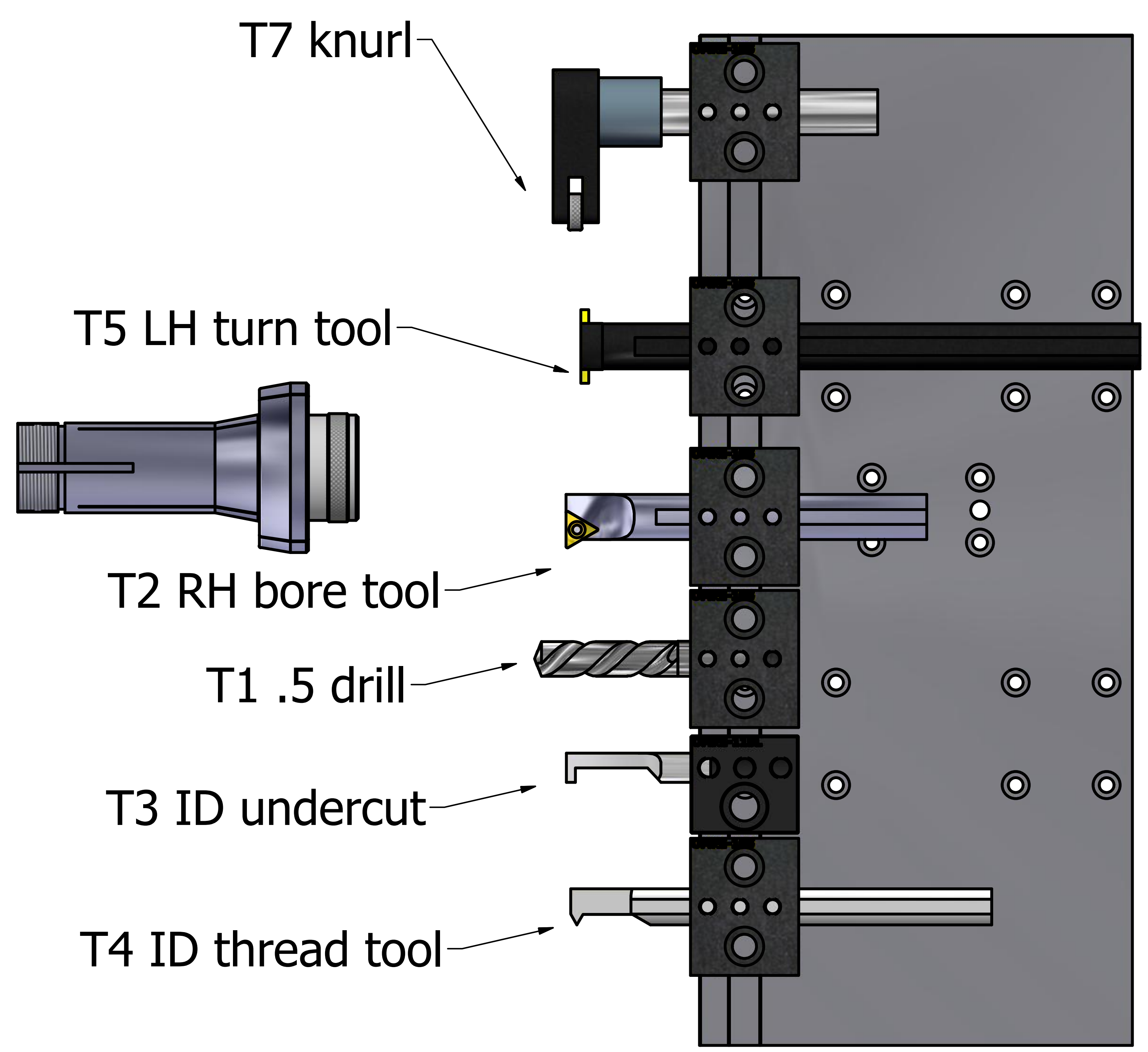








Sample 9



Process:
Load blank into collet. The left side of the part is done.
The 1.46 diameter is turned to the start of the knurl.
T1 (.5 DRILL) Drill to .54 deep
T2 (RH bore tool) Rough and finish bore
T3 (ID under cut) Clear out under cut
T4 (ID thread) Thread minor diameter .938", Major 1.000"
T5 (LH turn tool) Face and turn the OD
T7 (knurl) knurl OD

DRAWN JRichlin	3/17/2010	Richlin Machinery Inc **OmniTurn Training 631 694-9400**
		TITLE **Part #9**
Material:		SIZE C · DWG NO **OmniTurn Training** · REV
		SHEET 8 OF 11

Part: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\Part 9.ipt
Drawing: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.idw

```
G90G72G94F300 (PART-9)
G10X0Z2
M08
M03S1200
T1 (500 DRILL)
X0Z2
Z.2
G95F.002Z-.54
G94F300Z2
T2 (BORE TOOL)
X.6Z2
Z.1
G95F.003Z-.54
X.5
G94F300Z.1
X.8
G95F.003Z-.54
X.6
G94F300Z.1
X1.19
G95F.003X.96Z-.03
Z-.546
X-.01
G94F300Z2
T3 (GROOVE TOOL)
X0.9 Z2
Z-.3
G95F.003Z.546
X1.060F.002
G04F.5
X.9
Z-.444F.01D1
X1.060F.002
G04F.5
X.9
Z-.394
X1.060Z-.444
G94F300X.8
Z2
T4 (ID THREAD TOOL)
X.97Z2
Z.2
G95
G33X1Z-.444I.025K.05C
G94F300Z2
T5 (LH TURN TOOL)
X1.7Z2
Z0
G95F.003X.95
X1.46C.03
Z-.138
X1.5Z-.158
Z-.5
G94F300X1.7
Z2S500
T7 (KNURL TOOL)
X0Z2
Z0
G95F.04Z-.5
G94F300X.2
Z2
M30
```

Spindle Positioning - G3

Spindle positioning system specifications

Spindle power:	5HP
Voltage:	200 -230V 3 phase or single phase (contact the factory for wiring)
Resolution:	.02 °
Max Speed:	4000 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.

2.74

Spindle Positioning - G4 control

Spindle positioning system specifications

Spindle power:	5HP
Voltage:	200 -230V 3 phase
Resolution:	.02 °
Max Speed:	4000 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. M88 is for high precision orientation.
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 SFP at 1" diameter. With no sign the spindle will rotate in the M03 direction. The ”-” sign will cause the spindle to rotate in the M04 direction.

As an example, if you want 5" per minute feed rate for a milling cut at 1" diameter

S = 1.57 x feedrate / diameter of cut   S = 1.57 x 5 / 1   S = 7.9

As an example, if you want 5" per minute feed rate for a milling cut at .5" diameter

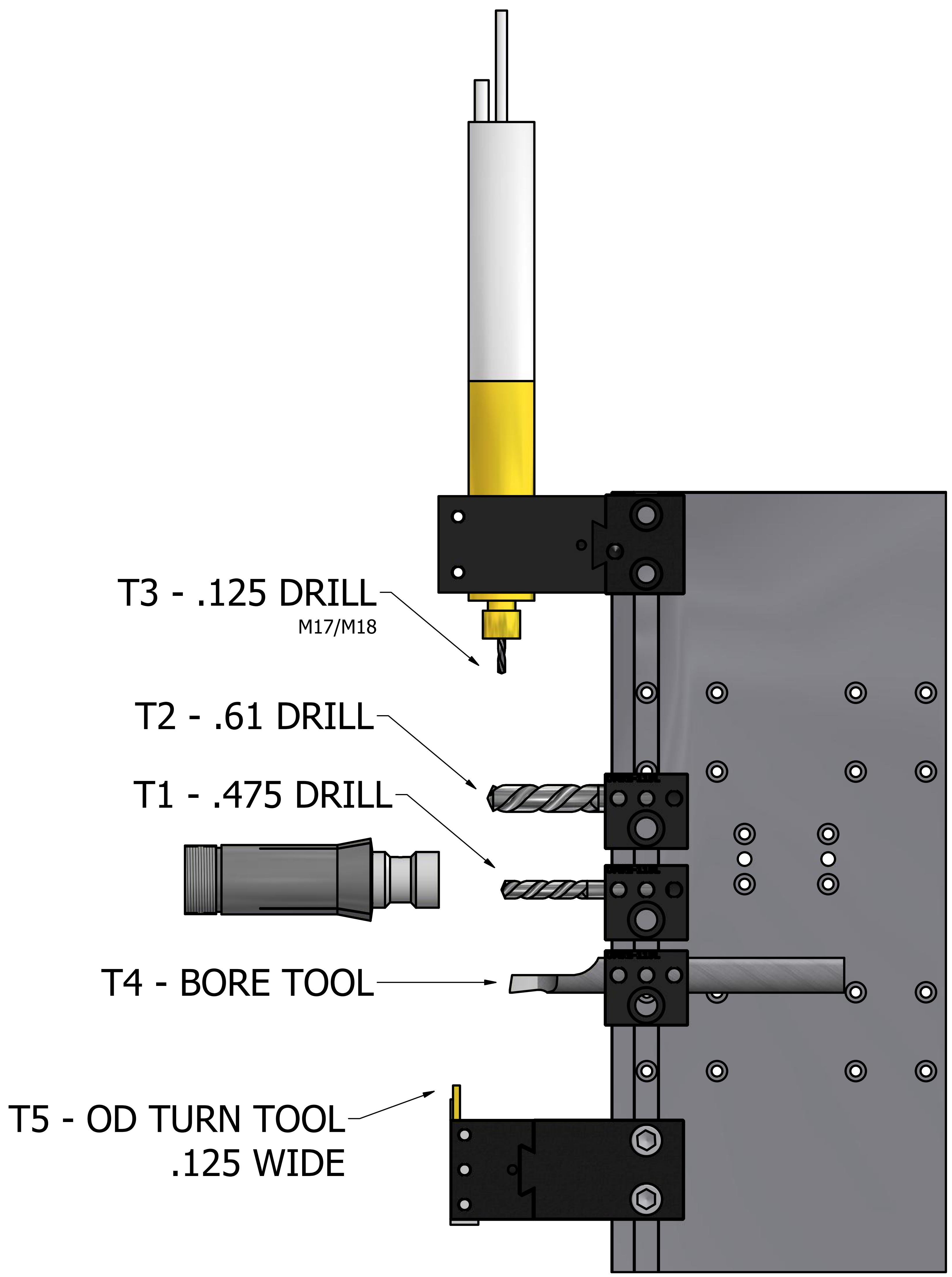
S = 1.57 x feedrate / diameter of cut   S = 1.57 x 5 / .5   S = 15.8

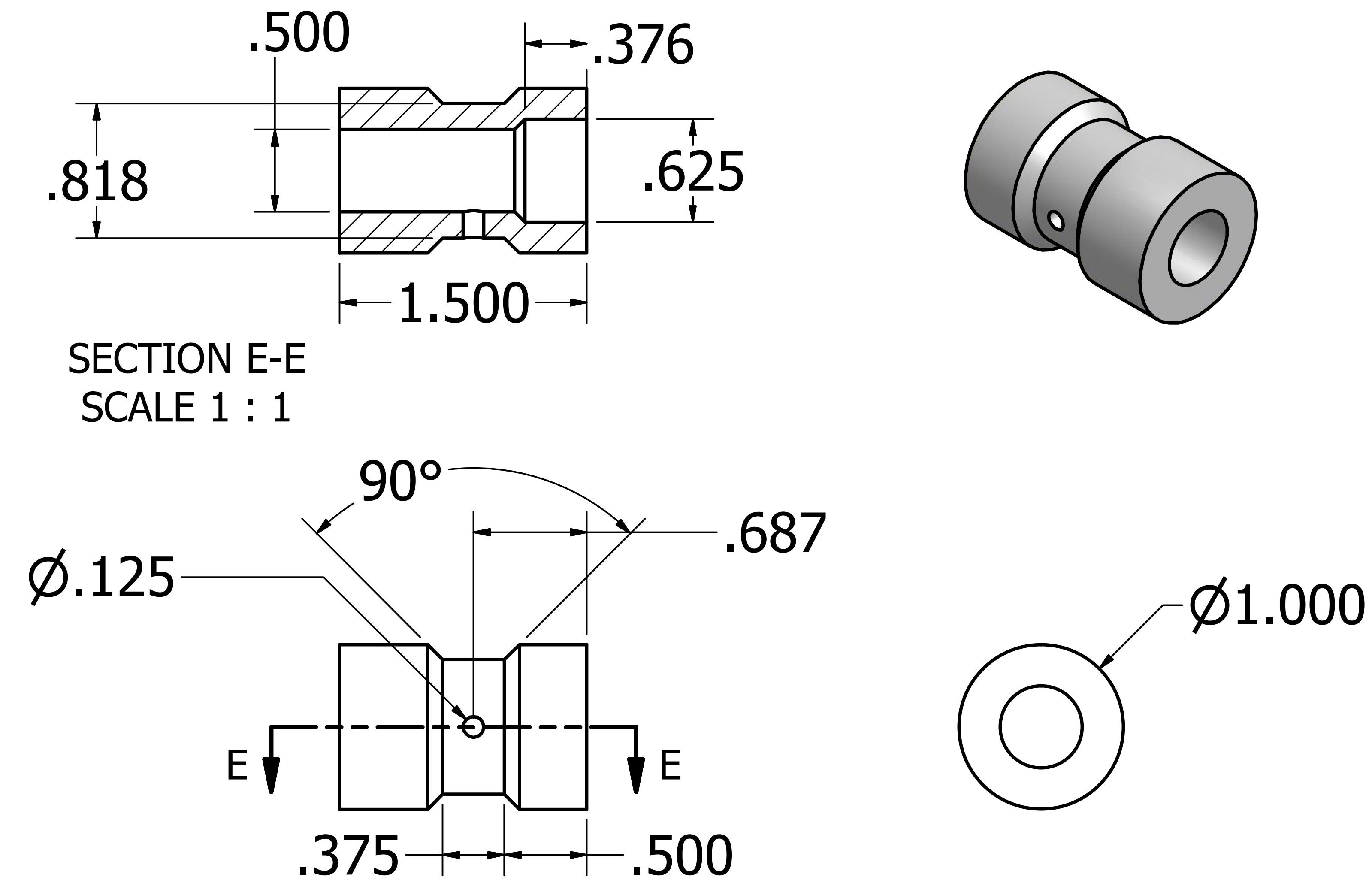
Notes on use:

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

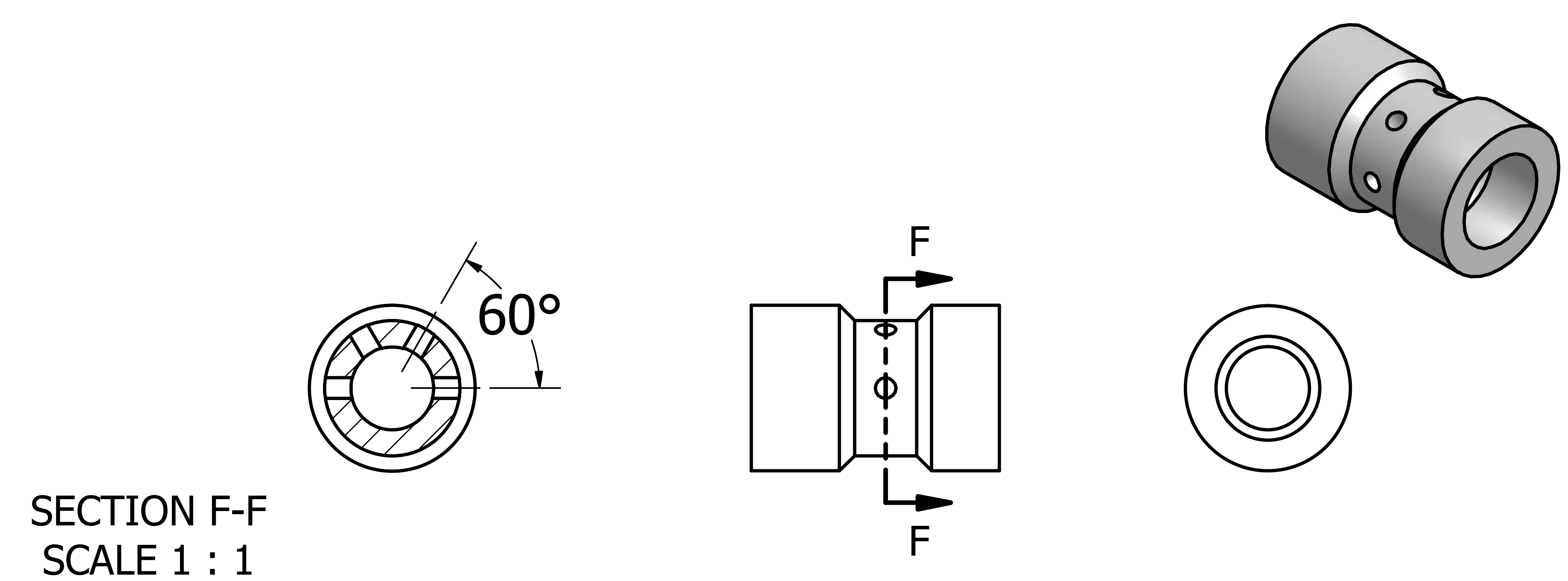
÷

Topic: Live tool cross drilling
Topic: C axis, M19, CA, CI
Topic: Looping





Process:
Load blank into collet.
T1 .475 DRILL
T2 .61 DRILL
T5 TURN OD
T3 .125 LIVE TOOL
T4 BORE TOOL



DRAWN JRichlin 3/17/2010
Material:

Richlin Machinery Inc
OmniTurn Training 631 694-9400

TITLE
Part #10

SIZE C
DWG NO **OmniTurn Training**
REV
SHEET 9 OF 11

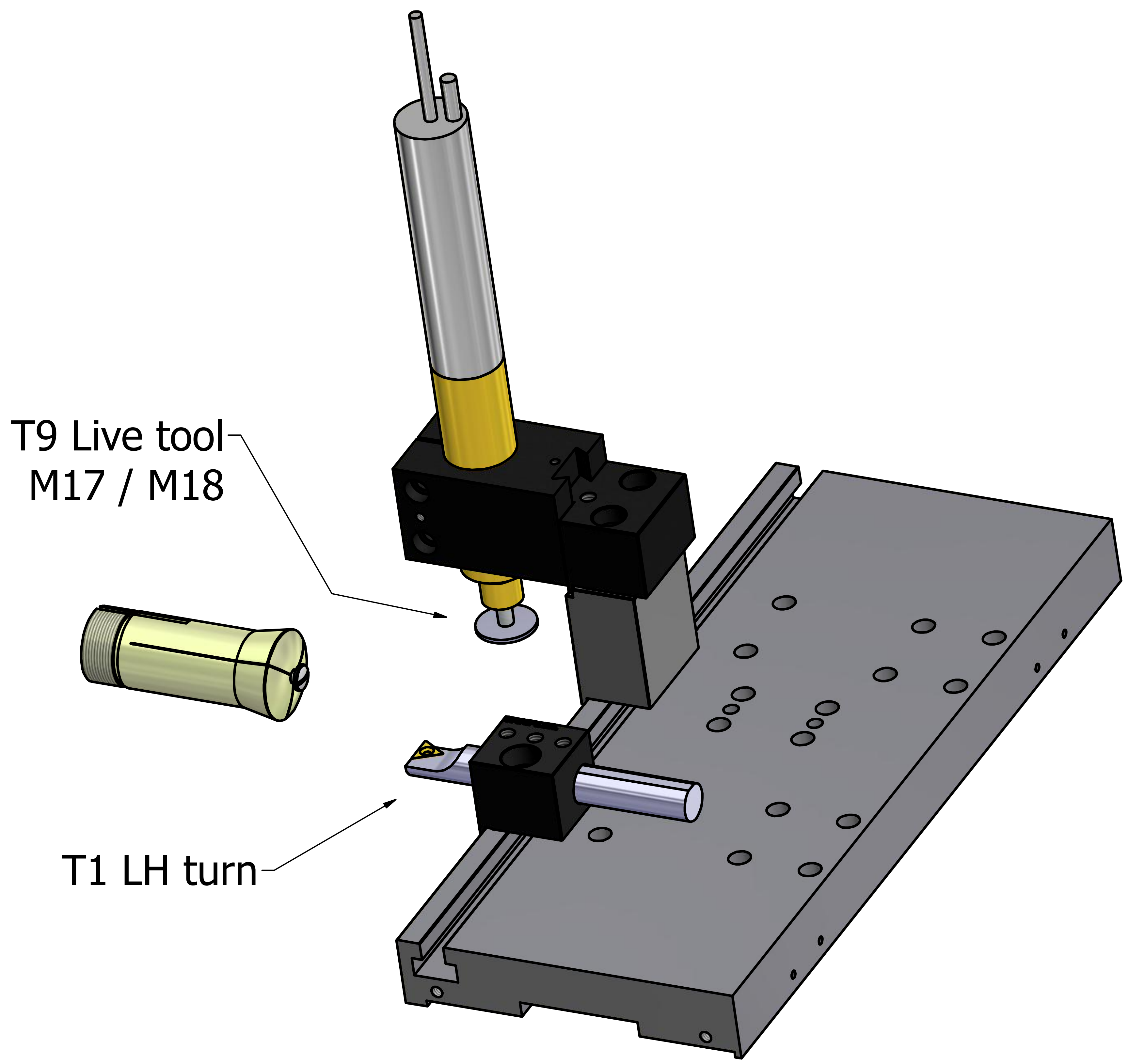
Part: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\part 10.ipt
Drawing: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.idw

```
G90G72G94300 (PART-10)
G10X0Z2
M03S1500
T1 (DRILL 475)
X0Z2
Z.2
G95F.003
G83Z-1.65K.5R.5L300
G94F300Z2
T2 (DRILL 61)
X0Z2
Z.1
G95F.003Z-.376
G94F300Z2
T5 (GROOVE TOOL)
X1.1Z2
Z-.875
G95F.003X.818
X1.1F.01
Z-1.016
X.818Z-.875F.003
X1.1F.01
Z-.775
X.818F.003
X1.1F.01
Z-.684D1
X.818Z-.5
G04F.3
G94F300X1.2
Z2
T4 (BORE TOOL)
X.625Z2
Z.1
G95F.003Z-.376
X.5Z-.438
Z-1.6
X.48
G94F300Z2
T3 (125 LIVE TOOL)
X1.1Z2
Z-.687
M19
M17
X.3F3
X1.1F300
M18
Z2
M30
```

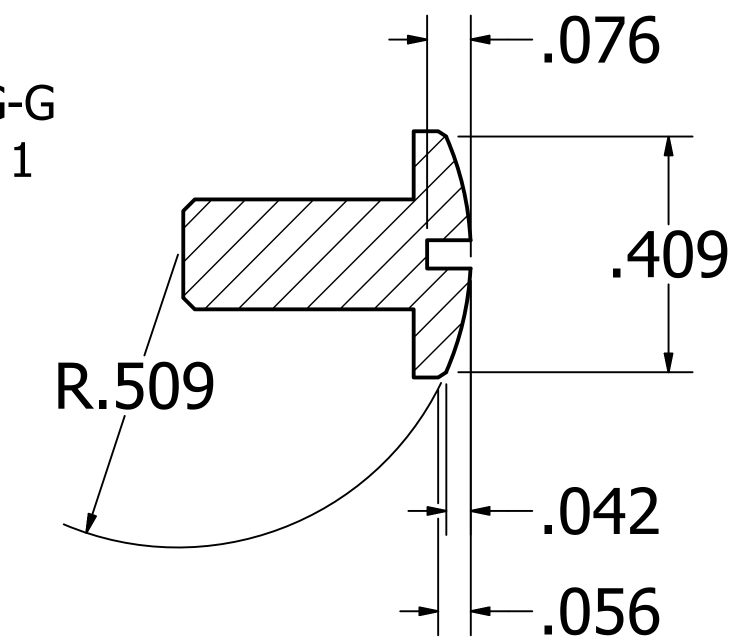
Alternative

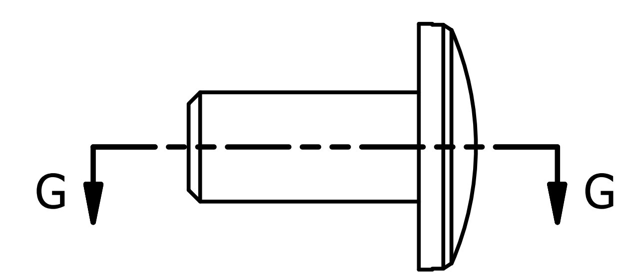
```
T3 (125 LIVE TOOL)
X1.1Z2
Z-.687
M19
M17
LS4
X.3F3
X1.1F300
CI60
LF
M18
Z2
M30
```

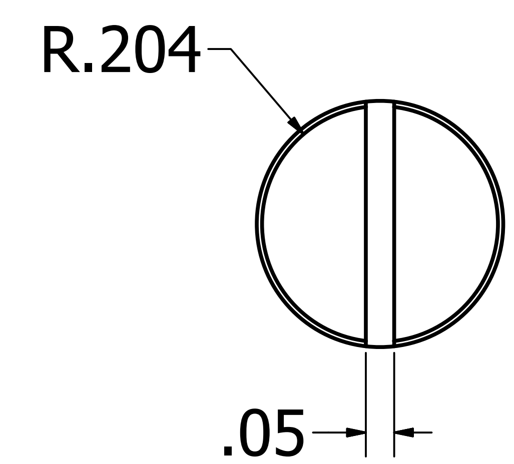
Topic: Arc Statement
Topic: Live tool slotting

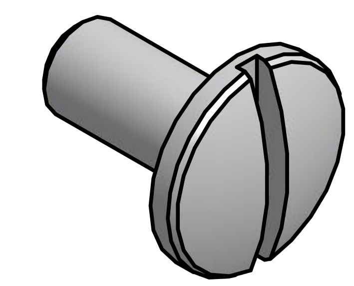


SECTION G-G
SCALE 3 : 1





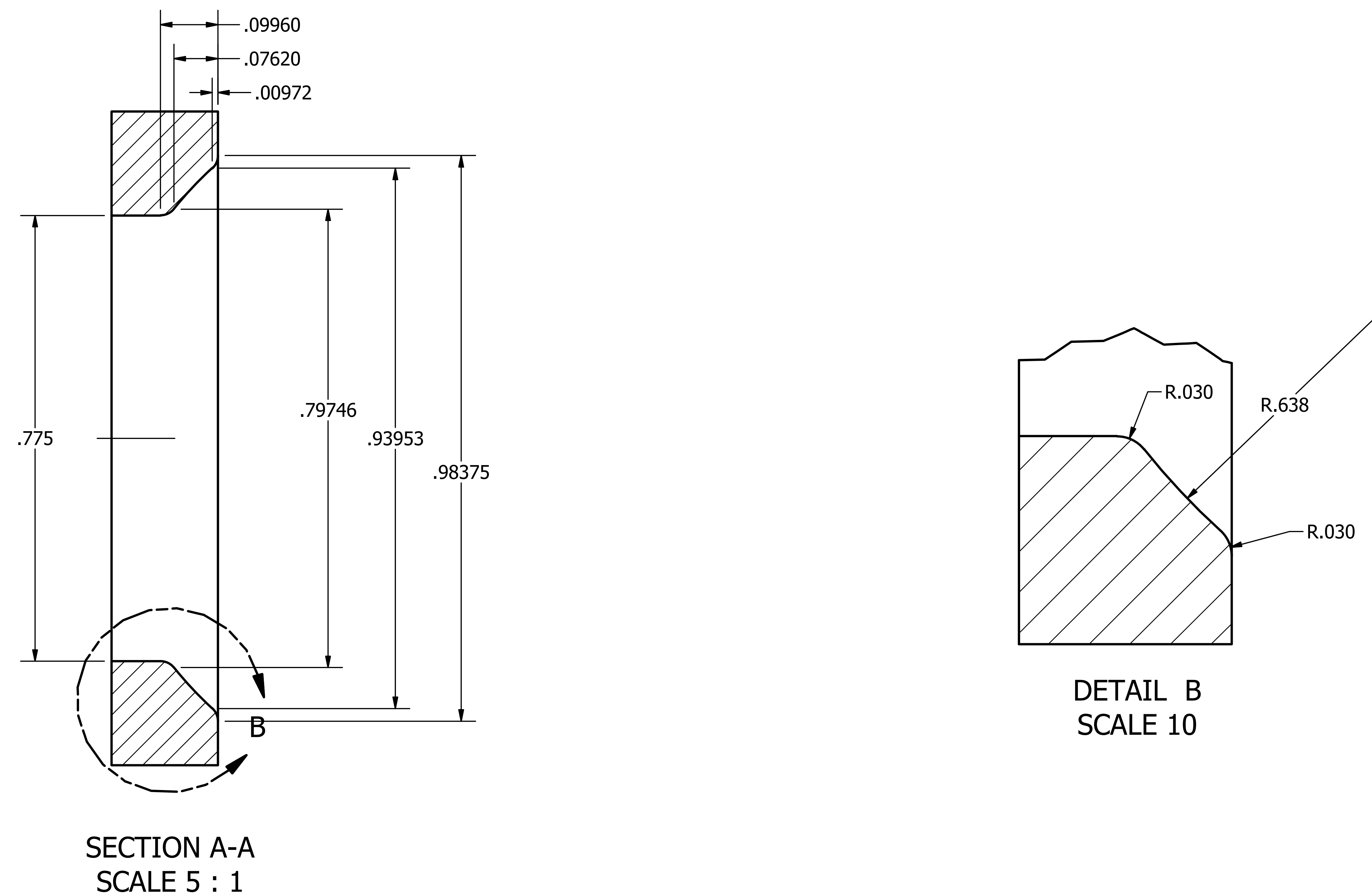


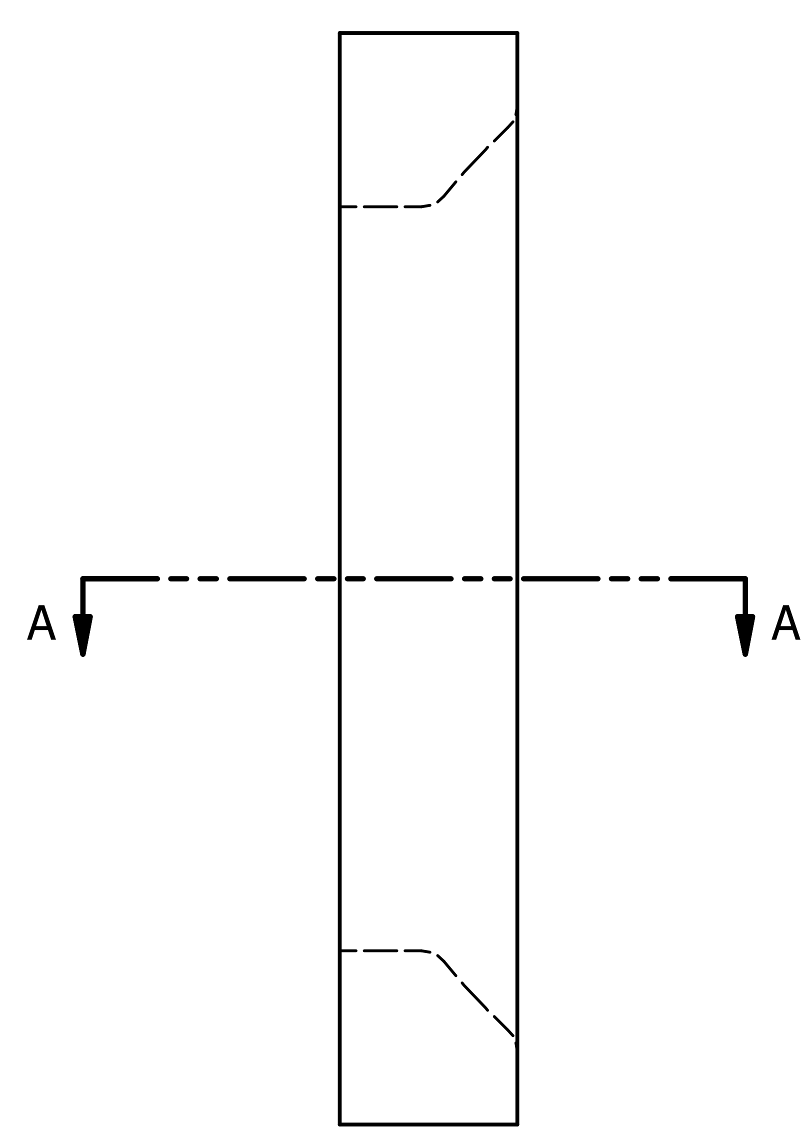


Process:
Load blank into collet.
T1 Turn face radius, chamfer and major diameter
T9 Live tool - create slot

Part: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\sample 11.ipt
Drawing: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.idw

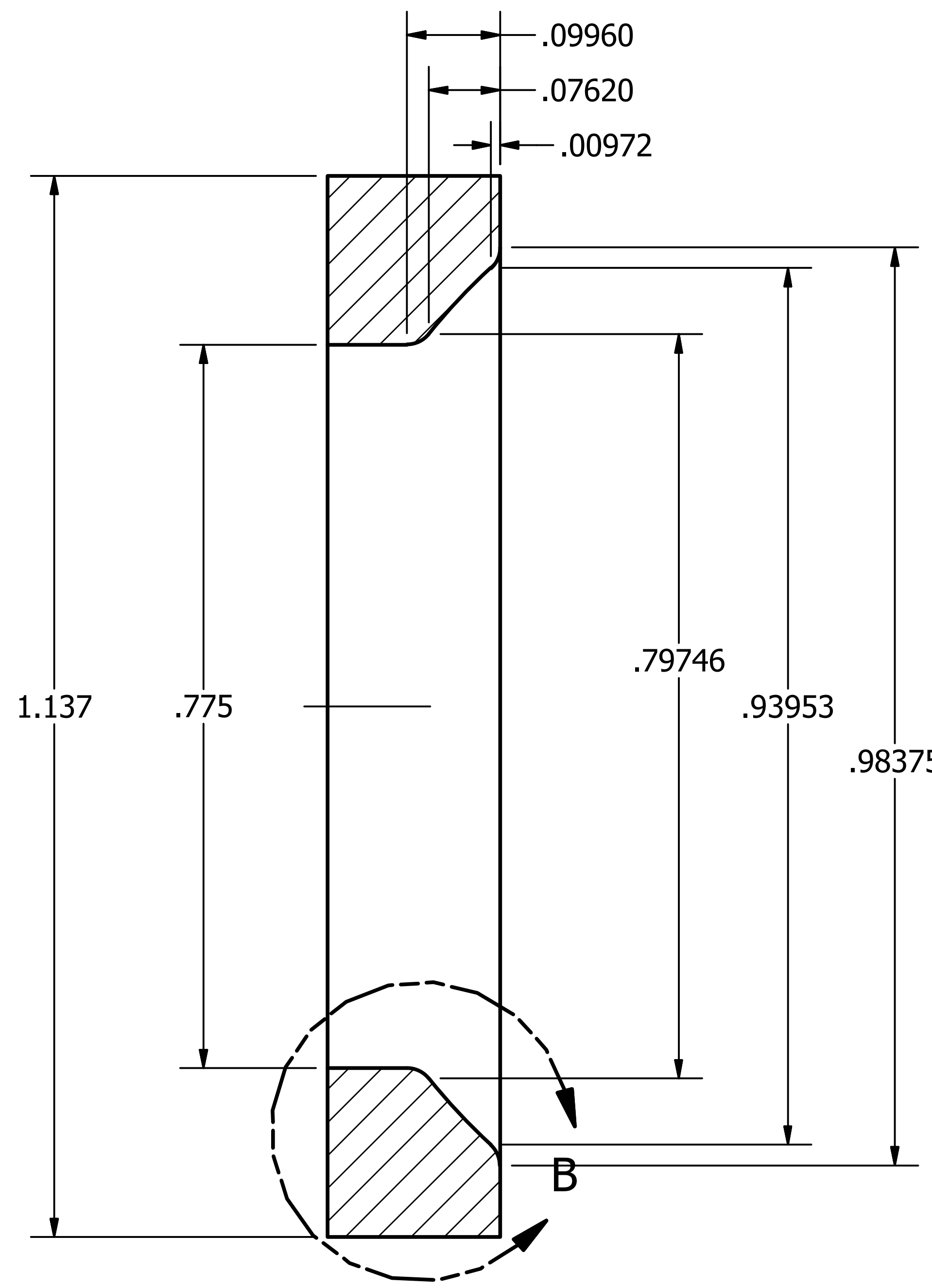
DRAWN		
JRichlin	3/17/2010	Richlin Machinery Inc **OmniTurn Training 631 694-9400**
		TITLE **Part 11**
Material:		SIZE C / DWG NO **OmniTurn Training** / REV
		SHEET 10 OF 11



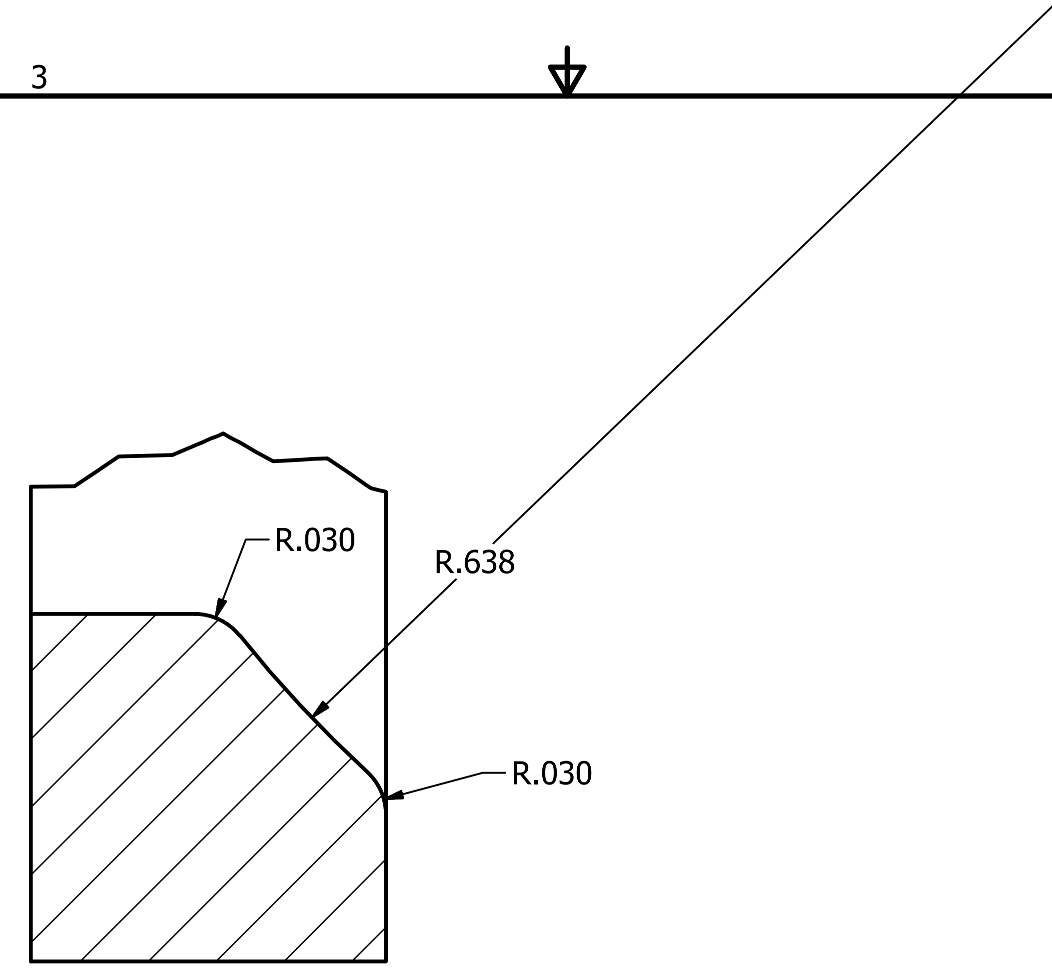


DRAWN JRichlin	3/7/2011	**Richlin Machinery Inc** Engineered - To - Order	
		TITLE	
		E-1-9694	
Rev 0		SIZE C	DWG NO / REV 0
Rev		SCALE	SHEET 1 OF 1

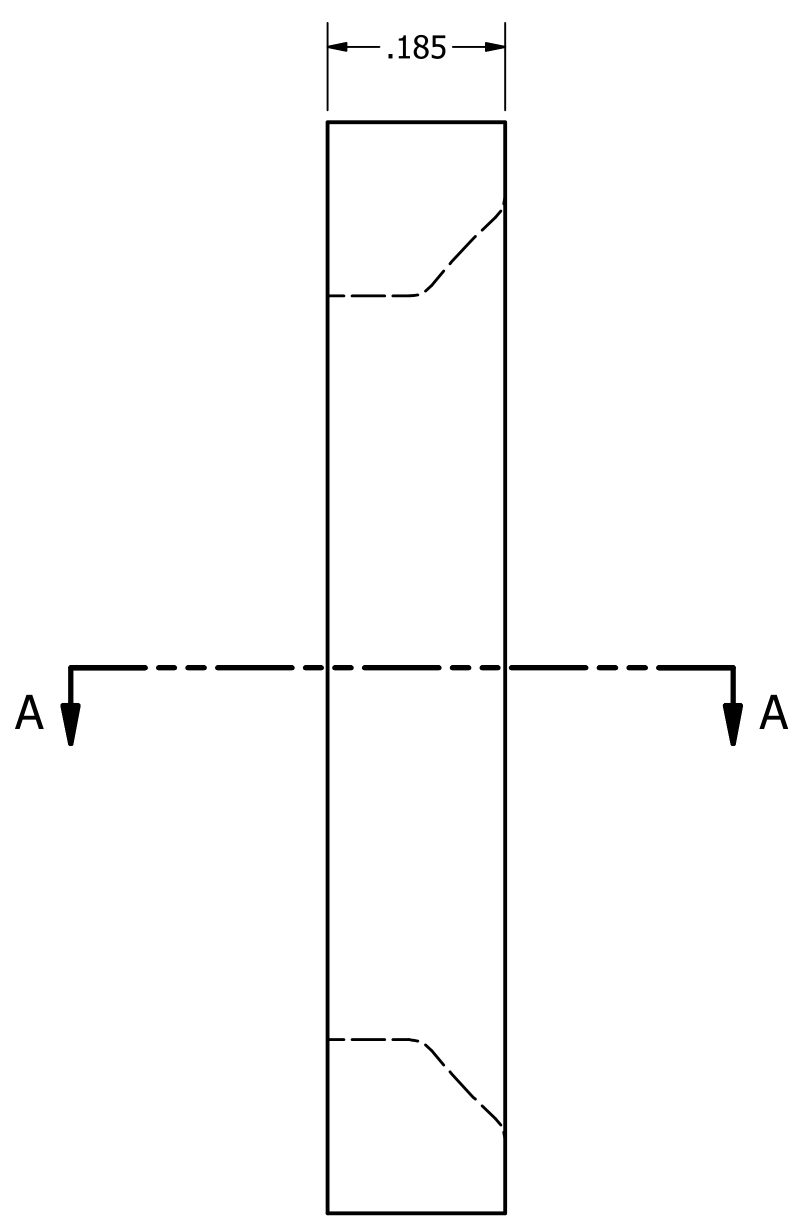
Using Tool Nose Radius Compensation
Solving partial arcs with G02 and G03



SECTION A-A
SCALE 5 : 1



DETAIL B
SCALE 10




```
t1 (lh turn tool 015 tnr)
x.8 z1
z0
g95f.003x1.2
z.1
g42
z0 x1.17 D1
x.98375
g03 x.93953 z-.00972 r.03
g02 x.79746 z-.0762 r.638
g03 x.775 z-.0996 r.03
z-2
x.7
g40
z1g00
```

D1 = x .015 z -.015 r .015

DRAWN JRichlin	3/7/2011	Richlin Machinery Inc **Engineered - To - Order**
		TITLE
		E-1-9694
Rev 0		SIZE C — DWG NO Partial arc solution — REV 0
Rev		SCALE — SHEET 1 OF 1

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 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 axes. 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	X*n.nnnn*Z*n.nnnn***D*n***

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.

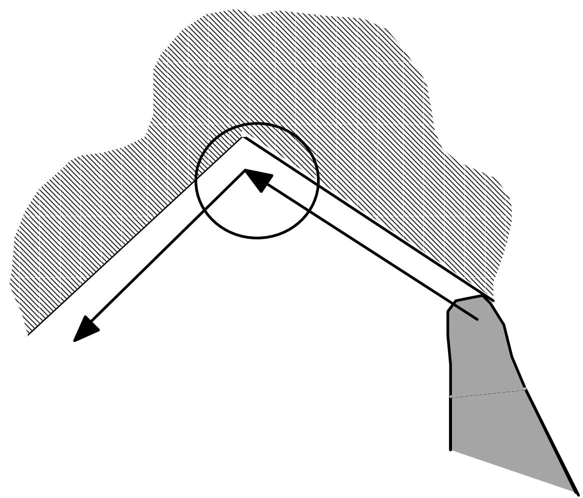
Sequence

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 comensation 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.

TOOL NOSE RADIUS COMPENSATION G41, G42, G40

• 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 except:
  ~~Drilling~~
  ~~Threading~~
• 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 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 atleast twice the TNR value.
• The compensation looks ahead at the next move to help eliminate over travel into corners

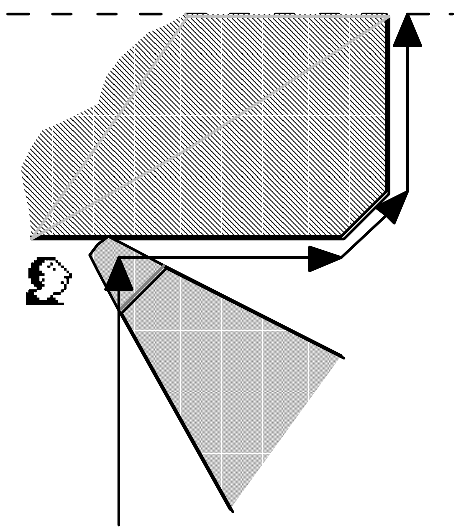


• When using the TNR compensation the tool path gets shifted off the finished size. This does not matter if the tool being used to take a finish pass is different than the roughing tool. The tool is shifted in the setup to give a correct finished size. If the same tool is used to do the rough and finish 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.

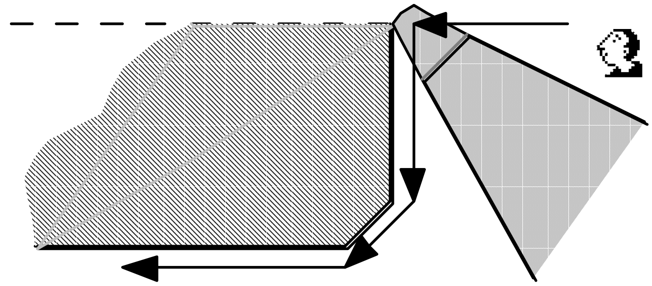
Right or left?:
The right and left compensations are based on the type of move you are performing, not the type of cutter. The type of compensation is described by looking at what side of the cut the center of the tool nose radius is. Imagine that you are sitting at the center of the tool nose radius, 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:



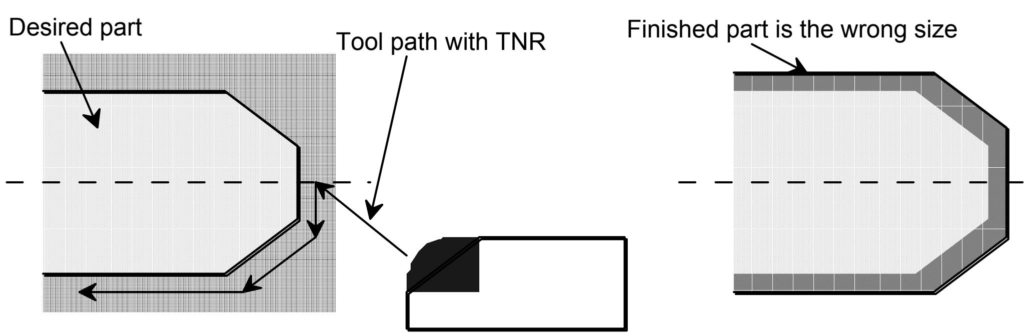
Right handed compensation
G42



Left handed compensation
G41

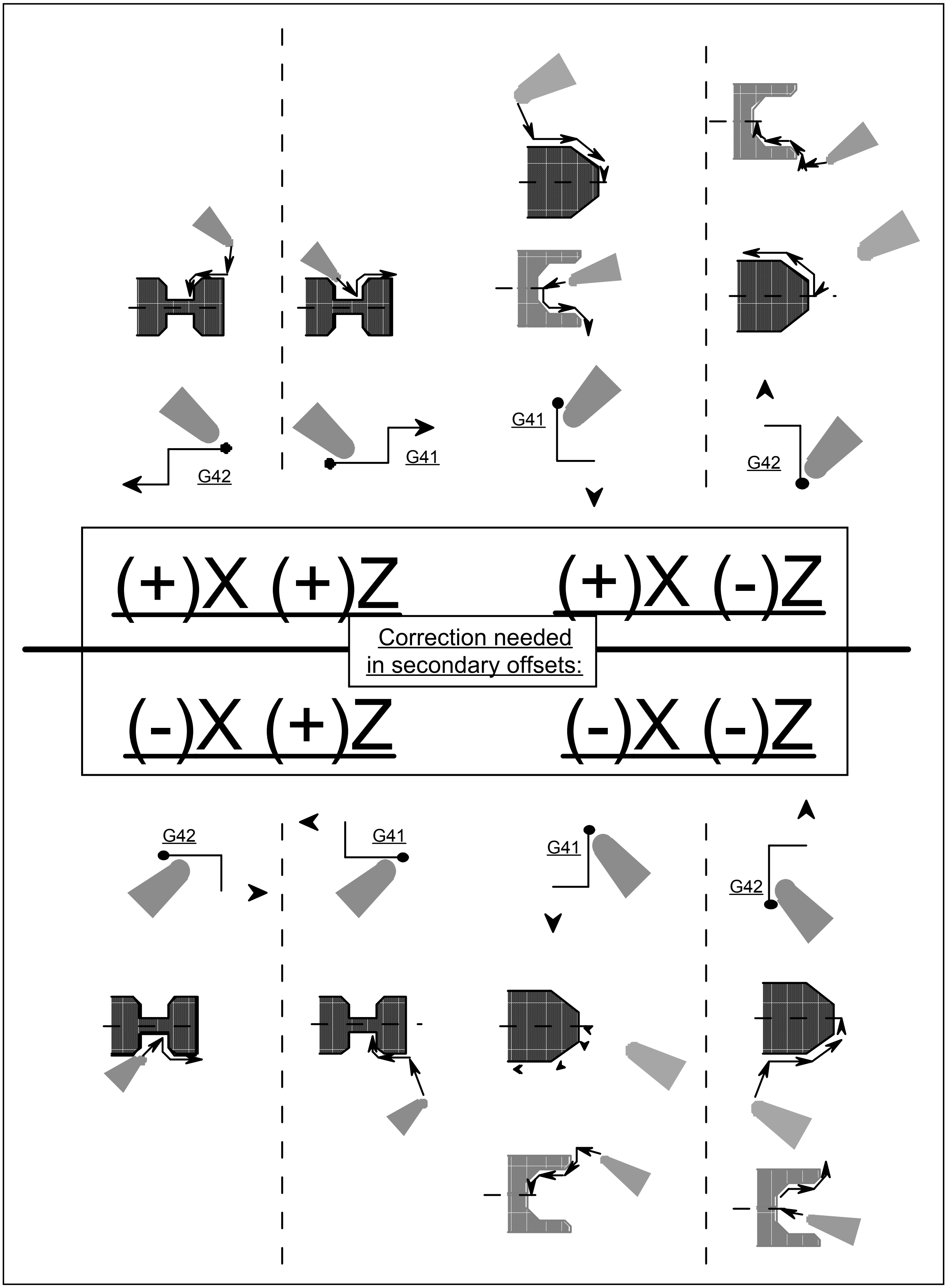
Shifting the TNR compensation

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



If you enter the incorrect sign for the secondary offset value the result will be the part will not be the right size.

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

1	X: +0.00000	Z: +0.00000	R:0.0000	**17**	X: +0.00000	Z: +0.00000	R:0.0000
2	X: +0.00000	Z: +0.00000	R:0.0000	**18**	X: +0.00000	Z: +0.00000	R:0.0000
3	X: +0.00000	Z: +0.00000	R:0.0000	**19**	X: +0.00000	Z: +0.00000	R:0.0000
4	X: +0.00000	Z: +0.00000	R:0.0000	**20**	X: +0.00000	Z: +0.00000	R:0.0000
5	X: +0.00000	Z: +0.00000	R:0.0000	**21**	X: +0.00000	Z: +0.00000	R:0.0000
6	X: +0.00000	Z: +0.00000	R:0.0000	**22**	X: +0.00000	Z: +0.00000	R:0.0000
7	X: +0.00000	Z: +0.00000	R:0.0000	**23**	X: +0.00000	Z: +0.00000	R:0.0000
8	X: +0.00000	Z: +0.00000	R:0.0000	**24**	X: +0.00000	Z: +0.00000	R:0.0000
9	X: +0.00000	Z: +0.00000	R:0.0000	**25**	X: +0.00000	Z: +0.00000	R:0.0000
10	X: +0.00000	Z: +0.00000	R:0.0000	**26**	X: +0.00000	Z: +0.00000	R:0.0000
11	X: +0.00000	Z: +0.00000	R:0.0000	**27**	X: +0.00000	Z: +0.00000	R:0.0000
12	X: +0.00000	Z: +0.00000	R:0.0000	**28**	X: +0.00000	Z: +0.00000	R:0.0000
13	X: +0.00000	Z: +0.00000	R:0.0000	**29**	X: +0.00000	Z: +0.00000	R:0.0000
14	X: +0.00000	Z: +0.00000	R:0.0000	**30**	X: +0.00000	Z: +0.00000	R:0.0000
15	X: +0.00000	Z: +0.00000	R:0.0000	**31**	X: +0.00000	Z: +0.00000	R:0.0000
16	X: +0.00000	Z: +0.00000	R:0.0000	**32**	X: +0.00000	Z: +0.00000	R:0.0000

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

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

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

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

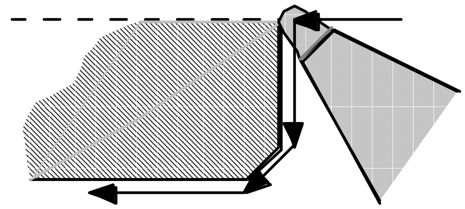
Then: Enter the value of tool nose compensation, IE .007 and then press ESC

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

TOOL NOSE RADIUS COMPENSATION G41, G42, G40

Worked examples

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



G90G94F300
M03S2000
T1 (LH turn tool with .015 tnr)
X0Z1
Z.05
G95F.003

G41	Turn on left hand tool nose radius compensation
X0Z0**D1**	Use the radius value found in secondary offset #1
X.22	
X.25Z-.03	
Z-.3	
X.27	
G94F300Z2	
G40	Turn off the TNR compensation on the Z2 move
M30	

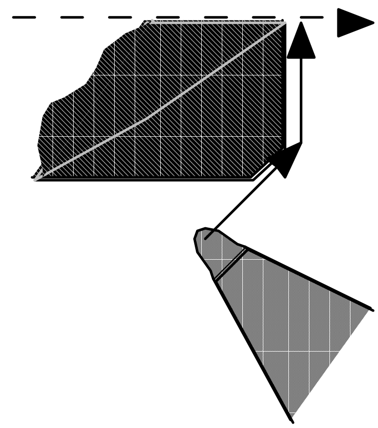
Before running the program the setup person must make sure that there are the correct values in the secondary offset D1.

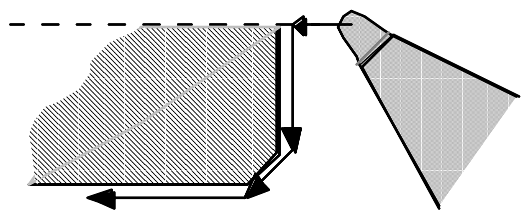
X = -.01500 Z = -.01500 R = .01500

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

The next example shows a tool being used with both right and left compensation. First the tool will be used to face onto the part. In this move the compensation is G42 (right). After the face move is done, the tool has to come off the material so it can come back on with G41 (left) compensation.





G90G94F300	
M03S2000	
T1	
X0Z1	
X.28	
G42	Turn on right hand tool nose radius compensation
X.28Z-.595**D1**	Use the radius value found in secondary offset #1
G95F.003X.225Z.005	
X0	
G94F200Z.1	
G40	Turn off the TNR compensation on the Z.1 move
G95F.003	
G41	Turn on left hand tool nose radius compensation
X0Z0**D2**	Use the radius value found in secondary offset #2
X.22	
X.25Z-.03	
Z-.3	
X.3	
G94F300Z2	
G40	Turn off the TNR compensation on the Z2 move
M30	

The values for the secondary offsets for TNR compensation should be:

D1	X = -.015	Z = -.015	R = .015
D2	X = -.015	Z = -.015	R = .015

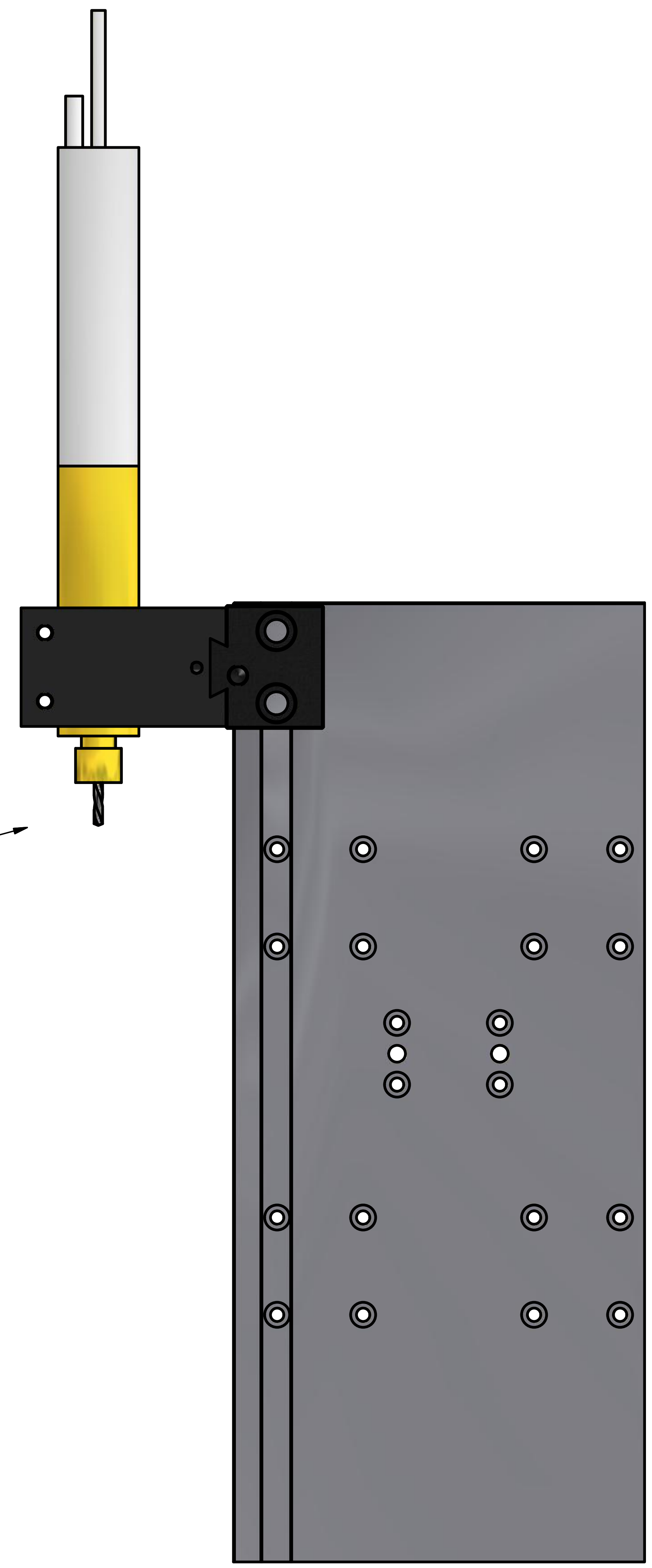
Notice in this example that the values of D1 and D2 are the same. You could use the same secondary offset (ie D1) for both the G41 and G42 offsets.

Running a program that used Tool Nose Radius Compensation

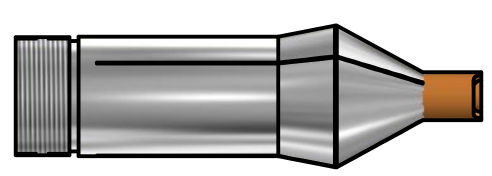
When you write a program with TNR compensation there is another program that is created automatically that has all of the moves that make up the compensated program. When you run the program you will see extra moves in your program that you did not write. This is normal. If you run a program in single 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.

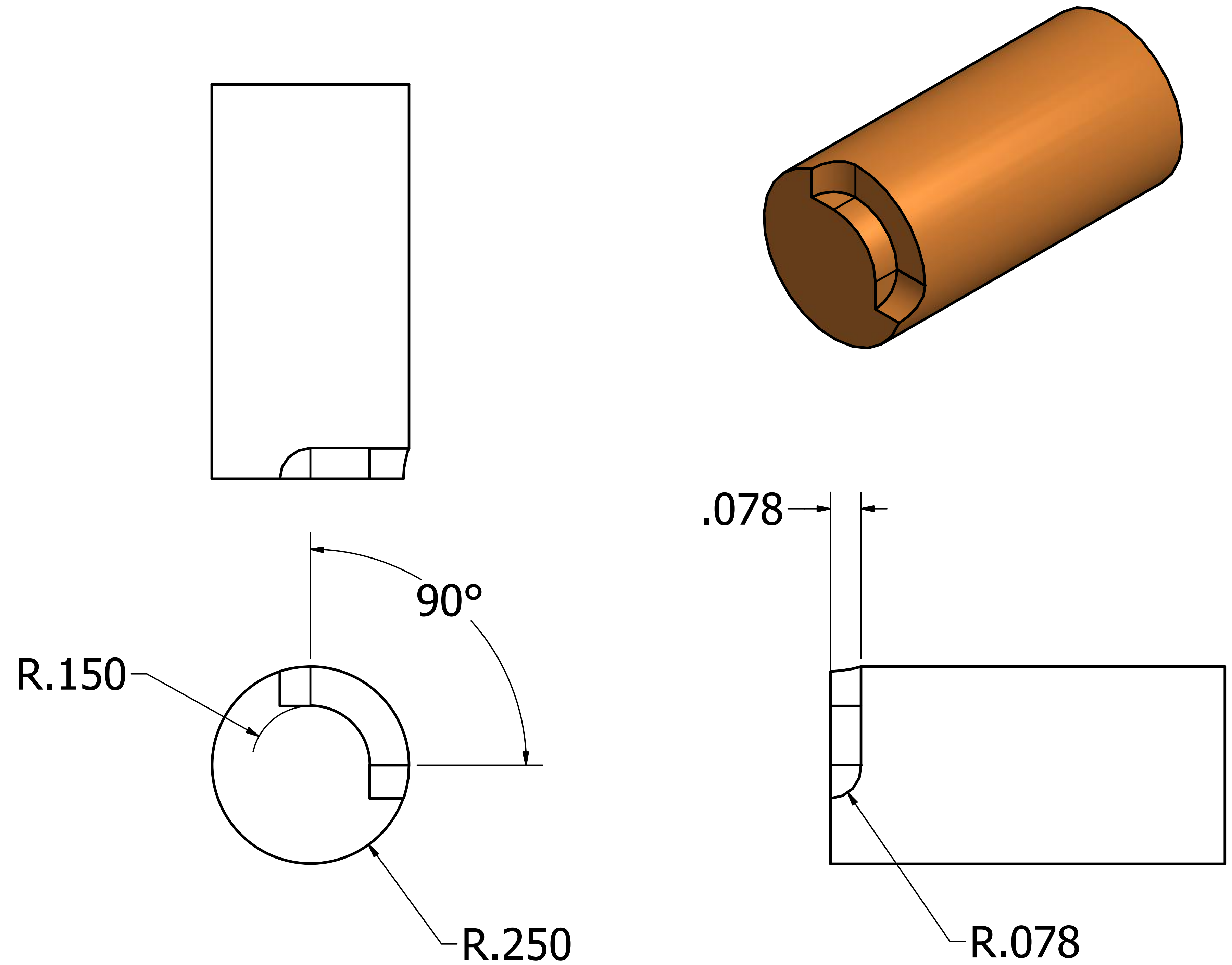
When you go to the editor you will be working on your original program. After you leave the editor the program will automatically be rewritten and stored so it is ready to run. Also every time you leave the secondary offset table the program will be rewritten to compensate for the new tool nose radius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

Topic: Live tool - milling a rotary slot



T19 Live tool end mill
M27 / M28





Process:
Load blank into collet.
T19 Live tool - create slot

DRAWN JRichlin	3/17/2010	Richlin Machinery Inc **OmniTurn Training 631 694-9400**
		TITLE **Part 12**
Material:		SIZE C / DWG NO **OmniTurn Training** / REV
		SHEET 11 OF 11

Part: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\sample 12.ipt
Drawing: C:\Richlin-Cad-Data\Machines\OmniTurn training demos\OmniTurn training setup 1.idw

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" 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.

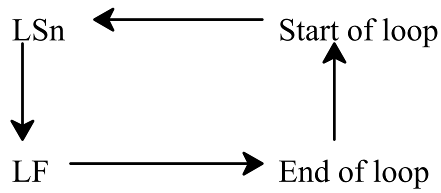
}n - Begin subroutine n
The first line in any subroutine must be the brace } followed by the subroutine number. No other text on that line.

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.

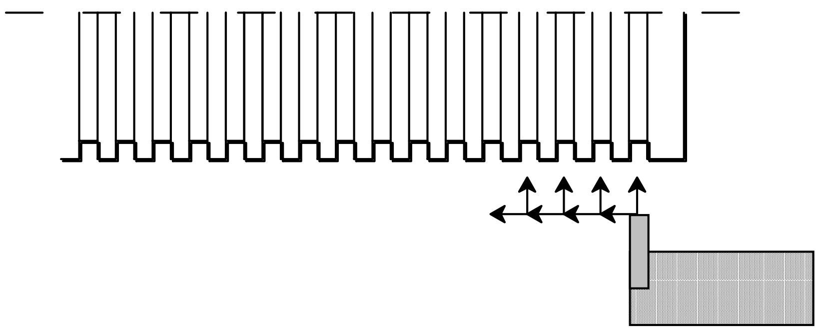
Looping

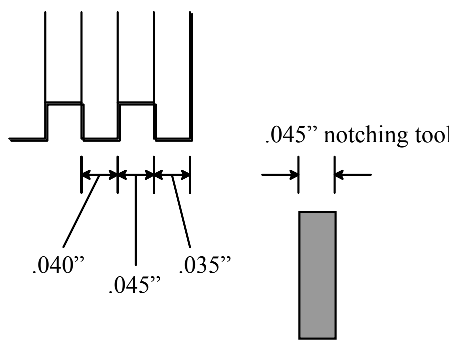
Looping is used to perform repetitive moves without having to write long programs. The start of a loop is defined by LS and then the number of times you want to execute the loop. IE: LS35 will start a loop with 35 repetitions. This command has to be on a line by itself. As the end of the loop put a LF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





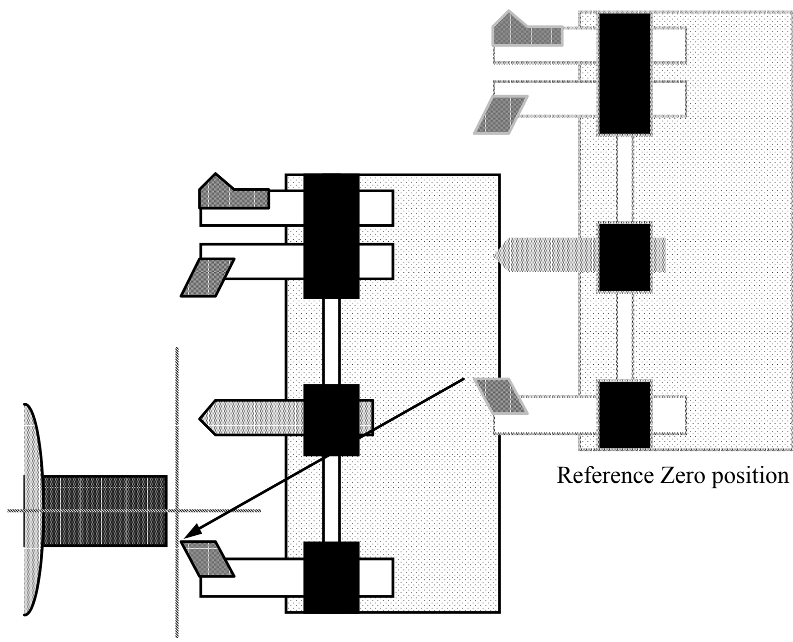
For this example the could could be:
G90G94F300
T1 (notch tool 045 wide)
X.25Z.2
Z-.035
G91 -------------------------------------- NOTE THIS LOOP IS DONE IN INCRIMENTAL
LS16
G95F.001X-.1
G04F.05
G00X.1
Z-.085
LF
G90 -------------------------------------- BACK TO ABSOLUTE MODE
Z1
M30

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:

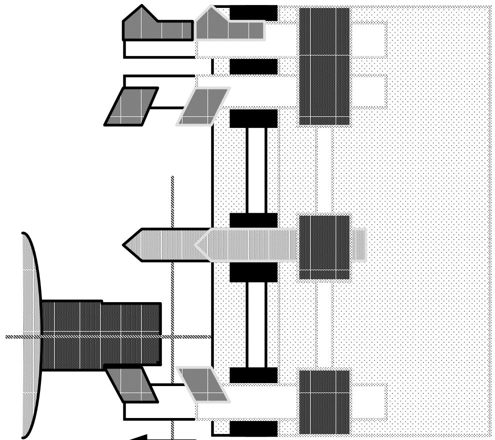
- The material is approx. .5" Ø.
- The part will be programmed so that all of the tools will start at the center of the part in X, and .1" away from the face in Z. ***Follow this format for the first few programs you write. Later you can be more efficient with time and movements after you have more experience!***

To Set Left Hand Turning Tools (Tool #3)

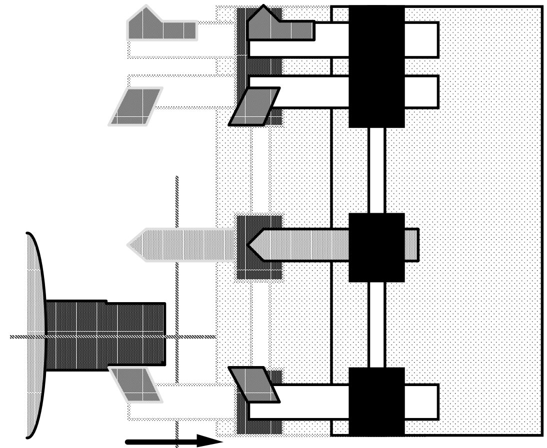
- **Be sure that the slide has been HOMED.**
- From the main screen go to the Jog mode by typing **"J"**
- Using the jog keys and joystick move the cutting tool until it is just off the material and slightly smaller than the major diameter.



•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

```
POSITION    : X +0.00000  Z +0.00000       FEED  10.0 IPM

COMMAND     : X +0.00000  Z +0.00000       PERCENT FEED:  100



MAKE JOG SELECTION

Jog       Automatic      Single Block       Manual Data Input
  1.Slow              7..1000
  2.Medium            8.1.0000                      -X
  3.Fast              9.Est. Home                    ↑
  4..00005            S.Set Zero              -Z←          →+Z
  5..0010             H.Go Home                      ↓
  6..0100             T.Set Tool                    +X


 PRESS 'ESC' TO RETURN TO MENU
```

Select "T" to start entering the 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.

```
POSITION    : X +0.00000  Z +0.00000

COMMAND     : X +0.00000  Z +0.00000


OFFSET NUMBER ?

Jog        Automatic       Single Block       Manual Data Input
  1.Slow              7..1000                    -X
  2.Medium            8.1.0000                    ↑
  3.Fast              9.Est Home          -Z←          → +Z
  4..00005            S.Set Zero                  ↓
  5..0010             H.Go Home                  +X
  6..0100             T.Set Tool
 O IS NOT A VALID OFFSET NUMBER
 PRESS ESCAPE TO RETURN TO JOG MENU
 PRESS 'ESC' TO RETURN TO MENU
```

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

```
POSITION    : X +0.00000  Z +0.00000

COMMAND     : X +0.00000  Z +0.00000


PRESS X OR Z TO STORE PRESENT X OR Z AXIS OFFSET

Jog        Automatic       Single Block       Manual Data Input
  1.Slow              7..1000                    -X
  2.Medium            8.1.0000                    ↑
  3.Fast              9.Est Home          -Z←          → +Z
  4..00005            S.Set Zero                  ↓
  5..0010             H.Go Home                  +X
  6..0100             T.Set Tool
 O IS NOT A VALID OFFSET NUMBER
 PRESS ESCAPE TO RETURN TO JOG MENU
 PRESS 'ESC' TO RETURN TO MENU
```

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.

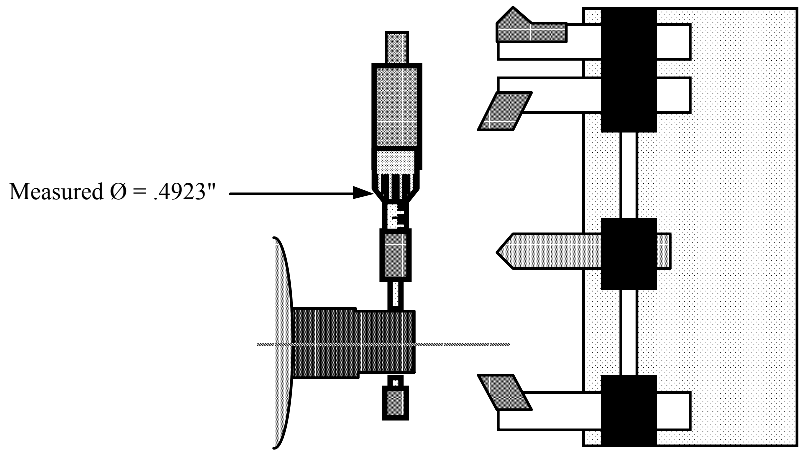
```
POSITION   : X +0.00000  Z +0.00000

COMMAND    : X +0.00000  Z +0.00000


WHAT WAS THE TURNED DIAMETER?

Jog       Automatic      Single Block      Manual Data Input
  1.Slow              7..1000                   -X
  2.Medium            8.1.0000
  3.Fast              9.Est Home
  4..00005            S.Set Zero          -Z         +Z
  5..0010             H.Go Home
  6..0100             T.Set Tool                +X
 O IS NOT A VALID OFFSET NUMBER
 PRESS ESCAPE TO RETURN TO JOG MENU
 PRESS 'ESC' TO RETURN TO MENU
```

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.

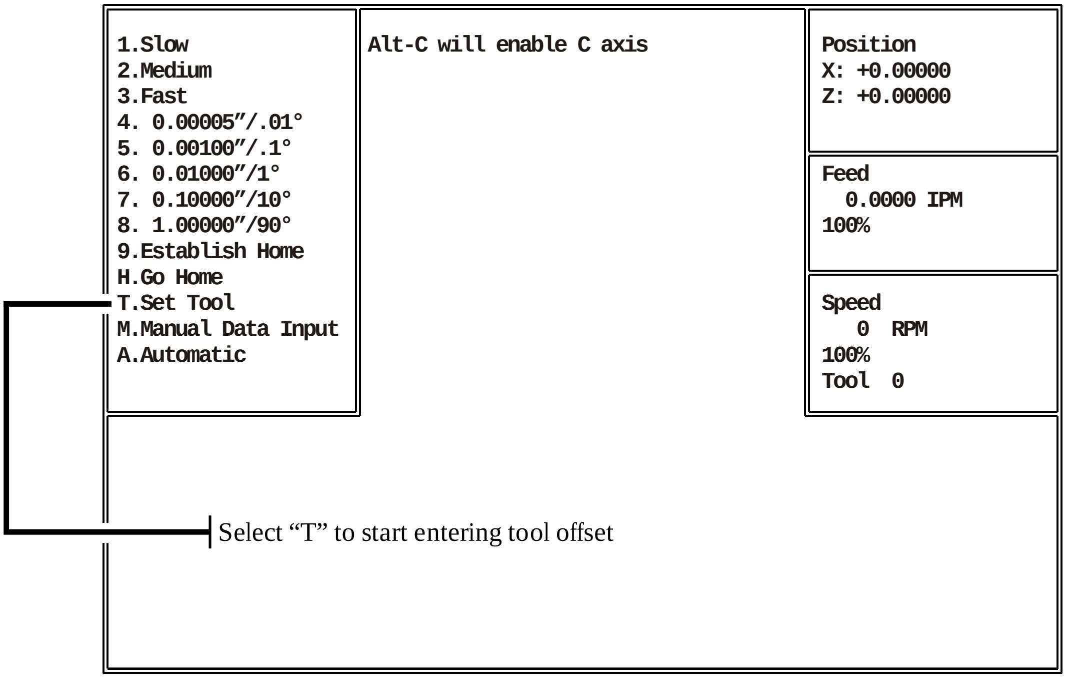
Touch the tool off in the Z axis and then press T like you did with the X axis.
The control will now ask for a tool number. In this example you would press 3 and then enter.
Then press Z when asked which axis you are setting.
Then the control askes what the location of the tool is from absolute zero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

```
POSITION    : X +0.00000  Z +0.00000
COMMAND     : X +0.00000  Z +0.00000

WHAT IS THE CURRENT Z LOCATION
Jog       Automatic       Single Block       Manual Data Input
  1.Slow             7..1000
  2.Medium           8.1.0000                    -X
  3.Fast             9.Est Home
  4..00005           S.Set Zero            -Z         +Z
  5..0010            H.Go Home
  6..0100            T.Set Tool                  +X
 O IS NOT A VALID OFFSET NUMBER
 PRESS ESCAPE TO RETURN TO JOG MENU
 PRESS 'ESC' TO RETURN TO MENU
```

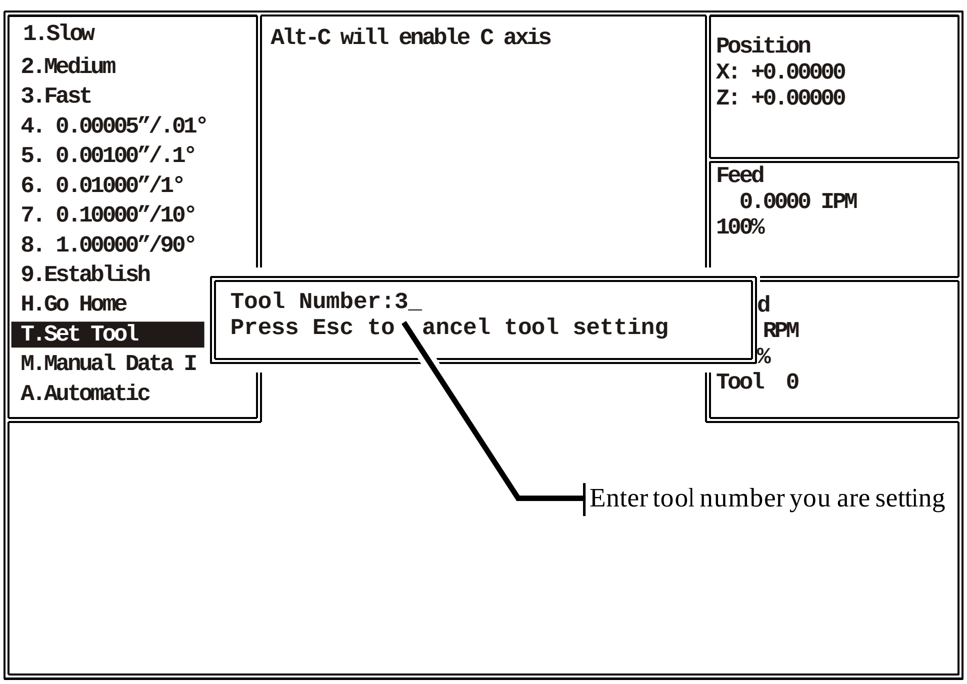
Enter where the tool is from absolute zero on the part

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.

To Set Left Hand Turning Tools continued

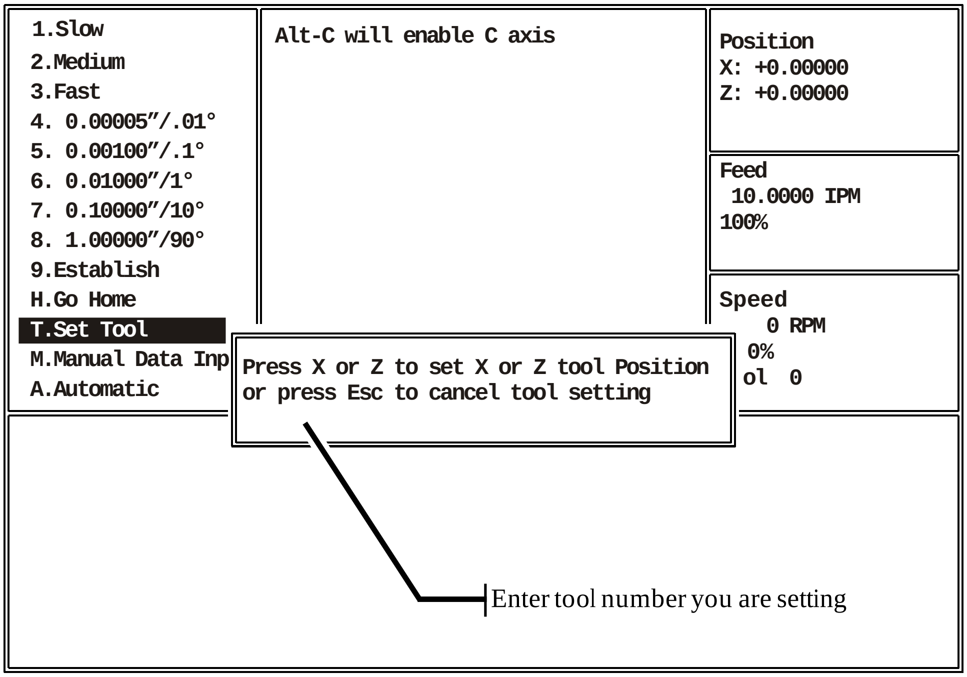


At this point, you are still in the jog mode. Instead of selecting a new jog speed now, select "T". The control will now begin the sequence for entering a tool offset. See the next screen.

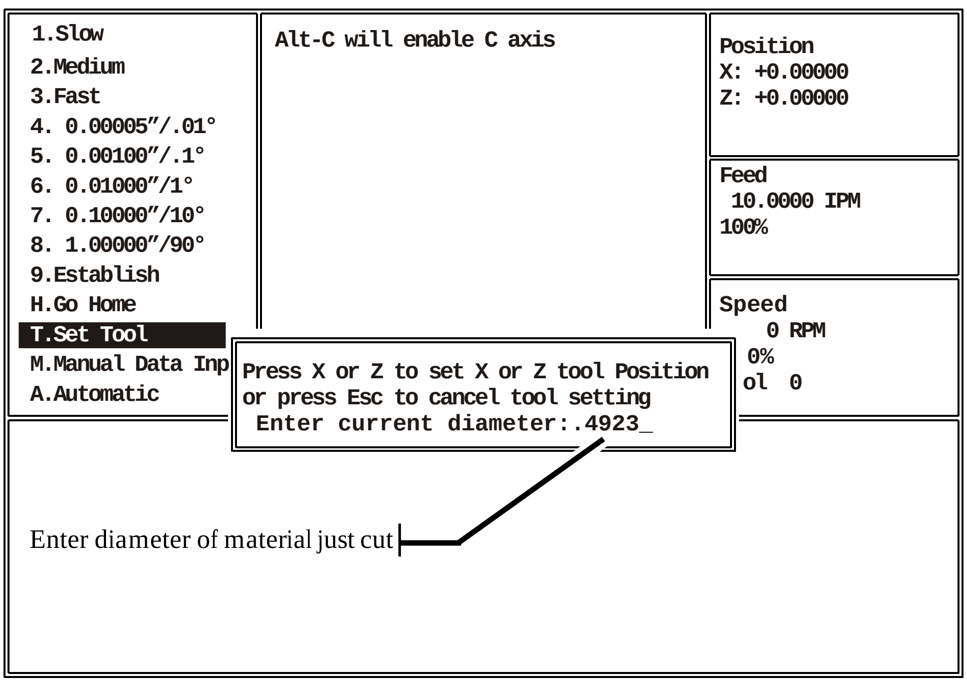


After you have selected T, the control will ask what tool it is that you are about to enter. Type the number tool, in this case 3. Then hit the "RETURN" key.

To Set Left Hand Turning Tools -continued



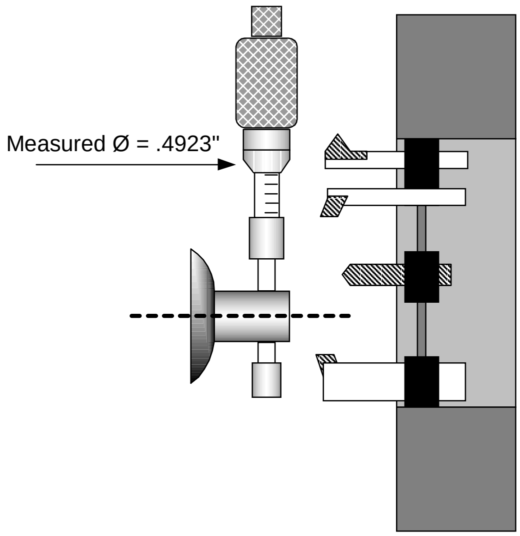
After you have selected the #3 tool offset, the control will ask you whether you want to enter the Z or X offset. In this case we have set the tool on the diameter of the material and we are ready to enter the X offset, so hit X.



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

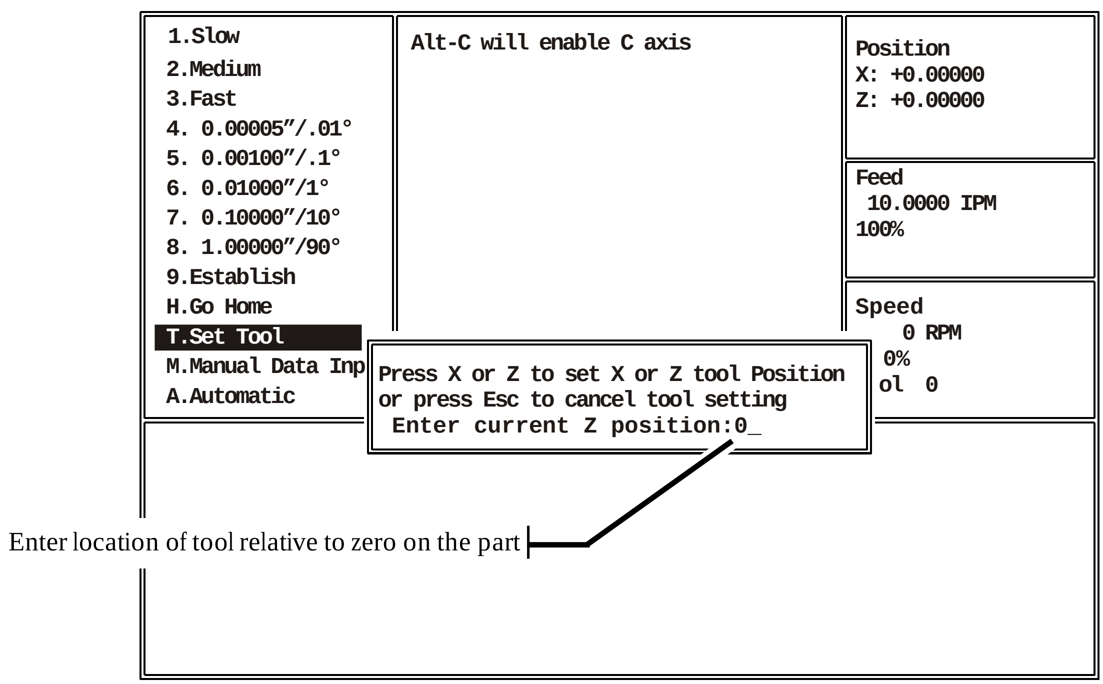


Take some care but do not be overly careful since any error made here can be easily corrected with the tool offset correction later when you are making the first piece. After typing .4923 hit "RETURN".

Now establish the Z offset, for tool #3

The setting of the Z offset is a little different.

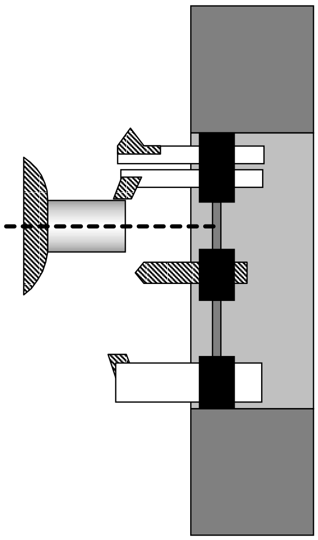
- Touch the tool off in the Z axis and then press T like you did with the X axis.
- The control will now ask for a tool number. In this example you would press 3 and then enter.
- Then press Z when asked which axis you are setting.
- Then the control asks what the location of the tool is from absolute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



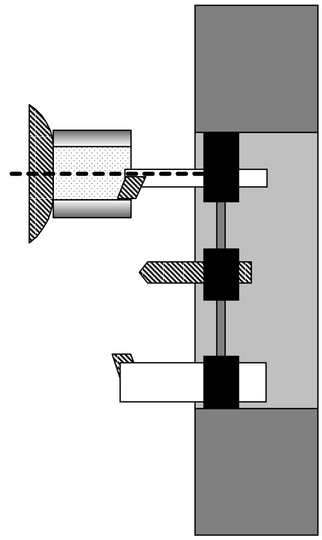
Using a finished part to touch off in Z:
If you put a finished part in the collet against a stop this would give you the absolute face of the part. The parts you machine should have the same location in Z when they are done. So you can jog the tool over the face of the part and touch off and when asked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

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

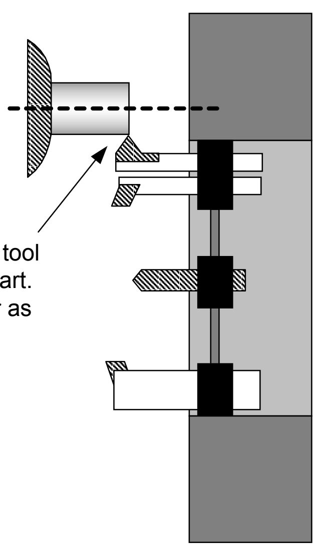


Setting a boring tool on the ID.
The measured Ø will be entered as a
POSITIVE

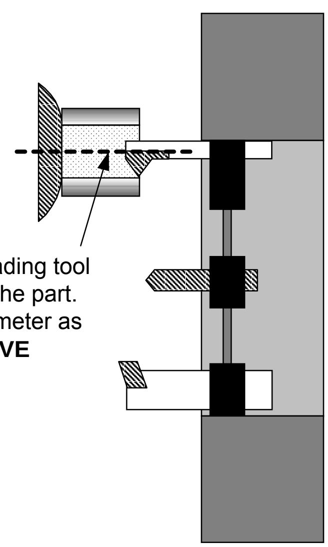
Setting Threading tools

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.

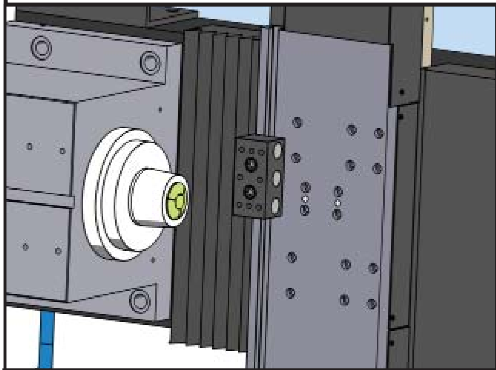


Setting a threading tool
on the OD of the part.
Enter the diameter as
POSITIVE

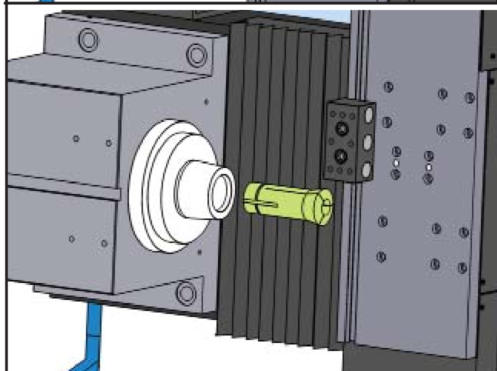


Setting a threading tool
on the ID of the part.
Enter the diameter as
POSITIVE

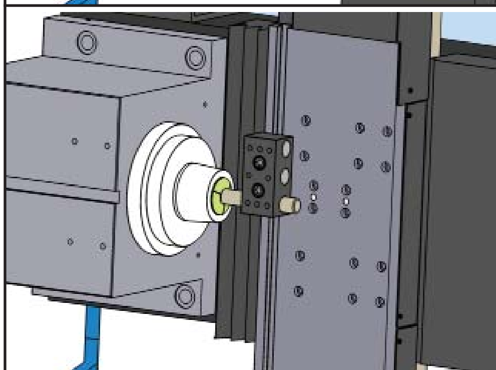
Finding X 0 for a Drill



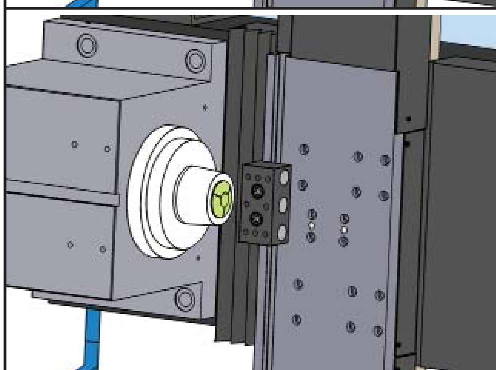
- Setting a drill to be exactly on center is easier than it looks.
- First get a 5/8” collet and a 5/8” shaft. A common shaft to use is a
- Multibar tool holder or a OTC drill shank holder.
-
- Put the drill holder on the tooling plate. The one shown here is an Omni-305 with three holes. Do not tighten it down yet, just hold it enough so it does not fall down. Put the holder where you want it for your setup



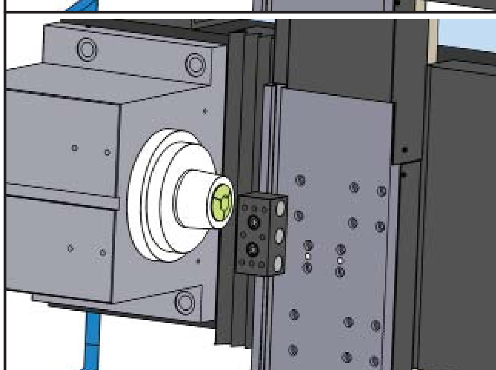
- Now put the 5/8” 5C collet in the spindle.



- Slide the 5/8” shank into the holder.
- Jog the tooling plate so that the 5/8” shaft lines up with the collet. Loosen the Omni-305 so you can slide the shaft into the collet.
- Then lock the collet on the shaft.
- Now tighten the Omni-305 to the tooling plate.
- The tool offset for this first location can now be set for X = 0.
- Do not move the slide yet in the X axis. You can move in Z needed.
- Open the collet, take the 5/8” shaft out.



- It is now easy to set X centerline for the other two holes. The center
- distance between all OmniTurn holders is 1.100”. So we can use the
- jog commands to bring the second hole on center to set it as well.
- Jog the X using the #7 (.1”) and #8 (1.0”) commands.
- With the second hole on center set the X offset value for this tool to 0.



- Repeat the same steps as the previous step to set the 3rd hole.
-
- Now go back and set the Z values with the drilling tools, work piece and correct work holding.

The ”F” keys have the following functions:

NOTE: These keys are not effective while program is running

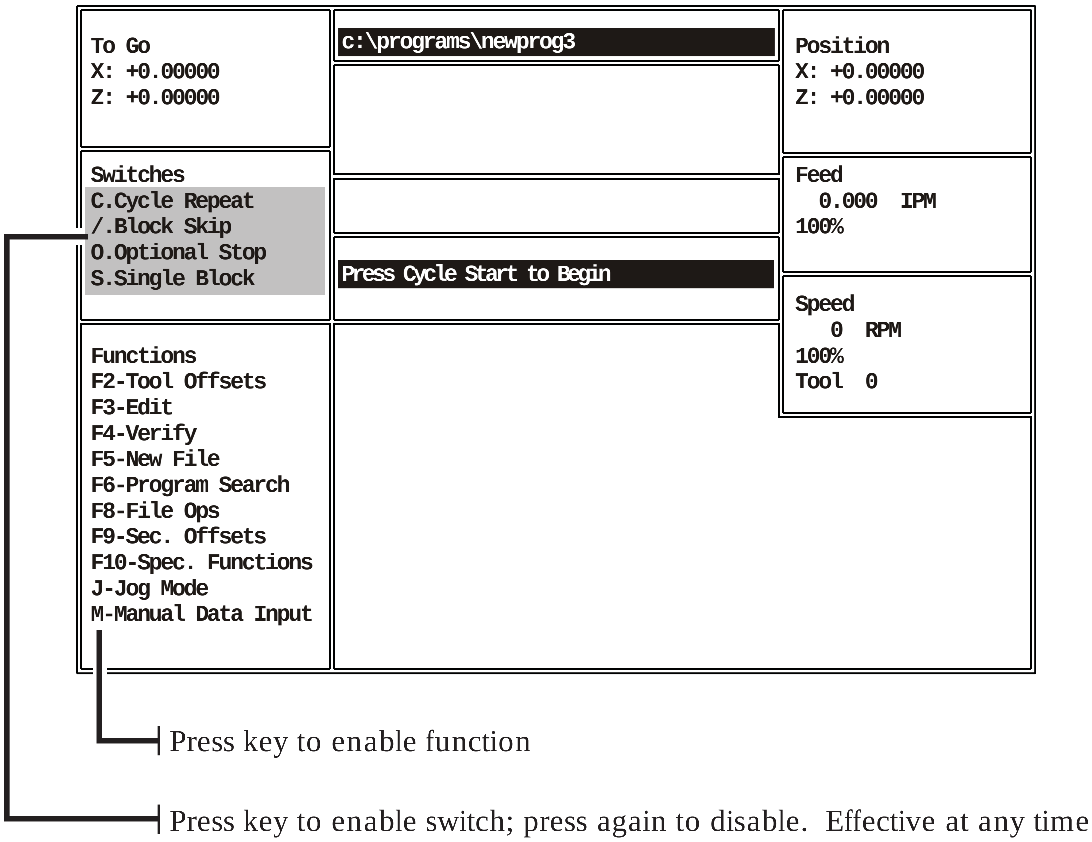
F 2	Tool Offsets	Adjust tool offsets.
F 3	Edit	Edit existing programs or create new programs.
F 4	Verify	Verify the program in memory and display the tool paths.
F 5	New File	This will remove the program from active memory and open the file handler screen.
F 6	Prog Search	This enables the program to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

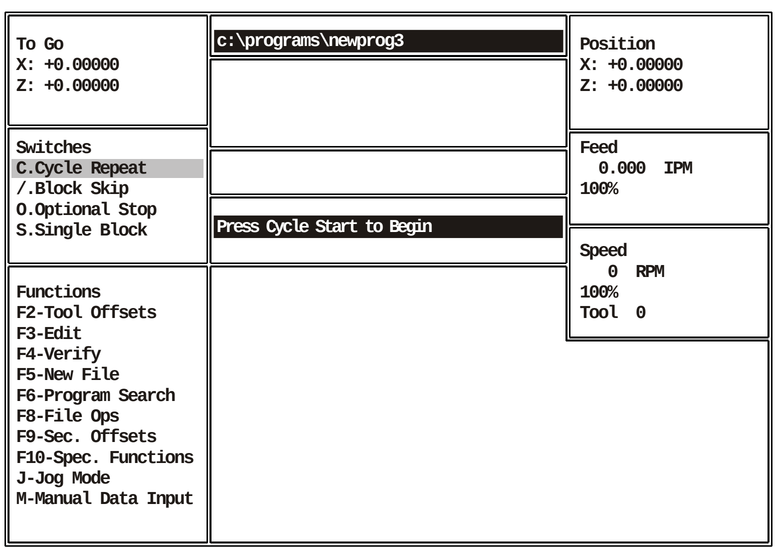
HOT keys on the keyboard while in the automatic mode

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

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

Functions and switches available from Automatic Mode

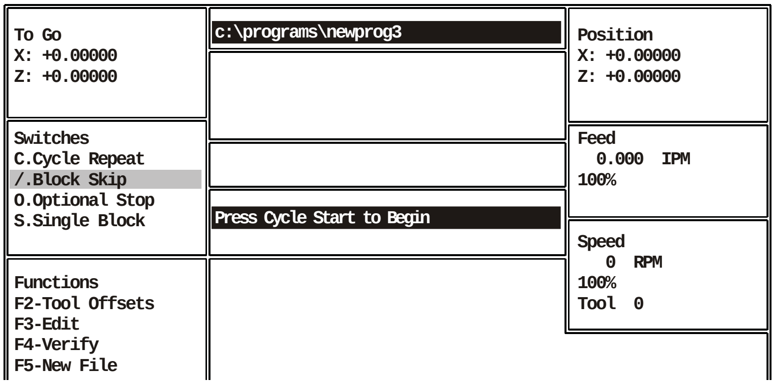




Cycle Repeat Mode

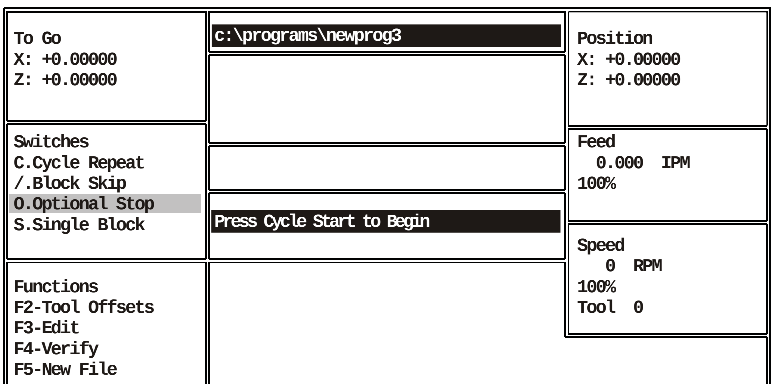
To run the program over and over without pressing Cycle Start, select Cycle Repeat by pressing the C key. This is the preferred mode for bar jobs and loaders, which automatically supply material for machining. To cancel the Cycle Repeat mode, press C again, or press A.

Automatic Mode, Switches, con't



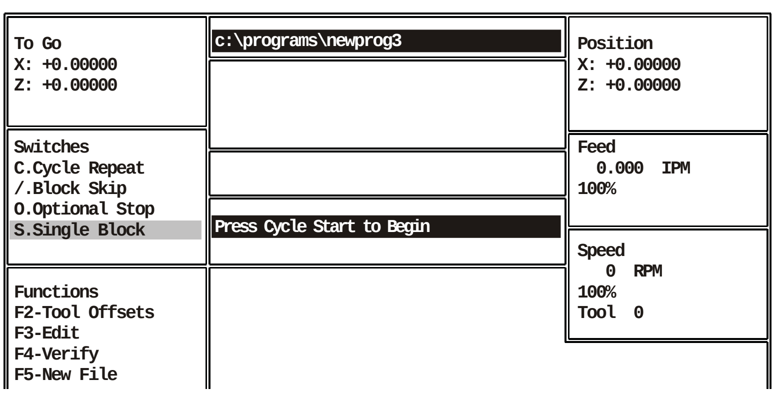
Block Skip Mode

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



Optional Stop Mode

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



Single Block Mode

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

Automatic Mode, Function Keys

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

Automatic Mode, F2: Tool Offsets

F2 Tool offset screen, used to modify tool offsets

This function key brings up the screen to adjust the tool offsets. Tool offsets are used to correct the starting location of the tools, and they will effect the finished part dimensions. These values are created when the tools are setup in the jog mode. When the F2 key is pressed the screen will then ask what tool number you want to adjust. The distances shown are the amount needed to travel from the Home position to the offset location. See below:
In the example above you see the offset screen with three tools being used. When it is necessary to correct a tool offset, enter the amount of change that is required. As an example, we will assume that tool 2 in the above example is a turning tool and is cutting apart .001” too large. So enter the offset change of -.001” for the X Diameter offset. When this value is entered you will notice that the total value of X has changed. This addition does not have to be done by the operator.

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

OFFSET NUMBER:
Press Esc to exit offset adjustment screen

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.

NEW G4 Control

Automatic Mode, F2: Tool Offsets, con't

Notes on Tool Offsets:

1. The control will allow you to clear all the offsets by pressing C (for clear). Please only do this when you have had experience with the control and understand what you are doing. Clearing offsets can cause you to crash tools if it is done incorrectly!

2. The smallest offset changes are:
   0.00005" in Z
   0.0001" in X, (this is equal to .00005" on the radius)

3. Tool Offset changes greater than .05" will invoke "*Do you really want to move [distance] (y/n)?*" prompt to alert the you of possible mistake. This is a safety feature to ensure that you do not put in a large correction in error, *ie* 0.1" instead of .01"

4. If you have no change to a offset value just press Return without inputting a value. The control assumes that you want zero change.

5. Tool offsets can be changed at any program stop: m00, m01 or Motion Stop.

Automatic Mode, F3, Program Editing

F3 Program Editing

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

The editor is described starting on page 5.19.

NEW G4 Control

Automatic Mode, F5: New File, F6: Program Search

F5 New Program; used to change the running program to a new one
This functionis used to change the program that is in the current file and allow the operator to enter in a new program name to be run. When F5 is pressed the file handler screen will appear. You will see the list of programs available. Use the arrow keys to highlight the desired program.
See page 1.17 - 1.20, Jog Mode, File Handler for details.

F6 Program Search; used to start the program at a certain tool instead of the beginning.
The program "DEMO" is used in the following example.
This function will allow you to start the program at a certain tool instead of the beginning.
NOTE: If there are less than two tools in your program, pressing F6 will generate an error as follows:

```
Program contains no Search Points
Pressany key to return to Auto mode
```

If there is more than one tool in your program they will be listed, with highlight on first tool. Use arrow keys to select desired tool, then press Enter.

```
T 1
T 2
T 0


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

When Enter is pressed, another screen will appear defining the tool and the line number, and any M-functions that are active at that point:

```
 Searched to tool  2  ,line 46
The following conditions will be established:
 M03S2500
 M08
Press cycle start to proceed or Esc to cancel
```

Pressing Cycle Start will return you to Auto screen, in Single Block mode, with highlight on selected tool:

```
To Go              | c:\programs\DEMO              | Position
X: +0.00000        |                               | X: +7.86285  ∅
Z: +0.00000        | G94F50                        | Z: -0.62225
                   | T2                            |
                   | x-.5Z.2                       |
Switches           |                               | Feed
C.Cycle Repeat     | THREADING TOOL                | 000.000  IPM
/.Block Skip       |                               | 100%
O.Optional Stop    | Press Cycle Start to continue |
S.Single Block     | or Esc to terminate program   | Speed
                   |                               |    0  RPM
Functions          |                               | 100%
F2-Tool Offsets    |                               | Tool  2
F3-Edit            |                               |
F4-Verify          |                               |
F5-New File        |                               |
```

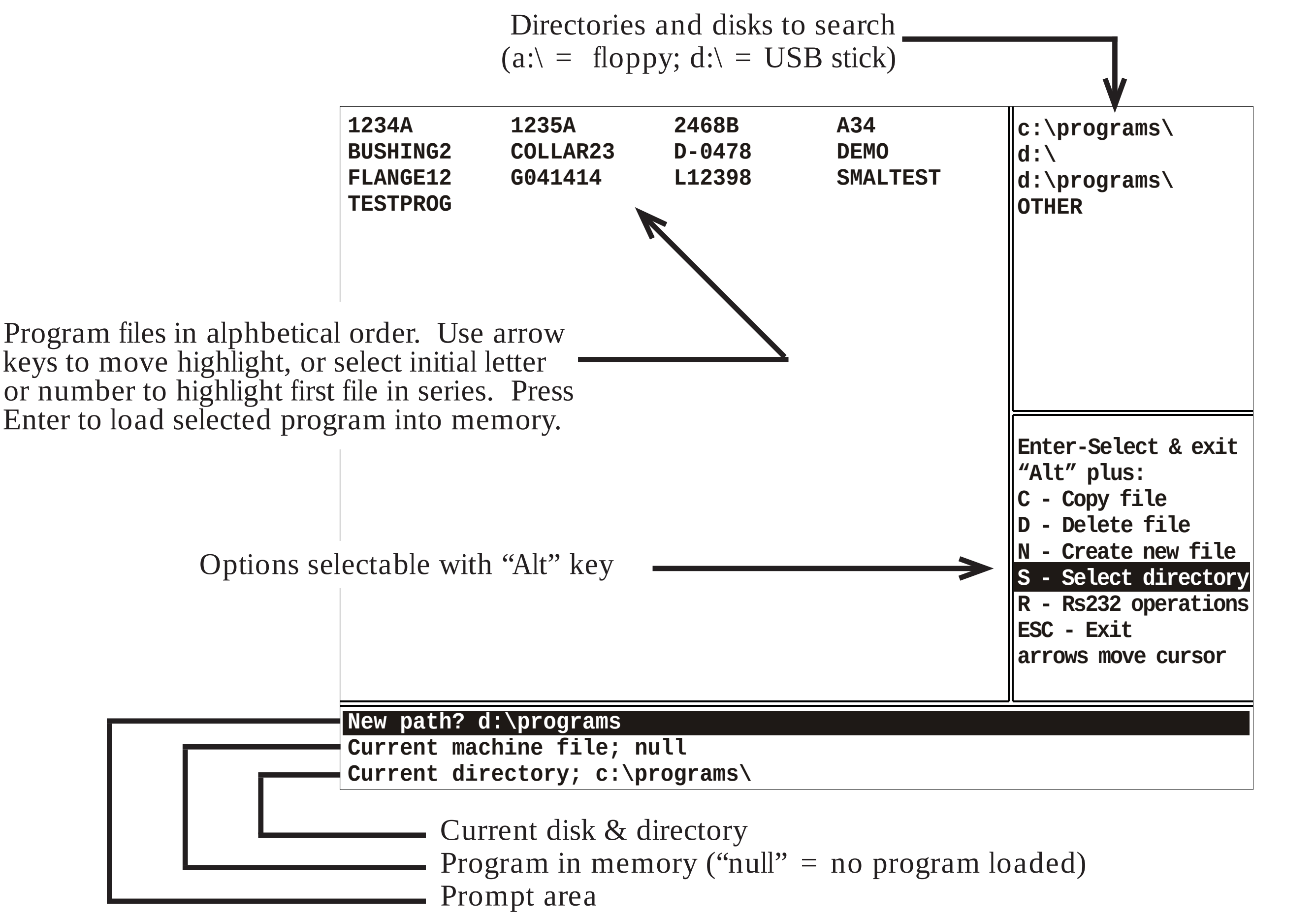
The Progam Search function reads every block of the program up to the tool selected, so the machine will start running as if it has just executed all the previous blocks. When Cycle Start is pressed, the current feed rate will be displayed and the tool will move to the part. Note that no collision avoidance calculation is done, so be sure that the tool can move safely to the part before pressing Cycle Start.

Automatic Mode, F8: File Operations

F8 File Ops; File Operations. This Function allows you to select, copy, delete and create new programs, and to send or receive programs via RS232 interface.

See page 1.17 - 1.20, Jog Mode, File Handler for details.

The screen is described below:



Options Selectable with “Alt” key

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

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

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

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

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

NEW G4 Control

Automatic Mode, F9: Secondary Offsets

F9 Secondary tool offset screen; used to modify secondary tool offsets

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

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

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

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

Changing a TNR compensation value:

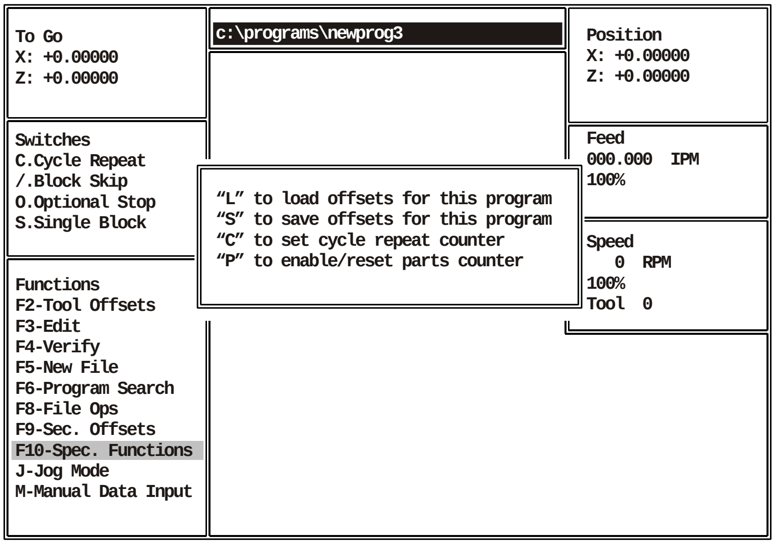
When you put a value in the R offset table it writes over the old value. So if you have a number already in the register that you want to use and it is not the value needed, all you have to do is enter the correct value. As an example if you have a value of .032 in the offset and you want to change it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

Clearing secondary offsets to Zero.

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

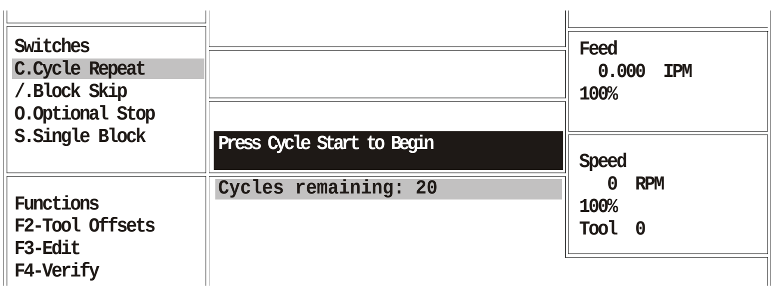
Individual offsets can be set to zero by pressing C when asked to enter a correction amount.

F10 **Special Functions**, used to call up a list of special functions.

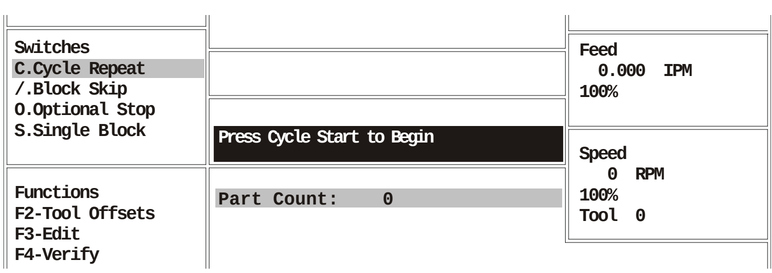


• Save and recall tool offset tables.
This function enables you to load or save the tool offsets with the program. Press L at this screen to load the offsets for the current program *from* the disk; press S to save the offsets for the current program *to* the disk. Use the OmniTurn 805 eight-position tool-holder (p/n 805PFB), which allows you to remove and replace tooling exactly as it was, and this function cuts setup time for periodic jobs to just a few moments. Also works well for re-loading offsets for different programs which use same set of tooling.

• Set the number of cycles to repeat:
Press C at this screen, then enter the number of cycles to repeat, then press Enter to return. Press C at Auto screen to enable Cycle Repeat. This is good for use with a barfeed. As an example you could set the machine up and tell it you need 20 pieces. The OmniTurn will count down to zero and then stop.



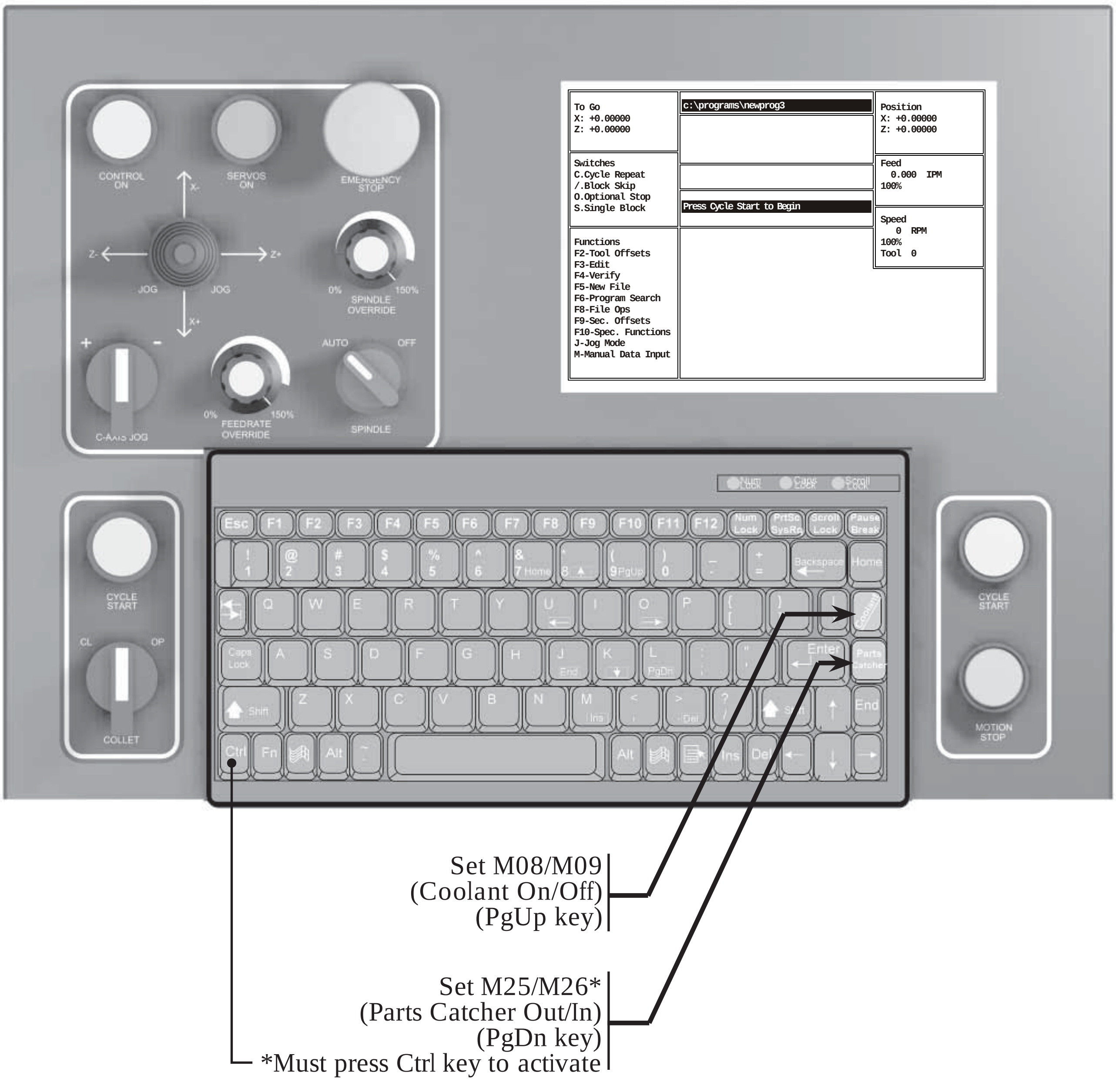
• Turn on the Parts Counter: Press P at this screen. You will return to the Auto mode screen, and there will be a counter on the screen. The counter will count up each time an M30 or M02 is executed. To clear the counter come back into the F10 screen and press P again.



NEW G4 Control

M function keyboard controls

Toggle M functions on and off with keyboard controls.
-Press the key once to turn the function on, Press again to turn it off
-Works only in Jog or Automatic mode



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

NEW G4 Control

Automatic Mode

The "F" keys have the following functions:

F1	Quit	Go back to the Main menus
F2	Offset	Adjust tool offsets, correct part size
F3	Edit	Input and correct programs
F4	DIR VER	When no file is in memory this will list all the programs on the user disk With a program in memory it will verify and plot the program
F5	Newprog	This will remove the program from active memory and allow a new one to be entered
F6	Searchto	This enables the program to be started someplace other than the start
F7	Prog	Runs Calcaid programming system
F8	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
F9	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

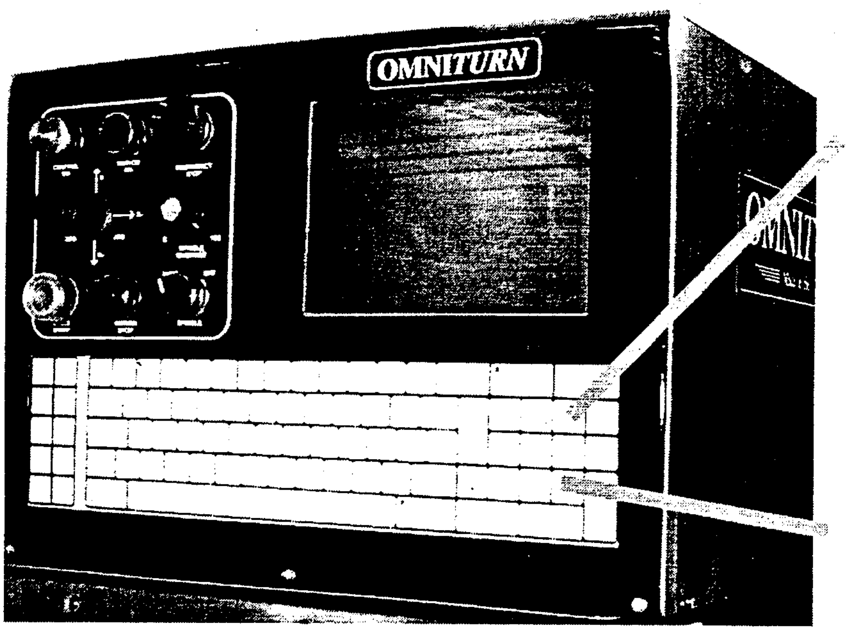
HOT keys on the keyboard while in the automatic mode

C	Continuous	The program will run continuously - This toggles on / off
O	Optional stop	This makes the M01 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 feedrates (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



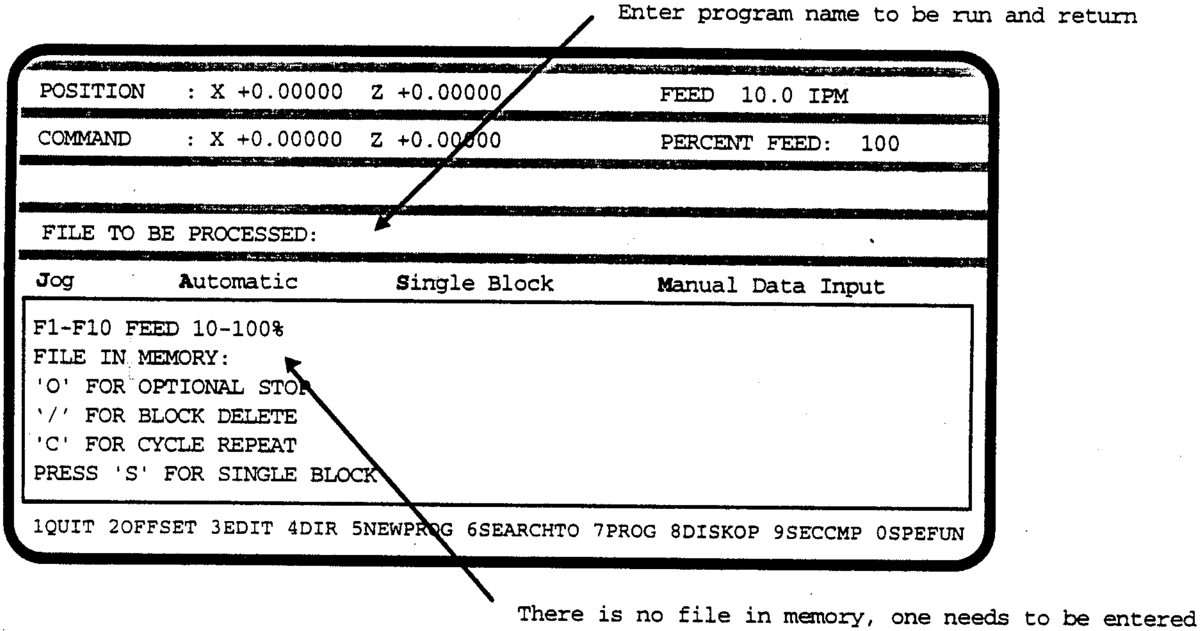
Pg Up Key
Coolant On/Off

Pg Dn key
Parts catcher In/Out

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

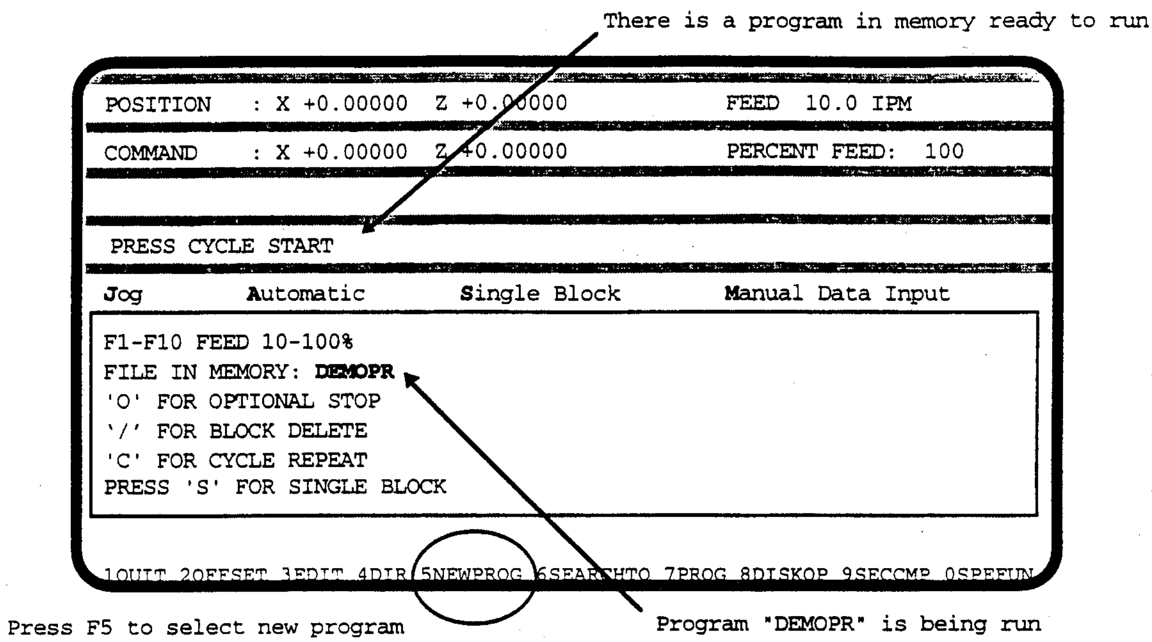
If you have a program saved on the program disk it's name can be entered now and the screen will show that this is the file now in memory. When the cycle start button is pressed the program will be executed.

---

2. Running a different existing program *(F5)*

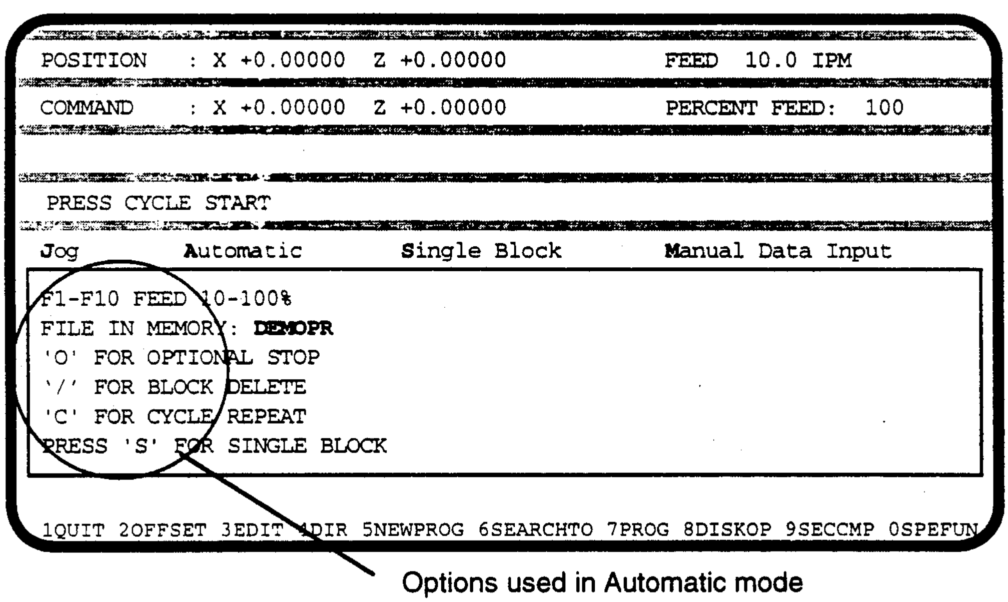
Once a program has been selected it stays in memory until it is changed. In the Automatic mode, F5 will delete the currently running program and ask for a new program name. You will notice that after F5 has been pressed the file in memory is blank. If you forget the exact name of the program that you 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



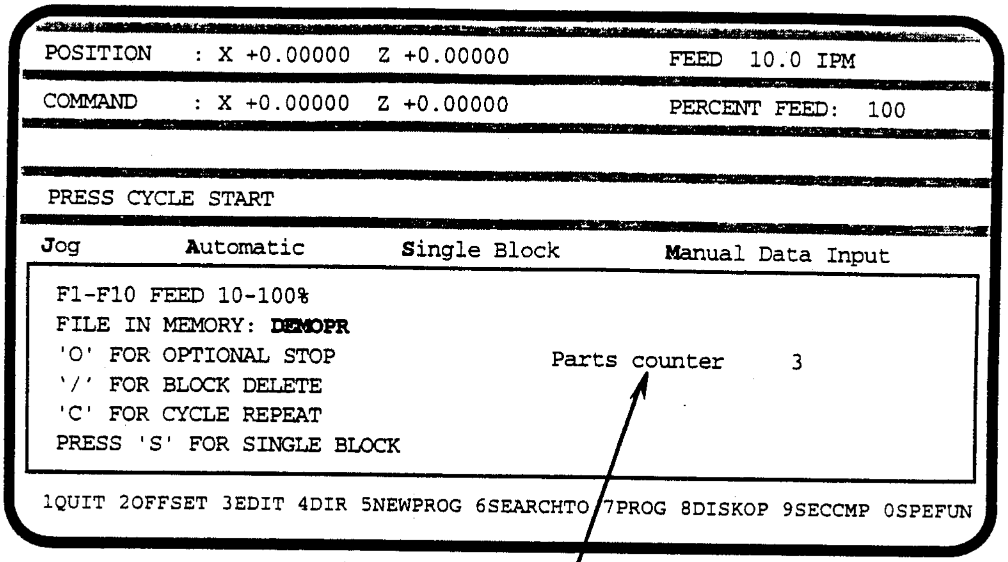
Program Run - Single step - "S"

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.

Parts counter - P



The parts counter is turned on in F10

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

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.

/ - Block Delete

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

PRESS CYCLE START
Jog         Automatic         Single Block         Manual Data Input
  F1-F10 FEED 10-100%
   FILE IN MEMORY: DEMOPR
   'O' FOR OPTIONAL STOP
   '/' FOR BLOCK DELETE              BLOCK DELETE ACTIVE
   'C' FOR CYCLE REPEAT
   PRESS 'S' FOR SINGLE BLOCK

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

The block delete is used to by pass lines of the program. Put the forward slash "/" at the begining of a line. When you want to skip the line just press the"/" 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 editor 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.

Function Keys

On the left side 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.

F2 *Tool offset screen, used to modify tool offsets*

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

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

OFFSET NUMBER:
Press Esc to exit offset adjustment screen

In the example above you see the offset screen with three tools being used. When it is necessary to correct a tool offset, enter the amount of change that is requiered. As an example, we will assume that tool 2 in the above example is a turning tool and is cutting a part .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.
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
1C	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
	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
1C	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.

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

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

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 5-1/4". Once you have reviewed the programs available press "Esc" to return to the Automatic mode screen.

F5 *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.

F6 *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
'O' FOR OPTIONAL STOP
'C' FOR CYCLE REPEAT
'/' FOR BLOCK DELETE
PRESS 'S' FOR SINGLE BLOCK

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

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.

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

F9 *Secondary tool offset screen, used to modify secondary tool offsets*

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

1	X: +0.00000	Z: +0.00000	R:0.0000	**17**	X: +0.00000	Z: +0.00000	R:0.0000
2	X: +0.00000	Z: +0.00000	R:0.0000	**18**	X: +0.00000	Z: +0.00000	R:0.0000
3	X: +0.00000	Z: +0.00000	R:0.0000	**19**	X: +0.00000	Z: +0.00000	R:0.0000
4	X: +0.00000	Z: +0.00000	R:0.0000	**20**	X: +0.00000	Z: +0.00000	R:0.0000
5	X: +0.00000	Z: +0.00000	R:0.0000	**21**	X: +0.00000	Z: +0.00000	R:0.0000
6	X: +0.00000	Z: +0.00000	R:0.0000	**22**	X: +0.00000	Z: +0.00000	R:0.0000
7	X: +0.00000	Z: +0.00000	R:0.0000	**23**	X: +0.00000	Z: +0.00000	R:0.0000
8	X: +0.00000	Z: +0.00000	R:0.0000	**24**	X: +0.00000	Z: +0.00000	R:0.0000
9	X: +0.00000	Z: +0.00000	R:0.0000	**25**	X: +0.00000	Z: +0.00000	R:0.0000
10	X: +0.00000	Z: +0.00000	R:0.0000	**26**	X: +0.00000	Z: +0.00000	R:0.0000
11	X: +0.00000	Z: +0.00000	R:0.0000	**27**	X: +0.00000	Z: +0.00000	R:0.0000
12	X: +0.00000	Z: +0.00000	R:0.0000	**28**	X: +0.00000	Z: +0.00000	R:0.0000
13	X: +0.00000	Z: +0.00000	R:0.0000	**29**	X: +0.00000	Z: +0.00000	R:0.0000
14	X: +0.00000	Z: +0.00000	R:0.0000	**30**	X: +0.00000	Z: +0.00000	R:0.0000
15	X: +0.00000	Z: +0.00000	R:0.0000	**31**	X: +0.00000	Z: +0.00000	R:0.0000
16	X: +0.00000	Z: +0.00000	R:0.0000	**32**	X: +0.00000	Z: +0.00000	R:0.0000

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

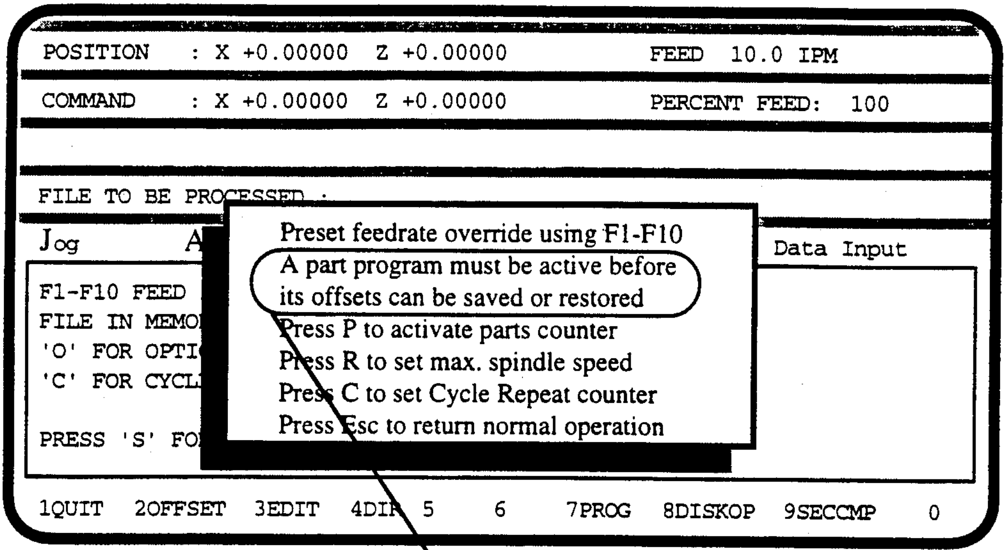
Changing a TNR compensation value:
When you put a value in the R offset table it writes over the old value. So if you have a number already in the register that you want to use and it is not the value needed, all you have to do is enter the correct value. As an example if you have a value of .032 in 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.

F10 *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 recalled. If you have a part program active the control will allow you to save and load the offsets from memory:

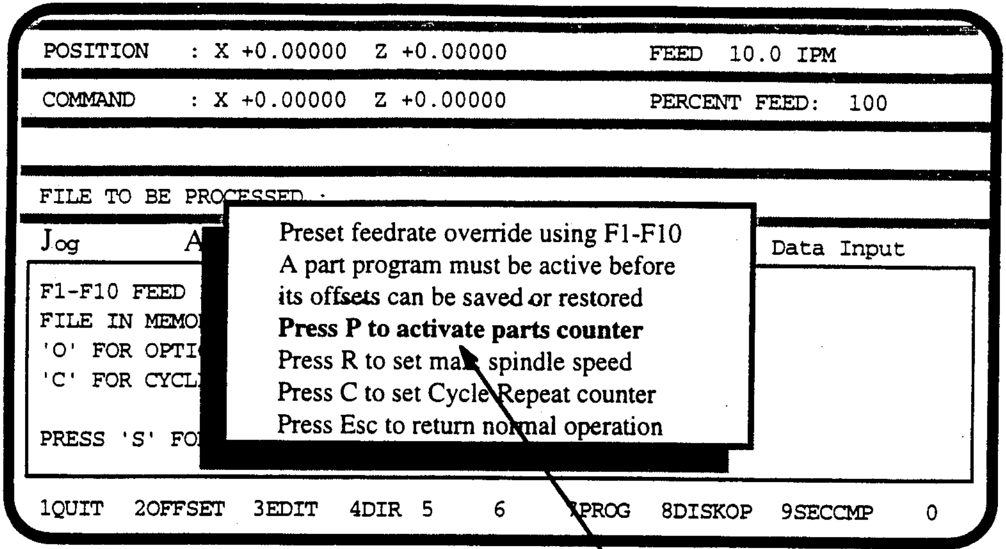
L - press L to load the offsets from memory
S - press S to save the offsets to the disk

• **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



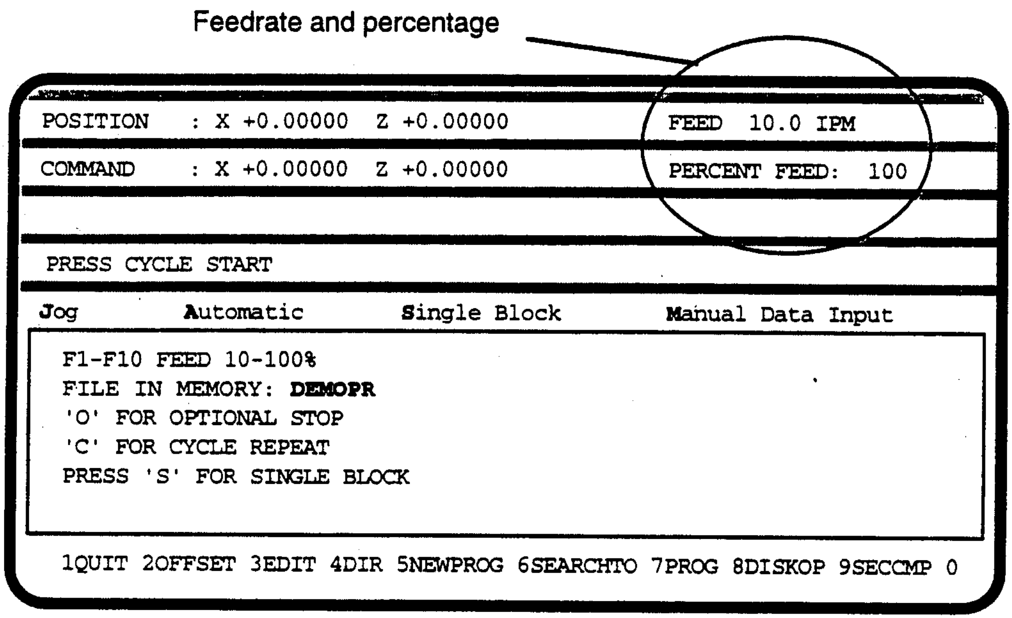
Press P now, it will turn
the counter on in the
Automatic mode

• **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 feedrates. 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.

Jeff Richlin
631 694 9400
jrichlin@gmail.com