Codes Honored by the OmniTurn control

(Sort by Code)

Codes Honored by the OmniTurn control

(Sort by Code)

Code	Usage	Description	Pages
M05	M05	Spindle off, stop	62,65,74
M08	M08	Coolant on	16,62,65
M09	M09	Coolant off	62
M12	M12	Collet clamp	62
M13	M13	Collet unclamp	62
M19	M19	Spindle Positioning (optional C-Axis only)	62,74
M25	M25	User assigned on	62
M26	M26	User assigned off	62
M30	M30	End of program - cancels all active ”M” functions	26,62,65
M31	M31	Cancels Cycle Repeat mode	62
M89	M89	Stop the spindle and lock it (optional: C-Axis only)	63
M91	M91	Wait for TB2-5 to be open circuit (optional: C-Axis only)	63
M92	M92	Wait for TB2-5 to be short to 0VDC (optional: C-Axis only)	63
M93	M93	Wait for TB2-7 to be open circuit (optional: C-Axis only)	63
M94	M94	Wait for TB2-7 + to be short to 0VDC (optional: C-Axis only)	63
M95	M95	Jump to subroutine 1 if TB2-9 is short to 0VDC (opt: C-Axis only)	63
M97	M97lnCnPn	Jump to subroutine, conditional (optional: PLC option only)	63
M98	M98Pn	Jump to subroutine	63
M99	M99	End subroutine	63
CI	CInnn.nn	Incremental spindle angle (optional: C-Axis only)	74
CA	CAnnn.nn	Absolute spindle angle (optional: C--Axis only)	74
C	XnZnCn	Automatic chamfer at intersection	15,16,67
D	Dn	Secondary offsets, axis correction or TNR comp value	68-71
F	Fn	Feedrates, dwell	48,56
LS	LSn	Loop start	72
LF	LF	Loop finish	73
R	XnZnRn	Automatic radius at intersection	15,16
S	Sn	Spindle speed selection, SFM or RPM	60,65,66,74
T	Tn	Tool offset call command	9
/	/	Block delete	*Section 5.3*
}	}n	Begin subroutine	63

Codes Honored by the OmniTurn control

(Sort by Description)

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

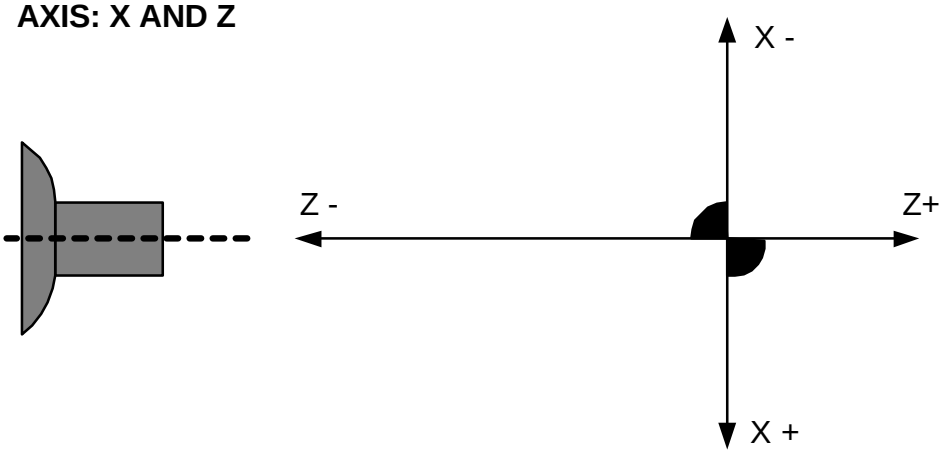
Codes Honored by the OmniTurn control

(Sort by Description)

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

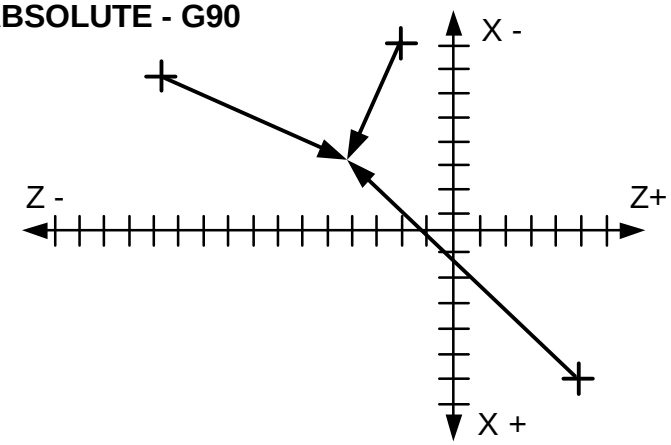
Nomenclature

AXIS: X AND Z



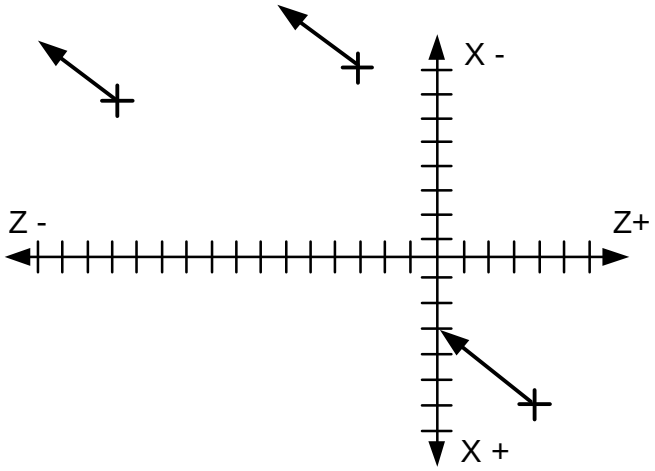
The slide has two axis's of travel.
X: Towards and away from you. Travel away from you is (-) minus. Towards you is plus (+).
Z: The slide that travels along the axis of the spindle. Going towards the spindle is (-) minus. Away from the spindle is (+) plus.

ABSOLUTE - G90



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

INCREMENTAL - G91



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

Programming Format

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

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

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

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

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

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

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

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

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

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

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

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

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

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

Programming Format

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

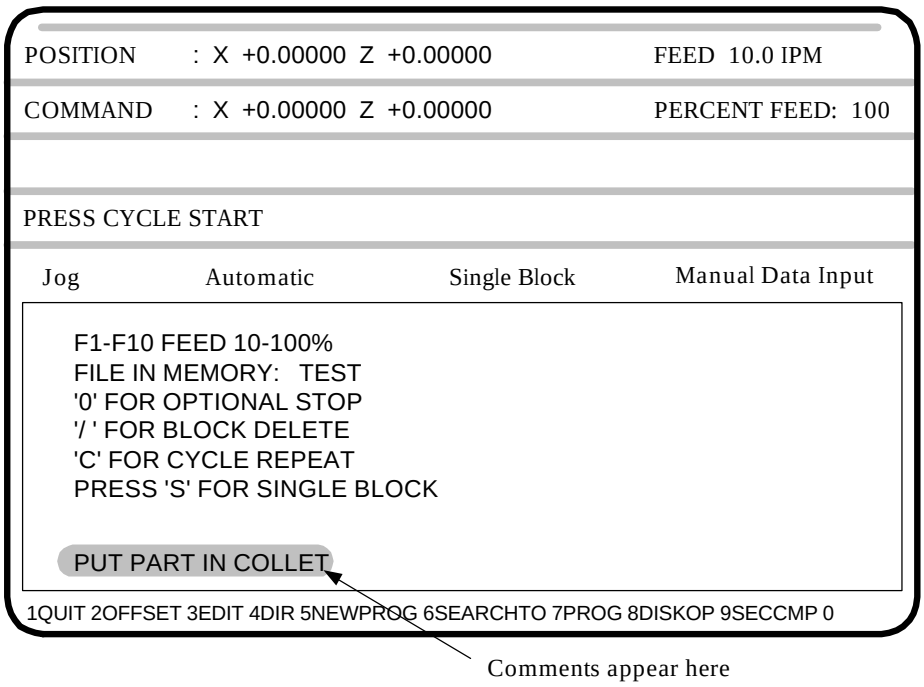
X.1Z-.4F7 This line of code has a feedrate of 7 IPM when in G94 (F300 max)

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

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

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

T1
X0Z2
M00 (PUT PART IN COLLET)



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

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

Creating a new program

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

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

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

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

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

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

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

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

Tool call statements

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

Tn
XnZn

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

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

T1.....Tool call for #1
X.25Z1 .....move the tool to a location of X = .25 and Z = I
Z.1 .....move the tool closer to the work, still in rapid mode

Linear moves X n Z n XnZn

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

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

Feed and Rapid moves are accomplished as follows:

Movement in X:

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

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

Movement in Z:

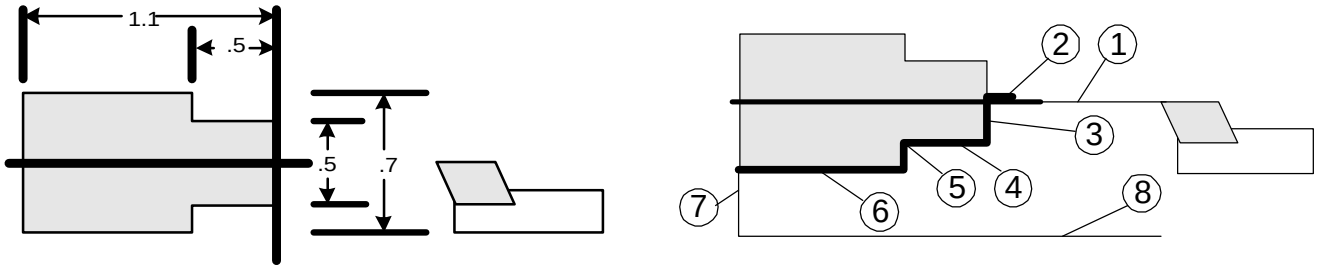
Zn	Single axis (Z) move
example: Z-.1	This will move the tool -.1 into the material

Z moves can be programmed directly off the print.

Movement in X and Z simultaneously

XnZn	Two axis linear move (X & Z)

Example for linear moves



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

	diameter mode:	**radius mode:**
	G90G94F300G72	G90G94F300G73
	M03S2000	M03S2000
	M08	M08
	T1	Tl
	X0 Z1	X0 Z1
(1)	Z.1	Z.1
(2)	Z0 G95f.003	Z0G95f.003
(3)	X.5	X.25
(4)	Z-.5	Z-.5
(5)	X.7	X.35
(6)	Z-1.1	Z-1.1
(7)	X 1	X . 5
(8)	Z1 G00	Z1G00
	M30	M30

Linear moves X n Z n XnZn

A note about feedrate modes:

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

G94 -Inch per minute mode

The G94 command will make all feedrate commands effective in inch/minute. If you have a feedrate command of F4 in this mode it will move the slide at 4"/minute. F.001 would be .001"/minute, Very Slow!!! This is a good way to rapid around, but it is not a good way to program feed (cutting) moves. If the spindle is not turned on the slide will still have the tool make the cut.

G95 -Inch per revolution mode

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

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

Example formats are:

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

Rapid travel (G00):

Rapid travel is accomplished by either:

Using G94 for rapid travels:

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

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

Note:

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

Note good tip:

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

Linear moves X n Z n XnZn

Using G00 for rapid travels

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

G90G94F300

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

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

Feedrate mode (G01):

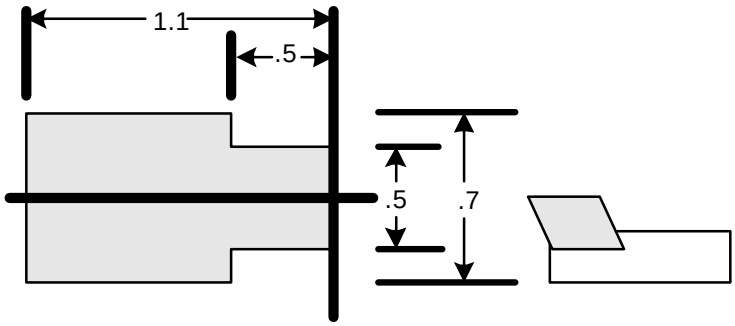
You can go from rapid travel mode to feedrate mode by either specifying a new feedrate and mode, IE: G94F300 for rapid travel, and G95F.n for feedrate. Or using G01 to revert to the last described feedrate mode in the program. This eliminates having to use the G95F.n over and over. The other advantage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

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

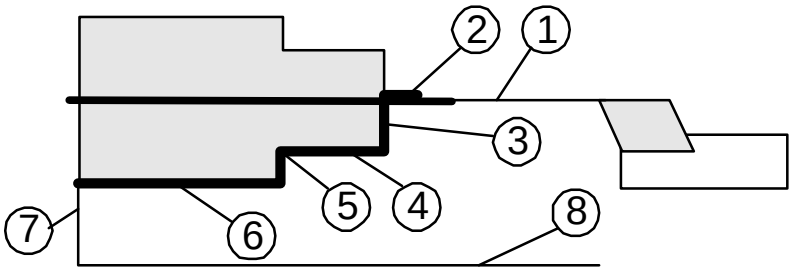
```
G94F300X0Z1
Z.1
G95F.003Z.005
X.5 G00Z.05
X0
G01Z0
X.5
G00Z1
```

Linear moves X n Z n XnZn

In the example below we will be finish turning a sample part with a left hand tool. The part dimensions are given in diameters. When the part is programmed these will have to be converted to radius moves.
Z zero is set at the face of the part, X zero is set at the center.



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



The same example in radius mode

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

The same example with G00 and G01

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

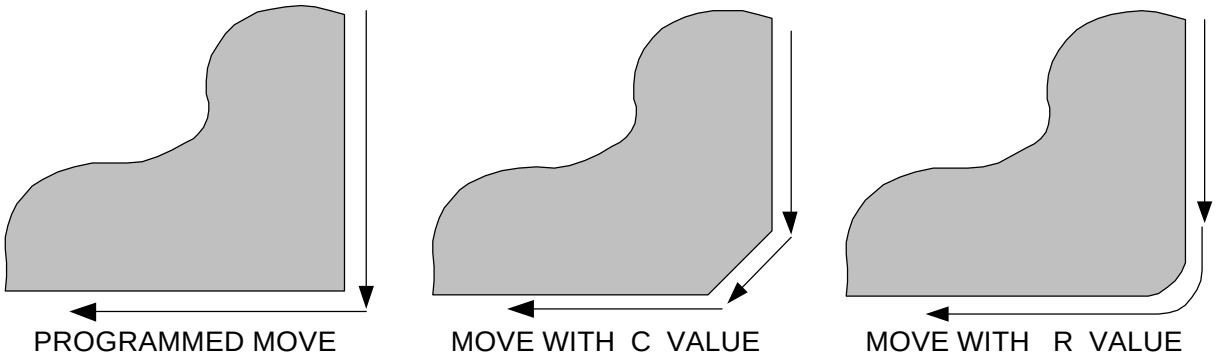
(7)	G00 X.5	Rapid feedrate, Pull the tool away in X
(8)	Z 1	Withdraw the tool in Z
	M30	End of program command

The same example written in diameter mode (G72)

	G90 G72 G94 F300	Establish Abs. positioning, Feed IPM @ 300”/minute, Radius mode
	T1 (turn tool)	Call Tool #1 offset
	X0 Z1	moves the tool to a safe start location
(1)	Z.1	Rapid to .1” from the face
(2)	G95 F.003 Z0	Set feedrate to IPR @ .003”/ rev, Feed to the face
(3)	X.5	Feed out to .25” radius (.5” diameter)
(4)	Z-.5	Feed to Z -.5
(5)	X.7	Feed to .7” diameter
(6)	Z-1.1	The moves to generate this tool path could be as shown below.
(7)	G94 F300 X1	Set feedrate to IPM @ 300”/minute, Pull the tool away in X
(8)	Z 1	Withdraw the tool in Z
	M30	End of program command

Automatic corner radiusing (R) and chamfering(C)

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



Format

X nZnRn -ZnRn -X nRn

X nZnCn -ZnCn -X nCn

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

RULES

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

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

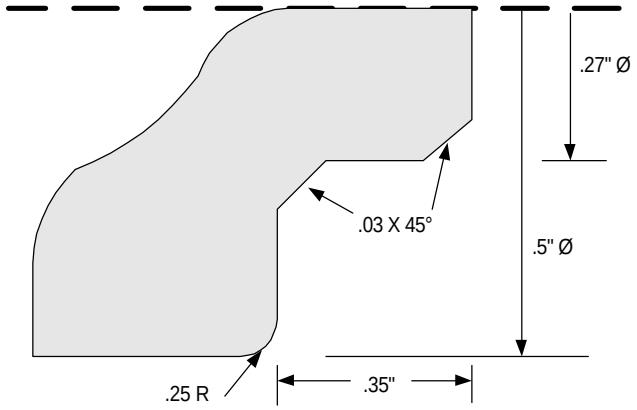
Running programs using C or R

When you use the automatic corner radius or chamfer commands the OmniTurn creates a number of moves to generate what you want. If you look at the command line while you run a program you will notice lines of code that you did write. In the single block mode you can see arc (G02 or G03) commands. This is normal. When you leave the editor the OmniTurn automatically recreates the new moves. The program is also recreated whenever you change the secondary offset table.

Automatic corner radiusing (R) and chamfering(C)

Example

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



G90G72G94F300	G72 Program is in diameter mode
M03S2500	Turn the spindle on at 2500 rpm
M08	Turn the coolant on
T1(LH turn tool)	
X0Z1	
Z.05	
G95F.001	
G41	Turn on LH cutter compensation
Z0D1	Use the radius value found in D1 for compensation
X.27**C.03**	Set chamfer amount
Z-.35**C.03**	Set chamfer amount
X.5**R.025**	Set radius amount
Z-.45	
X.54	Move off the part more than the compensation value
G00Z1	
G40	Turn cutter compensation off
M30	

Arc statements G02 and G03

The arcs G02 and G03 are one shot commands. They are used one time and then turned off.

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

G02 Xn Zn In Kn G03 Xn Zn In Kn

or

G02 Xn Zn Rn G03 Xn Zn Rn

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

Radius mode G73

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

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

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

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

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

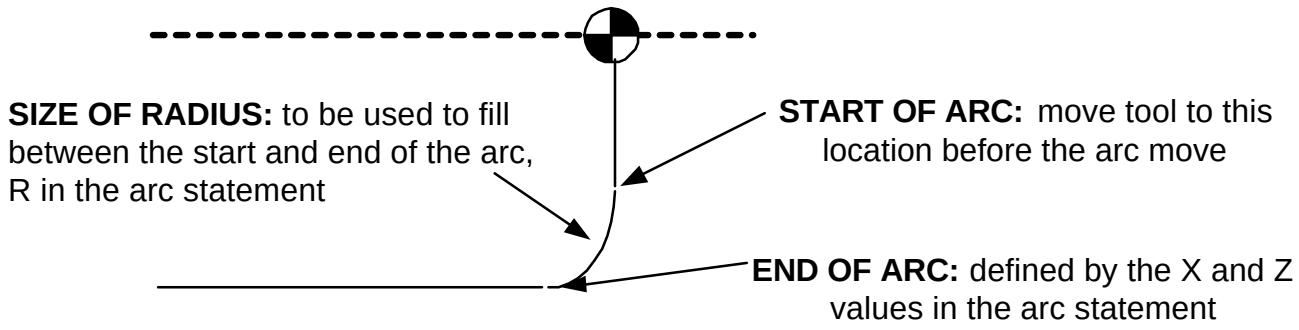
X the distance from the start of the arc to the end in X
Z the distance from the start of the arc to the end in Z
I the distance from the start of the arc to the center in X
K the distance from the start of the arc to the center in Z
R The length of the radius to be used to connect the start and end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

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

Arc statements G02 and G03

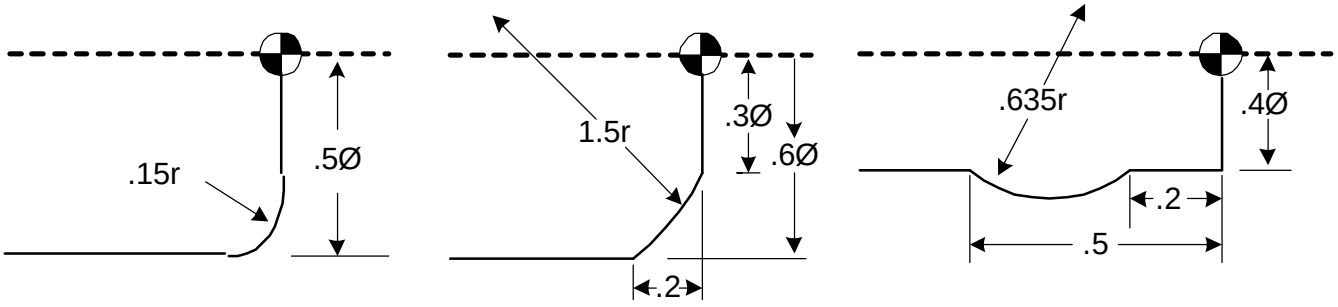
Using R version:

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



Examples of arc statements using R

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



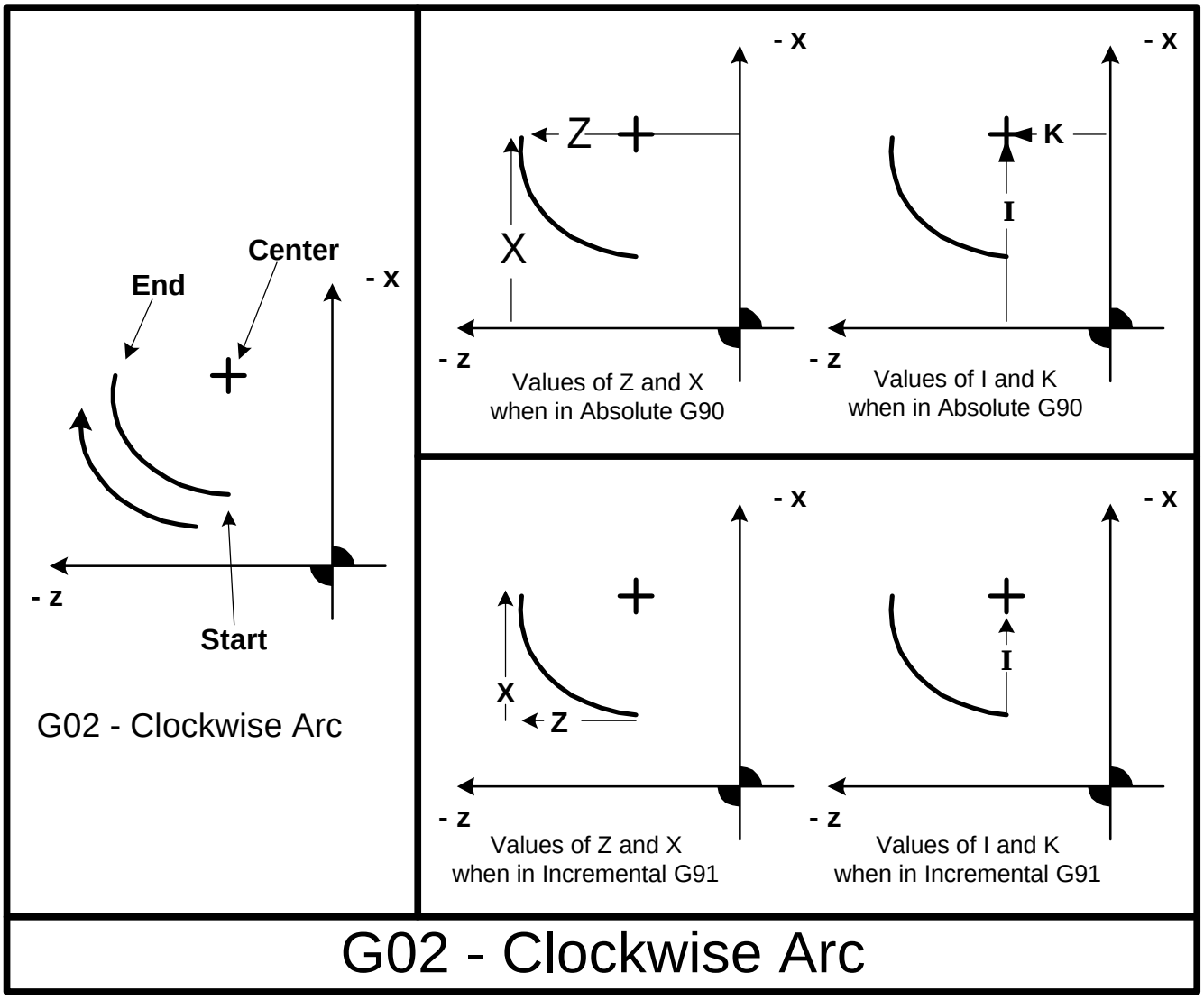

```
G90G94F300G73
M03S2500
M08
TI
X0Z1
Z.1
G95F.003Z0
X.1
G02X.25Z-.15R.15
Z-.3
X.275
G00Z1
M30
```

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

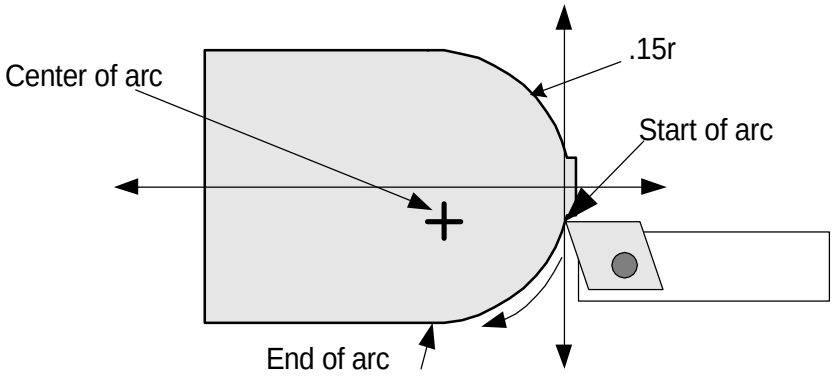
```
G90G94F300G73
M03S2500
M08
T1
X0Z1
Z.1
G95F.003Z0
X.2
Z-.2
G02X.2Z-.5R.635
Z-.7
G00X.3
Z1
M30
```

Arc statements G02 and G03

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



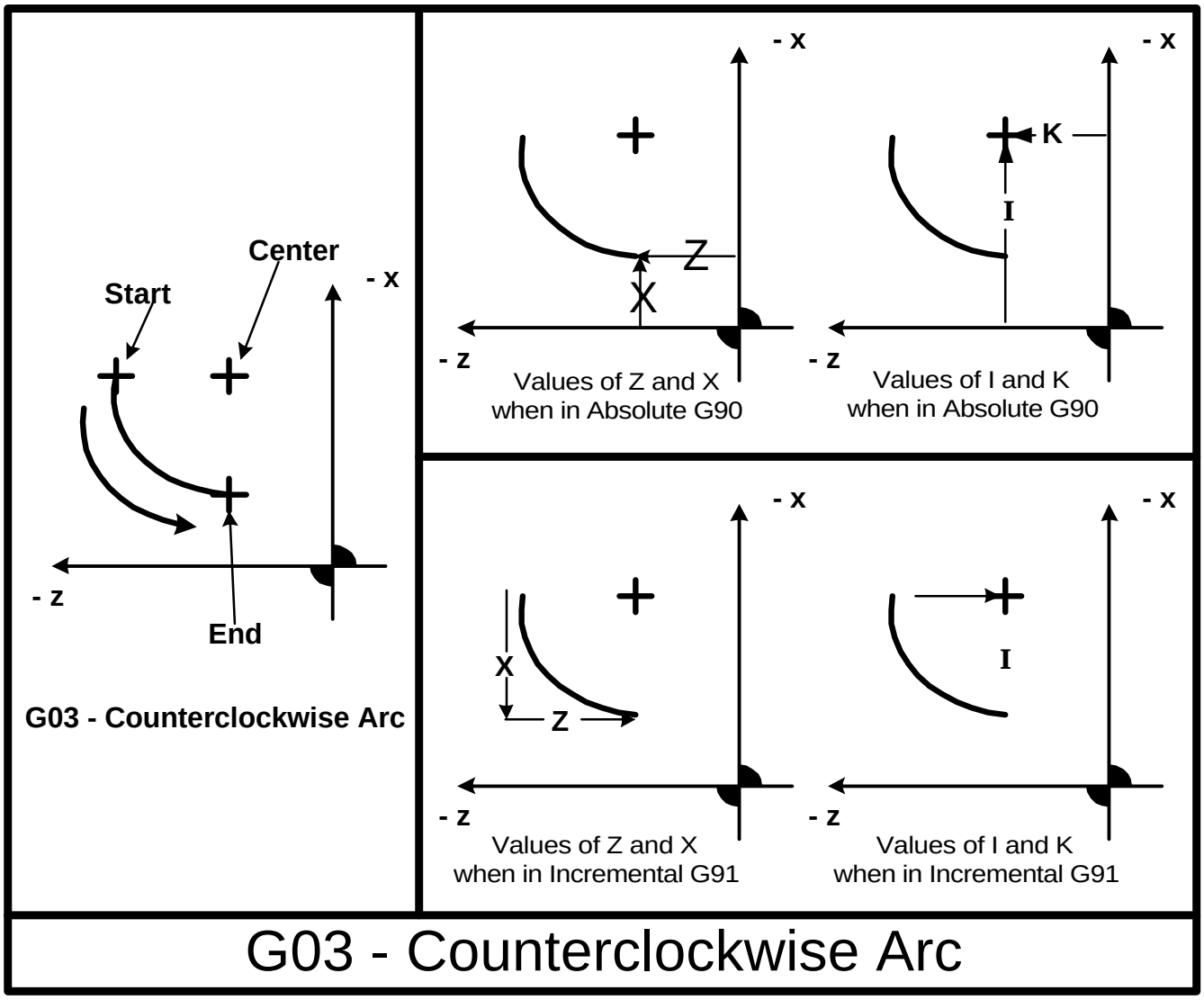
Using G02 - Clockwise Arc



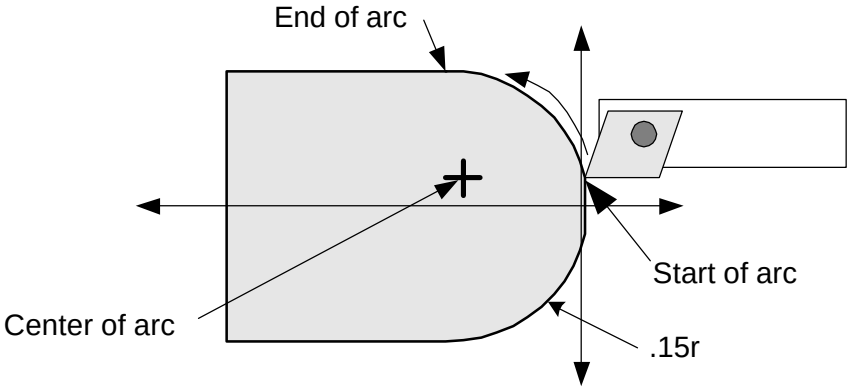
G90 G94 G73
T1F300
X0 Z.1
X.1 Z0
G02 X.25 Z-.15 I.1 K-.15
Z-.5
or the arc statment could be:
G02 X.25 Z-.15 R.15

Arc statements G02 and G03

G73 - Radius Mode



Using G03 - Counterclockwise Arc while in G73 - radius mode



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

Arc statements G02 and G03

Diameter mode

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

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

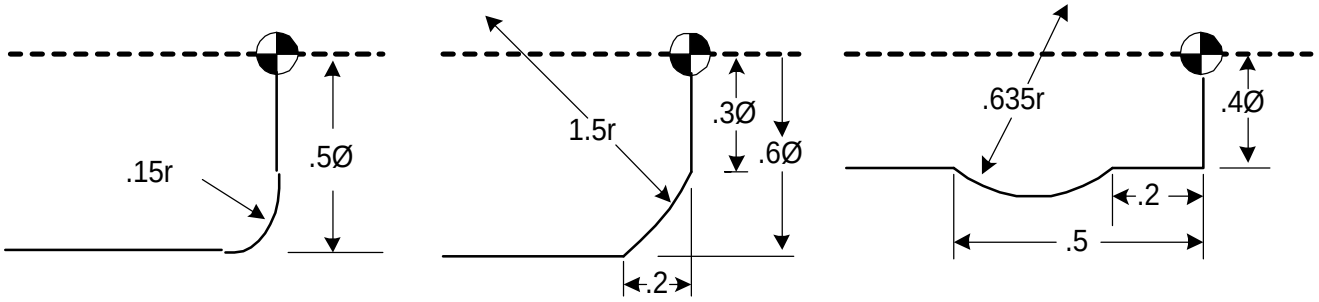
Note: -Using arc statements in the diameter mode (G72) be sure you are in absolute (G90)
-Position tool at start point before using arc move
-This format follows the Fanuc format more-closely than previously

G02XnZnInKn • G03XnZnInKn

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

Examples of arc statements using R in diameter mode:

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



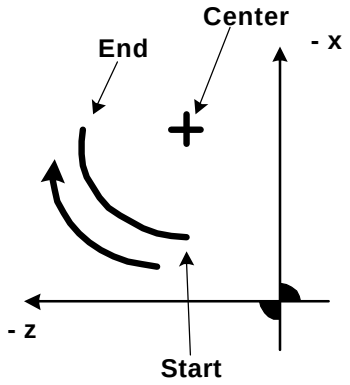

```
G90G94F300G72
M03S2500
M08
T1
X0Z1
Z.1
G95F.003Z0
X.2
G02X.5Z-.15R.15
Z-.3
X.55
G00ZI
M30
```

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

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

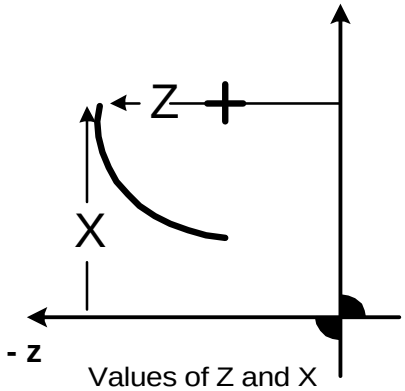
Arc statements G02 and G03

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



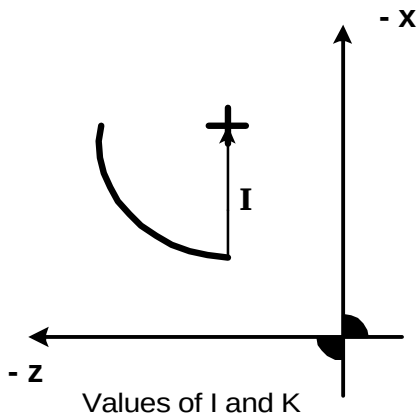
G02 - Clockwise Arc

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



Values of Z and X
when in Absolute G90

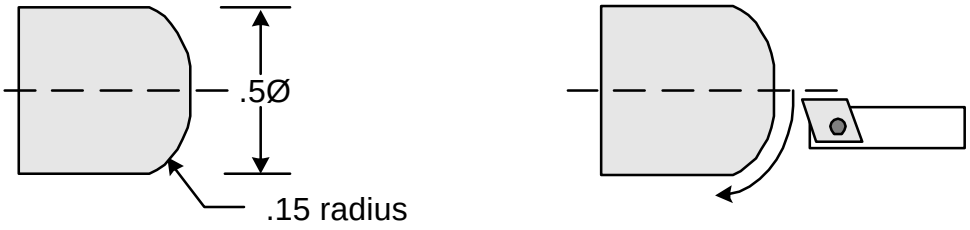
Arc center (In Kn): This is different. With diameter programming the arc center is defined as the incremental distance from the arc start to the center.



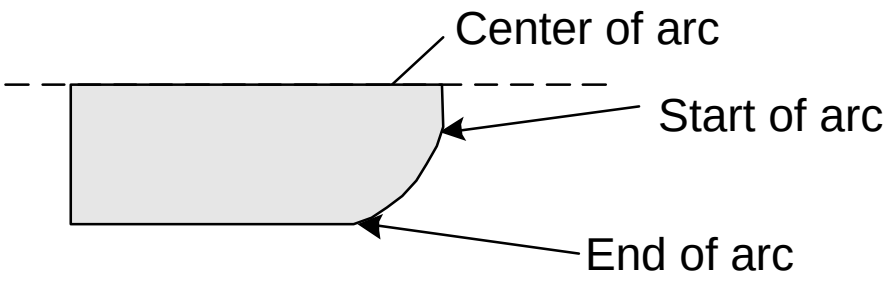
Values of I and K
Incrimental from start

Arc statements G02 and G03

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

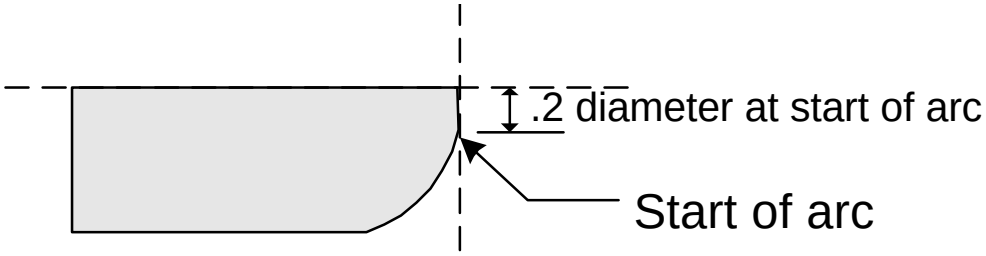


For this example first we show the three important locations that must be defined to write the arc statement:

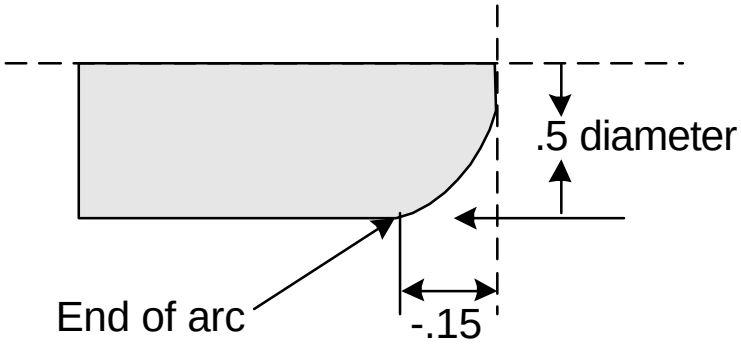


The starting point of the arc is where X = .2 (diameter value) and Z = 0. So these values will be used to write the position move before the arc statement:

X.2Z0

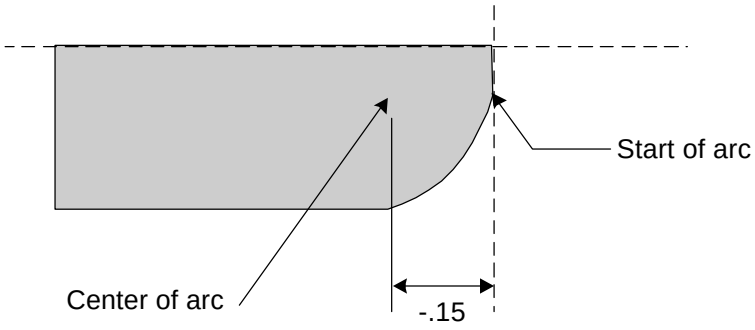


The end of the arc is defined from the absolute zero for the part. As shown on the following figure X = .5 and Z = -.15. These are used for the X and Z values in the G02



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

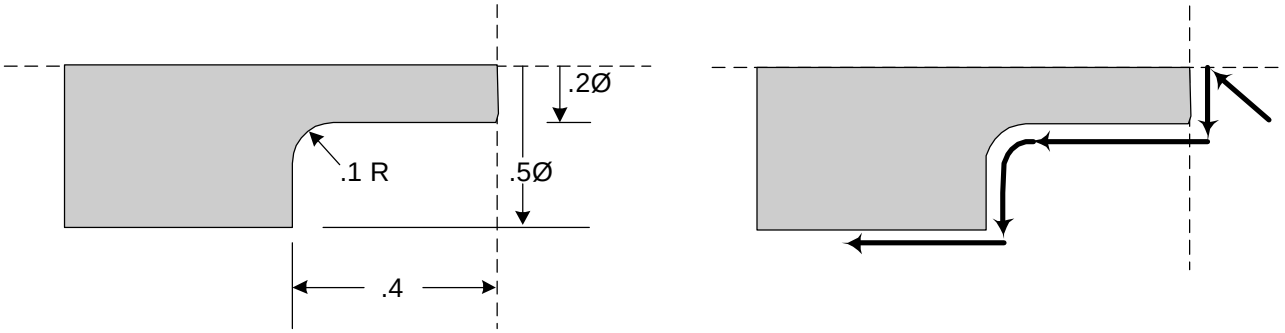
Arc statements G02 and G03



So the program lines could be:

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

For the next example we will show a G03:



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

Dwell - G04

The dwell statement format is:

G04Fn

Note:

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

Example

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

G10 Work shift

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

-Machining multiple parts off a single shootout of a bar.

-Shifting a program away from the spindle the first time it is run

G10XnZn

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

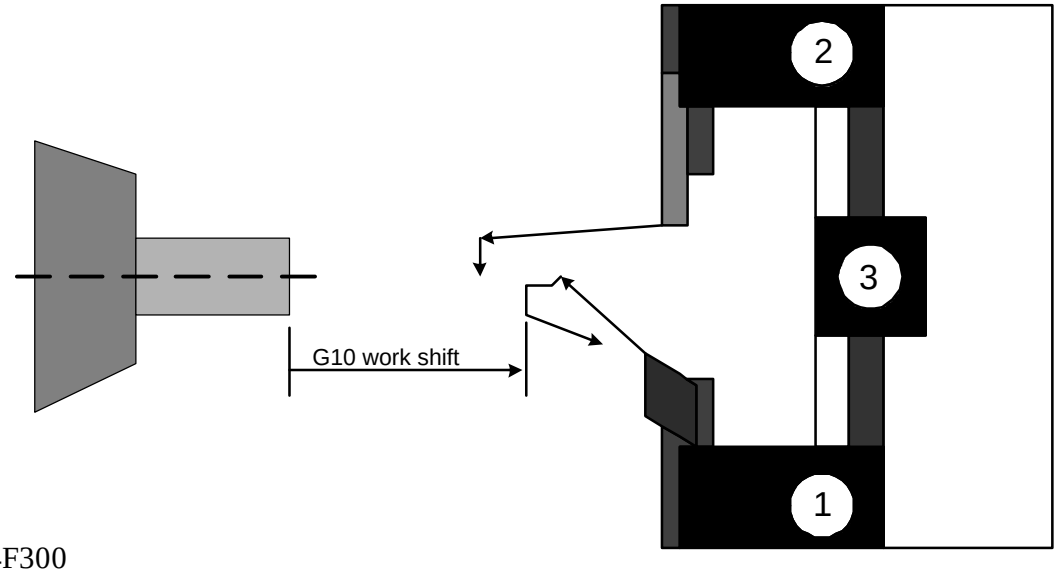
Note: The shift is executed on tool calls! Use the GIO command before tool calls otherwise there will be no effect.

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

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

Example of shifting a program for test running

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



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

G10 Work shift

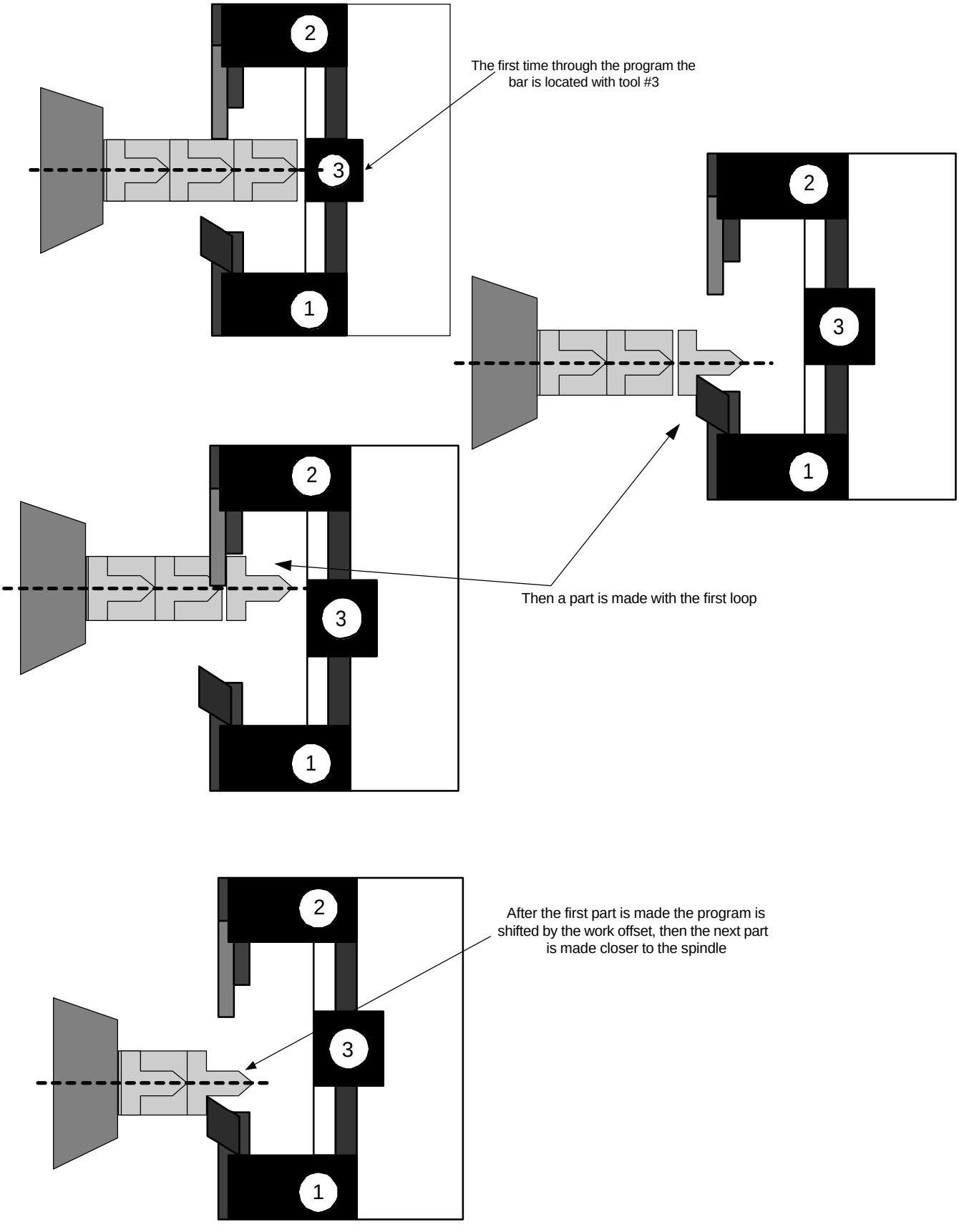
```
G96S150
G76S500
G95F.001 X.005
G94F300Z1
G97
M30
```

Example of shifting a program for bar work

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

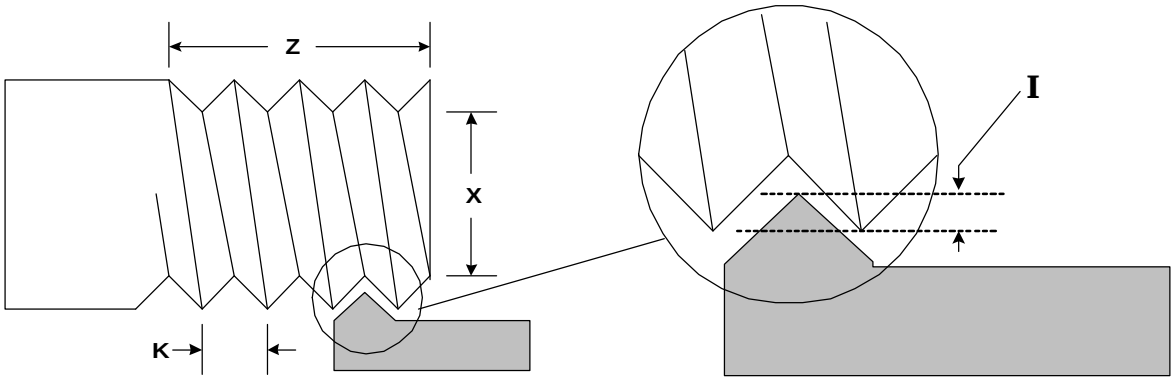
```
G90G72G94F300
M03S2000
T3(WORK STOP)
XOZ1
Z-1
M13
Z.025F50
M12
Z 1 G94F300
LS3                 The loop starts here and will repeat 3 times. Notice it is
T 1(LH TURN TOOL)   before a tool call.
X0Z.2
S2000
Z0G95F.01
X.1 Z-.05F.002
Z-.15
X.2
G94F300Z1
T2(PART OFF TOOL)
X0Z 1
X-.4
Z-.35
G96SI50
G76S500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

G10 Work shift



G33 Threading

The format is: G33XnZnInKnAnCnPO



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

Notes:

Diameter or radius mode

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

Starting position in Z

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

Starting position in X

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

Depth of each pass: I

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

G33 Threading

Retraction position between passes:

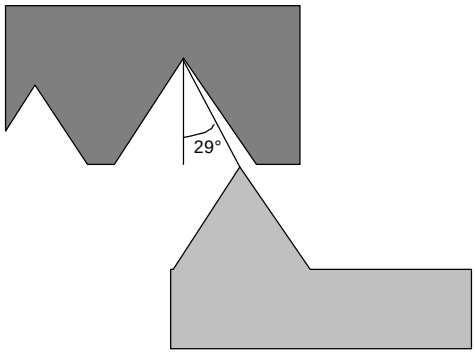
The tool will back away from the starting position plus 3 times the amount of I. Even as the tool gets deeper into the material it will always retract to the same point.

Pullout position in Z when using P option:

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

Angle infeed C option

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



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

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

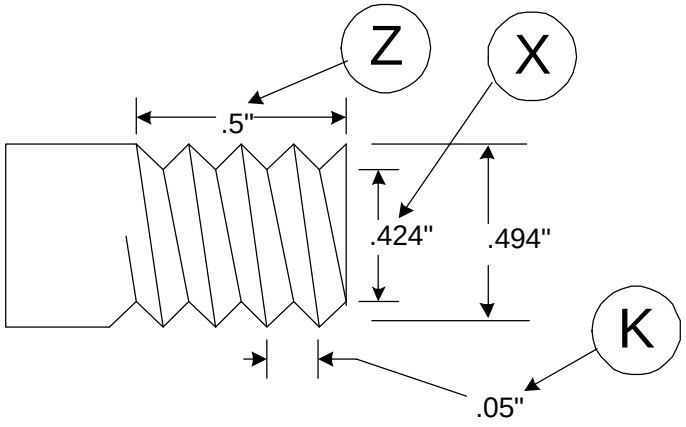
End of cycle position:

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

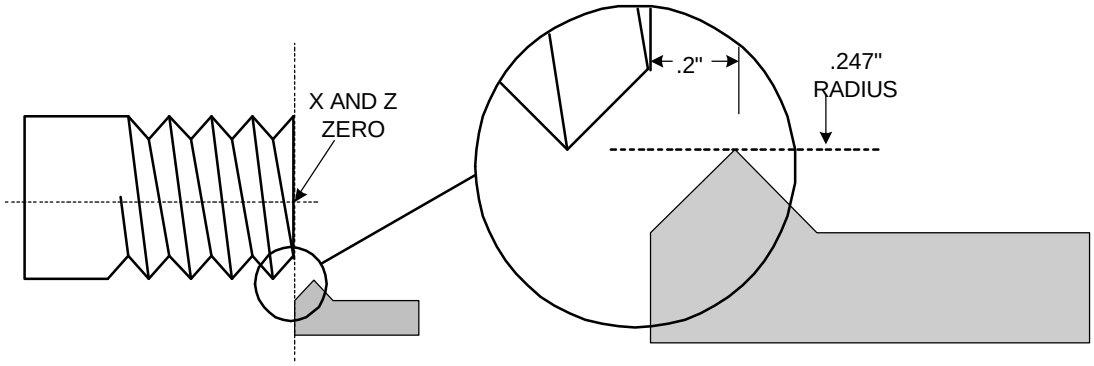
G33 Threading

Threading example

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



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



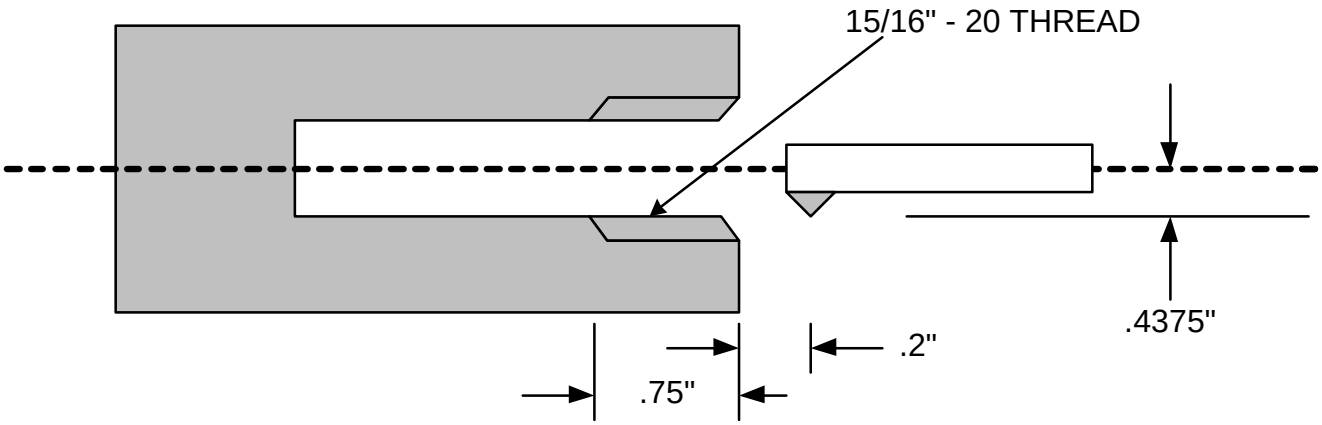
The locations shown above are given in radiuses. The starting location in X is the major diameter, less the amount of the first pass. In this case the major diameter is .494”, half of this is .247”. Less .O1 to be taken for the first pass: .237

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

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

G33 Threading

Internal threading example



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

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

Threading, Single Pass

The format is: G33XnZnInKnAnCnPO

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

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

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

This is useful for:

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

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

G33 Threading

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

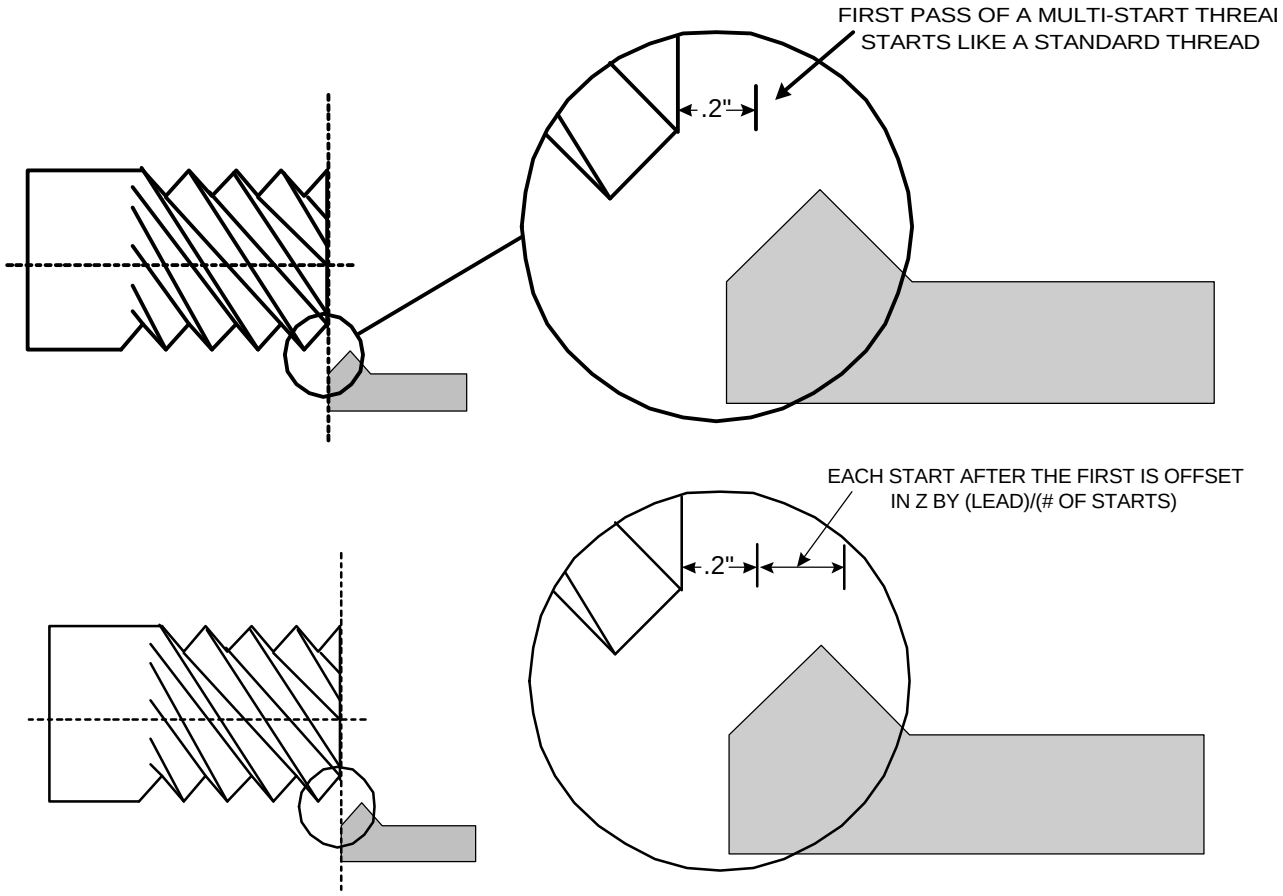
T4F300	Call the threading tool and position it at tool offset
X0Z.2	The offset puts the tool at the part center, .2" from the face in Z
X.237	Positions the tool at the starting radius to cut the thread
G33X.212**Z**-.5**I**.008**K**.05**PCO**	Cut a single pass thread

G33 Threading

Threading, Multi Start

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

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



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

Code for the above example:

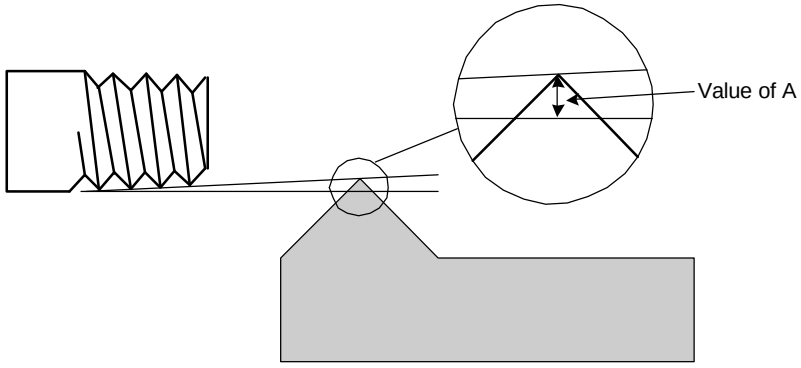
G90G94 F300
T4
X0Z.2
X.247
G33X.212Z-.5I.01K.15P
Z.25
G33X.212Z-.5I.01K.15P
Z.3
G33X.212Z-.5I.01K.15P

G33 Threading

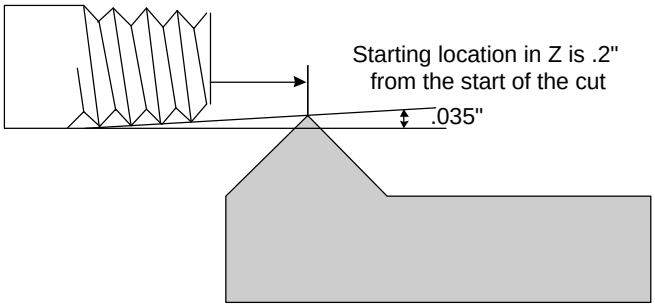
Threading, Tapered

G33XnZnInKnAnP

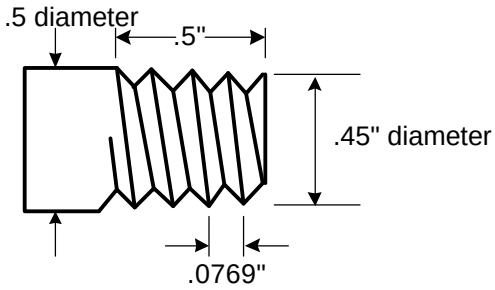
Tapered threading is done with the G33 command that includes an ”A”. This is the amount traveled in X over the distance traveled in Z.



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



Example



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

G33 Threading

Extra Coarse Feeds (IPR)

G35 -IPR feeds up to 1”

G35F2 -IPR feed up to 2”

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

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

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

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

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

Resolution limits are imposed with the use of these codes:

.00005” resolution in normal feed mode
.00025” resolution in G35 mode
.0005” resolution in G35F2 mode

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

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

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

TOOL NOSE RADIUS COMPENSATION G41,G42, G40

Tool nose radius compensation

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

Format

Right Compensation	G42
Left Compensation	G41
Cancel Compensation	G40

Compensation Value Location Xn.nnnnZn.nnnnDn

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

Sequence

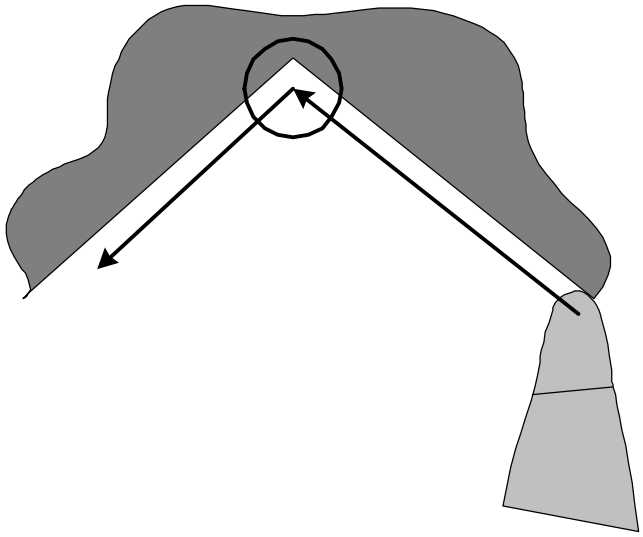
G42	Turn compensation on
XnZnDn	Move with secondary offset radius value used to turn on comp
XnZn	Move used to turn off compensation
G40	Turn compensation off

Rules

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

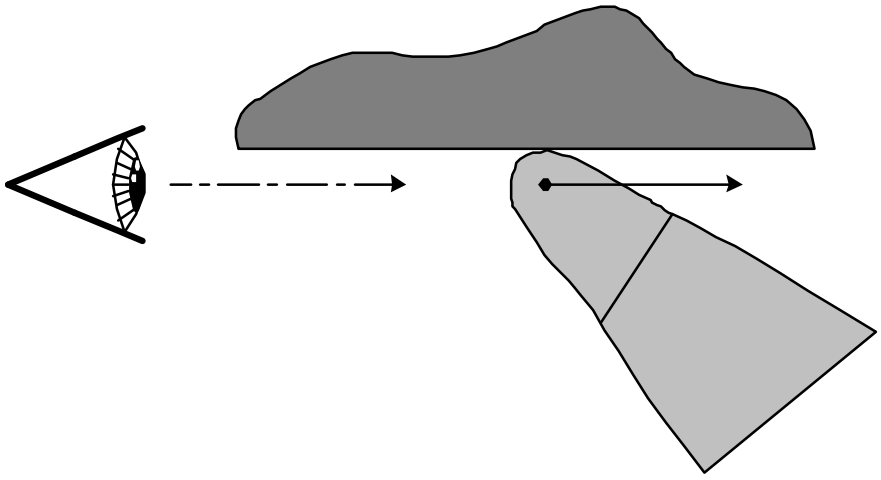
TOOL NOSE RADIUS COMPENSATION G41,G42, G40

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



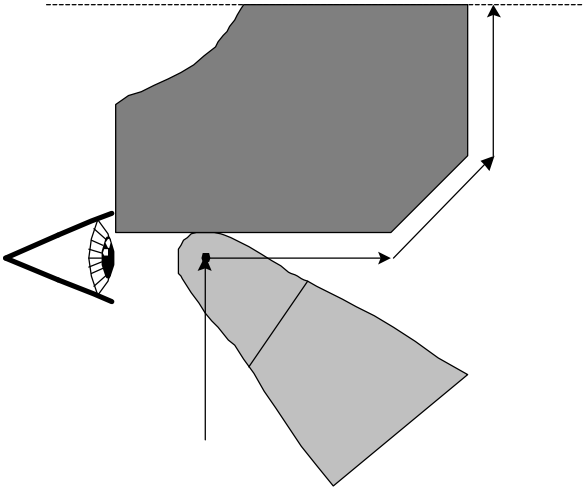
When using the TNR compensation the tool path gets shifted off the finished size. This does not matter if the tool being used to take a finish pass is different than the roughing tool. The tool is shifted in the setup to give a correct finished size. If the same tool is used to do the rough and finish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

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

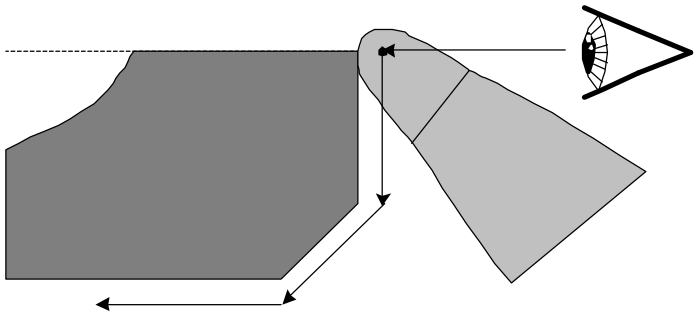


TOOL NOSE RADIUS COMPENSATION G41,G42, G40

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



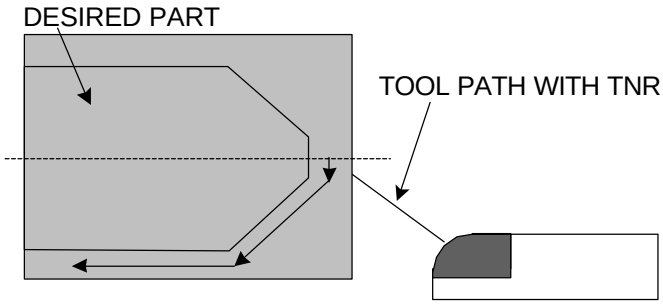
RIGHT HANDED COMPENSATION
G42

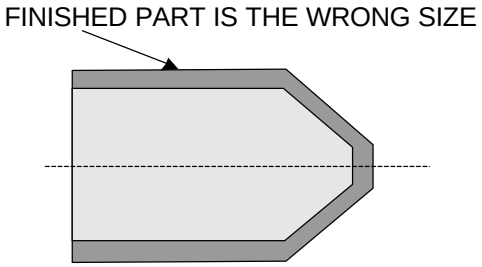


LEFT HANDED COMPENSATION
G41

Shifting the TNR compensation

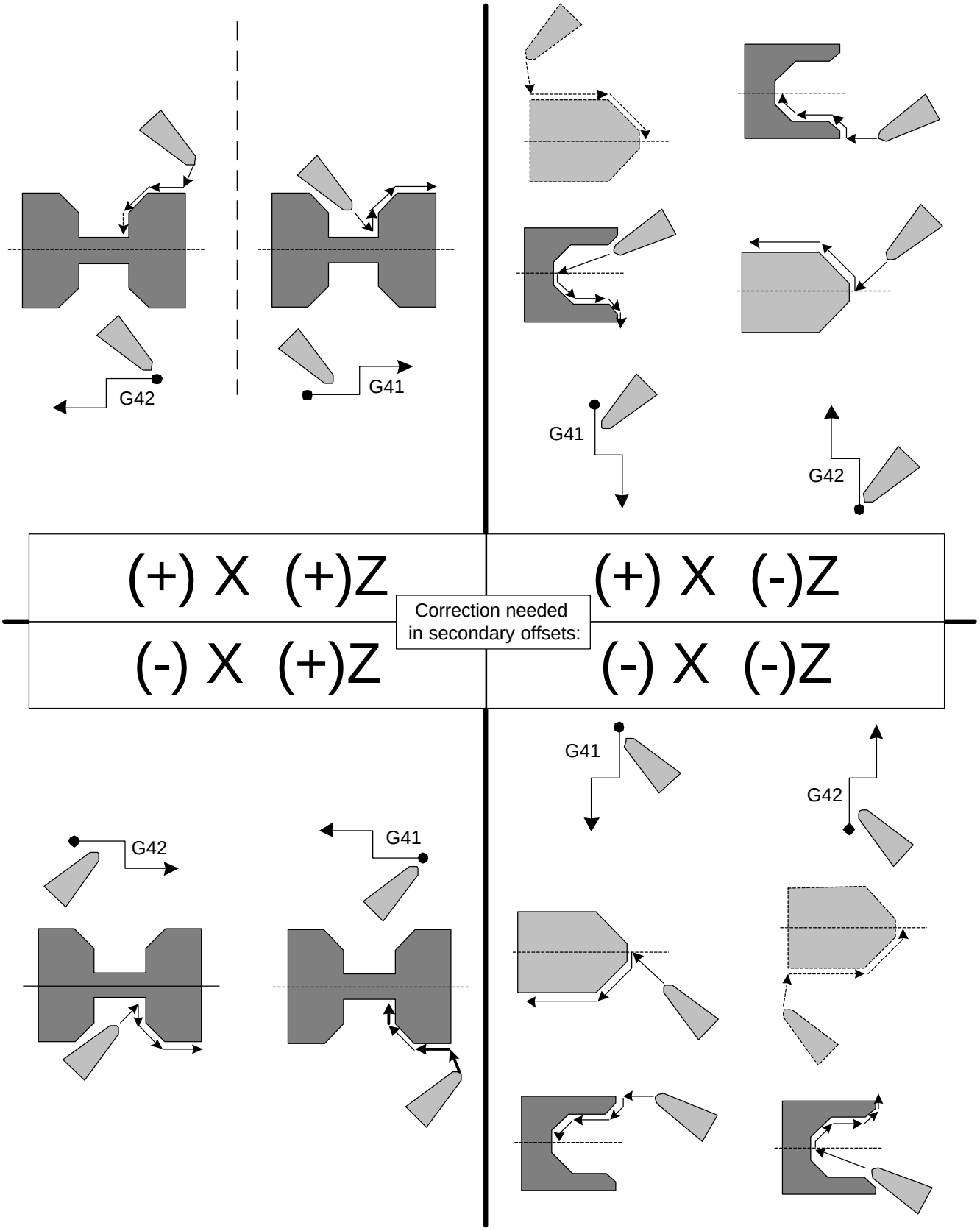
The direction of the correction will depend on the direction of the tool path and desired TNR compensation.





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

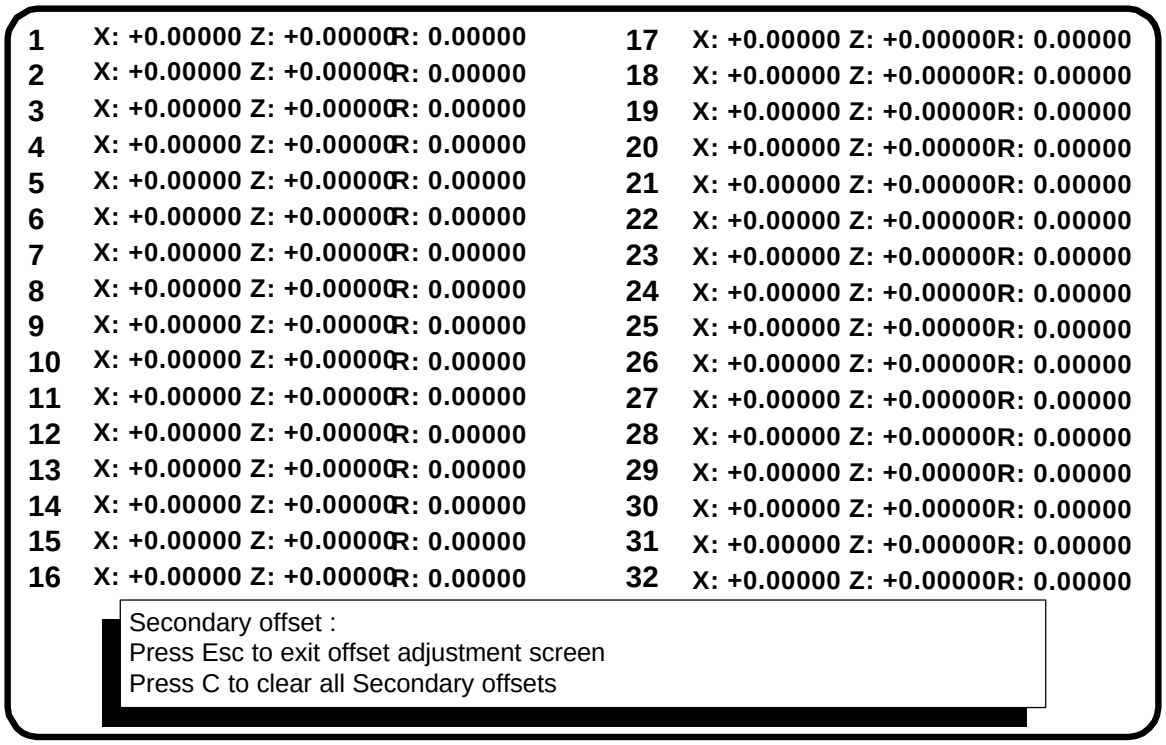
TOOL NOSE RADIUS COMPENSATION G41,G42, G40



TOOL NOSE RADIUS COMPENSATION G41,G42, G40

Setting the TNR value:

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



First: Select a secondary offset number

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

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

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

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

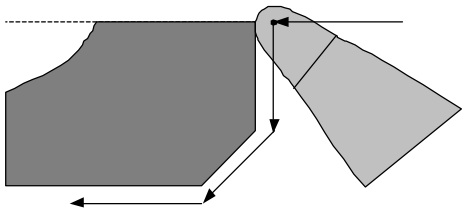
Changing a compensation value:

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

TOOL NOSE RADIUS COMPENSATION G41,G42, G40

Worked examples

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



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

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

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

G40	Turn off the TNR compensation on the Z2 move

M30

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

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

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

TOOL NOSE RADIUS COMPENSATION G41,G42, G40

Running a program that used Tool Nose Radius Compensation

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

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

G70 - G71 Inch - Metric Modes

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

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

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

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

G72 - G73 Diameter and Radius Programming

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

G72 Diameter Programming Mode

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

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

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

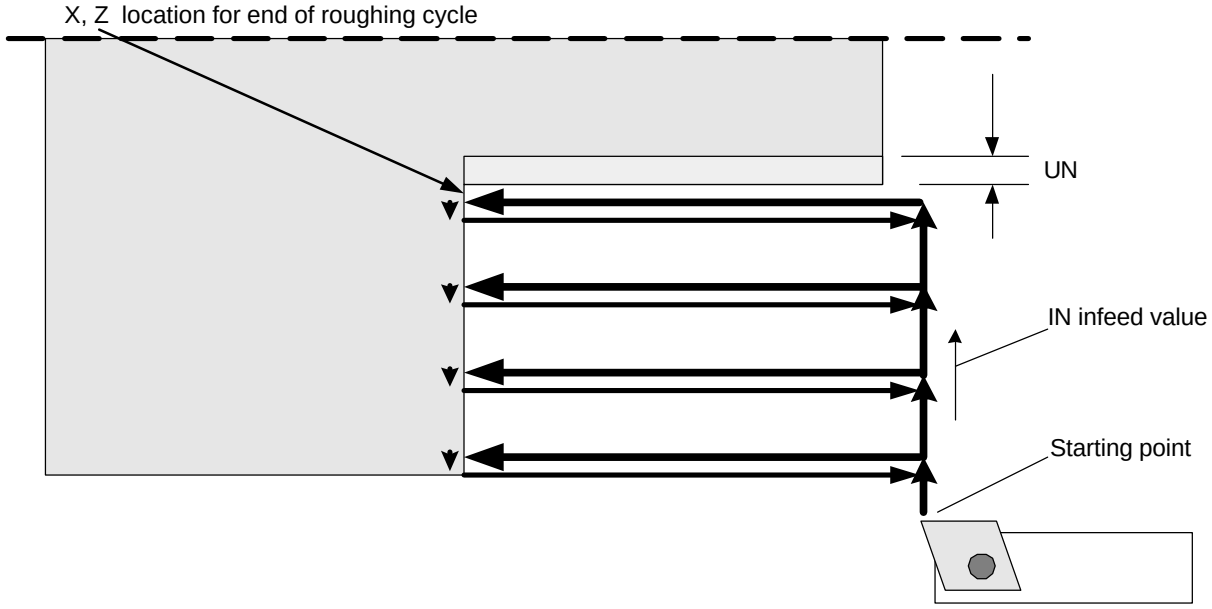
```
T1
G72X.5Z.1
```

G74 - Box Roughing Cycle

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

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.

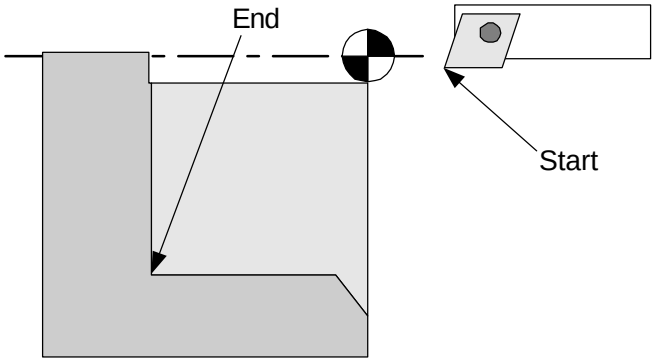
The feedrate is IPM (G94) or IPR (G95), depending on the mode when the cycle is started.

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

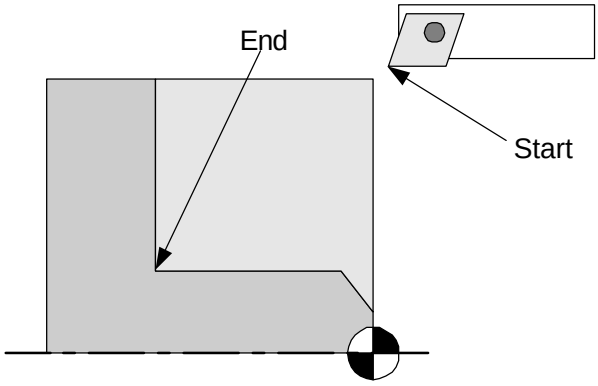
The return passes are at a fixed clearance distance (.02") from the last cutting pass.

G74 - Box Roughing Cycle continued

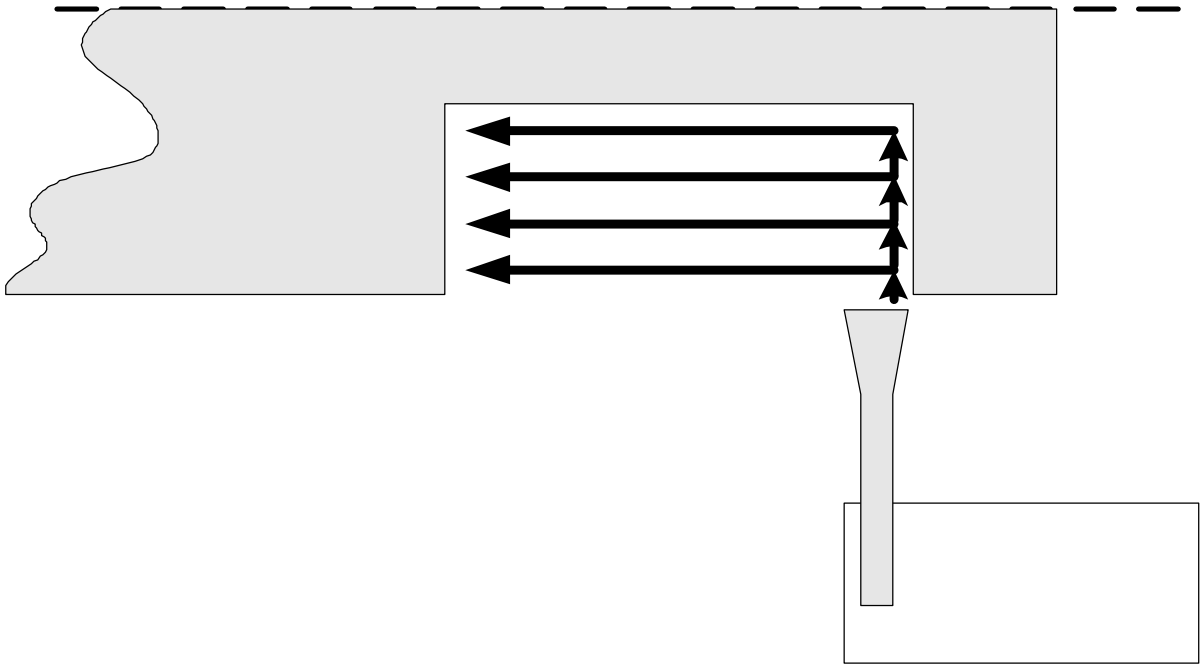
The G74 cycle can be used for either internal or external removal. It can also be used from the front (x+) or back (x-) of the part.



Internal example



External example from the back



Example behind a shoulder

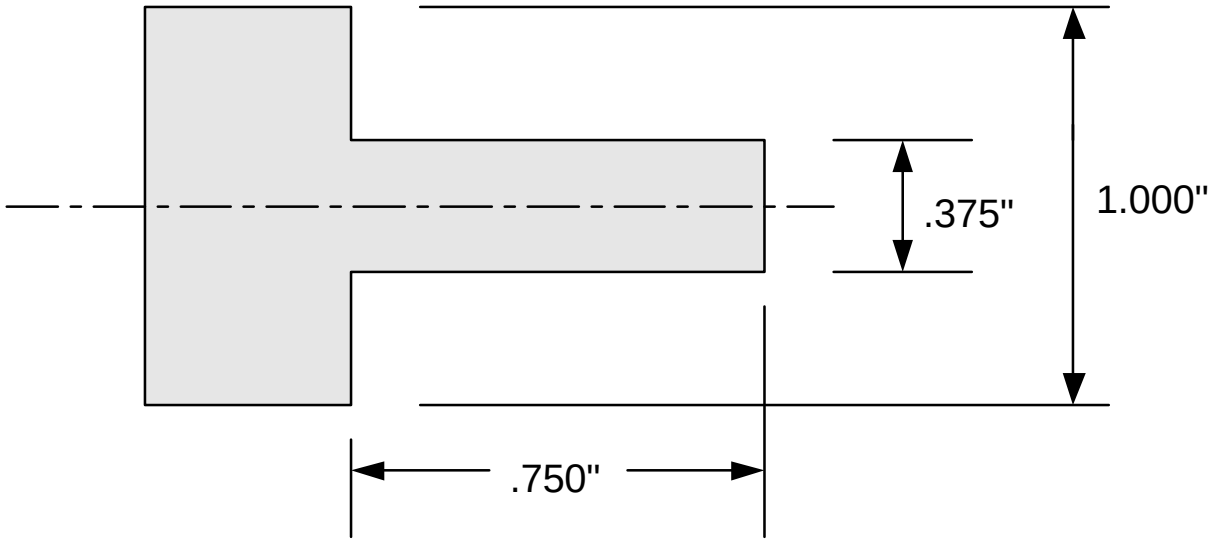
The G74 cycle can be used in radius (G73) or diameter (G72) mode. In both cases the In value is the amount to be taken off on a side. An In value of .05 would take .1 off the diameter of the part with each pass.

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

G74 - Box Roughing Cycle continued

Worked example for G74

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



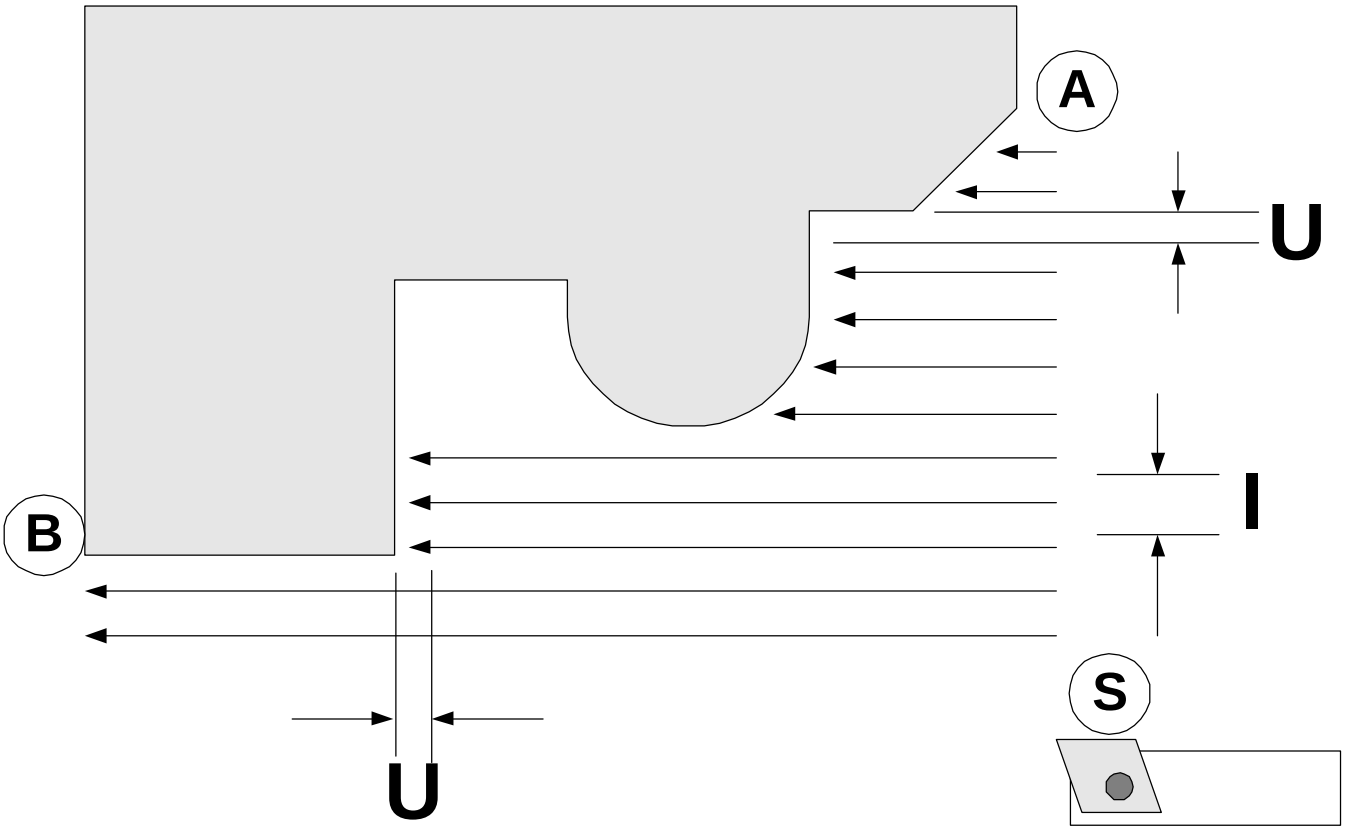
G90G94F30OG72
M03S2000
T1(LH Turn tool for roughing)
X1.05Z.5
Z.1
G95
G74X.375Z-.747I.075U.005F.003 programmed to leave .005 on X and Z for finish cut
M05
G00Z 1
X-1
M30

G75 - Box Contour Roughing Cycle continued

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

G75InUnFnPn

Un is the amount to be left on the part for the a finish pass
In is the maximum amount to be roughed per pass, defined as the depth of cut per side
Fn is the feedrate
Pn (optional) is a subroutine number



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

The location of the contour code to rough to:

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

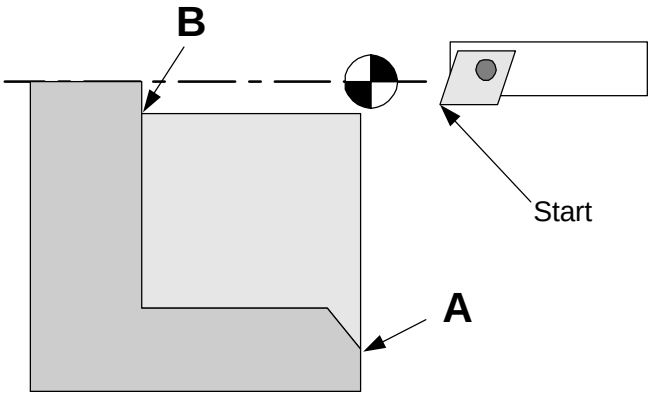
G75 - Box Contour Roughing Cycle continued

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

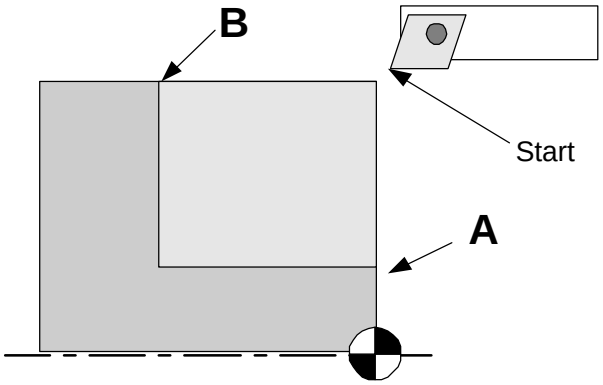
The RF code must be on a line by itself.

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

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



Internal example



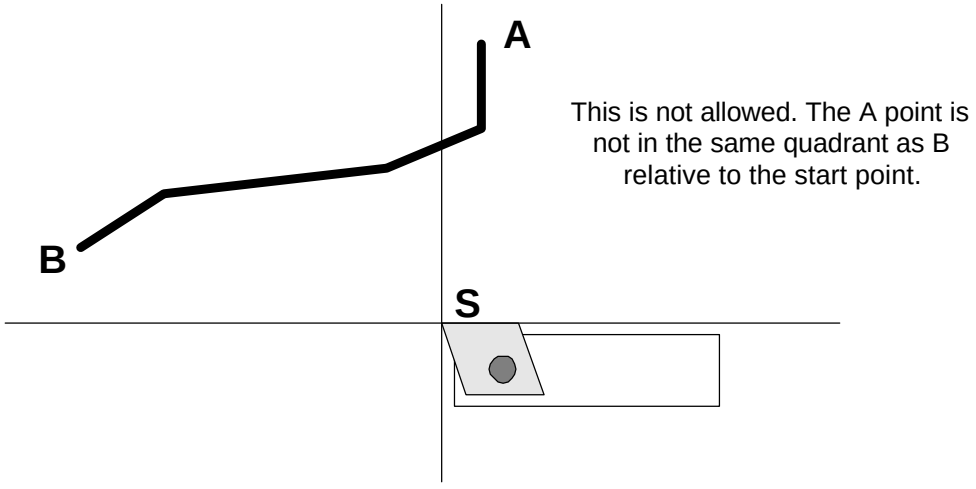
External example from the back

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

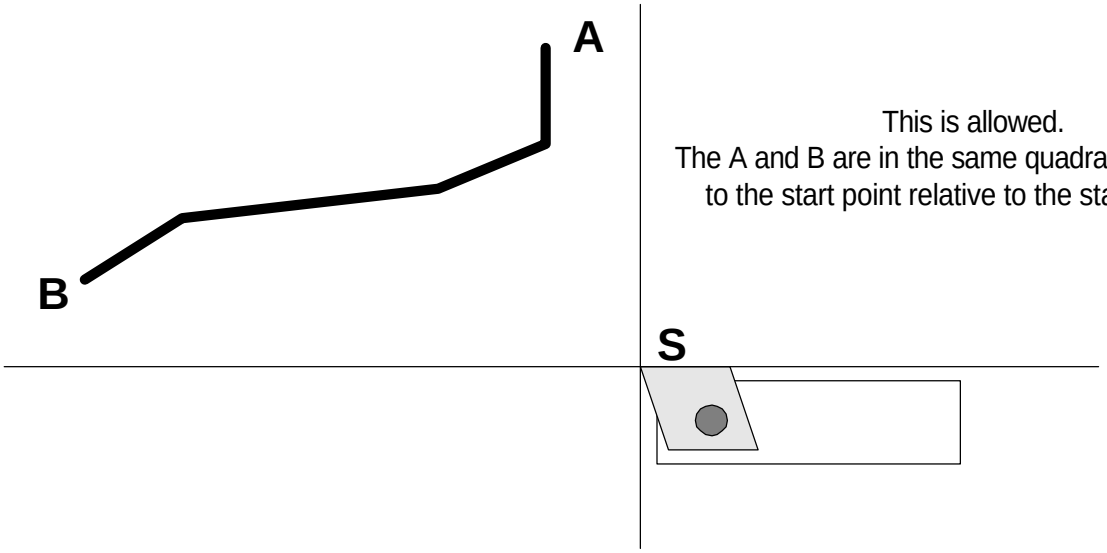
Tool nose radius compensation can not be active when using the roughing cycle.

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

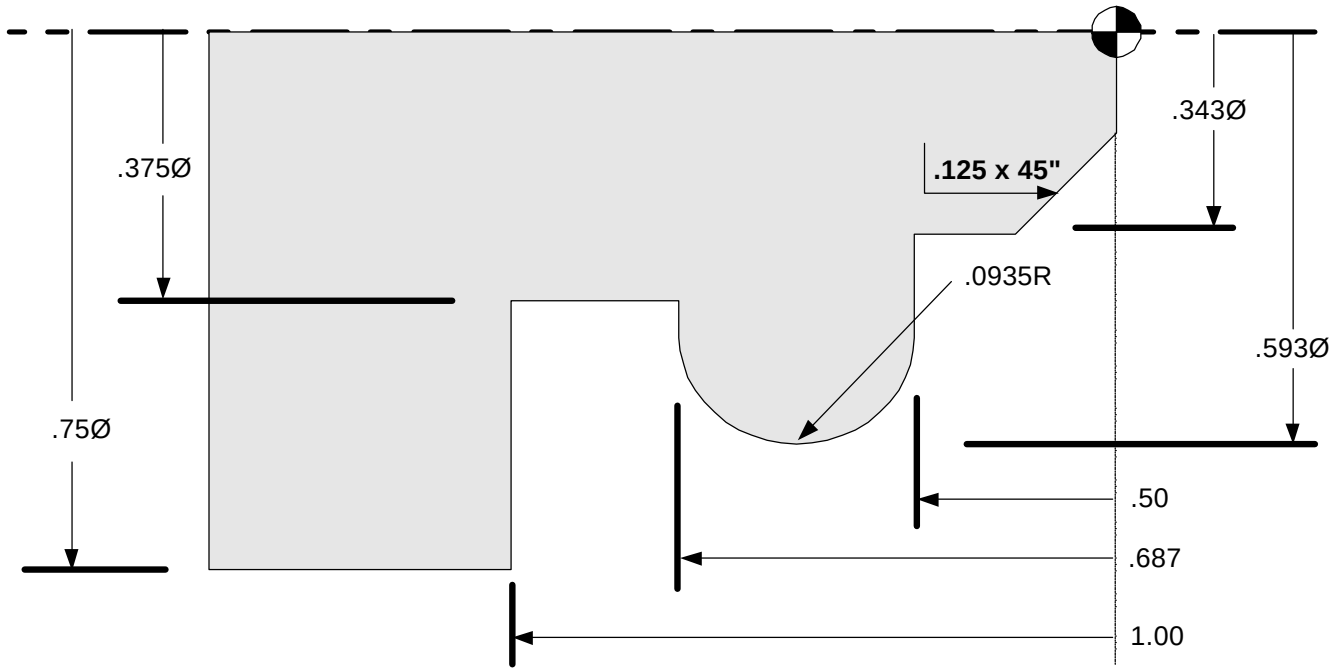
An error will be declared if the start and end point of the contour (A & B) are not in the same quadrant relative to the start point (S)



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



Worked external examples for G75



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

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

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

G75 - Box Contour Roughing Cycle continued

Written in Diameter mode

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

Written in Radius mode

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

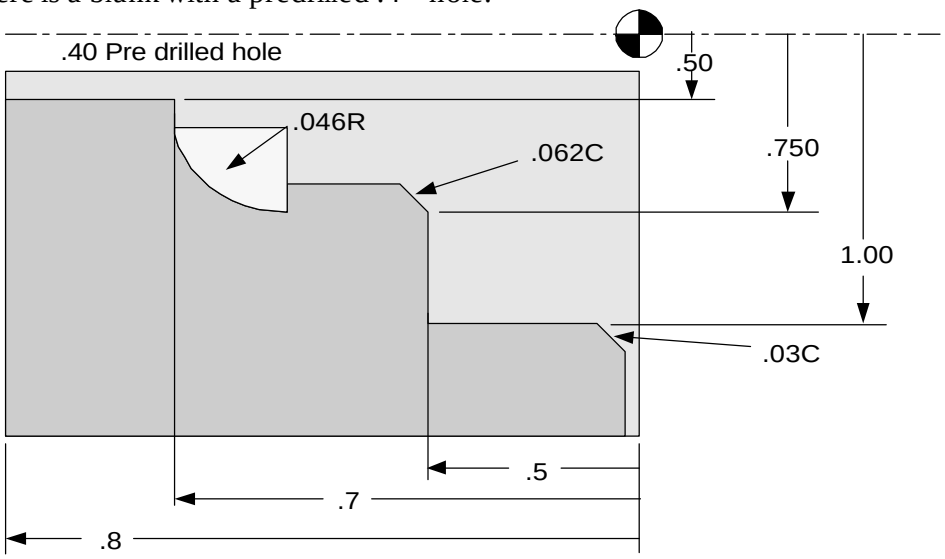
In this example the contour code is given as a subroutine. You can use the same subroutine for the G75 and G78 roughing passes. If you have to make a correction to the contour, it can be done in one place instead of for each cycle. You can also use it for the finishing pass if you are not going to use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

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

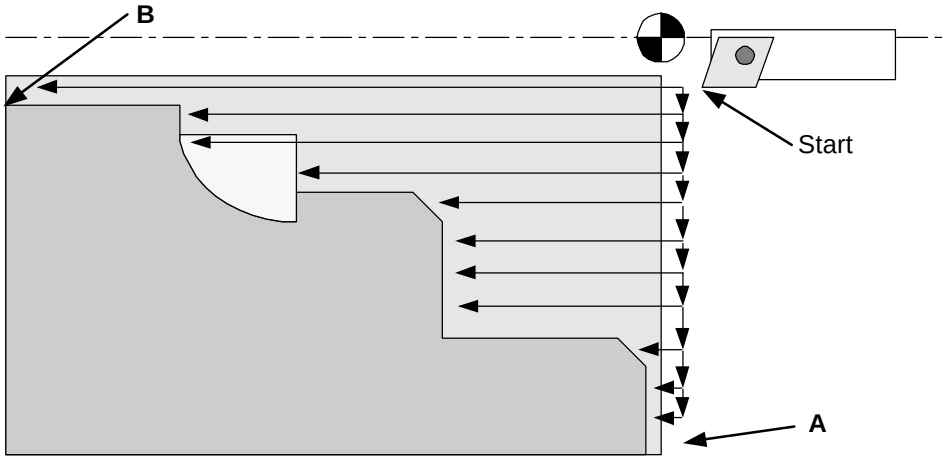
G75 - Box Contour Roughing Cycle continued

Worked internal example for G75

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



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



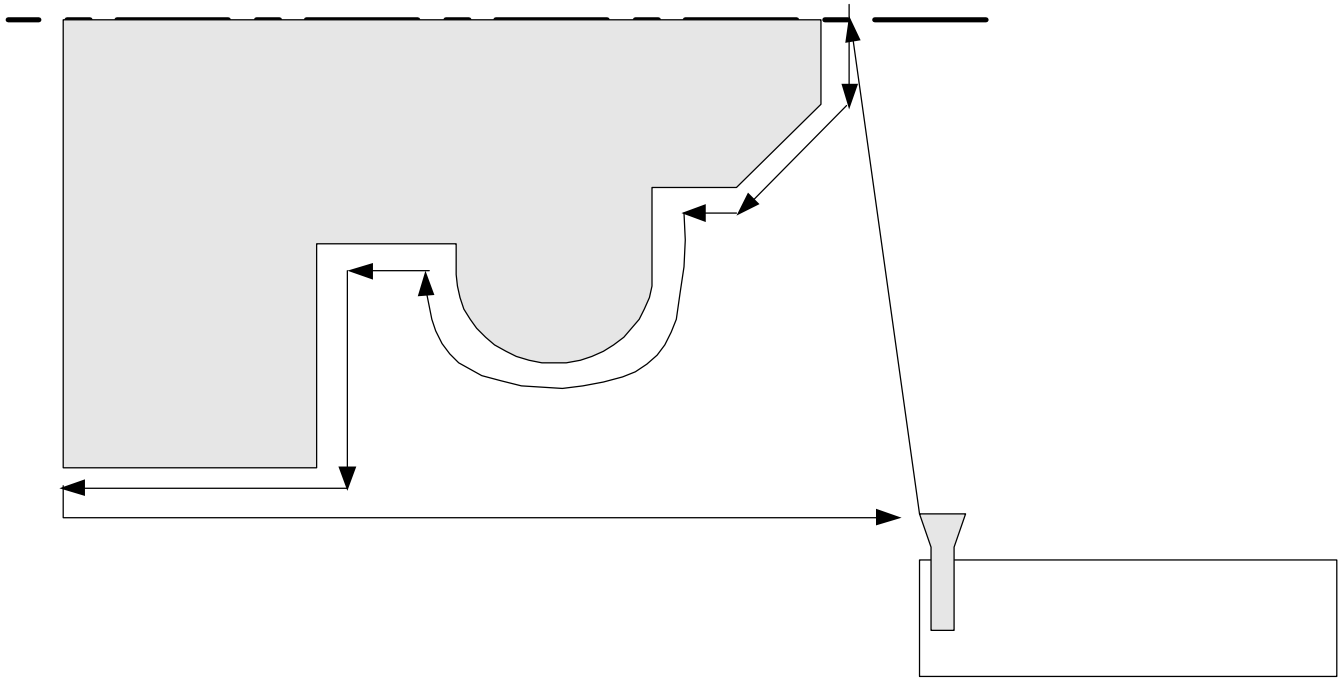

```
G90G94F300G72
M03S2000
T1
X.4Z.1
G95F.003
G751.05U.02F.003
X 1.2Z0
X1.06
X 1 Z-.03
Z-.5
X.75C.062
Z-.704
G03X.658Z-.75R.046
X.5
Z-1.
RF
G00Z 1
M30
```

G78 - Rough Contour Cycle

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

G78UnFnPn

Un is the amount to be left on the part for the a finish pass, (amount per side)
Fn is the feedrate
Pn (optional) is a subroutine number



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

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

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

Worked example for G78

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

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

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

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

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

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

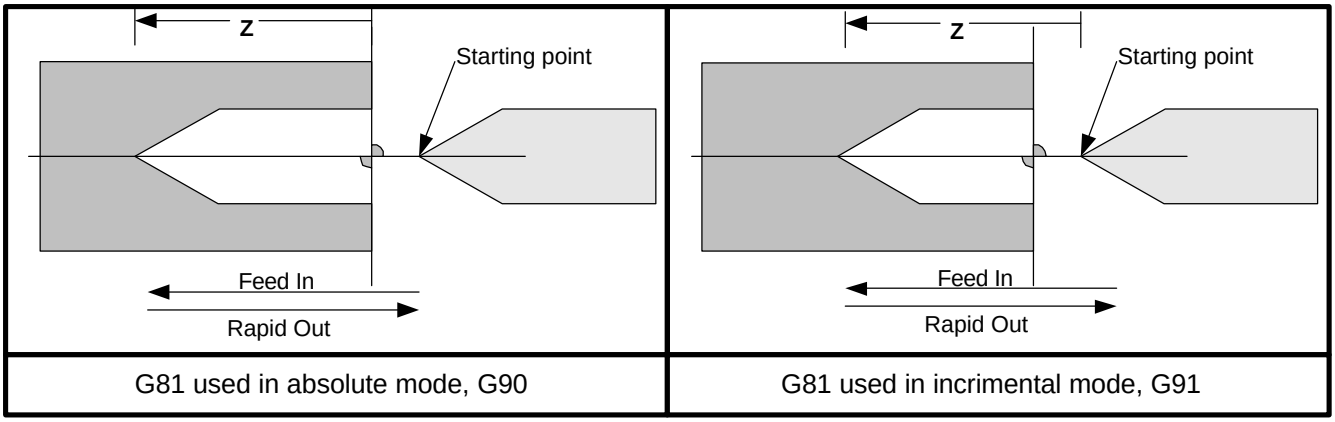
G78 - Rough Contour Cycle continued

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

G81 Drill Cycle

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

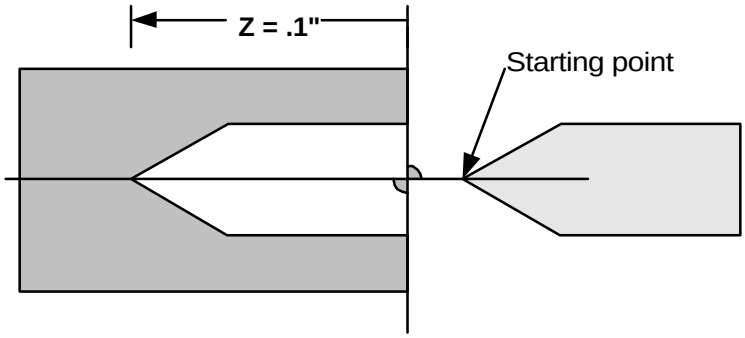
G81 Zn Fn



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

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

G81 Example



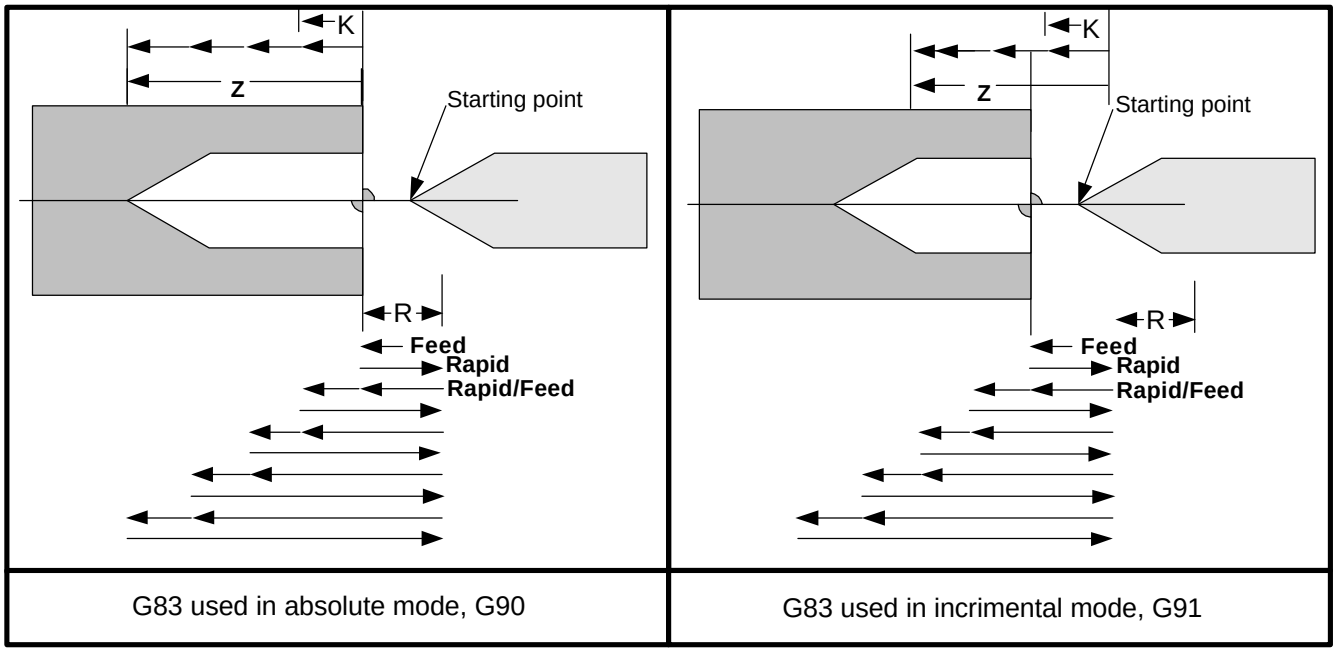
To drill a part .l" deep at a feed of .005" per revolution. The program would be:

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

G83 Peck Drill Cycle

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

G83 Zn Kn Fn Rn Ln Cn



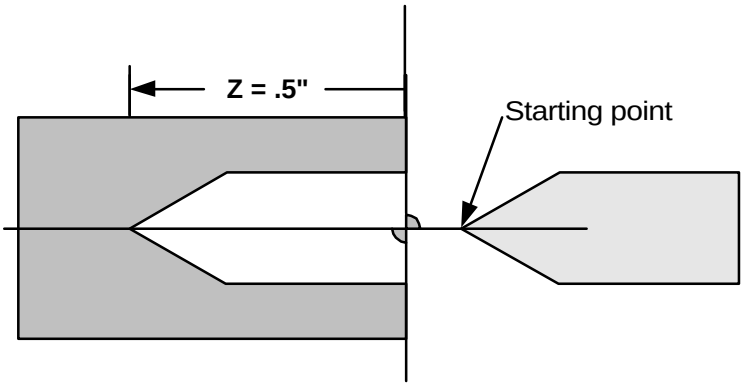
G83 used in absolute mode, G90 | G83 used in incrimental mode, G91

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

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

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

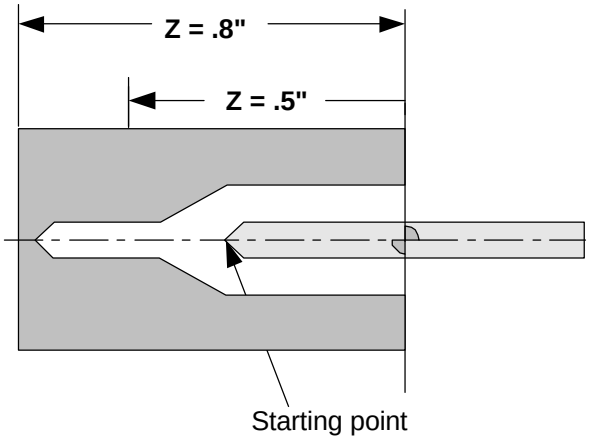
G83 Peck Drill Cycle



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

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

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



G90 G94F300	
T1	First drill (large one)
X0Z.01	
G95	
G83Z-.5K.2F.003R.5C.05L300	
G00Z1	
T2	Second drill (small one)
X0Z 1	
Z-.4	
G83Z-.8K. 1 F.002R.5C.05L300	

G90, G91 - G92 - G94, G95

G90 and G91 Absolute and Incremental mode selection

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

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

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

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

G92 is a position preset.

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

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

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

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

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

G94 and G95 Feedrate selections

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

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

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

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

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

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

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

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

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

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

Important Note

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

Simple formulas to convert these values are:

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

Sample program showing constant surface feet:

```
G90G94F300
M03S1500          Turn spindle speed on
T1(LH TURN TOOL .008 RADIUS)
X.25Z.2
G96S250           Set spindle to SFM mode @250sfm
G76S500           Establish minimum spindle speed at 500 rpm
G77S2500          Establish maximum spindle speed at 2500 rpm
Z0
G95F.002X0
G94F300Z2
G97               Switch to RPM mode
S2000             Set spindle speed at 2000 rpm
T2(DRILL)
X0Z.2
G95F.003Z-.5
G94F300Z2
M30
```

Program commands - “M-Codes”

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

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

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

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

M00 Program Stop

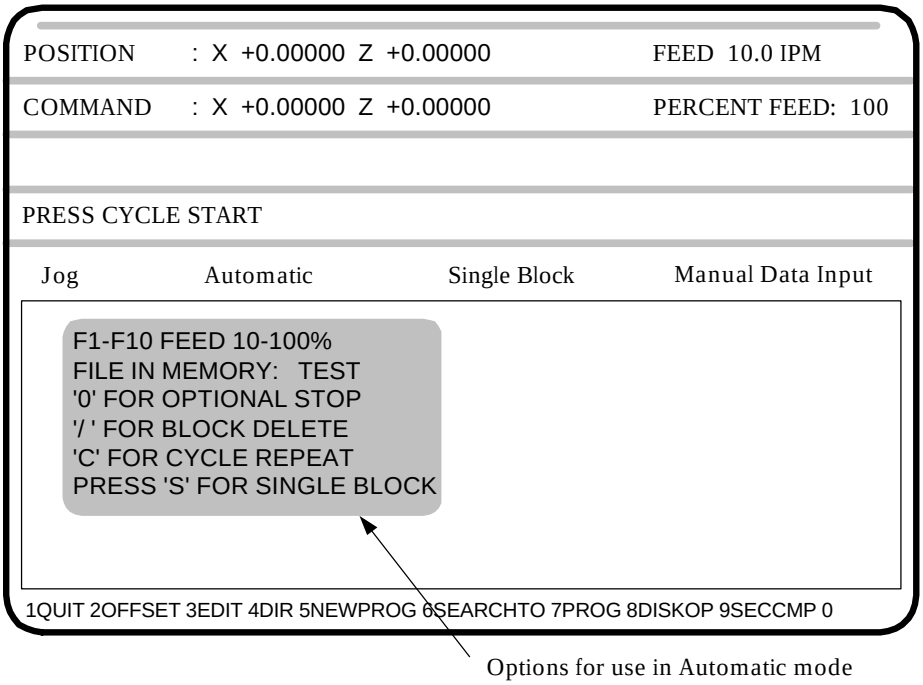
This program stop is not optional. When the control encounters the M00 it will stop and wait for the operator to press ”CYCLE START”. As an example: This command can be used to stop the program when the slide has a stop in position to allow an operator to push a piece of bar stock to the stop. Once the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

M01 Program Stop - Optional

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

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

-Have a M0l at the beginning of a program that is going to use an automatic bar feed or parts loader. This way you can have the optional stop activated when you are setting up the machine. Once the cycle and program are proven correct you can turn off the stop and let the machine run automatically.



M03, M04, M05 -Spindle control

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

In general these commands are:

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

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

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

M03S2000 This turns the spindle on forward, at 2000 RPM

M08, M09 -Coolant control

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

M12, M13 -Collet clamp control

These commands require optional equipment. Additional information about wiring in this command to your machine can be found in the "TECHNICAL" section.

M12 clamps the collet
M13 unclamps the collet

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

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

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

M30 & M02 -End of program

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

M31 - Cancel Cycle Repeat

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

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

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

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

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

Relay closure to 0VDC (COM) sets the input "on".
The input is "off" when the relay is open.
The inputs are located on TB2 in the spindle cabinet. (see page 6-22 for spindle panel layout).
The commands are as follows:

M91 Wait for TB2-5 to be open circuit

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

M93 Wait for TB2-7 to be open circuit

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

M95 - Conditional jump to subroutine (optional: C-Axis only)

This command will cause the program to jump to subroutine 1 if input 7 is "on" (shorted to 0VDC). Input 7 is located at TB2-9 in spindle cabinet (see page 6-22 for spindle panel layout). **The condition must exist *before* the command is executed.** Use dwell (G04) if necessary to insure that the state of the input is stable *before* the program executes the M97 command

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

This command will cause the program to jump to any subroutine if any available PLC input is either "on" or "off". The syntax is M97InCnPn.
In is the input which is being tested
Cn is the condition; either 1 ("ON") or 0 = ("OFF")
Pn is the subroutine which will be executed
The condition must exist *before* the command is executed. Use dwell (G04) if necessary to insure that the state of the input is stable *before* the program executes the M97 command

M98 - Jump to subroutine (unconditional)

When this command is executed, the program will jump to the specified subroutine.
The syntax is M98Pn, where n is the subroutine number.

}n - Begin subroutine n

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

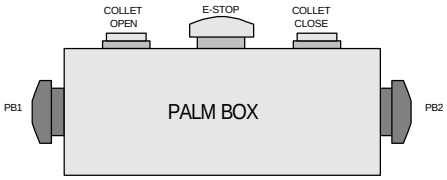
M99 - End subroutine

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

Spindle Control Option

Cycle Start

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



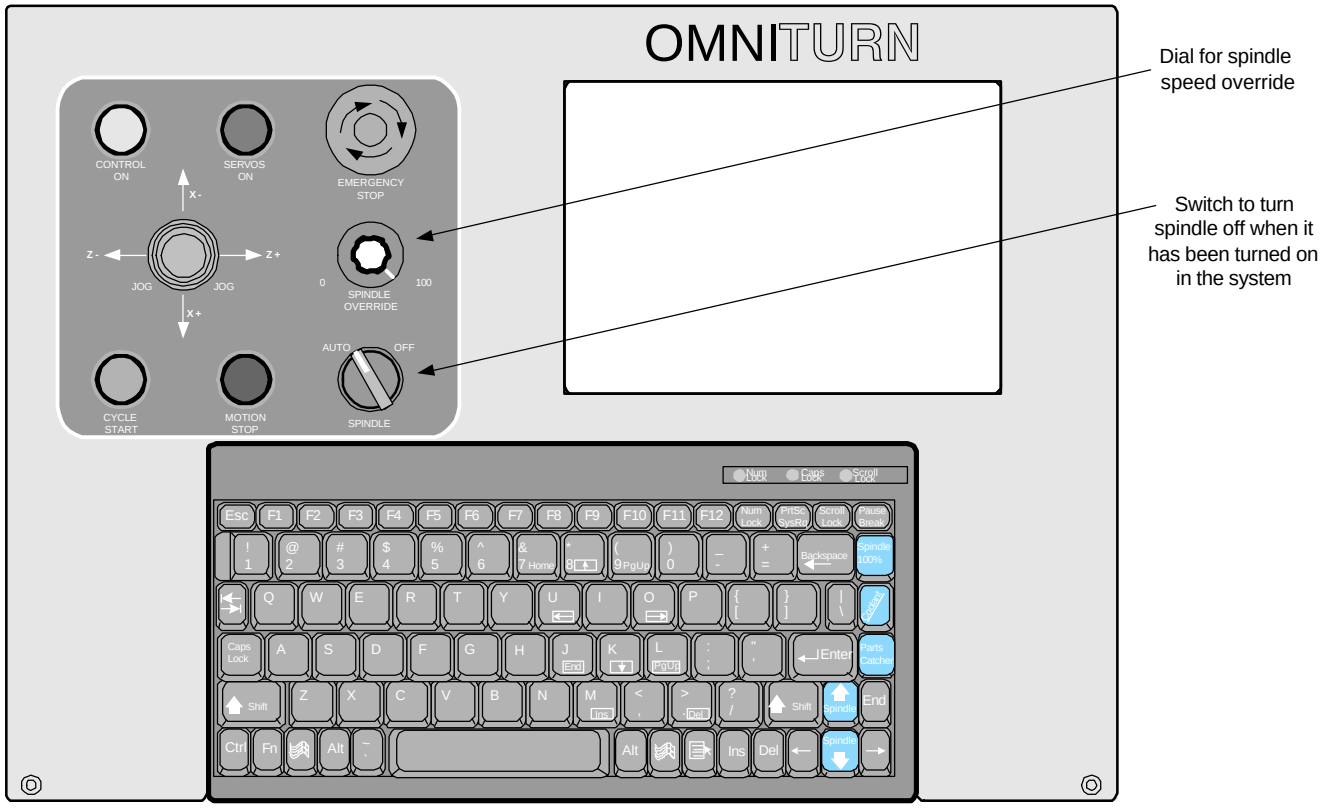
This station has two palm buttons on the sides of the box -PB 1 and PB2. Push both buttons together for cycle start.

Controlling the spindle (not C axis machines):

Manual control of the spindle

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

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



OmniTurn front panel Keyboard

From the program:

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

Spindle on, forward: **M03**S2000
Spindle on, reverse: **M04**S2000

Turning the spindle off:

Spindle off automatically at the end of the program: **M30**
Spindle off by command in the program: **M05**
Manual override: There is a switch on the face of the OmniTurn control that will turn the spindle off.

Leaving the spindle on at the end of the program:

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

Setting a spindle speed:

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

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

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

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

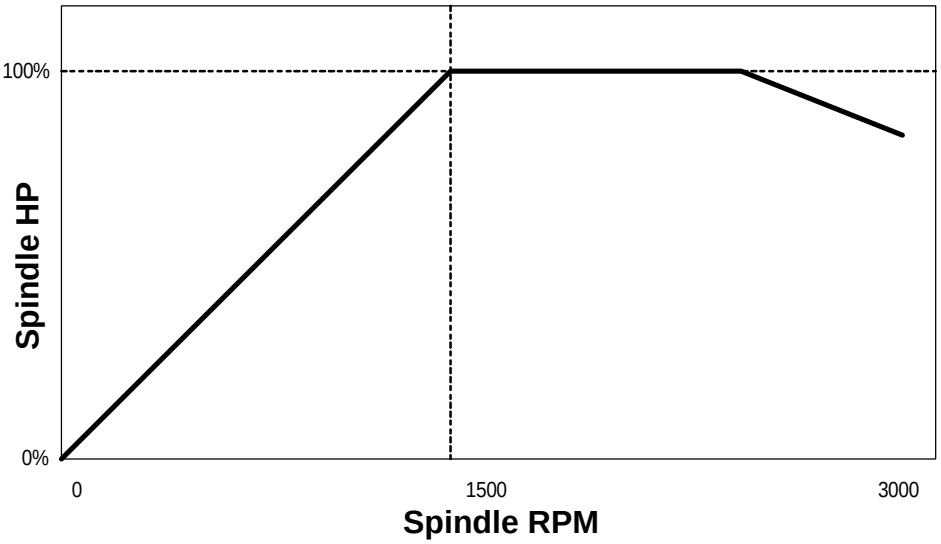
Tapping with the spindle control:

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

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

** This dwell is used to give the spindle some time to reverse before starting the tap back out. If the spindle reversal takes longer than you allow then the tap will start to pull out before the spindle is going reverse and not good things will happen, *snap*. This dwell time will vary depending on the system you have and the spindle speed you are running. The lower the spindle speed, the shorter the dwell time.

Setting the spindle speed range (for attachments only, GT-75 is done by the factory):
With the spindle drive it is possible to setup different maximum speeds to make available more HP at lower spindle speeds. The inverter drives that we use to vary the speed are constant torque at all RPMs. However the HP is lower at the lower RPMs. See chart below:



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

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

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

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

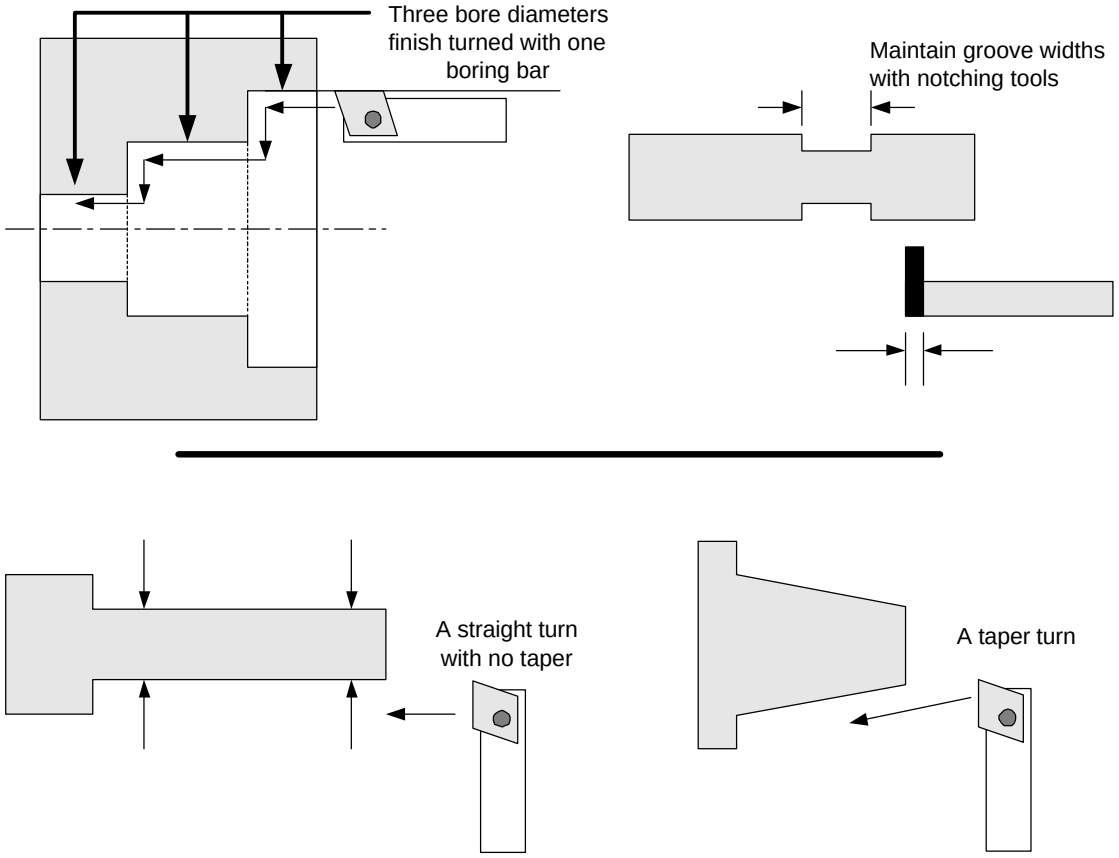
Setting the Max speed permanently:
This is done by changing the PRM.SER file. To do this look in chapter 7 -DOS notes for a procedure.

Secondary Offsets

What are secondary offsets?

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

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



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

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

NOTE: Clear secondary offsets before using them!

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

Secondary Offsets

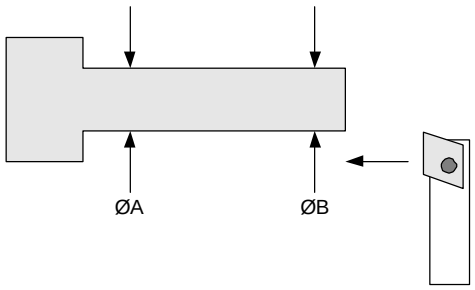
Using Secondary Offsets

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

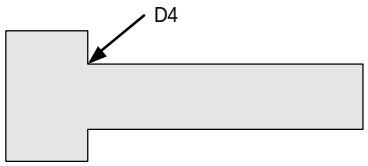
1	X: +0.00000 Z: +0.00000R: 0.00000	17	X: +0.00000 Z: +0.00000R: 0.00000
2	X: +0.00000 Z: +0.00000R: 0.00000	18	X: +0.00000 Z: +0.00000R: 0.00000
3	X: +0.00000 Z: +0.00000R: 0.00000	19	X: +0.00000 Z: +0.00000R: 0.00000
4	X: +0.00000 Z: +0.00000R: 0.00000	20	X: +0.00000 Z: +0.00000R: 0.00000
5	X: +0.00000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

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

Look at the following example:



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



The code to face and turn the shaft is:

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

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

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

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

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

X.1245Z.75

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

Canceling a secondary offset

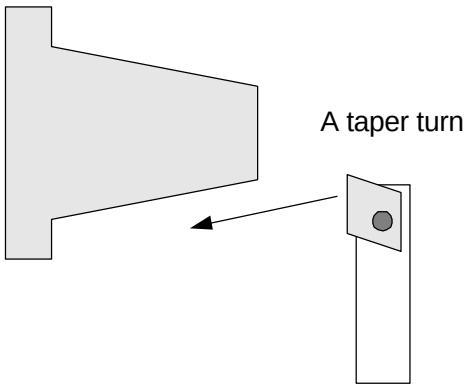
The secondary offset correction stays in effect until:

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

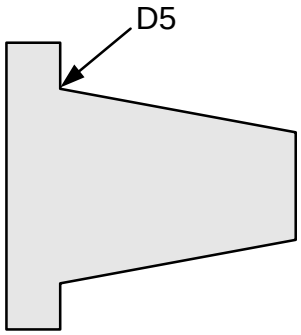
Secondary Offsets continued

Secondary offset examples:

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



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



The commands might look like this:

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

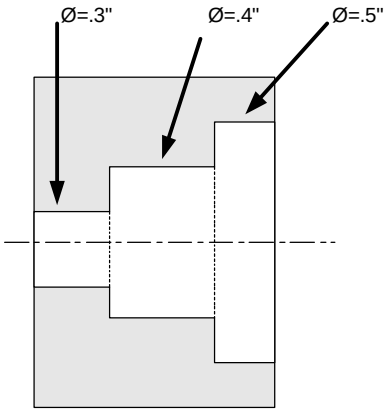
In this case D5 can be used to adjust two features:

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

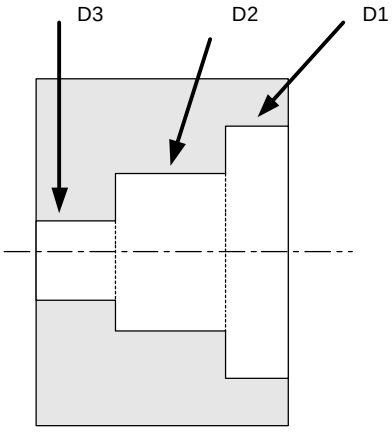
Neither, either, or both values can be entered. Each will effect the angle generated.

Secondary Offsets continued

Here is another example:



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



The coding for this is:

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

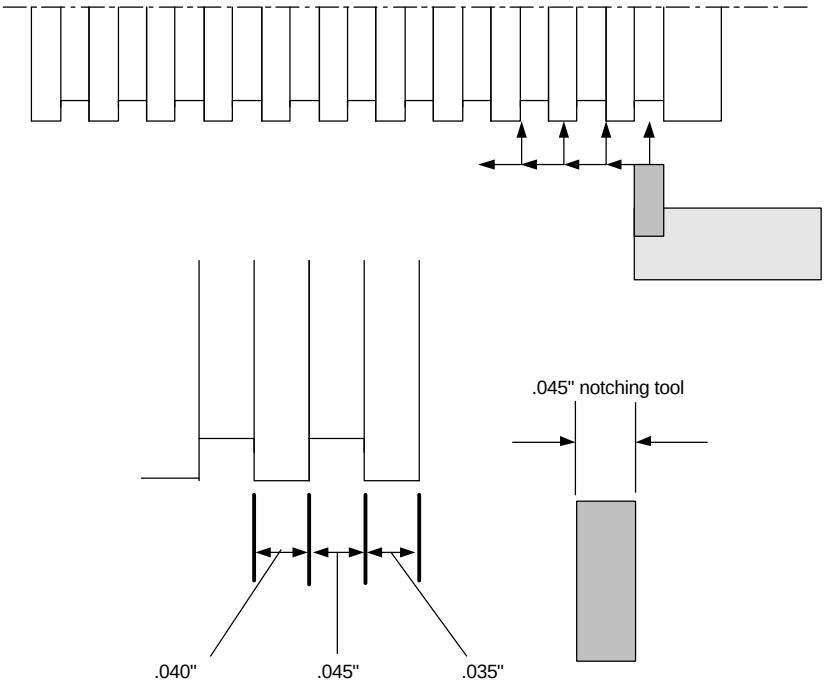
Here we have used the secondary offsets to help position the tool at the beginning of each of the bores. The offset correction will be added to the positioning move and then be maintained until a new offset is changed. For the .5”diameter we have added D1 to the move that brings the tool to the .5” diameter.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

Looping

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

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

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

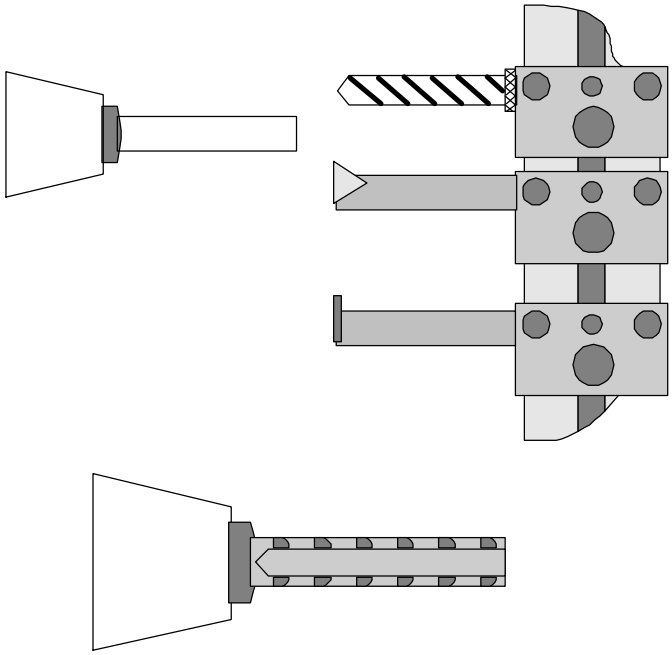


For this example the could be:
G90G94F300
T 1 (notch tool 045 wide)
X.25Z.2
Z-.035
G91----------NOTE THIS LOOP IS DONE IN INCRIMENTAL
LS16
G95F.001 X-.1
G04F.05
G00X.1
Z-.085
LF
G90----------BACK TO ABSOLUTE MODE
Zl
M30

Looping

Looping with Work Shift (G10)

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



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

Spindle Positioning

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

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

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

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

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

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

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

After G35 and G36 there must be a G92 command

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

Notes on use:

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

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

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

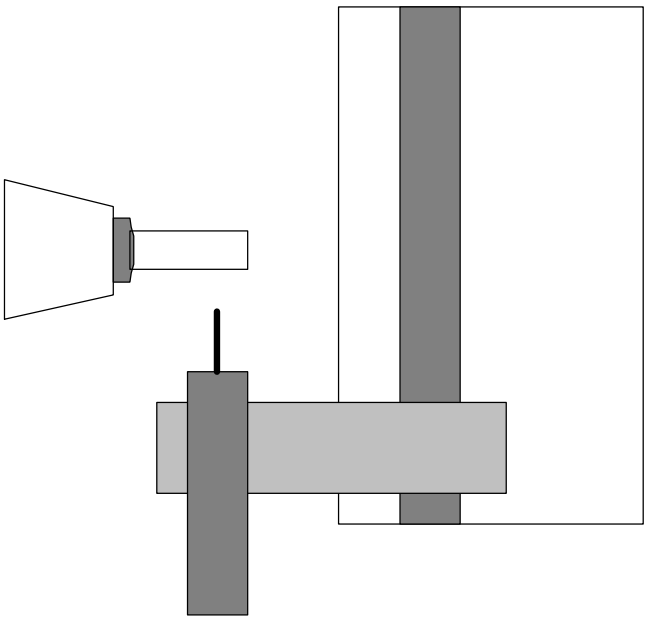
```
Z0
G35
G92X0Z0
G95F.25
C360Z-.3S5
G94F50Z0
G36
G92X0Z0
```

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

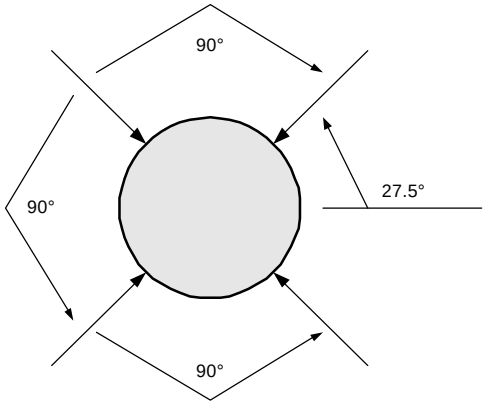
Spindle Positioning

Example showing positioning and cross drilling

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



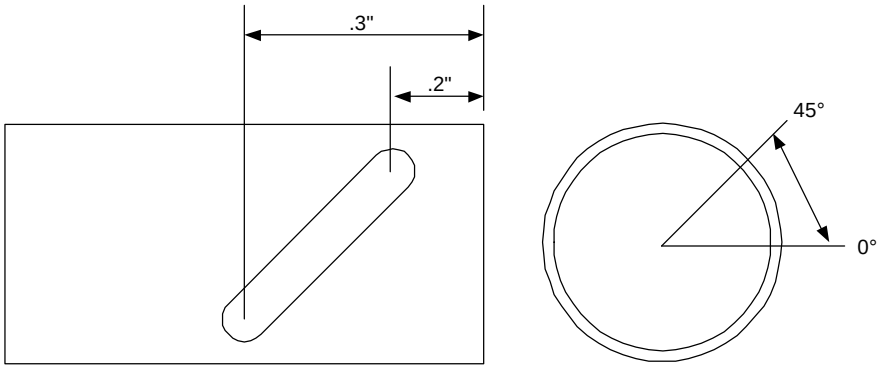
The slide will be used to drill the holes. We will drill (4) holes 90° apart, the first hole is located at 27.5° from a reference 0°.




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

Spindle Positioning

Two examples showing rotational milling, g95 & g94



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

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

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