

VF-4HS Series CNC MACHINE



Haas Automation Inc.

PROGRAMMING WORKBOOK



HAAS AUTOMATION, INC.
2800 Sturgis Rd.
Oxnard, CA 93030

January 2005



HAAS AUTOMATION INC.

2800 Sturgis Road
Oxnard, California 93030
Phone: 805-278-1800
www.HaasCNC.com

The information in this workbook is reviewed regularly and any necessary changes will be incorporated in the next revision. This material is subject to change without notice.

Warning: This workbook is for the exclusive use of Haas Customers, Distributors and Trainers and is protected by copyright law. The reproduction, transmission or use of this document or its contents for profit is not permitted.

All content is the property of Haas Automation, Inc., copyright 2004-2005. This Workbook may not be copied, distributed or reproduced for profit, in full or in part, without written permission from Haas Automation, Inc. This training information is being supplied for free to all Haas customers and schools that are learning to use Haas equipment. Haas information should never be modified unless you have written permission by Haas Automation.

Enquiries to Haas Automation about training information or a letter of authorization to copy, contact ebowman@haascnc.com

Copyright 2005, Haas Automation



CUSTOMER SATISFACTION PROCEDURE

Dear Haas customer,

Your complete satisfaction and goodwill are of the utmost importance to both Haas Automation, Inc., and the Haas distributor where you purchased your equipment. Normally, any concerns you may have about the sales transaction or the operation of your equipment will be rapidly resolved by your distributor.

However, if your concerns are not resolved to your complete satisfaction, and you have discussed your concerns with a member of the dealership's management, the General Manager or the dealership's owner directly, please do the following:

Contact Haas Automation's Customer Service Center by calling 800-331-6746 and ask for the Customer Service Department. So that we may resolve your concerns as quickly as possible, please have the following information available when you call:

- Your name, company name, address and phone number
- The machine model and serial number
- The dealership name, and the name of your latest contact at the dealership
- The nature of your concern

If you wish to write Haas Automation, please use this address:

Haas Automation, Inc.
2800 Sturgis Road
Oxnard, CA 93030
Att: Customer Satisfaction Manager
e-mail: Service@HaasCNC.com

Once you contact the Haas Automation Customer Service Center, we will make every effort to work directly with you and your distributor to quickly resolve your concerns. At Haas Automation, we know that a good Customer-Distributor-Manufacturer relationship will help ensure continued success for all concerned.

NOTE: Should you have a problem with your machine, please consult your operator's manual first. If this does not resolve the problem, call your authorized Haas distributor. As a final solution, call Haas directly at the number indicated below.

Haas Automation, Inc.
2800 Sturgis Road
Oxnard, California 93030-8933 USA
Phone: (805) 278-1800



CONTENTS

INTRODUCTION	1
THE COORDINATE SYSTEM	2
MACHINE HOME	5
ABSOLUTE AND INCREMENTAL POSITIONING	6
POSITIONING EXERCISE	8
PROGRAMMING WITH CODES	9
PROGRAM FORMAT	10
DEFINITIONS WITHIN THE FORMAT	11
OFTEN USED PREPARATORY "G" CODES	13
PREPARATORY "G" CODES LIST	14
MACHINE DEFAULTS	18
OFTEN USED MISCELLANEOUS "M" CODES	19
MISCELLANEOUS "M" CODES LIST	20
PROGRAM STRUCTURE	22
ALPHABETICAL ADDRESS CODES	24
RAPID POSITION COMMAND (G00)	28
LINEAR INTERPOLATION COMMAND (G01)	29
CIRCULAR INTERPOLATION COMMANDS (G02, G03)	30
INTERPOLATION EXERCISE	39
PROGRAM START-UP LINES	40
PROGRAM ENDING LINES	41
INCH / METRIC SELECTION (G20, G21)	42
WORK COORDINATE SELECTION (G54-59, G110-129 & G154 with P1-99)	43
MORE WORK COORDINATE SELECTION (G52, G53,)	44
TOOL LENGTH COMPENSATION (G43)	45
DWELL COMMAND (G04)	46
REFERENCE POINT AND RETURN (G28)	47
ANOTHER WAY TO RETURN TO MACHINE ZERO (G53)	48
CIRCULAR POCKET MILLING (G12, G13)	49
CIRCULAR POCKET MILLING EXERCISE	53



CONTENTS

CIRCULAR PLANE SELECTION (G17, G18, G19)	54
CUTTER COMPENSATION (G40, G41, G42)	58
CUTTER COMPENSATION EXERCISE #1	64
ADVANTAGES OF CUTTER COMPENSATION	65
CUTTER COMPENSATION EXERCISE #2	66
THREAD MILLING WITH HELICAL MOTION	68
CANNED CYCLES FOR DRILLING TAPPING AND BORING	70
CANCEL CANNED CYCLE (G80)	71
CANNED CYCLE RETURN PLANES (G98, G99)	72
DRILL CANNED CYCLE (G81)	73
SPOT DRILL/COUNTERBORE CANNED CYCLE (G82)	74
DEEP HOLE PECK DRILL CANNED CYCLE (G83)	75
CANNED CYCLE EXERCISE #1	78
TAPPING CANNED CYCLE (G84)	80
BORE IN - BORE OUT CANNED CYCLE (G85)	81
BORE IN - STOP - RAPID OUT CANNED CYCLE (G86)	82
BORE IN - MANUAL RETRACT CANNED CYCLE (G87)	83
BORE IN - DWELL - MANUAL RETRACT CANNED CYCLE (G88)	84
BORE IN - DWELL - BORE OUT CANNED CYCLE (G89)	85
CANNED CYCLE EXERCISE #2	86
HIGH SPEED PECK DRILL CANNED CYCLE (G73)	88
REVERSE TAPPING CANNED CYCLE (G74)	92
BORE IN - SHIFT OFF - RAPID OUT CANNED CYCLE (G76)	93
BACK BORE CANNED CYCLE (G77)	94
BOLT HOLE PATTERNS (G70, G71, G72)	95
BOLT HOLE CIRCLE (G70)	96
BOLT HOLE ARC (G71)	97
BOLT HOLES ALONG AN ANGLE (G72)	98
CANNED CYCLE EXERCISE #3	100
SUBROUTINE (M97, M98, M99)	103



CONTENTS

GENERAL PURPOSE POCKET MILLING (G150)	107
MISCELLANEOUS "M" CODES	114
FINAL EXERCISE	128





INTRODUCTION

This manual provides basic programming principles necessary to begin programming the HAAS C.N.C. Milling Machine.

In a "CNC" (Computerized Numerical Control) machine, the tool is controlled by a computer and is programmed with a machine code system that enables it to be operated with minimal supervision and with a great deal of repeatability.

The same principles used in operating a manual machine are used in programming a CNC machine. The main difference is that instead of cranking handles to position a slide to a certain point, the dimension is stored in the memory of the machine control once. The control will then move the machine to these positions each time the program is run.

In order to operate and program a CNC controlled machine, a basic understanding of machining practices and a working knowledge of math is necessary. It is also important to become familiar with the control console and the placement of the keys, switches, displays, etc., that are pertinent to the operation of the machine.

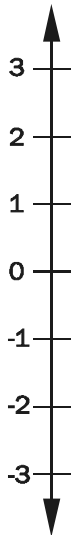
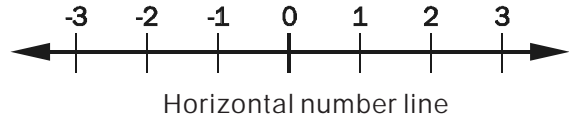
This workbook can be used for both operator's and programmer's. It is intended to give a basic understanding of CNC programming and it's applications. It is not intended as an in-depth study of all ranges of machine use, but as an overview of common and potential situations facing CNC programmers. Much more training and information is necessary before attempting to program on the machine.

This programming manual is meant as a supplementary teaching aid to users of the HAAS Mill. The information in this workbook may apply in whole or in part to the operation of other CNC machines. Its use is intended only as an aid in the operation of the HAAS Milling Machine. For a complete explanation and an in-depth description, refer to the Programming and Operation Manual that is supplied with your HAAS Mill.



THE COORDINATE SYSTEM

The first diagram we are concerned with is called a NUMBER LINE. This number line has a zero reference point that is called an ABSOLUTE ZERO and may be placed at any point along the number line.



The number line also has numbered increments on either side of absolute zero. Moving away from zero to the right are positive increments. Moving away from zero to the left are negative increments. The "+", or positive increments, are understood, therefore no sign is needed. We use positive and negative signs along with increment value's to indicate its relationship to zero on the line. If we choose to move to the third increment on the minus (-) side of zero, we would call for -3. If we choose the second increment in the plus range, we would call for 2. Our concern is the distance and the direction from zero.

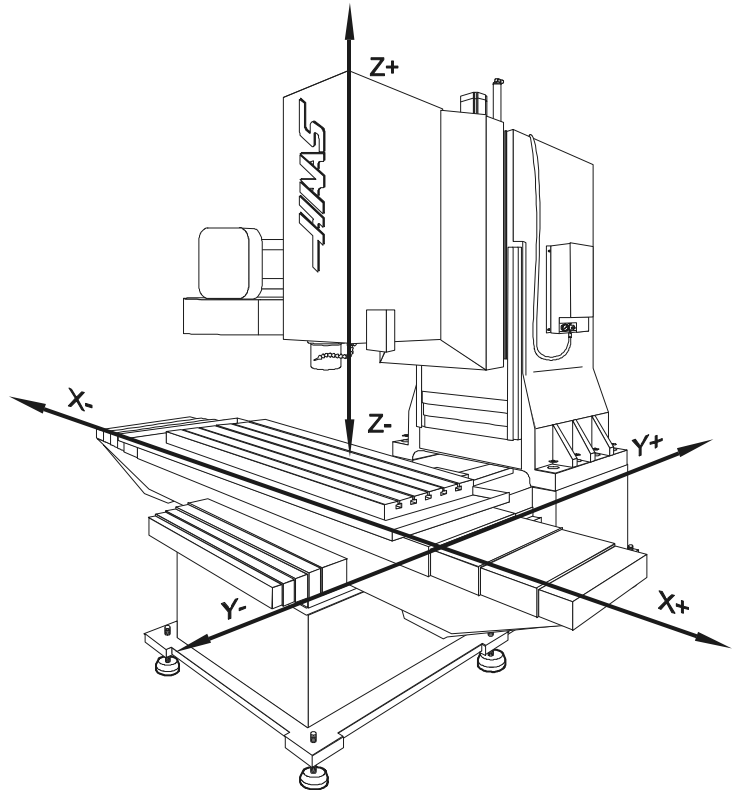
Remember that zero may be placed at any point along the line, and that once placed, one side of zero has negative increments and the other side has positive increments.

Vertical number line

The machine illustration shows three directions of travel available on a vertical machine center. To carry the number line idea a little further, imagine such a line placed along each axis of the machine. It shows the three directions to position the coordinates around a part origin, which is where these number lines intersect on a vertical machining center with the X, Y, and Z axis lines.

The first number line is easy to conceive as belonging to the left-to-right, or "X", axis of the machine. If we place a similar number line along the front-to-back, or "Y" axis, the increments (not the table) toward the operator, from Y zero, are the negative increments. The increments on the other side of zero away from the operator are positive increments.

The third axis of travel on our machine is the up-and-down, or "Z" axis. When we place a number line on the Z travel, the positive increments are up above zero, and the negative





values are down below zero. The increments of each number line on HAAS machining centers equals .0001 inches. Also, while a line theoretically travels infinitely in either direction once established, the three lines placed along the X, Y, and Z axes of the machine do not have unlimited accessibility. That is to say, we are limited by the range of travel on the model of machining center.

MACHINE	X-axis travel	Y-axis travel	Z-axis travel
Toolroom Mill	30"	12"	16"
Mini Mill's	16"	12"	10"
Mill Drill	12"	10"	12"
VF-E/ VF-0/ VF-1	20"	16"	20"
VF-EXT/ VF-OE	30"	16"	20"
VF-2	30"	16"	20"
VF-3	40"	20"	25"
VF-4	50"	20"	25"
VF-5	50"	25"	25"
VF-5 XT	60"	25"	25"
VF-6	64"	32"	30"
VF-7	84"	32"	30"
VF-8	64"	40"	30"
VF-9	84"	40"	30"
VF-10	120"	32"	30"
VF-11	120"	40"	30"
VR-11	120"	40"	30"
VS-1	84"	50"	50"
VS-3	150"	50"	50"
EC-400	20"	20"	20"
EC-1600	64"	40"	32"
Mini HMC	15"	15"	10"
HS-1/HS-1R	24"	20"	22"
HS-1RP	24"	20"	22"
HS-2RP	38"	35"	30"
HS-3	150"	50"	60"
HS-4	150"	66"	60"
HS-6	84"	50"	60"
HS-7	84"	66"	60"

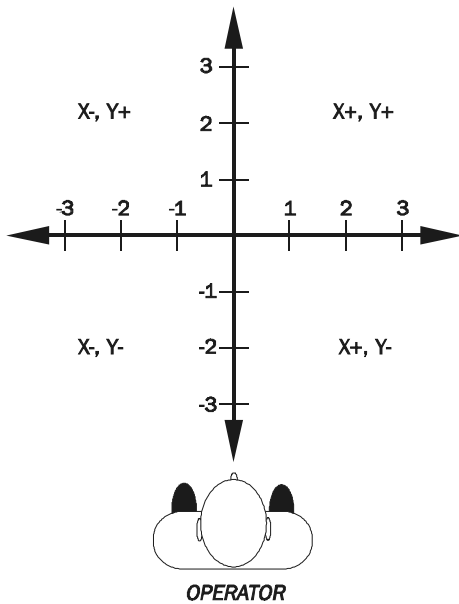


Remember, when we are moving the machine, we are concerned with positioning the center of the spindle in relation to X,Y and Z zero. Although the machine table is the moving part, we have to keep in mind our coordinates are based off our theoretical spindle movement.

Keep in mind that the part zero position may be defined at any point along each of the three axes, and will usually be different for each setup of the machine.

It is noteworthy to mention here that the Z-axis is set with the machine zero position in the upward position, or the tool change position. This will place most all Z moves in a negative range of travel.

This view shows the X,Y work zero grid from above. The work part zero for the Z-axis is usually set at the top of the part surface, and this will be entered in the tool length offset as a negative value for each tool. The range of Z-axis travel on the HAAS VF-1, for example, is 20 inches total; four of these inches are above tool change position and is



listed as a positive tool length offset, and 16 inches are below tool change position and listed as a negative. The diagram shows a top view of the grid as it would appear on the machine tool. This view shows the X and Y axes as the operator faces a vertical machine table. Note that at the intersection of the two lines, a common zero point is established. The four areas on each side and above and below the lines are called "QUADRANTS" and make up the basis for what is known as rectangular coordinate programming.

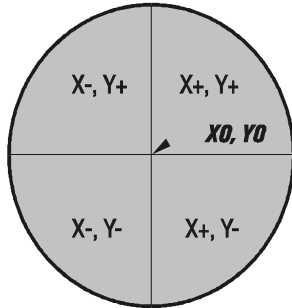
QUADRANT 1 IS ON THE TOP RIGHT = X+ Y+
 QUADRANT 2 IS ON THE TOP LEFT = X- Y+
 QUADRANT 3 IS ON THE BOTTOM LEFT = X- Y-
 QUADRANT 4 IS ON THE BOTTOM RIGHT = X+ Y-

Whenever we set a zero point somewhere on the X-axis and, a zero point somewhere on the Y-axis, we have automatically set a work zero point and an intersection of the two number lines. This intersection where the two zeros come together will automatically have the four quadrants to its sides, above, and below it. How much of a quadrant we will be able to access is determined by where we place the zero point within the travel of the machine axes. For example, for a VF-1, if we set zero exactly in the middle of the travel of X and Y (table center), we have created four quadrants that are 10 inches by 8 inches in size.



MACHINE HOME

The principal of machine home may be seen when doing a reference return of all machine axes at machine start-up. A zero return (POWER UP/RESTART) is performed when you



All four quadrants will
have

power on machine, all three axes are moved to extreme positive locations until limit switches are reached. When this condition is satisfied, the only way to move any of the three axes is in the negative direction (except for a positive four inches in Z-axis). This is because this position is defined as your MACHINE HOME for each of the three axes automatically when the machine was sent home with the POWER-UP/RESTART key. In effect, now the positive quadrants cannot be reached from machine home position in X and Y axes, and all the moves will be found to be in the X-, Y- quadrant. It is only by setting a new part zero somewhere within the travel of each axes that other quadrants are able to be reached.

Sometimes it is useful in the machining of a part to utilize more than one of these X,Y quadrants. An example of this is a round part that has it's datum lines running through the center. The setup of such a part may need machining to be performed in all four quadrants of a part. This is why you would want to make use of all four quadrants of the X and Y axes on a milling machine. As you gain more experience in machine tool programming and of setup techniques, you'll have a better understanding of how to position your machine tool and how to define a part zero origin and how to position a tool around that origin .

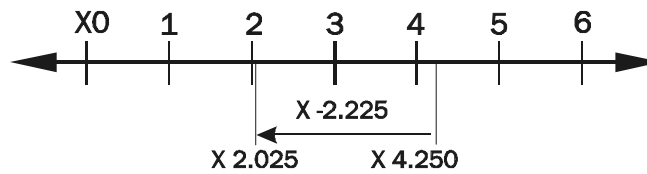


ABSOLUTE & INCREMENTAL POSITIONING

In Absolute positioning, all coordinate positions are given with regard to their relationship to a fixed zero, origin point, that is referred to as part zero. This is the most common type of positioning.

Another type of positioning is called incremental positioning. Incremental positioning concerns itself with distance and direction from the last position. A new coordinate is entered in terms of its relationship to the previous position, and not from a fixed zero or origin. In other words, after a block of information has been executed, the position that the tool is now at is the new zero point for the next move to be made.

An example of the use of the incremental system is below. Note that to move from X4.25 to X2.025 on the scale, an incremental move of X-2.225 is made, even though the move still places the tool on the plus side of the scale. Therefore the move was determined from the last point, with no regard for the part zero position. The + and - signs are used in terms of direction, and not in regard to the position of the part zero.



An example of an incremental move.

Keep in mind that when positioning in absolute, we are concerned with distance and direction from a fixed zero reference point, and when positioning in incremental we are concerned with distance and direction from the last position.

G90 ABSOLUTE POSITION COMMAND

When using a G90 absolute position command, each dimension or move is referenced from a fixed point, known as ABSOLUTE ZERO (part zero). Absolute zero is usually set at the corner edge of a part, or at the center of a square or round part, or an existing bore. ABSOLUTE ZERO is where the dimensions of a part program are defined from.

Absolute dimensions are referenced from a known point on the part, and can be any point the operator chooses, such as the upper-left corner, center of a round part, or an existing bore.

The Key to understanding ABSOLUTE dimensions is that they are always in reference to the ABSOLUTE ZERO (part zero). This part zero (work offset G codes G54-G59 and G110-G129) are set by the operator in the offset display using the Handle Jog operation mode. It can also be switched to a new part zero position during the program using a



different work offset G code that defines in it, another location (when machining with multiple vises and/or fixtures at separate locations on the machine table.)

Each dimension, or X-Y point is known as a coordinate. If a position 2 inches to the right, and 2 inches down (toward you) from part zero was programmed, the X coordinate would be X2.0 and the Y coordinate would be Y-2.0. And the machine would go to that exact location from part zero, regardless of where it began, within the travel of the machine tool.

X2.0 Y-2.0 could be a hole location, an arc end point, or the end of a line which are known coordinate values.

G91 INCREMENTAL POSITION COMMAND

This code is modal and changes the way axis motion commands are interpreted. G91 makes all subsequent commands incremental.

Incremental dimensions are referenced from one point to another. This can be a convenient way to input dimensions into a program (especially for G81-G89, G73, G74, and G77 canned cycles) depending on the blueprint.

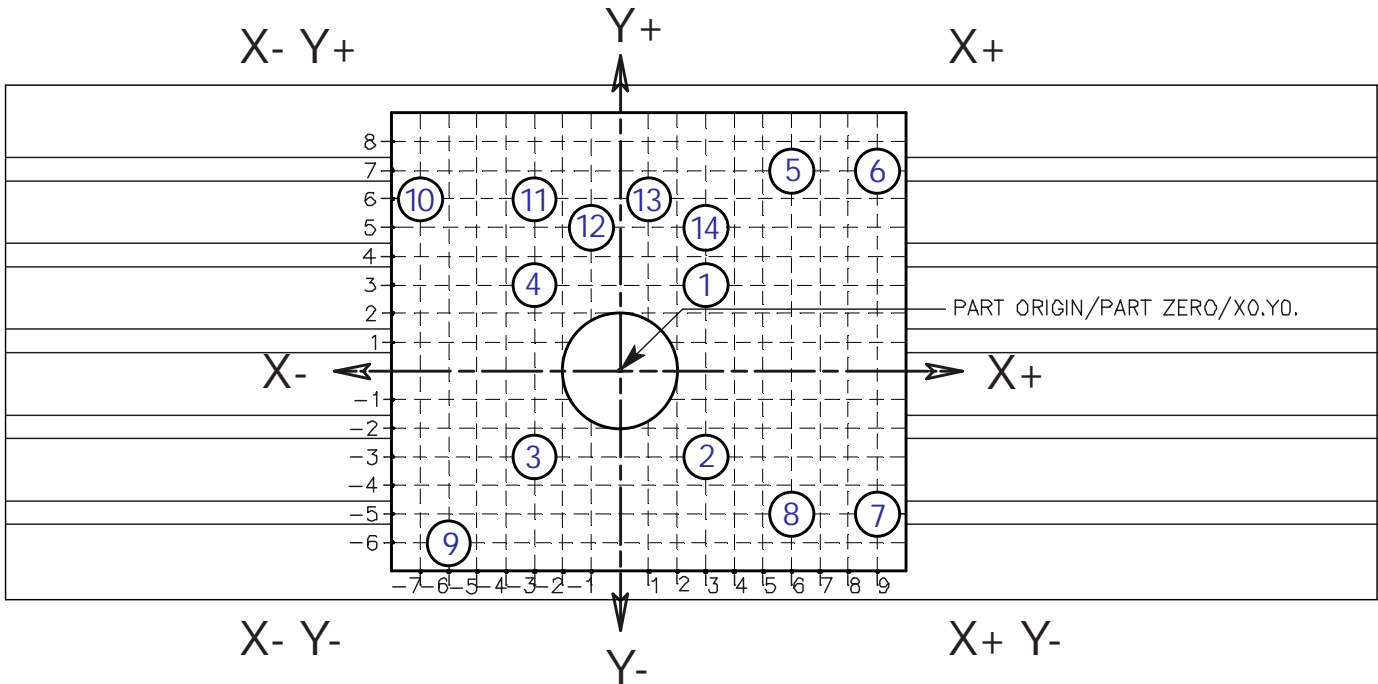
When using a G91 incremental position command, each measurement or move is the actual distance to the next location (whether it is a hole location, end of arc, or end of line) and is always in reference from the current location.

If you programmed a G91 with an X coordinate of X2.0 and a Y coordinate of Y-2.0, the machine would go that exact distance from where it is, regardless of where it began, within the travel of the machine tool.

Absolute mode should be your positioning mode of choice for most applications. There are times when incremental mode can be quite helpful. Repeating motions within a subroutine, for example, is one excellent example. If you have six identical pockets to machine on a Haas mill, you can save programming effort if you specify the motions incrementally to machine one pocket. Then just call up the subroutine again to repeat the commands to do another pocket at a new location.



POSITIONING EXERCISE



What is the value in X and Y for each hole in absolute G90 positioning when each move is defined from a single fixed part zero point of an X0 Y0 origin point.

- PT1 = X_____ Y_____
- PT2 = X_____ Y_____
- PT3 = X_____ Y_____
- PT4 = X_____ Y_____
- PT5 = X_____ Y_____
- PT6 = X_____ Y_____
- PT7 = X_____ Y_____
- PT8 = X_____ Y_____

What is the value for each hole in INCREMENTAL G91 positioning when each move is defined from the previous position and the zero point shifts with the new position.

- From PT8 to PT9 = X_____ Y_____
- From PT9 to PT10 = X_____ Y_____
- From PT10 to PT11 = X_____ Y_____
- From PT11 to PT12 = X_____ Y_____
- From PT12 to PT13 = X_____ Y_____
- From PT13 to PT14 = X_____ Y_____



PROGRAMMING WITH CODES

The definition of a part program for any CNC consists of movements of the tool, and speed changes to the spindle RPM. It also contains auxiliary command functions such as tool changes, coolant on or off commands, or external M code commands.

Tool movements consist of rapid positioning commands, straight line moves or movement along an arc of the tool at a controlled speed.

The HAAS mill has three (3) linear axes defined as X axis, Y axis, and Z axis. The X and Y axis will move the machine table below and around the spindles centerline, while the Z axis moves the tool spindle down toward or up and away from the machine table. The "machine zero" position is where the spindle is pointing down at the upper right corner, with the machine table all the way to the left in the X axis and all the way toward you in the Y axis and Z axis is up at the tool change position. Motion in the X axis will move the machine table to the right with negative values and to the left with positive values. The Y axis will move the machine table toward you with positive values and away from you with negative values. Motion in the Z axis will move the tool toward the machine table with negative values and away from the machine table with positive values.

A program is written as a set of instructions given in the order they are to be performed. The instructions, if given in English, might look like this:

LINE #1 = SELECT CUTTING TOOL.
LINE #2 = TURN SPINDLE ON AND SELECT THE RPM.
LINE #3 = RAPID TO THE STARTING POSITION OF THE PART.
LINE #4 = TURN COOLANT ON.
LINE #5 = CHOOSE PROPER FEED RATE AND MAKE THE CUT(S).
LINE #6 = TURN THE SPINDLE AND COOLANT OFF.
LINE #7 = RETURN TO CLEARANCE POSITION TO SELECT ANOTHER TOOL.

and so on. But our machine control understands only these messages when given in machine code, also referred to as G and M code programming. Before considering the meaning and the use of codes, it is helpful to lay down a few guidelines.



PROGRAM FORMAT

There is no positional requirement for the address codes. They may be placed in any order within the block. Each individual can format their programs many different ways. But, program format or program style is an important part of CNC machining. There are some program command formats that can be moved around, and some commands need to be a certain way, and there are some standard program rules that are just good to follow. The point is that a programmer needs to have an organized program format that's consistent and efficient so that any CNC machinist in your shop can understand it.

Some standard program rules to consider are:

Program X, Y and Z in alphabetical order on any block. The machine will read Z, X or Y in any order, but we want to be consistent. If more than one of X, Y or Z is on a line, they should be listed together and in order. Write X first, Y next, then Z.

You can put G and M codes anywhere on a line of code. But, in the beginning when N/C programming was being developed G codes had to be in the beginning of a line and M codes had to be at the end. And this rule, a lot of people still follow and is a good standard to continue.

Some CNC machines allow you to write more than one M code per line of code and some won't. On the HAAS, only one M code may be programmed per block and all M codes are activated or cause an action to occur after everything else on the line has been executed.

Program format is a series and sequence of commands that a machine may accept and execute. Program format is the order in which the machine code is listed in a program that consist of command words. Command words begin with a single letter and then numbers for each word. If it has a plus (+) value, no sign is needed. If it has a minus value, it must be entered with a minus (-) sign. If a command word is only a number and not a value, then no sign or decimal point is entered with that command. Program format defines the "language of the machine tool."

```
;  
N3 (SET DIA. OFFSET D03 AT .500) ;  
T3 M06 (½ DIA. 2 FLT END MILL) ;  
G90 G54 G00 X-2.3 Y2.3 S1400 M03 ;  
G43 H03 Z0.1 M08 ;  
G01 Z-0.625 F50. ;  
G41 Y2. D03 F11. ;  
X2. ;  
Y-2. ;  
X-2. ;  
Y2.25 ;  
G40 X-2.3 Y2.3 ;  
G00 Z1. M09 ;  
G28 G91 Y0. Z0. M05 ;
```



DEFINITIONS WITHIN THE FORMAT

1. **CHARACTER** : A single alphanumeric character value or the "+" and "-" sign.
2. **WORD** : A series of characters defining a single function such as a, "X" displacement, an "F" feedrate, or G and M codes. A letter is the first character of a word for each of the different commands. There may be a distance and direction defined for a word in a program. The distance and direction in a word is made up of a value, with a plus (+) or minus (-) sign. A plus (+) value is recognized if no sign is given in a word.
3. **BLOCK** : Series of words defining a single instruction. An instruction may consist of a single linear motion, a circular motion or canned cycle, plus additional information such as a feedrate or miscellaneous command (M-codes).
4. **POSITIVE SIGNS** : If the value following an address letter command such as A, B, C, I, J, K, R, U, V, W, X, Y, Z, is positive, the plus sign need not be programmed in. If it has a minus value it must be programmed in with a minus (-) sign.
5. **LEADING ZERO'S** : If the digits proceeding a number are zero, they need not be programmed in. The HAAS control will automatically enter in the leading zero's.
EXAMPLE: G0 for G00 and M1 for M01,
Trailing zeros must be programmed: M30 not M3, G70 not G7.
6. **MODAL COMMANDS** : Codes that are active for more than the line in which they are issued are called MODAL commands. Rapid traverse, feedrate moves, and canned cycles are all examples of modal commands. A NON-MODAL command which once called, are effective only in the calling block, and are then immediately forgotten by the control.
7. **PREPARATORY FUNCTIONS** : "G" codes use the information contained on the line to make the machine tool do specific operations, such as :
 - 1.) Move the tool at rapid traverse.
 - 2.) Move the tool at a feedrate along a straight line.
 - 3.) Move the tool along an arc at a feedrate in a clockwise direction.
 - 4.) Move the tool along an arc at a feedrate in a counterclockwise direction.
 - 5.) Move the tool through a series of repetitive operations controlled by "fixed cycles" such as, spot drilling, drilling, boring, and tapping.
8. **MISCELLANEOUS FUNCTIONS** : "M" codes are effective or cause an action to occur at the end of the block, and only one M code is allowed in each block of a program.
9. **SEQUENCE NUMBERS** : N1 thru N99999 in a program are only used to locate and identify a line or block and its relative position within a CNC program. A program can be with or without SEQUENCE NUMBERS. The only function of SEQUENCE NUMBERS is to locate a certain block or line within a CNC program.



AN EXAMPLE OF THE PROGRAM'S FIRST COUPLE OF LINES

The FIRST line or block in a program should be a tool number (T1) and a tool change (M06) command.

The SECOND line or block should contain an absolute (G90) command along with, a work offset (G54 is the default), part zero command . A rapid (G00) command to position to an X Y coordinate location, a spindle speed command (Snnnn), and a spindle ON clockwise command (M03), or you could have the spindle speed and clockwise command defined on a separate line.

The NEXT line or block contains a "Read tool length compensation" command (G43), a tool length offset register number (H01), a Z-axis positioning move (Z1.0), and an optional coolant ON command (M08).

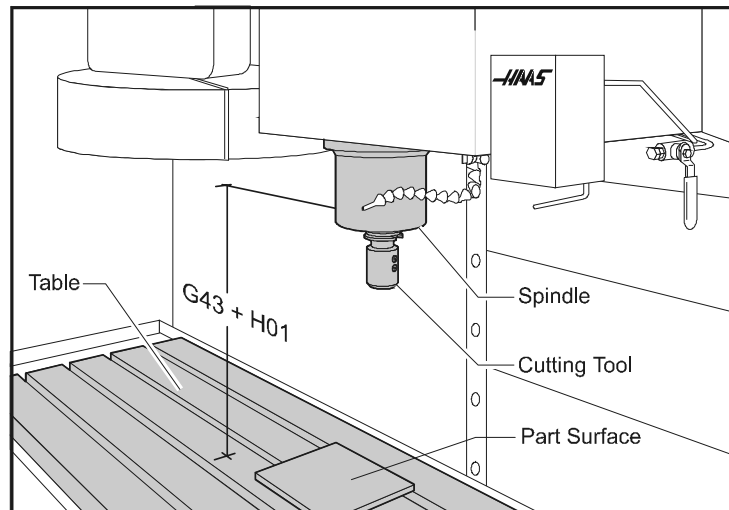
The tool start-up lines with the necessary codes for each tool are listed below. These formats are a good example for the start-up lines that are entered in for each tool.

```
T1 M06 (TEXT INFORMATION IN PARENTHESIS) ;
G90 G54 G00 X0.5 Y-1.5 S2500 M03 ;
G43 H01 Z1. M08 ;
```

Another format you might choose is:

```
M06 T1 (TEXT INFORMATION);
G00 G90 G54 X-1.5 Y2.5 ;
S2500 M03 ;
G43 Z1. H01 M08 ;
```

Note: A tool length offset number should usually always remain numerically matched with the tool number. Setting 15 (the H & T code agreement) will ensure the tool number and the tool length offset number will match. (Example: T1 in line #1 should have H01 in line #3 or an alarm will occur if Setting 15 is ON.)





OFTEN USED PREPARATORY "G" CODES

- G00 Rapid traverse motion; Used for non-cutting moves of the machine in positioning quick to a location to be machined, or rapid away after program cuts have been performed. Maximum rapid motion (I.P.M.) of a Haas machine will vary on machine model.
- G01 Linear interpolation motion; Used for actual machining and metal removal. Governed by a programmed feedrate in inches (or mm) per minute. Maximum feed rate (I.P.M.) of a Haas machine will vary on machine model.
- G02 Circular Interpolation, Clockwise
- G03 Circular Interpolation, Counterclockwise
- G28 Machine Home (Rapid traverse)
- G40 Cutter Compensation CANCEL
- G41 Cutter Compensation LEFT of the programmed path
- G42 Cutter Compensation RIGHT of the programmed path
- G43 Tool LENGTH Compensation +
- G53 Machine Coordinate Positioning, Non-Modal
- G54 Work Coordinate #1 (Part zero offset location)
- G80 Canned Cycle Cancel
- G81 Drill Canned Cycle
- G82 Spot Drill Canned Cycle
- G83 Peck Drill Canned Cycle
- G84 Tapping Canned Cycle
- G90 Absolute Programming Positioning
- G91 Incremental Programming Positioning
- G98 Canned Cycle Initial Point Return
- G99 Canned Cycle Rapid (R) Plane Return



PREPARATORY "G" CODES LIST

- 1) G Codes come in groups. Each group of G codes will have a specific group number.
- 2) A G code from the same group can be replaced by another G code in the same group. By doing this the programmer establishes modes of operation. The universal rule here, is that codes from the same group cannot be used more than once on the same line.
- 3) There are Modal G codes (All G-Codes except for Group 00) which once established, remain effective until replaced with another G code from the same group.
- 4) There are Non-Modal G codes (Group 00) which once called, are effective only in the calling block, and are immediately forgotten by the control.

The rules above govern the use of the G codes used for programming the Haas Mill. The concept of grouping codes and the rules that apply will have to be remembered to effectively program the Haas Mill. The following is a list of Haas G codes. If there's a (Setting number) listed next to a G code, that setting will in some way relate to that G code. A single asterisk (*) indicates that it's the default G code in a group. A double asterisk (**) indicates that it is an available option.

The first group (Group 1) control the manner in which the machine moves. These moves can be programmed in either absolute or incremental. The codes are G00, G01, G02, and G03.

<u>Code</u>	<u>Group</u>	<u>Function</u>
G00*	01	Rapid Positioning Motion (Setting 10, 56, 101)
G01	01	Linear Interpolation Motion
G02	01	Circular Interpolation Motion CW
G03	01	Circular Interpolation Motion CCW
G04	00	Dwell
G09	00	Exact Stop, Non-Modal
G10	00	Programmable Offset Setting
G12	00	Circular Pocket Milling CW
G13	00	Circular Pocket Milling CCW
G17*	02	Circular Motion XY Plane Selection (Setting 56)
G18	02	Circular Motion ZX Plane Selection
G19	02	Circular Motion YZ Plane Selection
G20*	06	Verify Inch Coordinate Positioning (Setting 9 will need to be INCH) (Setting 56)
G21	06	Verify Metric Coordinate Positioning (Setting 9 will need to be METRIC)
G28	00	Machine Zero Return Thru Reference Point (Setting 108)
G29	00	Move to location Thru G28 Reference Point
G31**	00	Feed Until Skip Function

(G codes continued next page)



<u>Code</u>	<u>Group</u>	<u>Function</u>
G35**	00	Automatic Tool Diameter Measurement
G36**	00	Automatic Work Offset Measurement
G37**	00	Automatic Tool Offset Measurement
G40*	07	Cutter Compensation Cancel G41/G42/G141 (Setting 56)
G41	07	2D Cutter Compensation Left (Setting 43, 44, 58)
G42	07	2D Cutter Compensation Right (Setting 43, 44, 58)
G43	08	Tool Length Compensation + (Setting 15)
G44	08	Tool Length Compensation - (Setting 15)
G47	00	Text Engraving (Macro Variable #599 to Change Serial Number)
G49*	08	Tool Length Compensation Cancel G43/G44/G143 (Setting 56)
G50*	11	Scaling G51 Cancel (Setting 56)
G51**	11	Scaling (Setting 71)
G52	12	Select Work Coordinate System G52 (Setting 33, YASNAC)
G52	00	Global Work Coordinate System Shift (Setting 33, FANUC)
G52	00	Global Work Coordinate System Shift (Setting 33, HAAS)
G53	00	Machine Coordinate Positioning, Non-Modal
G54*	12	Work Offset Positioning Coordinate #1 (Setting 56)
G55	12	Work Offset Positioning Coordinate #2
G56	12	Work Offset Positioning Coordinate #3
G57	12	Work Offset Positioning Coordinate #4
G58	12	Work Offset Positioning Coordinate #5
G59	12	Work Offset Positioning Coordinate #6
G60	00	Uni-Directional Positioning (Setting 35)
G61	13	Exact Stop, Modal
G64*	13	Exact Stop G61 Cancel (Setting 56)
G65**	00	Macro Sub-Routine Call
G68**	16	Rotation (Setting 72, 73)
G69*	16	Rotation G68 Cancel (Setting 56)
G70	00	Bolt Hole Circle with a Canned Cycle
G71	00	Bolt Hole Arc with a Canned Cycle
G72	00	Bolt Holes Along an Angle with a Canned Cycle
G73	09	High Speed Peck Drill Canned Cycle (Setting 22)
G74	09	Reverse Tapping Canned Cycle (Setting 130)
G76	09	Fine Boring Canned Cycle (Setting 27)
G77	09	Back Bore Canned Cycle (Setting 27)
G80*	09	Cancel Canned Cycle (Setting 56)
G81	09	Drill Canned Cycle
G82	09	Spot Drill / Counterbore Canned Cycle
G83	09	Peck Drill Deep Hole Canned Cycle (Setting 22, 52)
G84	09	Tapping Canned Cycle (Setting 130)
G85	09	Bore in~Bore out Canned Cycle
G86	09	Bore in~Stop~Rapid out Canned Cycle

(G codes continued next page)



<u>Code</u>	<u>Group</u>	<u>Function</u>
G87	09	Bore in~Manual Retract Canned Cycle
G88	09	Bore~Dwell~Manual Retract Canned Cycle
G89	09	Bore~Dwell~Bore out Canned Cycle
G90*	03	Absolute Positioning Command (Setting 56)
G91	03	Incremental Positioning Command (Setting 29)
G92	00	Set Work Coordinate Value (Fanuc) (HAAS)
G92	00	Global Work Coordinate System Shift (Yasnac)
G93	05	Inverse Time Feed Mode ON
G94*	05	Inverse Time Feed Mode OFF/Feed Per Minute ON (Setting 56)
G98*	10	Canned Cycle Initial Point Return (Setting 56)
G99	10	Canned Cycle "R" Plane Return
G100	00	Mirror Image Cancel
G101	00	Mirror Image (Setting 45, 46, 47, 48, 80)
G102	00	Programmable Output to RS-232
G103	00	Limit Block Look-a-head (P0-P15 for number of lines)
G107	00	Cylindrical Mapping
G110	12	Work Offset Positioning Coordinate #7
G111	12	Work Offset Positioning Coordinate #8
G112	12	Work Offset Positioning Coordinate #9
G113	12	Work Offset Positioning Coordinate #10
G114	12	Work Offset Positioning Coordinate #11
G115	12	Work Offset Positioning Coordinate #12
G116	12	Work Offset Positioning Coordinate #13
G117	12	Work Offset Positioning Coordinate #14
G118	12	Work Offset Positioning Coordinate #15
G119	12	Work Offset Positioning Coordinate #16
G120	12	Work Offset Positioning Coordinate #17
G121	12	Work Offset Positioning Coordinate #18
G122	12	Work Offset Positioning Coordinate #19
G123	12	Work Offset Positioning Coordinate #20
G124	12	Work Offset Positioning Coordinate #21
G125	12	Work Offset Positioning Coordinate #22
G126	12	Work Offset Positioning Coordinate #23
G127	12	Work Offset Positioning Coordinate #24
G128	12	Work Offset Positioning Coordinate #25
G129	12	Work Offset Positioning Coordinate #26
G136**	00	Automatic Work Offset Center Measurement
G141	07	3D+ Cutter Compensation
G143**	08	5 Axis Tool Length Compensation+ (Setting 117)
G150	00	General Purpose Pocket Milling
G153**	09	5 Axis High Speed Peck Drill Canned Cycle (Setting 22)
G154	09	Select Work Offset Positioning Coordinate P1-99

(G codes continued next page)



<u>Code</u>	<u>Group</u>	<u>Function</u>
G155**	09	5 Axis Reverse Tapping Canned Cycle
G161**	09	5 Axis Drill Canned Cycle
G162**	09	5 Axis Spot Drill/Counterbore Canned Cycle
G163**	09	5 Axis Peck Drill Canned Cycle (Setting 22)
G164**	09	5 Axis Tapping Canned Cycle
G165**	09	5 Axis Bore in, Bore out Canned Cycle
G166**	09	5 Axis Bore in, Stop, Rapid out Canned Cycle
G169**	09	5 Axis Bore, Dwell, Bore out Canned Cycle
G174	00	Special Purpose Non-Vertical Rigid Tapping CCW
G184	00	Special Purpose Non-Vertical Rigid Tapping CW
G187	00	Accuracy Control for High Speed Machining
G188	00	Get Program From PST (Program Schedule Table)

*Defaults

** Optional

Each G code defined in this control is part of a group of G codes. The Group 0 codes are non-modal; that is, they specify a function applicable to that block only and do not affect other blocks. The other groups are modal and the specification of one code in the group cancels the previous code applicable from that group. A modal G code applies to all subsequent blocks so those blocks do not need to re-specify the same G code.

There is also one case where the Group 01 G codes will cancel the Group 9 (canned cycles) codes. If a canned cycle is active, the use of G00 or G01 will cancel the canned cycle.



MACHINE DEFAULTS

A default is an automatic function of the machine tool control. After powering up the machine, the control will recognize the default "G" code values. The machine will go to the part zero that was entered in for G54 if no other work coordinate code was specified in the actual program, because the machine automatically recognizes the G54 column upon start-up. That is a default.

The control automatically recognizes these G codes when your HAAS mill is powered up:

G00	Rapid Traverse
G17	X,Y Circular Plane Selection
G20	Verify Inch (Setting 9 will need to be on INCH)
G40	Cutter Compensation Cancel
G49	Tool length Compensation Cancel
G50	G51 Cancel
G54	Work Coordinate Zero #1 (1 of 26 available)
G64	Exact Stop Cancel
G69	G68 Cancel (optional)
G80	Canned Cycle Cancel
G90	Absolute Programming
G94	Inverse Time Feed Deactivate
G98	Initial Point Return

There is no default feed rate (F code) or spindle speed (S code) , but once an F or S code is programmed, it will apply until another feed rate or spindle speed is entered or the machine is turned off.



OFTEN USED MISCELLANEOUS "M" CODES

- M00 The M00 code is used for a Program Stop command on the machine. It stops the spindle, turns off coolant and stops look-a-head processing. Pressing CYCLE START again will continue the program on the next block of the program.
- M01 The M01 code is used for an Optional Program Stop command. Pressing the OPT STOP key on the control panel signals the machine to perform a stop command when the control reads an M01 command. It will then perform like an M00.
- M03 Starts the spindle CLOCKWISE. Must have a spindle speed defined.
- M04 Starts the spindle COUNTERCLOCKWISE. Must have a spindle speed defined.
- M05 STOPS the spindle.
- M06 Tool change command along with a tool number will execute a tool change for that tool. This command will automatically stop the spindle, Z-axis will move up to the machine zero position and the selected tool will be put in spindle. The coolant pump will turn off right before executing the tool change.
- M08 Coolant ON command.
- M09 Coolant OFF command.
- M30 Program End and Reset to the beginning of program.
- M97 Local Subroutine call
- M98 Subprogram call
- M99 Subprogram return (M98) or Subroutine return (M97), or a Program loop.

NOTE: Only one "M" code can be used per line. And the M-codes will be the last command to be executed in a line, regardless of where it's located in that line.



MISCELLANEOUS "M" CODES LIST

All M codes are activated or cause an action to occur after everything else on a block has been completed. And only one M code is allowed per block in a program. If there is a (Setting number) listed next to an M code, that setting will in some way relate to that M code. The following list is a summary of Haas M codes. A * indicates options available.

- M00 Program Stop
- M01 Optional Program Stop (Setting 17)
- M02 Program End
- M03 Spindle On, Clockwise (S)
- M04 Spindle On, Counterclockwise (S)
- M05 Spindle Stop
- M06 Tool Change (T) (Setting 42, 87)
- M08 Coolant On (Setting 32)
- M09 Coolant Off
- M10** 4th Axis Brake On
- M11 4th Axis Brake Release
- M12** 5th Axis Brake On
- M13 5th Axis Brake Release
- M19 Orient Spindle (P,R)
- M21-M28 Optional User M Code Interface with M-Fin Signals
- M30 Program End and Reset (Setting 2, 39, 56, 83)
- M31 Chip Conveyor Forward (Setting 114,115)
- M32 Chip Conveyor Reverse (Setting 114, 115)
- M33 Chip Conveyor Stop
- M34 Coolant Spigot Position Down, Increment
- M35 Coolant Spigot Position Up, Decrement
- M36** Pallet Part Ready
- M39 Rotate Tool Turret (T) (Setting 86)
- M41 Spindle Low Gear Override
- M42 Spindle High Gear Override
- M50** Execute Pallet Change
- M51-M58 Optional User M Code Set
- M59 Output Relay Set
- M61-M68 Optional User M Code Clear
- M69 Output Relay Clear
- M75 Set G35 or G136 Reference Point
- M76 Control Display Inactive
- M77 Control Display Active
- M78 Alarm if Skip Signal Found
- M79 Alarm if Skip Signal Not Found
- M80** Automatic Door Open (Setting 131)



(M codes continued next page)

- M81 Automatic Door Close
- M82 Tool Unclamp
- M83** Auto Air Jet On
- M84 Auto Air Jet Off
- M86 Tool Clamp
- M88** Coolant Through the Spindle On (Setting 32)
- M89 Coolant Through the Spindle Off
- M93 Start Axis POS Capture
- M94 Stop Axis POS Capture
- M95 Sleep Mode
- M96 Jump if No Input (P, Q)
- M97 Local Sub-Routine Call (P, L)
- M98 Sub Program Call (P, L)
- M99 Sub Program Return or Loop (Setting 118)

** Options



PROGRAM STRUCTURE

A CNC part program consists of one or more blocks of commands. When viewing the program, a block is the same as a line of text. Blocks shown on the CRT are always terminated by the ";" symbol which is called an End Of Block (EOB). Blocks are made up of alphabetical address codes which are always an alphabetical character followed by a numeric value. For instance, the specification to move the X-axis would be a number preceded by the X symbol.

Programs must begin and end with a percent (%) sign. After the first percent (%) sign, the program must have a program number beginning with the letter O and then the number that defines that program (four digit number for older machines and five digit number for newer machines). Those numbers are used to identify and select a main program to be run, or as a subprogram called up by the main program. The % will "not" be seen on the control, but they must be there when you load a program into the control. And they will be seen when you download a program from the machine. The % signs are automatically entered in for you, if you entered the program in on the HAAS control.

A program may also contain a "/" symbol. The "/" symbol, sometimes called a slash, is used to define an optional block. If a block contains this symbol, any information that follows the slash in a program block, will be ignored when the BLOCK DELETE button is selected when running a program.

On the following page is a sample program as it would appear on the control screen. The words following the ":" are not part of the actual program but are put there as further explanation.

This program will drill four holes and mill a two-inch hole in a four-inch square plate with X and Y zero at the center. The program with comment statements would appear like this.



%	:PROGRAMS MUST BEGIN AND END WITH % AND WILL NOT BE SEEN IN PROGRAM DISPLAY
O00023 ;	:LETTER "O" AND UP TO A FIVE DIGIT PROGRAM NUMBER
(MILL PART PROGRAM EXAMPLE) ;	:(COMMENTS IN PARENTHESIS ARE IGNORED BY CONTROL)
N1 (DRILL 4 PLACES) ;	:FIRST OPERATION , (NOTES TO OPERATOR)
T1 M06 (½ DIA. STUB DRILL) ;	:TOOL CHANGE TO TOOL #1, (NOTES TO OPERATOR)
G90 G54 G00 X-1.5 Y1.5 S1400 M03 ;	:ABS POSIT, WORK OFFSET#, RAPID X Y, SPINDLE ON CW
G43 H01 Z1. M08 ;	:TOOL LENGTH COMP #1, Z POSITION, COOLANT ON
G73 G99 Z-0.625 Q0.2 R0.1 F5. ;	:HIGH SPEED PECK DRILLING TO Z-.625 DEEP, .2 PECK
Y-1.5 ;	:DRILL ANOTHER HOLE RAPID PLANE IS AT R.1
/ X1.5 ;	:DRILL THIRD HOLE WITH AN OPTIONAL BLOCK DELETE
/ Y1.5 ;	:DRILL FOURTH HOLE WITH OPTIONAL BLOCK DELETE
G80 G00 Z1. M09 ;	:CANCEL CANNED CYCLE, RAPID Z1. COOLANT OFF
G28 G91 Z0. M05 ;	:RETURN Z TO MACHINE ZERO, SPINDLE OFF
;	
N2 (COUNTERSINK 4 PLACES) ;	:(COMMENTS IN PARENTHESIS ARE IGNORED BY CONTROL)
T2 M06 (5/8 DIA. 90 DEG. C'SINK) ;	:TOOL CHANGE TO TOOL #2, (NOTES TO OPERATOR)
G90 G54 G00 X-1.5 Y1.5 S900 M03 ;	:ABS POSIT, WORK OFFSET#, RAPID X Y, SPINDLE ON CW
G43 H02 Z1. M08 ;	:TOOL LENGTH COMP #2, Z POSITION, COOLANT ON
G82 G99 Z-0.27 P0.5 R0.1 F12. ;	:SPOT DRILL CYCLE TO Z-.27 DEEP, DWELL .5 SECOND
Y-1.5 ;	:SECOND HOLE , RAPID PLANE IS AT R.1
/ X1.5 ;	:THIRD HOLE WITH AN OPTIONAL BLOCK DELETE
/ Y1.5 ;	:FOURTH HOLE WITH AN OPTIONAL BLOCK DELETE
G80 G00 Z1. M09 ;	:CANCEL CANNED CYCLE, RAPID Z1., COOLANT OFF
G28 G91 Z0. M05 ;	:RETURN Z TO MACHINE ZERO, SPINDLE OFF
M00 (ADD BOLTS AND REMOVE CLAMPS) ;	:PROGRAM STOP COMMAND TO PERFORM A TASK
;	
N3 (SET DIA. OFFSET D03 AT .500) ;	:(COMMENTS IN PARENTHESIS ARE IGNORED BY CONTROL)
T3 M06 (½ DIA. 4 FLT END MILL) ;	:TOOL CHANGE TO TOOL #3 (NOTES TO OPERATOR)
G90 G54 G00 X-2.3 Y2.3 S1604 M03 ;	:ABS POSIT, WORK OFFSET#, RAPID X Y, SPINDLE ON CW
G43 H03 Z0.1 M08 ;	:TOOL LENGTH COMP #3, Z POSITION, COOLANT ON
G01 Z-0.625 F50. ;	:FAST FEED TO DEPTH
G41 Y2. D03 F16. ;	:CUTTER COMP. LEFT OF LINE WITH DIA. COMP D03
X2.0 ;	:CUT A 4.0 IN. SQUARE
Y-2.0 ;	: " " "
X-2.0 ;	: " " "
Y2.25 ;	: " " "
G40 X-2.3 Y2.3 ;	:G40 CANCELS CUTTER COMP MOVING AWAY FROM PART
G00 Z1. M09 ;	:RAPID Z1., COOLANT OFF
G28 G91 Y0. Z0. M05 ;	:RETURN Y AND Z TO MACHINE ZERO
T1 M06 ;	:TOOL CHANGE BACK TO TOOL #1
M30 ;	:PROGRAM STOP AND RETURN TO BEGINNING
%	:PROGRAM BEGINS AND ENDS WITH %

To change tools, all that is needed is an M06 even without a G28 in the previous line. A G28 can be specified to send all axes to machine home, or it can be defined to send a specific axis home with G28 G91 Z0 and/or Y0 and/or X0 to send just these axis specified to home position.



ALPHABETICAL ADDRESS CODES

The following is a list of the Address Codes used in programming the Mill.

A FOURTH AXIS ROTARY MOTION (Setting 30, 34, 48, 108)

The A address character is used to specify motion for the optional fourth, A, axis. It specifies an angle in degrees for the rotary axis. It is always followed by a signed number and up to three fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/1000 degrees.

Setting 30 - 4TH AXIS ENABLE - When this setting is off, it disables the 4th axis and no commands can be sent to that axis. When it is on, it is selected to one of the rotary table types to choose from in this setting. In order to change this setting the servos must be turned off (Emergency Stop in).

Setting 34 - 4TH AXIS DIAMETER - This is a numeric entry. When this setting is set correctly, the surface feed rate, on the entered in diameter for the rotary cut will be exactly the feed rate programmed into the control.

B FIFTH AXIS ROTARY MOTION (Setting 78, 79, 80,108)

The B address character is used to specify motion for the optional fifth, B, axis. It specifies an angle in degrees or the rotary axis. It is always followed by a signed number and up to three fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/1000 degrees.

Setting 78 - 5TH AXIS ENABLE - When this setting is off, it disables the 4th axis and no commands can be sent to that axis. When it is on, it is selected to one of the rotary table types to choose from in this setting. In order to change this setting the servos must be turned off (Emergency Stop in).

Setting 79 - 5TH AXIS DIAMETER - This is a numeric entry. When this setting is set correctly, the surface feed rate, on the entered in diameter for the rotary cut will be exactly the feed rate programmed into the control.

C AUXILIARY EXTERNAL ROTARY AXIS (Setting 38)

The C address character is used to specify motion for the optional external sixth, C, axis. It specifies an angle in degrees for the rotary axis. It is always followed by a signed number and up to three fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/1000 degrees.

Setting 38 - AUX AXIS NUMBER - This is a numeric entry between 0 and 4. It is used to select the number of external auxiliary axes added to the system.

D TOOL DIAMETER OFFSET SELECTION (Setting 40, 43, 44, 58)

The D address character is used to select the tool diameter or radius used for cutter compensation. The number following must be between 0 and 200 (100 programs on an older machine). The Dnn selects that number offset register, that is in the offset display, which contains the tool diameter/radius offset amount when using cutter compensation (G41 G42). D00 will cancel cutter compensation so that the tool size is zero and it will cancel any previously defined Dnn.

Setting 40 - TOOL OFFSET MEASURE - Selects how the tool size is specified for cutter compensation, Radius or Diameter.



E ENGRAVING FEED RATE or CONTOURING ACCURACY (Setting 85)

The E address character is used, with G187, to select the accuracy required when cutting a corner during high speed machining operations. The range of values possible is 0.0001 to 0.25 for the E code. Refer to the "Contouring Accuracy" section of your machine manual for more information.

Setting 85 - Is also used to designate the same condition for Contouring Accuracy.

F FEED RATE (Setting 19, 77)

The F address character is used to select the feed rate applied to any interpolation functions, including pocket milling and canned cycles. It is either in inches per minute with four fractional positions or mm per minute with three fractional positions.

Setting 77 - Allows the operator to select how the control interprets an F address code that does not contain a decimal point, (It is recommended that the programmer always use a decimal point).

G PREPARATORY FUNCTIONS (G codes)

The G address character is used to specify the type of operation to occur in the block containing the G code. The G is followed by a two or three digit number between 0 and 187. Each G code defined in this control is part of a group of G codes. The Group 0 codes are non-modal; that is, they specify a function applicable to this block only and do not effect other blocks. The other groups are modal and the specification of one code in the group cancels the previous code applicable from that group. A modal G code applies to all subsequent blocks so those blocks do not need to re-specify the same G code. More than one G code can be placed in a block in order to specify all of the setup conditions for an operation.

H TOOL LENGTH OFFSET SELECTION (Setting 15)

The H address character is used to select the tool length offset entry from the offsets memory. The H is followed by a two digit number between 0 and 200 (100 programs on an older machine). H0 will clear any tool length offset and Hnn will use the tool length entered in on n from the Offset display. You must select either G43 or G44 to activate a tool length (H) offsets. The G49 command is the default condition and this command will clear any tool length offsets. A G28, M30 or pressing Reset will also cancel tool length offsets.

Setting 15 - When this setting is on, a check is made to ensure that the H offset code matches the tool presently in the spindle. This check can help prevent crashes.

I CIRCULAR INTERPOLATION or CANNED CYCLE DATA

The I address character is used to specify data for either canned cycles or circular motions. It is defined in inches with four fractional positions or mm with three fractional positions.

J CIRCULAR INTERPOLATION or CANNED CYCLE DATA

The J address character is used to specify data for either canned cycles or circular motions. It is defined in inches with four fractional positions or mm with three fractional positions.

K CIRCULAR INTERPOLATION or CANNED CYCLE DATA

The K address character is used to specify data for either canned cycles or circular motions. It is defined in inches with four fractional positions or mm with three fractional positions.



L LOOP COUNT TO REPEAT A COMMAND LINE

The L address character is used to specify a repeat count for some canned cycles and auxiliary functions. It is followed by a number between 0 and 32767.

M M CODE MISCELLANEOUS FUNCTIONS

The M address character is used to specify an M code. These codes are used to control miscellaneous machine functions. Note that only one M code is allowed per block in a CNC program and all M codes are performed secondary in a block.

N NUMBER OF BLOCK

The N address character is entirely optional. It can be used to identify or number each block of a program. It is followed by a number between 0 and 99999. The M97 functions needs to reference an N line number.

O PROGRAM NUMBER (PROGRAM name in parenthesis)

The O address character is used to identify a program. It is followed by a number between 0 and 99999. A program saved in memory always has a Onnnnn identification in the first block. Altering the Onnnnn in the first block causes the program to be renumbered. If you enter a program name (Program Text Name) between parenthesis in the first three lines of a program, that program name will also be seen in your list of programs. You can have up to 500 program numbers (200 programs on an older machine) in your List of Programs. You can delete a program number from the LIST PROG display, by cursor selecting the program, and pressing the ERASE PROG key. You can also delete a program in the advanced editor using the menu item DELETE PROGRAM FROM LIST.

P DELAY OF TIME or M98 PROGRAM NUMBER Call or M97 SEQUENCE NUMBER Call

The P address character is used for either a dwell time in seconds with some canned cycles or in a G04, or P is used as a program number reference for an M98, or a program sequence number reference for an M97. The P value is a positive number without decimal point up to 99999 when used with an M98 or M97. When used as a dwell time, it may be a positive decimal number between 0.001 and 1000.0 in seconds.

Q CANNED CYCLE OPTIONAL DATA

The Q address character is used in canned cycles and is always a positive number in inches between 0.001 and 100.0.

R CIRCULAR INTERPOLATION or CANNED CYCLE DATA (Setting 52)

The R address character is used in canned cycles or circular interpolation. It's either in inches with four fractional positions or mm with three fractional positions. It is followed by number in inches or metric. It's usually used to define the reference plane for canned cycles.

S SPINDLE SPEED COMMAND (Setting 20)

The S address character is used to specify the spindle speed in conjunction with M41 and M42. The S is followed by an unsigned number between 1 - 99999. The S command does not turn the spindle on or off; it only sets the desired speed. If a gear change is required in order to set the commanded speed, this command will cause a gear change to occur



even if the spindle is stopped. If spindle is running, a gear change operation will occur and the spindle will start running at the new speed.

T TOOL SELECTION CODE (Setting 15)

The T address character is used to select the tool for the next tool change. The number following must be a positive number between 1 and (20) the number in Parameter 65. It does not cause the tool change operation to occur. The Tnn may be placed in the same block that starts tool change (M06 or M16) or in any previous block.

U AUXILIARY EXTERNAL LINEAR AXIS

The U address character is used to specify motion for the optional external linear, U-axis. It specifies a position of motion in inches. It is always followed by a signed number and up to four fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/10000 inches. The smallest magnitude is 0.0001 inches, the most negative value is -8380.0000 inches, and the largest number is 8380.0000 inches.

V AUXILIARY EXTERNAL LINEAR AXIS

The V address character is used to specify motion for the optional external linear, V-axis. It specifies a position of motion in inches. It is always followed by a signed number and up to four fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/10000 inches.

W AUXILIARY EXTERNAL LINEAR AXIS

The W address character is used to specify motion for the optional external linear, W-axis. It specifies a position of motion in inches. It is always followed by a signed number and up to four fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/10000 inches.

X LINEAR X-AXIS MOTION (Setting 45)

The X address character is used to specify motion for the X-axis. It specifies a position or distance along the X-axis. It is either in inches with four fractional positions or mm with three fractional positions. It is followed by a signed number in inches or metric. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/1000 mm.

Y LINEAR Y-AXIS MOTION (Setting 46)

The Y address character is used to specify motion for the Y-axis. It specifies a position or distance along the Y-axis. It is either in inches with four fractional positions or mm with three fractional positions. It is followed by a signed number in inches or metric. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/1000 mm.

Z LINEAR Z-AXIS MOTION (Setting 47)

The Z address character is used to specify motion for the Z-axis. It specifies a position or distance along the Z-axis. It is either in inches with four fractional positions or mm with three fractional positions. It is followed by a signed number in inches or metric. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/1000 mm.

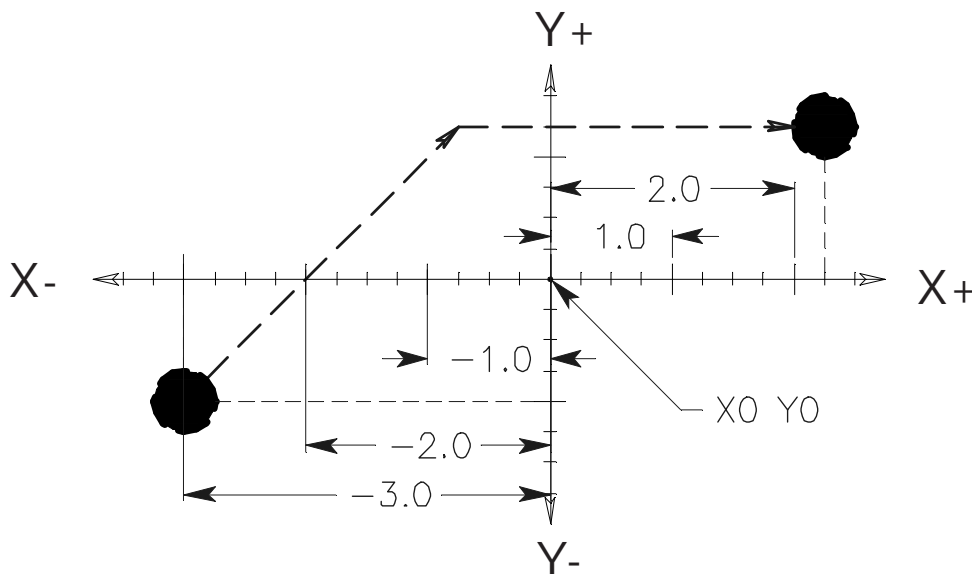


RAPID POSITION COMMAND

G00 RAPID POSITIONING MOTION

- *X Positioning X axis motion
- *Y Positioning Y axis motion
- *Z Positioning Z axis motion
- *A Positioning A axis motion
- * indicates optional

This G code is for rapid traverse of the three or four axes of the machine. This G00 code is modal and causes all the following blocks to be in rapid (710 in./min.) motion until another Group 01 code is specified. Generally, rapid motions "will not" be in a straight line. All the axes specified are moved at the maximum speed and will not necessarily complete each axis move at the same time. It activates each axis drive motor independantly of each other and, as a result, the axis with the shortest move will reach its desination first. So you need to be careful of any obstructions to avoid with this type of rapid move. The tool will first move from the current position in a straight line along a 45 degree angle to an intermediate location when one of these axes has completed its move. Then the machine will position parallel to the X or Y axis to complete the move to the final location. If the Z axis is also in the program move, it will operate in the same manner along with the X and Y axes. Only the axes specified are moved and the commands for absolute (G90) or incremental (G91) will change how the values are interpreted.



ABSOLUTE POSITIONING - G90 G00 X2.25 Y1.25

or

INCREMENTAL POSITIONING - G91 G00 X5.25 Y2.25



LINEAR INTERPOLATION COMMAND

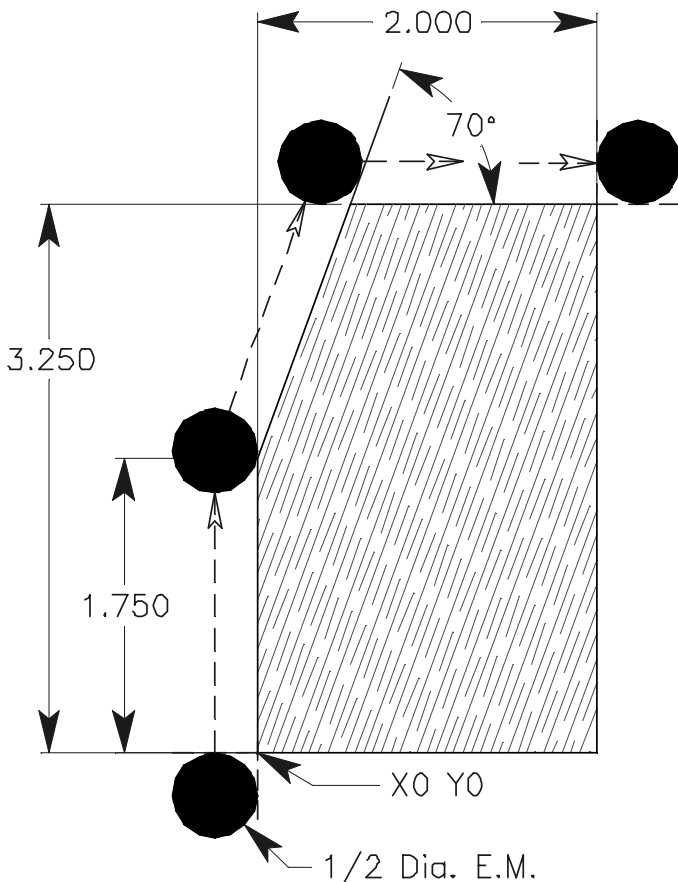
G01 LINEAR INTERPOLATION MOTION

- *X Linear X-axis motion
- *Y Linear Y-axis motion
- *Z Linear Z-axis motion
- *A Linear A-axis motion
- *F Feed rate in inches (mm) per minute
- * indicates optional

This G code provides for straight line (linear) motion from point to point. Motion can occur in 1, 2 or 3 axes. All axis specified will start at the same time and proceed to their destination and arrive simultaneously at the specified feedrate. The rotary axis may also provide motion around an axis or center point. The speed of all axes are controlled by a feedrate specified along with axis moves. Rotary axis feedrate is dependent on rotary axis diameter setting (Setting 34) and will provide a controlled motion. The F command is modal and may be specified in a previous block. Only the axes specified are moved in either absolute (G90) or incremental (G91) modal commands which change how values are interpreted.

Location are defined around part geometry using cutter comp.

```
(Absolute with Cutter Comp.)
G01 G41 X0. Y-0.25 D01 F12.
G90 Y1.75 (Absolute G90 Command)
X0.546 Y3.25
X2.
Y0.
X0.
G40 X-.35 Y-.25
      or
(Incremental with Cutter Comp.)
G01 G41 X0. Y-0.25 D01 F12.
G91 Y2. (Incremental G91 Command)
X0.546 Y1.5
X1.454
Y-3.25
X-2.
G40 X-.35 Y-.25
.....
```





CIRCULAR INTERPOLATION COMMANDS

G02 CW CIRCULAR INTERPOLATION MOTION &
G03 CCW CIRCULAR INTERPOLATION MOTION

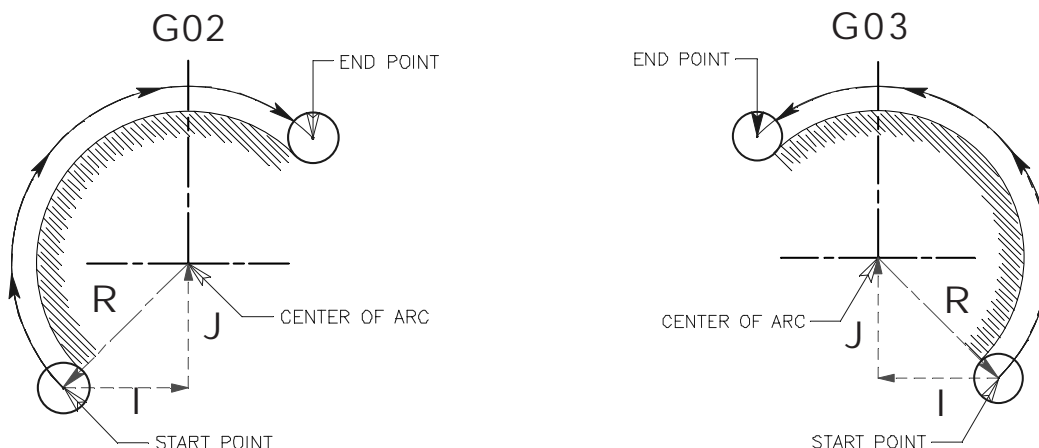
- *X Circular end point X-axis motion
- *Y Circular end point Y-axis motion
- *Z Circular end point Z-axis motion
- *A Circular end point A-axis motion
- *I X-axis Distance from start point to arc center (If R is not used)
- *J Y-axis Distance from start point to arc center (If R is not used)
- *K Z-axis Distance from start point to arc center (If R is not used)
- *R Radius of the arc to be machined (If I, J, K are not used)
- *F Feed rate in inches (or mm) per minute

* indicates optional

G03 will generate a counterclockwise circular motion, but is otherwise defined the same way as G02.

These G codes are used to specify a clockwise or counterclockwise motion of two, of the linear axes. Circular motion is possible in two of the three axes in either the X, Y, and/or Z axes as selected by G17, G18, and G19. The X, Y, and Z in a circular command (G02 or G03) is used to define the end point of that motion in either absolute (G90) or incremental (G91) motion. If any of the axes, X, Y, or Z for the selected plane is not specified, the endpoint location of the arc will then be recognized the same as the starting point of the arc, for that axis. There are two basic command formats for defining circular interpolation, depending on whether the IJK method or the R method is used to define the arc center.

Circular interpolation commands are used to move a tool along a circular arc to the commanded end position. Five pieces of information are required for executing a circular interpolation command:





The Five pieces of information for executing a circular interpolation command.

Item	Command	Remark
① Plane selection command	G17	Arc parrallel to XY-plane
Plane selection command	G18	Arc parrallel to ZX-plane
Plane selection command	G19	Arc parrallel to YZ-plane
② Arc start position coordinates	X,Y,Z	Coordinates of the start position
③ Rotation direction	G02	Clockwise direction
	G03	Counterclockwise direction
④ Arc end position (G90) Absolute or Arc end position (G91) Incremental	X,Y,Z	Coordinates of the end position on the work coordinate system
	X,Y,Z	Distance from start position to end position in X, Y, and Z axes, respectively
⑤ I J K method (arc center coordinate) or R method (arc radius)	I,J,K	Distance from start position to arc center in X, Y, and Z axes, respectively
	R	Arc radius value

There are two ways to specify the center of the circular arc that your rotating around. The first uses I, J, or K to specify the distance from the starting point to the center of the arc. The second uses R to specify the radius of the arc. These two ways will be further described below:

The "I", "J" and "K" values are "INCREMENTAL" distances from the tools starting point (START POINT) on arc to the ARC CENTER.

"I" = Incremental distance from Start Point to arc center in the "X axis."

"J" = Incremental distance from Start Point to arc center in the "Y axis."

"K" = Incremental distance from Start Point to arc center in the "Z axis."

R is easier to define, though it's easier to make a mistake and get an incorrect radius. If R is used, and you make a mistake with the start point, or the end point, and the machine can still do the radius, and does it, then you'll have an incorrect radius. If you make a mistake with the IJK method, machine will be more likely to stop and give you an alarm before executing it. Though, using an R in a G02 or G03 is still easier and the preffered method to manually program an arc.

R is the distance from the starting point to the center of the circle. With a positive R, the control will generate a circular path of 180 degrees or less, but to generate a circular path of over 180 degrees, then specify a negative R. When R is used, a complete 360 degree arc is not possible. X, Y, or Z is required to specify an endpoint different from the starting point. So anything under a 360 degree arc can be performed with an R command in a G02 or G03.



G02 CW CIRCULAR INTERPOLATION

Contouring a part profile using a circular interpolation command, uses the axis information contained in a block, to move the tool in a CLOCKWISE circular move, up to 360 degrees.

The velocity at which the tool is moving is controlled by the feedrate (F) command.

All circles are defined and machined by programming in three pieces of information to the control, they are :

START POINT of the arc

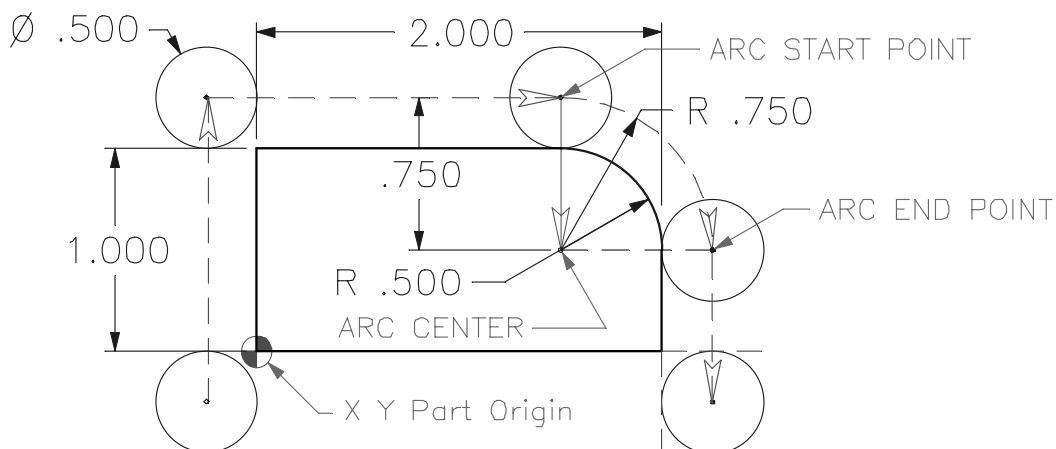
END POINT of the arc

ARC CENTER incremental distance from the start point to the arc center.

The START POINT is the point defined prior to the G02 line, usually by a G01 linear positioning move.

The END POINT is defined by the "X" and "Y" coordinates contained in the G02 line, when in the G17- XY PLANE.

The ARC CENTER is defined in the G02 line with the "I" for the Xaxis and "J" for the Y axis incremental distance and direction to the arc center, when in the G17- XY PLANE. Or by the "R" Radius value.



```
N6 G01 Y1.250 F12.  
N7 X1.500 (to start point)  
N8 G02 X2.250 Y.500 (I0. J-.750 or R.750)  
N9 G01 Y-.25
```

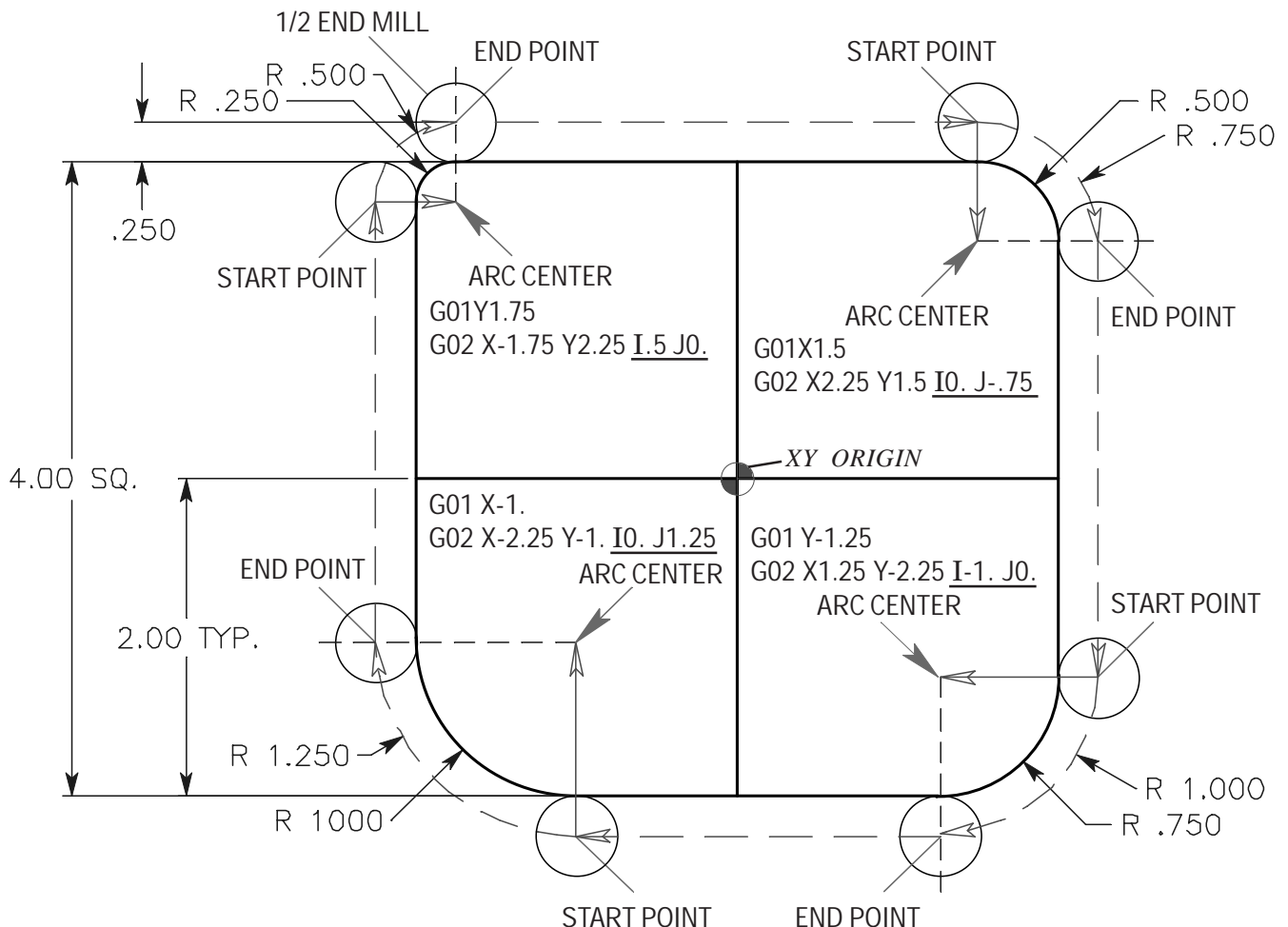



G02 CW CIRCULAR INTERPOLATION USING "I", & "J" FOR THE X & Y AXIS

The "I", "J" and "K" values are INCREMENTAL distances from where the tool starts cutting the arc (START POINT) to the ARC CENTER.

When I, J, or K are used to specify the center of the arc, R is not used. Only the I, J, or K specific to the selected plane (IJ for G17, IK for G18, JK for G19) are allowed. If only one of the I, J, K is specified, the others are assumed to be zero. The I, J, or K is the distance from the starting point to the center of the circle. Using I, J, or K is the only way to cut a complete 360 degree arc; in this case, the starting point is the same as the ending point and no X, Y, or Z is needed. To cut a complete circle of 360 degrees (360°), you do not need to specify an ending point X, Y, or Z; just program I, J, or K to define the center of the circle.

NOTE: Example of circular moves are not using cutter compensation, so the circular moves are defined from the center of the cutter around arc. For 90 degree corners or fillets the "I" and "J" values can be defined like you see below.





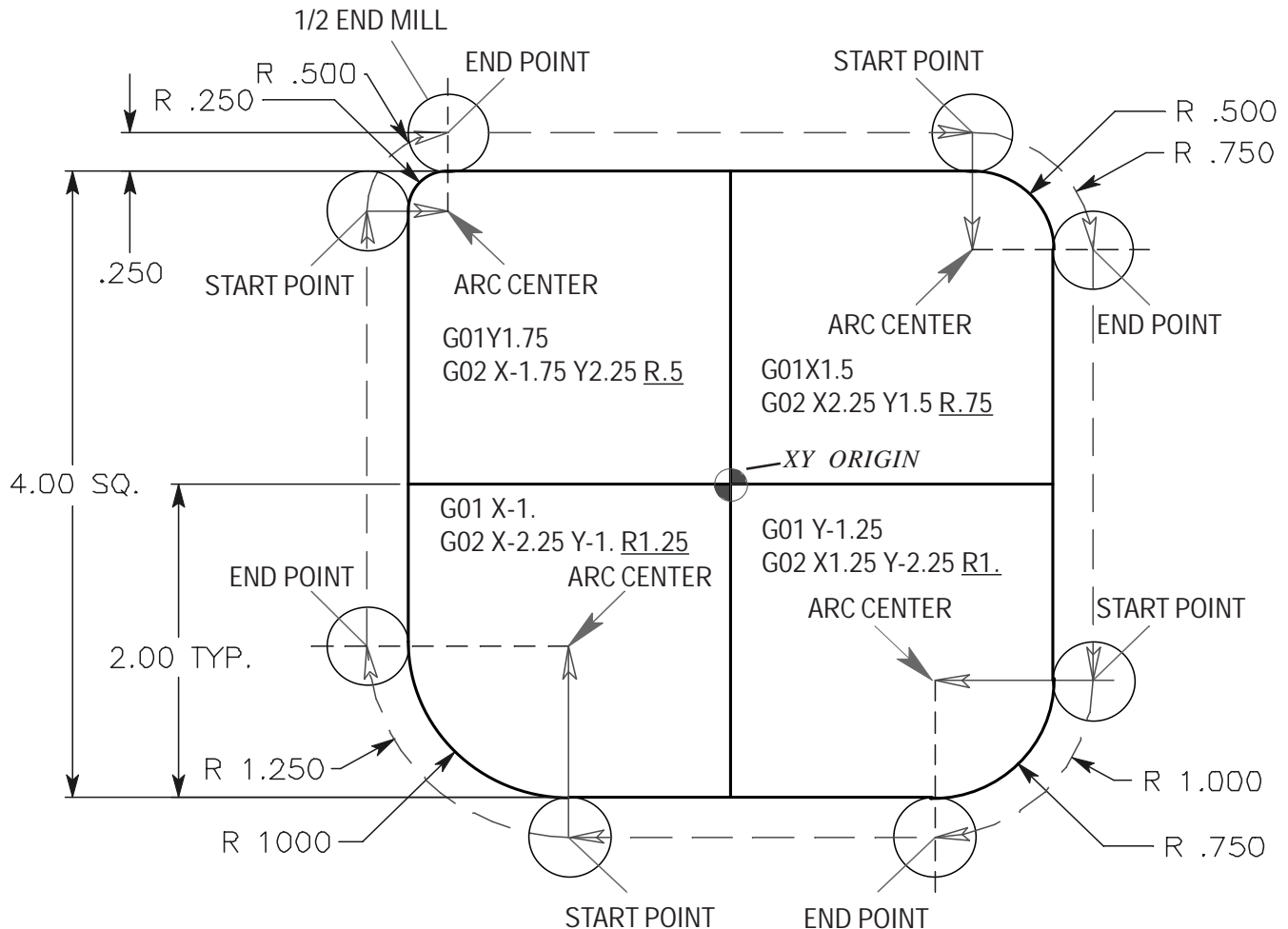
G02 CW CIRCULAR MOTION USING "R"

The letter address "R" can be substituted for the letter addresses "I", "J" and/or "K" for various corners or fillets and the "R" value is more easily defined.

Just as "I" "J" and "K", are used to locate the ARC CENTER in relation to the START POINT, "R" does the same function with less input.

When R is used to specify the center of the circle, a complete 360 degree arc is not possible. When defining an arc less then 360 degrees using the R command, the X, Y, or Z needs to specify an endpoint different from the starting point. And R defines the distance from the starting point to the center of the circle. With a positive R, the control will generate a radius of 180 degrees or less; to generate a radius of over 180 degrees, you need to specify an R command with a negative value.

NOTE: Example of circular moves are not using cutter compensation, so the circular moves are defined from the center of the cutter around arc. For 90 degree corners or fillets the "R" values can be defined like you see below.





G03 CCW CIRCULAR INTERPOLATION

Contouring a part profile using a circular interpolation command, uses the axis information contained in a block, to move the tool in a COUNTERCLOCKWISE circular move, up to 360 degrees.

The velocity at which the tool is moving is controlled by the feedrate (F) command.

All circles are defined and machined by programming in three pieces of information to the control, they are :

START POINT of the arc

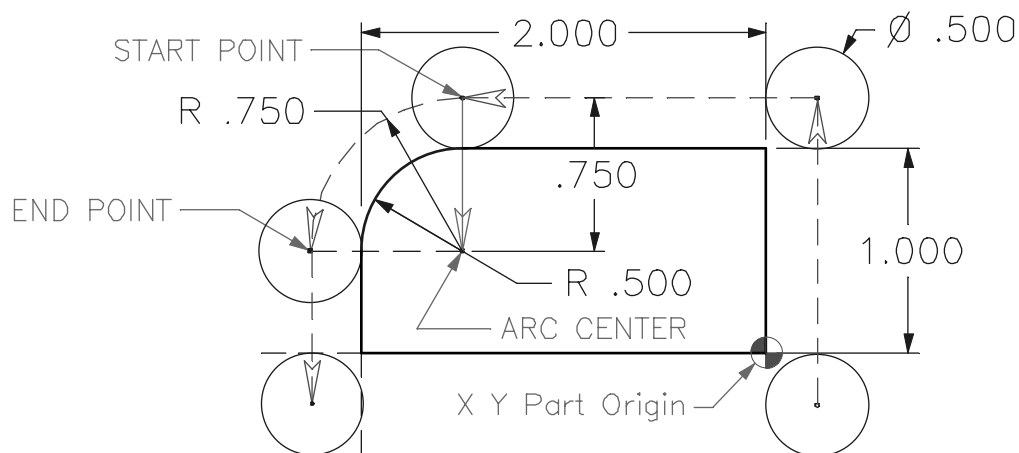
END POINT of the arc

ARC CENTER incremental distance from the start point to the arc center.

The START POINT is the point defined prior to the G03 line, usually by a G01 linear positioning move.

The END POINT is defined by the "X" and "Y" coordinates contained in the G03 line, when in the G17- XY PLANE.

The ARC CENTER is defined in the G03 line with the "I" for the Xaxis and "J" for the Y axis incremental distance and direction to the arc center, when in the G17- XY PLANE. Or by the "R" Radius value.



```

N6 G01 Y1.250 F12.
N7 X-1.500 (to start point)
N8 G03 X-2.250 Y.500 (I0. J-.750 or R.750)
N9 G01 Y-.25

```

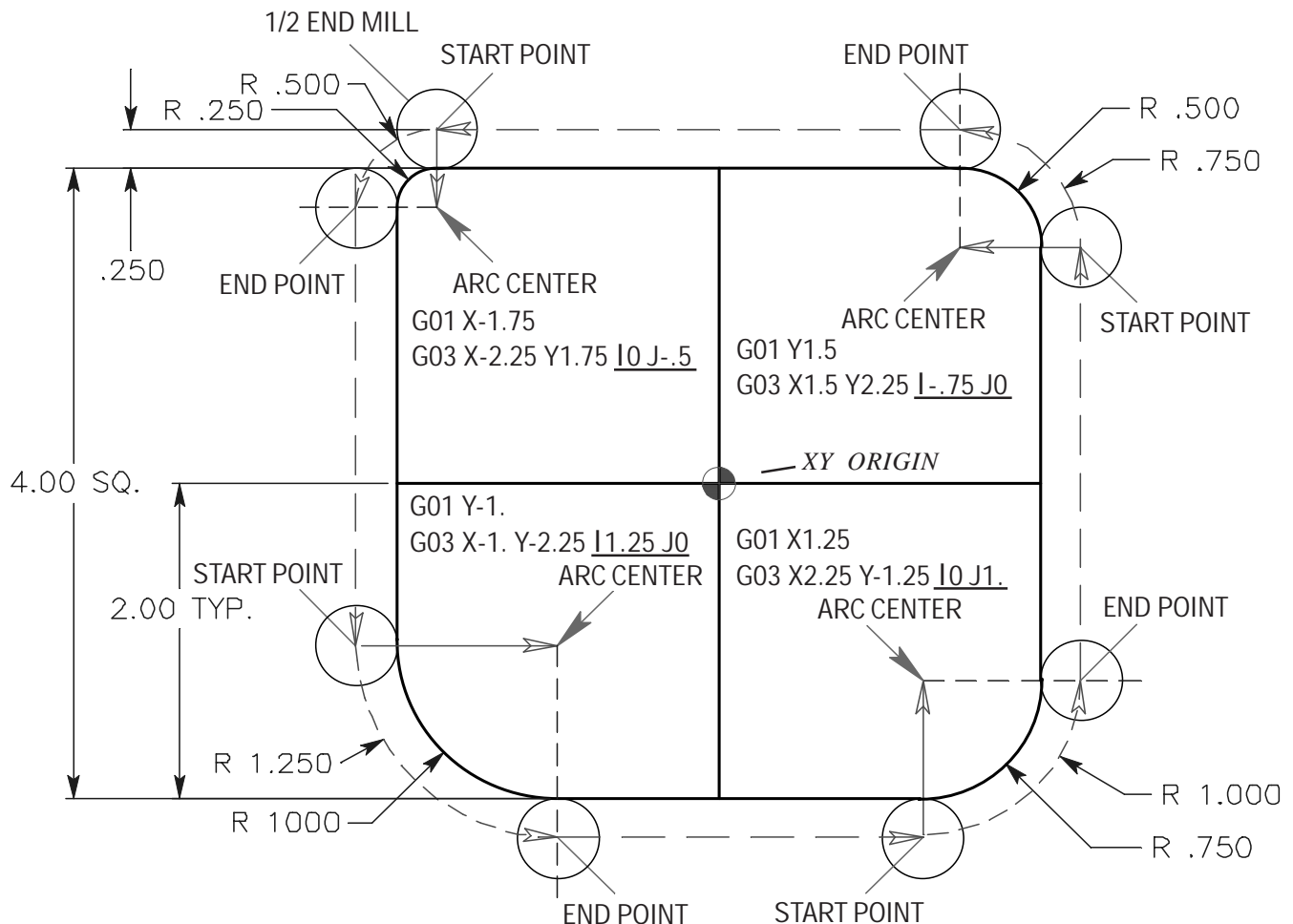


G03 CCW CIRCULAR INTERPOLATION USING "I", & "J" FOR THE X & Y AXIS

The "I", "J" and "K" values are INCREMENTAL distances from where the tool starts cutting the arc (START POINT) to the ARC CENTER.

When I, J, or K are used to specify the center of the arc, R is not used. Only the I, J, or K specific to the selected plane (IJ for G17, IK for G18, JK for G19) are allowed. If only one of the I, J, K is specified, the others are assumed to be zero. The I, J, or K is the distance from the starting point to the center of the circle. Use of I, J, or K is the only way to cut a complete 360 degree arc; in this case, the starting point is the same as the ending point and no X, Y, or Z is needed. To cut a complete circle of 360 degrees (360°), you do not need to specify an ending point X, Y, or Z; just program I, J, or K to define the center of the circle.

NOTE: Example of circular moves are not using cutter compensation, so the circular moves are defined from the center of the cutter around arc. For 90 degree corners or fillets the "I" and "J" values can be defined like you see below.





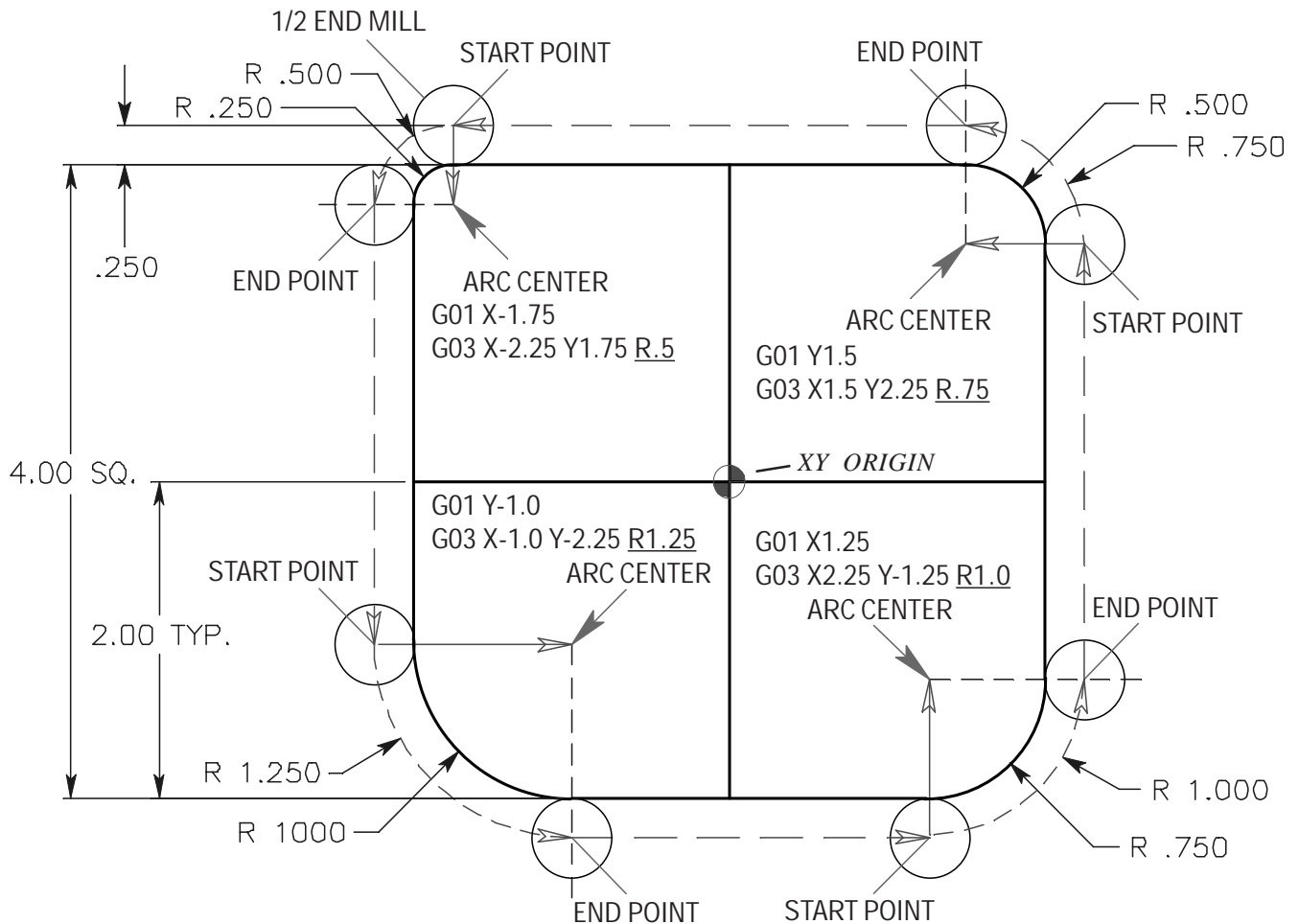
G03 CCW CIRCULAR MOTION USING "R"

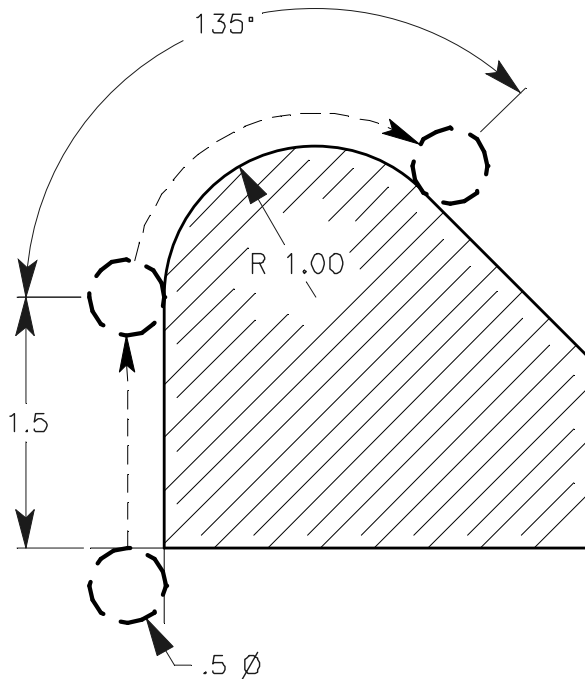
The letter address "R" can be substituted for the letter addresses "I", "J" and/or "K" for various corners or fillets and the "R" value is more easily defined.

Just as "I" "J" and "K", are used to locate the ARC CENTER in relation to the START POINT, "R" does the same function with less input.

When R is used to specify the center of the circle, a complete 360 degree arc is not possible. When defining an arc less then 360 degrees using the R command, the X, Y, or Z needs to specify an endpoint different from the starting point. And R defines the distance from the starting point to the center of the circle. With a positive R, the control will generate a radius of 180 degrees or less; to generate a radius of over 180 degrees, you need to specify an R command with a negative value.

NOTE: Example of circular moves are not using cutter compensation, so the circular moves are defined from the center of the cutter around arc. For 90 degree corners or fillets the "R" values can be defined like you see below.

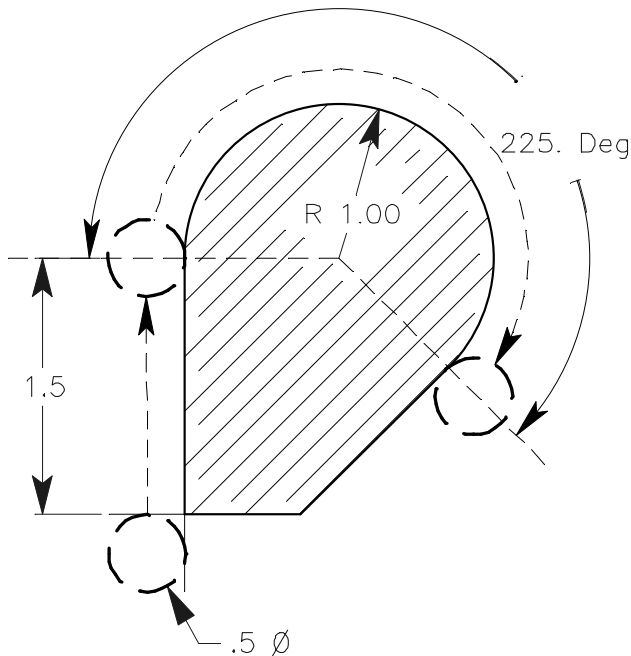




The following line will cut an arc less than 180 degrees (180°) using a positive R value:

```
G90 G54 G00 X-0.25 Y-0.25
G01 Y1.5 F12.
G02 X1.884 Y2.384 R1.25
```

To generate an arc of over 180 degrees you need to specify a negative R value(180°) using an R minus:



```
G90 G54 G00 X-0.25 Y-0.25
G01 Y1.5 F12.
G02 X1.884 Y0.616 R-1.25
```

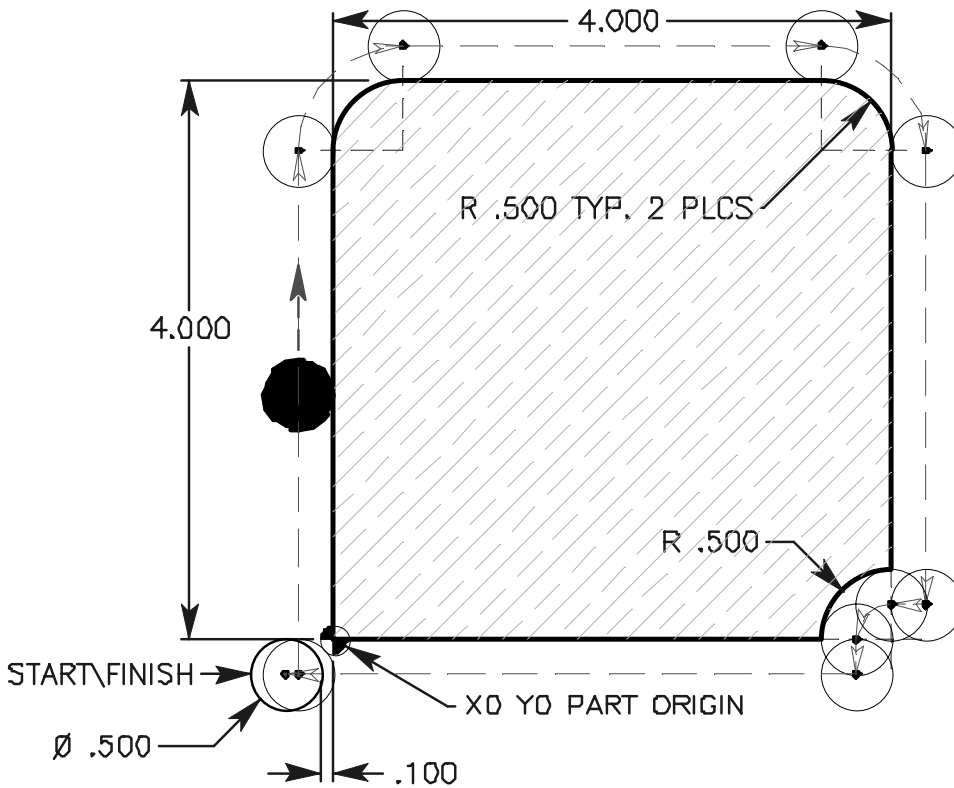
A complete 360 degree arc using an R command is not possible. To do a 360 degree arc in a G02 or G03 you need to use I and/or J to define the center of a circle for a complete 360 degree arc.

G03 CCW CIRCULAR INTERPOLATION MOTION

G03 will generate counterclockwise circular motion but is otherwise the same as G02.



INTERPOLATION EXERCISE



Cutter will be a .500 dia. end mill. Start contour from the lower left corner of part and mill around outside of part .625 deep. When defining a circular move (G02 or G03) you can use either an IJK or an R command but not both. Mill around outside of part with tool by defining the very center of the cutter to position around part.

"I" = X axis incremental distance and direction from the start point to the arc center.

"J" = Y axis incremental distance and direction from the start point to the arc center.

O00010 (INTERPOLATION EXERCISE)

T1 M06 (1/2 DIA. 4 FLT END MILL)

G90 G54 G00 X_____ Y_____

S1500 M03

G43 H01 Z0.1 M08

G01 Z_____ F50. (feed fast to depth non-cutting move)

X_____ F12. (feed onto left side of part programming to the center of tool)

Y_____ (feed to the upper left radius and continue around part to the end)

G0__ X_____ Y_____ R_____ (or use I_____ and J_____ instead of R)

G0__ X_____

G0__ X_____ Y_____ R_____ (or use I_____ and J_____ instead of R)

G0__ Y_____

X_____

G0__ X_____ Y_____ R_____ (or use I_____ and J_____ instead of R)

G0__ Y_____

X_____

G00 Z1. M09

G28 G91 Z0. M05

M30



PROGRAM START-UP LINES

T1 M06	T1	- Selects tool number one to be loaded into the spindle.
	M06	- Activates the tool change sequence.
G90 G54 G00 X___ Y_____	G90	- Activates control to be in ABSOLUTE.
	G54	- Selects work coordinate system No. 1
	G00	- Preparatory function for RAPID.
	X__	- Axis move to initial X position.
	Y__	- Axis move to initial Y position.
S1200 M03	S1200	- Informs the control that 1200 has been selected as the RPM for this tool.
	M03	- Turns the spindle on in a clockwise direction at a speed of 1200 RPM.
G43 H01 Z0.1 M08	G43	- Recognizes the tool length offset value stored in the Hnn code offset display register in the offset LENGTH GEOMETRY display.
	H01	- Defines to the control which offset register the tool offset value is stored in.
	Z0.1	- Informs the control to move from full spindle retract to this Z value and apply the tool length offset.
	M08	- Turns the coolant on.



PROGRAM ENDING LINES

G00 Z1. M09

- G00 - Preparatory function for rapid.
- Z1. - Retracts tool to 1.0 above part zero in preparation for full spindle retract.
- M09 - Turns the coolant off.

G28 G91 Y0. Z0. M05

- G28 - This preparatory function causes a return to machine zero of all axes. If an X, Y and Z are specified, only those axes specified will be moved. G28 will not recognize any tool length compensation in this move.
- G91 - Preparatory function for incremental mode, which is required for rapid move to the G28 reference point.
- Y0. - This will select the Y axis to position the table toward you to its machine zero.
- Z0. - This will insure a full retract, of Z axis in the up direction to machine zero
- M05 - This command will turn off spindle.

----- USING G53 TO SEND MACHINE HOME INSTEAD OF G28 G91 -----

G53 G49 Y0. Z0. M05

- G53 - This preparatory function temporarily ignores works offsets, and positions the machine axes from machine home. It is non-modal; so the next block will revert back to the previously selected work offset.
- G49 - This preparatory function cancels tool length compensation.
- Y0. - This will select the Y axis to position the table toward you to its machine zero.
- Z0. - This will insure a full retract, of Z axis in the up direction to machine zero
- M05 - This command will turn off spindle.

M30

- M30 - Signals End of Program and Reset, to the control. The program will reset to the first block of the program and stop. It also stops the spindle, turns off the coolant and cancels tool length offsets. It also makes the control ready for the next cycle.



INCH / METRIC SELECTION

G20 Inch programming selection

G21 metric programming selection

Selection between inch and metric programs can only be done with Setting 9.

Changing Setting

9 from inches to metric or back again will not convert the programs in memory. You must change your program axis values with metric or inch units and reload them after changing this setting.

Inch programming allows displacements up to +/-15400.000 inches and a resolution of 0.0001 inches. When in inch, the feed rate is defined as inches per minute, with a range of 0.0001 to 500.0000 mm/min.

Metric programming allows displacements up to +/-39300.000 mm and a resolution of 0.001 mm. When in metric, the feed rate is defined as millimeters per minute with a range of 0.001 to +/-12700.000 mm/min.

When jogging in metric, the speeds and units on the keypad are interpreted as mm/min but the value used is ten times larger than shown on the keypad.

The optional fourth and fifth axis programming is not effected by the selection of metric. It is always programmed in degrees. The auxiliary C axis is also always in degrees.

The standard G codes G20 and G21 are sometimes used to switch over between inch and metric but, on the Haas control, the G20 (inch) and G21 (mm) codes can only be used to ensure that the inch/metric Setting 9 is set correctly for that program.

Setting 9 DIMENSIONING - This is an Inch/Metric setting, When it is set to Inch, the programmed units for X, Y, and Z are inches to 0.0001. When it's set to Metric, programmed units are in millimeters to 0.001. Changing this setting will not convert a program over to the new setting. You must change the program axis values for the new units. When set to Inch, the Group 6 default G code is G20. When Set to Metric, the default G Coce is G21. When this setting is changed from inches to metric, or visa versa, all the offset values will be converted over accordingly.



WORK COORDINATE SELECTION

G54-59 WORK COORDINATE OFFSET SYSTEM #1-6

These G-codes will select one of the six work coordinate offsets that are entered in the Work Zero Offset display page. All subsequent references to axes' positions will be recognized in the new coordinate system.

G110-G129 WORK COORDINATE OFFSET SYSTEM #7-26

These G-codes are the same as work offsets G54 to G59.

G154 P1-99 WORK COORDINATE OFFSET SYSTEM #7-105

G154 was added in mill software version 12.02 and above. This feature provides 99 more work offsets. In all previous versions, the user was limited to a maximum of 27 work offsets. These included the standard work offsets (designated G54 through G59), and twenty additional work offsets (G110 through G129). Now with G154 P1 to P99, you are able to activate additional work offsets. For example G154 P10 will select work offset 10 from the list of additional work offset. Note that G110 to G129 refer to the same work offsets as the G154 P1 through P20. I.e., they can be selected by using either method. The Work Offset display screens have been adjusted accordingly. The Position page display has also been enhanced so that when a G154 work offset is active, the heading in the upper right work offset will show the G154 P value.

Note: In software 12.02, G154 renamed to G155 - In all previous versions, G154 was 5-AXIS REV TAP CYCLE. This has been changed to G155.



MORE WORK COORDINATE SELECTION

Note: *The G52 command works differently depending on the value of Setting 33. This setting selects the FANUC, HAAS, or YASNAC style of coordinates, which are listed below.*

G52WORK COORDINATE OFFSET SYSTEM YASNAC

This code selects the G52 work coordinate system. The G52 works the same as G54 etc., except that the G52 system will be set by a G92 code as well as from the offsets display.

G52 SET LOCAL COORDINATE WORK OFFSET SHIFT VALUE FANUC

This code sets the origin of the local (child) coordinate system to the command location, relative to the current work system origin. G52 is a non-modal, no motion code. The G52 coordinate system will stay in effect for all work systems until it is canceled. The G52 is canceled when RESET is pressed and at the end of a program. It is also canceled during a program by M30, G52, X0 Y0 Z0, or by a G92 command.

G52 SET LOCAL COORDINATE WORK OFFSET SHIFT VALUE HAAS

This code acts the same as in the Fanuc control except that G52 is not cleared at power-up, RESET, or when an M30 is performed. It is canceled with a G52, X0 and/or Y0 and/or Z0, or by a G92 command.

G53NON-MODAL MACHINE POSITION COORDINATE SYSTEM

This code temporarily ignores work coordinates offset and uses the machine coordinate system (machine zero). It is non-modal; so the next block will revert to whatever work offset command that was previously selected.



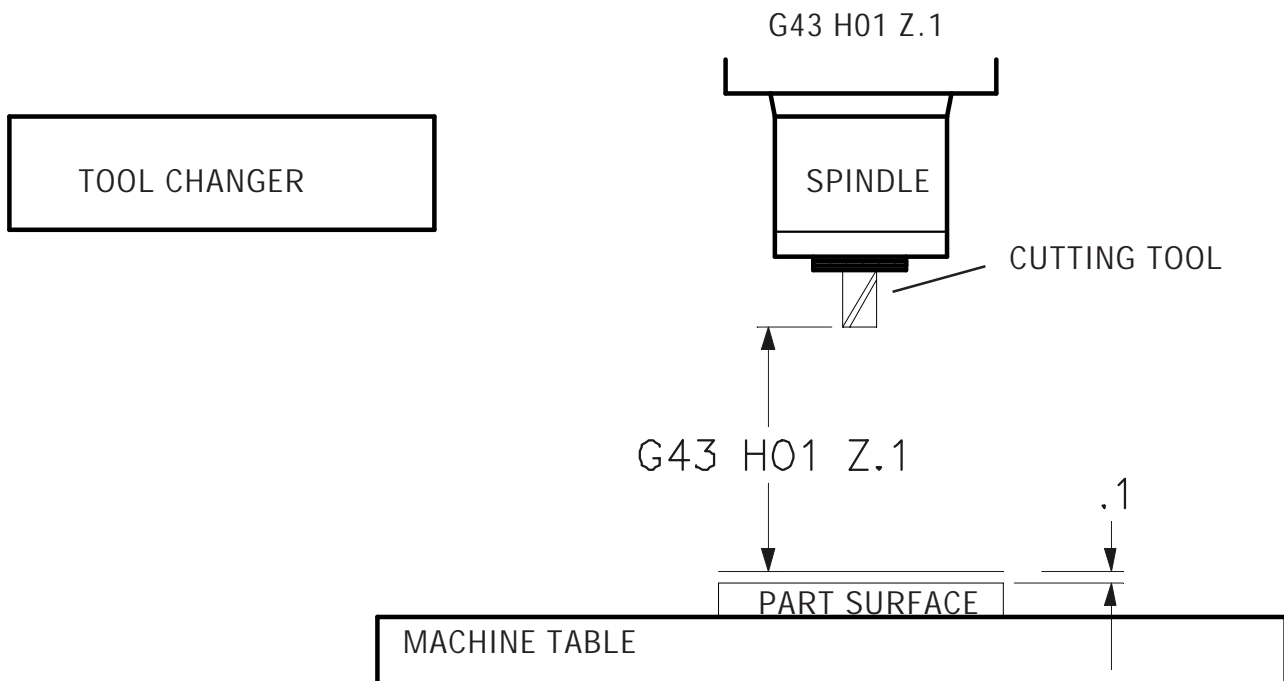
TOOL LENGTH COMPENSATION

G43 Tool Length Compensation + (plus)

This code selects tool length compensation in a positive direction. That is; the tool length offsets are added to the commanded axis positions. An Hnn must be programmed to select the correct offset register from the offset display for that tool being used.

During the setup process, each tool point was touched-off to the part zero surface. From this position a TOOL LENGTH DISTANCE offset was recorded for that tool with the TOOL OFFSET MESUR key. This TOOL LENGTH is referred to as the "Z" axis origin move to the part zero surface.

The programmed code to position tool:



G44 TOOL LENGTH COMPENSATION - (MINUS)

This code selects tool length compensation in a negative direction. That is; the tool length offsets are subtracted from the commanded axis positions. A Hnn must be programmed to select the correct entry from offsets memory.

G49 CANCELS G43/G44

This G code cancels tool length compensation. Putting in a H00 will also cancel tool length compensation. M30, and RESET will also cancel tool length comp.



DWELL COMMAND

G04 Dwell

P The dwell time in seconds or milliseconds

G04 is used to cause a delay or dwell in the program. The block containing G04 will delay for the time specified in the P code. When programmed on a line following some motion such as G00, G01, G02 and G03 all motion will be stopped for the amount of time specified in the P command in seconds. If the P has no fraction part, the delay is in milliseconds (0.001 seconds); otherwise the delay is in seconds. The slide motion is stopped, but the spindle will continue to rotate at the requested RPM, and the coolant stays on.

G04 P_____

Minimum value - P.001 of a second

Maximum value - P1000.0 seconds

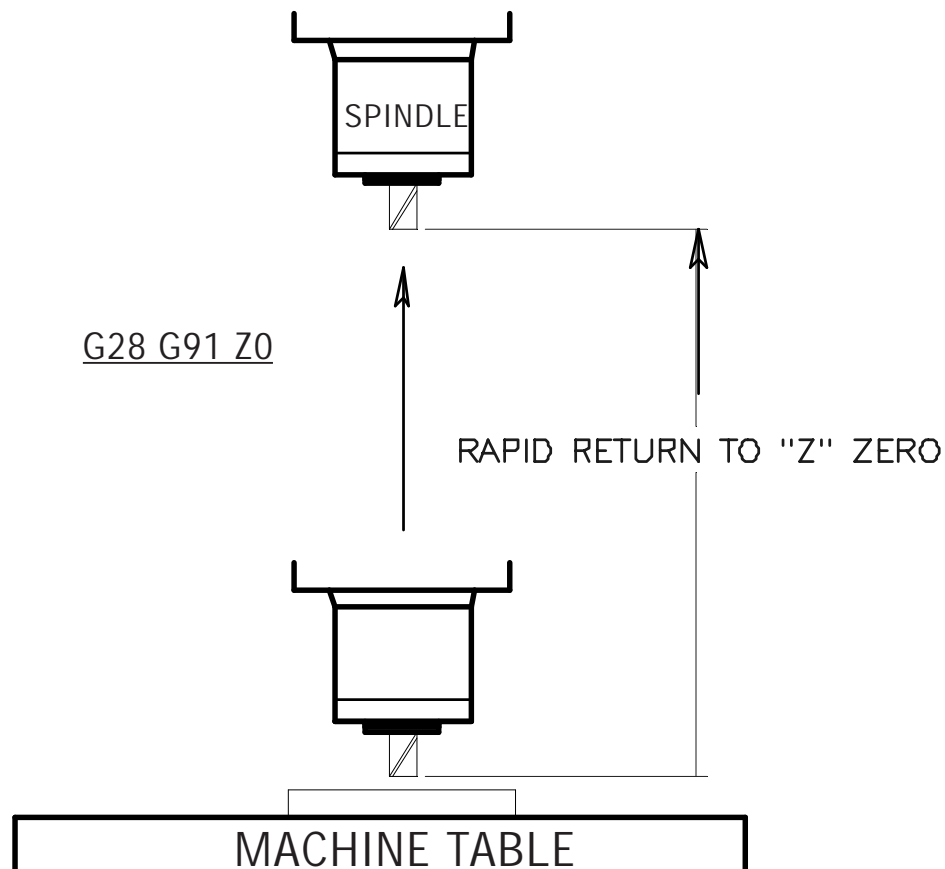


REFERENCE POINT AND RETURN

G28 Return To Reference Point, set optional intermediate point

The G28 code is used to return to the machine zero position on all axes. If an X, Y, Z, or A axis is on the same block and specifies a location, only those axes will move and return to the machines' zero reference point and the movement to the machines' zero reference point will be through that specified location. This point is called the intermediate point and is saved, if needed, for use in G29. If you do not want to position through an intermediate point while specifying a specific axis to position to machine zero, then add a incremental (G91) command to this line along with a Z0, Y0, and/or X0 for the specific axis you want to send to machine zero. This will command those axes specified to position incrementally to a zero distance as an intermediate point, and then those axes specified will then position to machine zero. Just be sure to program in an absolute (G90) command in the start-up lines for the next tool which is usually needed for the beginning of each tool.

If no X, Y, Z, or A is specified, all axes will be moved directly to machine zero. Any auxiliary axes (B, C,...) are returned to there machine home after the X, Y, Z, and A axes. G28 will not recognize any tool length offsets in this move.





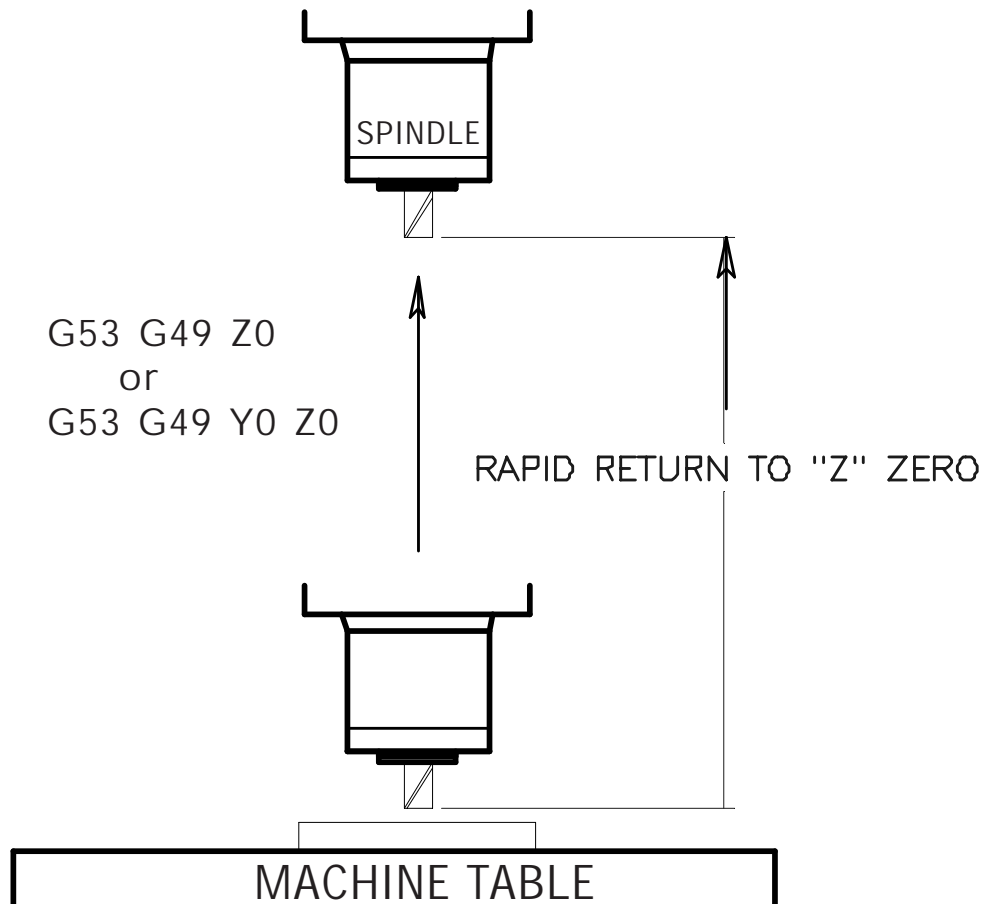
ANOTHER WAY TO RETURN TO MACHINE ZERO

G53 Non-Modal Machine Coordinate Selection

This code temporarily ignores work coordinates offset and uses the machine coordinate system (machine zero). This gives you a command to move to a specific location defined from the machine zero reference point. It is non-modal; so the next command block will revert back to the work coordinate offset command, that was previously active.

This G53 command may be used to send the machine home for a tool change, or to send the machine home in Y, Z axes at the end of a program, rather than using a G28. A G28 works good for sending the machine home, and most people have learned to send it home this way. But to send it straight home with a G28, and a specific axis, you need to also define a G91 so that it positions through the intermediate point incrementally to go straight home (see p.42). And then, you will usually need to be sure to switch back to a G90 absolute command, for your next move. And instead of dealing with switching in and out of absolute and incremental you could command a G53 along with a Z0, Y0, and/or X0 for the specific axis you want to send to machine zero.

Just be sure to cancel your tool length offset with a (G49) command when you define a G53 G49 Z0 to send Z axis to machine home for a tool change.





CIRCULAR POCKET MILLING

There are two G codes, G12 and G13 that will provide for pocket milling of a circular shape. They're different only in which direction of rotation is used. G12 and G13 are non-modal.

G12 Circular Pocket Milling Clockwise

- X Position in X axis to center of circular pocket
- Y Position in Y axis to center of circular pocket
- Z Z depth of cut, or it's the increment depth of cuts when used with G91
- I Radius Of First Circle (Or it's the finish radius if K is not used)
- K Radius Of Finished Circle (If specified)
- Q Radius cut increment step-over of the spiral out (Q is used with K only)
- L Loop count for repeating incremental depth of cuts (L is used with G91)
- D* Cutter Comp. Offset Number (Enter cutter size in offset display register)
- F Feed Rate in inches (mm) per minute

This G12 code implies the use of G42 cutter compensation right

G13 Circular Pocket Milling Counterclockwise

This G13 code implies the use of G41 cutter compensation left and will be machining in a counterclockwise direction, but is otherwise the same as G12.

G13 is usually preferred instead of G12, since G13 will be climb cutting when used with a standard right handed tool.

*In order to get the exact programmed circular pocket diameter, the control uses the selected D code tool size. If this compensation is not desired, program D0.

The tool must be positioned at the center of the circular pocket either in a previous block or in this command using an X and Y position. The cuts are performed entirely with circular motions of varying radiuses.

The G12 Code implies the use of G42 cutter compensation right.

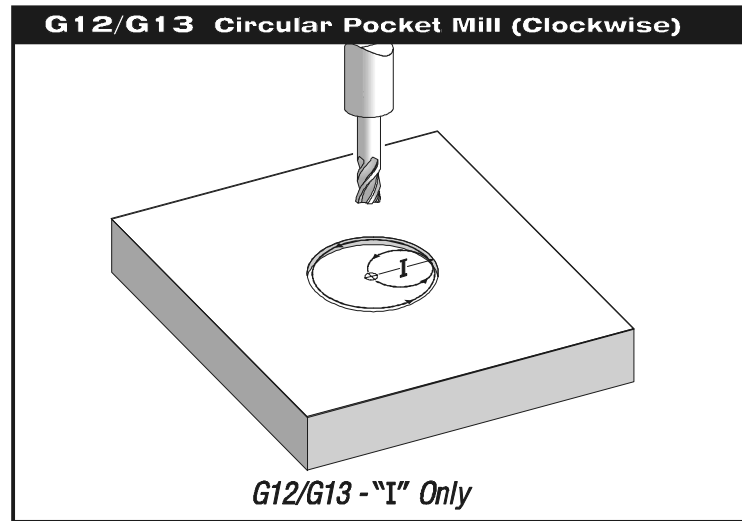
The G13 Code implies the use of G41 cutter compensation left.

To remove all the material within the circle use an I and Q value less than the tool diameter and a K value equal to the circle radius. G12 and G13 belongs to Group zero and thus is non-modal.

If no K & Q is specified, the center roughing passes of this command are removed completely and only one finish pass of the circular pocket is performed.

If G91 (incremental) is specified and an L count is included, the Z increment is repeated L times at the F feed rate command.

NOTE: This command will not bring the tool back out of the circular pocket after it's done. So be sure to move the tool up in the Z axis, above the fixture and part, before you position to another XY location to machine.

**EXAMPLE: G12/G13 ONE PASS "I" ONLY**

(G13 is usually preferred, in machining a counterclockwise climb cut on the material.)

O00050

N11 (D01 DIA. OFFSET IS .500)

N12 T1 M06 (1/2 DIA. 2 FLT END MILL)

N13 G90 G54 G00 X2.5 Y2.5 (position to X Y center of circular pocket)

N14 S1900 M03

N15 G43 H01 Z0.1 M08

N16 G13 Z-0.5 I0.5 D01 F10. (1.0 Dia. x.05 deep circular pocket 1 pass)

N17 G00 Z1. M09

N18 G53 G49 Y0. Z0.

N19 M30

*Moving the Z axis down with a faster or slower feedrate than what's in the G12/G13 line, for a different Z axis feedrate, then what's machining the circular pocket in X and Y axes.

O00050

N21 (D01 DIA. OFFSET IS .500)

N22 T1 M06 (1/2 DIA. 2 FLT END MILL)

N23 G90 G54 G00 X2.5 Y2.5 (position to X Y center of circular pocket)

N24 S1900 M03

N25 G43 H01 Z0.1 M08

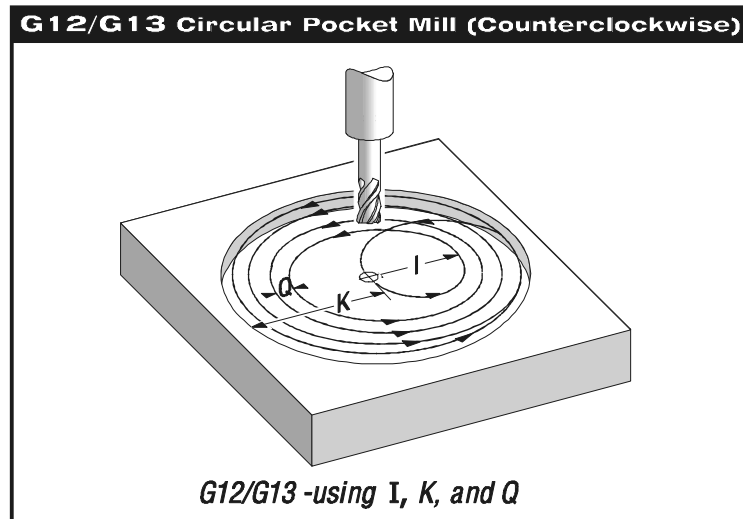
N26 G01 Z-0.5 F6. (feeding Z axis down slower or faster before the G13 line)*

N27 G13 I0.5 D01 F10. (1.0 Dia. x .5 deep circular pocket 1 pass)

N28 G00 Z1. M09

N29 G53 G49 Y0. Z0.

N30 M30



EXAMPLE: G12/G13 MULTIPLE PASSES I, K & Q

(G13 is usually preferred, in machining a counterclockwise climb cut of the material.)

O00051

N31 (D02 DIA. OFFSET IS .625)

N32 T2 M06 (5/8 DIA. 2 FLT END MILL)

N33 G90 G54 G00 X2.5 Y2.5 (X Y center location of circular pocket)

N34 S1500 M03

N35 G43 H02 Z0.1 M08

N36 G13 Z-0.5 I0.3 K1.5 Q0.3 D02 F10. (3.0 Dia. x.5 deep circular pocket)*

N37 G00 Z1. M09

N38 G53 G49 Y0. Z0.

N39 M30

*Moving the Z axis down with a faster or slower feedrate than what's in the G12/G13 line, for a different Z axis feedrate, then what's machining the circular pocket in X and Y axes.

O00051

N41 (D02 DIA. OFFSET IS .625)

N42 T2 M06 (5/8 DIA. 2 FLT END MILL)

N43 G90 G54 G00 X2.5 Y2.5 (X Y center location of circular pocket)

N44 S1500 M03

N45 G43 H02 Z0.1 M08

N46 G01 Z-0.5 F6. (feeding Z axis down slower or faster than being in G13 line)*

N47 G13 I0.3 K1.5 Q0.3 D02 F10. (3.0 Dia. x .5 dp circular pocket)*

N48 G00 Z1. M09

N49 G53 G49 Y0. Z0.

N50 M30



EXAMPLE: G12/G13 MULTIPLE "Z" PASSES TO DEPTH USING INCREMENTAL G91 AND AN "L" LOOP COUNT

```
O00052
N51 (D02 DIA. OFFSET IS .625)
N52 T2 M06 (5/8 DIA. 2 FLT END MILL)
N53 G90 G54 G00 X2.5 Y2.5 (X Y center location of circular pocket)
N54 S1500 M03
N55 G43 H02 Z0.1 M08
N56 G01 Z0. F30. (Move down to start point to begin increment down)
N57 G13 G91 Z-0.375 I0.325 K2. Q0.3 D02 L4 F12. (4.0 Dia. x 1.5
Deep circular pocket)
N58 G00 G90 Z1. M09
N59 G53 G49 Y0. Z0.
N60 M30
```

The above program uses G91 and a "L" count of four. This cycle is multiplied by the "L" command and will do it a total of four times at the Z depth increment of .375 to a total depth of 1.5 in. The G91 and "L" count can also be used for G12 and G13 "I" only line.

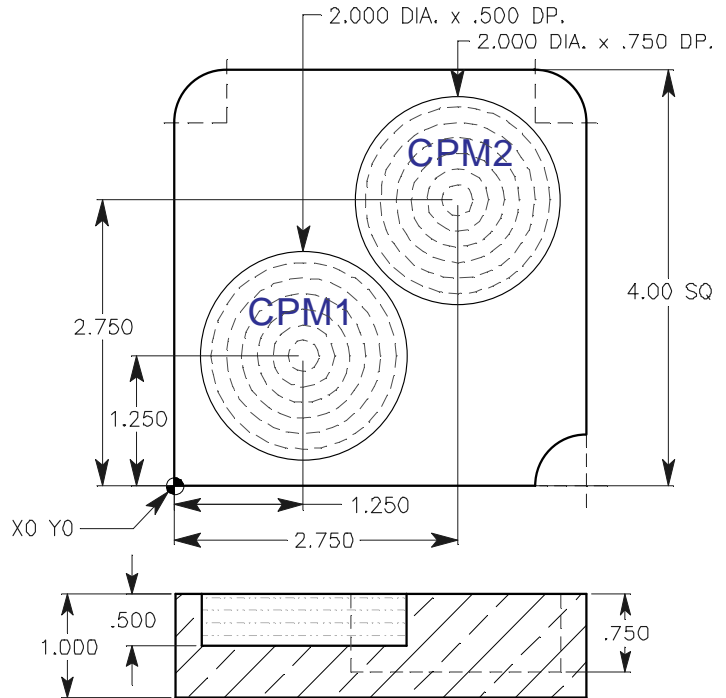
Since the G91 incremental Z depth is looped together within the G13 circular pocket command, you are unable to separate the Z axis feedrate from the X and Y axis feedrate. The feedrate in the Z axis will be the same as X and Y axis.

You may want to fast feed down to the Z surface, of where the circular pocket begins, to start incrementally step down to establish the desired depth.



CIRCULAR POCKET MILLING EXERCISE

- G12 Circular Pocket Milling CW
or
- G13 Circular Pocket Milling CCW
- X Position to center of pocket
- Y Position to center of pocket
- Z Depth of cut or increment down
- I Radius of First Circle (or the finish radius if K is not used).
- K Radius of Finished Circle (if specified).
- Q Radius step over Increment (must be used with K).
- D Cutter Comp. number (Enter cutter size into offset display register number).
- L Loop count for repeating deeper cuts.
- F Feedrate in inch (mm) per min.



TOOL 2 IS A 5/8 DIA. 2 FLUTE END MILL WITH A SPINDLE SPEED OF 1250 RPM AT 8.0 FEED.

Circular Pocket Mill CPM 1 which is a 2.0 Dia. x .500 dp. pocket, spiraling out to a rough 1.980 diameter using I0.25, K0.99 and Q0.2 roughing out pocket. Then mill another circular pocket command to finish CPM 1 using 1.0 "I only" as a circular pocket finish pass.

Circular Pocket Mill CPM 2 which is a 2.0 Dia. x .750 dp. pocket incrementally stepping down 0.25 depth using an L3 count and an incremental G91 command to do 3 passes on a circular pocket using I0.3, K1.0 and Q0.35 to machine out pocket.

O00020 (CIRCULAR POCKET MILLING EXERCISE)

```

T___ M___ (T2 IS A 5/8 DIA. 2 FLT CENTER CUTTING END MILL)
G___ G___ G___ X_____ Y_____ (X Y position to center of CPM1)
S_____ M_____
G___ H___ Z___ M___ (Rapid to .1 above part)
G___ Z___ I___ K___ Q___ D___ F___ (CPM1 with I,K& Q)
G___ I___ D___ F11.5 (I only for a finish pass of CPM1)
G___ Z___ (Rapid Z to a .1 clearance position above part)
X_____ Y_____ (X Y position to center of CPM2)
G01 Z0. F20. (Feed down to the start point to start incremental depth down)
G___ G___ Z___ I___ K___ Q___ D___ L___ F___ ( I, K, Q & G91)
G___ G___ Z___ M___ (Switch to absolute, Rapid 1.0 above part, coolant off)
G53 G49 Z0. M05
M30
    
```

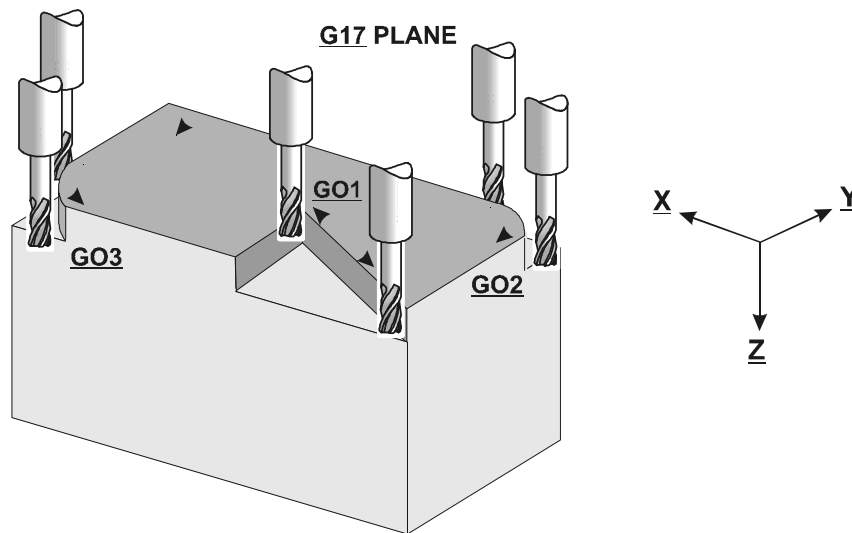


CIRCULAR PLANE SELECTION

The axis plane used for machining in a circular motions must be comprised of two of the three axes in X, Y, or Z. The plane selection is modal and stays in effect for all subsequent circular interpolation moves until you command another plane selection code. There are three G codes used to select the circular plane; G17 for the XY plane, G18 for the XZ plane, and G19 for the YZ plane.

G17 XY CIRCULAR PLANE SELECTION

The G17 code is used to select the XY plane for circular motion. In this plane, circular motion is defined as clockwise for the operator looking down onto the XY table from above



O00054 (G17 XY CIRCULAR PLANE)

N1 T1 M06 (1/2 DIA. 2 FLT E.M.)

N2 G90 G54 G00 X4. Y3.25 S2600 M03 (XY start point of arc)

N3 G43 H01 Z0.1 M08

N4 G01 Z-0.5 F50.

N5 G17 G02 X5.25 Y2. R1.25 F10. (G17 circular motion XY plane)

N6 G00 Z0.1

N7 X5. Y-.25 (G17 is default when you power up machine)

N8 G01 Z-0.5 F50.

N9 X3.25 Y0.8

N10 Y-.25

N11 G00 Z1.

N12 X-0.25 Y1.

N13 G01 Z-0.5 F50.

N14 G17 G03 X1. Y-0.25 R1.25 F10. (G17 circular motion XY plane)

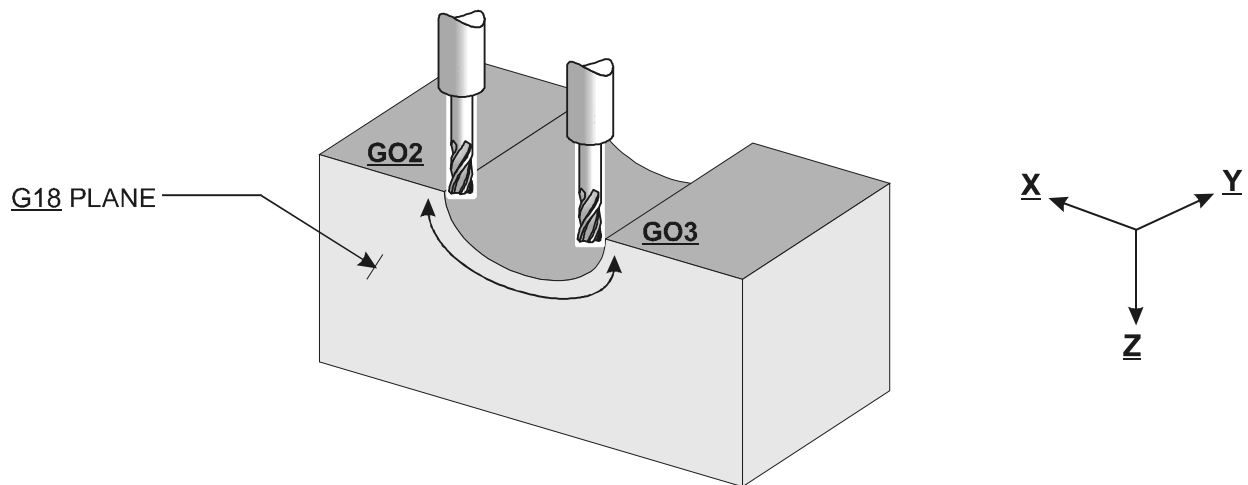


```

N15 G00 Z1. M09
N16 G53 G49 Y0. Z0. M05
N12 M30
  
```

G18 ZX CIRCULAR PLANE SELECTION

The G18 code is used to select the ZX plane for circular motion. In the XZ plane (G18), circular motion is defined as clockwise for the operator looking from the rear of the machine out toward the control panel.



```

O00055 (G18 ZX CIRCULAR PLANE)
N1 T1 M06 (1/2 DIA. 2 FLT BALL E.M.)
N2 G90 G54 G17 G00 X1.5 Y0. S2600 M03 (XY start point of arc)
N3 G43 H01 Z0.1 M08
N4 G01 Z0. F20.
N5 M97 P100 L80 (Local sub-routine call done 80 times with L80)
N6 G17 G00 Z1. M09 (Switch back to G17 XY-plane when finished with G18)
N7 G53 G49 Y0. Z0. M05
N8 M30
  
```

(Local sub-routine N100 called by M97 P100 in line N5, repeated 80 times with L80)

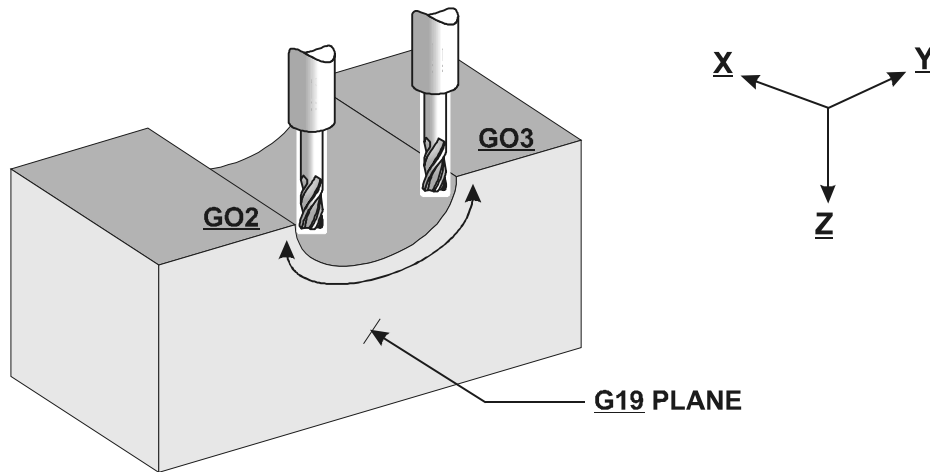
```

N100 G91 G01 Y-0.01 (Local sub-routine N100 called by an M97 on line N5)
N101 G90
N102 G18 G03 X3. Z0. R0.75 F12. (G18 circular motion ZX plane)
N103 G91 G01 Y-0.01
N104 G90
N105 G18 G02 X1.5 Z0. R0.75 F12. (G18 circular motion ZX plane)
N106 M99 (An M99 will cause the program to jump back to the next
line after the M97 sub-routine call in the main program.)
  
```



G19 YZ CIRCULAR PLANE SELECTION

The G19 code is used to select the YZ plane for circular motion. In the YZ plane (G19), circular motion is defined as clockwise for the operator looking across the table from the side of the machine the control panel is mounted.



O00056 (G19 YZ CIRCULAR PLANE)

N1 T1 M06 (1/2 DIA. 2 FLT BALL E.M.)

N2 G90 G54 G17 G00 X0. Y3. S2600 M03 (XY start point of arc)

N3 G43 H01 Z0.1 M08

N4 G01 Z0. F20.

N5 M97 P100 L80 (Local sub-routine call done 80 times with L80)

N6 G17 G00 Z1. M09 (Switch back to G17 XY-plane when finished with G19)

N7 G53 G49 Y0. Z0. M05

N8 M30

(Local sub-routine N100 called by M97 P100 in line N5, repeated 80 times with L80)

N100 G91 G01 X-0.01

N101 G90

N102 G19 G03 Y1.5 Z0. R0.75 F12. (G19 circular motion YZ plane)

N103 G91 G01 X-0.01

N104 G90

N105 G19 G02 Y3. Z0. R.75 F12. (G19 circular motion YZ plane)

N106 M99 (An M99 will cause the program to jump back to the next line after the M97 sub-routine call in the main program.)



The default plane selection when the machine is powered on is G17 for the XY plane. This means that circular motion in the plane of the XY table axis may be programmed without first selecting G17.

A helical motion is possible with G02 or G03 by programming the linear axis which is not in the selected plane. This third axis will be interpolated along the specified axis in a linear manner while the other two axes will be moved in the circular motion. The speed of each axis will be controlled so that the helical rate matches the programmed feed rate.

If cutter radius compensation (G41 or G42) is selected, you can only use it for the G17 XY circular planes. Cutter compensation for circular motions in the G18 XZ or G19 YZ circular planes can be done using the command G141 for 3D+ cutter compensation.



CUTTER COMPENSATION

Cutter compensation is used to offset the center of the cutter, and shift it the distance of the radius, to the specified side of the programmed path. Complex part geometries having angled lines, lines tangent to arcs, and lines intersecting arcs involve substantial trigonometric computations to determine the center of the cutter. Cutter compensation involves programming the part geometry directly instead of the tool center. The cutter compensation commands are Cutter Comp. Left (G41), Cutter Comp Right (G42) and Cutter Comp Cancel (G40).

G40 Cutter Comp Cancel

G40 will cancel the G41 or G42 cutter compensation commands. A tool using cutter compensation will change from a compensated position to an uncompensated position. Programming in a D00 will also cancel cutter compensation. Be sure to cancel cutter compensation, when you're done with each milling cut series that's using compensation.

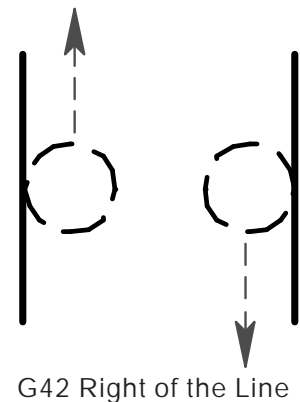
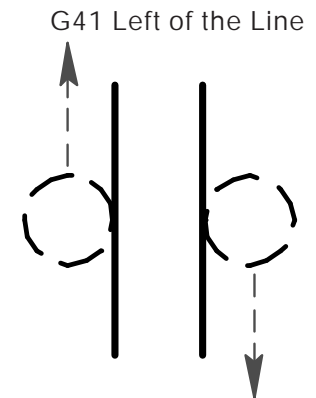
G41 Cutter Compensation Left

G41 will select cutter compensation left; that is the tool is moved to the left of the programmed path to compensate for the radius of the tool. A Dnn must also be programmed to select the correct tool size from the DIAMETER/RADIUS offset display register.

G42 Cutter Compensation Right

G42 will select cutter compensation right; that is the tool is moved to the right of the programmed path to compensate for the size of the tool. A Dnn must also be programmed to select the correct tool size from the DIAMETER/RADIUS offset display register.

Dnn Cutter Comp value - The actual offset amount must be input in the specified tool offset display number. On the HAAS you have 200 tool DIAMETER/RADIUS offsets to use. Usually, you have one cutter offset for each tool, and it is best to use the same offset number as is the tool number. On the HAAS tool geometry OFFSET DISPLAY page, the two columns to the right side of this display are for your cutter diameter offsets. The (RADIUS/DIAMETER) GEOMETRY column, in the offset display, is to set your initial cutter offset value and can be defined as either a DIAMETER or RADIUS value, by selecting the one you want to use in SETTING 40. The selection you choose will be listed at the top of the offset geometry column. The WEAR column to the right of the tool (RADIUS/DIAMETER) GEOMETRY column is for any adjustments you need to make from the initial tool GEOMETRY offset. And these value are added together by the control and used as one value.





- G41 Cutter Compensation Left
- G42 Cutter Compensation Right
- G40 Cancel Cutter Comp G41 and G42

Understanding cutter compensation can be simplified if one has a basic understanding of manual machining. There are two common types of cutting conditions associated with milling machines. They are CLIMB and CONVENTIONAL cutting.

Two common rules for these types of cuts are:

If the programmed cutter path needs to mill CLIMB cutting, which is what's usually desired on CNC machines, since the machines are rigid enough to handle this type of cut, and the conditions from this type of cut are usually preferred. And if it's a standard right handed tool, it will then be programmed with G41 cutter LEFT of the programmed path.

If the programmed cutter path needs to mill with CONVENTIONAL cutting, and it's a standard right handed tool, it will then be programmed with G42 cutter RIGHT of the programmed path.

Program without cutter compensation.

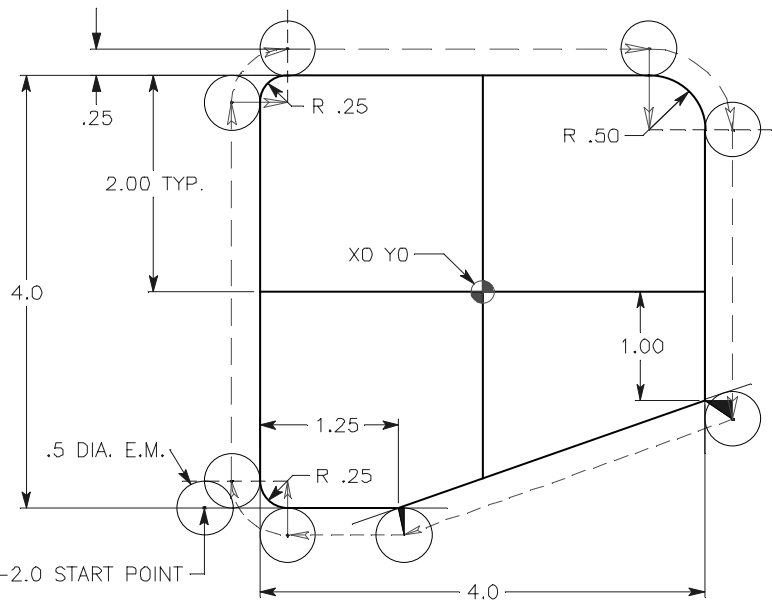
```

N104 G00 X-2.5 Y-2.0
N105 G01 Z-0.45 F50.
N106 X-2.25 F12.
N107 Y1.75
N108 G02 X-1.75 Y2.25 R0.5
N109 G01 X1.5
N110 G02 X2.25 Y1.5 R0.75
N111 G01 Y???? (Calculate point)
N112 X???? Y-2.25 (Calculate point)
N113 G01 X-1.75
N114 G02 X-2.25 Y-1.75 R0.5
N115 G01 X-2.35 Y-2.0
    
```

Program with cutter compensation.
Dia. value for D01 would be .500 in DIAMETER offset register #1.

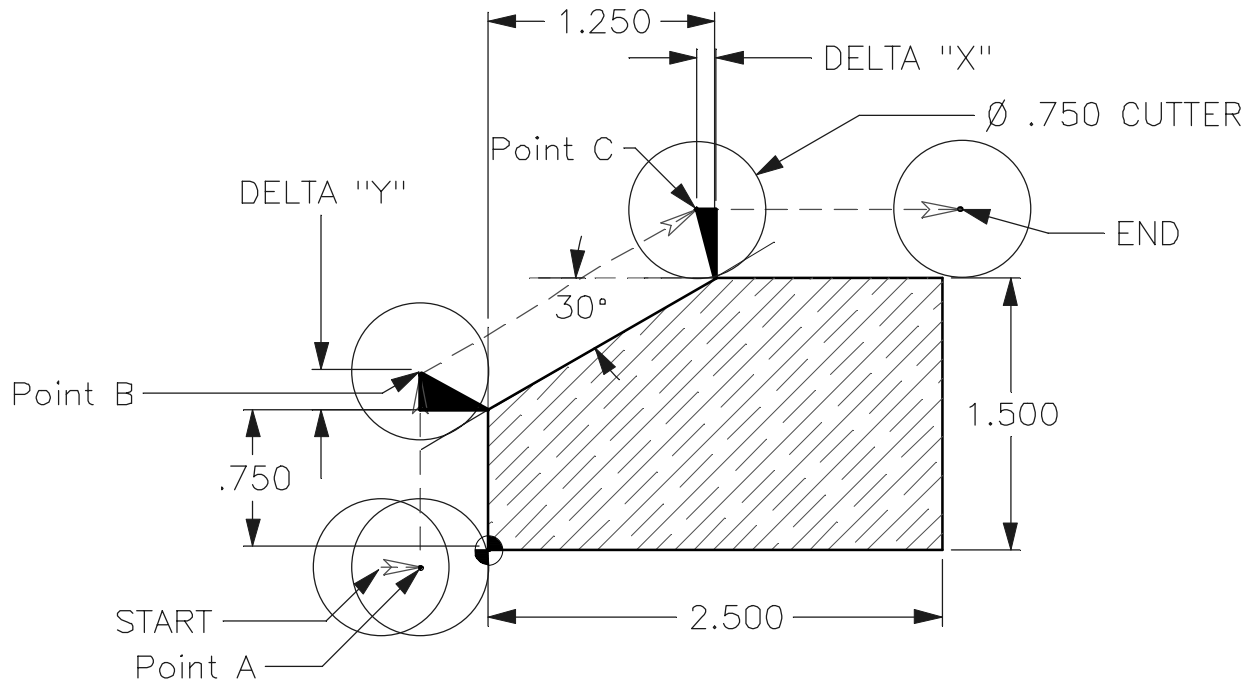
```

N204 G00 X-2.35 Y-2.0
N205 G01 Z-0.45 F50.
N206 G41 X-2. D01 F12. (turn on C.C. with an X and/or Y move)
N207 Y1.75
N208 G02 X-1.75 Y2. R0.25
N209 G01 X1.5
N210 G02 X2. Y1.5 R0.5
N211 G01 Y-1.
N212 X-0.75 Y-2.
N213 X-1.75
N214 G02 X-2. Y-1.75 R0.25
N215 G40 G01 X-2.35 (turn off C.C. with an X and /or Y move)
    
```





When programming without cutter compensation, to the center of the cutter, a problem occurs cutting angle geometry. The cutter center must be offset to the part geometry to maintain the cutter tangency. For example, the Y axis move from Point A to Point B must have the Delta "Y" calculated dimension added to the .75 dimension. The X axis move from Point B to Point C must have the Delta "X" calculated dimension subtracted from the 1.25 dimension.



Program is to the center of the cutter manually calculating the cutter compensation of tool.
O00060

N1 T3 M06 (3/4 DIA. 4 FLT END MILL)

N2 G90 G54 G00 X-0.475 Y-0.1 (X Y position away from part, with center of tool)

N3 S1275 M03

N4 G43 H03 Z0.1 M08

N5 G01 Z-0.25 F50.

N6 X-0.375 F12.

N7 Y?.???? (center of tool)

N8 X?.???? (center of tool) Y1.875

N9 X2.6

N10 G00 Z1. M09

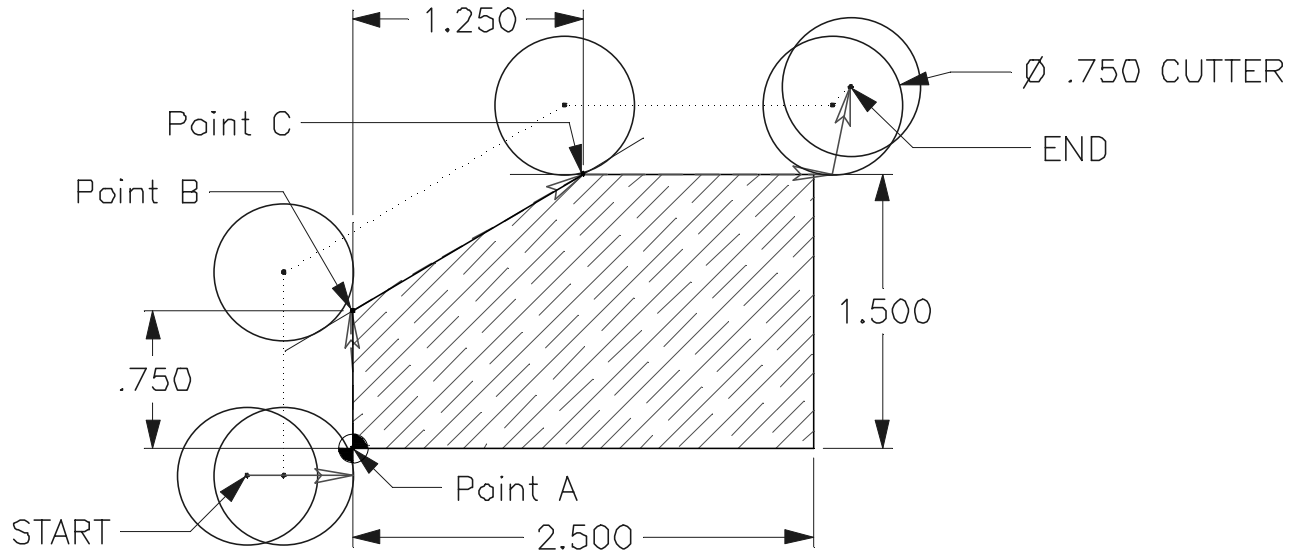
N11 G53 G49 Y0. Z0. M05

N12 M30

If however, the programming is being done with cutter compensation in effect, the moves from Point A, to Point B, to Point C will reflect the actual geometry or part



print X and Y coordinate values. The control will offset the cutter center around Point B and Point C to maintain the required cutter tangency.



O00061 (Part program using cutter compensation)

N1 T3 M06 (3/4 DIA. 4 FLT END MILL)

N2 G90 G54 G00 X-0.475 Y-0.1 (X Y position away from part, with center of tool)

N3 S1275 M03

N4 G43 H03 Z0.1 M08

N5 G01 Z-0.25 F50.

N6 G41 X0. D03 F12. (Position onto part line, turning on Cutter Compensation)

N7 Y0.75

N8 X1.25 Y1.5

N9 X2.6

N10 G40 X2.7 Y2. (Position off of part to the X Y center of tool, turning off C.C.)

N11 G00 Z1. M09

N12 G53 G49 Y0. Z0. M05

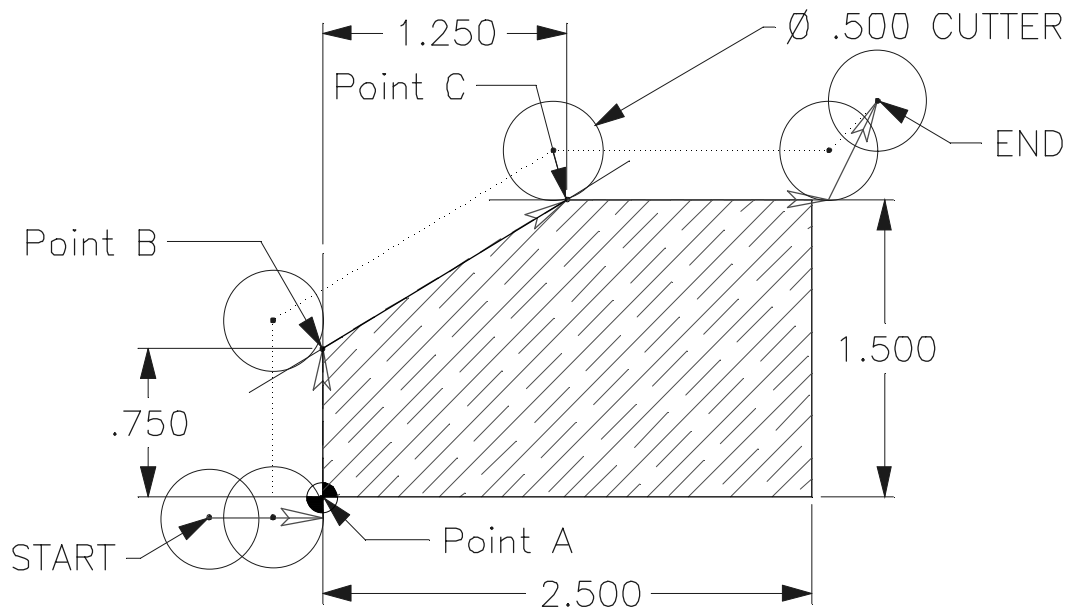
M30

When the part has been programmed using cutter compensation we are, in effect, programming with a zero diameter cutter, to the center of the tool. The tool diameter value is entered in on the OFFSET display page under the RADIUS/DIAMETER geometry offset column. The control will position the tool the offset amount, off of the programmed part line that is programmed so that the edge of the tool is positioning around the part geometry.



If, at the time of setup, the cutting tool requested by the planning is for a diameter not available currently in your shop, then input the available size on the offset page. DO NOT change the program. Cutter compensation takes the stored value for the diameter and calculates the cutter path offset from that value.

If a larger tool is going to be used, you will need to make sure that you change the starting and ending positions so that the distance of the cutter is positioned half the diameter off the part for clearance when you lead onto and off of the part.



Machining part with C.C. using an undersize tool.

```
O00062 (Part program using cutter compensation)
N1 T3 M06 (1/2 DIA. 4 FLT END MILL)
N2 G90 G54 G00 X-0.475 Y-0.1 (X Y position away from part, with center of tool)
N3 S1275 M03
N4 G43 H03 Z0.1 M08
N5 G01 Z-0.25 F50.
N6 G41 X0. D03 F12. (Position onto part line, turning on Cutter Compensation)
N7 Y0.75
N8 X1.25 Y1.5
N9 X2.6
N10 G40 X2.7 Y2. (Position off of part to the X Y center of tool, turning off C.C.)
N11 G00 Z1. M09
N12 G53 G49 Y0. Z0. M05
M30
```



NOTE: Advantage of cutter compensation can be taken when a rough and finish pass must be done with one tool. As we have seen, if we program the finish geometry and use cutter compensation, we will get a finish pass.

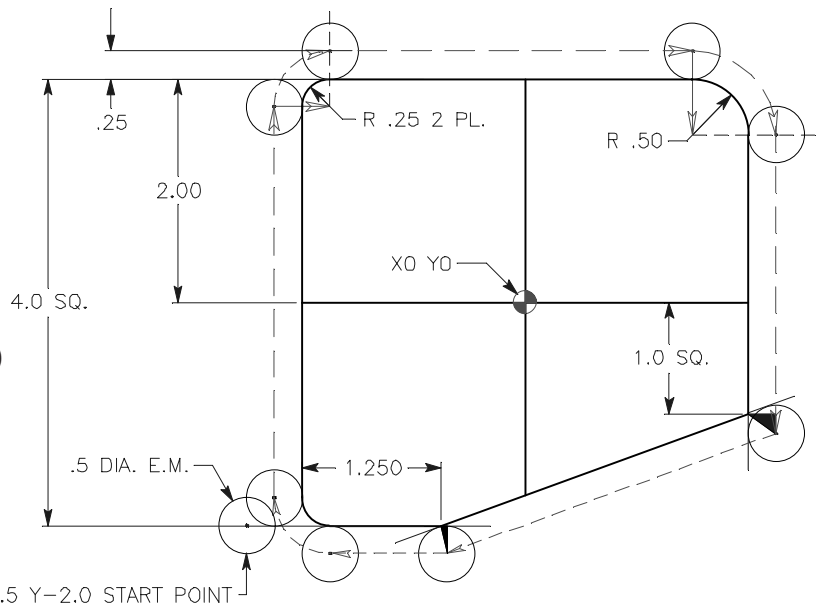
If however, prior to the rough pass, we input a diameter offset value that is oversize to the physical size of the end mill, and then program the finish geometry, the control will offset the end mill more, leaving stock for a finish pass. Then program the same end mill using another offset with the correct offset value and the same geometry.

EXAMPLE: If the cutter is .500 diameter, and you want to leave .010 stock for a finish pass. Then enter a diameter offset number (Dn) command for tool DIAMETER Geometry of .520 for machining a roughing pass around part. And then define another pass around part using the same program coordinates but a different diameter offset number (Dn), and enter in the correct diameter offset value of .500 to machine part to the finish size.

TOOL 1 HAS .520 IN D21
AND .500 IN D01

```

O00063
N1 T1 M06
N2 G90 G54 G00 X-2.5 Y-2. S1600 M03
N3 G43 H01 Z0.1 M08
N4 G01 Z-0.45 F30.
N5 G41 X-2. D21 F12. (C.C. on using D21)
N6 Y1.75
N7 G02 X-1.75 Y2. R0.25
N8 G01 X1.5
N9 G02 X2. Y1.5 R0.5
N10 G01 Y-1.
N11 X-.75 Y-2.
N12 X-1.75
N13 G02 X-2. Y-1.75 R0.25
N14 G40 G01 X-2.35 Y-1.8 (Turn off C.C. with an X and/or Y move off of part and then do a finish pass around part using the same part geometry but a different C.C. offset.)
N15 G41 X-2. D01 F8. (Turn on C.C. using D01 for a finish pass using the same program geometry.)
N16 Y1.75
N17 G02 X-1.75 Y2. R0.25
N18 G01 X1.5
N19 G02 X2. Y1.5 R0.5
N20 G01 Y-1.
N21 X-.75 Y-2.
N22 X-1.75
N23 G02 X-2. Y-1.75 R0.25
N24 G40 G01 X-2.35 Y-1.75 (Turn off C.C. with an X and/or Y move off of part.)
N25 G00 Z1. M09
N26 G53 G49 Z0. Y0. M05
N27 M30
    
```



4.0 SQ.

X0 Y0

1.0 SQ.

.5 DIA. E.M.

1.250

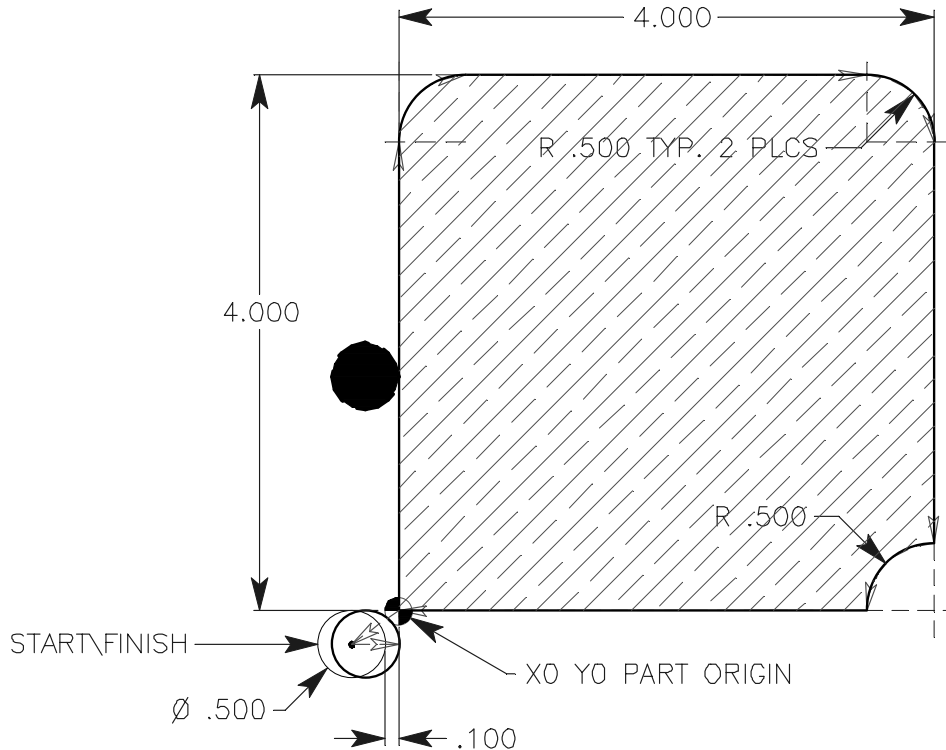
X=-2.5 Y=-2.0 START POINT

R .25 2 PL.

R .50



CUTTER COMPENSATION EXERCISE #1



Cutter will be a .500 dia. end mill. Start contour from the lower left corner of part and mill around outside of part .625 deep. Mill around outside of part using cutter compensation to offset the cutter, and then define the actual part geometry to position the tool around part.

Use a 1/2 dia. End Mill. Mill around outside of part .625 deep using Cutter Comp.

O00030 (CUTTER COMPENSATION EXERCISE #1)

T1 M06 (1/2 DIA. 4 FLT END MILL)

G90 G54 G00 X_____ Y_____ (Position to the center of tool)

S1500 M03

G43 H01 Z0.1 M08

G01 Z_____ F50. (Fast feed to depth non-cutting move)

G___ X_____ D___ F12. (Turn on C.C. moving onto part)

Y_____

G0__ X_____ Y_____ R_____ (or use I_____ and J_____ instead of R)

G0__ X_____

G0__ X_____ Y_____ R_____ (or use I_____ and J_____ instead of R)

G0__ Y_____

G0__ X_____ Y_____ R_____ (or use I_____ and J_____ instead of R)

G0__ X_____ (Position off part at least half the cutter diameter before canceling C.C.)

G___ X_____ Y_____ (Cancel cutter compensation moving off of part)

G00 Z1. M09

G53 G49 Z0. M05

M30



ADVANTAGES OF CUTTER COMPENSATION

1. The mathematical computations for determining a tool path are greatly simplified.
2. Because the geometry and not the tool center is programmed, the same program can be used for a variety of different cutter diameters.
3. When using cutter compensation you are then able to control and adjust for part dimensions using your cutter diameter/radius offsets register.
4. The same program path can be used for the roughing passes as well as finishing cuts by using different cutter offset numbers.

SOME RESTRICTIONS WITH CUTTER COMPENSATION

1. A cutter compensation command (G41, G42 or G40) must be on the same block with an X and/or Y linear command when moving onto or off of the part using cutter comp. You cannot turn on or off cutter compensation with a Z axis move.
2. You can use cutter comp. in the G18 (X, Z) or G19 (Y, Z) planes using G141.
3. You cannot turn ON or OFF cutter compensation in a G02 or G03 circular move, it must be in a linear G00 or G01 straight line move.

WHEN ACTIVATING CUTTER COMPENSATION, CARE MUST BE TAKEN TO:

1. Select a clearance point, without cutter compensation, to a start point in the X and Y axis at least half the cutter diameter off the part before you start initiating cutter compensation.
2. Bring the Z axis down without cutter compensation in effect.
3. Make an X and/or Y axis move with a G41 or G42 call-out on the same line, with a diameter offset Dnn command, which has the cutter diameter value in the offset display register being used.

WHEN DEACTIVATING CUTTER COMPENSATION, CARE MUST BE TAKEN TO:

1. Select a clearance point in X and/or Y axis, at least half the cutter diameter off the part.
2. DO NOT cancel cutter compensation on any line that is still cutting the part.
3. Cancel of cutter compensation (G40) may be a one or two axis move, but you may need values entered for both X and Y axis. This may need to be done to ensure that both axes will position to the location you want, or remain fixed and not move during the cancel (G40) process. This is a programming technique that may be a programmer's preference. The code can be written with only a one axis move, but the programmer should be aware of the results. If only one of the axes, X or Y, is on the cancel (G40) command line, the control will move both axes during the cancel process. And the machine will position the axis that was not defined back to center position of that tool to the last known axis coordinate values.



CUTTER COMPENSATION EXERCISE #2

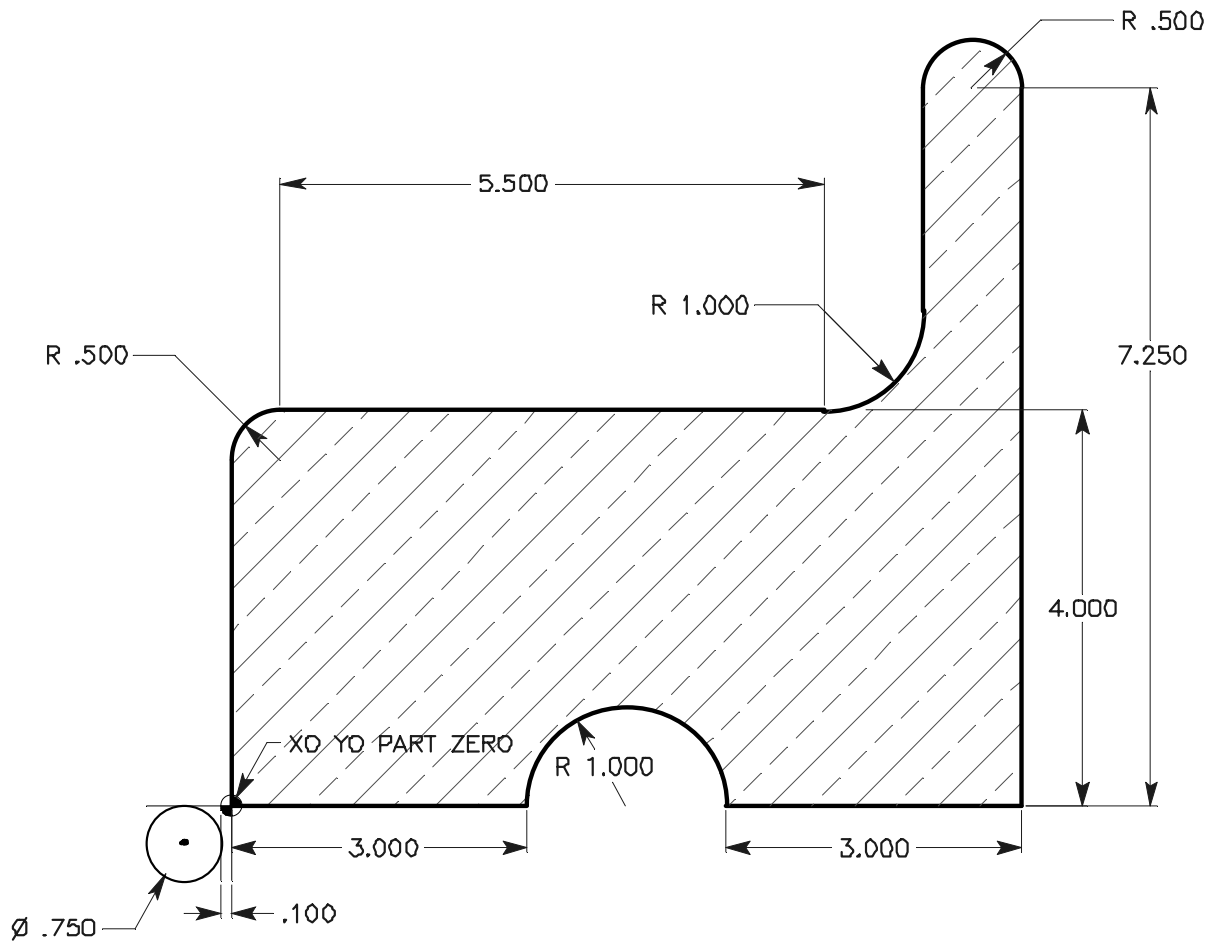
To program this part profile, use the .750 diameter end mill, starting at the lower left hand corner. Use CUTTER COMPENSATION, and climb cut part profile with only one finish pass .450 deep.

(Use the HAAS calculator to get the spindle speed and feed)

TOOL 1 - 3/4 DIA. 4 FLT. END MILL

Surface Speed (FPM) = 220.

Chip Per Flute = .0025





THREAD MILLING WITH HELICAL MOTION

Thread milling uses a cutter formed with the pitch of the thread to be cut. The cutters are solid carbide, fragile and expensive.

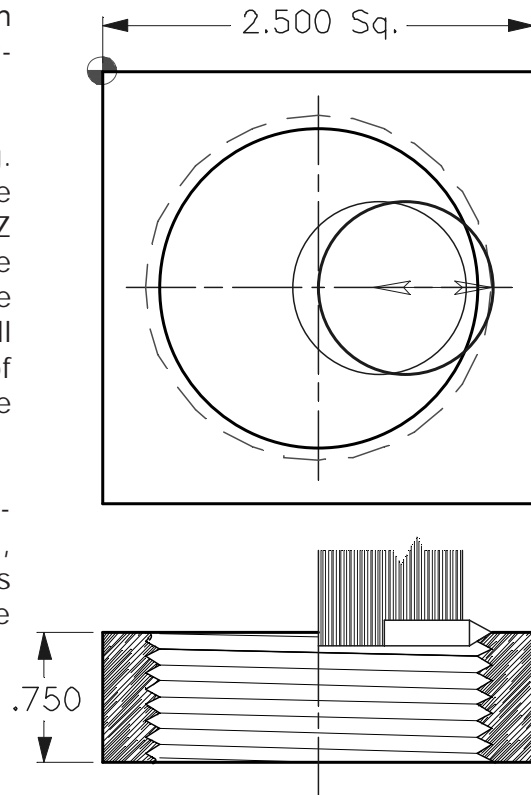
Thread milling is accomplished with Helical milling. Use a standard G02 or G03 move to create the circular move in the X-Y axis and then insert a Z move on the same block corresponding to the thread pitch. This will generate one turn of the thread. If you are using a multiple tooth cutter, it will generate the rest of the thread depth. The speed of each axis will be controlled so that the helical rate matches the programmed feed rate.

Thread milling can be used to machine larger internal or external threads. You can also thread mill, port threads, blind hole threads, metric threads and most all the special type of threads that are good to thread mill.

You can also adjust for the thread size using diameter compensation.

2-12 UN 3B

Minor Dia. - 1.9100/1.9198



To do a complete 360 degree thread pass using a G02 or G03 you need to use an I and/ or J to define the center of a circle for a complete thread pass. Because you cannot do a 360 degree arc using an R command.

O00068 (Looping a Helical Move with a Single Point Tool)

N1 T4 M06 (1.0 dia. single point thread tool to do a 2.0-12 thread)

N2 G90 G54 G00 X1.6 Y-1.25

N3 S1500 M03

N4 G43 H04 Z0.1 M08

N5 G01 Z-0.8 F50.

N6 G41 X2.25 D04 F10. (Move over to thread and turn on C.C.)

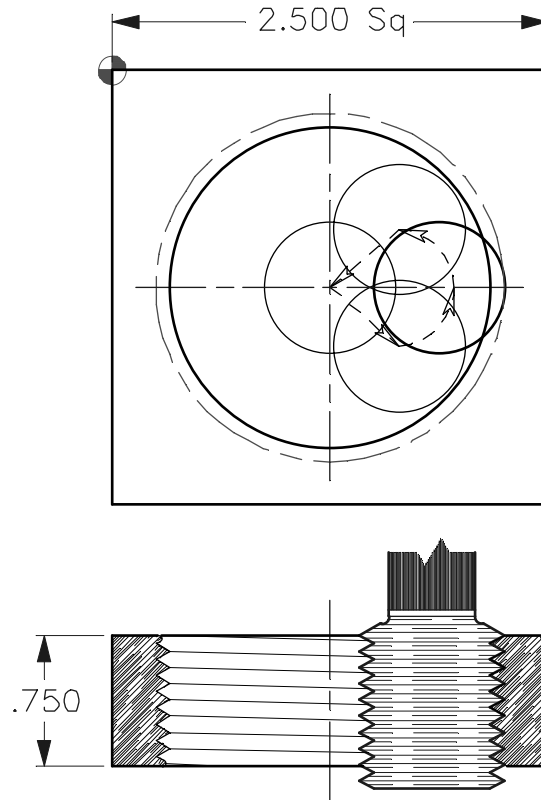
N7 G91 G03 X0. Y0. I-1. J0. Z.0833 F3. L10 (Repeat 10x to mill thread)

N8 G90 G40 G01 X1.6 Y-1.25 (Move off of thread and turn off C.C.)

N9 G00 Z0.1 M09

N10 G53 G49 Z0.

N11 M30



2-12 UN 3B
 Minor Dia. - 1.9100/1.9198

O00069 (A Single Helical Move with a Thread Hob)

N1 T5 M06 (12 pitch x 3/4 inch dia. 4 flt. thread hob for a 2.0-12 thread)

N2 G90 G54 G00 X1.25 Y-1.25

N3 S1500 M03

N4 G43 H05 Z0.1 M08

N5 G01 Z-1.0 F50.

N6 G41 X1.75 Y-1.75 D05 (position over to turn on cutter comp)

N7 G03 X2.25 Y-1.25 R0.5 F10. (arc into thread)

N8 G03 X2.25 Y-1.25 I-1. J0. Z-0.9167 F12. (to mill complete thread)

N9 G03 X1.75 Y-.75 R0.5 (arc off of thread)

N10 G40 G01 X1.25 Y-1.25 (position over to turn off cutter comp)

N11 G00 Z0.1 M09

N12 G53 G49 Z0.

N13 M30



CANNED CYCLES FOR DRILLING TAPPING AND BORING

A canned cycle is used to simplify programming of a part. Canned cycles are defined for the most common Z-axis repetitive operation such as drilling, tapping, and boring. There are 13 canned cycles to choose from and once selected a canned cycle is active until canceled with G80. When active, the canned cycle is executed every time an X and/or Y-axis move is programmed. Those X Y moves are executed as RAPID commands and the Z motions are executed as feed commands, and the canned cycle is performed again for any X-Y coordinates that follow. The five operations in a canned cycle are:

- 1) Positioning of the X and Y axes (and/or optional A axis).
- 2) Rapid traverse to the R plane.
- 3) The drilling, tapping, and boring for the canned cycle being used.
- 4) Operation at the bottom of hole.
- 5) Retraction to the R plane (G99) or initial starting point (G98).

The positioning of a canned cycle in the X and/or Y axes can be done in either in absolute (G90) or incremental (G91). Incremental (G91) motion in a canned cycle is often useful with a loop (Lnn) count which will repeat the canned cycle operation that many times with each incremental X or Y move for the canned cycle.

Incremental example using an "L" repeat command:

```
N15 G81 G99 Z-0.5 R0.1 F6.5 (This will drill one hole at the present location.)  
N16 G91 X-0.5625 L9 (To drill 9 more holes .5625 in the minus direction equally spaced.)  
N17 G90 X~~~~ Y~~~~
```

If an L0 is in the canned cycle line, the cycle will not execute until the control reads the next X and/or Y position location. One reason to define an L0 is for not drilling a hole in the center of a bolt hole pattern.

Using L0 so as to not drill a hole in the center of bolt circle example:

```
N21 X1.25 Y-0.75 (Center location of bolt hole pattern)  
N22 G81 G99 Z-0.5 R0.1 F6.5 L0 (L0 on G81 line so as not to drill hole in the center of BHC)  
N23 G70 I0.75 J10. L6 (6 hole Bolt-Hole-Circle)
```

Once a canned cycle is defined, that operation is performed at every X-Y position subsequently listed in a block. Some of the canned cycle numerical values can also be changed after the canned cycle is defined. The most important of these are the R plane value and the Z depth value, and all subsequent cycles are performed with the new R or Z value.

The G98 and G99 are modal commands which change the way the canned cycles operate. When G98 (machine default value) is active, the Z-axis will return to the initial start point at the completion of the canned cycle for clearance over to the next X Y hole location. When G99 is active, the Z-axis will return to the rapid (R) plane when the canned cycle is completed for clearance to the next X Y location. Changes to the G98/G99 selection can also be made after the canned cycle is active. If changed, the new G98/G99 value will change all subsequent canned cycle.



CANNED CYCLES

G CODE	Z Drilling Operation	Operation at the End of Hole	Z axis Retraction	Application
G73	intermittent feed in	dwel (optional)	rapid out	high speed peck drilling
G74	spindle CCW feed in	spindle reverse CW	feed out	left hand tapping
G76	feed in	dwel, spindle stop, orient, shift	rapid out	fine boring
G77	spindle stop, shift, rapid in, shift, feed up	orient spindle, shift	rapid out	back boring
G81	feed in	none	rapid out	drilling
G82	feed in	dwel	rapid out	spot drilling
G83	intermittent feed in	dwel (optional)	rapid out	peck drilling cycle
G84	spindle CW feed in	spindle reverse CCW	feed out	tapping cycle
G85	feed in	none	feed out	boring cycle
G86	feed in	spindle stop,orient	rapid out	boring cycle
G87	feed in	spindle stop, orient	manual jog out	boring cycle
G88	feed in	dwel, spindle stop, orient	manual jog out	boring cycle
G89	feed in	dwel	rapid out	boring cycle

Setting 27 - G76/G77 SHIFT DIR. - More information aobout this setting is defined with the G76 and G77 codes.

Setting 28 - CAN CYCLE ACT W/O X/Y - For all canned cycles

This is an On/Off setting. If a canned cycle is defined in a block without an X or Y location, there are two common actions taken by other controls; some will execute the canned cycle at that time and some will not. When it is off, an initial canned cycle definition without an X or Y motion will not cause the canned cycle to be executed. When it is on, the initial canned cycle definition will cause one cycle to be executed even if there is no X or Y motion in that Canned Cycle command block. Most users prefer having this ON.

Setting 52 - G83 RETRACT ABOVE R - More information is defined with G83 about this setting.

Setting 57 - EXACT STOP CANNED X-Y - For all canned cycles

This is an On/Off setting. When it is off, the rapid X-Y motion associated with a canned cycle may not get exact stop; according to other conditions. When it is on, the X-Y motion always gets exact stop. This will make canned cycles slower but less likely to run into a close tolerance/interference fixture. Most users will have this OFF.

G80 CANNED CYCLE CANCEL

The G80 code is used to cancel a canned cycle. This G code is modal in that it deactivates all canned cycles G73, G74, G76, G77, or G81-G89 until a new one is selected. In addition to this, a G00 or G01 code will also cancel any active canned cycle.



CANNED CYCLE RETURN PLANES

G98 CANNED CYCLE INITIAL POINT RETURN

This G code is modal and changes the way a canned cycle operates. With G98, the Z axis returns to its initial starting point (Z position it was at when the canned cycle was first defined) between each new X and/or Y location. This allows for positioning up and around areas of a part, fixture, and/or clamp that is in the way of a tool to the next location.

G99 CANNED CYCLE R PLANE RETURN

This G code is modal and changes the way a canned cycle operates. With G99, the Z axis will stay down at the R plane between each new X and/or Y location, when there is no obstruction is in the way of the tool to the next location.

O00072 (G98/G99 Return Plane in a Canned Cycle)

N1 T2 M06 (7/16 DIA. CARBIDE DRILL)

N2 G90 G54 G00 X1.5 Y-0.5 (1)

N3 S1200 M03

N4 G43 H02 Z1. M08 (initial start point is Z1.0)

N5 G83 G99 Z-.625 Q.2 R0.1 F8. (G99 rapids to R.1 between holes)

N6 X0.5 Y-0.75 (2)

N7 Y-2.25 (3)

N8 G98 X1.5 Y-2.5 (G98 position to initial start point after drilling) (4)

N9 G99 X3.5 Z-1.2 R-0.4 (stays down at R plane after drilling hole) (5)

N10 X4.5 Y-2.25 (6)

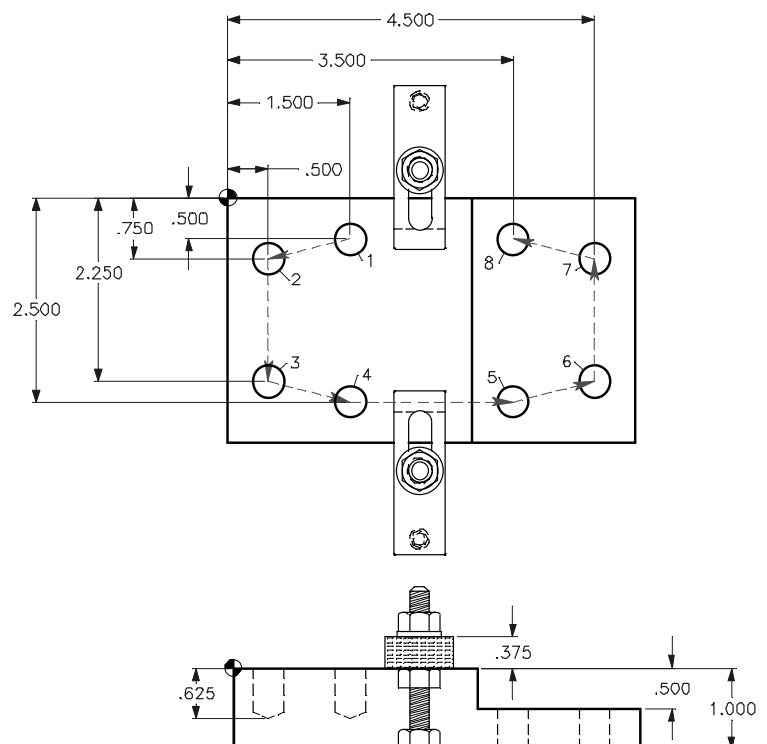
N11 Y-0.75 (7)

N12 X3.5 Y-0.5 (8)

N13 G80 G00 Z1. M09

N14 G53 G49 Z0. M05

N15 M30

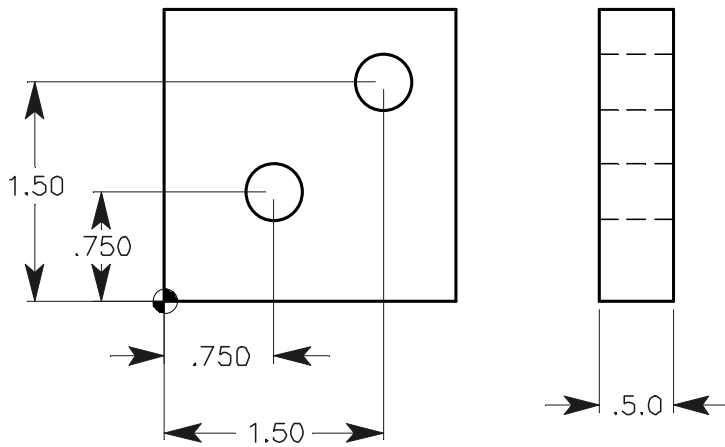




G81 DRILL CANNED CYCLE

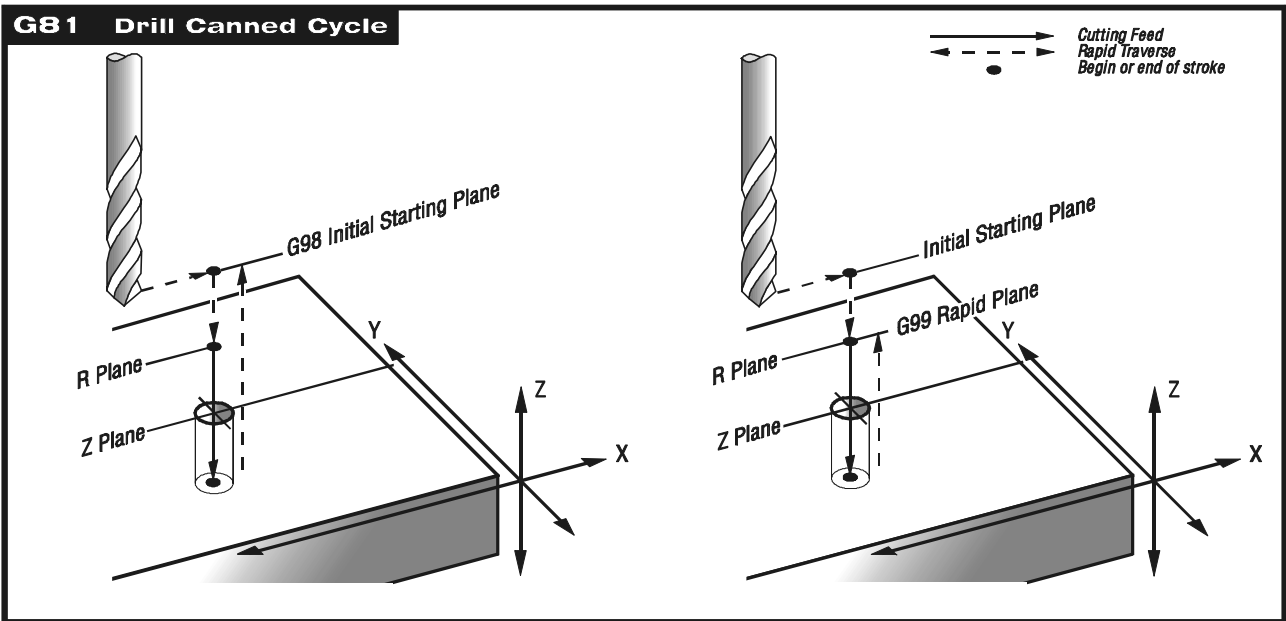
- X* Rapid X-axis location
- Y* Rapid Y-axis location
- Z Z-depth (feed to Z-depth starting from R plane)
- R R-plane (rapid point to start feeding)
- F Feed rate in inches (mm) per minute
- * Indicates optional

This G code is modal so that it is activated every X and/or Y axis move, and it will rapid to that position and then cause this canned cycle to be executed again, until it's canceled. Use G98 and G99 for the Z position clearance location for positioning between holes.



```

O00073 (G81 Drilling Cycle)
N1 T1 M06 (1/2 DIA. DRILL)
N2 G90 G54 G00 X0.75 Y0.75
N3 S1450 M03
N4 G43 H01 Z1. M08
N5 G81 G99 Z-0.625 R0.1 F10.
N6 X1.5 Y1.5
N7 G80 G00 Z1. M09
N8 G53 G49 Z0. M05
N9 M30
    
```

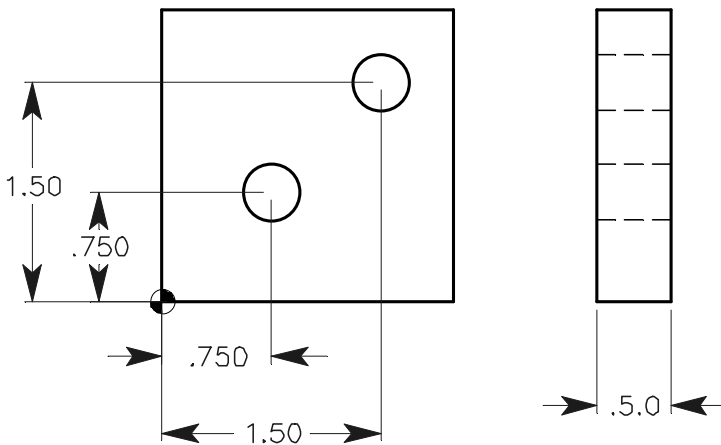




G82 SPOT DRILL~COUNTERBORE CANNED CYCLE

- X* Rapid X-axis location
- Y* Rapid Y-axis location
- Z Z-depth (feed to Z-depth starting from R plane)
- P Dwell time at Z-depth
- R R-plane (rapid point to start feeding)
- F Feed rate in inches (mm) per minute
- * Indicates optional

This G code is modal so that it is activated every X and/or Y axis move, and it will rapid to that position and then cause this canned cycle to be executed again, until it's canceled. A dwell in seconds/milliseconds is caused at the bottom of each Z-depth in this cycle which is defined with P. Use G98 and G99 for the Z position clearance location for positioning between holes.



O00074 (G82 Drill-Dwell Cycle)

N1 T2 M06 (1/2 DIA. DRILL)

N2 G90 G54 G00 X0.75 Y0.75

N3 S1450 M03

N4 G43 H02 Z1. M08

N5 G82 G99 Z-0.625 P1.5 R0.1 F10.

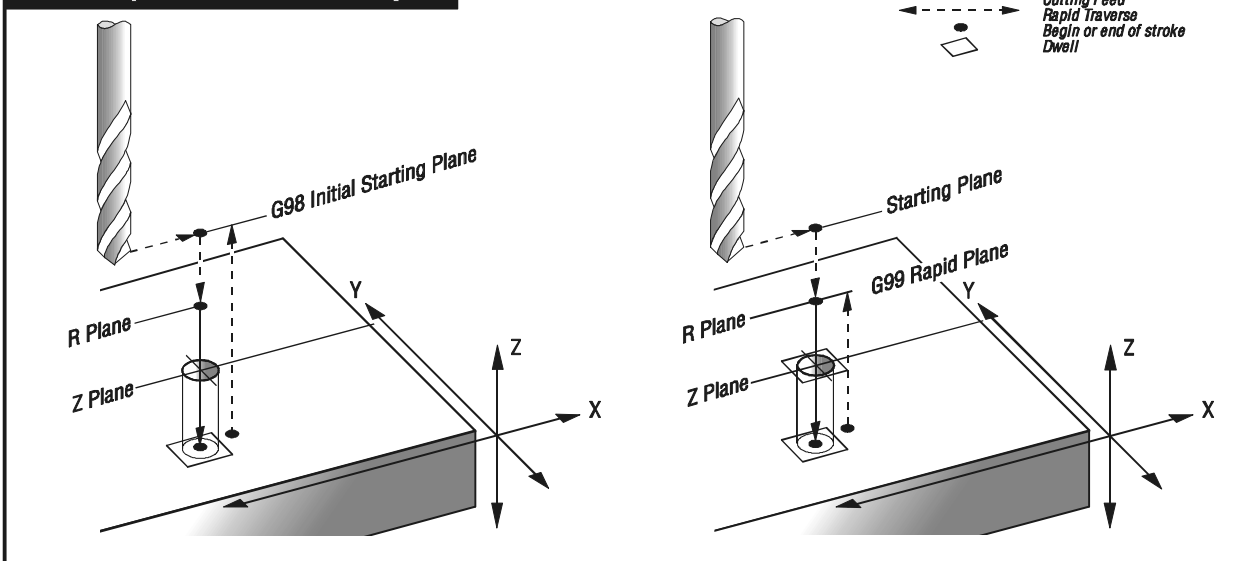
N6 X1.5 Y1.5

N7 G80 G00 Z1. M09

N8 G53 G49 Z0. M05

N9 M30

G82 Spot Drill Canned Cycle



**G83 DEEP HOLE PECK DRILL CANNED CYCLE**

- X* Rapid X-axis location
- Y* Rapid Y-axis location
- Z Z-depth (feed to Z-depth starting from R plane)
- Q* Pecking equal incremental depth amount (if I, J and K are not used)
- I* Size of first peck depth (if Q is not used)
- J* Amount reducing each peck after first peck depth (if Q is not used)
- K* Minimum peck depth (if Q is not used)
- P Dwell time at Z-depth
- R R-plane (rapid point to start feeding)
- F Feed rate in inches (mm) per minute

* Indicates optional

This G code is modal so that it is activated every X and/or Y axis move, and it will rapid to that position and then cause this canned cycle to be executed again, until it's canceled. And the depth for each peck in this cycle will be the amount defined with Q. Then the tool will rapid up to the R plane after each peck and then back in for the next peck until Z depth is reached. Use G98 and G99 for the Z position clearance location for positioning between holes.

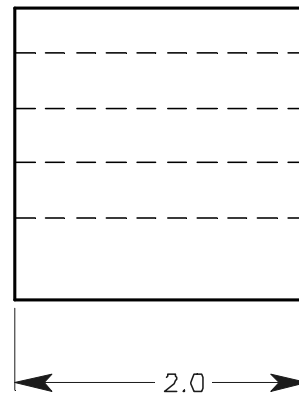
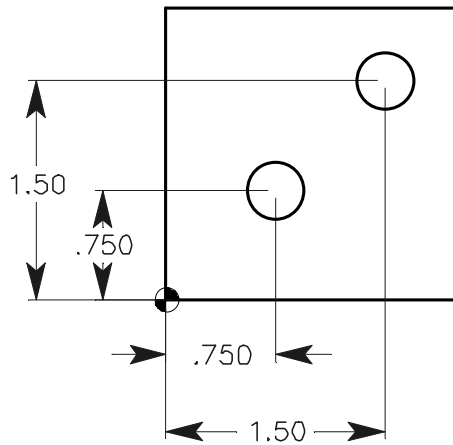
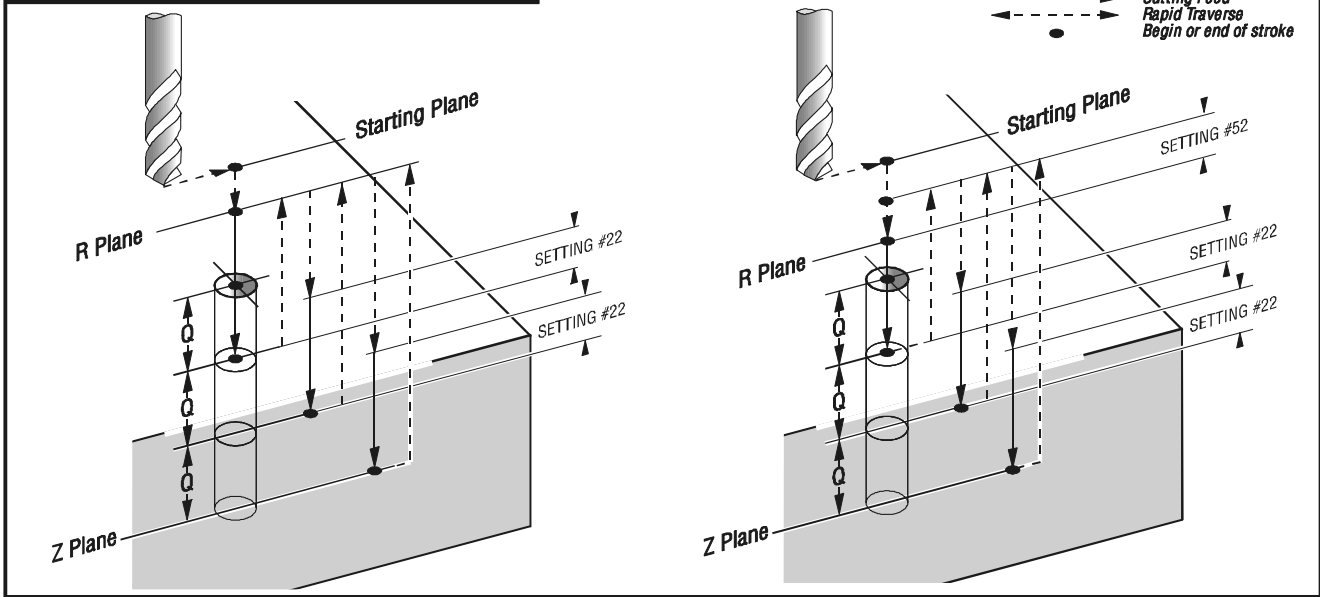
If I, J, and K are specified, a different operating mode is selected. The first pass will cut in by I, each succeeding cut will be reduced by amount J, and the minimum cutting depth is K.

Setting 22 - As the tool pecks deeper into the hole, with each peck it rapids out to the R-plane, and then back in to a constant specified distance above the bottom of the hole that was created by the previous peck. That specified distance is defined in Setting 22.

Setting 52 - Changes the way G83 works when it returns to the R-plane. Most programmers set the R-plane well above the cut to insure that the chip clear motion actually allows the chips to get out of the hole but this causes a wasted motion when first drilling through this "empty" space. If Setting 52 is set to the distance required to clear chips, the R plane can be put much closer to the part being drilled. When the clear move to R occurs, the Z will be moved above R by this setting.

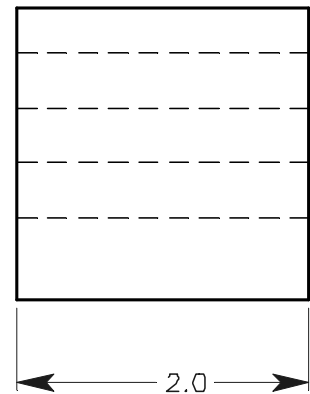
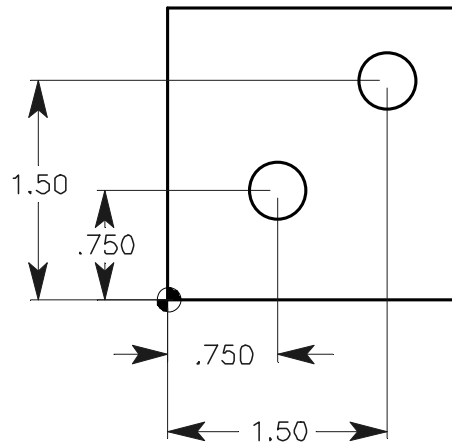
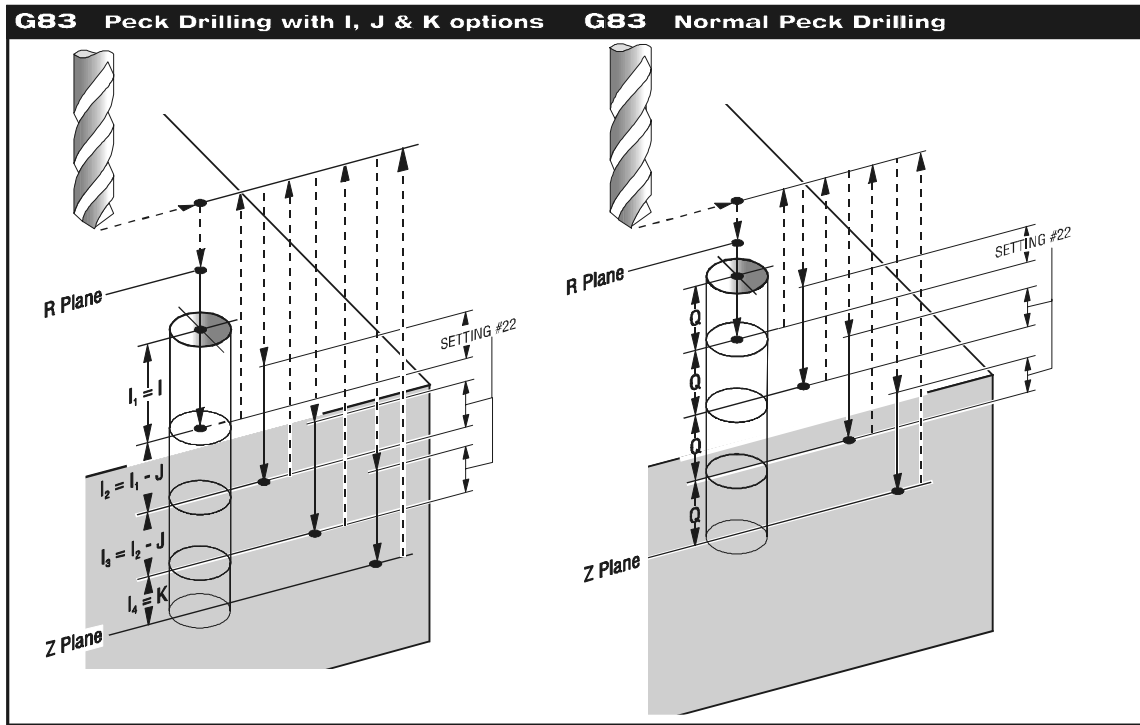


G83 Peck Drilling Canned Cycle



```

O00076 (G83 Deep Hole Peck Drill Using Q)
N1 T3 M06 (1/2 DIA. DRILL)
N2 G90 G54 G00 X0.75 Y0.75
N3 S1451 M03
N4 G43 H03 Z1. M08
N5 G83 G99 Z-2.16 Q0.5 R0.1 F10.
N6 X1.5 Y1.5
N7 G80 G00 Z1. M09
N8 G53 G49 Z0. M05
N9 M30
  
```

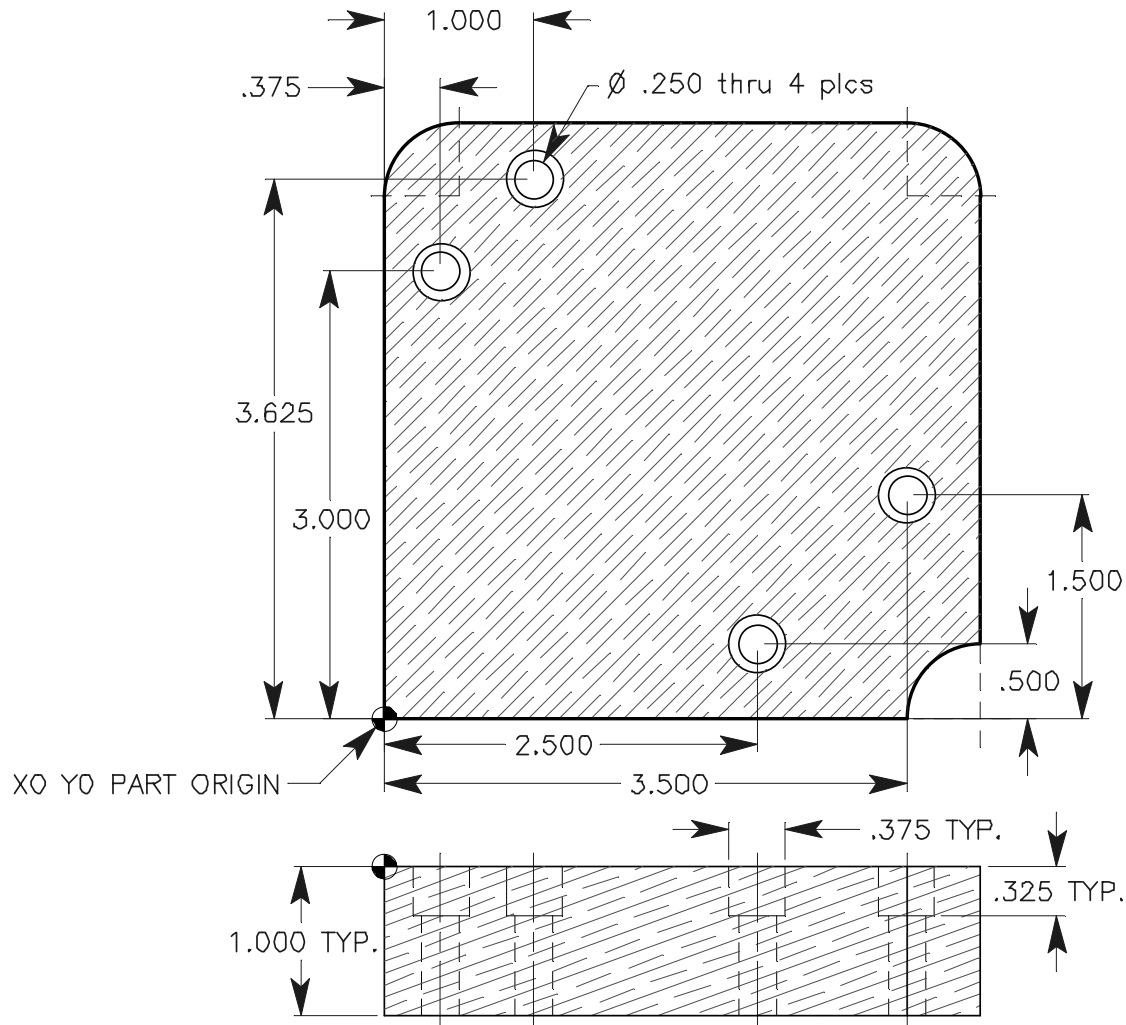


```

O00077 (G83 Deep Hole Peck Drill Using I, J & K)
N1 T3 M06 (1/2 DIA. DRILL)
N2 G90 G54 G00 X0.75 Y0.75
N3 S1451 M03
N4 G43 H03 Z1. M08
N5 G83 G99 Z-2.16 I0.5 J0.1 K0.2 R0.1 F10.
N6 X1.5 Y1.5
N7 G80 G00 Z1. M09
N8 G53 G49 Z0. M05
N9 M30
    
```



CANNED CYCLE EXERCISE #1



TOOL 3 - is a 1/2 DIA. SPOT DRILL - Drill To a depth of .200 to leave a .400 dia. spot for a counterbore chamfer. (use a G81 canned cycle)

S1604 Spindle Speed

(Surface Speed is 210.)

F6.5 Feed Per Minute

(Chip Load is .002)

TOOL 4 is a 1/4 DIA. DRILL - Drill thru to a 1.2 depth (use a G83 canned cycle with a .250 peck amount).

S2903 Spindle Speed

(Surface Speed is 190.)

F14.5 Feed Per Minute

(Chip Load is .0025)

TOOL 5 is a 3/8 DIA. 4 FLT. END MILL - Drill to a .325 depth (use a G82 canned cycle with a dwell for a 1/2 second).

S2037 Spindle Speed

(Surface Speed is 200.)

F12.2 Feed Per Minute

(Chip Load is .0015)



CANNED CYCLE EXERCISE #1

O00050 (CANNED CYCLE EXERCISE #1)

T___ M___ (T3 - 1/2 DIA. SPOT DRILL)

G___ G___ G___ X___ Y___

S___ M___

G___ H___ Z___ M___

G___ G___ Z___ R___ F___

X___ Y___

X___ Y___

X___ Y___

G___ G___ Z___ M___

G___ G___ Z___ M___

M___ (Optional stop)

T___ M___ (T4 - 1/4 DIA. DRILL)

G___ G___ G___ X___ Y___

S___ M___

G___ H___ Z___ M___

G___ G___ Z___ Q___ R___ F___

X___ Y___

X___ Y___

X___ Y___

G___ G___ Z___ M___

G___ G___ Z___ M___

M___ (Optional stop)

T___ M___ (T5 - 3/8 DIA. 4 FLT. END MILL)

G___ G___ G___ X___ Y___

S___ M___

G___ H___ Z___ M___

G___ G___ Z___ P___ R___ F___

X___ Y___

X___ Y___

X___ Y___

G___ G___ Z___ M___

G___ G___ Z___ M___

M___ (Program End and Reset)



G84 TAPPING CANNED CYCLE

- X* Rapid X-axis location
- Y* Rapid Y-axis location
- Z Z-depth (tapping Z-depth starting from R plane)
- J* Tapping Retract Speed (Rev. 10.13 and above)
- R R-plane (rapid point to start feeding)
- F Feed rate in inches (mm) per minute
- * Indicates optional

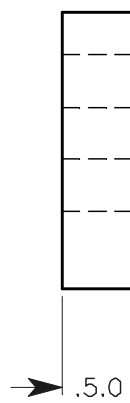
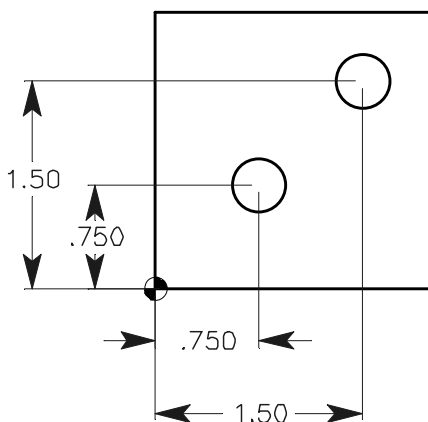
You don't need to start the spindle with a M03 for a tap that's using G84 because this cycle will turn on the spindle for you automatically and it will do it quicker.

This G code is modal. Use G98 and G99 for the Z position clearance location.

On older machines without vector motors, if your using a spindle speed that's in low gear you, may want to command M42 to force it into high gear, because most tapping (a smaller size tap) operations don't need the torque of low gear. And in high gear the tapping operation performs quicker.

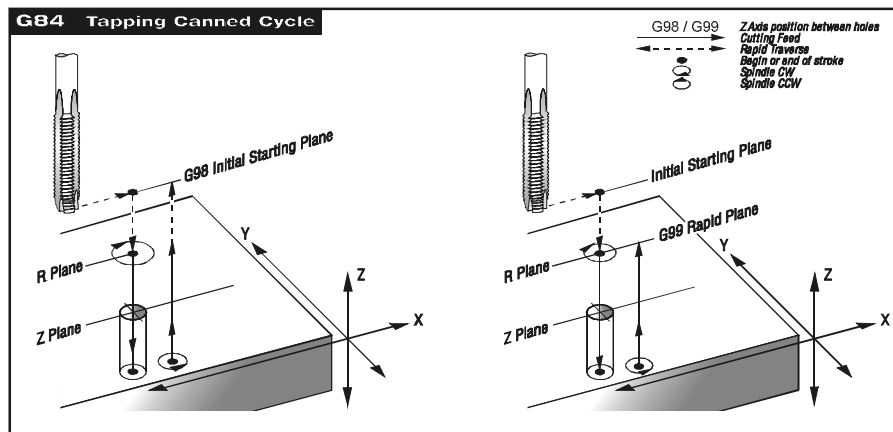
Newer machines have Setting 130, Tap Retract Speed, can be set with, 1 thru 9, to quick-reverse-out of thread up to 9 times faster then going in, If J is not used.

With Rigid Tapping, the ratio between feedrate and spindle speed must be calculated for thread pitch being cut. The calculation is $1 \div \text{Threads Per Inch} \times \text{rpm} = \text{tapping feedrate}$. Use the Haas calculator for the speed and feed numbers.



```

O00080 (G84 R.H. Tapping Cycle)
N1 T4 M06 (7/16-14 TAP)
N2 G90 G54 G00 X0.75 Y0.75
N3 S450 (You don't need M03, the
G84 turns on the spindle for you.)
N4 G43 H04 Z1. M08
N5 G84 G99 Z-0.65 R0.1 J3 F32.1429
N6 X1.5 Y1.5
N7 G80 G00 Z1. M09
N8 G53 G49 Z0. M05
N9 M30
  
```

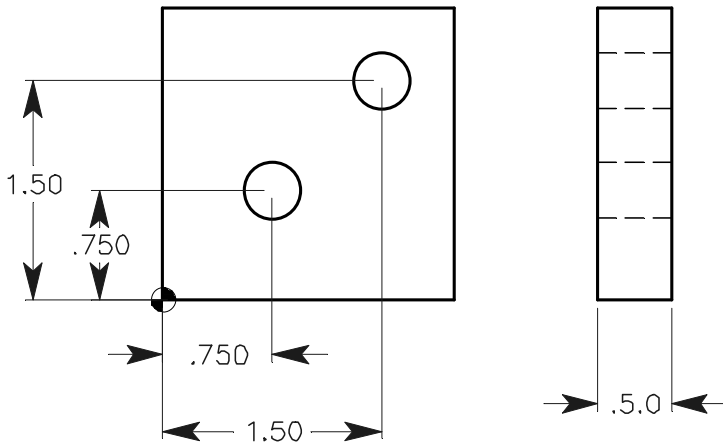




G85 BORE IN~BORE OUT CANNED CYCLE

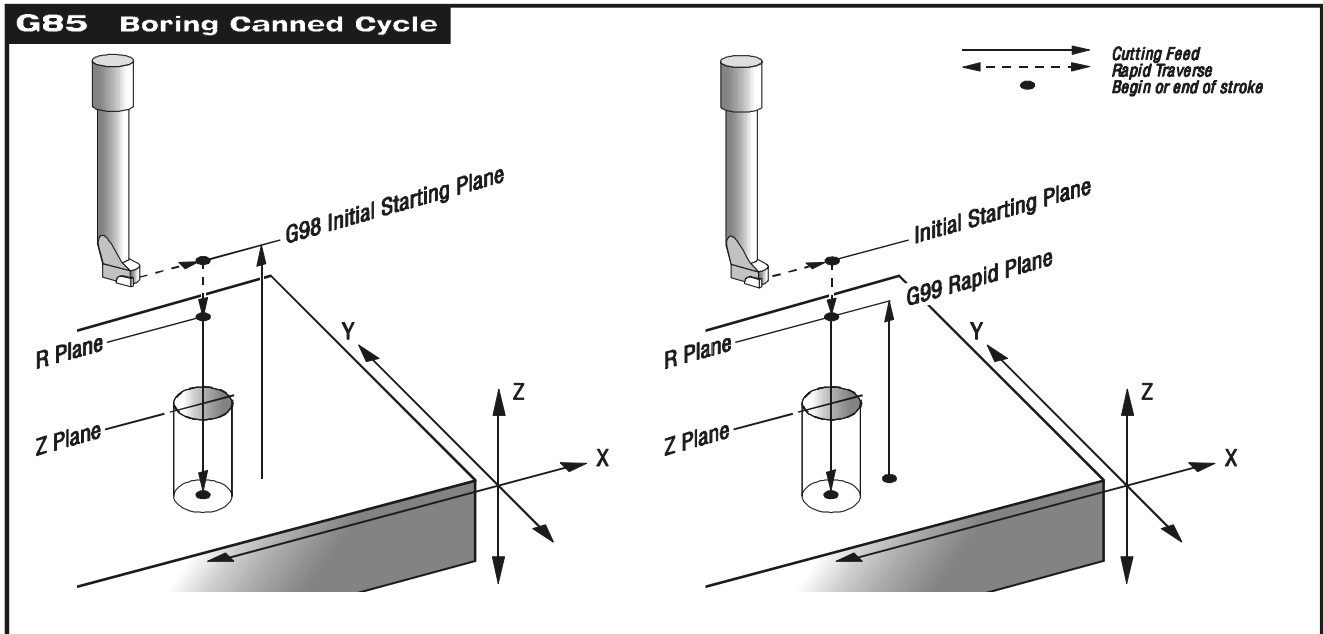
- X* Rapid X-axis location
- Y* Rapid Y-axis location
- Z Z-depth (feed to Z-depth starting from R plane)
- R R-plane (rapid point to start feeding)
- F Feed rate in inches (mm) per minute
- * Indicates optional

This G code is modal so that it is activated every X and/or Y axis move, and it will rapid to that position and then cause this canned cycle to be executed again, until it's canceled. Use G98 and G99 for the Z position clearance location for positioning between holes.



```

O00081 (G85 Bore In-Bore Out)
N1 T5 M06 (BORING BAR)
N2 G90 G54 G00 X0.75 Y0.75
N3 S1451 M03
N4 G43 H05 Z1. M08
N5 G85 G99 Z-.55 R0.1 F4.5
N6 X1.5 Y1.5
N7 G80 G00 Z1. M09
N8 G53 G49 Z0. M05
N9 M30
    
```

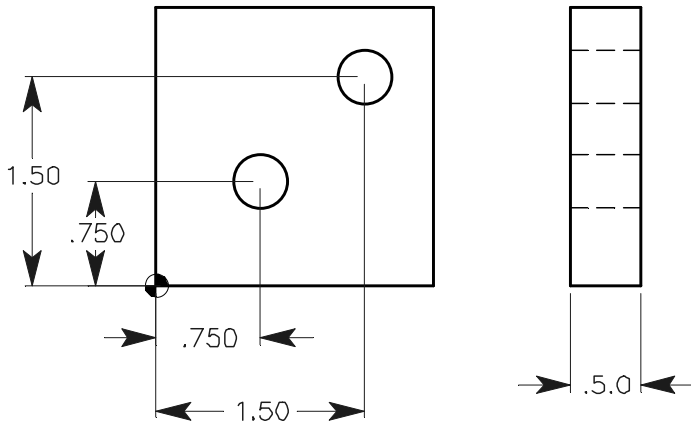




G86 BORE IN~STOP~RAPID OUT CANNED CYCLE

- X* Rapid X-axis location
- Y* Rapid Y-axis location
- Z Z-depth (feed to Z-depth starting from R plane)
- R R-plane (rapid point to start feeding)
- F Feed rate in inches (mm) per minute
- * Indicates optional

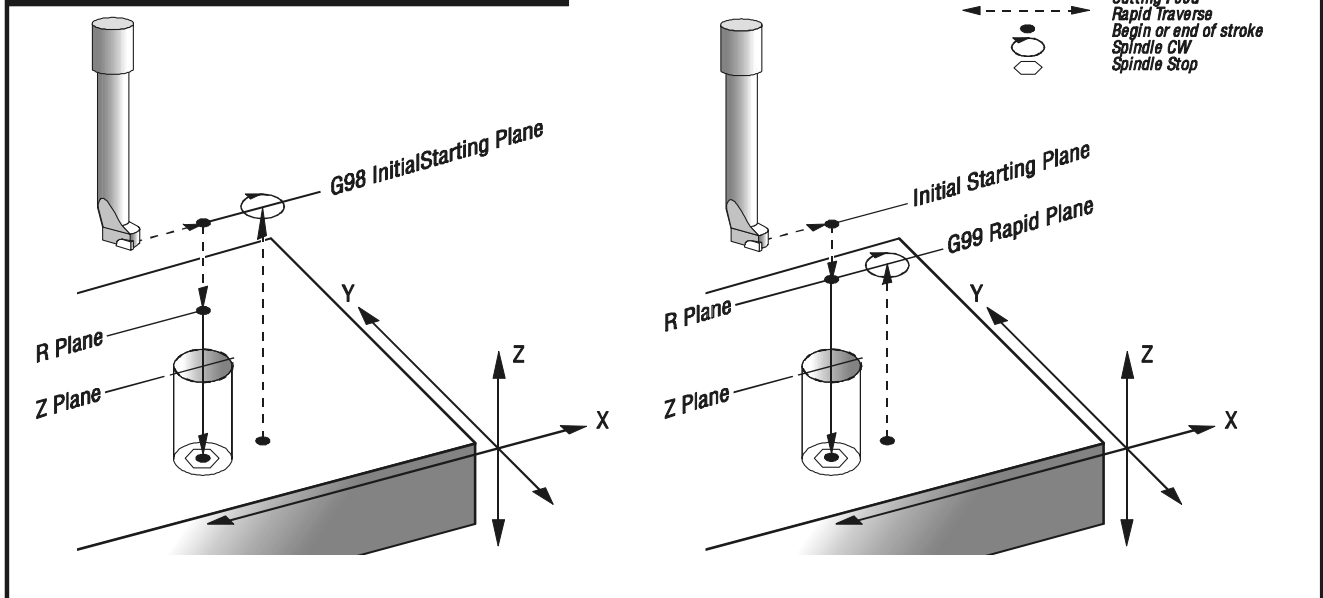
This G code is modal so that it is activated every X and/or Y axis move, and it will rapid to that position and then cause this canned cycle to be executed again, until it's canceled. Use G98 and G99 for the Z position clearance location for positioning between holes.



```

O00082 (G86 Bore-Stop-Rapid Out)
N1 T6 M06 (BORING BAR)
N2 G90 G54 G00 X0.75 Y0.75
N3 S1451 M03
N4 G43 H06 Z1. M08
N5 G86 G99 Z-0.55 R0.1 F4.5
N6 X1.5 Y1.5
N7 G80 G00 Z1. M09
N8 G53 G49 Z0. M05
N9 M30
  
```

G86 Bore and Stop Canned Cycle

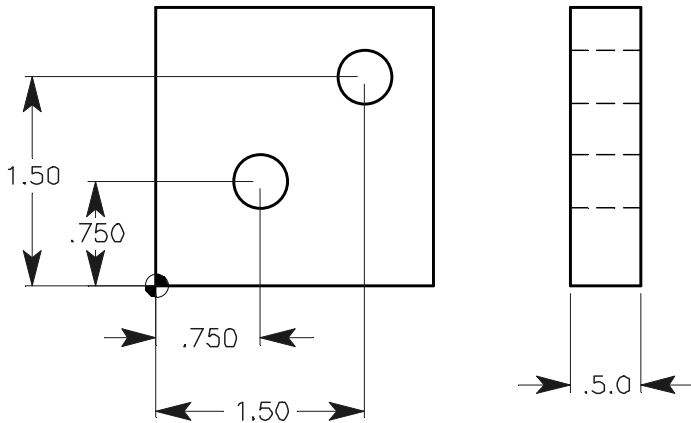




G87 BORE IN-MANUAL RETRACT CANNED CYCLE

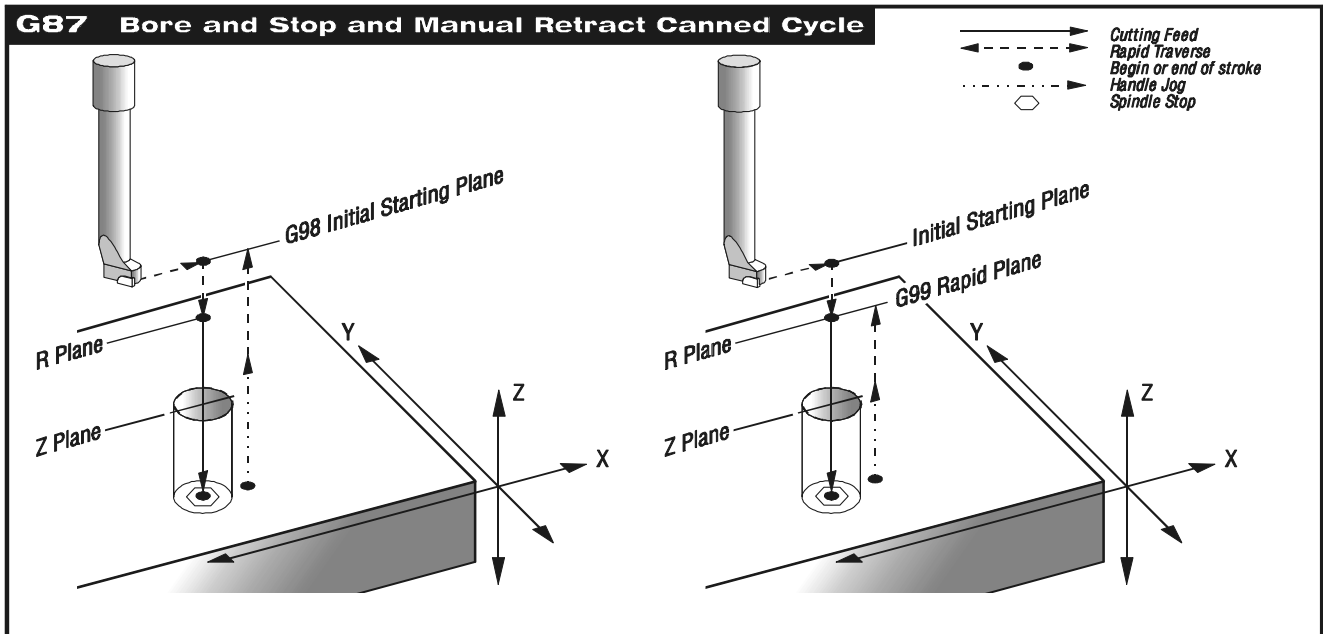
- X* Rapid X-axis location
- Y* Rapid Y-axis location
- Z Z-depth (feed to Z-depth starting from R plane)
- R R-plane (rapid point to start feeding)
- F Feed rate in inches (mm) per minute
- * Indicates optional

This G code is modal so that it is activated every X and/or Y axis move, and it will rapid to that position and then cause this canned cycle to be executed again, until it's canceled. Use G98 and G99 for the Z position clearance location for positioning between holes.



```

O00083 (G87 Bore-Manual Retract)
N1 T7 M06 (BORING BAR)
N2 G90 G54 G00 X0.75 Y0.75
N3 S1451 M03
N4 G43 H07 Z1. M08
N5 G87 G99 Z-0.55 R0.1 F4.5
N6 X1.5 Y1.5
N7 G80 G00 Z1. M09
N8 G53 G49 Z0. M05
N9 M30
    
```

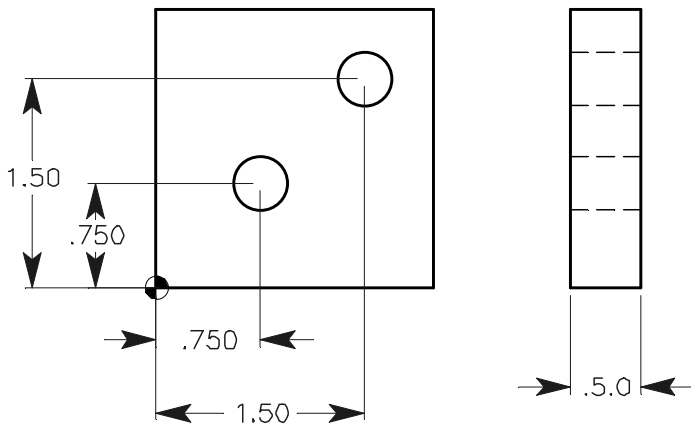




G88 BORE IN~DWELL~MANUAL RETRACT CANNED CYCLE

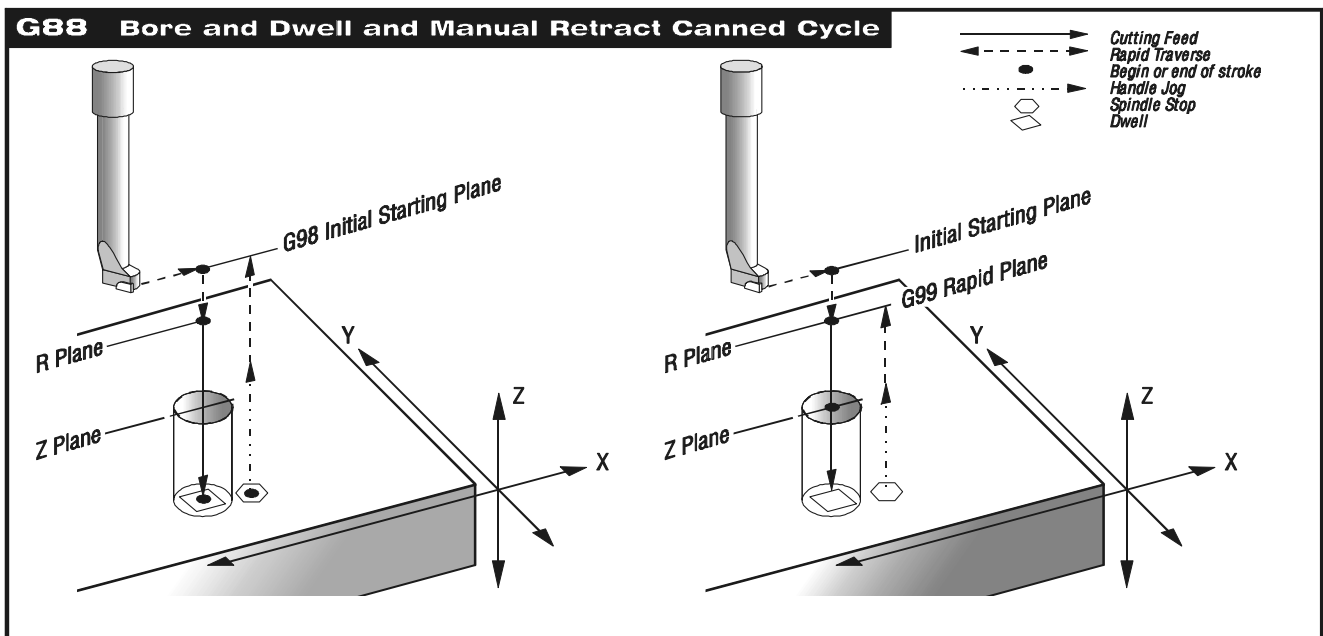
- X* Rapid X-axis location
- Y* Rapid Y-axis location
- Z Z-depth (feed to Z-depth starting from R plane)
- P Dwell time at Z-depth
- R R-plane (rapid point to start feeding)
- F Feed rate in inches (mm) per minute
- * Indicates optional

This G code is modal so that it is activated every X and/or Y axis move, and it will rapid to that position and then cause this canned cycle to be executed again, until it's canceled. Use G98 and G99 for the Z position clearance location for positioning between holes.



```

O00084 (G88 Bore-Dwell-Manual)
N1 T8 M06 (BORING BAR)
N2 G90 G54 G00 X0.75 Y0.75
N3 S1451 M03
N4 G43 H08 Z1. M08
N5 G88 G99 Z-.55 P1.5 R0.1 F4.5
N6 X1.5 Y1.5
N7 G80 G00 Z1. M09
N8 G53 G49 Z0. M05
N9 M30
  
```

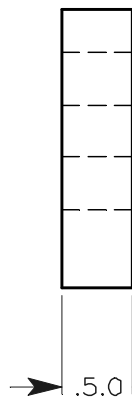
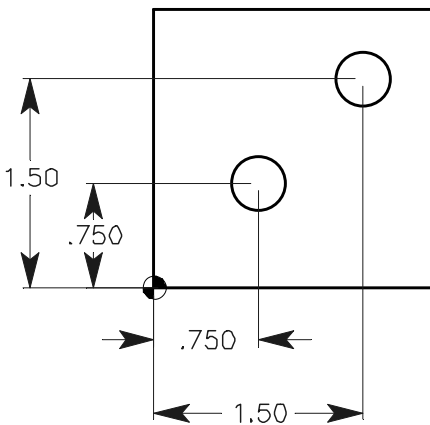




G89 BORE IN~DWELL~BORE OUT CANNED CYCLE

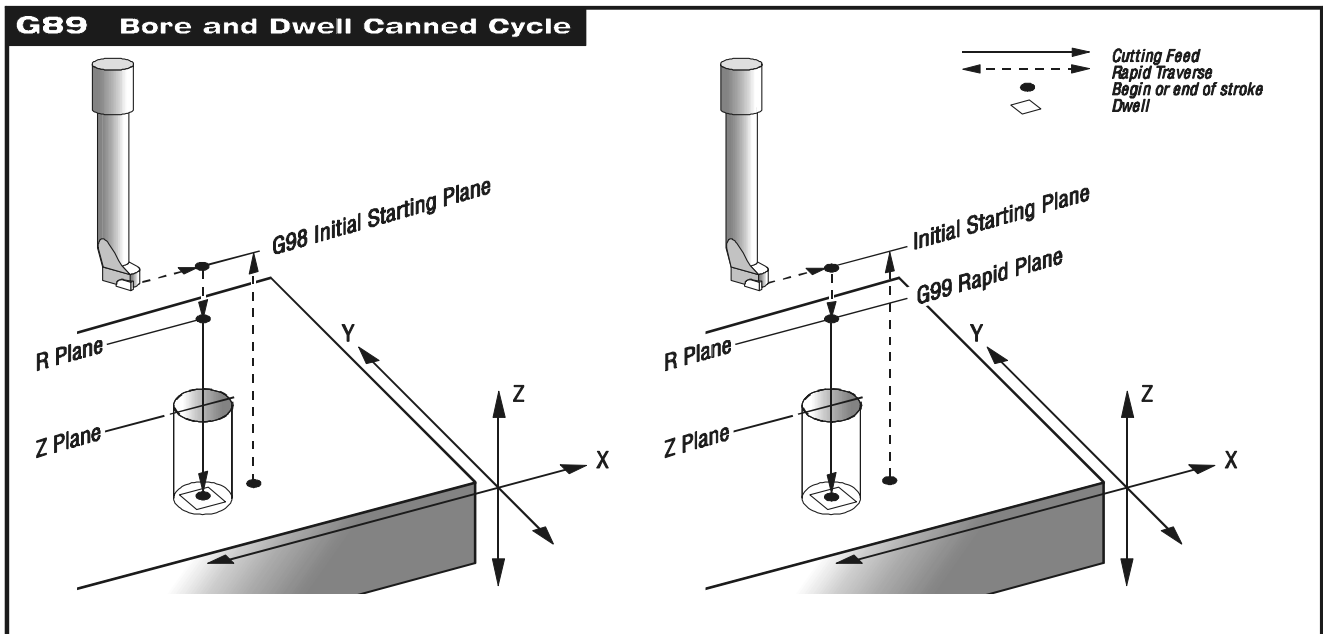
- X* Rapid X-axis location
- Y* Rapid Y-axis location
- Z Z-depth (feed to Z-depth starting from R plane)
- P Dwell time at Z-depth
- R R-plane (rapid point to start feeding)
- F Feed rate in inches (mm) per minute
- * Indicates optional

This G code is modal so that it is activated at every X and/or Y axis move, and will rapid to that position and cause this canned cycle to be executed again, until it's canceled. A dwell in this cycle in seconds". "milliseconds will happen at the end of the Z-depth with P defined. Use G98 and G99 for the Z position clearance location for positioning between holes.



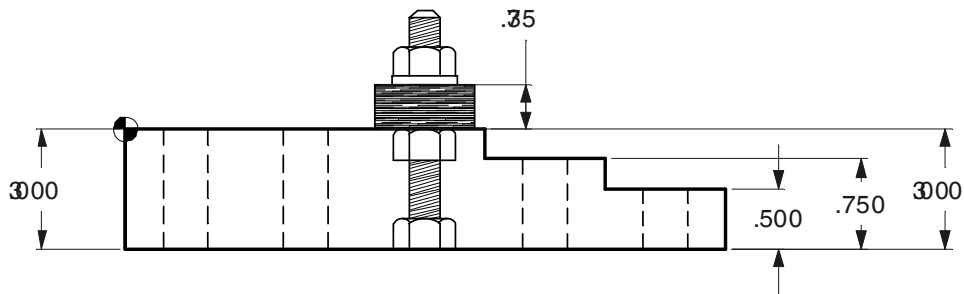
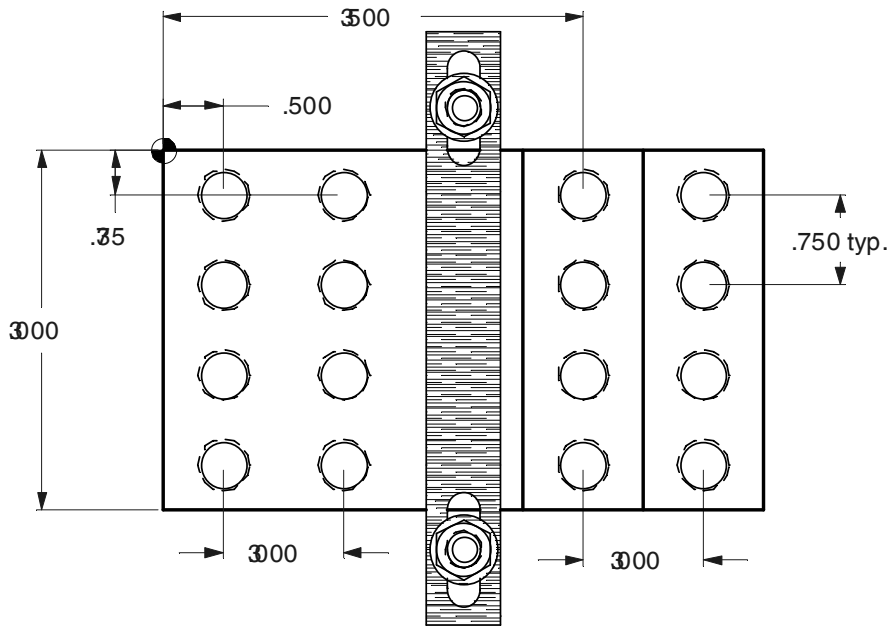
```
O00085 (G89 Bore In-Dwell-Bore Out)
N1 T9 M06 (BORING BAR)
N2 G90 G54 G00 X0.75 Y0.75
N3 S1451 M03
N4 G43 H09 Z1. M08
N5 G89 G99 Z-0.5 P1.0 R0.1 F4.5
N6 X1.5 Y1.5
N7 G80 G00 Z1. M09
N8 G53 G49 Z0. M05
N9 M30
```

G89 Bore and Dwell Canned Cycle





CANNED CYCLE EXERCISE #2



Be sure to calculate depth of tools from the surface that is being machined.

Change the R-plane according to the surface you are machining.

Use the Haas CALC display for spindle speed and feedrate.

TOOL #1 - 90 DEG. 5/8 (.625) DIA. SPOT DRILL TO A .25 DEPTH

Use G81 Canned Cycle

Surface Speed is 230. SFM with a Chip Load of .0025

S _____

F _____

TOOL #2 - 27/64 (.4219) DIA. DRILL TO 1.15 THRU

Use G83 Canned Cycle with a Q.25 peck amount

Surface Speed is 230. SFM with a Chip Load of .003

S _____

F _____

TOOL #3 - 1/2-13 TAP WITH RIGID TAP THRU TO 1.2 DEPTH

Use G84 Canned Cycle

Use tapping calculator to get a feedrate for 600 RPM

S 600

F _____



000060 (CANNED CYCLE EXERCISE #2)

Lined area for writing the canned cycle exercise code.



G73 HIGH SPEED PECK DRILL CANNED CYCLE

- X* Rapid X-axis location
 - Y* Rapid Y-axis location
 - Z Z-depth (feed to Z-depth starting from R plane)
 - Q* Pecking equal incremental depth amount (if I, J and K are not used)
 - I* Size of first peck depth (if Q is not used)
 - J* Amount reducing each peck after first peck depth (if Q is not used)
 - K* Minimum peck depth (if Q is not used)
 - P* Dwell time at Z-depth
 - R R-plane (rapid point to start feeding)
 - F Feed rate in inches (mm) per minute
- * Indicates optional

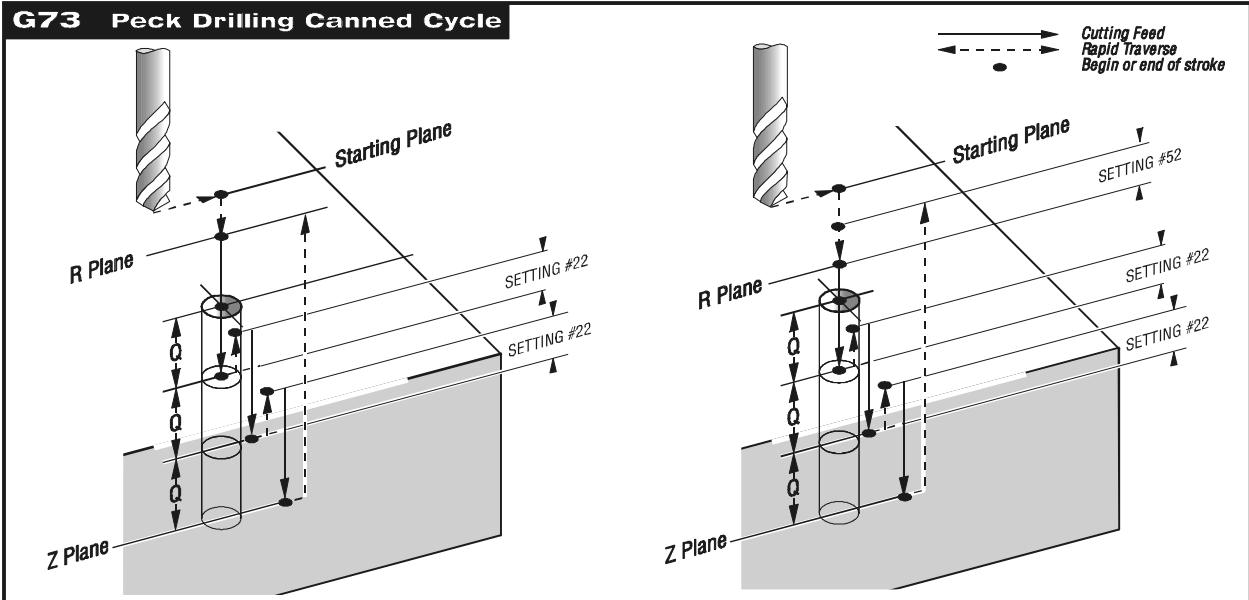
This G code is modal so that it is activated every X and/or Y axis move, and it will rapid to that position and then cause this canned cycle to be executed again, until it's canceled. The depth of each peck in this cycle will be the amount defined with Q or using IJ & K or K & Q. The tool will pull back after each peck and then back in for the next peck until Z depth is reached. This cycle is a high speed peck cycle where the retract distance it pulls back after each peck is set by Setting 22. Use G98 and G99 for the Z position clearance location for positioning between holes.

SETTING 22 - As the tool pecks deeper into the hole, after each peck, it will pull back a constant specified distance above the bottom of the hole that was created by the previous peck, to break the chip. That specified distance is defined in Setting 22.

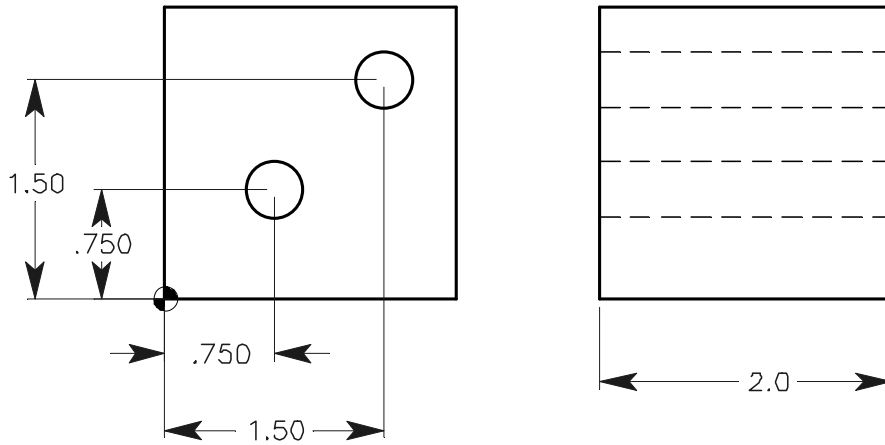
SETTING 52 - Changes the way G73 works when it returns to the R plane when using K and Q. When the clear move to R occurs, the Z will be moved above R by this setting distance. Most programmers set the R-plane well above the cut to insure that the chip clear motion actually allows the chips to get out of the hole but this causes a wasted motion when first drilling through this "empty" space. Or you may need to define a clearance move above the part, in which the R-plane may be down inside a part or pocket. If Setting 52 is set to the distance required to clear chips, the R-plane can be put much closer to the part being drilled. The Z axis will be moved above the R-plane by this amount in Setting 52.



G73 HIGH SPEED PECK DRILL CANNED CYCLE USING Q



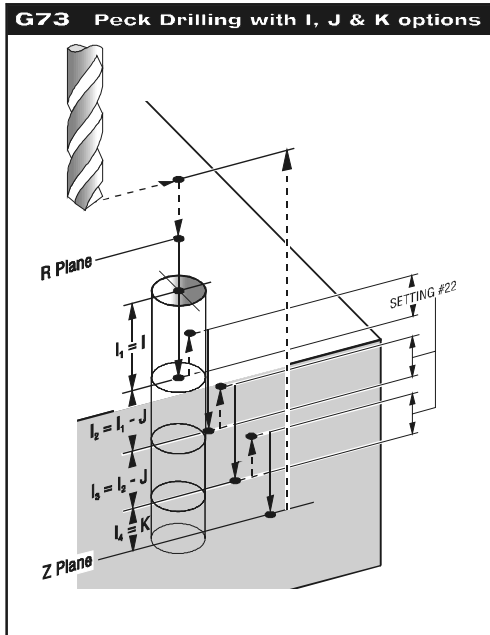
If Q is used, it specifies the amount each peck will be in a G73 canned cycle.



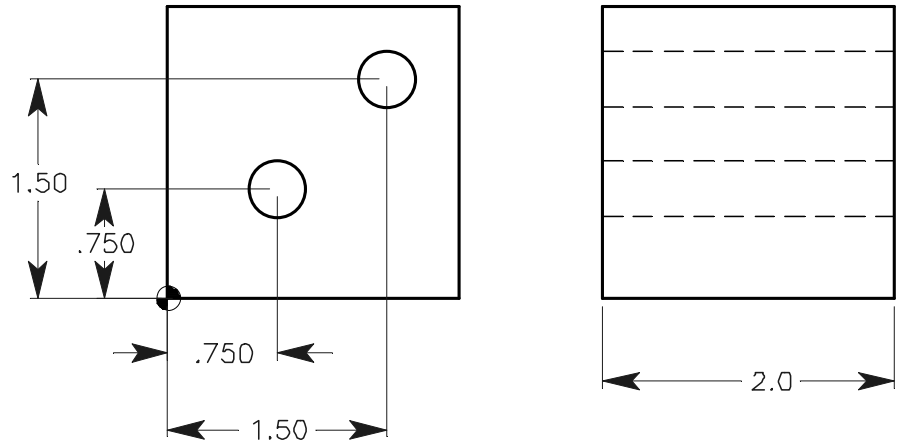
O00089 (G73 High Speed Peck Drill Using Q)

```

N1 T3 M06 (1/2 DIA. DRILL)
N2 G90 G54 G00 X0.75 Y0.75
N3 S1451 M03
N4 G43 H03 Z1. M08
N5 G73 G99 Z-2.15 Q0.1 R0.1 F10.
N6 X1.5 Y1.5
N7 G80 G00 Z1. M09
N8 G53 G49 Z0. M05
N9 M30
    
```

**G73 HIGH SPEED PECK DRILL CANNED CYCLE USING I,J,K**

If I, J, and K are specified, a different operating mode is selected. The first peck will be in by I, each succeeding peck will be reduced by I, with a minimum peck being defined with K.



O00090 (G73 High Speed Peck Drill Using I, J & K)

N1 T3 M06 (1/2 DIA. DRILL)

N2 G90 G54 G00 X0.75 Y0.75

N3 S1451 M03

N4 G43 H03 Z1. M08

N5 G73 G99 Z-2.15 I0.5 J0.1 K0.2 R0.1 F10.

N6 X1.5 Y1.5

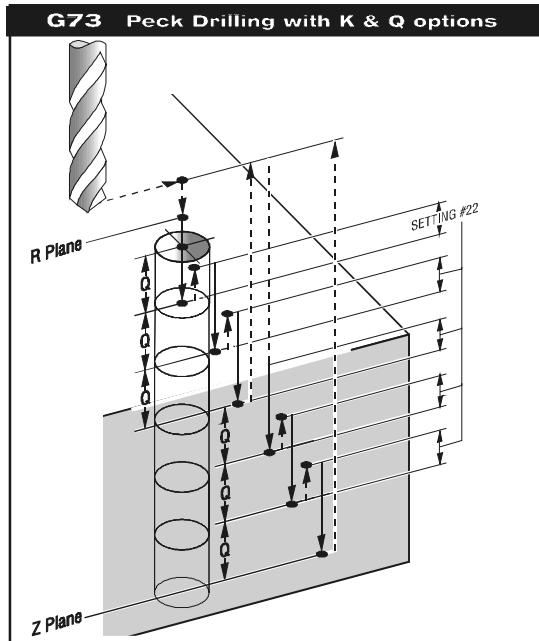
N7 G80 G00 Z1. M09

N8 G53 G49 Z0. M05

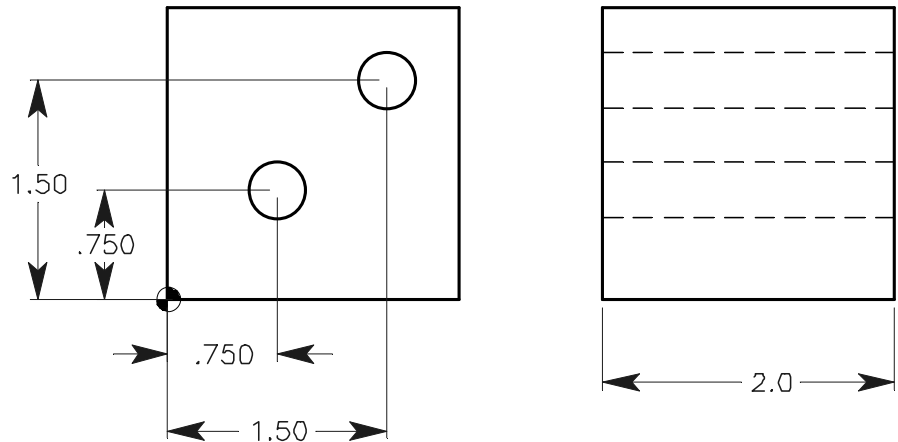
N9 M30



G73 HIGH SPEED PECK DRILL CANNED CYCLE USING K & Q



If K and Q commands are specified together with a G73, a different operating mode is selected in this cycle. After a number of pecks of Q distance down into the part totals up to the K amount, and then multiples of K thereafter, the tool will then return to the R-plane. This allows much faster drilling than G83, but still returns to the R plane occasionally to clear chips.



O00091 (G73 High Speed Peck Drill Using K & Q)

N1 T3 M06 (1/2 DIA. DRILL)

N2 G90 G54 G00 X0.75 Y0.75

N3 S1451 M03

N4 G43 H03 Z1. M08

N5 G73 G99 Z-2.15 K1. Q0.2 R0.1 F10.

N6 X1.5 Y1.5

N7 G80 G00 Z1. M09

N8 G53 G49 Z0. M05

N9 M30

**G74 REVERSE (LEFT HAND) TAPPING CANNED CYCLE**

- X* Rapid X-axis location
- Y* Rapid Y-axis location
- Z Z-depth (tapping Z-depth starting from R plane)
- J* Tapping Retract Speed (Rev. 10.13 and above)
- R R-plane (rapid point to start feeding)
- F Feed rate in inches (mm) per minute
- * Indicates optional

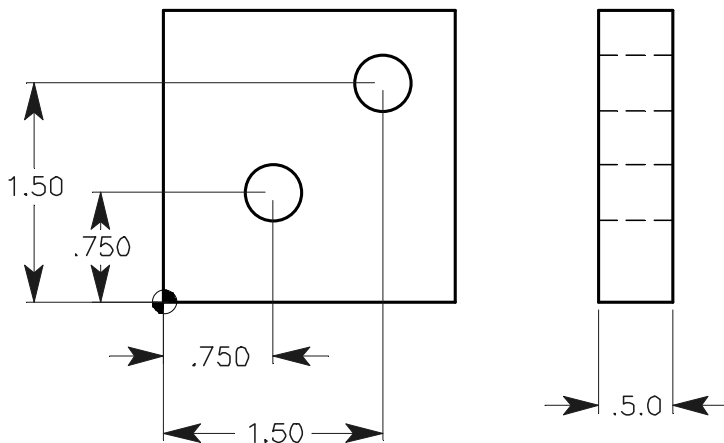
You don't need to start the spindle with a M04 for a tap that's using G74 because this cycle will turn on spindle for you automatically, and it will do it quicker.

This G code is modal. Use G98 and G99 for the Z position clearance location.

On older machines without vector motors, if your using a spindle speed that's in low gear you, may want to command M42 to force it into high gear, because most tapping (a smaller size tap) operations don't need the torque of low gear. And in high gear the tapping operation performs quicker.

Newer machines have Setting 130, Tap Retract Speed, can be set with, 1 thru 9, to quick-reverse-out of thread up to 9 times faster then going in, If J is not used.

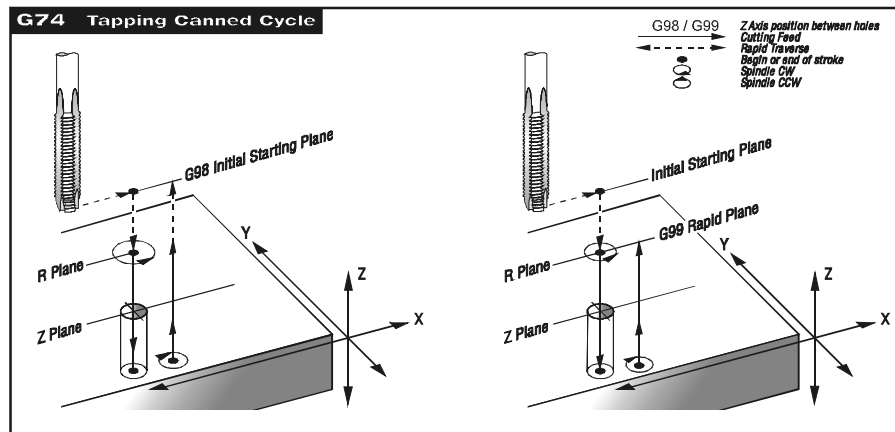
With Rigid Tapping, the ratio between feedrate and the spindle speed must be calculated for the thread pitch being cut. The calculation is $1/\text{Threads Per Inch} \times \text{rpm} = \text{tapping feedrate}$. Use the Haas calculator for the speed and feed numbers.

**O00092 (G74 L.H. Tapping Cycle)**

```

N1 T4 M06 (1/2-20 L.H. TAP)
N2 G90 G54 G00 X0.75 Y0.75
N3 S450 (You don't need M04, the
G84 turns on the spindle for you.)
N4 G43 H04 Z1. M08
N5 G74 G99 Z-0.65 R0.1 J5 F22.5
N6 X1.5 Y1.5
N7 G80 G00 Z1. M09
N8 G53 G49 Z0. M05
N9 M30

```





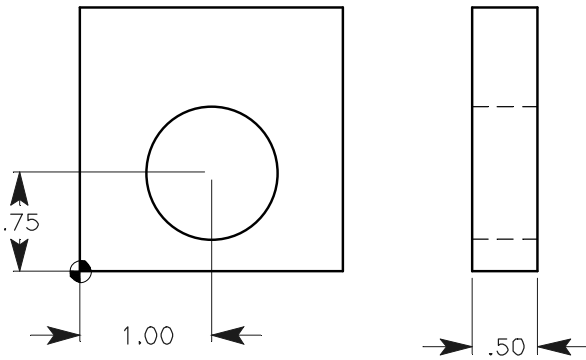
G76 BORE IN~STOP~SHIFT~RAPID OUT CANNED CYCLE

- X* Rapid X-axis location
- Y* Rapid Y-axis location
- Z Z-depth (feed to Z-depth starting from R plane)
- P Dwell time at Z-depth
- Q* Shift value, always incremental (if I or J are not used)
- I* Shift value X-axis (if Q is not used)
- J* Shift value Y-axis (if Q is not used)
- R R-plane (rapid point to start feeding)
- F Feed rate in inches (mm) per minute

* Indicates optional

The Q value distance will shift in the direction set by setting 27. If Q is not specified, the optional I and J values are used to define the shift distance and direction.

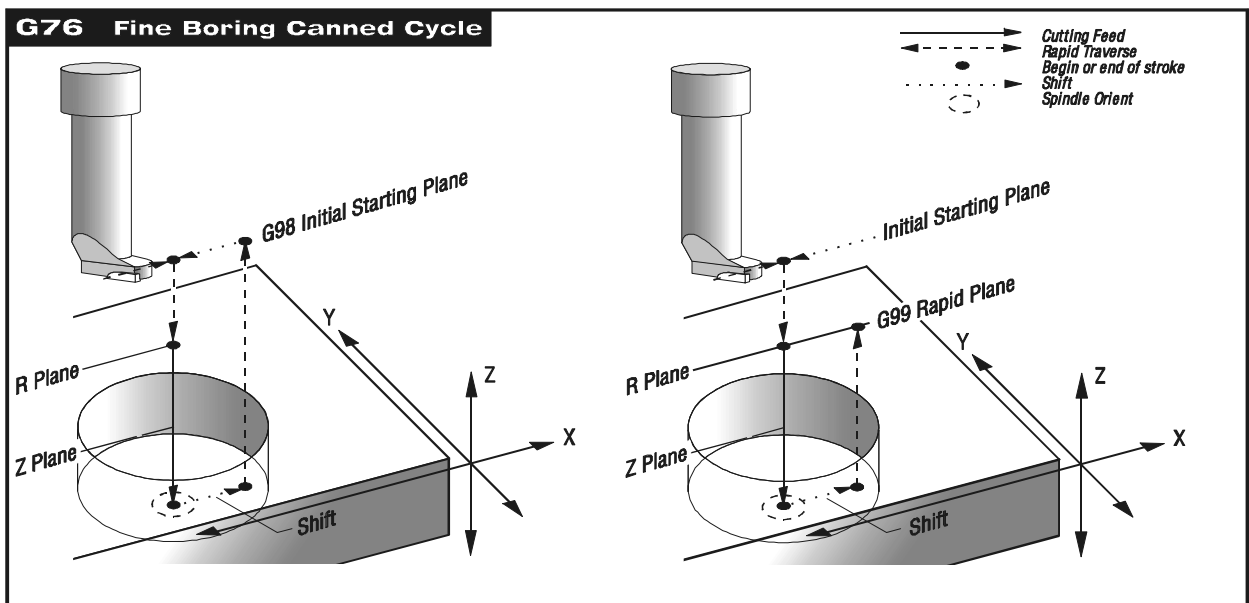
This G code is modal so that it is activated every X and/or Y axis move, and it will rapid to that position and then cause this canned cycle to be executed again, until it's canceled. Use G98 and G99 for the Z position clearance location for positioning between holes.



O00093 (G76 Bore-Stop-Shift-Rapid)

```

N1 T6 M06 (BORING BAR)
N2 G90 G54 G00 X1.0 Y0.75
N3 S1451 M03
N4 G43 H06 Z1. M08
N5 G76 G99 Z-0.55 P0.5 I-0.01 R0.1 F4.5
N6 G80 G00 Z1. M09
N7 G53 G49 Z0. M05
N8 M30
    
```



**G77 BACK BORE CANNED CYCLE**

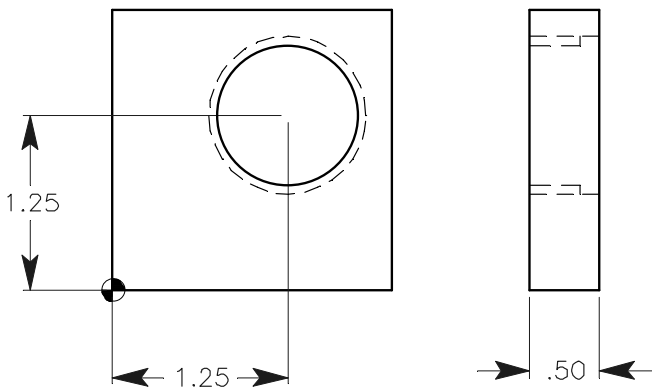
STOP~SHIFT~RAPID IN~SHIFT~SPINDLE ON
BORE UP~STOP~SHIFT~RAPID OUT CANNED CYCLE

- X* Rapid X-axis location
- Y* Rapid Y-axis location
- Z Z-depth (feed to Z-depth starting from R plane)
- Q* Shift value, always incremental (if I or J are not used)
- I* Shift value X-axis (if Q is not used)
- J* Shift value Y-axis (if Q is not used)
- R R-plane (rapid point to start feeding)
- F Feed rate in inches (mm) per minute

* Indicates optional

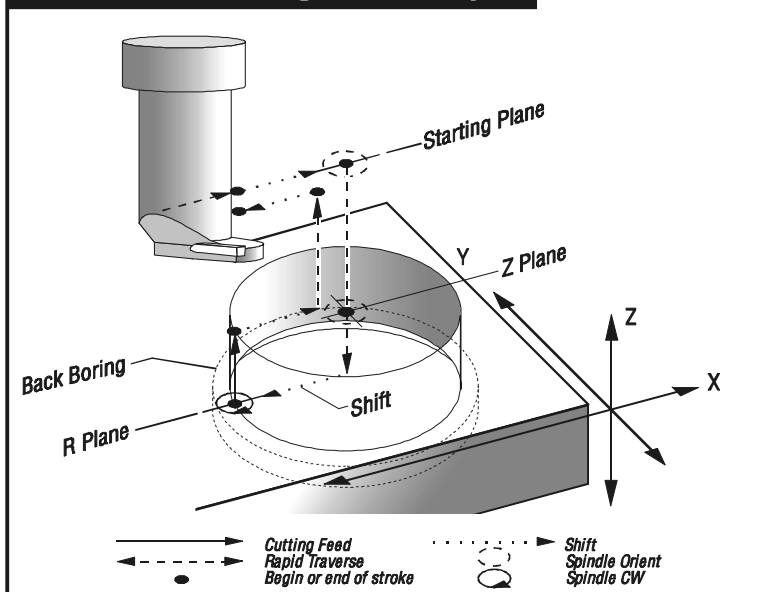
This G code is modal so that it is activated every X and/or Y axis move, and it will rapid to that position and then cause this canned cycle to be executed again, until it's canceled. Use G98 and G99 for the Z position clearance location for positioning between holes.

The Q value distance will shift in the direction set by setting 27. If Q is not specified, the optional I and J values are used to define the shift distance and direction.



```

O00094 (G76 Back Bore)
N1 T7 M06 (BACK BORING BAR)
N2 G90 G54 G00 X1.25 Y1.25
N3 S1451 M03
N4 G43 H07 Z1. M08
N5 G77 G99 Z-0.35 R-0.6 Q0.6 F4.5
N6 G80 G00 Z1. M09
N7 G53 G49 Z0. M05
N8 M30
  
```

G77 Back Boring Canned Cycle

This cycle will shift the X and/or Y axis prior to and after cutting in order to clear the tool while entering and exiting the part. The Q value shift direction is set with setting 27. If Q is not specified, the optional I and J values can be used to determine shift direction and distance. The tool will rapid down to the R clearance depth and feed up to the Z command depth for back counterbore.



BOLT HOLE PATTERNS

There are three G codes that provide commands used to do bolt holes patterns. They are G70, G71, and G72. These G codes are defined with one of the canned cycles G73, G74, G76, G77, or G81-G89. You define the angle of the bolt hole pattern, 0 to 360.0 degrees horizontal from three O'clock CCW. A minus sign will reverse angles CW.

G70 BOLT HOLE CIRCLE COMMAND

- I = Radius of the bolt hole circle
- J = Starting angle of first hole from three o'clock, 0 to 360.0 deg.
- L = Number of evenly spaced holes around bolt hole circle

The tool must be positioned at the center of the circle either in a previous block or in the G70 block. G70 belongs to Group zero and thus is non-modal. For a G70 to work correctly, a canned cycle must be active to perform the desired drill, tap or bore cycle.

G71 BOLT HOLE ARC COMMAND

- I = Radius of the bolt hole arc
- J = Starting angle of first hole from three o'clock, 0 to 360.0 deg.
- K = Angular spacing between holes (+ or -)
- L = Number of evenly spaced holes around bolt hole arc

The tool must be positioned at the center of the arc either in a previous block or in the G71 block. G71 belongs to Group zero and thus is non-modal. For a G71 to work correctly, a canned cycle must be active to perform the desired drill, tap or bore cycle. Using a K minus value (K-30.0) defines positioning around the bolt hole arc CW starting from the J angle.

G72 BOLT HOLES ALONG AN ANGLE

- I = Distance between bolt holes along an angle
- J = Angle of holes from three o'clock, 0 to 360.0 deg.
- L = Number of evenly spaced holes along an angle

This G72 code drills L holes in a straight line at the specified angle. G72 belongs to Group zero and thus is non-modal. For a G72 to work correctly, a canned cycle must be active to perform the desired drill, tap or bore cycle.



BOLT HOLE CIRCLE

You must position to the center of the bolt hole circle in the XY axis coordinates, either in the G70 command block, or define the XY coordinates in a previous command block.

If an L0 is on the canned cycle line, the cycle will not execute that command until the control reads the next line for a G70 command, so as not to drill a hole in the center of a bolt hole circle. Or you can combine the drill cycle with the G70 on the same line, and it will also, not drill a hole in the center.

A minus sign will reverse angle CW.

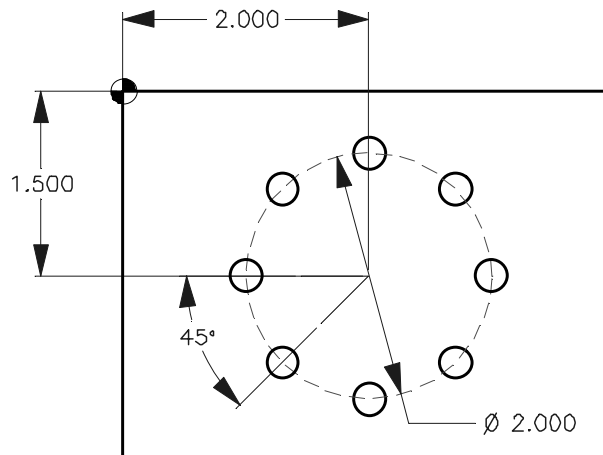
Be sure to use a decimal point with angle command codes.

G70 BOLT HOLE CIRCLE COMMAND

I = Radius of the bolt hole circle

J = First hole starting angle from three o' clock (0 to 360.0 deg. CCW)

L = Number of evenly spaced holes around bolt hole circle



O00096 (Bolt Hole Circle)

N1 T1 M06 (1/2 DIA. DRILL)

N2 G90 G54 G00 X2. Y-1.5 (Center position of bolt hole circle)

N3 S1451 M03

N4 G43 H01 Z1. M08

N5 G81 G99 Z-0.45 R0.1 F8. L0

N6 G70 I1. J0. L8

N7 G80 G00 Z1. M09

N8 G53 G49 Z0. M05

N9 M30

(Having L0 on line N5 will cause machine to not do this command until the control reads the next line, so as not to drill a hole in the center of bolt circle. Or you can combine N5 and N6 together, minus the L0, to also not drill a hole in the center.)



BOLT HOLE ARC

You must position to the center of the bolt hole arc in the XY axis coordinates, either in the G71 command block, or define the XY coordinates in a previous command block.

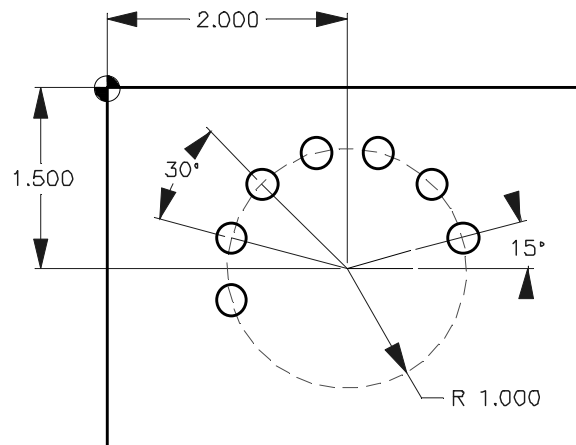
If an L0 is on the canned cycle line, the cycle will not execute that command until the control reads the next line for a G71 command, so as not to drill a hole in the center of a bolt hole arc. Or you can combine the drill cycle with the G71 on the same line, and it will also not drill a hole in the center.

A minus sign will reverse angles CW.

Be sure to use a decimal point with angle command codes.

G71 BOLT HOLE ARC COMMAND

- I = Radius of the bolt hole arc
- J = First hole starting angle from three o' clock (0 to 360.0 deg. CCW)
- K = Angular spacing between holes (+ or -)
- L = Number of evenly spaced holes around bolt hole arc



O00097 (Bolt Hole Arc)

N1 T2 M06 (1/2 DIA. DRILL)

N2 G90 G54 G00 X2. Y-1.5 (Center position of bolt hole arc)

N3 S1451 M03

N4 G43 H02 Z1. M08

N5 G81 G99 Z-0.45 R0.1 F8. L0

N6 G71 I1.0 J15. K30. L7

N7 G80 G00 Z1. M09

N8 G53 G49 Z0. M05

N9 M30

(Having L0 on line N5 will cause machine to not do this command until the control reads the next line, so as not to drill a hole in the center of bolt circle. Or you can combine N5 and N6 together, minus the L0, to also not drill a hole in the center.)



BOLT HOLES ALONG AN ANGLE

If an L0 is on the canned cycle line, the cycle will not execute that command until the control reads the next line for the G72 command. Or you can combine the G72 with the drill cycle on the same line.

A minus sign will reverse angle CW.

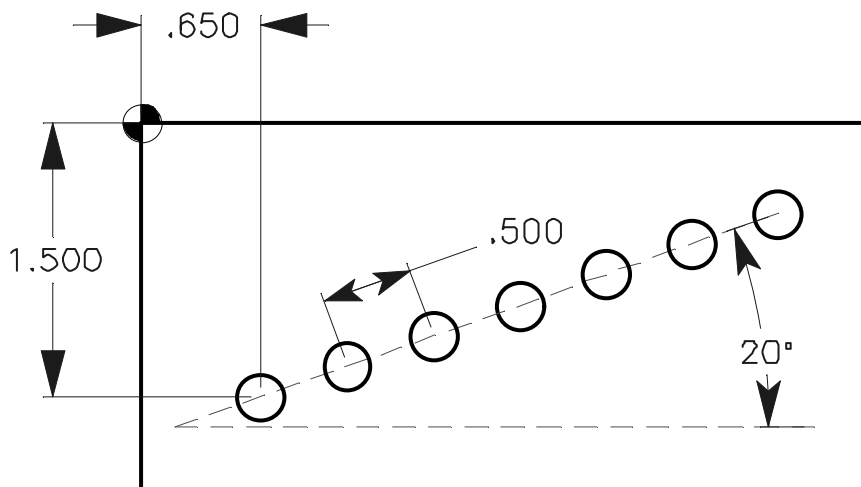
Be sure to use a decimal point with angle command codes.

G72 BOLT HOLES ALONG AN ANGLE

I = Distance between bolt holes along an angle

J = Angle of holes from three o' clock (0 to 360.0 deg. CCW)

L = Number of evenly spaced holes along an angle



O00098 (Bolt Holes Along An Angle)

N1 T3 M06 (1/2 DIA. DRILL)

N2 G90 G54 G00 X0.65 Y-1.5 (Start position of bolt holes along an angle)

N3 S1451 M03

N4 G43 H03 Z1. M08

N5 G81 G99 Z-0.45 R0.1 G72 I0.5 J20. L7 F8.

N6 G80 G00 Z1. M09

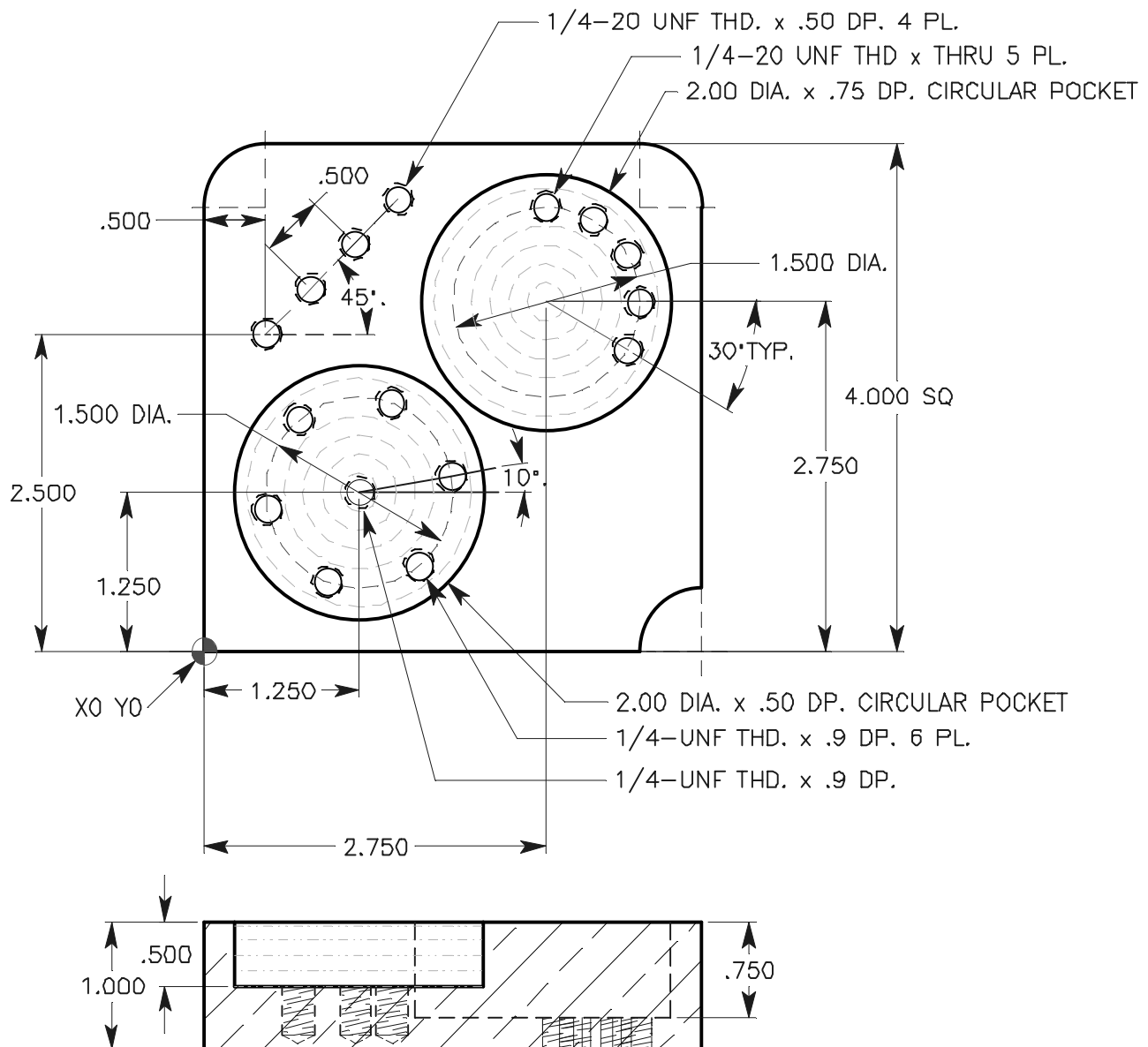
N7 G53 G49 Z0. M05

N8 M30





CANNED CYCLE EXERCISE #3



Program a #7 (.201 Dia.) Drill at 3200 RPM with a 11.5 feedrate to a Z-.95 drill tip depth for the Bolt Hole Circle, drill thru part to Z-1.1 for the Bolt Hole Arc, and change it to a Z-.6 drill tip depth for the Bolt Holes Along an Angle. Define the rapid plane .100 up from the bottom of the circular pockets of the Bolt Hole Circle and Bolt Hole Arc and change the rapid plane for the Bolt Hole Arc and Bolt Holes along an Angle to be .1 above where it begins to start drilling from.

NOTE: If an L0 is on the canned cycle line, it will not execute that command until the control reads the next location or the bolt hole command in the following line so as not to drill a hole in the center of bolt hole circle or bolt hole arc.



```

O00070 (CANNED CYCLE EXERCISE #3)
T6 M06 (#7 .201 DIA. CARBIDE STUB DRILL)
G___ G___ G___ X_____ Y_____ (X Y center position of Bolt Hole Circle)
S3200 M_____
G___ H___ Z_____ M___ (Position to Z.1 for the initial point, coolant on)
G___ G___ Z_____ R_____ F11.5 (G81 with G98 initial point return)
G___ I_____ J_____ L___ (Bolt Hole Circle command for 6 holes)
X_____ Y_____ L___ (Go to Bolt Hole Arc center, but no hole here using L0)
G___ Z___ R___ I_____ J_____ K_____ L___ (Bolt hole ARC 5 holes)
G___ X___ Y___ Z___ R___ I___ J___ L___ (Define a Bolt Holes
Along an Angle, with XY start location, new Z depth and change the R plane)
G___ G___ Z_____ M___ (Cancel canned cycle and turn off coolant)
G___ G___ Y_____ Z_____ M___ (Send Y and Z home and turn off spindle)
M30 (End of Program)

```

NOTE: The circular pockets are .50 deep, and .75 deep, so the rapid R-plane should be defined .100 above the bottom surface of pockets (R-.4 and R-.65) , so as not to drill air from top of part down to bottom surface of the pocket. But be careful, because the R-plane needs to be changed back to R.1 to the rapid plane above part surface when your positioning over to drill and tap Bolt Holes Along an Angle.

Be sure to use a decimal point with angle command codes.





SUBROUTINE

A subprogram is a separate program called up by another program. The use of subprograms can significantly reduce the amount of programming on some parts. Subroutines allow the CNC programmer to define a series of commands which might be repeated several times in a program and, instead of repeating them many times, they can be "called up" when needed. A subroutine call is done with M97 or M98 and a Pnnnn. The P code command identifies the O program number being used when executed with M98 or an N sequence number to identify the block where a local subroutine starts when executed with M97.

Local subroutines are called with an M97. This can be easier to use than the M98 because the subroutine is contained within the main program without the need to define a separate Onnnn program. With local subroutines, you define an M30 for the end of your main program portion. And after the M30 list all of your subroutines starting with a N sequence number to define the beginning of a local subroutine. And then end every subroutine with an M99 to send the control back to the very next line in the main program after the subroutine call.

A subroutine call from the main program calls up program blocks in a subroutine to be executed just as if they were included in the main program. Then to return back to the main program, you end a subroutine with an M99 which sends it back to the next line after the subroutine call in the main program.

Another important feature of a "subroutine call" is that the M97 and M98 block may also include an L (loop) or repeat count. If there is an Ln with the subroutine call it is repeated that number of times before the main program continues with the next block.

The most common use of subroutines is in the definition of a series of holes which may need to be center drilled, peck drilled, tapped, and/or chamfered. If a subroutine is defined that consists only of the X-Y position of the holes, the main program can define the canned cycles, and the hole locations can be called up in the subroutine to do each of the tool operations. Thus, the X-Y positions can be entered only once and used several times for each tool.



O00104 (PROGRAM CALLING SUB-PROGRAM WITH AN M98)

T1 M06 (90 DEG. 5/8 DIA. SPOT DRILL)

G90 G54 G00 X1.5 Y-0.5 (1)

S1406 M03

G43 H01 Z1. M08

G81 G99 Z-0.26 R0.1 F7. (G81 Drilling Canned Cycle)

M98 P105 (Call Sub-Program 000105)

T2 M06 (27/64 DIA. DRILL)

G90 G54 G00 X1.5 Y-0.5 (1)

S2082 M03

G43 H02 Z1. M08

G83 G99 Z-.75 Q0.2 R0.1 F12.5 (G83 Peck Drilling Canned Cycle)

M98 P105 (Call Sub-Program 000105)

T3 M06 (1/2-20 TAP)

G90 G54 G00 X1.5 Y-0.5 (1)

S750 (G84 will turn on the spindle, so no M03 is needed)

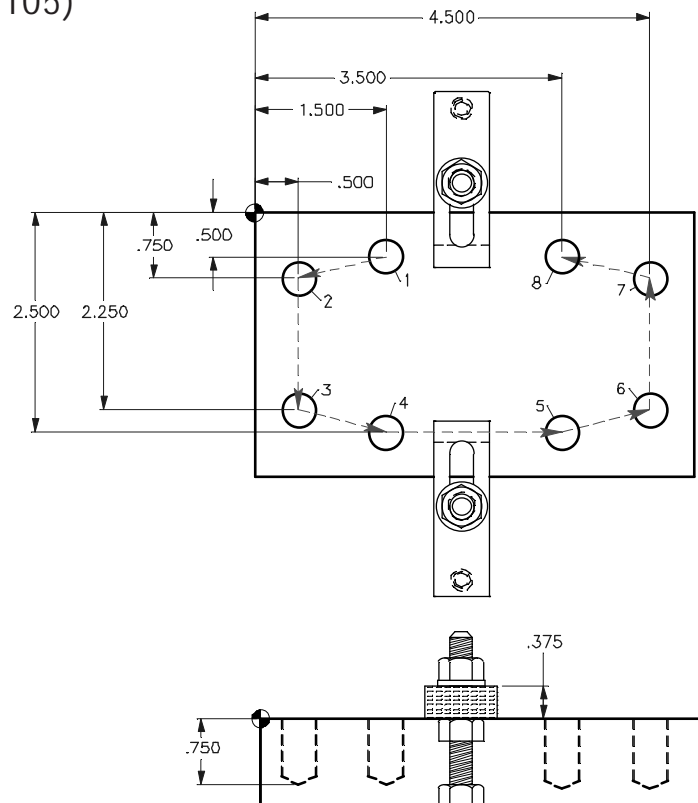
G43 H03 Z1. M08

G84 G99 Z-.6 R0.1 F37.5 (G84 Tapping Canned Cycle)

M98 P105 (Call Sub-Program 000105)

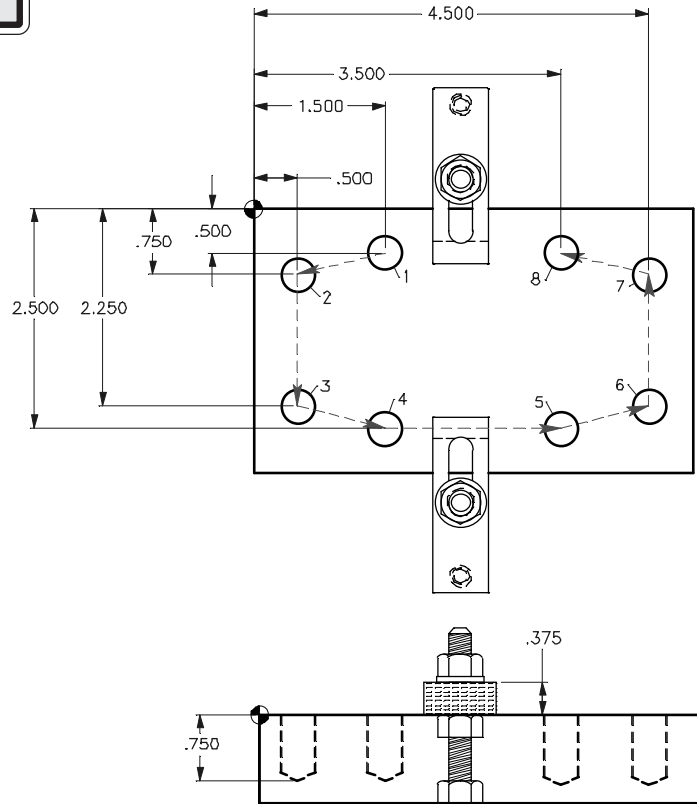
G53 G49 Y0.

M30 (End Program)





(Sub-program)
 (Listing all the hole locations)
 O00105
 X.5 Y-.75 (2)
 Y-2.25 (3)
 G98 X1.5 Y-2.5 (4)
 G99 X3.5 (5)
 X4.5 Y-2.25 (6)
 Y-.75 (7)
 X3.5 Y-.5 (8)
 G80 G00 Z1.0 M09
 G53 G49 Z0. M05
 M99 (An M99 ends a sub-program and returns back to the next line in the main program after the M98 sub-program call.)





O00106 (PROGRAM CALLING LOCAL SUB-ROUTINE WITH M97)

T1 M06 (90 DEG. 5/8 DIA. SPOT DRILL)

G90 G54 G00 X1.5 Y-0.5

S1406 M03

G43 H01 Z1. M08

G81 G99 Z-0.26 R0.1 F7. (G81 Drilling Canned Cycle)

M97 P10 (Call Local Sub-Routine N10)

T2 M06 (27/64 DIA. DRILL)

G90 G54 G00 X1.5 Y-0.5

S2082 M03

G43 H02 Z1. M08

G83 G99 Z-.625 Q0.2 R0.1 F12.5 (G83 Peck Drilling Canned Cycle)

M97 P10 (Call Local Sub-Routine N10)

T3 M06 (1/2-20 TAP)

G90 G54 G00 X1.5 Y-0.5

S750 (G84 will turn on the spindle, so no M03 is needed)

G43 H03 Z1. M08

G84 G99 Z-.5 R0.1 F37.5 (G84 Tapping Canned Cycle)

M97 P10 (Call Local Sub-Routine N10)

G53 G49 Y0.

M30 (End of main program)

(Local Sub-Routine)

N10 (Local subroutine example listing all the hole locations)

X.5 Y-.75 (2)

Y-2.25 (3)

G98 X1.5 Y-2.5 (4)

X3.5 R-.4 (5)

X4.5 Y-2.25 (6)

Y-.75 (7)

X3.5 Y-.5 (8)

G80 G00 Z1.0 M09

G53 G49 Z0. M05

M99 (An M99 ends a sub-routine and returns back to the next line in the main program after the M97 sub-routine call.)



GENERAL PURPOSE POCKET MILLING

G150 General Purpose Pocket Milling

- X X position of starting hole (not needed if you're already at that location)
- Y Y position of starting hole (not needed if you're already at that location)
- Z Final depth of pocket
- Q Incremental Z-axis depth, starting from R plane
- R R plane clearance position
- I X-axis shift over cut increment (use I or J, not both)
- J Y-axis shift over cut increment (use I or J, not both)
- K Finishing cut allowance
- D Cutter compensation geometry offset register number
- G41 Left or G42 Right (of program path) Cutter compensation
- F Feed rate
- P Subprogram call of program number that defines pocket geometry

This G code provides for general purpose pocket milling. The shape of the pocket to be machined must be defined by a series of motions within a subprogram. A series of motions in either the X or Y axis will be used to cut out the specified shape, followed by a finishing pass to clean up the outer edge. Only one of either I or J must be specified. If I is used, the pocket is cut by shifting over in the X axis, and the strokes will be along the Y axis. If J is used, the pocket is cut by shifting over in the Y axis, and the strokes will be along the X axis. I and J must be positive numbers. The K command is a finish pass and must be a positive number. There is no finishing pass for the Z depth using the K command. A Z depth finish pass can be calculated using Z, Q, and R commands.

Multiple Q cuts can be defined to get to the final Z depth. The feed down to the final Z depth starts from the R plane, feeding down by the Q amount for each pass until the Z depth is reached. Q must be positive.

If an L count is specified on the G150 command line, with a G91 and an X and/or Y value, the entire block is repeated again at an incremental X or Y distance, L number of times.

The subroutine must define a closed loop area with a series of G01, G02, or G03 motions in the X and Y axes, and must end with an M99. G codes G90 and G91 can also be used in the subprogram to select absolute or incremental motion. Any codes other than G, I, J, R, X, or Y are ignored in the subprogram, which must consist of no more than 40 moves. (In control software version 11.11, G150 Pocket Milling was increased to 40 geometry moves. In all previous versions, G150 could be defined with no more than 20 geometry moves.)



With a G150 pocket milling command, you may need a clearance hole drilled to Z depth of pocket, prior to the endmill entering for a G150 pocket cycle, since it plunges straight down in the Z axis. You may choose to specify an XY starting hole location to be drilled, prior to, or in the G150 command.

The first motion of a subprogram should define a move from the starting hole location onto a start point of the pocket geometry. The final move in the subprogram should close the loop at the same point where you began the pocket geometry. In the example on the following page, the start point of the G150 is X1.5 Y3.25, and the first move of the subprogram is X0.5. Therefore, the last move of the subprogram must return to X0.5 Y3.25.

If K is specified, the roughing cuts will cut inside the programmed pocket size by the amount of K. The finishing pass will follow along the pocket geometry edge and is done at the full pocket depth.

(Tool 1 drills a clearance hole in the pocket for endmill to plunge into)
O00108 (PROGRAM NAME HERE, SHOWS UP IN LIST PROGRAM)
N100 (Starting hole for endmill to enter pocket down in Z axis for G150 pocket cycle)
N101 T1 M06 (1/2 DIA. DRILL)
N102 G90 G54 G00 X1.5 Y3.25 (Starting location for drilling)
N103 S1451 M03
N104 G43 H01 Z1. M08
N105 G83 G99 Z-1.25 Q0.25 R0.1 F10.
N106 G80 G00 Z1. M09
N107 G53 G49 Z0. M05

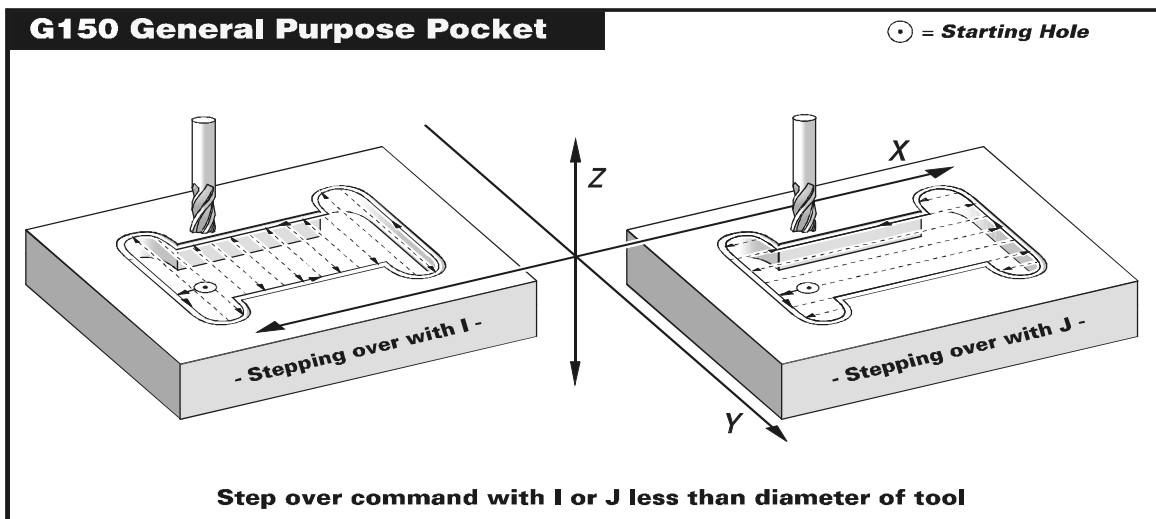
N200 (G150 Shift X axis with I, milling down Q steps to Z depth)
N201 T2 M06 (1/2 DIA. 4-FLT. ENDMILL)
N202 G90 G54 G00 X1.5 Y3.25 (Start location for pocket milling routine)
N203 S2070 M03
N204 G43 H02 Z1. M08
N205 G01 Z.01 F30. (Feed down to start point in Z for incremental Q depth)
N206 G150 P109 G41 D02 I.4 (or J.4) K.02 Z-1.25 Q.63 R.01 F16.5
N207 G40 G01 X1.5 Y3.25 (Be sure to cancel cutter compensation)
N208 G00 Z0.1 M09
N209 G53 G49 Y0. Z0. M05
N210 M30

The G150 pocket milling sub-program is called up on line N206 (this page) with G150 P109. This tells the control to look for program O00109 (next page) for the finish geometry of the pocket being machined.



```

(SUBPROGRAM)
O00109 (G150 POCKET GEOMETRY)
N1 G01 X0.5
N2 Y2.
N3 G03 X2.25 R0.875
N4 G01 X5.25
N5 G03 X7. R0.875
N6 G01 Y5.
N7 G03 X5.25 R0.875
N8 G01 X2.25
N9 G03 X0.5 R0.875
N10 G01 Y3.25
N11 M99 (Returns to main program)
  
```



G150 NOTES

1. Position the endmill to the starting point inside the pocket. Do not have cutter overlap on any line of the pocket geometry with the entry of the endmill.
2. The G150 command line calls up a subprogram with a P command (P12345) that specifies the separate subprogram (O12345) which defines the geometry of a pocket. This pocket geometry must be defined in 40 strokes (moves) or less. In control software version 11.11, G150 Pocket Milling was increased to 40 geometry moves. In all previous versions, a G150 could not be defined with more than 20 geometry moves in the subprogram called up by the G150.
3. The G150 pocket geometry command cannot be listed after the M30 command, like the way M97 (Local subprogram call) is used. Instead, G150 uses a P command to call up a separate program which contains the pocket geometry.



4. You can also define an island within the G150 pocket command. Remember that you only have a total of 40 moves to define the pocket geometry (only 20 moves for machines with ver.11.11 and earlier).

5. When positioning on and off of an island inside a G150 subprogram that's using cutter compensation (G41/G42). The cutter will be either left or right of the line you're positioning onto. If you're positioning onto the island or pocket at the same location you position off the island or pocket, you will leave a large scallop the size of the offset radius. To eliminate this scallop, you need to overlap the entry and exit moves by a little more than the offset amount, in the offset register, so that the center of the cutter overlaps the path on the entry and exit moves, when defining an Island in a G150 pocket command.

6. Define the pocket geometry tool path in the direction you wish to cut. Define the path counterclockwise to climb cut using G41, or clockwise to conventional cut with a G42. The cutting path direction needs to be reversed when positioning onto an island inside the pocket when climb or conventional cutting with endmill.

(Tool 1 drills a clearance hole in the pocket for endmill to plunge into.)

O00110 (PROGRAM NAME)

N100 (Starting hole for endmill to enter pocket down in Z axis for G150 pocket cycle)

N101 T1 M06 (1/2 DIA. DRILL)

N102 G90 G54 G00 X1.5 Y3.25 (Start location for drilling)

N103 S1451 M03

N104 G43 H01 Z1. M08

N105 G83 G99 Z-1.25 Q0.25 R0.1 F10.

N106 G80 G00 Z1. M09

N107 G53 G49 Z0. M05

N200 (G150 Shift Y axis with J, milling down Q steps to Z depth.)

N201 T2 M06 (1/2 DIA. 4-FLT. ENDMILL)

N202 G90 G54 G00 X1.5 Y3.25 (Start location for pocket milling routine)

N203 S2070 M03

N204 G43 H02 Z0.1 M08

N205 G01 Z.01 F30. (Feed down to start point in Z axis for incremental Q depth)

N206 G150 P111 G41 D02 J.4 (or I.4) K.02 Z-1.25 Q.63 R.01 F16.5

N207 G40 G01 X1.5 Y3.25 (Be sure to cancel cutter compensation)

N208 G00 Z0.1 M09

N209 G53 G49 Y0. Z0. M05

N210 M30



The following is a sub-program for a G150 pocket geometry milling cycle. The main program, O00110, is calling P111 on line N206. The G150 P111 commands cause the control to look for program O00111 for the finish geometry of the pocket being machined.

(SUBPROGRAM)

O00111 (G150 Pocket geometry)

N1 G01 X0.5

N2 Y1.5

N3 G03 X2.25 R0.875

N4 G01 X4.

N5 Y3.

N6 X2.5

N7 G02 Y3.5 R0.25

N8 G01 X5.

N9 G02 Y3. R0.25

N10 X3.4

N11 Y1.5

N12 X5.25

N13 G03 X7. R0.875

N14 G01 Y5.

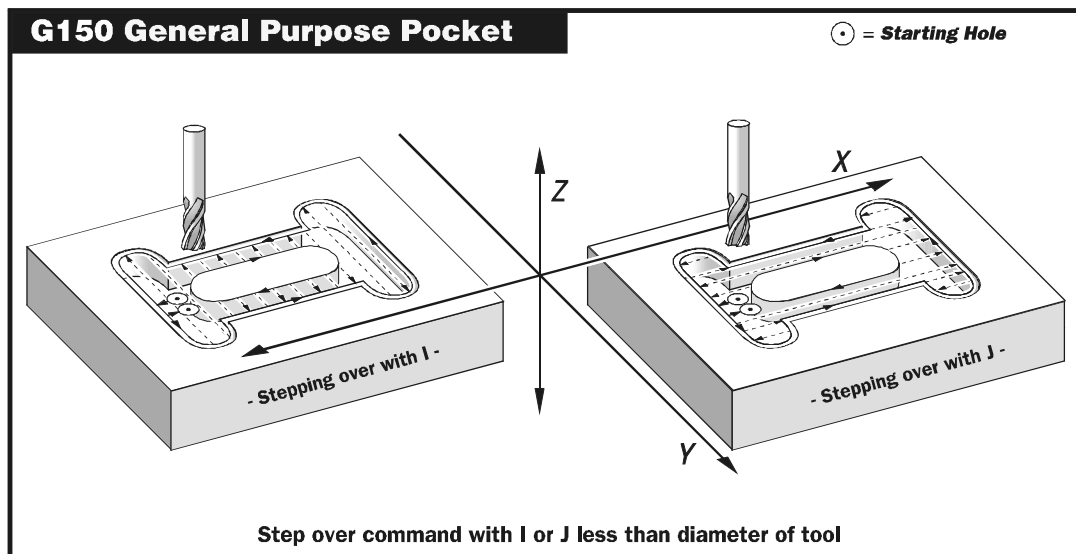
N15 G03 X5.25 R0.875

N16 G01 X2.25

N17 G03 X0.5 R0.875

N18 G01 Y3.25

N19 M99 (Returns to main program)





7. When you're defining the pocket geometry in the subprogram, define it from a starting point inside the pocket. Then define the pocket geometry, beginning with a move onto the side of the pocket. Close the loop where you began, and end the subprogram there with an M99. Don't define a move back to the starting point where you began.
8. The Q value must be defined in a G150 command, even if you're only doing one pass to the final Z depth. The Q amount is defined from the R plane. If you only want one pass, and your final depth is Z-0.25, and you're starting the pocket at R0.1 above part, then your Q value will be Q0.35. If, however, you give it a Q0.25, it will move from R0.1 down to Z-0.15 (the amount Q0.25) into the pocket. Then it will take a second pass to the final depth of Z-0.25.
9. If you execute a G150 pocket milling command and have an invalid move in your pocket geometry with an alarm 370, Pocket Definition Error, the control will stop on the G150 command line in the main program. To locate this invalid move quicker (within the pocket geometry). Run the pocket subprogram separately in Graphics to find the errors. To run the subprogram in Graphics, you'll need to enter a feedrate (G01) command on the first line of the subprogram; troubleshoot the pocket geometry in Graphics. After correcting the moves in the subprogram, take the feedrate command back out, and verify the fix in Graphics on the main program.





MISCELLANEOUS M CODES

Only one M code may be programmed per block of a program. All M codes are effective or cause an action to occur at the end of the block. However, when Parameter 278 bit "CNCR SPINDLE" is set to 1, spindle start occurs at the beginning of a block and will be concurrent with axis motion. The settings listed next to the M code, in some way effects that M-code. The M-codes with double asterisk (**) are machine options.

M00	Program Stop (Setting 42)
M01	Optional Program Stop (Setting 17)
M02	Program End
M03	Spindle Clockwise (S) (Setting 144)
M04	Spindle Counterclockwise (S) (Setting 144)
M05	Spindle Stop
M06	Tool Change (Setting 42, 87)
M08	Coolant On (Setting 32)
M09	Coolant Off
M10**	Engage 4th Axis Brake
M11 **	Release 4th Axis Brake
M12 **	Engage 5th Axis Brake
M13 **	Release 5th Axis Brake
M16	Tool Change (same as M06)
M19	Orient Spindle (P and R values optional)
M21-M28	Optional Pulsed User M Function with Fin
M30	Program End and Rewind (Setting 2, 39, 56, 83)
M31 **	Chip Auger Forward (Setting 114,115)
M32 **	Chip Auger Reverse (Setting 114,115)
M33 **	Chip Auger Stop
M34 **	Coolant Spigot Position Down, Increment
M35 **	Coolant Spigot Position Up, Decrement
M36 **	Pallet Part Ready
M39	Rotate Tool Turret (T) (Setting 86)
M41	Spindle Low Gear Override
M42	Spindle High Gear Override
M50 **	Execute Pallet Change (Setting 121 thru 129)
M51-M58	Optional User M Code Set
M59	Output Relay Set (N)
M61-M68	Optional User M Code Clear
M59	Output Relay Clear (N)
M75	Set G35 or G136 Reference Point
M76	Control Display Inactive
M77	Control Display Active
M78	Alarm if Skip Signal Found



M79	Alarm if Skip Signal not Found
M80 **	Automatic Door Open (Setting 131)
M81 **	Automatic Door Close (Setting 131)
M82	Tool Unclamp
M83 **	Auto Air Jet On
M84 **	Auto Air Jet Off
M86	Tool Clamp
M88 **	Coolant Through the Spindle ON (Setting 32)
M89 **	Coolant Through the Spindle OFF
M93	Axis POS Capture Start
M94	Axis POS Capture Stop
M95	Sleep Mode
M96	Jump if no Input (P,Q)
M97	Local Sub-Routine Call (P, L)
M98	Sub Program Call (P, L)
M99	Sub Program/Sub-Routine Return Or Loop (P) (Setting 118)
M109**	Interactive User Input (P)



M CODE DETAILED DESCRIPTION

M01 OPTIONAL PROGRAM STOP (SETTING 42)

The M00 code is used to stop a program. It also stops the spindle and turns off the coolant and stops interpretation lookahead processing. The program pointer will advance to the next block and stop. A cycle start will continue program operation from the next block. If the Through the Spindle Coolant option is ON, M00 will shut it off.

M01 OPTIONAL PROGRAM STOP (SETTING 17)

The M01 code is identical to M00 except that it only stops if OPTIONAL STOP is turned on from the front panel. A cycle start will continue program operation from the next block. If the Through the Spindle Coolant option is ON, M01 will shut it off.

M02 PROGRAM END

The M02 code will stop program operation the same as M00 but does not advance the program pointer to the next block.

M03 SPINDLE FORWARD (S) (SETTING 144)

The M03 code will start the spindle moving in a clockwise direction at whatever speed was previously set. The block will delay until the spindle reaches about 90% of commanded speed.

When Parameter 278 bit 10, "CNCR SPINDLE", is set to 1, the machine will not wait for the spindle to come up to speed immediately after an M03 command. Instead, it will check and/or wait for the spindle to come up to speed immediately before the next interpolated motion is initiated.

M04 SPINDLE REVERSE (S) (SETTING 144)

The M04 code will start the spindle moving in a counterclockwise direction at whatever speed was previously set. The block will delay until the spindle reaches about 90% of commanded speed.

When Parameter 278 bit 10, "CNCR SPINDLE", is set to 1, the machine will not wait for the spindle to come up to speed immediately after an M04 command. Instead, it will check and/or wait for the spindle to come up to speed immediately before the next interpolated motion is initiated.

Running an M04 command with Through the Spindle Coolant (TSC) on is not recommended.

M05 SPINDLE STOP

The M05 code is used to stop the spindle. The block is delayed until the spindle slows below 10 RPM.



M06 TOOL CHANGE (SETTING 42, 87)

The M06 code is used to initiate a tool change. The previously selected tool (Tn) is put into the spindle. If the spindle was running, it will be stopped. No previous axis commands are required before the tool change unless there is a problem with tool/part/fixture clearance. The Z-axis will automatically move up to the machine zero position and the selected tool will be put into the spindle. The Z-axis is left at machine zero. The spindle will not be started again after the tool change but the Snnnn speed and gear will be unchanged. The Tnn must be in the same block or in a previous block. The coolant pump will be turned off during a tool change.

When the Through the Spindle Coolant (TSC) is ON, M06 will orient the spindle and move the Z-axis to tool change position, turn off the TSC pump, purge the coolant from the drawbar, then perform a tool change. TSC will remain OFF until an M88 is called.

M08 COOLANT ON (SETTING 32)

The M08 code will turn on the coolant supply. Note that the M code is performed at the end of a block; so that if a motion is commanded in the same block, the coolant is turned on after the motion. The low coolant status is only checked at the start of a program so a low coolant condition will not stop a program which is already running.

M09 COOLANT OFF

The M09 code will turn off the coolant supply.

M10 4TH AXIS BRAKE ON (OPTION)

The M10 code is used to apply the brake to the 4th axis. The brake is normally engaged, so M10 is only required when M11 is used to release the brake. M11 activates a relay that releases the brake. M10 deactivates this relay, engaging the brake.

M11 4TH AXIS BRAKE RELEASE (OPTION)

The M11 code will "pre-release" the 4th axis brake. This is useful in preventing the delay that otherwise occurs when a 4th axis is used with a brake and a motion is commanded in that axis. It is not required but, without a prior M11, there will be a delay in motion in order to release the air.

M12 5TH AXIS BRAKE ON (OPTION)

The M12 code is used to apply the brake to the 5th axis. The brake is normally engaged, so M12 is only required when M13 is used to release the brake. M13 activates a relay that releases the brake. M12 deactivates this relay, engaging the brake.



M13 5TH AXIS BRAKE RELEASE (OPTION)

The M13 code will "pre-release" the 5th axis brake. This is useful in preventing the delay that otherwise occurs when a 5th axis is used with a brake and a motion is commanded in that axis. It is not required but, without a prior M13, there will be a delay in motion in order to release the air.

M19 ORIENT SPINDLE (P, R) (OPTION)

The M19 code is used to orient the spindle to a fixed position. The spindle is oriented electronically. A P value can be added as an option that will cause the spindle to be oriented to a particular angle (in degrees). For example, M19 P270 will orient the spindle to 270 degrees. An R value will recognize up to four places to the right of the decimal point. An M19 R123.4567 will position the spindle to the angle specified by the R value.

M21-M28 OPTIONAL USER M CODE

The M21 through M28 codes are optional for user interfaces. They will activate one of relays 1132 through 1139, wait for the M-fin signal, release the relay, and wait for the M-fin signal to cease. The RESET button will terminate any operation that is hung-up waiting for M-fin.

M30 PROGRAM END AND RESET (SETTING 2, 39, 56, 83)

The M30 code is used to stop a program. It also stops the spindle and turns off the coolant. The program pointer will be reset to the first block of the program and stop. The parts counters displayed on the Current Commands display are also incremented. M30 will also cancel tool length offsets. When the Through the Spindle Coolant (TSC) option is ON, M30 will shut it OFF, and then perform an M30 operation.

M31 CHIP AUGER FORWARD (SETTING 114, 115)

M31 starts the chip conveyor motor in the forward direction. The forward direction is defined as the direction that the conveyor must move to transport chips out of the work cell. The conveyor will not turn if the door is open. This may be overridden by setting bit 17 of parameter 209 (CNVY DR OVRD).

M32 CHIP AUGER REVERSE (SETTING 114, 115)

M32 starts the chip conveyor motor in the reverse direction. The reverse direction is defined as the direction opposite of forward. The conveyor will not turn if the door is open. This may be overridden by Setting bit 17 of parameter 209 (CNVY DR OVRD).

M33 CHIP AUGER STOP



M34 COOLANT SPIGOT POSITION DOWN, INCREMENT

M34 Increments the current spigot position down one place. Incrementing the spigot position causes the spigot to move down one place from the home position. The home position is designated as zero. If the current home position is designated as 5 and M34 is executed, then the current spigot position will move down to position 6. The spigot home position is with the spigot at the most positive Z axis location the spigot can attain. Incrementing the spigot then lowers the coolant stream direction.

M35 COOLANT SPIGOT POSITION UP, DECREMENT

M35 decrements the coolant spigot position up one place. Decrementing the spigot position causes the spigot to move up toward the spigot home position. The home position is designated as zero. If the current spigot position is 5 and M35 is executed, then the current spigot position will move up to 4. The spigot home position for a horizontal mill places the spigot at the most positive Z axis location. Decrementing the spigot will raise the coolant stream direction.

M36 WAIT PALLET READY (OPTION)

This is used on Horizontal mills only. Flashes PART READY button on the front switch box. It delays the pallet change execution until the PART READY button is depressed.

M39 ROTATE TOOL TURRET (SETTING 86)

The M39 code is used to rotate the tool turret without performing a tool change. The desired tool pocket number (Tn) must be programmed previous to the M39. This M code may be useful to move an empty pocket to face the spindle. This is not normally required but is useful for diagnostic purposes or to recover from a tool changer crash. Remember that the pocket facing the spindle must always be empty for a tool change.

M41 SPINDLE LOW GEAR OVERRIDE

The M41 code is used to override the spindle gear implied by the Snnn command. With M41, the spindle gear will always be low. If the speed commanded is above the low gear limit, the spindle speed will be the low gear limit. This M code does not turn the spindle on or off. If the spindle was turning before this command, it will be started again. If it was stopped before this command it will be left off. M41 is ignored if there is no gear box.

M42 SPINDLE HIGH GEAR OVERRIDE

The M42 code is used to override the spindle gear implied by the Snnn command. With M42, the spindle gear will always be high. Note that this may reduce the torque at the tool. This M code does not turn the spindle on or off. If the spindle was turning before this command, it will be started again. If it was stopped before this command it will be left off. M42 is ignored if there is no gear box.



M50 EXECUTE PALLET CHANGE (OPTION) (SETTING 121 THRU 129)
Signals the calling program and executes a pallet change sequence.

M51-M58 OPTIONAL USER M CODE SET

The M51 through M58 codes are optional for user interfaces. They will activate one of relays 1132 through 1139 and leave it active. These are the same relays used for M21-M28. Use M61-M68 to turn these off. The RESET key will turn off all of these relays.

M59 OUTPUT RELAY SET (N)

M61-M68 OPTIONAL USER M CODE CLEAR

The M61 through M68 codes are optional for user interfaces. They will deactivate one of relays 1132 through 1139. These are the same relays used for M21-M28.

M69 OUTPUT RELAY CLEAR (N)

M75 SET G35 OR G136 REFERENCE POINT

This code is used to set the reference point used for G35 and G136. It must be used after a motion which is terminated with the skip function.

M76 CONTROL DISPLAY INACTIVE

This code is used to disable the updating of the screen displays. It is not necessary for machine performance.

M77 CONTROL DISPLAY ACTIVE

This code is used to enable the updating of the screen displays. It is only used when M76 has been used to disable the displays.

M78 ALARM IF SKIP SIGNAL FOUND

This code is used to generate an alarm if the previous skip function actually got the skip signal. This is usually used when a skip signal is not expected and may indicate a probe crash. This code can be placed in a block with the skip function or in any subsequent block. The skip functions are G31, G36, and G37.

M79 ALARM IF SKIP SIGNAL NOT FOUND

This code is used to generate an alarm if the previous skip function did not actually get the skip signal. This is usually done when the absence of the skip signal means a positioning error of a probe. This code can be placed in a block with the skip function or in any subsequent block. The skip functions are G31, G36, and G37.



M80 AUTOMATIC DOOR OPEN (OPTION) (SETTING 131)

The M-Codes that control the Auto Door have been changed. The new M-codes are M80 to open. Note: Setting 51 DOOR HOLD OVERRIDE must be set to ON, parameter 57 bit 31 DOOR STOP SP must be set to zero and setting 131 AUTO DOOR set to ON, an M80 will cause the door to open and an M81 will cause it to close. Also, the control will beep while the door is in motion.

M81 AUTOMATIC DOOR CLOSE (OPTION) (SETTING 131)

An M80 will cause the door to open and an M81 will cause it to close. Also, the control will beep while the door is in motion.

M82 TOOL UNCLAMP

This code is used to release the tool from the spindle. It is not normally needed as tool change operations do this automatically and a manual TOOL RELEASE button is available to the operator.

THIS M CODE IS NOT RECOMMENDED FOR USE AS THE TOOL WILL BE DROPPED FROM THE SPINDLE AND MAY DAMAGE THE TOOL, THE MACHINE, OR YOUR SETUP.

M83 AUTO AIR JET ON (OPTION)

An M83 will turn the Air Gun on. Additionally, an M83 Pnnn (where nnn is in millisecon-nds) will turn it on for the specified time, then off automatically.

Note: The Air Blast is assigned to discrete output #1138. An M84 will turn it off

M84 AUTO AIR JET OFF (OPTION)

An M83 will turn the Air Gun on, and an M84 will turn the Air Gun off.

M86 TOOL CLAMP

This code will clamp a tool into the spindle. It is not normally needed as tool change operations do this automatically and a manual TOOL RELEASE button is available to the operator.

M88 COOLANT THRU THE SPINDLE ON (SETTING 32)

This code is used to turn on the Through the Spindle Coolant (TSC) option.

When M88 is called, the spindle will stop, then the TSC pump will turn on, and the spindle will restart.

M89 COOLANT THRU THE SPINDLE OFF

This code is used to turn off the Through the Spindle Coolant (TSC) option.

When M89 is called, the spindle will stop and the TSC pump will shut off. Turns on purge for the amount of time specified in parameter 237 then turns off purge.



M93 START AXIS POS CAPTURE (P, Q)

These M codes permit the control to capture the position of an auxiliary axis when a discrete input goes high. The format is:

M93 Px Qx (where P is the axis number and Q is a discrete input number from 0 to 63) M94

M93 causes the control to watch the discrete input specified by the Q value, and when it goes high, captures the position of the axis specified by the P value. The position is then copied to hidden macro variable 749.

M94 STOP AXIS POS CAPTURE

M94 stops the AXIS POS CAPTURE.

M95 SLEEP MODE

Sleep mode is essentially a long dwell. Sleep mode can be used when the user wishes his machine to begin warming itself up early in the morning and be ready for use upon his arrival. The format of the M95 command is:

M95 (hh:mm)

The comment immediately following the M95 must contain the hours and minutes that the machine is to sleep for. For example, if the current time were 6pm and the user wanted the machine to sleep until 6:30am the next morning, the following command would be used:

M95 (1230)

Up to 99 hours can be specified thus allowing the machine to sleep for over 4 days. If the time is specified using an incorrect format, alarm 324 DELAY TIME RANGE ERROR will be generated. When the machine enters sleep mode and the following message is displayed:

HAAS
SLEEP MODE
REMAINING TIME nnn MIN.

The message will be re-displayed in a different position on the screen each second so that the user can see at a glance that the machine is sleeping.

This has the advantage of preventing the message from being "burned" into one spot on the screen.

When less than one minute of sleep time remains, the message will change to:



REMAINING TIME nn SEC.

If the user presses any key or opens the door, sleep mode will be cancelled and the active program will wait at the block following the M95 until the user presses the Cycle Start key.

For the last 30 seconds of the sleep time, the machine will beep and display an additional message:

WAKE UP IN nn SECONDS

When the sleep time has elapsed and the active program will continue at the block following M95.

M96 JUMP IF NO INPUT (P, Q)

P Block to branch to when conditional test succeeds

Q Discrete input to test, 0.31

This code is used to test a discrete input for 0 status. When this block is executed and the input signal specified by Q is 0, a branch to the block specified by P is performed. A Pnnnn code is required and must match a line number within the same program. The Q value must be in the range of 0 to 31. These correspond to the discrete inputs found on the diagnostic display page with the upper left being input 0 and the lower right being 31. Q is not required within the M96 block. The last specified Q will be used. This command stops the lookahead queue until the test is made at runtime. Since the lookahead queue is exhausted, M96 cannot be executed when cutter compensation is invoked. M96 cannot be executed from a main DNC program. If you wish to use M96 in DNC, it must be in a resident subroutine called from the DNC program.

The following is an M96 example:

```

N05 M96 P5 Q8      (TEST INPUT DOOR S, UNTIL CLOSED);
N10      (START OF SOME PROGRAM LOOP);
.
.      (PROGRAM THAT MACHINES PART);
.
N85 M21      (EXECUTE AN EXTERNAL USER FUNCTION)
N90 M96 P10 Q27  (LOOP TO N10 IF SPARE INPUT IS 0);
N95 M30      (IF SPARE INPUT IS 1 THEN END PROGRAM);

```



M97 LOCAL SUB-ROUTINE CALL (P, L)

This code is used to call a sub-routine, referenced by a line number N within the same program. A Pnnnnn code is required and must match the N line number. This is used for simple sub-routines within a program and does not require the complication of having a separate program. A local sub-routine must still end with an M99. If there is an L count on the M97 line, the sub-routine will be repeated that number of times.

Main program:

O04321 (Start of main program)

...

... (Part program)

...

M97 P123 (Jumps to line N123, after the M30, to execute a local sub-routine.)

... (The M99 at the end of the sub-routine will cause it to jump back here.)

...

... (Finish part program)

...

M30 (End of main program)

N123 (Identifies the start of the Local Sub-Routine called up by M97 P123)

...

... (Local sub-routine portion of part)

...

M99 (Jumps back to the line after the local sub-routine call in the main program)



M98 SUB-PROGRAM CALL (P, L)

This M98 code is used to call a sub-program. The Pnnnn code is the sub-program number being called; it must be in the same block as the M98. The sub-program number being called must already be loaded into the control, and it must contain an M99 at the end in order to return to the next line in the main program. An L count can also be included on the line containing the M98, which will cause the subroutine to be repeated L times before continuing to the next block.

Main program:

O05432 (Start of main program)

...

... (Part program)

...

M98 P234 (Jumps to program O00234 to execute sub-program)

... (The M99 at the end of the sub-program will jump back here)

...

... (Finish part program)

...

M30 (End of main program)

Sub-program:

O00234 (Identifies the start of a separate sub-program)

...

... (Sub-program portion of part)

...

M99 (Jumps back to the line after the sub-program call in the main program)



M99 SUB-PROGRAM RETURN OR LOOP (SETTING 118)

In the main program, an M99 will cause the program to loop back to the beginning and repeat over and over again without stopping.

Main program:

O06543

...

... (Complete part program)

...

...

M99 (This will cause the program to jump back to the beginning and repeat itself)

An M99 without a P code at the end of a sub-program will return to the main program after executing the sub-routine or macro called.

Main program:

O07654

...

... (Part program)

...

M98 P345 (Jumps to program O00345 to run)

... (The M99 at the end of the sub-program will jump back here)

...

... (Finish part)

...

M30 (End of main program)

Sub-program:

O00345 (Identifies sub-program)

...

... (Sub-program portion of part)

...

M99 (Jumps back to the line after the sub-program call)

Note: If M99 Pnnnn is used at the end of the sub-program, it will cause a jump to line number Nnnnn (containing the same number as Pnnnn) in the main program.

A condition of using an M99 Pnnnn in the Haas control varies from that seen in Fanuc-compatible controls. In Fanuc-compatible controls, M99 Pnnnn will return to the main program and resume execution at block N specified by M99 Pnnnn. For the Haas control, M99 will NOT return to block N specified in the M99 return call, but (like always) will jump to the line after the sub-program call in the main program.



You can simulate Fanuc behavior by using the following code-calling program:

<u>Fanuc</u>	<u>Haas</u>
Main program:	Main program:
O00432	O00543
...	...
N50 M98 P9876	N50 M98 P9876
...	N51 M99 P101
...	...
N100 (to continue here)	...
...	N101 (to continue here)
...	...
...	...
M30	M30
Sub-program:	Sub-program:
O09876	O09876
M99 P101	M99

If you have macros, you can use a global variable and specify a block to jump to by adding #nnn=dddd in the subroutine and then using M99 P#nnn after the subroutine call. There are many ways to jump conditionally after an M99 return when using macros.

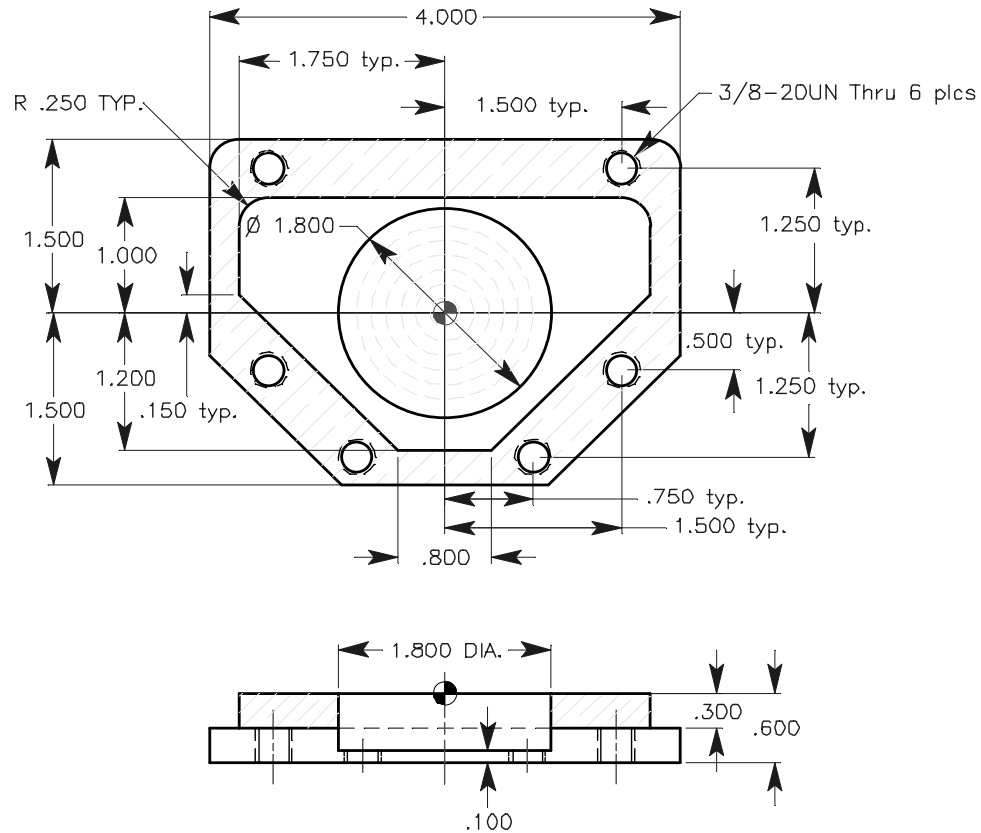
M109 INTERACTIVE USER INPUT (P) (OPTION)

This M code allows a G-code program to place a short prompt on the screen, get a single character input from the user and store it in a macro variable. The first 15 characters from the comment following the M109 will be displayed as a prompt in the lower left corner of the screen. A macro variable in the range 500 through 599 must be specified by a P code. Note also that due to the look-ahead feature, it is necessary to include a loop in the program following the M109 to check for a non-zero response before continuing. The program can check for any character that can be entered from the keyboard by comparing with the decimal equivalent of the ASCII character.

- M164 ROTATE APL GRIPPERS
- M165 OPEN APL GRIPPER 1 (RAW MATERIAL)
- M166 CLOSE APL GRIPPER 1 (RAW MATERIAL)
- M167 OPEN APL GRIPPER 2 (RAW MATERIAL)
- M168 CLOSE APL GRIPPER 2 (RAW MATERIAL)



FINAL EXERCISE



TOOL 1 - 1.00 Dia. 4 flt. End Mill - finish mill contour .3 depth 1 pass.
250 SFM / .003 chip per tooth. S _____ F _____

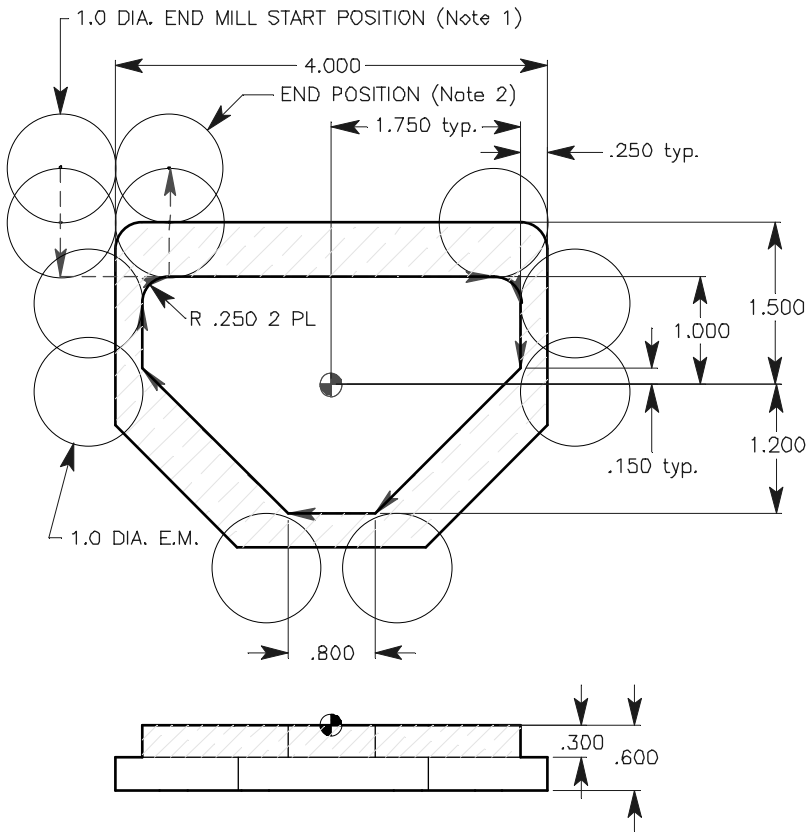
TOOL 2 - 1 3/4 Dia. Insert Drill - rough drill circular pocket .480 deep
with a G81 canned cycle. 250 SFM / .005 chip per tooth.
S _____ F _____

TOOL 3 - 5/8 Dia. 4 flt. End Mill - finish 1.800 dia. x .5 deep circular
pocket. 250 SFM / .0025 chip per tooth. S _____ F _____

TOOL 4 - 1/2 Dia. Spot Drill - drill 6 holes .3 deep with a G82 canned
cycle. 200 SFM / .003 chip per tooth. S _____ F _____

TOOL 5 - 5/16 Dia. Drill - drill 6 holes thru using G73 canned cycle
200 SFM / .0025 chip per tooth. S _____ F _____

TOOL 6 - 3/8-20 UN Tap - tap 6 holes thru with a G84 rigid tapping
machine. Tap at 650 RPM. S650 F _____



TOOL #1

Note 1: Be sure to position the cutter to the center of tool at least half the cutter diameter off of part surface before activating cutter comp.

TOOL #1 - 1.00 dia.
4 flt. End Mill.
Climb cut a finish pass around contour shape that is .3 depth 1 pass using cutter comp.
250. FPM x .003 chip per tooth.

O00080 (FINAL EXERCISE)

T___ M___ (1.00 DIA. END MILL)

G___ G___ G___ X_____ Y_____

S___ M___

G___ H___ Z___ M___

G___ Z___ F50. (Non-cutting fast feed to depth)

G___ Y_____ D___ F___ (Turn on Cutter Comp.)

X_____

G___ X_____ Y_____ R_____

G___ Y_____

X_____ Y_____

X_____

X_____ Y_____

Y_____

G___ X_____ Y_____ R_____

G___ G___ X_____ Y_____ (Cancel Cutter Comp.)

G___ Z___ M___

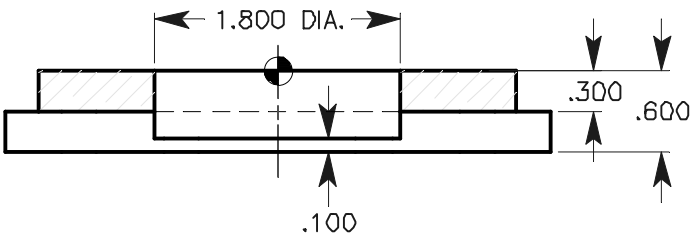
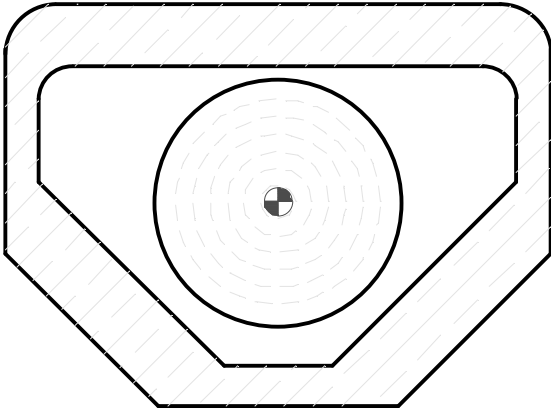
G___ G___ Z___ M___

Note 2: Be sure to position the End Mill at least half the cutter diameter off the part surface, before canceling cutter comp. Cancel back to the center of tool.



TOOL #2

TOOL #2 - 1 3/4 Dia. 2 flute Insert Drill.
Rough drill the 1.800 circular pocket
.480 deep using a G81 canned cycle.
250 FPM x .005 chip per tooth



```

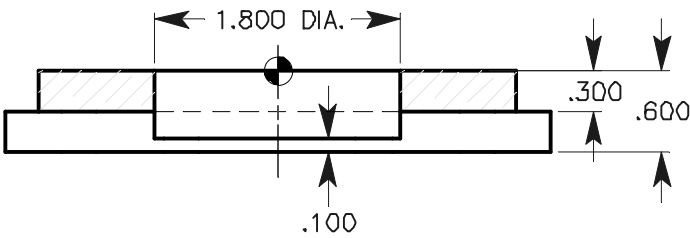
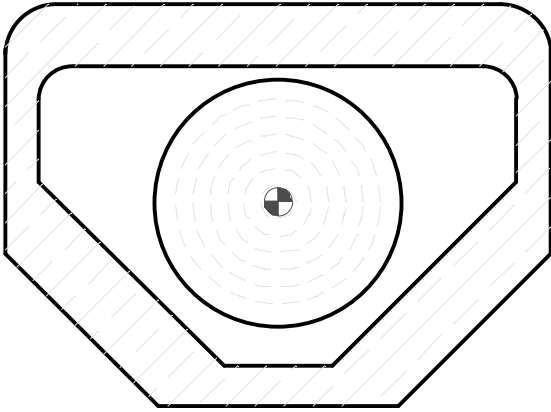
T___ M___ (1 3/4 DIA. INSERT DRILL)
G___ G___ G___ X___ Y___
S___ M___
G___ H___ Z___ M___
G___ Z___ R___ F___
G___ G___ Z___ M___
G___ G___ Z___ M___

```



TOOL #3

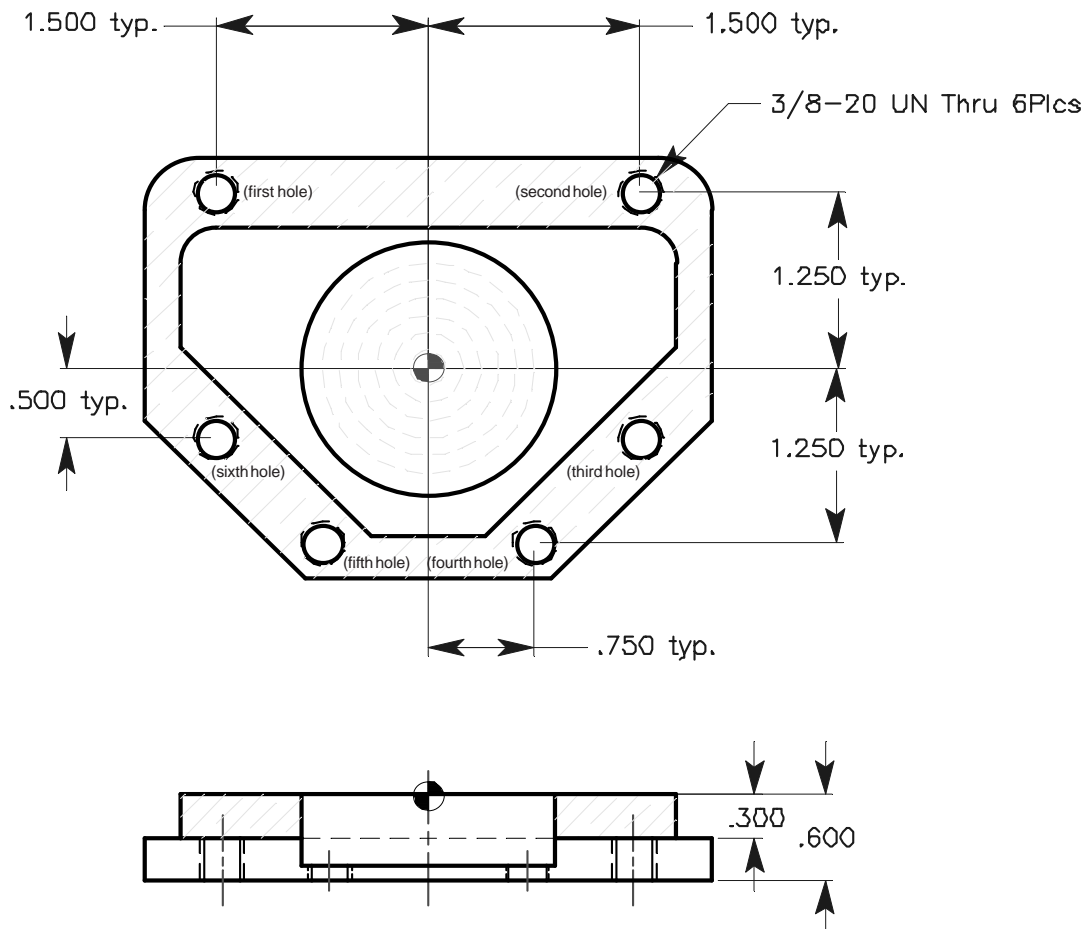
TOOL #3 - 5/8 dia. 4 flt. End Mill
 Finish machine 1.800 circular pocket
 to a .500 depth, using a G13 Circular
 Pocket Milling Command.
 250 FPM x .0025 chip per tooth



```
T___ M___ (5\8 DIA. 4 FLT. END MILL)
G___ G___ G___ X___ Y___
S___ M___
G___ H___ Z___ M___
G___ Z___ F___ (Plunge Z axis down in part at a feedrate)
G___ I___ K___ Q___ Z___ D___ F___
G___ Z___ M___
G___ G___ Z___ M___
```



TOOL 4, 5 & 6



The rapid plane for the drill and tap is R-.2 down from the top surface of part.

TOOL #4 - 1/2 dia. Spot Drill

Use a G82 to spot drill for 6 holes to a depth of -.5 with a rapid plane of -.2 and a 1/2 second dwell at the Z depth.
200 FPM x .003 chip per tooth.

TOOL #5 - 21/64 dia. Drill

Use a G73 to drill 6 holes thru part to a depth of -.75 with a .080 peck depth amount and a -.2 rapid plane.
200 FPM x .0025 chip per tooth.

TOOL #6 is a 3/8-20 Tap

Use a G84 to tap 6 holes thru part to a -.75 depth and a -.2 rapid plane, with a 650 RPM. Machine equipped with rigid tapping.



TOOL 4, 5 & 6

T___ M___ (1\2 DIA. SPOT DRILL)
 G___ G___ G___ X_____ Y_____
 S_____ M_____
 G___ H___ Z_____ M____
 G___ G___ Z_____ P_____ R_____ F____
 M___ P_____

T___ M___ (5\16 DIA. DRILL)
 G___ G___ G___ X_____ Y_____
 S_____ M_____
 G___ H___ Z_____ M____
 G___ G___ Z_____ Q_____ R_____ F____
 M___ P_____

T___ M___ (3\8-16 TAP)
 G___ G___ G___ X_____ Y_____
 S_____ (M03 not needed with G84)
 G___ H___ Z_____ M____
 G___ G___ Z_____ R_____ F____
 M___ P_____

G53 Y0

T1 M06 (Change back to tool #1 for the next part to be run)

M___ (End of program)

Use G98 and G99 for
 the Z position clearance
 location for positioning
 between holes.

G98 Initial Point Return
 G99 R Plane Return

(SUB-PROGRAM)

O00081

G___ X_____
 G___ Y_____
 G___ X_____ Y_____
 G___ X_____
 X_____ Y_____
 G___ G___ Z_____ M____
 G___ G___ Z_____ M____
 M_____

Conversion Formulas

inch x 25.4 = mm
 foot x 304.8 = mm
 mile x 1.609 = km



mm x 0.03937 = inch
 meter x 39.37 = inch
 km x 0.6214 = mile

Fahrenheit to Celsius: $(^{\circ}\text{F} - 32) \div 1.8 = ^{\circ}\text{C}$

Celsius to Fahrenheit: $(^{\circ}\text{C} \times 1.8) + 32 = ^{\circ}\text{F}$

Degrees, Minutes, Seconds to Decimal Degrees: Degrees + (Minutes/60) + (Seconds/3600) = Decimal Degrees

M i l l a n d L a t h e C o n v e r s i o n s

To Find:		Formula:
Revolutions Per Minute	RPM =	(SFM x 3.82) ÷ D
Surface Feet Per Minute	SFM =	RPM x D x 0.262
Surface Meters Per Minute (Metric)	SMPM =	SFM x 0.3048
Feed Per Minute Milling Feedrate	FPM =	FPT x T x RPM
Feed Per Revolution	FPR =	FPM ÷ RPM
Feed Per Tooth - Mill	FPT =	FPM ÷ (T x RPM)
Feed Per Minute - Lathe	FPM =	FPR x RPM
Metal Removal Rate	MMR =	W x d x F
Advance Per Revolution	ADV/R =	F ÷ RPM

T h r e a d s

Mill Tapping Feedrate	FPM =	1 ÷ TPI x RPM
Lathe Threading Feedrate (Thread Lead)	FPR =	1 ÷ TPI
Tap Drill Size	=	Major Dia. of Tap - $\frac{\% \text{ of Thread Height} \times .01299}{\text{TPI}}$
Percent of Full Thread	=	TPI x $\frac{\text{Major Dia. of Tap} - \text{Drill Dia.}}{.01299}$
Mill Tapping Feedrate (Metric)	=	RPM x Metric Pitch
Tap Drill Size (Metric)	=	Tap Major Dia. (mm) - $\frac{\% \text{ of Thread Height} \times \text{Metric Pitch}}{76.980}$
Percent of Full Thread (Metric)	=	$\frac{76.980}{\text{Metric Pitch} \times \text{Basic Major Diameter (mm)} - \text{Drilled Hole (mm)}}$

M i s c e l l a n e o u s

Radius of Circle	=	Circumference x 0.159155
Diameter of Circle	=	Circumference x 0.31831
Circumference of Circle	=	D x 3.1416
Area of Circle	=	R ² x 3.1416
Cutting Time in Minutes (Mill)	=	L ÷ FPM
Cutting Time in Seconds (Lathe)	=	$\frac{\text{Distance to go} \times 60 \text{ sec}}{\text{FPR} \times \text{RPM}}$

A b b r e v i a t i o n s a n d M e a s u r e m e n t U n i t s

D = Diameter of Milling Cutter or Lathe Part	RPM = Revolutions per Minute (Spindle Speed)
F = Feed (Inch or metric)	SFM = surface feet per minute
d = depth of cut	SMPM = Surface Meters per Minute
FPR = Feed per Revolution	T = Number of Teeth in the Cutter
FPT = Feed per Tooth	TPI = Threads per Inch
FPM = Feed per Minute (Table Travel Feedrate)	W = Width of Cut
FPR = Feed per Revolution	°C = Degrees Celsius
L = Length of Cut (Inches)	°F = Degrees Fahrenheit

