

ACR-MotionMax Programmer's Reference Manual

IMPORTANT

User Information



Warning!



ACR Series products are used to control electrical and mechanical components of motion control systems. You should test your motion system for safety under all potential conditions. Failure to do so can result in damage to equipment and/or serious injury to personnel.

ACR series products and the information in this guide are the proprietary property of Parker Hannifin Corporation or its licensors, and may not be copied, disclosed, or used for any purpose not expressly authorized by the owner thereof.

Since Parker Hannifin constantly strives to improve all of its products, we reserve the right to change this guide, and software and hardware mentioned therein, at any time without notice.

In no event will the provider of the equipment be liable for any incidental, consequential, or special damages of any kind or nature whatsoever, including but not limited to lost profits arising from or in any way connected with the use of the equipment or this guide.

© 2004 Parker Hannifin Corporation
All Rights Reserved

Technical Assistance

Contact your local automation technology center (ATC) or distributor.

North America and Asia

Parker Hannifin
5500 Business Park Drive
Rohnert Park, CA 94928
Telephone: (800) 358-9070 or (707) 584-7558
Fax: (707) 584-3793
Email: emn_support@parker.com
Internet: <http://www.parkermotion.com>

Germany, Austria, Switzerland

Parker Hannifin
Postfach: 77607-1720
Robert-Bosch-Str. 22
D-77656 Offenburg
Telephone: +49 (0) 781 509-0
Fax: +49 (0) 781 509-176
Email: sales.hauser@parker.com

Europe (non-German speaking)

Parker Hannifin
21 Balena Close
Poole, Dorset
England BH17 7DX
Telephone: +44 (0) 1202 69 9000
Fax: +44 (0) 1202 69 5750
Email: EMDTech.Help@parker.com

Italy

Parker Hannifin
20092 Cinisello Balsamo
Milan, Italy via Gounod, 1
Telephone: +49 (0) 781 509-0
Fax: +49 (0) 781 509-176
Email: sales.sbc@parker.com



Technical Support E-mail

emn_support@parker.com

Contents

<i>Programming Information</i>	7
Introduction	7
Comments	7
Leading Zeros	7
Zero vs. "Oh"	7
Looping	8
Canned Cycles	8
ACR-MOTIONMAX™ Format Statement	9
G-Code List	10
M-Code List	11
<i>ACR-MOTIONMAX G-Codes</i>	13
G00 RAPID TRAVERSE	13
G01 LINEAR INTERPOLATION	14
G02 and G03 CIRCULAR INTERPOLATION	15
G02 and G03 HELICAL INTERPOLATION	18
G04 DWELL	20
G09 EXACT STOP CHECK	21
G17, G18 and G19 CIRCULAR-PLANE SELECTION	22
G20 and G21 INCH AND METRIC UNITS	22
G28 ZERO RETURN	23
G40, G41, and G42 CUTTER COMPENSATION	24
G43 TOOL LENGTH COMPENSATION (PLUS)	26
G44 TOOL LENGTH COMPENSATION (MINUS)	27
G49 CANCEL TOOL LENGTH COMPENSATION	27
G50 CANCEL SCALING	28
G51 SCALING	28
G53 CANCEL WORK COORDINATE SYSTEMS (WCS)	30
G54-G59 WORK COORDINATE SYSTEMS (WCS)	30

G61 EXACT STOP CHECK MODE	31
G64 CANCEL EXACT STOP CHECK MODE	31
G68 COORDINATE ROTATION	32
G69 CANCEL COORDINATE ROTATION	34
G73 PECK DRILLING CYCLE (CHIP BREAKING)	35
G74 LEFT-HAND TAPPING CYCLE	36
G80 CANCEL CANNED CYCLE	37
G81 STANDARD DRILLING CYCLE	37
G82 DRILL CANNED CYCLE with DWELL	38
G83 PECK DRILLING CYCLE (DEEP HOLE)	39
G84 RIGHT-HAND TAPPING CYCLE	40
G85 REAMING CANNED CYCLE	41
G89 BORING CYCLE WITH DWELL	42
G90 ABSOLUTE POSITIONING MODE	43
G91 INCREMENTAL POSITIONING MODE	43
G92 PROGRAM ZERO DESIGNATION	44
G93 INVERSE TIME FEEDRATE	44
G94 UNITS PER MINUTE FEEDRATE	44
G95 UNITS PER REVOLUTION FEEDRATE	45
G98 and G99 INITIAL PLANE and RAPID PLANE	45
<i>ACR-MOTIONMAX M-Codes</i>	47
M00 PROGRAM STOP COMMAND	47
M01 OPTIONAL PROGRAM STOP	47
M02 END OF PROGRAM	47
M03, M04, and M05 SPINDLE CONTROL COMMANDS	48
M06 TOOL CHANGE	48
M07, M08, M09 COOLANT CONTROL	49
M10 CLAMP ROTARY TABLE	49
M11 UNCLAMP ROTARY TABLE	49
M18 ORIENT TOOL CHANGER	50
M19 ORIENT SPINDLE	50
M30 END OF PROGRAM COMMAND	50
M98 and M99 SUBPROGRAM COMMANDS	50
<i>ACR-MOTIONMAX Programming Words</i>	53

PROGRAMMING WORDS	53
A, B, C.....	53
D.....	53
F.....	53
G.....	53
H.....	54
I, J, K.....	54
M.....	54
N.....	54
O.....	54
P.....	54
Q.....	55
R.....	55
S.....	55
T.....	55
X.....	55
Y.....	55
Z.....	55
Other Programming Characters.....	55
/.....	55
.....	56
().....	56
%.....	56
;.....	56
:.....	56
=.....	56
<i>Glossary of Terms</i>	<i>57</i>
<i>Index.....</i>	<i>59</i>

This page is intentionally left blank.

Programming Information

Introduction

This Programmer's Reference Manual is provided to ACR-MOTIONMAX™ control users for G and M code reference.

The manual discusses each G and M code in numerical order. Unless specifically stated otherwise, all commands in this manual are standard on all ACR-MOTIONMAX™ controls. We will tell you if a code is initialized, modal and which level of product.

Comments

Comments are used to help the programmer or operator understand the program and aid in the editing, debugging, updating, and running of the program. A comment is not executed as part of the CNC program. Comments can be enclosed in parentheses (), or they can follow a percent sign (%), a colon (:) or a semi-colon (;).

A visible comment appears on the screen in the program area when the program is being run. Everything following the comment character on that line is considered a comment and is not executed.

Example commands:

```
G90          (absolute mode)
G00 Z1       : rapid move to Z = 1
G00 X2       ; rapid move to X2
```

Leading Zeros

For all G and M codes beginning with zero (i.e. G01, M06, etc.) entering the zero is optional. In other words, the control will recognize G1 as G01, M6 as M06, etc.

Zero vs. "Oh"

Many CNC program bugs are introduced when the numeral 0 (Zero) is entered instead of the letter O (Oh) or vice versa. Likewise, many errors are caused by entering the letter l (el) or I (eye) for the numeral 1 (one). Some computer fonts make it very difficult to distinguish these characters from one another. The programmer must take care to ensure the proper characters are used.

Looping

The ACR-MOTIONMAX™ control allows Looping within a G-Code program. The command format is: =N100/5. In this example, the “=” indicates that a loop program follows. “N100” indicates the last line of the loop segment. The “5” directs the control to repeat the loop five times, then proceed with the command line following N100.

Example commands:

```
N70 G90 G00 X1 Y1      ;Absolute positioning mode, rapid to start point (1)
=N90/3                ;Execute until line N90, loop three times
N75 G91 G01 X3 F20    ;Incremental positioning mode, feed to point 2
N80 X1.5, Y2          ;Feed to point 3
N85 X-1.5 Y-2         ;Feed to point 1
N90 G00 X5            ;Rapid to new start point, ends loop
N95 G90 G00 X1 Y1    ;Absolute positioning, return to initial start point
```

Looping Notes:

- The specified end-of-loop N number must exist in the file.
- If the same N number is used or specified more than once, the control will use the first one it finds, and ignore any others.
- The = word must be the first character in the looping command line, and there can only be one N word per line, so no sequence number is allowed for the line with the = word.
- An exact stop check is performed at the end of each loop, before repeating the loop or returning to the main program.
- Cutter compensation is allowed in loops, but the user must use caution. You must ensure all parameters will be the same within each loop. For example, assume cutter compensation is enabled when you start a loop, and then you cancel cutter compensation within the loop. If you do not re-instate cutter compensation within the loop, the first loop will cut a compensated path, but subsequent paths will not be compensated.

Nested Loops

Nested loops (a loop within a loop) are allowed up to 10 deep. Each must be completed within the original loop; i.e. loops must be ended in reverse order of how they started. For example:

```
=N100/2              ;begins first loop
=N200/3              ;begins second loop
=N300/2              ;begins third loop
:
N300                 ;ends third loop
N200                 ;ends second loop
N100                 ;ends first loop
```

Canned Cycles

Since the most frequent operation on a mill or machining center is hole making, ACR-MOTIONMAX provides a number of canned cycles for performing these operations. A canned cycle is a fixed sequence of motions performed using variable data at one or many locations. Most of the common canned cycles are controlled by G codes in the range from G80 to G89. A basic set of Fanuc-compatible codes is augmented by extensions documented here.

Canned cycles may be applied to any accessible plane by using the G68 Coordinate Rotation cycle.

ACR-MOTIONMAX™ Format Statement

ACR-MOTIONMAX uses random word address decimal program notation. An address letter precedes every numerical value. Letters may be either upper or lower case. Numbers are represented by either an integer (Ex: H27) or real number (Ex: X1234.5678). Integers have no decimal point, and the number represents the number of places (i.e. the size) of the number. Real numbers require a decimal. In the Format Statement, the number before the decimal represents the number of digits that may precede the point, and the second number represents the number of digits that may follow the decimal point. Note that X3.0000, X3.0, and X3 are all valid. Trailing zeros are not required and have no effect on accuracy. Letters grouped together all have the same format. Multiple occurrences of a code are shown where allowed. Spaces are optional and ignored. Signed numbers are indicated in the format statement by a – (minus sign) following the letter address. The + (Plus sign) may be included but is ignored.

Exceptions:

- When P is used with M98 to specify a sub program, the format is P9.

G-Code List

G-code	Description	Modal	Initialized	Product Level
G00	Rapid traverse	Yes	Yes	1,2,3
G01	Linear Interpolation	Yes	No	1,2,3
G02	Circular Interpolation – CW	Yes	No	1,2,3
G03	Circular Interpolation – CCW	Yes	No	1,2,3
G04	Dwell command	No	No	3
G09	Exact Stop Check	No	No	3
G17	XY plane selection	Yes	Yes	3
G18	XZ plane selection	Yes	No	3
G19	YZ plane selection	Yes	No	3
G20	Inch units	Yes	Yes	1,2,3
G21	Metric units	Yes	No	1,2,3
G28	Send axis to machine home position	No	No	2,3
G40	Cancel cutter radius compensation	Yes	Yes	3
G41	Cutter radius compensation left	Yes	No	3
G42	Cutter radius compensation right	Yes	No	3
G43	Tool length compensation (plus)	Yes	No	2,3
G44	Tool length compensation (minus)	Yes	No	2,3
G49	Tool length compensation cancel	Yes	Yes	2,3
G50	Cancel Scaling	Yes	Yes	3
G51	Scaling	Yes	No	3
G53	Cancel work coordinate systems (WCS)	Yes	No	1,2,3
G54	WCS select for system #1	Yes	No	1,2,3
G54.XX	WCS select alternate 01-99	Yes	No	2,3
G55	WCS select for system #2	Yes	No	1,2,3
G56	WCS select for system #3	Yes	No	1,2,3
G57	WCS select for system #4	Yes	No	1,2,3
G58	WCS select for system #5	Yes	No	1,2,3
G59	WCS select for system #6	Yes	No	1,2,3
G61	Exact stop check mode	Yes	No	1,2,3
G64	Continuous motion mode	Yes	Yes	1,2,3
G68	Coordinate System Rotation	Yes	No	3
G69	Cancel Coordinate System Rotation	Yes	Yes	3
G73	Peck drilling cycle	Yes	No	3
G74	Left hand tapping cycle	Yes	No	3
G80	Cancel canned cycle	Yes	Yes	3
G81	Standard drilling cycle	Yes	No	3
G82	Drilling with dwell	Yes	No	3
G83	Deep hole drilling cycle	Yes	No	3
G84	Right hand tapping cycle	Yes	No	3
G85	Boring cycle	Yes	No	3
G89	Boring cycle with dwell	Yes	No	3
G90	Absolute programming mode	Yes	Yes	1,2,3
G91	Incremental programming mode	Yes	No	1,2,3
G92	Set X, Y, Z locations	Yes	No	1,2,3
G93	Inverse Time Feedrate	Yes	No	3
G94	Units per Minute Feedrate	Yes	Yes	1,2,3
G95	Units per Revolution Feedrate	Yes	No	2,3
G98	Set initial plane as default	Yes	Yes	3
G99	Set rapid plane as default	Yes	No	3

M-Code List

<u>M Code</u>	<u>Description</u>
M00	Program stop
M01	Optional stop
M02	End of program (used by many older programs)
M03	Spindle on in a clockwise direction
M04	Spindle on in a counter clockwise direction
M05	Spindle stop
M06	Tool change command
M07	Mist coolant on (optional)
M08	Flood coolant on
M09	Coolant off
M10	Clamp rotary table
M11	Unclamp rotary table
M18	Orient Tool Changer
M19	Orient Spindle
M30	End of program
M98	Subprogram call
M99	End of subprogram
M101	Hi Pressure coolant on (optional)
M102	Hi Pressure coolant off (optional)
M103	Chip Conveyor On (optional)
M104	Chip Conveyer Off (optional)

This page intentionally left blank.

ACR-MOTIONMAX G-Codes

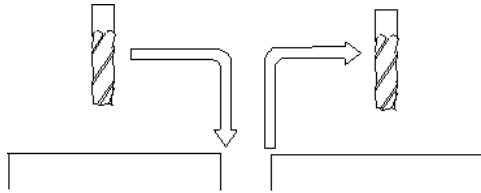
⌘ **NOTE:** Throughout this manual, references to the Z-axis will apply to the tool axis.

G00 RAPID TRAVERSE

Initialized: Yes Modal: Yes
Related Commands: G90, G91, G01, G02, G03

Primary Function:

The G00 command directs the machine axes to ignore the programmed feedrate and move at maximum rate to a specified position. This command is normally used to position the tool to and from its cutting positions, minimizing the dead time during the machining cycle.



The ACR-MOTIONMAX™ control executes an XYZ rapid traverse as a straight line. The control moves the axis with the longest travel at the rapid rate, and adjusts the feedrate of the other axes so that all axes finish the move at the same time.

The G00 end point location depends on the programming mode in use. If in the absolute mode (G90), the end point is taken from program zero (see G92, and G54-G59). If programming in the incremental mode (G91), the end point is taken from the tool's current position prior to the motion command.

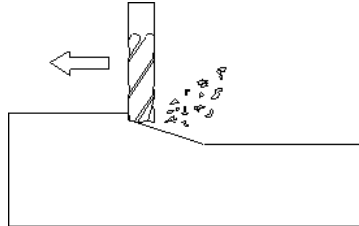
Example command: N010 G00 X1. Y1.

G01 LINEAR INTERPOLATION

Initialized: No Modal: Yes
 Related commands: G90, G91, G00, G02, G03

Primary function:

The linear interpolation command causes the tool to move along a straight-line path, in a cutting mode, at a programmed feedrate. The feedrate word is also modal, so it stays in effect until it is changed. A feedrate of F3.5 equals 3.5 inches per minute (IPM), or millimeters per minute in G21 metric mode.

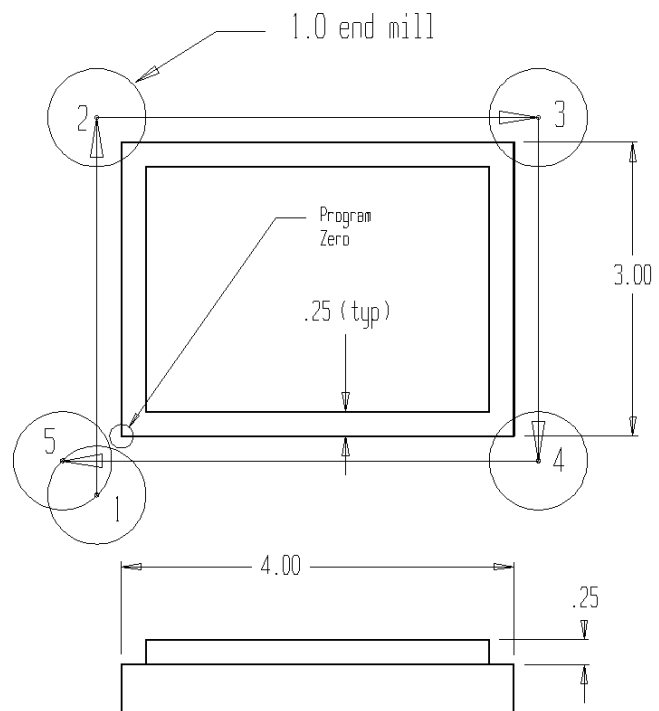


The G01 command is used primarily to machine straight surfaces on your workpiece, such as facing, milling a straight surface, and milling angular surfaces. G01 can command moves on a single axis, or simultaneous moves of all axes.

The G01 end point location depends on the programming mode in use. If in the absolute mode (G90), the end point is taken from program zero (see G92, and G54-G59). If programming in the incremental mode (G91), the end point is measured from the tool's current position prior to the motion command.

Example commands (shows the absolute mode):

O0004	(Program number)
N005 G54 G90 S300 M03	(Select WCS one, absolute mode, spindle CW at 300 rpm)
N010 G00 X-.25 Y-.6	(Rapid to point 1)
N015 G43 H01 Z-.25	(Instate tool length compensation, rapid Z move)
N020 G01 Y3.25 F3.5	(Linear move at feed to point 2)
N025 X4.5	(Feed to point 3)
N030 Y-.25	(Feed to point 4)
N035 X-.6	(Feed to point 5)
N050 M30	



G02 and G03 CIRCULAR INTERPOLATION

Initialized: No

Modal: Yes

Related words: G90, G91, G00, G01

Primary function:

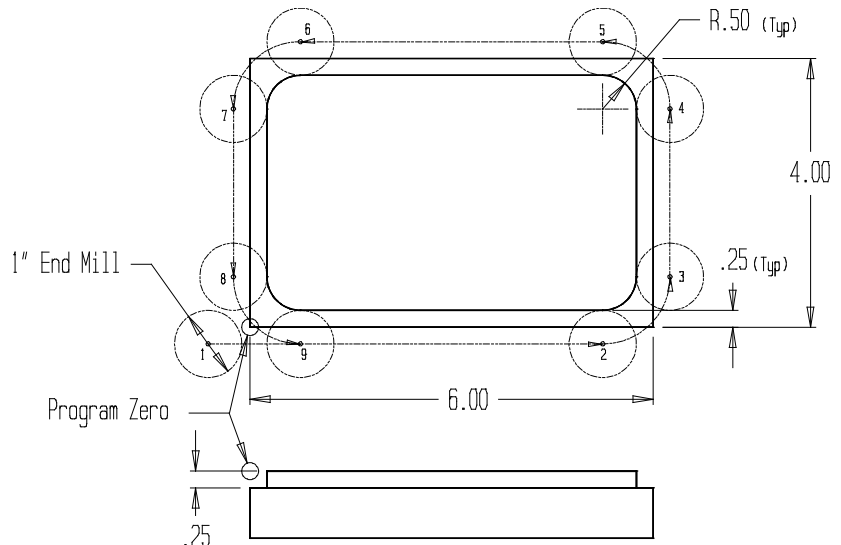
The circular motion commands (G02, G03) produce an arc of a circle by coordinated motion of two axes at a specified feedrate. These commands let you make a smooth radius motion in one command. G02 and G03 are modal, so if a linear move follows a circular motion, you must include a G01 for that motion.

G02 commands a clockwise (CW) circular motion, and G03 commands counterclockwise (CCW). If you intend to make a circular motion along the XZ or YZ plane (which is possible), look at the circular movement from the plus side of the uninvolved axis. For example, if making a circular move along the XZ plane, look at the movement from the plus side of the Y-axis. This would require you to look at the motion as if you were standing on the column side of a vertical machining center. (Before XZ or YZ circular motion can be made, the proper circular plane must be selected. See G17, G18 and G19 for more information.)

The control needs to know three things to cut a circular path: the arc starting point, ending point, and the center point or radius. When a G02 or G03 command is read, the tool will be at the last commanded position, so the machine knows the arc *starting point*. The programmer then designates the *end point* of the arc. The arc center point is defined using either the R-word or with I, J, and K.

Specifying circular motion with the R-word.

The R-word specifies the *arc radius*, and the control interpolates the arc center point. The R-word can be used with arcs of 180 degrees or less or greater than 180 degrees if the R-word is negative. By programming the arc radius and the arc end point, the control calculates and commands the desired arc travel. The *R-word is not modal*. If you have several consecutive circular movements to make, you must program an *R-word* in every circular command. In the command **G02 X2 Y3 R1.5**, X and Y define the arc end point, and R is the arc radius.



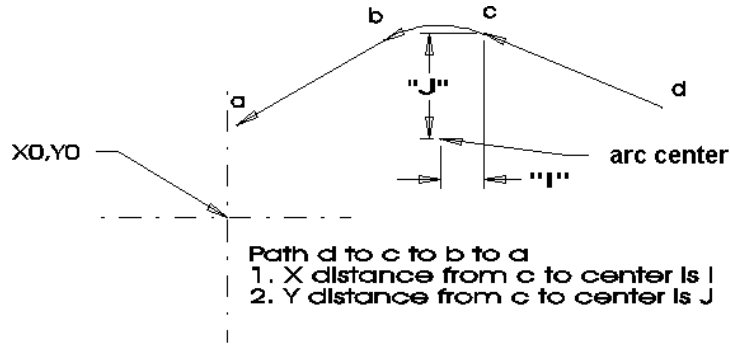
Example program (absolute positioning mode):

```
O0005 (Optional program number)
N010 G54 G90 S350 M03 (Select WCS one, absolute mode, spindle CW at 350 RPM)
N015 G00 X-.625 Y-.25 (Rapid to point 1)
N020 G43 H01 (Instate tool length compensation)
N023 G01 Z-.25 F6 (Feed tool down to work surface)
N025 G01 X5.25 F3.5 (Feed to point 2)
N030 G03 X6.25 Y.75 R1.0 (CCW circular motion to point 3; .5 part radius + .5 cutter radius = R1.0)
N035 G01 Y3.25 (Feed to point 4)
N040 G03 X5.25 Y4.25 R1.0 (CCW circular motion to point 5)
N045 G01 X.75 (Feed to point 6)
N050 G03 X-.25 Y3.25 R1.0 (CCW circular motion to point 7)
N055 G01 Y.75 (Feed to point 8)
N060 G03 X.75 Y-.25 R1.0 (CCW circular motion to point 9)
N065 G00 Z.1 (Rapid away from workpiece)
```

Specifying circular motion with I, J, and K.

Many programs use I, J, and K to define the arc center point, especially when defining an arc greater than 180 degrees. I relates to the X-axis, J to the Y-axis, and K to the Z-axis.

If you are programming using *absolute* positioning, X, Y, and Z define the arc end point based on the origin. With *incremental* positioning, all positions are based on the *previous* position, so X, Y, and Z define the arc end point based on the previous X, Y, Z end point (arc start).



By default, the ACR-MOTIONMAX™ control uses *incremental dimensions* to measure arc center points, regardless of what programming mode – incremental or absolute – you are using. I, J, and K define the incremental distances from the X, Y, Z arc start point to the arc center point (see figure below). If you elect to use absolute arc center points you must change the configuration by selecting F8 Setup, expand the CNC_CONFIG group of the Machine Parameters and change the parameter AbsoluteArcCenters to equal 1 instead of 0.

Two axes will be moving during a circular command (usually X and Y), so two axes are usually commanded. However, there is a way to make one half a circle by specifying only one axis departure. If an axis is not included in the circular command, the control assumes the end point and beginning point are the same.

Circular Interpolation Examples

Using Incremental Arc Center Point:

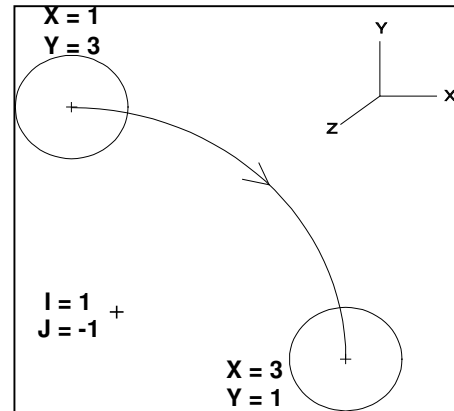
```
G90 ;set absolute positioning
G00 X1.0 Y3.0 ;rapid move to X1Y3
G02 X3.0 Y1.0 I1.0 J-1.0 F20 ;CW arc to X3 Y1, incremental center
point at I1 J-1, feedrate 20
```

These instructions use absolute positioning to mill a clockwise arc in the XY plane. Milling takes place from the initial XY position along an arc ending at a point with coordinates X = 3.0 and Y = 1.0 at a feedrate of 20 units/minute. The arc center point is at X = 1.0 and Y = 1.0. Use the arc center point for full circles and for arcs greater than 180 degrees.

Using Arc Radius:

```
G90 ;set absolute positioning
G00 X1 Y3 ;rapid move to X1 Y3
G02 X3 Y1 R2 F20 ;CW arc to X3 Y1, feedrate 20
```

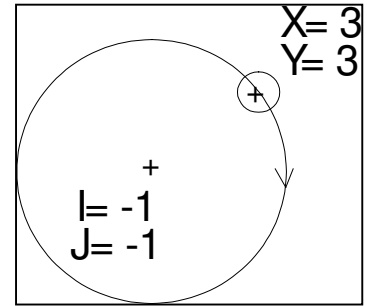
These instructions perform the same milling operation as the previous example. Instead of using the arc center point, though, we've specified the arc radius. The arc radius method is easy to visualize and program; we recommend using it when you don't need to use the arc center point.



One-Command Circle:

```
G00 X3 Y3
G02 I-1 J-1 F20
```

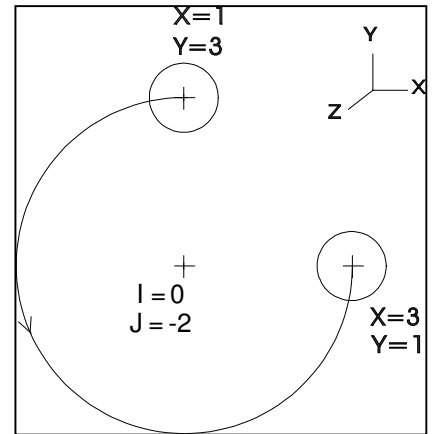
These instructions direct the control to mill a clockwise circle, with the incremental arc center point at X = -1.0 and Y = -1.0 (measured from the start point, X3 Y3). The circle begins and ends at absolute mode coordinates X = 3.0 and Y = 3.0. The radius of the circle is 1.414. The X and Y coordinates could be specified in the G02 block if desired (i.e. G02 X3 Y3 I-1 J-1) but are not required (if using incremental mode, since the end point is the same as the previous position, the command would be G02 X0 Y0I-1 J-1).



Counterclockwise Arc

```
G90 ;set absolute positioning
G00 X1.0 Y3.0
G03 X3.0 Y1.0 I0.0 J-2.0 F20
```

These instructions direct the control to mill a CCW arc in the XY plane using absolute positioning. Milling starts at the initial XY position (X1 Y3), and completes a circular arc ending at X3 Y1, at a feedrate of 20 units/minute. The incremental arc center point is at I0 J-2 (select arc center point type in the F5/F8 Setup Modes screen).

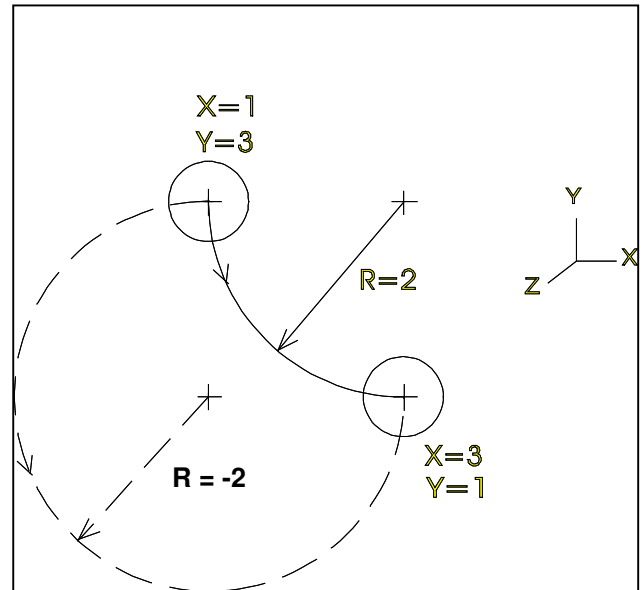


A Caution about Arc Radius

```
G90 ;set absolute positioning
G00 X1 Y3 ;rapid to X1 Y3
G03 X3 Y1 R2 F20 ;CCW arc to X3 Y1
```

The instructions above are the same as the previous example, except that the arc radius has been specified instead of the arc center point. Although you might expect these instructions to perform the same milling operation as the previous example, they do not. The arc radius method can be used to program arcs of less than 180-degrees or greater than 180 degrees. In fact, there are always two CCW arcs with the same start point, end point, and radius. One of the two arcs is greater than 180 degrees (negative R-word), and one is less than 180 degrees (positive R-word).

Although the center point method must be used for full circles, the radius method is recommended for other cases as it is usually easier to program and to visualize.



Other uses: Helical motion (see next topic)

Example commands:

This short program shows how the ACR-MOTIONMAX™ control can be programmed for thread milling, using a thread milling cutter that forms the entire thread in one pass around (the second type of tool discussed). Only the thread milling operation is shown.

```
O0001                                (Thread milling cutter)
N005 G54 G90 S400 M03                (Instate WCS 1, absolute mode, spindle CW at 400 rpm)
N010 G00 X2. Y2.5                    (Segment 1)
N015 G43 H01 Z-.85                   (Rapid through part)
N018 M08                              (Coolant on)
N020 G01 G42 D01 X1.                 (Segment 2)
N025 G02 X2. Y3.5 Z-.8812 R1. F5.    (Segment 3)
N030 G02 X2. Y.5 Z-.9437 R-1.5      (Segment 4)
N035 G02 X2. Y3.5 Z-1.0062 R1.5     (Segment 5)
N040 G02 X3. Y2.5 Z-1.0374 R-1.     (Segment 6)
N045 G00 G40 X2. Y2.5               (Segment 7)
N050 Z1                              (rapid to Z1)
N055 M30                             (program end)
```

The Z-axis departure value in this program may be confusing. Remember that each circular motion must take into consideration the pitch of the thread. For example, in sequence N025, since the circular motion makes a quarter circle, the amount of Z departure is 1/4 of the pitch (.125 pitch divided by 4 equals .0312, and -.85 minus .0312 equals -.8812).

G04 DWELL

Initialized: No

Modal: No

Related commands: G82, G89

Primary function:

This command is used to force the axis motion to pause for a specified length of time. The most common use for this feature is to relieve tool pressure. For example, if you plunge a center cutting end mill into solid stock prior to machining a pocket, it is wise to pause the tool at the bottom of its motion prior to milling in XY. The counter boring cycles G82 and G89 automatically include a way of programming a dwell during the cycle.



Example commands:

N055 G01 Z-.5 F3 (Plunge into stock in Z; feedrate 3 ipm to Z -.5)

N060 G04 P0.500 (Pause all axis motion for .5 second)

The P-word (think P for pause) in the G04 command specifies the length of time for the dwell in seconds. Here are some examples of how the P-word is formatted:

P0.500 = .5 second dwell

P1.000 = 1 second dwell

G09 EXACT STOP CHECK

Initialized: No

Modal: No

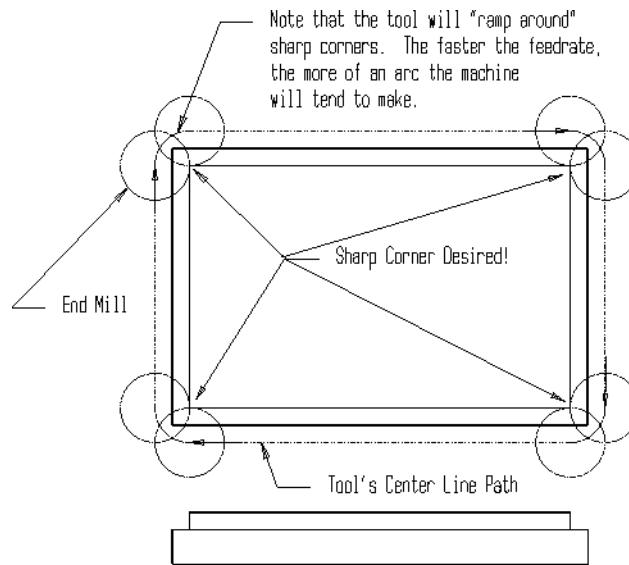
Related commands: G04, G61, G64, Looping

Primary Function:

When you program a series of motion commands the control automatically rounds corners between motions. The control can't form a flowing motion from one command to the next without cheating a little. The amount of rounding is dependent on the feedrate and how a series of parameters are set. In most cases, the rounding is almost unnoticeable.

The corner rounding is not an exact radius. It is caused by the deceleration of one axis, and the acceleration of the next. This must not be confused with the actual radius that results from offsetting the centerline of the cutter when using Cutter Diameter Compensation (G41, G42).

The drawing below illustrates the rounding. As you can see, even though the programmed motions were supposed to form sharp corners, the rounding from one command to the next may cause an undesirable shape.



G09 forces the control to physically stop between commands to keep this rounding from occurring. G09 is a one-time G code, meaning it will only take effect in one command. G61 has the same effect but is modal, and stays in effect until cancelled by G64.

Example Command:

```
N050 G01 G09 X5. F15.
```

```
N055 G01 Y3.
```

Note: The tool stops at the end of block N050.

G17, G18 and G19 CIRCULAR-PLANE SELECTION

Initialized: G17 (for XY Plane) Modal: Yes
Related words: G02, G03, G40, G41, G42, G68, G69

Primary function:

These commands control the plane selection (XY, XZ, or YZ) to be used with circular commands. There are times when the control must be told which plane you will be working in so that it can make decisions accordingly. Much of the time a programmer is working in the XY plane. Since the XY plane (G17) is initialized, there is no need for the G17 to be programmed.

If you intend to do work in another R-plane (XZ or YZ) using circular commands, you must first command the proper R-plane selection.

G18 is XZ, and G19 is YZ. When finished with these commands, you must remember to go back to the XY plane with G17. Some programmers tend to forget G18 and G19 until some alarm occurs on the machine as they execute the program. That is, it is so uncommon to need G18 and G19, you may forget they even exist.

Example command:

N050 G18 (Selects XZ plane)
N055 G19 (Selects YZ plane)

G20 and G21 INCH AND METRIC UNITS

Initialized: G20 Modal: Yes
Related words: All motion words, G70, G71

Primary function:

G20 specifies inch mode. G21 specifies metric mode. Inch mode is initialized.

If you intend to work in the metric mode, *all related dimensions must be input in metric*. Everything in the program, from program zero point measurements to tool length and radius offsets to feedrates must be in millimeters! For this reason, most programmers shy away from using the metric mode for one or two applications. Unless a great percentage of your drawings are produced with metric dimensions, it is hardly worth changing over to the metric mode.

If your company intends to use both modes for the various parts you produce, we strongly recommend that you include a G20 or G21 at the beginning of all programs to avoid the possibility of being in the wrong mode when the program is activated. You will have real problems if you try to run a metric program in the inch mode!

Example commands:

N050 G20 (Selects inch move)
N055 G21 (Selects metric mode)

G28 ZERO RETURN

Initialized: No

Modal: No

Primary function:

G28 is a two part command, causing two movements. The first move is a rapid movement to a specified set of coordinates to ensure fixture avoidance. The second part is a rapid move to machine zero, one axis at a time. Any axis not included in the G28 command will remain in its current position. In the single block mode, you will notice that the control makes two distinct movements whenever a G28 command is executed.

The machine must have been previously homed to safely use a G28. If you haven't homed the machine, the movement will be unspecified, and may result in moving to a limit. Software limits are not active during a G28 move.

The intermediate position is essentially a G00 command, and is affected by the absolute (G90) or incremental (G91) mode. If in the absolute mode, the machine will go to that absolute position, then go home. If in the incremental mode, the machine will move that incremental amount, then go home.

The basic reason for the intermediate position is to allow the programmer to clear obstructions on the way to the machine zero. We recommend that beginning programmers always clear obstructions with G00 *before* using a G28 command.

Once fixture clearance is assured, precede the G28 with a G91 to force the control into incremental mode. By specifying an incremental move of zero, the control will move the designated axes directly to home position. We recommend using this command:

```
N055 G91 G28 X0 Y0 Z0
```

This command tells the machine to go straight to machine zero from its current position. The literal translation is as follows: Incremental mode, move nothing in X, Y, or Z, then go to machine zero. For all intents and purposes, the machine goes straight to machine zero. The ACR-MOTIONMAX™ control has a built-in safety feature that moves Z to home before moving the X and Y axes home.

If you forget to include the G91 in the G28 command, the subsequent absolute mode movement to X0, Y0, and Z0 would send the tool to *program zero*, and a very likely crash!

Example commands:

N055 G91 G28 X0 Y0 Z0	Move nothing in X, Y, and Z, then go to machine zero (essentially straight to machine zero).
N060 G91 G28 Z0	Move nothing in Z, then go to machine zero in Z.
N065 G91 G28 X0 Y0	Move nothing in X and Y, then go to machine zero in X and Y.
N070 G91 G28 X0 Y0 Z4.	Move four inches (up) in Z, then go to machine zero X, Y, and Z.
N080 G90 G28 X3. Y3. Z3.	This command keeps the control in absolute mode. The control is probably in this mode when the G28 command is read. This command tells the machine to go to an absolute position of X3., Y3., and Z3., then to go to machine zero. Using G28 in the absolute mode is possible, but requires more thought and is more error prone.

G40, G41, and G42 CUTTER COMPENSATION

Initialized: G40 (cancel) Modal: Yes
Related Words: G00, G01, G02, G03, G90, G91

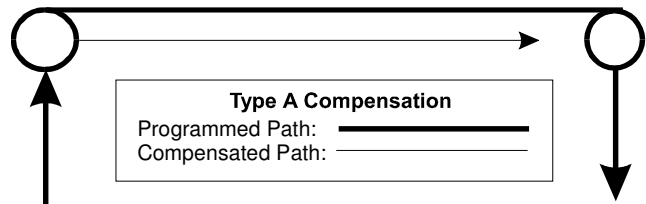
Primary function:

Cutter compensation bases the cutting path on the tool or spindle center line, providing a means for parts programming without having to manually compensate for the diameter of the cutting tool. Cutter compensation is used when milling on the outside diameter of the tool, such as when milling the sides of a part.

Cutter compensation offsets the cutting tool one half of the tool diameter to the **left (G41)** or the **right (G42)** of the work piece relative to the direction of travel. When either code is in use, it must be cancelled with a **G40** before the other can be instated. Cutter compensation is also cancelled by an M30 End of File code.

The lead-in path should be a right angle to the compensated path. The cutter will move along the programmed path until it is one tool radius away from the next segment of the programmed path. The cutter will maintain this offset until cutter radius compensation is cancelled (G40). The offset will be removed during the lead-out move after the G40 command (also at a right angle to the part path).

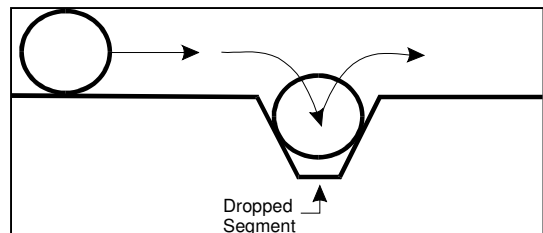
The offset motion occurs during the first linear move (G00 or G01) after G41 or G42 is commanded. The ACR-MOTIONMAX™ control uses Type A compensation (the same as Fanuc Type A) to control how this first motion occurs. The cutter will move from the beginning of the lead-in move to the compensated start of the second move (see drawing at right). On the lead-out move, the cutter will move from the end of the compensated path to the end of the lead-out move.



With cutter compensation, different tools may be used with the same part drawing without manual recalculation. The operator enters the diameter of each tool into a table, and the control calculates the tool radius as an "offset" value used to determine the actual compensated tool path.

The offset value of the tool in use must be set using the "D" word (diameter) in the command line. If no D-word is specified, any previously issued D-word will be used, which is likely to cause a machining error. If no D-word has been issued, an on-screen error is generated.

Lookahead – ACR-MOTIONMAX™ *looks ahead* for crossing conflicts along the compensated path. ACR-MOTIONMAX™ will automatically drop segments where the tool cannot fit.



Cutter Compensation Restrictions

- Cutter compensation can be used with these G-Codes: G00, G01, G02, G03, G04, G90, G91.
- Cutter compensation is not available with canned cycles or rotary axis movements.
- Cutter compensation can be used with all M-Codes *EXCEPT*: M06, M91, M92. Use of any disallowed code will terminate cutter compensation and generate an error.
- D and T-words are not allowed while cutter compensation is active; if included they will be ignored. You must specify cutter and tool information *BEFORE* enabling cutter compensation. If a tool change is required, use an G40 to cancel cutter compensation, change the tool and specify cutter diameter, then re-instate cutter comp.
- If no D or T-word is specified, the default tool radius is 0.0. Negative diameters are allowed. Diameters greater than 1000 or less than -1000 are not allowed.

- Paths within the look ahead may NOT contain crossings except on lead-in or lead-out moves. A path on top of another path is not considered to be a crossing. Crossing paths may result in unpredictable results with no error messages.
- The lead-in and lead-out moves must be lines, not arcs, and must be greater than compensation length.
- There will be no look ahead during the lead-in move, and no look back during the lead-out move. This may generate an over cut during either move with no error message. *Ensure that the first compensated cutter position after lead-in, and the last compensated position after lead-out do not cause an over cut!*
- Cutter compensation may fail if the length of the first or last compensated segments is equal to or less than the compensation length.
- Arcs that are too small for the specified cutter to fit into will be changed over to single points where the edge of the cutter touches the start and stop points of the arc.
- Since G41 and G42 are modal, a G40 command must be included to end the cutter compensation program segment.

Cutter compensation example – milling an outside ACR-MOTIONMAX

Planned cutter size – 1.000 inch in diameter

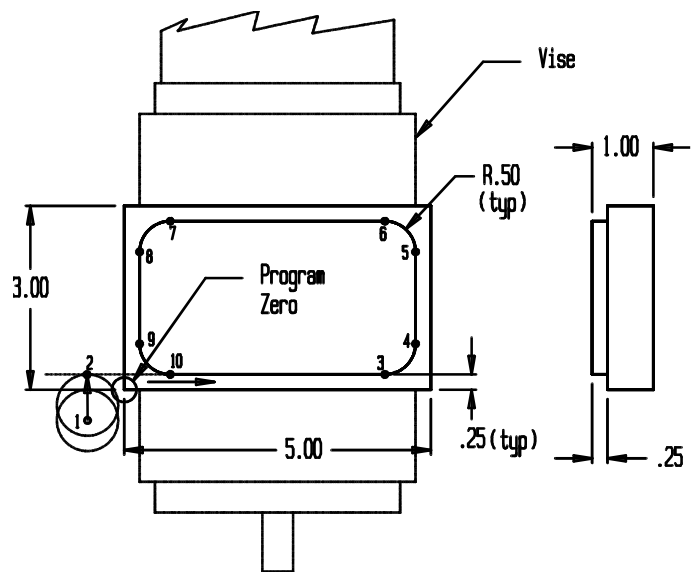
Offset #1 – 1.000 inch

Example Program:

```

O0005 ;Program number
N005 G54 ;Select WCS number one
N010 G90 S350 M03 ;Select ABS mode, turn spindle on at 350 RPM
N015 G00 X-.6 Y-.6 ;Rapid to point 1
N020 G43 H01 Z.1 ;Instate length comp, rapid above part
N025 G01 Z-.25 F30. M08 ;Fast feed to surface, turn on coolant
N030 G42 D01 Y.25 F4. ;Instate cutter comp, feed to point 2
N035 X4.25 ;Feed to point 3
N040 G03 X4.75 Y.75 R.5 ;Circular move to point 4
N045 G01 Y2.25 ;Feed to point 5
N050 G03 X4.25 Y2.75 R.5 ;Circular move to point 6
N055 G01 X.75 ;Feed to point 7
N060 G03 X.25 Y2.25 R.5 ;Circular move to point 8
N065 G01 Y.75 ;Feed to point 9
N070 G03 X.75 Y.25 R.5 ;Circular move to point 10
N080 G00 Z.1 M09 ;Rapid to clearance point above part, turn off coolant
N082 G00 X-.6 ;Lead out move
N085 G40 ;Cancel cutter compensation
N090 M30 ;End of program

```

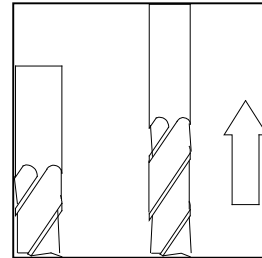


G43 TOOL LENGTH COMPENSATION (PLUS)

Initialized: No Modal: Yes
Related words: G00, G44, G49, G90, G91

Primary Function:

This command allows the programmer to forget about the length of each tool in the program as the program is written. The ACR-MOTIONMAX™ control allows the operator to measure the length value for each tool and input the value to a table of up to 200 tools.



The G43 command is used to instate tool length compensation. The command **must include an H word** that calls up a tool offset containing the tool length compensation value set by the operator. Although the ACR-MOTIONMAX™ control will accept any offset number, it is easiest to use the offset corresponding to the current tool. Normally, then, if using tool number one, H01 should be used for the offset.

However, if an application requires it, the ACR-MOTIONMAX™ control allows you to use one tool, but use the length offset for a different tool. The H word can be added on any line to change the tool offset when G43 is in force. G49 will cancel the H word (see second example below)

Also included in the G43 command is a Z departure, which is the new Z coordinate to move to. Tool length compensation should be instated with the first Z-axis approach to the workpiece for each tool.

Once instated, tool length compensation remains in effect until the next tool is selected. Since the next tool will instate a new tool length compensation value on *its* first Z-axis move to the workpiece, there is no need to cancel tool length compensation when using our method for programming. G49 cancels tool length compensation.

If a tool is active in F4 Run mode, it becomes inactive if you change to F5 Setup mode. If you break out of a part file and change to setup, even though the same tool in is the spindle you must manually reactivate it in the Tool Offset screen.

Example command: N015 G90 G00 G43 H01 Z.1

This command tells the control to bring the tip of the tool to a Z ending point of .1 inch above the program zero point.

NOTE: G43 must be instated, with an H-word defined for the current tool.

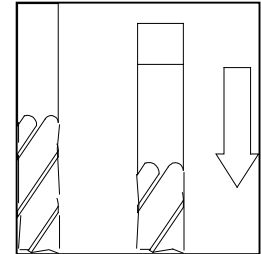
G44 TOOL LENGTH COMPENSATION (MINUS)

Initialized: No Modal: Yes
Related words: G00, G43, G49, G90, G91

Primary function:

This command is identical to the G43 command with one exception. The offset value's sign (+ or -) must be reversed when using G44. Everything else you know about G43 applies directly to G44.

No matter which method you use to measure the tool length offset value (length of tool or distance from tool point at home to program zero), the sign of the value will come out correctly on the machine for use with G43. That is, if measuring the tool length on the machine (by our given method), the sign will come out plus. If measuring the distance from the tip of the tool down to program zero, the sign will come out minus.



For this reason, G43 is much more commonly used in the industry. We recommend using G43 to maintain consistency between machines, but have included G44 for compatibility with older programs.

G49 CANCEL TOOL LENGTH COMPENSATION

Initialized: Yes Modal: Yes
Related words: G00, G43, G44, G49, G90, G91

Primary function:

Generally, there is no need for this command. However, some programmers like to include this word in the command to return the tool to its machine zero for a tool change. Here is an example of how they do it:

N050 G91 G28 G49 Z0

There is no real advantage to doing this, but it doesn't hurt, either. As long as the tool length compensation command (G43) is given in the next tool's first Z-axis command, the new tool length compensation value will be correctly instated.

You can cause a hazardous situation if you touch off the part with the spindle nose for the reference (*not recommended!*) and you forget to include the G43 in the next tool's first Z-axis move. If you do not cancel compensation, at least you will have an even chance of avoiding a crash. You may be lucky and the next tool may be shorter than the current tool. But if tool length compensation is canceled on the way up for a tool change, the control would assume that the nose of the spindle is the tool tip.

We recommend using the Z home position and the negative signed lengths for reference. Then, if you forget to call the length, you'll miss the part by the length of the tool. Much better than a crash!

G50 CANCEL SCALING

Initialized: Yes Modal: Yes
Related words: G00, G01, G02, G03, G51

Primary Function:

This code cancels G51 Scaling mode.

G51 SCALING

Initialized: No Modal: Yes
Related words: G00, G01, G02, G03

Primary Function:

Scaling allows the user to enlarge or reduce the dimensions of a part around a three coordinate center point. Whatever motions are commanded between the G51 and G50 will be scaled.

X, Y, Z identify the scaling center, and must be specified in absolute coordinates. If X, Y, or Z are not specified, the value defaults to zero. If operating in G91 incremental mode, the scaling center is ignored.

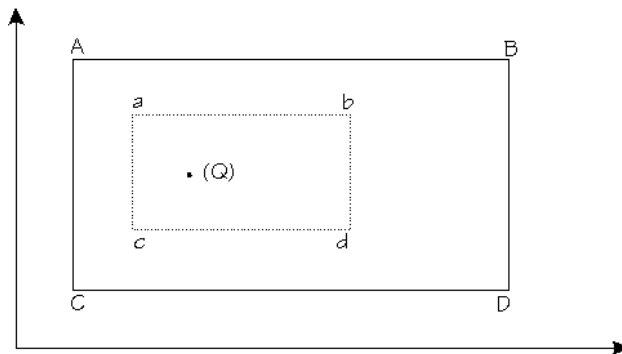
P defines the scaling factor, and may be in the range of 0.001 to 999.99. An error is generated if G51 is commanded for dependent axis scaling and P is not specified. When scaling a rotary axis, P scales the angular amount of axis rotation. Scaling by P will increase ($P > 1$) or decrease ($0 < P < 1$) the distance that a point is from the scaling center.

When G51 is active, each X, Y, and/or Z value in the following program is multiplied by the scale factor and relocated in relation to the scale center. If the original X value is .5, the scale factor is 2.0, and the scale center is (0,0,0) the X value after scaling becomes 1. Multiple G51's of dependent axes is not allowed.

Independent axis scaling is commanded using I, J or Z to define the axis to be scaled. A scaling center should be defined, but if XYZ are not specified, the scaling center will be 0,0,0. The command G51 I0.5 will scale all following X coordinates by 0.5, centered on the point X0 Y0 Z0, until cancelled by G50. Arcs are not allowed during independent axis scaling.

If two axis are scaled independently and simultaneously, the scaling centers must coincide. For example, if two independent axis G51's are commanded before a G50, the second G51 must have the same center as the first.

In the example below, the original part defined by points A, B, C, D will be scaled with scale center Q and $P = 0.5$, resulting in a new part defined by a, b, c, d.



Scaling affects these values and functions:

- Coordinate rotation
- Linear and arc feed moves
- Linear rapid moves
- Drilling cycle depth values (including rapid plane)

Scaling has no effect on these values and features:

- Cutter compensation value
- Tool offset value
- Tool length compensation value
- Tool changes
- G73 and G83 peck depth values

Example commands

Absolute mode:

G90 (absolute mode; starting position 2,0,0)
G51 X3 Y0 Z0 P0.5
G1 X2 (moves to X 2.5)
G1 Y2 (moves to Y 1)
G1 X-2 (moves to X 0.5)
G1 Y-2 (moves to Y -1)
G50 (cancel scaling)

Incremental mode:

G91 (incremental mode; starting position 2,0,0)
G51 X3 Y0 Z0 P0.5 (the center does not become (5,0,0) since XYZ are in absolute coordinates)
G1 X2 (moves to X 3 increment of 1)
G1 Y2 (moves to Y 1 increment of 1)
G1 X-2 (moves to X 2 increment of -1)
G1 Y-2 (moves to Y 0 increment of -1)
G50 (cancel scaling)

G53 CANCEL WORK COORDINATE SYSTEMS (WCS)

Initialized: No Modal: Yes
Related words: G54-G59

Primary function:

G53 cancels the current work coordinate system (also called Fixture Offsets). G53 is modal in the ACR-MOTIONMAX™ control, and returns to the machine coordinate system that was in place prior to instating a WCS.

Work coordinate systems may be re-instated at any time using the G54, G55, G56, G57, G58, or G59 commands.

Example program:

```
N050 G00 X0.000 Y0.000           ;Move to program zero position  
N055 G54                         ;Instate WCS 54  
N060G01 X10.000 Y2.000         ;Move to X10 Y2 in new coordinate system  
N065 G53                         ;Return to program coordinate system
```

G54-G59 WORK COORDINATE SYSTEMS (WCS)

Initialized: No Modal: Yes
Related words: G53, G92

Primary function:

The operator stores the values for the coordinate system in the ACR-MOTIONMAX™ WCS display screen. These WCS measurements are taken *from* the machine zero point *to* the program zero point for each axis. For vertical machining centers, these values will usually be negative since machine home is at or near the positive limit.

It is simple to activate work coordinate systems from the program. G54 specifies that you want to work in WCS number one, G55 for number two, and so on. When the control reads a G54 command, it uses the values included in the G54 setting parameters to calculate the distances from machine zero to program zero.

Additional work offsets... Additional Work Coordinate Systems can be utilized by combining a call to G54 with a decimal value to indicate the offset in the offset table. G54.01 is the same as calling the standard G54. The following table reflects all of the additional WCS...

G54	G54.01
G55	G54.02
G56	G54.03
G57	G54.04
G58	G54.05
G59	G54.06

G54.07 (NOTE: G54.07-G54.99 is only available in level 2 or 3)

.
. .
. .

G54.99

G61 EXACT STOP CHECK MODE

Initialized: No

Modal: Yes

Related words: G09, G64

Primary function:

This command is identical to the G09 command, except that the G61 is modal. That is, you do not have to include a G61 in every command after which you want the motion to stop. See the G09 description for complete information on Exact Stop Check.

Example Commands:

N050 G61

N055 G01 X5. F15.

N060 Y5.

N065 X0

N070 Y0

N075 G64 (Back to normal cutting mode)

The machine will pause after every command from block N055 to block N070.

G64 CANCEL EXACT STOP CHECK MODE

Initialized: Yes

Modal: Yes

Related words: G09, G61

Primary function:

This command is used to cancel the G61 (exact stop check) mode. The machine will power up under the influence of a G64, meaning no command is necessary unless you will be using the G61 command.

Example command: N050 G64

G68 COORDINATE ROTATION

Initialized: No Modal: Yes
Related words: G17, G18, G19, G69

Primary function:

This command is used to rotate the coordinate frame in either two or three dimensions. Rotating in two dimensions, for example, allows an identical shape to be cut at different angles in the XY plane. In three dimensions, a shape can be cut on any arbitrary plane. A subroutine may be called after each rotation to cut a series of identical shapes

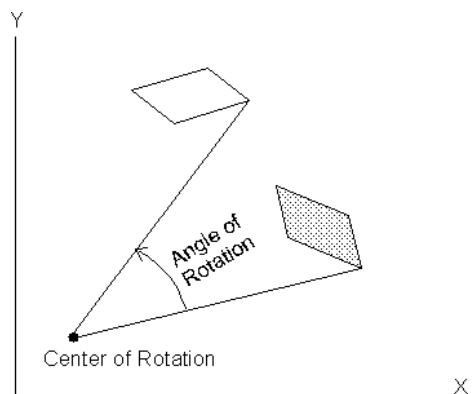
The coordinate display will always display non-rotated coordinates.

⚠ **CAUTION:** G68 is an advanced programming technique, and should not be used until thoroughly understood. Improper programming or incorrect setup may result in damage to the tool and/or workpiece. Ensure the machine is safely positioned for the new coordinate system before issuing a G68 command.

Two-dimensional rotation – In two-dimensional mode, the rotational plane depends on which plane is active, the default XY (G17), XZ (G18), or YZ (G19). The desired plane must be selected before commanding a G68. The rotated plane remains in the active plane. Programming words involved with G68 three-dimensional rotation:

X, Y, or Z – define a two-dimensional point which defines the new origin in the rotated coordinate system. If no X, Y, or Z coordinate is included, the current origin becomes the rotational point for that axis.

R – the angle of plane rotation. Positive (+) R commands a counterclockwise rotation.



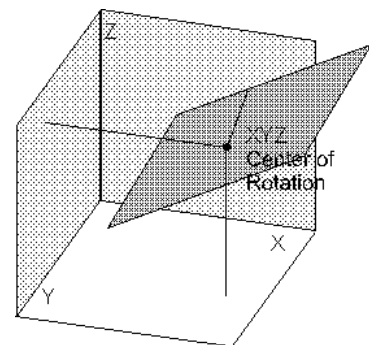
Example command: (assumes G17 – XY plane – is active): **G68 X4 Y3.5 R45.**

Three-dimensional Rotation – Allows user to identify an arbitrary plane, independent of G17, G18, or G19 commands. Programming words involved with G68 three-dimensional rotation:

X, Y, and Z – define the new origin in the rotated coordinate system. If no X, Y, or Z coordinate is included, the current origin becomes the center of rotation for that axis.

I, J, and K – identify the vector to rotate around. I indicates rotation about the X-axis; J indicates rotation about the Y-axis; K indicates rotation about the Z-axis. I, J, and K must be included; if no I, J, and K are defined the rotation will be in two dimensions.

R – commands the angle of rotation. A positive (+) R commands a counterclockwise rotation.



Example command: **G68 X5 Y4 Z5 I0 J1 K0 R45.**

Multiple G68 commands – Multiple plane rotations may be used to rotate the coordinate field, for example to cut on a unique face of a workpiece. If more than one coordinate rotation is specified, the XYZ coordinates and IJK vectors of the second G68 will be based on the rotated coordinate system defined by the first G68.

To illustrate multiple plane rotations, assume we have a simple shape to cut, but it needs to be on a unique plane of the workpiece (see Figure 1). The command **G68 X.75 Y.5 Z.75 I0 J1 K0 R-45** identifies the new origin of the rotated plane as X.75 Y.5 Z.75, specifies the positive Y-axis as the rotation vector, and 45° CW as the angle. In this example, the Y^1 orientation is unchanged from the original Y-axis orientation. X^1 and Z^1 are rotated 45° clockwise.

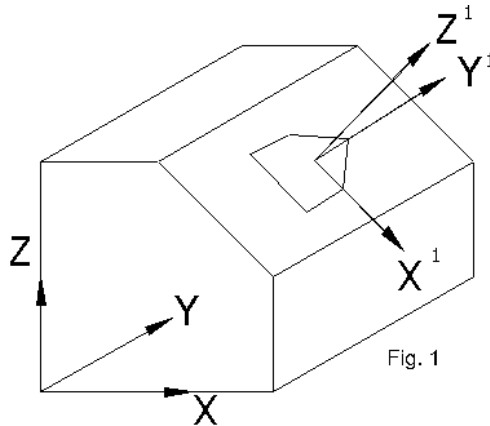


Fig. 1

To obtain the proper orientation for the shape, we need to rotate the plane again (see Figure 2). The command **G68 I0 J0 K1 R90** will rotate the plane 90° CCW about the Z^1 axis. Note that the origin of the plane remains the same, since we did not include an XYZ coordinate in the second command.

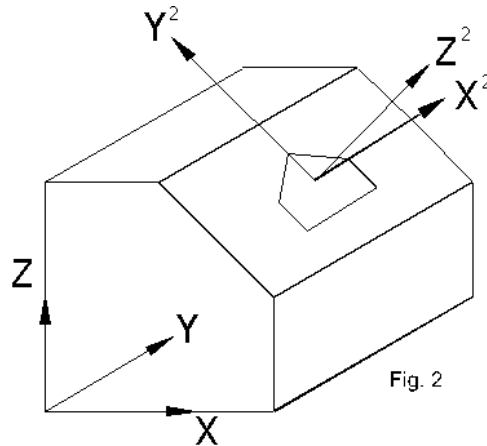


Fig. 2

G68 Limitations

- You cannot command G17, G18, or G19 while in G68 mode.
- G68 cannot be specified while cutter compensation is enabled.
- Do not use G28 while in G68 mode.
- The first G68 command must include the X, Y or Z origin point. Subsequent G68 commands will use the previous G68 origin point if X, Y or Z is not specified.
- If a rotary-axis-only move is specified directly after a G68, then only the specified rotary axis will move; no XYZ movement will occur.

- If a linear move is specified directly after a G68, provide a coordinate value for each axis to prevent an unintentional motion due to the G68 rotation.

G69 CANCEL COORDINATE ROTATION

Initialized: Yes Modal: Yes

Related words: G17, G18, G19, G68

Primary function:

This command is used to cancel the G68 Coordinate Rotation mode. The ACR-MOTIONMAX™ control will power up with G69 in effect, meaning no command is necessary unless you have commanded G68 and wish to cancel it.

G73 PECK DRILLING CYCLE (CHIP BREAKING)

Initialized: No

Modal: Yes

Related Words: G83, G87, G98, G99

Primary function:

This command allows the programmer to specify that a hole be peck drilled to depth in one command. This peck style is best used for chip breaking. If you have trouble with the chips packing around the drill, use this cycle to break the chips at specified intervals going into the hole.

Programming words involved with G73:

N - Sequence number

X - X position of hole center

Y - Y position of hole center

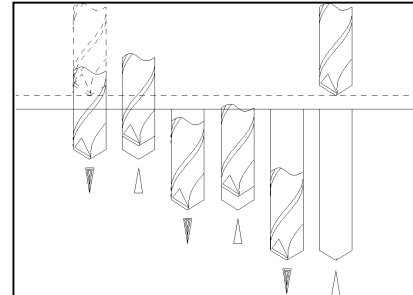
Z - Z position at hole bottom (absolute value)

R - Rapid Plane (Z position of tool BEFORE the hole is drilled)

Q - Peck increment per pass.

F - Feedrate in inches per minute

G98 or G99 - G98 is default. If G99 is specified, the tool comes out of the hole to the R-plane.



Here is how the hole will be machined by this cycle:

- 1) Rapid move to the X and Y position given in the canned cycle command.
- 2) The tool will rapid in Z to the R-plane (if not already there).
- 3) The tool will feed into the hole the initial peck amount, specified by Q.
- 4) The tool will rapid out of the hole the peck break distance (see Machine Setup Parameter) to break the chip.
- 5) The tool will feed into the hole the peck break distance plus another peck amount (Q).
- 6) Steps 4 and 5 will be repeated until the hole bottom depth (commanded by Z) is reached.
- 7) Tool will rapid out of the hole to the initial plane (unless G99 specified).

Example command:

N050 G73 X4. Y3. R.1 Z-1.5 Q.1 F2. G99

G74 LEFT-HAND TAPPING CYCLE

Initialized: No

Modal: Yes

Related words: G84, G98, G99

Primary function:

This command directs the machine to complete a left-hand thread Tapping cycle. Tapping can be completed as either compression (standard) or rigid (optional) tapping. The sequence of each type is described below.

Programming words involved with G84:

N - Sequence number

X - X position of hole center

Y - Y position of hole center

Z - Z position at hole bottom (absolute value)

R - Rapid Plane (Z position of tool before the hole is drilled)

F - Tapping feedrate in inches per minute ($F = \text{RPM} / \text{Pitch}$ or $\text{RPM} \times \text{Lead}$)

S - Spindle speed in RPM

G98 or G99 - If G98 is specified in the command, the tool comes out of the hole to the initial plane. If G99 is specified, the tool comes out of the hole to the R-plane. G98 is default.

NOTE: Feedrate and spindle speed must be determined together depending on the pitch of the thread. The RPM must be programmed such that the calculated feedrate is within the allowable range of the machine, the workpiece material, the tap material, and the dynamics of spindle reversal. These parameters must be appropriately set for the proper push-in and draw-out of the tapping tool.

Compression tapping (used on machines without spindle rotation feedback) follows this sequence. The spindle must be running in the counterclockwise (M04) direction at the correct speed before this command is issued. The

S-word is required for compression tapping.

- 1) Rapid move to the specified XY position.
- 2) The tool will rapid in Z to the R-plane (if not already there).
- 3) The tool will feed into the hole to the hole bottom.
- 4) The spindle direction will reverse (to clockwise).
- 5) The tool will feed back out of the hole to the R-plane.
- 6) The tool will rapid up to the initial plane (unless G99 is specified).
- 7) The spindle will reverse direction to counterclockwise.

Rigid tapping (requires spindle rotation feedback) follows this sequence. If you desire to be able to re-tap the hole, then the "Orient Spindle Before Tapping" feature must be selected ON. The spindle should be stopped before issuing the G74; if it is running when the cycle begins, then the spindle will stop and orient.

- 1) Rapid move to specified XY position.
- 2) The tool will rapid in Z to the R-plane (if not already there). If Orient Spindle Before Tapping is selected, then the spindle will stop and orient.
- 3) The spindle will start at the programmed speed in the counterclockwise direction.
- 4) The tool will feed into the hole to the hole bottom.
- 5) The spindle will reverse direction to clockwise.
- 6) The tool will feed back out of the hole to the R-plane and the spindle will stop.
- 7) The tool will rapid up to the initial plane (unless G99 is specified).

Example command: N050 G74 X4. Y3. R.25 Z-1.5 F50 S1200 G99

G80 CANCEL CANNED CYCLE

Initialized: No Modal: Yes
Related words: G73, G74, G81-89

Primary Function:

The G80 command is used to cancel any of the canned cycles. You must remember to cancel a cycle or the control will remain under the influence of the canned cycle for all subsequent commands.

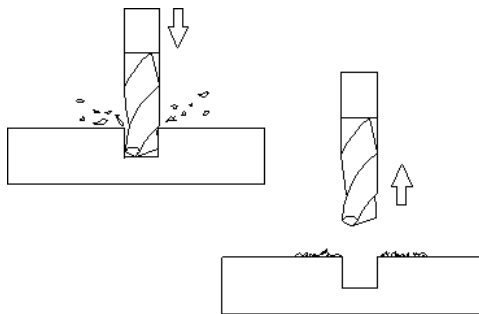
This command may occur on a line by itself, or in combination with any other non-cycle command.

G81 STANDARD DRILLING CYCLE

Initialized: No Modal: Yes
Related words: G73, G83, G98, G99

Primary function:

This command allows the programmer to specify that a hole be drilled to depth in one command. The G81 command makes an excellent command for center drilling and reaming. However, there are other commands that may be better for drilling (G73 and G83).



Programming words involved with G81:

- N - Sequence number
- X - X position of hole center
- Y - Y position of hole center
- Z - Z position at hole bottom (absolute value)
- R - Rapid Plane (Z position of tool before the hole is drilled)
- F - Feedrate in inches per minute
- G98 or G99 - G98 is default. If G99 is specified, the tool comes out of the hole to the R-plane.

Here is how the hole will be machined when under the influence of this cycle:

- 1) Rapid move to X and Y position in the canned cycle command.
- 2) The tool will rapid in Z to the R-plane (if not already there).
- 3) The tool will feed into the hole until the hole depth (Z value) is reached.
- 4) The tool will rapid out of the hole to the initial plane (unless G99 is specified).

Example Command: N050 G81 X4. Y3. R.1 Z-1.5 F2.

G82 DRILL CANNED CYCLE with DWELL

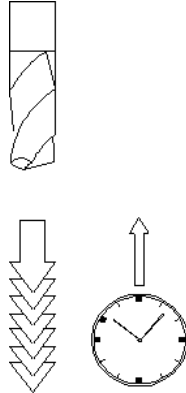
Initialized: No

Modal: Yes

Related words: G81, G98, G99

Primary function:

This command allows the programmer to specify that a hole be counter bored. The only difference between this command and the G81 canned cycle is that the tool will pause at the bottom of the hole for a specified length of time. This allows any tool pressure to be relieved from the cutting operation, which holds the hole depth accuracy more closely.



Programming words involved with G82:

N - Sequence number

X - X position of hole center

Y - Y position of hole center

Z - Z position at hole bottom (absolute value)

R - Rapid Plane (Z position of tool before the hole is drilled)

P - Pause time (P0.500 = .5 second, P1.000 = 1 second, etc).

F - Feedrate in inches per minute

G98 or G99 - G98 is default. If G99 is specified, the tool comes out of the hole to the R-plane.

Here is how the hole will be machined when under the influence of this cycle:

- 1) Rapid move to X and Y position in the canned cycle command.
- 2) The tool will rapid in Z to the R-plane (if not already there).
- 3) The tool will feed into the hole until the hole depth (commanded by Z) is reached.
- 4) The tool will pause for a specified length of time (specified by P)
- 5) The tool will rapid out of the hole to the initial plane (unless G99 is specified).

Example command: N050 G82 X4. Y3. R.1 Z-1.5 P0.500 F2. G99

G83 PECK DRILLING CYCLE (DEEP HOLE)

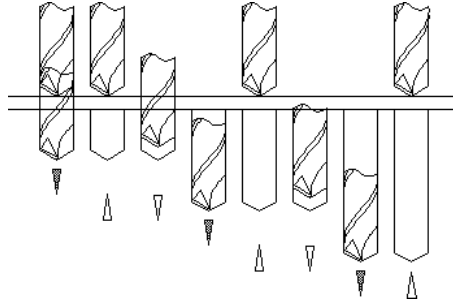
Initialized: No

Modal: Yes

Related words: G73, G98, G99

Primary function:

This command allows the programmer to specify a peck drilled hole to depth in one command. This peck cycle is best used for deep hole machining. As a hole is machined, the chips tend to collect in the flutes of the drill. G83 pecks the tool the value of Q per pass, then rapids the tool out of the hole to clear the chips.



Programming words involved with G83:

N - Sequence number

X - X position of hole center

Y - Y position of hole center

Z - Z position at hole bottom (absolute value)

R - Rapid Plane (Z position of tool before the hole is drilled)

Q - Unsigned peck increment per pass

F - Feedrate in inches per minute

G98 or G99 - G98 is default. If G99 is specified, the tool comes out of the hole to the R-plane.

Here is how the hole will be machined when under the influence of this cycle:

1) Rapid move to X and Y position.

2) The tool will rapid in Z to the R-plane (if not already there).

3) The tool will peck into the hole the amount specified by the "Q" word.

4) The tool will rapid out of the hole to the R-plane (to clear chips).

5) The tool will rapid back into the hole to the peck break distance (see Machine Setup Parameter)

above the previous peck depth.

6) The tool will feed to the incremented peck amount.

7) Steps 3 through 5 will be repeated until the hole depth (commanded by Z) is reached.

8) The tool will rapid out of the hole to the initial plane (unless G99 is specified).

Example Command: N050 G83 X4. Y3. R.1 Z-4.5 Q1.5 F2

G84 RIGHT-HAND TAPPING CYCLE

Initialized: No Modal: Yes

Related Words: G74, G98, G99

Primary function:

This command directs the machine to complete a right-hand thread Tapping cycle (most holes are tapped this way). Tapping can be completed as either compression (standard) or rigid (optional) tapping.

Programming words involved with G84:

N - Sequence number

X - X position of hole center

Y - Y position of hole center

Z - Z position at hole bottom (absolute value)

R - Rapid Plane (Z position of tool before the hole is drilled)

F - Tapping feedrate in inches per minute ($F = \text{RPM} / \text{Pitch}$ or $\text{RPM} \times \text{Lead}$)

S - Spindle speed in RPM (for Rigid Tapping)

G98 or G99 - If G98 is specified in the command, the tool comes out of the hole to the initial plane. If G99 is specified, the tool comes out of the hole to the R-plane. G98 is default.

NOTE: Feedrate and spindle speed must be determined together depending on the pitch of the thread. The RPM must be programmed such that the calculated feedrate is within the allowable range of the machine, the workpiece material, the tap material, and the dynamics of spindle reversal. These parameters must be appropriately set for the proper push-in and draw-out of the tapping tool.

Compression tapping (used on machines without spindle rotation feedback) follows this sequence. The spindle must be running in the clockwise (M03) direction at the correct speed before this command is issued. The S-word is required for compression tapping.

- 1) Rapid move to the specified XY position.
- 2) The tool will rapid in Z to the R-plane (if not already there).
- 3) The tool will feed into the hole to the hole bottom.
- 4) The spindle direction will reverse (to counterclockwise).
- 5) The tool will feed back out of the hole to the R-plane.
- 6) The tool will rapid up to the initial plane (unless G99 is specified).
- 7) The spindle will reverse direction to clockwise.

Rigid tapping (requires spindle rotation feedback) follows this sequence. If you desire to be able to re-tap the hole, then the "Orient Spindle Before Tapping" feature (ACR-MOTIONMAX F5/F8 System Settings screen) must be selected ON. The spindle should be stopped before issuing the G84; if it is running when the cycle begins, then the spindle will stop and orient.

- 1) Rapid move to specified XY position.
- 2) The tool will rapid in Z to the R-plane (if not already there). If Orient Spindle Before Tapping is selected, then the spindle will stop and orient.
- 3) The spindle will start at the programmed speed in the clockwise direction.
- 4) The tool will feed into the hole to the hole bottom.
- 5) The spindle will reverse direction to counterclockwise.
- 6) The tool will feed back out of the hole to the R-plane and the spindle will stop.
- 7) The tool will rapid up to the initial plane (unless G99 is specified).

Example command: N050 G84 X4. Y3. R.25 Z-1.5 F50 S1200 G99

See Training Manual chapter 6 for complete discussion of canned cycles.

G85 REAMING CANNED CYCLE

Initialized: No

Modal: Yes

Related words: G81, G98, G99

Primary function:

The reaming cycle is similar to the G81 cycle for reaming (G81 rapids the tool out of the hole). The G85 command, though, feeds the tool into the hole and then feeds out of the hole.

Programming words involved with G85:

N - Sequence number

X - X position of hole center

Y - Y position of hole center

Z - Z position at hole bottom (absolute value)

R - Rapid Plane (Z position of tool before the hole is drilled)

F - Feedrate in inches per minute

G98 or G99 - G98 is default. If G99 is specified, the tool comes out of the hole to the R-plane.

Here is how the hole will be machined when under the influence of this cycle:

- 1) Rapid move to X and Y position in the canned cycle command.
- 2) The tool will rapid in Z to the R-plane (if not already there).
- 3) The tool will feed into the hole until the hole depth (commanded by Z) is reached.
- 4) The tool will feed out of the hole to the R-plane.
- 5) The tool will rapid to the initial plane.

Example Command: N050 G85 X4. Y3. R.1 Z-1.5 F2. G99

G89 BORING CYCLE WITH DWELL

Initialized: No

Modal: Yes

Related words: G85, G98, G99

Primary function:

This cycle can be used for boring a flat bottom hole when a witness mark is permissible in the hole. It is like having a combination of G85 and G82 commands; the tool will pause at the hole bottom to relieve tool pressure.

Programming words involved with G89:

N - Sequence number

X - X position of hole center

Y - Y position of hole center

Z - Z position at hole bottom (absolute value)

R - Rapid Plane (Z position of tool before the hole is drilled)

P - Pause time at hole bottom (P0.500 = .5 second, P1.000 = 1 second, etc.); default is 0

F - Feedrate in inches per minute

G98 or G99 - G98 is default. If G99 is specified, the tool comes out of the hole to the R-plane.

Here is how the hole will be machined when under the influence of this cycle:

- 1) Rapid move to X and Y position in the canned cycle command.
- 2) The tool will rapid in Z to the R-plane (if not already there).
- 3) The tool will feed into the hole until the hole depth (commanded by Z) is reached.
- 4) The tool will pause a specified length of time (specified by the P-word).
- 5) The spindle will stop.
- 6) The tool will rapid out of the hole to the initial plane (unless G99 is specified).
- 7) The spindle re-starts in original direction once the tool clears the hole.

Note: On the way out of the hole (in step 6) the tool will “drag” a line on the inside of the hole.

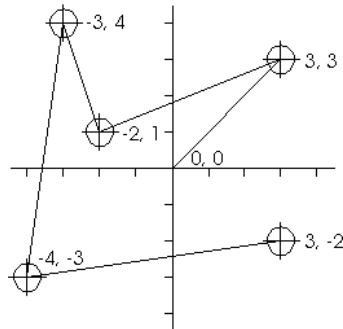
Example Command: N050 G89 X4 Y3 R.1 Z-1.5 P0.500 F2

G90 ABSOLUTE POSITIONING MODE

Initialized: Yes Modal: Yes
Related words: All motion commands

Primary function:

The G90 command tells the control to work from the program zero point. All axis coordinates (X, Y, Z, etc.) programmed in the absolute mode are taken from program zero.



The drawing shows a tool path measured with absolute coordinates. Each point is measured from the origin (0, 0).

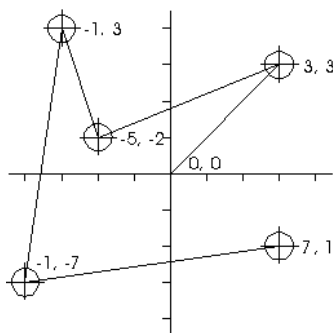
Example Command: N055 G90

G91 INCREMENTAL POSITIONING MODE

Initialized: No Modal: Yes
Related words: All motion commands

Primary function:

The G91 command tells the control to reference the next motion from the current tool position.



The drawing shows a tool path measured with incremental coordinates. Each point is measured from the previous point.

Other uses:

G91 is used in conjunction with the G28 command to help send the machine directly to machine zero. We do not consider this truly programming in the incremental mode. We think of it as tricking the control into going straight to machine zero. And, since a G91 command is given prior to any other axis motion, we really didn't make any real movements in the incremental mode.

Example Command: N050 G91

G92 PROGRAM ZERO DESIGNATION

Initialized: No Modal: Yes
Related words: G54-G59, All motion commands

Primary function:

G92 command is one method to assign program zero, and the only way to do it on some older controls. There are several limitations that have been overcome by work coordinate systems. Since work coordinate systems are standard on all ACR-MOTIONMAX™ controls, we strongly recommend that you program using Work Coordinate Systems, G54 – G59, instead of G92.

Example commands: G92 X0 Y0 Z0

This command makes the present tool position the origin position.

G92 X2.0 Y2.0 Z0

This command sets the current position of the tool (for each axis) at X = 2.0, Y = 2.0, and Z = 0.

G93 INVERSE TIME FEEDRATE

Initialized: No Modal: Yes
Related words: G01, G02, G03, G94, G95, F

Primary function:

G93 provides a different way to specify feedrate, primarily for use when completing 5-axis operations. When G93 is active, feedrate is measured as one move per minute. Therefore, F1 directs the machine to complete the commanded movement in one minute. F2 = 30 seconds, F5 = 12 seconds, F.5 = 120 seconds, etc. F0 commands the maximum allowed cutting feedrate for the machine.

G93 is modal, but the F-word in G93 mode is not. Every line containing a G01, G02, or G03 must contain an F-word, or an error will be generated. The ACR-MOTIONMAX™ feedrate override control is active in G93 mode.

G94 UNITS PER MINUTE FEEDRATE

Initialized: Yes Modal: Yes
Related words: G01, G02, G03, G93, G95, F

Primary function:

This command selects units per minute feedrate mode. The ACR-MOTIONMAX™ control will power up with G94 in effect, meaning no command is necessary unless you have commanded G93 or G95 and wish to cancel it.

G95 UNITS PER REVOLUTION FEEDRATE

Initialized: No Modal: Yes
Related words: G01, G02, G03, G93, G94, F

Primary function:

This command selects units per revolution feedrate mode. The F-word indicates linear units per revolution of the spindle. Example: F.001 with S1000 yields 1 ipm feed.

G98 and G99 INITIAL PLANE and RAPID PLANE

Initialized: G98 - Initial Plane Modal: Yes
Related words: All canned cycles (G73-G89)

Primary function:

The G98 and G99 commands allow the programmer to clear clamps and obstructions while in a canned cycle mode. If G98 (default) is in effect, the tool will come out of the hole to the initial plane. If G99 is in effect, the tool will come out of the hole to the R-plane.

The initial plane is defined as the last Z position of the tool prior to the canned cycle command. The Rapid plane is the R value in the canned cycle itself.

ACR-MOTIONMAX M-Codes

M00 PROGRAM STOP COMMAND

Initialized: No Modal: No
Related words: M01, M02, M30

Primary function:

The M00 command allows the programmer to stop the machine until the cycle start button is pressed. It allows the programmer to give the operator time to do something, such as applying tapping compound, turning the part over, or re-clamping the part.

Whenever you give this command, make it very clear what you expect the operator to do. Place a message to the operator in parentheses in the command line with the M00 command. If M00 is included in a line with X, Y, or Z movements, these movements will be completed before the program stop is executed.

Example Command: N050 X1.225 M00 (Apply tapping compound)

M01 OPTIONAL PROGRAM STOP

Initialized: No Modal: No
Related words: M00, M02, M30

Primary function:

This command functions the same as M00 when "optional stop" is enabled in the ACR-MOTIONMAX™ control (see ACR-MOTIONMAX™ Operations Manual). If optional stop is disabled, M01 is ignored.

M01 can be helpful during program verification, for example. Adding an M01 at the end of every tool allows the operator to inspect what the tool has done before going on to the next tool.

M01 pauses the program *after* the line it is in has been executed. Use caution combining it with other codes.

Example command: N050 M01

M02 END OF PROGRAM

Initialized: No Modal: No
Related words: M00, M01, M30

Primary function:

This command stops all program operation, and turns off the spindle and coolant. M02 and M30 function the same. M02 is included for compatibility with older programs.

(This command was used in the old days of numerical control when tapes were used. If a loop tape - a tape spliced together to form a loop - was used, the M02 command kept the tape from rewinding to get back to the beginning.)

Example command: N150 G00 X1.2 Y2.3 M02

This command will rapid the X-axis to 1.2, the Y-axis to 2.3, and the machine will stop. Spindle and coolant are turned off, and this message will appear on the screen: "END OF FILE."

M03, M04, and M05 SPINDLE CONTROL COMMANDS

Initialized: No

Modal: Yes

Primary function:

These commands control the spindle:

M03 - Spindle on CW (right hand tools).

M04 - Spindle on CCW (left hand tools).

M05 - Spindle off.

NOTES:

1. M03 and M04 turn spindle power ON, M05 turns spindle power OFF.
2. CW and CCW motion are determined by relay output connections.
3. Ensure proper wiring of relay and auxiliary output pins is maintained (see ACR-MOTIONMAX™ Service and Installation Manual).
4. If no S-word has been used to designate spindle RPM, the M03 and M04 commands have no effect.
5. M05 issues an M09 command, turning off all coolant options. Coolant must be turned on again if required.

M06 TOOL CHANGE

Initialized: No

Modal: No

Primary function:

The M06 command executes an automatic tool change, but does not necessarily include tool selection. Use the T command to select the tool. On manual tool change machines, an instruction box appears on the screen.

Example command: N110 T5 S200 M6

This instruction causes the tool to retract to the tool change position, and automatically commands spindle off (M05) and coolant off (M09). A rapid traverse is made for any required X and Y moves to the tool change position.

After inserting the proper tool and restarting the spindle, the operator resumes the program by pressing the START button.

NOTE: The Z-axis position is not physically moved to account for the new tool length. By using a G43 or G44 command after an M06, the Z-axis position will be changed internally and displayed on the controller screen. The new tool length is compensated for during the first Z-axis move after a G43 or G44.

●* **CAUTION:** M06 may automatically issues spindle off (M05) and coolant off (M09) commands. If they are subsequently required, they must be turned back on using M03 or M04 (spindle on, CW or CCW), and M07 or M08 commands.

M07, M08, M09 COOLANT CONTROL

Initialized: No Modal: No
Related words: M05, M06

Primary function:

These commands are simple programmable on/off switches for the coolant.

- **M07** turns on the air mist coolant (optional)
- **M08** turns on the flood coolant
- **M09** turns off all coolant systems

⚠ **NOTE:** If coolant is required after an M05 or M06 is issued, an M08 must be reissued (M05 and M06 may contain an automatic M09, coolant off).

Example Commands:

```
N100 M06 T2           ;coolant off, spindle off, change tool, press START to continue
N110 M08              ;flood coolant on
N120 M03              ;spindle on CW
N220 G00 X0 Y0 Z0 M09;move to 0, turn coolant off
N230 M05              ;turn spindle off
```

M10 CLAMP ROTARY TABLE

Initialized: Yes Modal: Yes
Related words: M11

Primary function:

On machines equipped with a rotary table, M10 engages a mechanical brake to prevent table rotation. Clamping is recommended when cutting a part mounted on the rotary table and rotary motion is not required.

The rotary table default status is clamped. The mechanical brake requires compressed air to be released, so the brake is clamped whenever power or air pressure are lost or removed from the machine.

●* **CAUTION:** The table must be unclamped before issuing a command to rotate the axis.

M11 UNCLAMP ROTARY TABLE

Initialized: No Modal: Yes
Related words: M10

Primary function:

M11 uses compressed air to unclamp the mechanical brake. The rotary table must be unclamped to perform simultaneous contouring using rotary motion. If air pressure is lost, the brake will engage and clamp the table.

M18 ORIENT TOOL CHANGER

Initialized: No

Modal: No

Related words:

Primary function:

After an M18 code is commanded, the tool changer will go to the orient position, usually tool pocket #1. This function is only available on machines with automatic tool changers.

M19 ORIENT SPINDLE

Initialized: No

Modal: No

Related words: M05, M06

Primary function:

After an M19 code is commanded, the spindle will stop and rotate to orient position. This function is only available on machines with spindle orientation capability, such as those with automatic tool changers. On AMCV machines, the M19 is not needed, as the M06 orients the spindle as part of the tool change sequence.

M30 END OF PROGRAM COMMAND

Initialized: No

Modal: No

Related words: M00, M01, M02

Primary function:

The M30 command stops all program operation -- including turning off spindle and coolant. After an M30 command is encountered, the machine is ready to run another part file. M30 and M02 function in the same manner.

Example command: N330 G00 X1.2 Y2.3 Z0 M30

This line commands a Z-axis rapid movement to 0, then a simultaneous XY move to X1.2, Y2.3, then the machine stops. Spindle and coolant are turned off, and an END OF FILE message appears on the controller screen.

M98 and M99 SUBPROGRAM COMMANDS

Initialized: No

Modal: No

Primary function:

These commands allow the program flow to jump to a subprogram. Subprograms are useful, because they relieve the programmer from having to enter redundant commands in a part file. M98 is used to jump to a subprogram, and M99 is used to return from a subprogram. Nested subprograms are allowed up to 10 levels deep. When using nested programs, the subprogram must be below the actual calling M98 block.

Programming words involved with M98:

N - Sequence number

P - program number

L - number of times to loop (optional...assumes 1 time)

Example program:

```
N005 G70 G90           ; inches, absolute positioning
N010 T01 M06          ; tool change
N015 G00 X1 Y1 Z.1    ; rapid to X=1 , Y=1, Z=.1
N025 M98 P100         ; jump to subroutine number 100
N030 T02 M06          ; tool change
N035 G00 X3.0 Y3.0    ; rapid to X=3, Y=3
N040 M98 P100 L2      ; jump to subroutine number 100 2 times
N050 M30              ; end of program
```

(Subroutine for a square)

```
O100                  ; the O word must be on a line by itself
N1 G91 S300          ; incremental positioning
N2 G01 Z-.25 F6      ; feed the tool into the piece
N3 X1                ; incremental square
N4 Y1
N5 X-1
N6 Y-1
N7 G90              ; absolute position
N8 G00 Z.1          ; rapid out of the piece
M99                  ; jump back
```

This page intentionally left blank.

ACR-MOTIONMAX Programming Words

PROGRAMMING WORDS

NOTE: ACR-MOTIONMAX™ controls allow the programmer to include a decimal point with some words. Some older controls do not. The programming words that allow decimal points are: A, B, C, F, I, J, K, Q, R, X, Y, Z.

A, B, C

a. If your machine has a rotary table of some kind (not a simple indexing device), the rotation of the device is considered a true axis of motion, just like X, Y, and Z. Depending on the machine's orientation, the rotary axis can be designated with the letter address A, B, or C. The A-word is used to define the starting angle (in degrees) of a bolt hole circle.

D

- a. The D-word is used in Cutter Compensation to tell the control to compensate for the radius of a cutter other than the cutter specified by the T-word. The D-word can point to any tool number between 1 and 200.
- b. The D-word is also used to define the diameter of a bolt hole circle.
- c. In some probing cycles, D indicates the expected width or diameter of a feature to be probed.

F

With G94 in effect, the F-word specifies the desired feedrate in inches per minute (IPM) in a cutting command (G01, G02, and G03). The F-word allows a decimal point, so a feedrate of 3-1/2 inches per minute is programmed as "F3.5".

With G93 in effect, feedrate is measured as one move per minute. See G93 for more information.

G

This code specifies a preparatory function. Preparatory functions allow various modes to be set in the program. There are many G words, but only a few that are used on a consistent basis. No decimal point is allowed with G words.

H

The H word specifies the offset number to be used with tool length compensation (see Chapter 4 of the ACR-MOTIONMAX™ Training Manual). No decimal point is allowed with the H word. Generally speaking, the programmer should make the H word the same number as the tool station number.

I, J, K

- a. Using I, J, and K is a way to specify the arc in a G02 or G03 circular move. I, J, and K always specify the *incremental* distance and direction in X, Y, and Z (I=X, J=Y, K=Z) from the arc start point to the arc center. I, J, and K allow a full circle to be generated in one CNC command. I, J, and K follow the same decimal point rules as the X word.
- b. In peck drilling cycles – G73, G83, G87 – I, J, and K are used to define peck parameters. I defines the initial peck depth. J defines the amount each peck will be reduced until the peck increment Q is reached. K specifies the distance above the depth of the preceding peck to which the tool will rapid traverse back into the hole.
- c. In G76 Fine Boring Cycle, I and J define the incremental amount the tool will move over before exiting the hole. I and J are not modal for this application.

M

The M word specifies a series of miscellaneous functions. You can think of M words as programmable ON/OFF switches that turn on and off things like coolant and the spindle. One more point to make about M words is that each machine tool manufacturer selects its own set of M words. While there are many standardized M words, consult your machine tool manufacturer's manual to find the exact list for your machine. No decimal point is allowed with the M word.

N

This word specifies a sequence number used for program line identification. It allows the programmer to organize each line in a program by number. This allows easy editing of a program at the machine. Sequence numbers are not required to be in numerical order. In fact, they do not even have to be in the program at all. No decimal point is allowed with the N word.

O

This is the word most controls use for a program number. All controls discussed in this course allow the user to store multiple programs in the memory of the control. The programmer may assign the program a number from 0001 through 9999. The O word will be the very first word in the program. No decimal point is allowed with the O word. Using the O word is optional with the ACR-MOTIONMAX™ control, unless using subprograms. Ensure you use the letter "O", and not the character zero.

P

- a. The P-word is used to specify the length of time in seconds for a G04 dwell command. Dwell commands are used to make the axis motion (for all axes) pause for a specified length of time. A time of three seconds would be specified as P3.000. Other examples: P2.500 = 2.5 seconds, P0.500 = .5 second, and P10.000 = 10 seconds.
- b. A secondary use for the P-word is with subroutines to specify the subroutine program number to be searched. No decimal point is allowed with the P-word. See the ACR-MOTIONMAX™ Training Manual for more information on subroutines.
- c. In G51 Scaling, P defines the scaling factor with a value range 0.001 to 999.99.

Q

The Q-word is used with peck drilling canned cycles (G73, G83) to specify the peck depth for each pass. Decimal point related formatting for the X-word also applies to the Q-word.

R

- a. The primary use for the *R-word* is to specify the radius of a G02 or G03 circular move. All decimal point related functions are the same as for the X word.
- b. The secondary use for the *R-word* is to specify the rapid plane for a canned cycle command such as G73 (See the ACR-MOTIONMAX™ Training Manual for more information).
- c. A fourth use of the R-word is to specify the angle of plane rotation within the G68 Coordinate Rotation code.

S

The S-word specifies a spindle speed. ACR-MOTIONMAX™ controls with spindle interface option allow the programmer to specify an exact RPM. That is, a spindle speed of 350 RPM is programmed "S350". No decimal point is allowed with the S-word.

T

The T-word specifies a three-digit tool station to be placed in the waiting position of the tool changer magazine. At the beginning of one tool we usually program the T-word of the next tool so that when the tool in the spindle is finished cutting, the next tool will be ready. The T-word does not perform a tool change, which requires an M06 code. No decimal point is allowed with the T-word.

X

- a. The X word specifies a position along the X-axis. The X word can be specified with or without a decimal point. With the decimal point, an X position of 10 inches is specified as "X10."
- b. The X-word is also used to define the X center point of a bolt hole circle.

Y

The Y word specifies a position along the Y-axis. All decimal point related functions are the same as for the X word. The Y-word is also used to define the Y center point of a bolt hole circle.

Z

- a. The primary use for the Z word is to specify a position along the Z-axis. All decimal point related functions are the same as for the X word.
- b. The secondary use for the Z word is to specify the hole bottom position in a canned cycle command.

Other Programming Characters

/

- a. A forward slash (/) is the optional block delete word. When the forward slash is the first character in a command line, it works with the Block Delete switch in the ACR-MOTIONMAX™ menu. If the switch is on

when the control reads the slash code, the control will ignore any line that begins with the slash code. If the switch is off, the control will execute the command in the normal manner.

b. Used in a looping command. In the looping command **=N100/5**, the 5 following the forward slash indicates the number of loops to complete.

•

A period (.) preceding a sequence number ensures the line won't be skipped during any Run/Search operation.

()

Parentheses can enclose a comment statement within a program. A comment is not executed as part of the CNC program.

%

A percent sign (%) can be used to indicate a comment statement. A comment is not executed as part of the CNC program.

;

A semi-colon (;) can be used to indicate a comment statement. A comment is not executed as part of the CNC program.

:

A colon (:) can be used to indicate a comment statement. A comment is not executed as part of the CNC program.

=

An equals sign (=) is used to indicate a looping command. The command **=N100/5** indicates that a looping program follows.

Glossary of Terms

Canned cycle

A series of operations to complete a machining process using only one command.

Initialized

A mode that is in effect when the control is first turned on, such as inch mode. Some are user-programmable.

Instate

To initiate, enable, or activate a function or code.

Machine zero

An accurate reference point along each axis; for a vertical machining center, home is typically the extreme positive end of each axis. When the machine has been "homed" the control knows the precise location of the table and spindle

Modal

A command is modal when it stays in effect until specifically cancelled.

Program Zero

A common, logical point on a workpiece from which the part coordinates are based.

This page intentionally left blank.

Index

A

Absolute Positioning, 8, 15–17, 15–17, 43, 51
Arc, 15–18, 15–18, 25, 29, 54
 Center Point, 15–18, 15–18, 25, 29, 54
 End Point, 15–18, 15–18, 25, 29, 54
 Radius, 15–18, 15–18, 25, 29, 54

B

Bolt Hole Circles, 35–42, 35–42
Boring cycle with dwell, 42

C

Canned cycle, 8, 24, 45, 55
CAUTION, 8, 17, 32, 47
Chip breaking, 10, 35, 39
Circular Interpolation, 10, 15–16, 15–16
Comments, 7
Coolant control, 49
Coordinate rotation, 29, 32–34, 32–34, 55
Counter boring cycle, 10
Cutter compensation, 8, 24–25, 24, 29, 33, 53

D

Deep hole machining, 39
Dwell, 10, 20, 42, 54
D-word, 24, 53

E

End of program, 11, 16, 25, 47, 50, 51
Exact Stop Check, 8, 10, 21, 31

F

Feedrate, 10–16, 10–17, 20–21, 20–21, 35–36, 35–36, 37–42,
 35–36, 44–45, 44–45, 53
Fine boring cycle, 54
F-word, 44–45, 44–45, 53

H

Helical Interpolation, 18

I

Inch mode, 22
Incremental Positioning, 8, 16, 43, 51
Initial Plane, 10, 35–36, 35–36, 37–42, 35–36, 45

L

Linear Interpolation, 10, 14
Look Ahead, 25
Looping, 8, 21, 56

M

Machine zero, 23, 27, 30, 43
Metric mode, 22

N

NOTE, 9, 21, 33, 40, 48, 53–54

O

Optional stop, 11

P

Pause, 20, 31, 38, 42, 54
Peck drilling cycle, 10, 35, 39
Plane Selection, 10, 22
Probing, 53
Program stop, 11, 47
Program zero, 13–14, 22–23, 26, 27, 30, 43, 44
Programming words, 32, 35, 36, 37–39, 40–42, 50, 53
P-word, 10, 20, 42, 54

Q

Q-word, 55

R

Rapid plane, 10, 29, 35, 36, 37–39, 40–41, 45, 55
Rapid Traverse, 10, 13, 48, 54
Reaming, 10, 37, 41
Reaming cycle, 41
R-word, 15, 55

S

Spindle control, 48
Standard boring cycle, 42
Subprogram commands, 50
S-word, 36, 40, 48, 55

T

Tapping, 10, 36, 40, 47
Thread milling, 18–19
Tool change, 11, 24, 27, 48, 50, 51, 55
T-word, 24, 53, 55

W

Work coordinate systems, 10, 30, 44

Z

Zero return, 23