Chapter 7: G-codes

7.1 Introduction

This chapter gives detailed descriptions of the G-codes that you use to program your machine.

7.1.1 List of G-codes

CAUTION: *The sample programs in this manual have been tested for accuracy, but they are for illustrative purposes only. The programs do not define tools, offsets, or materials. They do not describe workholding or other fixturing. If you choose to run a sample program on your machine, do so in Graphics mode. Always follow safe machining practices when you run an unfamiliar program.*

NOTE: *The sample programs in this manual represent a very conservative programming style. The samples are intended to demonstrate safe and reliable programs, and they are not necessarily the fastest or most efficient way to operate a machine. The sample programs use G-codes that you might choose not to use in more efficient programs.*

Code	Description	Group	Page
G00	Rapid Motion Positioning	01	**303**
G01	Linear Interpolation Motion	01	**304**
G02	CW Circular Interpolation Motion	01	**311**
G03	CCW Circular Interpolation Motion	01	**311**
G04	Dwell	00	**313**
G09	Exact Stop	00	**314**
G10	Set Offsets	00	**314**
G14	Secondary Spindle Swap	17	**315**

Code	Description	Group	Page
G15	Secondary Spindle Swap Cancel	17	**315**
G17	XY Plane	02	**315**
G18	XZ Plane	02	**315**
G19	YZ Plane	02	**315**
G20	Select Inches	06	**316**
G21	Select Metric	06	**316**
G28	Return To Machine Zero Point	00	**316**
G29	Return From Reference Point	00	**316**
G31	Skip Function	00	**316**
G32	Thread Cutting	01	**317**
G40	Tool Nose Compensation Cancel	07	**320**
G41	Tool Nose Compensation (TNC) Left	07	**321**
G42	Tool Nose Compensation (TNC) Right	07	**321**
G50	Spindle Speed Limit	00	**321**
G50	Set Global coordinate Offset FANUC	00	**322**
G52	Set Local Coordinate System FANUC	00	**322**
G53	Machine Coordinate Selection	00	**322**
G54	Coordinate System #1 FANUC	12	**322**
G55	Coordinate System #2 FANUC	12	**322**
G56	Coordinate System #3 FANUC	12	**322**
G57	Coordinate System #4 FANUC	12	**322**
G58	Coordinate System #5 FANUC	12	**322**
G59	Coordinate System #6 FANUC	12	**322**

Code	Description	Group	Page
G61	Exact Stop Modal	15	**323**
G64	Exact Stop Cancel G61	15	**323**
G65	Macro Subprogram Call Option	00	**323**
G70	Finishing Cycle	00	**323**
G71	O.D./I.D. Stock Removal Cycle	00	**324**
G72	End Face Stock Removal Cycle	00	**327**
G73	Irregular Path Stock Removal Cycle	00	**331**
G74	End Face Grooving Cycle	00	**333**
G75	O.D./I.D. Grooving Cycle	00	**336**
G76	Threading Cycle, Multiple Pass	00	**339**
G80	Canned Cycle Cancel	09	**342**
G81	Drill Canned Cycle	09	**343**
G82	Spot Drill Canned Cycle	09	**343**
G83	Normal Peck Drilling Canned Cycle	09	**345**
G84	Tapping Canned Cycle	09	**347**
G85	Boring Canned Cycle	09	**350**
G86	Bore and Stop Canned Cycle	09	**351**
G89	Bore and Dwell Canned Cycle	09	**351**
G90	O.D./I.D. Turning Cycle	01	**352**
G92	Threading Cycle	01	**353**
G94	End Facing Cycle	01	**355**
G95	Live Tooling Rigid Tap (Face)	09	**356**
G96	Constant Surface Speed On	13	**357**

Code	Description	Group	Page
G97	Constant Surface Speed Off	13	**357**
G98	Feed Per Minute	10	**357**
G99	Feed Per Revolution	10	**357**
G100	Disable Mirror Image	00	**358**
G101	Enable Mirror Image	00	**358**
G103	Limit Block Lookahead	00	**358**
G105	Servo Bar Command	09	**359**
G107	G107 Cylindrical Mapping	00	**359**
G110	Coordinate System #7	12	**361**
G111	Coordinate System #8	12	**361**
G112	XY to XC Interpolation	04	**357**
G113	Cancel G112	04	**362**
G114	Coordinate System #9	12	**362**
G115	Coordinate System #10	12	**362**
G116	Coordinate System #11	12	**362**
G117	Coordinate System #12	12	**362**
G118	Coordinate System #13	12	**362**
G119	Coordinate System #14	12	**362**
G120	Coordinate System #15	12	**362**
G121	Coordinate System #16	12	**362**
G122	Coordinate System #17	12	**362**
G123	Coordinate System #18	12	**362**
G124	Coordinate System #19	12	**362**

Code	Description	Group	Page
G125	Coordinate System #20	12	**362**
G126	Coordinate System #21	12	**362**
G127	Coordinate System #22	12	**362**
G128	Coordinate System #23	12	**362**
G129	Coordinate System #24	12	**362**
G154	Select Work Coordinates P1-99	12	**362**
G156	Broaching Canned Cycle	09	**364**
G167	Modify Setting	00	**370**
G184	Reverse Tapping Canned Cycle For Left Hand Threads	09	**371**
G186	Reverse Live Tool Rigid Tap (For Left Hand Threads)	09	**371**
G187	Accuracy Control	00	**372**
G195	Forward Live Tool Radial Tapping (Diameter)	09	**373**
G196	Reverse Live Tool Radial Tapping (Diameter)	09	**373**
G198	Disengage Synchronous Spindle Control	00	**355**
G199	Engage Synchronous Spindle Control	00	**374**
G200	Index on the Fly	00	**376**
G211	Manual Tool Setting	-	**378**
G212	Auto Tool Setting	-	**378**
G241	Radial Drill Canned Cycle	09	**379**
G242	Radial Spot Drill Canned Cycle	09	**380**
G243	Radial Normal Peck Drilling Canned Cycle	09	**382**
G245	Radial Boring Canned Cycle	09	**384**
G246	Radial Bore and Stop Canned Cycle	09	**385**

Code	Description	Group	Page
G249	Radial Bore and Dwell Canned Cycle	09	**388**
G266	Visible Axes Linear Rapid %Motion	00	**389**

Introduction to G-codes

G-codes are used to command specific actions for the machine: such as simple machine moves or drilling functions. They also command more complex features which can involve optional live tooling and the C Axis.

Each G-code has a group number. Each group of codes contains commands for a specific subject. For example, Group 1 G-codes command point-to point moves of the machine axes, Group 7 are specific to the Cutter Compensation feature.

Each group has a dominant G-code; referred to as the default G-code. A default G-code means they are the one in each group the machine uses unless another G-code from the group is specified. For example programming an X, Z move like this, `X-2. Z-4.` will position the machine using `G00`.

NOTE: *Proper programming technique is to preface all moves with a G-code.*

Default G-codes for each group are shown on the **`Current Commands`** screen under **`All Active Codes`**. If another G-code from the group is commanded (active), that G-code is displayed on the **`All Active Codes`** screen.

G-code commands are either modal or non-modal. A modal G-code stays in effect until the end of the program or until you command another G-code from the same group. A non-modal G-code affects only the line it is in; it does not affect the next program line. Group 00 codes are non-modal; the other groups are modal.

NOTE: *The Haas Intuitive Programming System (IPS) is a programming mode that either hides G-codes or completely bypasses the use of G-codes.*

Canned Cycles

Canned cycles simplify part programming. Most common Z-axis repetitive operations, such as drilling, tapping, and boring, have canned cycles. When active, a canned cycle executes at every new axis position. Canned cycles execute axis motions as rapid commands (`G00`) and the canned cycle operation is performed after the axis motion. This applies to `G17`, `G19` cycles, and Y-Axis movements on Y-Axis lathes.

Using Canned Cycles

Modal canned cycles stay in effect after you define them, and they execute in the Z-axis for each position of the X, Y, and C-Axes.

NOTE: *X, Y, or C-Axis positioning moves during a canned cycle are rapid moves.*

Canned cycles operate differently, depending on whether you use incremental (U,W) or absolute (X, Y, or C) positions.

If you define a loop count (`Lnn` code number) in the canned cycle block, the canned cycle repeats that many times with an incremental (`U` or `W`) move between each cycle.

Enter the number of repeats (`L`) each time you want to repeat a canned cycle. The control does not remember the number of repeats (`L`) for the next canned cycle.

You should not use spindle control M-codes while a canned cycle is active.

Canceling a Canned Cycle

`G80` cancels all canned cycles. `G00` or `G01` code also cancel a canned cycle. A canned cycle stays active until `G80`, `G00`, or `G01` cancels it.

Canned Cycles with Live Tooling

The canned cycles `G81`, `G82`, `G83`, `G85`, `G86`, `G87`, `G88`, `G89`, `G95`, and `G186` can be used with axial live tooling, and `G241`, `G242`, `G243`, `G245`, and `G249` can be used with radial live tooling. Some programs must be checked to be sure they turn on the main spindle before running the canned cycles.

NOTE: *`G84` and G184 are not usable with live tooling.*

G00 Rapid Motion Positioning (Group 01)

*__B__ - B-axis motion command
*__C__ - C-Axis motion command
*__U__ - X-axis incremental motion command
*__W__ - Z-axis incremental motion command
*__X__ - X-axis absolute motion command
*__Y__ - Y-axis absolute motion command
*__Z__ - Z-axis absolute motion command
* __E__ - Optional code to specify the rapid rate of the block as a percent.

* indicates optional

This G code is used to move the machine's axes at the maximum speed. It is primarily used to quickly position the machine to a given point before each feed (cutting) command. This G code is modal, so a block with `G00` causes all following blocks to be rapid motions until another cutting move is specified.

NOTE: *Generally, rapid motion will not be in a straight line. Each axis specified is moved at the same speed, but all axes will not necessarily complete their motions at the same time. The machine will wait until all motions are complete before starting the next command.*

G01 Linear Interpolation Motion (Group 01)

F - Feed rate
* **B** - B-axis motion command
* **C** - C-Axis motion command
* **U** - X-axis incremental motion command
* **W** - Z-axis incremental motion command
* **X** - X-axis absolute motion command
* **Y** - Y-axis absolute motion command
* **Z** - Z-axis absolute motion command
* **A** - Optional angle of movement (used with only one of `X`, `Z`, `U`, `W`)
* **I** - X-axis chamfering from Z to X (the sign does not matter, only for 90 degree turns)
* **K** - Z-axis chamfering from X to Z (the sign does not matter, only for 90 degree turns)

* **,C** - Distance from center of intersection where the chamfer begins (the sign does not matter, can chamfer non-90 degree lines)
* **,R / R** - Radius of the fillet or arc (the sign does not matter)

This G code provides for straight line (linear) motion from point to point. Motion can occur in 1 or more axes. You can command a `G01` with 3 or more axes All axes will start and finish motion at the same time. The speed of all axes is controlled so that the feed rate specified is achieved along the actual path. The C-Axis may also be commanded and this will provide a helical (spiral) motion. A C-Axis feed rate is dependent on the C-Axis diameter setting (Setting 102) to create a helical motion. The `F` address (feedrate) command is modal and may be specified in a previous block. Only the axes specified are moved.

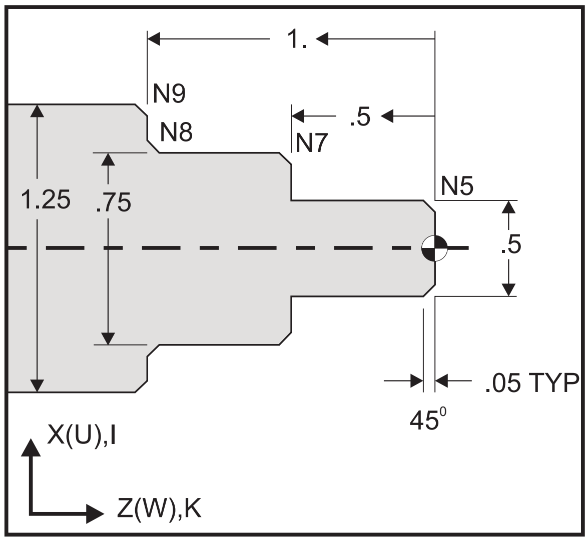
Corner Rounding and Chamfering Example

A chamfer block or a corner rounding block can be automatically inserted between two linear interpolation blocks by specifying `,C` (chamfering) or `,R` (corner rounding).

NOTE: *Both of these variables use a comma symbol (,) before the variable.*

There must be a terminating linear interpolation block after the beginning block (a `G04` pause may intervene). These two linear interpolation blocks specify a theoretical corner of intersection. If the beginning block specifies a `,C` (comma C) the value after the `C` is the distance from the corner of intersection where the chamfer begins and also the distance from that same corner where the chamfer ends. If the beginning block specifies a `,R` (comma R) the value after the R is the radius of a circle tangent to the corner at two points: the beginning of the corner rounding arc block that is inserted and the endpoint of that arc. There can be consecutive blocks with chamfer or corner rounding specified. There must be movement on the two axes specified by the selected plane (the active plane X-Y (`G17`), X-Z (`G18`) or Y-Z (`G19`). For chamfering a 90° angle only, an `I` or `K` value can be substituted where `,C` is used.

F7.1: Chamfering




```
%
o60011 (G01 CHAMFERING) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is an OD cutting tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, Spindle on CW) ;
G00 G54 X0 Z0.25 (Rapid to 1st position) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z0 F0.005 (Feed to Z0) ;
N5 G01 X0.50 K-0.050 (Chamfer 1) ;
G01 Z-0.5 (Linear feed to Z-0.5) ;
```

```
N7 G01 X0.75 K-0.050 (Chamfer 2) ;
N8 G01 Z-1.0 I0.050 (Chamfer 3) ;
N9 G01 X1.25 K-0.050 (Chamfer 4) ;
G01 Z-1.5 (Feed to Z-1.5) ;
(BEGIN COMPLETION BLOCKS) ;
G00 X1.5 M09 (Rapid Retract, Coolant off) ;
G53 X0 (X home) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

This G-code syntax automatically includes a 45° chamfer or corner radius between two blocks of linear interpolation which intersect a right (90 degree) angle.

Chamfering Syntax

```
G01 X(U) x Kk ;
G01 Z(W) z Ii ;
```

Corner Rounding Syntax

```
G01 X(U) x Rr ;
G01 Z(W) z Rr ;
```

Addresses:

I = chamfering, Z to X

K = chamfering, X to Z

R = corner rounding (X or Z axis direction)

Notes:

1. Incremental programming is possible if `U` or `W` is specified in place of `X` or `Z`, respectively. So its actions are as follows:
   X(current position + i) = Ui

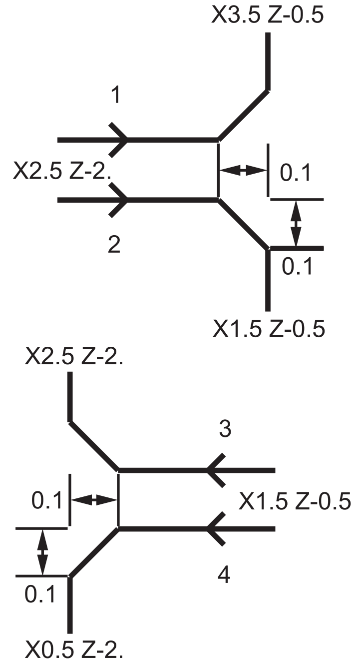
   Z(current position + k) = Wk

   X(current position + r) = Ur

   Z(current position + r) = Wr
2. Current position of X or Z Axis is added to the increment.
3. `I`, `K` and `R` always specify a radius value (radius programming value).

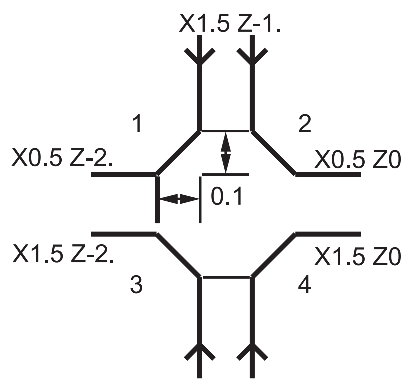
F7.2: Chamfering Code Z to X: [A] Chamfering, [B] Code/Example, [C] Movement.

A	B	C
1. Z+ to X+	X2.5 Z-2; G01 Z-0.5 I0.1; X3.5;	X2.5 Z-2; G01 Z-0.6; X2.7 Z-0.5; X3.5;
2. Z+ to X-	X2.5 Z-2.; G01 Z-0.5 I-0.1; X1.5;	X2.5 Z-2.; G01 Z-0.6; X2.3 Z-0.5; X1.5;
3. Z- to X+	X1.5 Z-0.5.; G01 Z-2. I0.1; X2.5;	X1.5 Z-0.5 G01 Z-1.9; X1.7 Z-2.; X2.5;
4. Z- to X-	X1.5 Z-0.5.; G01 Z-2. I-0.1; X0.5;	X1.5 Z-0.5; G01 Z-1.9; X1.3 Z-2. X0.5;



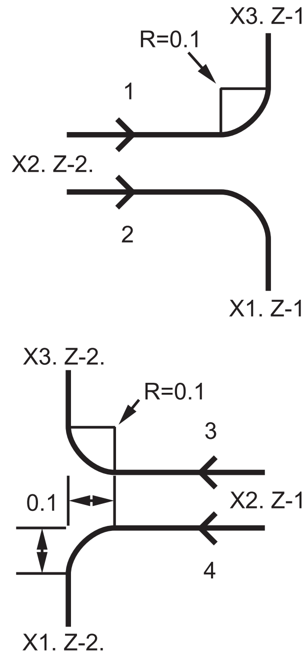
F7.3: Chamfering Code X to Z: [A] Chamfering, [B] Code/Example, [C] Movement.

A	B	C
1. X- to Z-	X1.5 Z-1.; G01 X0.5 K-0.1; Z-2.;	X1.5 Z-1.; G01 X0.7; X0.5 Z-1.1; Z-2.
2. X- to Z+	X1.5 Z-1.; G01 X0.5 K0.1; Z0.;	X1.5 Z-1.; G01 X0.7; X0.5 Z-0.9; Z0.;
3. X+ to Z-	X0.5 Z-1.; G01 X1.5 K-0.1; Z-2.;	X0.5 Z-1.; G01 X1.3; X1.5 Z-1.1; Z-2.
4. X+ to Z+	X0.5 Z-1.; G01 X1.5 K0.1; Z0.;	X0.5 Z-1.; G01 X1.3; X1.5 Z-0.9; Z0.;



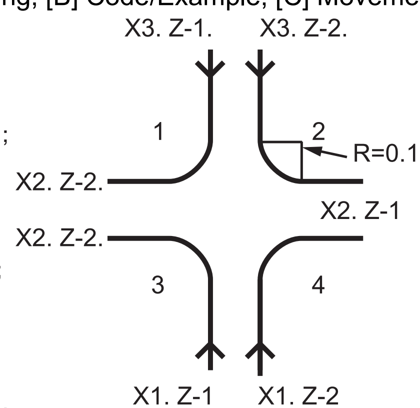
F7.4: Corner Rounding Code Z to X: [A] Corner rounding, [B] Code/Example, [C] Movement.

A	B	C
1. Z+ to X+	X2. Z-2.; G01 Z-1 R0.1; X3.;	X2. Z-2.; G01 Z-1.1; G03 X2.2 Z-1. R0.1; G01 X3.;
2. Z+ to X-	X2. Z-2.; G01 Z-1. R-0.1; X1.;	X2. Z-2.; G01 Z-1.1; G02 X1.8 Z-1 R0.1; G01 X1.;
3. Z- to X+	X2. Z-1.; G01 Z-2. R0.1; X3.;	X2. Z-1.; G01 Z-1.9; G02 X2.2 Z-2. R0.1; G01 X3.;
4. Z- to X-	X2. Z-1.; G01 Z-2. R-0.1; X1.;	X2. Z-1.; G01 Z-1.9; G03 X1.8 Z-2. R0.1; G01 X1.;



F7.5: Corner Rounding Code X to Z: [A] Corner rounding, [B] Code/Example, [C] Movement.

A	B	C
1. X- to Z-	X3. Z-1.; G01 X0.5 R-0.1; Z-2.;	X3. Z-1; G01 X0.7; G02 X0.5 Z-1.1 R0.1; G01 Z-2.;
2. X- to Z+	X3. Z-2.; G01 X0.5 R0.1; Z0.;	X3. Z-2.; G01 X0.7; G03 X0.5 Z-0.9 R01; G01 Z0.;
3. X+ to Z-	X1. Z-1.; G01 X1.5 R-0.1; Z-2.;	X1. Z-1.; G01 X1.3; G03 X1.5 Z-1.1 R0.1; G01 Z-2.;
4. X+ to Z+	X1. Z-2.; G01 X1.5 R0.1; Z0.;	X1. Z-21.; G01 X1.3; G02 X1.5 Z-0.9 R0.1; G01 Z0.;



Rules:

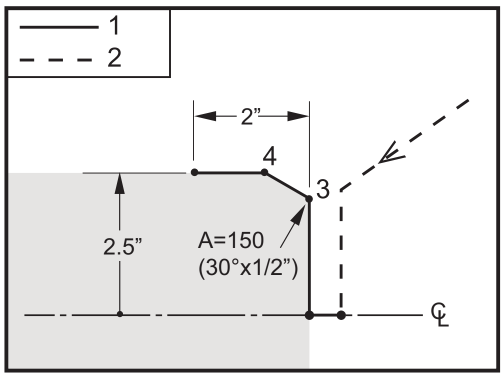
1. Use `K` address only with `X(U)` address. Use `I` address only with `Z(W)` address.
2. Use `R` address with either `X(U)` or `Z(W)`, but not both in the same block.

3. Do not use I and K together on the same block. When using R address, do not use I or K.
4. The next block must be another single linear move that is perpendicular to the previous one.
5. Automatic chamfering or corner rounding cannot be used in a threading cycle or in a Canned cycle.
6. Chamfer or corner radius must be small enough to fit between the intersecting lines.
7. Use only a single X or Z-axis move in linear mode (G01) for chamfering or corner rounding.

G01 Chamfering with A

When specifying an angle (A), command motion in only one of the other axes (X or Z), the other axis is calculated based on the angle.

F7.6: G01 Chamfering with A: [1] Feed, [2] Rapid, [3] Start Point, [4] Finish Point.



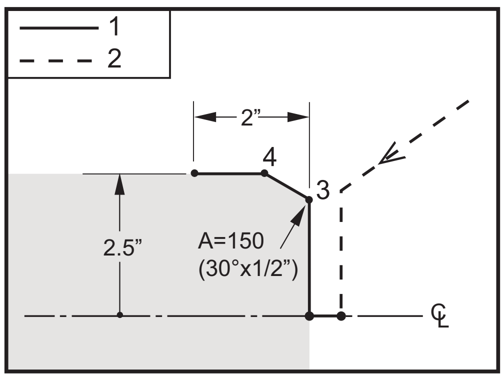

```
%
o60012 (G01 CHAMFERING WITH 'A') ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is an OD cutting tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, Spindle on CW) ;
G00 G54 X4. Z0.1 (Rapid to clear position) ;
M08 (Coolant on) ;
X0 (Rapid to center of diameter) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z0 F0.01 (Feed towards face) ;
G01 X4. (position 3) ;
X5. A150. (position 4) ;
```

```
Z-2. (Feed to back of part) ;
(BEGIN COMPLETION BLOCKS) ;
G00 X6. M09 (Rapid Retract, Coolant off) ;
G53 X0 (X home) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

NOTE: *A -30 = A150; A -45 = A135*

When specifying an angle (A), command motion in only one of the other axes (X or Z), the other axis is calculated based on the angle.

F7.7: G01 Chamfering with A: [1] Feed, [2] Rapid, [3] Start Point, [4] Finish Point.




```
%
o60012 (G01 CHAMFERING WITH 'A') ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is an OD cutting tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, Spindle on CW) ;
G00 G54 X4. Z0.1 (Rapid to clear position) ;
M08 (Coolant on) ;
X0 (Rapid to center of diameter) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z0 F0.01 (Feed towards face) ;
G01 X4. (position 3) ;
X5. A150. (position 4) ;
```

```
Z-2. (Feed to back of part) ;
(BEGIN COMPLETION BLOCKS) ;
G00 X6. M09 (Rapid Retract, Coolant off) ;
G53 X0 (X home) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

NOTE: *A -30 = A150; A -45 = A135*

G02 CW/G03 CCW Circular Interpolation Motion (Group 01)

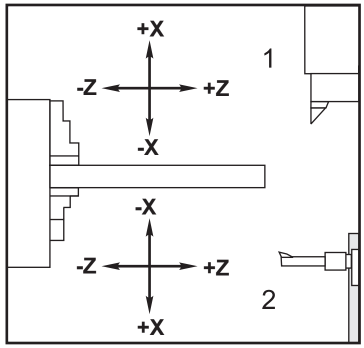
F - Feed rate
*__I__ - Distance along X-axis to center of circle
*__J__ - Distance along Y-axis to center of circle
*__K__ - Distance along Z-axis to center of circle
*__R__ - Radius of arc
*__U__ - X-axis incremental motion command
*__W__ - Z-axis incremental motion command
*__X__ - X-axis absolute motion command
*__Y__ - Y-axis absolute motion command
*__Z__ - Z-axis absolute motion command

* indicates optional

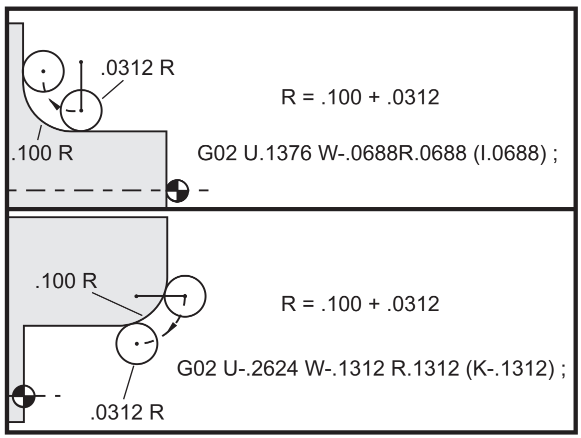
These G codes are used to specify a circular motion (CW or CCW) of the linear axes (Circular motion is possible in the X and Z axes as selected by `G18`). The `X` and `Z` values are used to specify the end point of the motion and can use either absolute (`X` and `Z`) or incremental motion (`U` and `W`). If either the `X` or `Z` is not specified, the endpoint of the arc is the same as the starting point for that axis. There are two ways to specify the center of the circular motion; the first uses `I` or `K` to specify the distance from the starting point to the center of the arc; the second uses `R` to specify the radius of the arc.

For information on `G17` and `G19` Plane Milling, see the Live Tooling section.

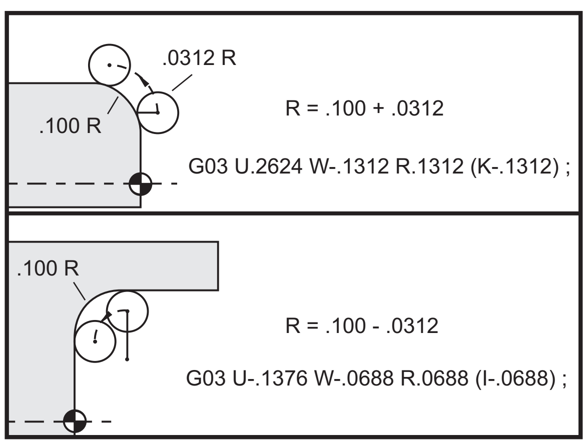
F7.8: `G02` Axis Definitions: [1] Turret Lathes, [2] Table Lathes.



F7.9: `G02` and `G03` Programs



G02



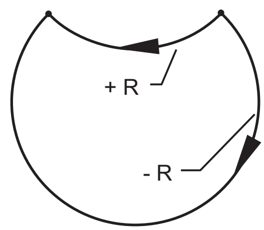
G03

R is used to specify the radius of the arc. With a positive R, the control will generate a path of 180 degrees or less; to generate a radius of over 180 degrees, specify a negative R. X or Z is required to specify an endpoint if different from the starting point.

The following lines cut an arc of less than 180 degrees:

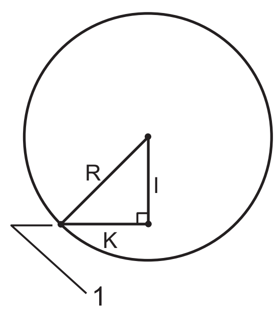
```
G01 X3.0 Z4.0 ;
G02 Z-3.0 R5.0 ;
```

F7.10: G02 Arc Using Radius



I and K are used to specify the center of the arc. When I and K are used, R may not be used. The I or K is the signed distance from the starting point to the center of the circle. If only one of I or K is specified, the other is assumed to be zero.

F7.11: G02 Defined X and Z: [1] Start.



G04 Dwell (Group 00)

P - The dwell time in seconds or milliseconds

NOTE: *The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.*

G04 specifies a delay or dwell in the program. The block with G04 delay for the time specified by the P address code. For example:

```
G04 P10.0. ;
```

Delays the program for 10 seconds.

NOTE: *G04 P10. is a dwell of 10 seconds; G04 P10 is a dwell of 10 milliseconds. Make sure you use decimal points correctly so that you specify the correct dwell time.*

G09 Exact Stop (Group 00)

G09 code is used to specify a controlled axes stop. It affects only the block in which it is commanded. It is non-modal and does not affect the blocks that come after the block where it is commanded. Machine moves decelerate to the programmed point before the control processes the next command.

G10 Set Offsets (Group 00)

G10 lets you set offsets within the program. G10 replaces manual offset entry (i.e. Tool length and diameter, and work coordinate offsets).

L - Selects offset category.

- L2 Work coordinate origin for COMMON and G54-G59
- L10 Geometry or shift offset
- L1 or L11 Tool wear
- L20 Auxiliary work coordinate origin for G110-G129

P - Selects a specific offset.

- P1-P50 - References geometry, wear or work offsets (L10-L11)
- P0 - References COMMON work coordinate offset (L2)
- P1-P6 - G54-G59 references work coordinates (L2)
- P1-P20 G110-G129 references auxiliary coordinates (L20)
- P1-P99 G154 P1-P99 reference auxiliary coordinate (L20)

Q - Imaginary tool nose tip direction
R - Tool nose radius
***U** - Incremental amount to be added to X-axis offset
***W** - Incremental amount to be added to Z-axis offset
***X** - X-axis offset
***Z** - Z-axis offset

* indicates optional

G14 Secondary Spindle Swap / G15 Cancel (Group 17)

`G14` causes the secondary spindle to become the primary spindle, so that the secondary spindle reacts to commands normally used for the main spindle. For example, `M03`, `M04`, `M05` and `M19` affect the secondary spindle, and `M143`, `M144`, `M145`, and `M119` (secondary spindle commands) cause an alarm.

NOTE: *`G50` limits the secondary spindle speed, and `G96` sets the secondary spindle surface feed value. These G-codes adjust the secondary spindle speed when there is motion in the X Axis. `G01` Feed Per Rev feeds based on the secondary spindle.*

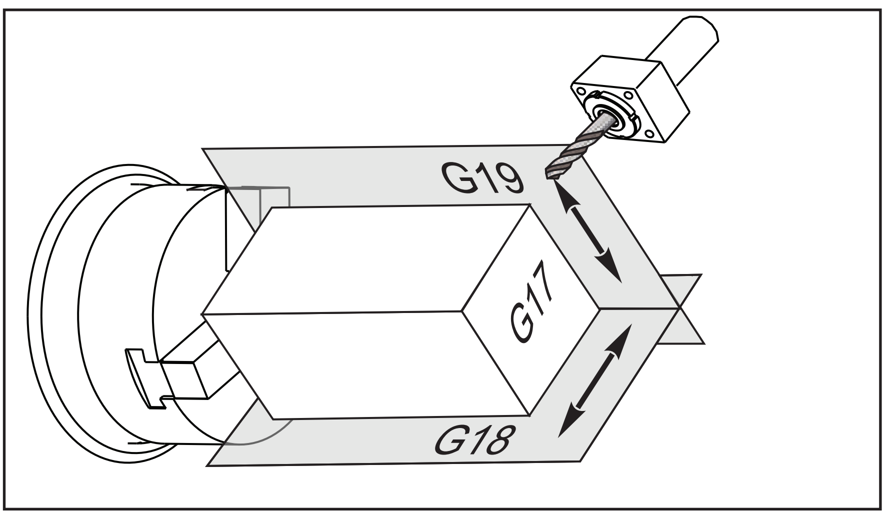
`G14` automatically activates Z-Axis mirroring. If the Z Axis is already mirrored (Setting 47 or `G101`), the mirror function is canceled.

`G14` is canceled by `G15`, `M30`, at the end of a program, or when you press **[RESET]**.

G17 XY Plane / G18 XZ Plane / G19 YZ Plane (Group 02)

This code defines the plane in which tool path motion is performed. Programming tool nose radius compensation `G41` or `G42` applies Tool Radius cutter compensation in the `G17` plane, regardless of whether `G112` is active or not. For more information, refer to Cutter Compensation in the Programming section. Plane selection codes are modal and remain in effect until another plane is selected.

F7.12: `G17`, `G18`, and `G19` Plane Selection



Program format with tool nose compensation:

```
G17 G01 X_ Y_ F_ ;
G40 G01 X_ Y_ I_ J_ F_ ;
```

G20 Select Inches / G21 Select Metric (Group 06)

Use G20 (inch) and G21 (mm) codes are to make sure that the inch/metric selection is set correctly for the program. Use Setting 9 to select between inch and metric programming. G20 in a program causes an alarm if Setting 9 is not set to inch.

G28 Return to Machine Zero Point (Group 00)

The G28 code returns all axes (X, Y, Z, B and C) simultaneously to the machine zero position when no axis is specified on the G28 line.

Alternatively, when one or more axes locations are specified on the G28 line, G28 will move to the specified locations and then to machine zero. This is called the G29 reference point; it is saved automatically for optional use in G29.

```
G28 X0 Z0 (moves to X0 Z0 in the current work coordinate system
then to machine zero) ;
G28 X1. Z1. (moves to X1. Z1. in the current work coordinate
system then to machine zero) ;
G28 U0 W0 (moves directly to machine zero because the initial
incremental move is zero) ;
G28 U-1. W-1 (moves incrementally -1. in each axis then to
machine zero) ;
```

G29 Return From Reference Point (Group 00)

G29 moves the axes to a specific position. The axes selected in this block are moved to the G29 reference point saved in G28, and then moved to the location specified in the G29 command.

G31 Feed Until Skip (Group 00)

(This G-code is optional and requires a probe.)

This G-code is used to record a probed location to a macro variable.

NOTE: *Turn on the probe before using `G31`.*

F - Feedrate in inches (mm) per minute
*__U__ - X-axis incremental motion command
*__V__ - Y-axis incremental motion command
*__W__ - Z-axis incremental motion command
X - X-axis absolute motion command
Y - Y-axis absolute motion command
Z - Z-axis absolute motion command
C - C-Axis absolute motion command

* indicates optional

This G-code moves the programmed axes while looking for a signal from the probe (skip signal). The specified move is started and continues until the position is reached or the probe receives a skip signal. If the probe receives a skip signal during the `G31` move, the control beeps and the skip signal position is recorded to macro variables. The program then executes the next line of code. If the probe does not receive a skip signal during the `G31` move, the control does not beep, the skip signal position is recorded at the end of the programmed move, and the program continues.

Macro variables `#5061` through `#5066` are designated to store skip signal positions for each axis. For more information about these skip signal variables see Macros in the Programming section of this manual.

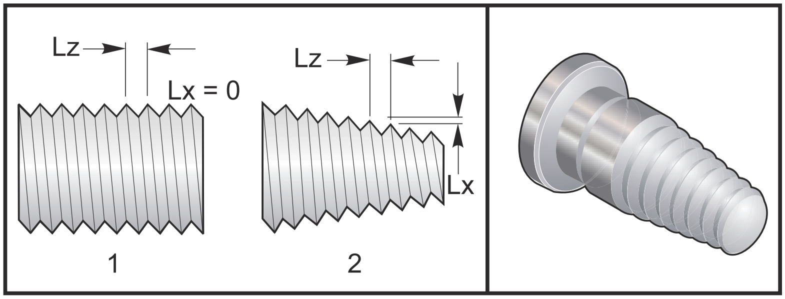
Do not use Cutter Compensation (`G41` or `G42`) with a `G31`.

G32 Thread Cutting (Group 01)

F - Feedrate in inches (mm) per minute
Q - Thread Start Angle (optional). See example on the following page.
U/W - X/Z-axis incremental positioning command. (Incremental thread depth values are user specified)
X/Z - X/Z-axis absolute positioning command. (Thread depth values are user specified)

NOTE: *Feedrate is equivalent to thread lead. Movement on at least one axis must be specified. Tapered threads have lead in both X and Z. In this case set the feedrate to the larger of the two leads. `G99` (Feed per Revolution) must be active.*

F7.13: G32 Definition of Lead (Feedrate): [1] Straight thread, [2] Tapered thread.



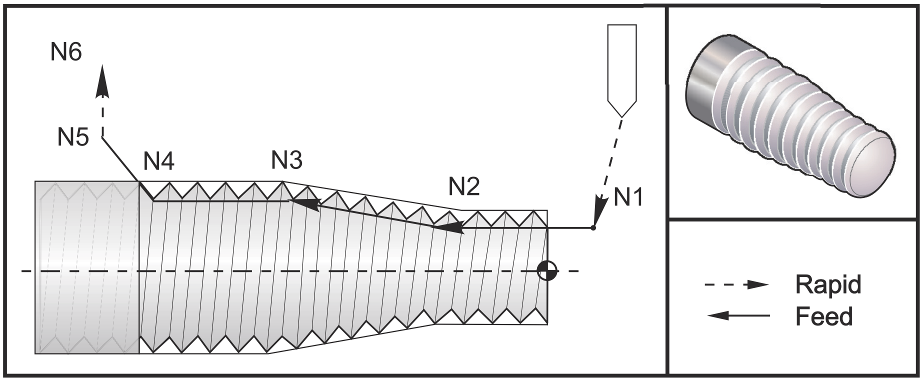
G32 differs from other thread cutting cycles in that taper and/or lead can vary continuously throughout the entire thread. In addition, no automatic position return is performed at the end of the threading operation.

At the first line of a G32 block of code, axis feed is synchronized with the rotation signal of the spindle encoder. This synchronization remains in effect for each line in a G32 sequence. It is possible to cancel G32 and recall it without losing the original synchronization. This means multiple passes will exactly follow the previous tool path. (The actual spindle RPM must be exactly the same between passes).

NOTE: *Single Block Stop and Feed Hold are deferred until last line of a G32 sequence. Feedrate Override is ignored while G32 is active, Actual Feedrate will always be 100% of programmed feedrate. M23 and M24 have no affect on a G32 operation, the user must program chamfering if needed. G32 must not be used with any G-code Canned Cycles (i.e.: G71). Do Not change spindle RPM during threading.*

CAUTION: *G32 is Modal. Always cancel G32 with another Group 01 G-code at the end of a threading operation. (Group 01 G-codes: G00, G01, G02, G03, G32, G90, G92, and G94.*

F7.14: Straight-to-Taper-to-Straight Thread Cutting Cycle



NOTE: *Example is for reference only. Multiple passes are usually required to cut actual threads.*

```
%
o60321 (G32 THREAD CUTTING WITH TAPER) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is an OD thread tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, Spindle on CW) ;
N1 G00 G54 X0.25 Z0.1 (Rapid to 1st position) ;
M08 (coolant on) ;
(BEGIN CUTTING BLOCKS) ;
N2 G32 Z-0.26 F0.065 (Straight thread, Lead = .065) ;
N3 X0.455 Z-0.585 (Blend to tapered thread) ;
N4 Z-0.9425 (Blend back to straight thread) ;
N5 X0.655 Z-1.0425 (Pull off at 45 degrees) ;
(BEGIN COMPLETION BLOCKS) ;
N6 G00 X1.2 M09 (Rapid Retract, Coolant off) ;
G53 X0 (X home) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

G40 Tool Nose Compensation Cancel (Group 07)

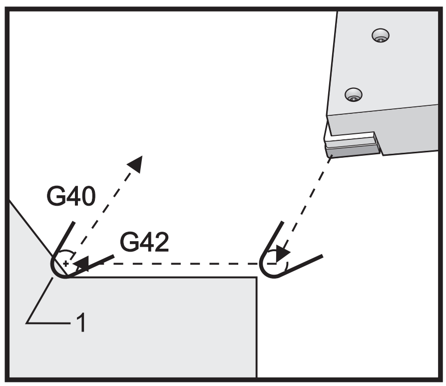
*__X__ - X Axis absolute location of departure target
*__Z__ - Z Axis absolute location of departure target
*__U__ - X Axis incremental distance to departure target
*__W__ - Z Axis incremental distance to departure target

* indicates optional

`G40` cancels `G41` or `G42`. Programming `Txx00` will also cancel tool nose compensation. Cancel tool nose compensation before the end of a program.

The tool departure usually does not correspond with a point on the part. In many cases overcutting or undercutting can occur.

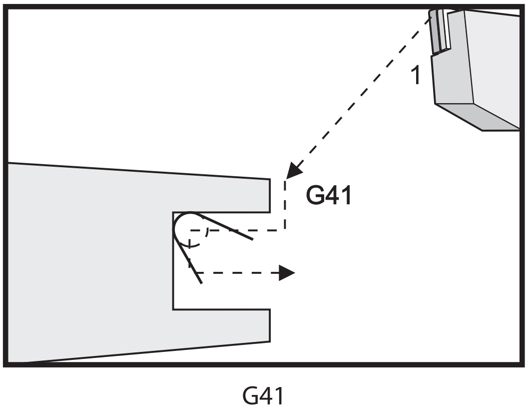
F7.15: `G40` TNC Cancel: [1] Overcut.

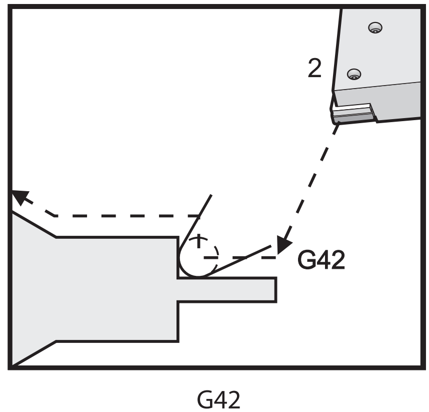


G41 Tool Nose Compensation (TNC) Left / G42 TNC Right (Group 07)

G41 or G42 will select tool nose compensation. G41 moves the tool to the left of the programmed path to compensate for the size of a tool and vice versa for G42. A tool offset must be selected with a Tnnxx code. where xx corresponds to the offsets that are to be used with the tool. For more information, see Tool Nose Compensation in the Operation section of this manual.

F7.16: G41 TNC Right and G42 TNC Left: [1] Tip = 2, [2] Tip = 3.





G50 Spindle Speed Limit

G50 can be used to limit the maximum spindle speed. The control will not allow the spindle to exceed the S address value specified in the G50 command. This is used in constant surface feed mode (G96).

This G code will also limit the secondary spindle on DS-Series machines.

```
N1G50 S3000 (Spindle rpm will not exceed 3000 rpm) ;
N2G97 M3 (Enter constant surface speed cancel, spindle on) ;
```

NOTE: *To cancel this command, use another G50 and specify the maximum spindle RPM for the machine.*

G50 Set Global Coordinate Offset FANUC (Group 00)

U - Incremental amount and direction to shift global X coordinate.
X - Absolute global coordinate shift.
W - Incremental amount and direction to shift global Z coordinate.
Z - Absolute global coordinate shift.
S - Limit spindle speed to specified value

`G50` performs several functions. It sets and shifts the global coordinate and it limits the spindle speed to a maximum value. Refer to the Global Coordinate System topic in the Programming section for a discussion of these.

To set the global coordinate, command `G50` with an `X` or `Z` value. The effective coordinate becomes the value specified in address code `X` or `Z`. Current machine location, work offsets, and tool offsets are taken into account. The global coordinate is calculated and set. For example:

```
G50 X0 Z0 (Effective coordinates are now zero) ;
```

To shift the global coordinate system, specify `G50` with a `U` or `W` value. The global coordinate system is shifted by the amount and direction specified in `U` or `W`. The current effective coordinate displayed changes by this amount in the opposite direction. This method is often used to place the part zero outside of the work cell. For example:

```
G50 W-1.0 (Effective coordinates are shifted left 1.0) ;
```

G52 Set Local Coordinate System FANUC (Group 00)

This code selects the user coordinate system.

G53 Machine Coordinate Selection (Group 00)

This code temporarily cancels work coordinates offsets and uses the machine coordinate system. This code will also ignore tool offsets.

G54-G59 Coordinate System #1 - #6 FANUC (Group 12)

`G54` - `G59` codes are user-settable coordinate systems, #1 - #6, for work offsets. All subsequent references to axes' positions are interpreted in the new coordinate system. Work coordinate system offsets are entered from the **`Active Work Offset`** display page. For additional offsets, refer to `G154` on page **362**.

G61 Exact Stop Mode (Group 15)

The `G61` code is used to specify exact stop. Rapid and interpolated moves will decelerate to an exact stop before another block is processed. In exact stop, moves will take a longer time and continuous cutter motion will not occur. This may cause deeper cutting where the tool stops.

G64 Cancels Exact Stop Mode (Group 15)

`G64` code cancels exact stop and selects the normal cutting mode.

G65 Macro Subprogram Call Option (Group 00)

`G65` is described in the Macros programming section.

G70 Finishing Cycle (Group 00)

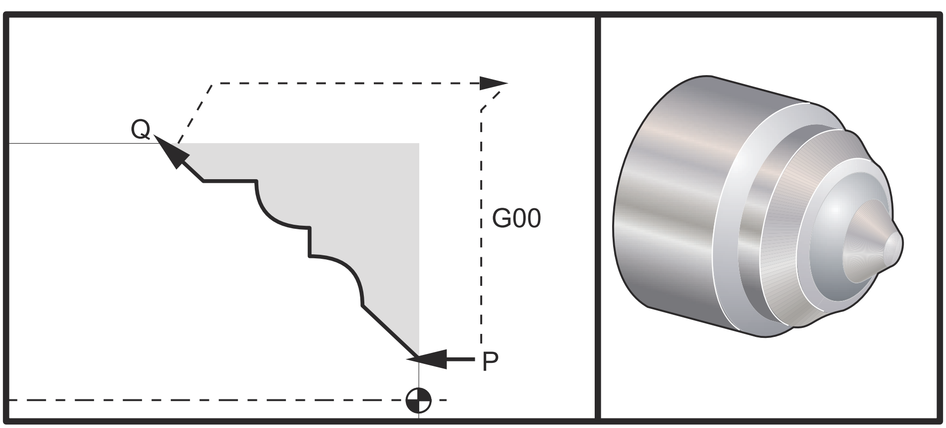
The `G70` Finishing Cycle can be used to finish cut paths that are rough cut with stock removal cycles such as `G71`, `G72` and `G73`.

P - Starting Block number of routine to execute
Q - Ending Block number of routine to execute

`G18` Z-X plane must be active

NOTE: *The P values are modal. This means if you are in the middle of a canned cycle and a `G04 Pnn` or an `M97 Pnn` is used the P value will be used for the dwell / subprogram as well as the canned cycle.*

F7.17: `G70` Finishing Cycle: [P] Starting block, [Q] Ending Block.



%

```
G71 P10 Q50 F.012 (rough out N10 to N50 the path) ;
N10 ;
F0.014 ;
... ;
N50 ;
... ;
G70 P10 Q50 (finish path defined by N10 to N50) ;
%
```

The G70 cycle is similar to a local subprogram call. However, the G70 requires that a beginning block number (P code) and an ending block number (Q code) be specified.

The G70 cycle is usually used after a G71, G72 or G73 has been performed using the blocks specified by P and Q. Any F, S, or T codes with the PQ block are effective. After execution of the Q block, a rapid (G00) is executed returning the machine to the start position that was saved before the starting of the G70. The program then returns to the block following the G70 call. A subprogram in the PQ sequence is acceptable providing that the subprogram does not contain a block with an N code matching the Q specified by the G70 call. This feature is not compatible with FANUC controls.

After a G70, the block following the G70 will be executed, not the block with an N code matching the Q code specified by the G70 call.

G71 O.D./I.D. Stock Removal Cycle (Group 00)

First Block (Only use when using two block G71 notation)
*__U__ - Depth of cut for each pass of stock removal, positive radius
*__R__ - Retract height for each pass of stock removal
Second Block
*__D__ - Depth of cut for each pass of stock removal, positive radius (Only use when using one block G71 notation)
*__F__ - Feedrate in inches (mm) per minute (G98) or per revolution (G99) to use throughout G71 PQ block
*__I__ - X-axis size and direction of G71 rough pass allowance, radius
*__K__ - Z-axis size and direction of G71 rough pass allowance
P - Starting Block number of path to rough
Q - Ending Block number of path to rough
*__S__ - Spindle speed to use throughout G71 PQ block
*__T__ - Tool and offset to use throughout G71 PQ block
*__U__ - X-axis size and direction of G71 finish allowance, diameter
*__W__ - Z-axis size and direction of G71 finish allowance

* indicates optional

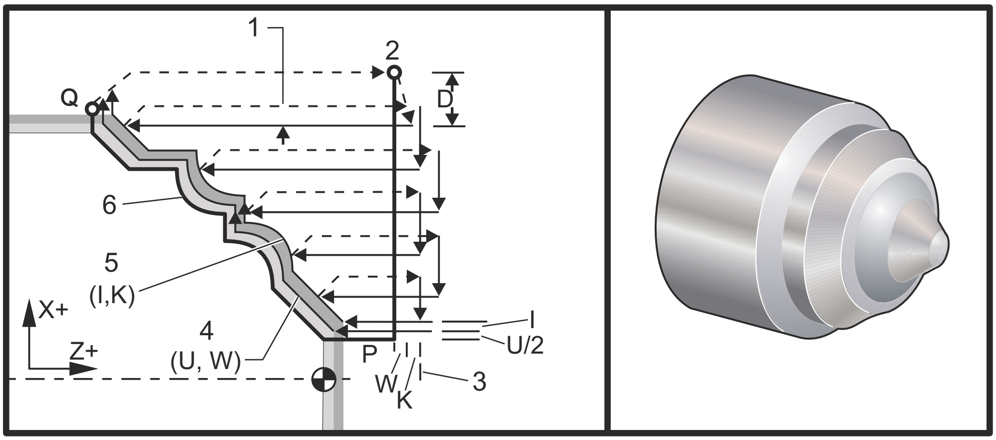
G18 Z-X plane must be active.

2 Block G71 Programming Example:

```
G71 U... R...
G71 F... I... K... P... Q... S... T... U... W...
```

NOTE: *The P values are modal. This means if you are in the middle of a canned cycle and a `G04 Pnn` or an `M97 Pnn` is used the P value will be used for the dwell / subprogram as well as the canned cycle.*

F7.18: `G71` Stock Removal: [1] Setting 287, [2] Start position, [3] Z-Axis clearance plane, [4] Finishing allowance, [5] Roughing allowance, [6] Programmed path.



This canned cycle roughs material on a part given the finished part shape. Define the shape of a part by programming the finished tool path and then use the `G71 PQ` block. Any `F`,`S` or `T` commands on the `G71` line or in effect at the time of the `G71` is used throughout the `G71` roughing cycle. Usually a `G70` call to the same `PQ` block definition is used to finish the shape.

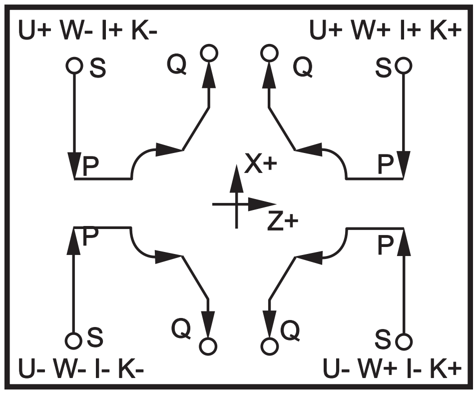
Two types of machining paths are addressed with a `G71` command. The first type of path (Type 1) is when the X-Axis of the programmed path does not change direction. The second type of path (Type 2) allows the X-Axis to change direction. For both Type 1 and Type 2, the programmed path of the Z-axis cannot change direction. If the `P` block contains only an X-Axis position, then Type 1 roughing is assumed. If the `P` block contains both an X-Axis and Z-Axis position, then Type 2 roughing is assumed.

NOTE: *The Z-Axis position given in the `P` block to specify Type 2 roughing does not have to cause axis motion. You can use the current Z-Axis position. For example, In the program example on page* ***11****, note that the `P1` block (indicated by the comment in parentheses) contains the same Z-Axis position as the start position `G00` block above.*

Any one of the four quadrants of the X-Z plane can be cut by specifying address codes D, I, K, U, and W properly.

In the figures, the start position S is the position of the tool at the time of the G71 call. The Z clearance plane [3] is derived from the Z-axis start position and the sum of W and optional K finish allowance.

F7.19: G71 Address Relationships



Type I Details

When Type I is specified by the programmer it is assumed that the X-axis tool path does not reverse during a cut. Each roughing pass X-axis location is determined by applying the value specified in D to the current X location. The nature of the movement along the Z clearance plane for each roughing pass is determined by the G code in block P. If block P contains a G00 code, then movement along the Z clearance plane is a rapid mode. If block P contains a G01 then movement will be at the G71 feed rate.

Each roughing pass is stopped before it intersects the programmed tool path allowing for both roughing and finishing allowances. The tool is then retracted from the material, at a 45 degree angle. The tool then moves in rapid mode to the Z-axis clearance plane.

When roughing is completed the tool is moved along the tool path to clean up the rough cut. If I and K are specified an additional rough finish cut parallel to the tool path is performed.

Type II Details

When Type II is specified by the programmer the X axis PQ path is allowed to vary (for example, the X-axis tool path can reverse direction).

The X axis PQ path must not exceed the original starting location. The only exception is the ending Q block.

Type II, must have a reference move, in both the X and Z axis, in the block specified by P.

Roughing is similar to Type I except after each pass along the Z axis, the tool will follow the path defined by PQ. The tool will then retract parallel to the X axis. The Type II roughing method does not leave steps in the part prior to finish cutting and typically results in a better finish.

G72 End Face Stock Removal Cycle (Group 00)

First Block (Only use when using two block G72 notation)
*__W__ - Depth of cut for each pass of stock removal, positive radius
*__R__ - Retract height for each pass of stock removal
Second Block
*__D__ - Depth of cut for each pass of stock removal, positive radius (Only use when using one block G72 notation)
*__F__ - Feedrate in inches (mm) per minute (G98) or per revolution (G99) to use throughout G71 PQ block
*__I__ - X-axis size and direction of G72 rough pass allowance, radius
*__K__ - Z-axis size and direction of G72 rough pass allowance
P - Starting Block number of path to rough
Q - Ending Block number of path to rough
*__S__ - Spindle speed to use throughout G72 PQ block
*__T__ - Tool and offset to use throughout G72 PQ block
*__U__ - X-axis size and direction of G72 finish allowance, diameter
*__W__ - Z-axis size and direction of G72 finish allowance

*indicates optional

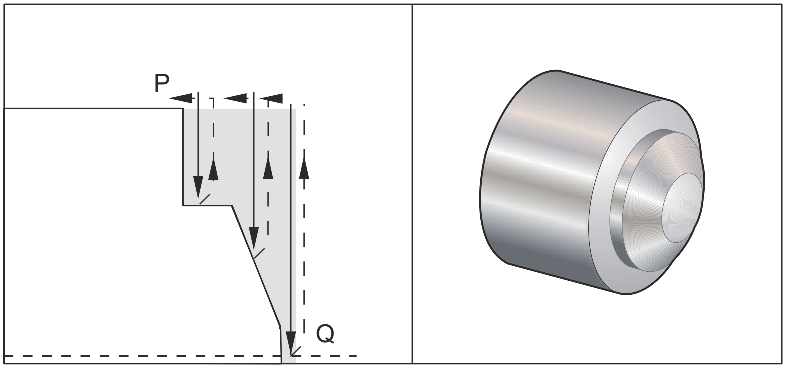
G18 Z-X plane must be active.

2 Block G72 Programming Example:

```
G72 W... R...
G72 F... I... K... P... Q... S... T... U... W...
```

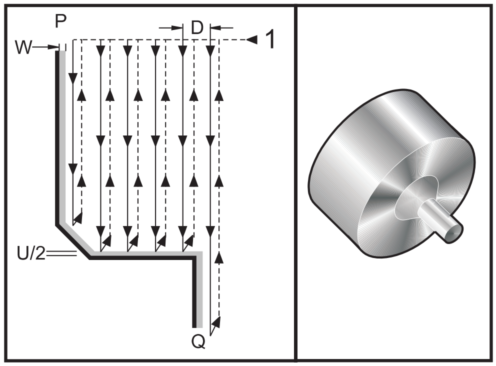
NOTE: *The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.*

F7.20: `G72` Basic G Code Example: [P] Starting block, [1] Start position, [Q] Ending block.




```
%
O60721 (G72 END FACE STOCK REMOVAL EX 1) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is an end face cutting tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS, spindle on CW) ;
G00 G54 X6. Z0.1 (Rapid to clear position) ;
M08 (Coolant on) ;
G96 S200 (CSS on) ;
(BEGIN CUTTING BLOCKS) ;
G72 P1 Q2 D0.075 U0.01 W0.005 F0.012 (Begin G72) ;
N1 G00 Z-0.65 (P1 - Begin toolpath);
G01 X3. F0.006 (1st position);
Z-0.3633 (Face Stock Removal);
X1.7544 Z0. (Face Stock Removal) ;
X-0.0624 ;
N2 G00 Z0.02 (Q2 - End toolpath);
G70 P1 Q2 (Finish Pass) ;
(BEGIN COMPLETION BLOCKS) ;
G97 S500 (CSS off) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

F7.21: `G72` Tool Path: [P] Starting block, [1] Start position, [Q] Ending block.



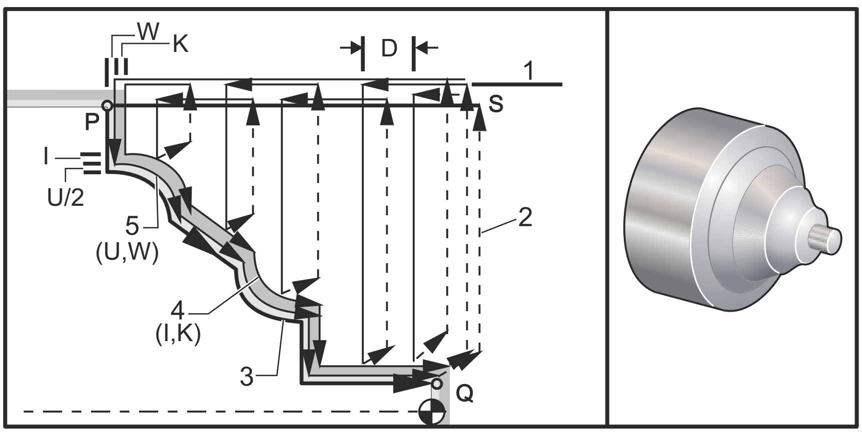

```
%
O60722(G72 END FACE STOCK REMOVAL EX 2) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is an end face cutting tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS, spindle on CW) ;
G00 G54 X4.05 Z0.2 (Rapid to 1st position) ;
M08 (Coolant on) ;
G96 S200 (CSS on) ;
(BEGIN CUTTING BLOCKS) ;
G72 P1 Q2 U0.03 W0.03 D0.2 F0.01 (Begin G72);
N1 G00 Z-1.(P1 - Begin toolpath) ;
G01 X1.5 (Linear feed) ;
X1. Z-0.75 (Linear feed) ;
G01 Z0 (Linear feed) ;
N2 X0(Q2 - End of toolpath) ;
G70 P1 Q2 (Finishing cycle) ;
(BEGIN COMPLETION BLOCKS) ;
G97 S500 (CSS off) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

This canned cycle removes material on a part given the finished part shape. It is similar to G71 but removes material along the face of a part. Define the shape of a part by programming the finished tool path and then use the G72 PQ block. Any F,S or T commands on the G72 line or in effect at the time of the G72 is used throughout the G72 roughing cycle. Usually a G70 call to the same PQ block definition is used to finish the shape.

Two types of machining paths are addressed with a G72 command.

- The first type of path (Type 1) is when the Z Axis of the programmed path does not change direction. The second type of path (Type 2) allows the Z Axis to change direction. For both the first type and the second type of programmed path the X Axis cannot change direction. If Setting 33 is set to FANUC, Type 1 is selected by having only an X-axis motion in the block specified by P in the G72 call.
- When both an X-axis and Z-axis motion are in the P block then Type 2 roughing is assumed.

F7.22: G72 End Face Stock Removal Cycle: [P] Starting block, [1] X-Axis clearance plane, [2] G00 block in P, [3] Programmed path, [4] Roughing allowance, [5] Finishing allowance.

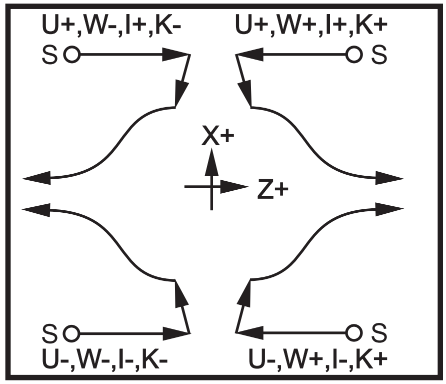


The G72 consists of a roughing phase and a finishing phase. The roughing and finishing phase are handled differently for Type 1 and Type 2. Generally the roughing phase consists of repeated passes along the X-axis at the specified feed rate. The finishing phase consists of a pass along the programmed tool path to remove excess material left by the roughing phase while leaving material for a G70 finishing cycle. The final motion in either type is a return to the starting position S.

In the previous figure the start position S is the position of the tool at the time of the G72 call. The X clearance plane is derived from the X-axis start position and the sum of U and optional I finish allowances.

Any one of the four quadrants of the X-Z plane can be cut by specifying address codes I, K, U, and W properly. The following figure indicates the proper signs for these address codes to obtain the desired performance in the associated quadrants.

F7.23: G72 Address Relationships



G73 Irregular Path Stock Removal Cycle (Group 00)

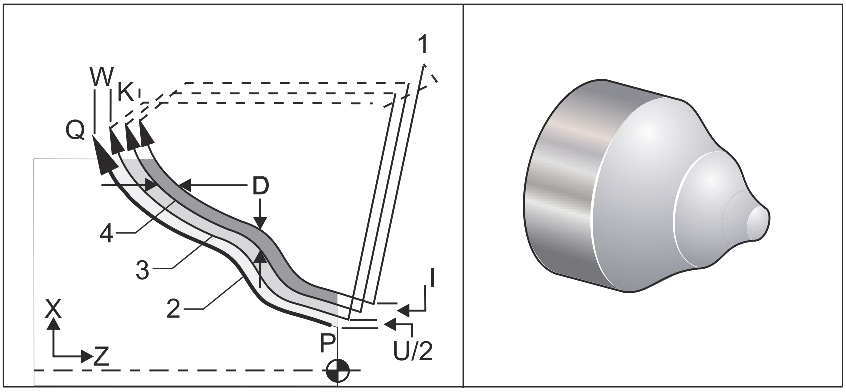
D - Number of cutting passes, positive integer
“**F** - Feedrate in inches (mm) per minute (G98) or per revolution (G99) to use throughout G73 PQ block
I - X-axis distance and direction from first cut to last, radius
K - Z-axis distance and direction from first cut to last
P - Starting Block number of path to rough
Q - Ending Block number of path to rough
*__S__ - Spindle speed to use throughout G73 PQ block
*__T__ - Tool and offset to use throughout G73 PQ block
*__U__ - X-axis size and direction of G73 finish allowance, diameter
*__W__ - Z-axis size and direction of G73 finish allowance

* indicates optional

G18 Z-X plane must be active

NOTE: *The P values are modal. This means if you are in the middle of a canned cycle and a G04 Pnn or an M97 Pnn is used the P value will be used for the dwell / subprogram as well as the canned cycle.*

F7.24: G73 Irregular Path Stock Removal: [P] Starting block, [Q] Ending block [1] Start position, [2] Programmed path, [3] Finish allowance, [4] Roughing allowance.



The G73 canned cycle can be used for rough cutting of preformed material such as castings. The canned cycle assumes that material has been relieved or is missing a certain known distance from the programmed tool path PQ.

Machining starts from the current position (S), and either rapids or feeds to the first rough cut. The nature of the approach move is based on whether a G00 or G01 is programmed in block P. Machining continues parallel to the programmed tool path. When block Q is reached a rapid departure move is executed to the Start position plus the offset for the second roughing pass. Roughing passes continue in this manner for the number of rough passes specified in D. After the last rough is completed, the tool returns to the starting position S.

Only F, S and T prior to or in the G73 block are in effect. Any feed (F), spindle speed (S) or tool change (T) codes on the lines from P to Q are ignored.

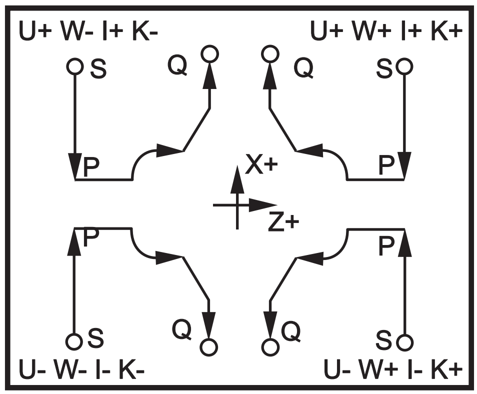
The offset of the first rough cut is determined by (U/2 + I) for the X Axis, and by (W + K) for the Z Axis. Each successive roughing pass moves incrementally closer to the final roughing finish pass by an amount of (I/(D- 1)) in the X Axis, and by an amount of (K/(D-1)) in the Z Axis. The last rough cut always leaves finish material allowance specified by U/2 for the X Axis and W for the Z Axis. This canned cycle is intended for use with the G70 finishing canned cycle.

The programmed tool path PQ does not have to be monotonic in X or Z, but care has to be taken to insure that existing material does not interfere with tool movement during approach and departure moves.

NOTE: *Monotonic curves are curves that tend to move in only one direction as x increases. A monotonic increasing curve always increases as x increases, i.e. f(a)>f(b) for all a>b. A monotonic decreasing curve always decreases as x increases, i.e. f(a)<f(b) for all a>b. The same sort of restrictions are also made for the monotonic non-decreasing and monotonic non-increasing curves.*

The value of `D` must be a positive integral number. If the `D` value includes a decimal, an alarm is generated. The four quadrants of the `ZX` plane can be machined if the following signs for `U`, `I`, `W`, and `K` are used.

F7.25: `G71` Address Relationships

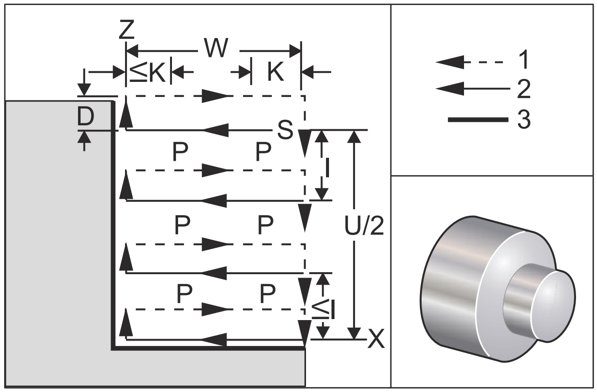


G74 End Face Grooving Cycle (Group 00)

* **D** - Tool clearance when returning to starting plane, positive radius
* **F** - Feed rate
* **I** - X-axis size of increment between peck cycles, positive radius
K - Z-axis size of increment between pecks in a cycle
* **U** - X-axis incremental distance away from current X position before returning to the start plane.
W - Z-axis incremental distance to total pecking depth
X - X-axis absolute location of furthest peck cycle (diameter)
Z - Z-axis absolute location total pecking depth

*indicates optional

F7.26: `G74` End Face Grooving Cycle Peck Drilling: [1] Rapid, [2] Feed, [3] Programmed Path, [S] Start position, [P] Peck retraction (Setting 22).



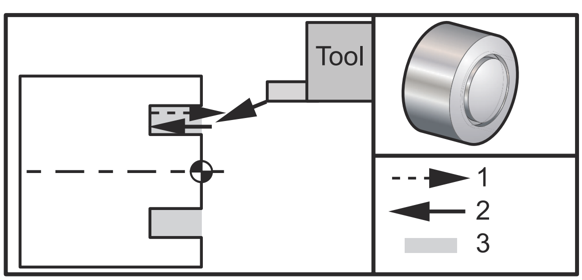
The G74 canned cycle is used for grooving on the face of a part, peck drilling, or turning.

***Warning: The D code command is rarely used and should only be used if the wall on the outside of the groove does not exist like the figure above. The D code can be used in grooving and turning to provide a tool clearance shift, in the X axis, before returning in the Z axis to the "C" clearance point. But, if both sides to the groove exist during the shift, then the groove tool would break. So you wouldn't want to use the D command.

A minimum of two pecking cycles occur, if an X, or U, code is added to a G74 block and X is not the current position. One at the current location and then at the X location. The I code is the incremental distance between X-Axis pecking cycles. Adding an I performs multiple pecking cycles between the starting position S and X. If the distance between S and X is not evenly divisible by I then the last interval is less than I.

When K is added to a G74 block, pecking is performed at each interval specified by K, the peck is a rapid move opposite the direction of feed with a distance defined by Setting 22. The D code can be used for grooving and turning to provide material clearance when returning to starting plane S.

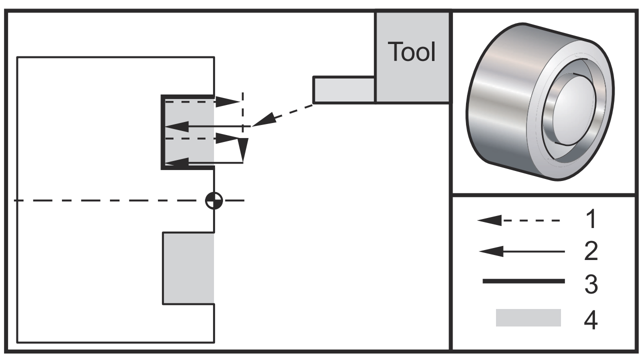
F7.27: `G74` End Face Grooving Cycle: [1] Rapid, [2] Feed, [3] Groove.




```
%
O60741 (G74 END FACE) ;
(G54 X0 is at the center of rotation) ;
```

```
(Z0 is on the face of the part) ;
(T1 is an end face cutting tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, Spindle on CW) ;
G00 G54 X3. Z0.1 (Rapid to 1st position) ;
M08 (Coolant on) ;
G96 S200 (CSS on) ;
(BEGIN CUTTING BLOCKS) ;
G74 Z-0.5 K0.1 F0.01 (Begin G74) ;
(BEGIN COMPLETION BLOCKS) ;
G97 S500 (CSS off) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

F7.28: `G74` End Face Grooving Cycle (Multiple Pass): [1] Rapid, [2] Feed, [3] Programmed path, [4] Groove.




```
%
O60742 (G74 END FACE MULTI PASS) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is an end face cutting tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, spindle on CW) ;
G00 G54 X3. Z0.1 (Rapid to 1st position) ;
```

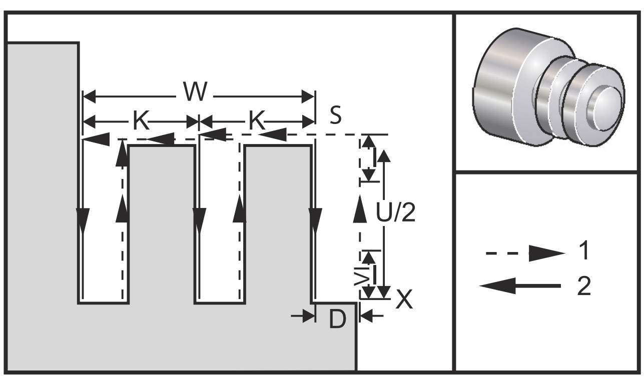
```
M08 (Coolant on) ;
G96 S200 (CSS on) ;
(BEGIN CUTTING BLOCKS) ;
G74 X1.75 Z-0.5 I0.2 K0.1 F0.01 (Begin G74) ;
(BEGIN COMPLETION BLOCKS) ;
G97 S500 (CSS off) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

G75 O.D./I.D. Grooving Cycle (Group 00)

*__D__ - Tool clearance when returning to starting plane, positive
*__F__ - Feed rate
*__I__ - X-axis size of increment between pecks in a cycle (radius measure)
*__K__ - Z-axis size of increment between peck cycles
*__U__ - X-axis incremental distance to total pecking depth
W - Z-axis incremental distance to furthest peck cycle
X - X-axis absolute location total pecking depth (diameter)
Z - Z-axis absolute location to furthest peck cycle

* indicates optional

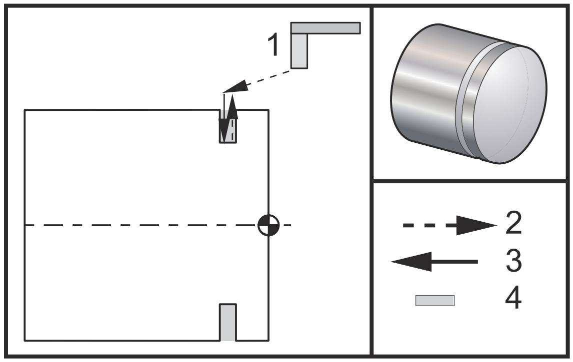
F7.29: `G75` O.D./I.D. Grooving Cycle: [1] Rapid, [2] Feed, [S] Start position.



The `G75` canned cycle can be used for grooving an outside diameter. When a `Z`, or `W`, code is added to a `G75` block and `Z` is not the current position, then a minimum of two pecking cycles occur. One at the current location and another at the `Z` location. The `K` code is the incremental distance between `Z` axis pecking cycles. Adding a `K` performs multiple, evenly spaced, grooves. If the distance between the starting position and the total depth (`Z`) is not evenly divisible by `K` then the last interval along `Z` is less than `K`.

NOTE: *Chip clearance is defined by Setting 22.*

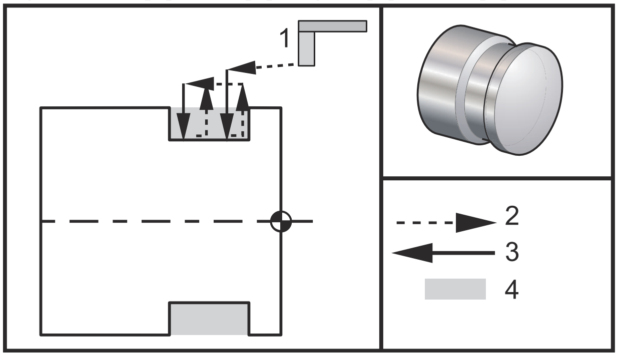
F7.30: `G75` O.D. Single Pass




```
%
O60751 (G75 OD GROOVE CYCLE) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is an OD groove tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, spindle on CW) ;
G00 G54 X4.1 Z0.1 (Rapid to 1st position) ;
M08 (Coolant on) ;
G96 S200 (CSS on) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z-0.75 F0.05 (Feed to Groove location) ;
G75 X3.25 I0.1 F0.01 (Begin G75) ;
(BEGIN COMPLETION BLOCKS) ;
G97 S500 (CSS off) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

The following program is an example of a `G75` program (Multiple Pass):

F7.31: `G75` O.D. Multiple Pass: [1] Tool, [2] Rapid, [3] Feed, [4] Groove.




```
%
O60752 (G75 OD GROOVE CYCLE 2) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is an OD groove tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, spindle on CW) ;
G00 G54 X4.1 Z0.1 (Rapid to 1st position) ;
M08 (Coolant on) ;
G96 S200 (CSS on) ;
(BEGIN CUTTING BLOCKS) ;
G01 Z-0.75 F0.05 (Feed to Groove location) ;
G75 X3.25 Z-1.75 I0.1 K0.2 F0.01 (Begin G75) ;
(BEGIN COMPLETION BLOCKS) ;
G97 S500 (CSS off) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

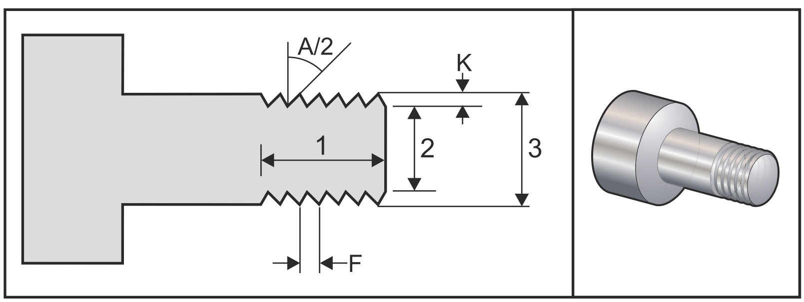
G76 Threading Cycle, Multiple Pass (Group 00)

***A** - Tool nose angle (value: 0 to 120 degrees) Do not use a decimal point
D - First pass cutting depth
F(E) - Feed rate, the lead of the thread
***I** - Thread taper amount, radius measure
K - Thread height, defines thread depth, radius measure
***P** - Single Edge Cutting (load constant)
***Q** - Thread Start Angle (Do not use a decimal point)
***U** - X-axis incremental distance, start to maximum thread Depth Diameter
***W** - Z-axis incremental distance, start to maximum thread length
***X** - X-axis absolute location, maximum thread Depth Diameter
***Z** - Z-axis absolute location, maximum thread length

* indicates optional

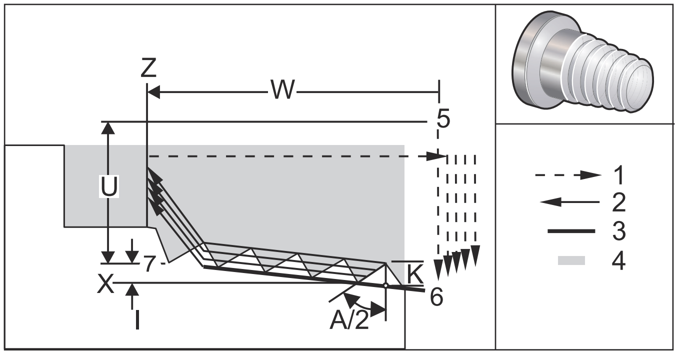
NOTE: *The P values are modal. This means if you are in the middle of a canned cycle and a `G04 Pnn` or an `M97 Pnn` is used the P value will be used for the dwell / subprogram as well as the canned cycle.*

F7.32: `G76` Threading Cycle, Multiple Pass: [1] Z depth, [2] Minor diameter, [3] Major diameter.



Setting 95/Setting 96 determine chamfer size/angle; `M23`/`M24` turn chamfering **ON**/**OFF**.

F7.33: G76 Threading Cycle, Multiple Pass Tapered: [1] Rapid, [2] Feed, [3] Programmed path, [4] Cut allowance, [5] Start position, [6] Finished diameter, [7] Target, [A] Angle.



The G76 canned cycle can be used for threading both straight or tapered (pipe) threads.

The height of the thread is defined as the distance from the crest of the thread to the root of the thread. The calculated depth of thread (K) is the value of K less the finish allowance (Setting 86, Thread Finish Allowance).

The thread taper amount is specified in I. Thread taper is measured from the target position X, Z at point [7] to position [6]. The **I** value is the difference in radial distance from the start to the end of the thread, not an angle.

NOTE: *A conventional O.D. taper thread will have a negative I value.*

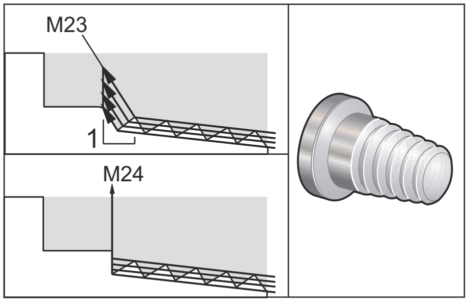
The depth of the first cut through the thread is specified in D. The depth of the last cut through the thread can be controlled with Setting 86.

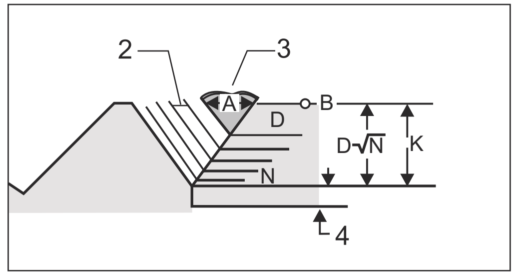
The tool nose angle for the thread is specified in A. The value can range from 0 to 120 degrees. If A is not used, 0 degrees is assumed. To reduce chatter while threading use A59 when cutting a 60 degree included thread.

The F code specifies the feed rate for threading. It is always good programming practice to specify G99 (feed per revolution) prior to a threading canned cycle. The F code also indicates the thread pitch or lead.

At the end of the thread an optional chamfer is performed. The size and angle of the chamfer is controlled with Setting 95 (Thread Chamfer Size) and Setting 96 (Thread Chamfer Angle). The chamfer size is designated in number of threads, so that if 1.000 is recorded in Setting 95 and the feed rate is .05, then the chamfer will be .05. A chamfer can improve the appearance and functionality of threads that must be machined up to a shoulder. If relief is provided for at the end of the thread then the chamfer can be eliminated by specifying 0.000 for the chamfer size in Setting 95, or using M24. The default value for Setting 95 is 1.000 and the default angle for the thread (Setting 96) is 45 degrees.

F7.34: G76 Using an A Value: [1] Setting 95 and 96 (see Note), [2] Setting 99 (Thread Minimum Cut), [3] Cutting Tip, [4] Setting 86 - Finish Allowance.





NOTE: *Setting 95 and 96 will affect the final chamfer size and angle.*

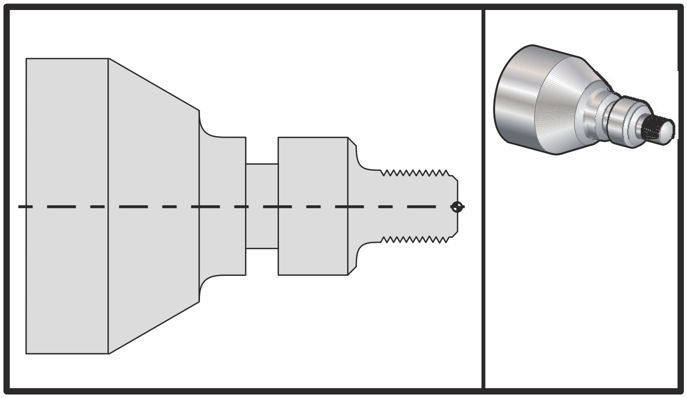
Four options for G76 Multiple Thread Cutting are available:

1. P1:Single edge cutting, cutting amount constant
2. P2:Double edge cutting, cutting amount constant
3. P3: Single edge cutting, cutting depth constant
4. P4: Double edge cutting, cutting depth constant

P1 and P3 both allow for single edge threading, but the difference is that with P3 a constant depth cut is done with every pass. Similarly, P2 and P4 options allow for double edge cutting with P4 giving constant depth cut with every pass. Based on industry experience, double edge cutting option P2 may give superior threading results.

D specifies the depth of the first cut. Each successive cut is determined by the equation D*sqrt(N) where N is the Nth pass along the thread. The leading edge of the cutter does all of the cutting. To calculate the X position of each pass you have to take the sum of all the previous passes, measured from the start point the X value of each pass

F7.35: G76 Thread Cutting Cycle, Multiple Pass



```
%
o60761 (G76 THREAD CUTTING MULTIPLE PASSES) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is an OD thread tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, Spindle on CW) ;
G00 G54 X1.2 Z0.3 (Rapid to 1st position) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G76 X0.913 Z-0.85 K0.042 D0.0115 F0.0714 (Begin G76) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

G80 Canned Cycle Cancel (Group 09)

G80 cancels all active canned cycles.

NOTE: *G00 or G01 also cancel canned cycles.*

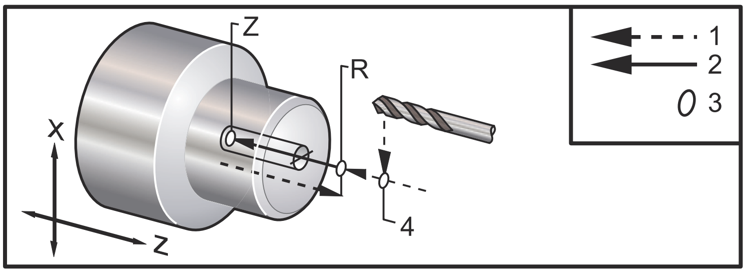
G81 Drill Canned Cycle (Group 09)

*__C__ - C-Axis absolute motion command (optional)
F - Feed Rate
*__L__ - Number of repeats
R - Position of the R plane
*__X__ - X-axis motion command
*__Y__ - Y-axis absolute motion command
Z - Position of bottom of hole

* indicated optional

Also see `G241` for radial drilling and `G195`/`G196` for radial tapping with live tooling.

F7.36: `G81` Drill Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting plane, [R] R plane, [Z] Position at the bottom of the hole.



G82 Spot Drill Canned Cycle (Group 09)

*__C__ - C-Axis absolute motion command (optional)
F - Feed Rate in inches (mm) per minute
*__L__ - Number of repeats
P - The dwell time at the bottom of the hole
R - Position of the R plane
*__X__ - X-axis motion command
*__Y__ - Y-axis motion command
Z - Position of bottom of hole

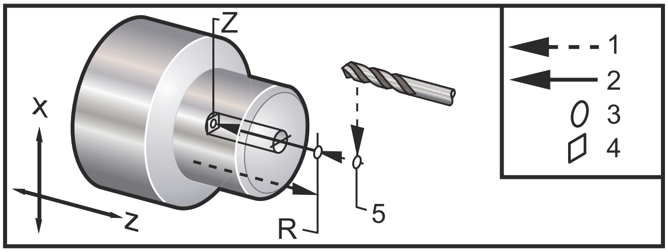
* indicates optional

This G code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. Once activated, every motion of X will cause this canned cycle to be executed.

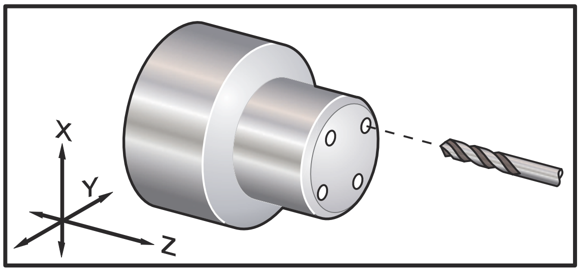
Also, see `G242` for radial live tool spot drilling.

NOTE: *The P values are modal. This means if you are in the middle of a canned cycle and a `G04 Pnn` or an `M97 Pnn` is used the P value will be used for the dwell / subprogram as well as the canned cycle.*

F7.37: `G82` Spot Drill Canned Cycle:[1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Dwell, [5] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.



F7.38: `G82` Y-Axis Drill




```
%
o60821 (G82 LIVE SPOT DRILL CYCLE) ;
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is a spot drill) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G98 (Feed per min) ;
G00 G54 X1.5 C0. Z1. (Rapid to 1st position) ;
P1500 M133 (Live tool CW at 1500 RPM) ;
M08 (coolant on) ;
(BEGIN CUTTING CYCLE) ;
G82 C45. Z-0.25 F10. P80 (Begin G82) ;
C135. (2nd position) ;
C225. (3rd position) ;
C315. (4th position) ;
(BEGIN COMPLETION BLOCKS) ;
M135 (Live tool off) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 (Z home) ;
M30 (End program) ;
%
```

To calculate how long you should dwell at the bottom of your spot drill cycle, use the following formula:

P = Dwell Revolutions x 60000/RPM

If you want the tool to dwell for two full revolutions at its full Z depth in the program above (running at 1500 RPM), you would calculate:

2 x 60000 / 1500 = 80

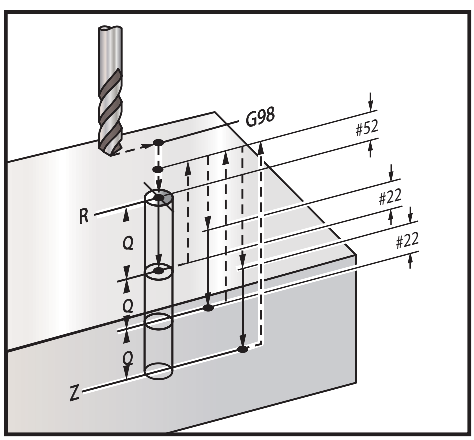
Enter `P80` (80 milliseconds or P.08 (.08 seconds) on the `G82` line, to dwell for 2 revolutions at 1500 RPM.

G83 Normal Peck Drilling Canned Cycle (Group 09)

*__C__ - C-Axis absolute motion command (optional)
F - Feed Rate in inches (mm) per minute
*__I__ - Size of first cutting depth
*__J__ - Amount to reduce cutting depth each pass
*__K__ - Minimum depth of cut
*__L__ - Number of repeats
*__P__ - The dwell time at the bottom of the hole
*__Q__ - The cut-in value, always incremental
*__R__ - Position of the R plane
*__X__ - X-axis motion command
*__Y__ - Y-axis motion command
Z - Position of bottom of hole

* indicates optional

F7.39: `G83` Peck Drilling Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Dwell, [#22] Setting 22, [#52] Setting 52.



NOTE: *If `I`, `J`, and `K` are specified, a different operating mode is selected. The first pass will cut in the value of `I`, each succeeding cut will be reduced by amount `J`, and the minimum cutting depth is `K`. Do not use a `Q` value when programming with `I`,`J`, and `K`.*

Setting 52 changes the way `G83` works when it returns to the `R` plane. Usually the `R` plane is set well outside the cut to insure that the chip clearing motion allows the chips to clear the hole. However, this is wasted motion when first drilling through this empty space. If Setting 52 is set to the distance required to clear chips, the `R` plane can be put much closer to the part being drilled. When the clear move to `R` occurs, the `Z` will be moved past `R` by this value in Setting 52. Setting 22 is the amount to feed in `Z` to get back to the same point at which the retraction occurred.

```
%
o60831 (G83 NORMAL PECK DRILLING) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is a drill) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, spindle on CW) ;
G00 G54 X0 Z0.25 (Rapid to 1st position) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G83 Z-1.5 F0.005 Q0.25 R0.1 (Begin G83)
(BEGIN COMPLETION BLOCKS)
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 ;
%
```

```
%
(LIVE PECK DRILL - AXIAL) ;
T1111 ;
G98 ;
G00 G54 X6. C0. Y0. Z1. ;
G00 X1.5 Z0.25 ;
G97 P1500 M133 ;
M08 ;
G83 G98 C45. Z-0.8627 F10. Q0.125 ;
C135. ;
```

```
C225. ;
C315. ;
G00 G80 Z0.25 ;
M135 ;
M09 ;
G28 H0. (Unwind C-Axis) ;
G00 G54 X6. Y0. Z1. ;
G18 ;
G99 ;
M01 ;
M30 ;
%
```

G84 Tapping Canned Cycle (Group 09)

F - Feed Rate
* **R** - Position of the R plane
S - RPM, called prior to G84
* **X** - X-axis motion command
Z - Position of bottom of hole

* indicates optional

Programming Notes:

- It is not necessary to start the spindle CW before this canned cycle. The control does this automatically.
- When `G84` tapping on a lathe, it is simplest to use `G99` Feed Per Revolution.
- The Lead is the distance traveled along a screw's axis, with each full revolution.
- The feedrate, when using `G99`, is equal to the Lead of the tap.
- An `S` value must be called prior to the `G84`. The `S` value determines the RPM of the tapping cycle.
- In Metric Mode (`G99`, with Setting 9 = **`MM`**), the feedrate is the metric equivalent of the lead, in **`MM`**.
- In Inch Mode (`G99`, with Setting 9 = **`INCH`**), the feedrate is the Inch equivalent of the lead, in inches.
- The lead (and `G99` feedrate) of an M10 x 1.0mm tap is 1.0mm, or .03937" (1.0/25.4=.03937).

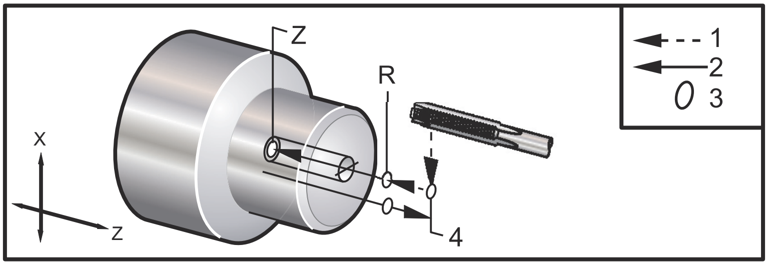
Examples:

1. The lead of a 5/16-18 tap is 1.411 mm (1/18*25.4 = 1.411), or .0556" (1/18 = .0556)
2. This canned cycle can be used on the secondary spindle of a Dual Spindle DS lathe, when prefaced by a `G14`.
   Refer to the `G14` Secondary Spindle Swap on page **315** for more information.
3. For Axial Live-Tool tapping, use a `G95` or `G186` command.

4. For Radial Live-Tool tapping, use a `G195` or `G196` command.
5. For Reverse Tapping (left-hand thread) on the Main or Secondary Spindle, refer to page **371**.

More programming examples, in both Inch and Metric, are shown below:

F7.40: `G84` Tapping Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting plane, [R] R plane, [Z] Position at the bottom of the hole.




```
%
o60841 (IMPERIAL TAP, SETTING 9 = MM) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part)
(T1 is a 1/4-20 Tap) ;
G21 (ALARM if setting 9 is not MM) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G40 G80 G99 (Safe startup) ;
G00 G54 X0 Z12.7 (Rapid to 1st position) ;
M08 (Coolant on) ;
S800 (RPM OF TAP CYCLE) ;
(BEGIN CUTTING BLOCK) ;
G84 Z-12.7 R12.7 F1.27 (1/20*25.4 = 1.27) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

```
%
o60842 (METRIC TAP, SETTING 9 = MM) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part)
(T1 is an M8 x 1.25 Tap) ;
G21 (ALARM if setting 9 is not MM) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
```

```
G00 G18 G40 G80 G99 (Safe startup) ;
G00 G54 X0 Z12.7 (Rapid to 1st position) ;
M08 (Coolant on) ;
S800 (RPM OF TAP CYCLE) ;
(BEGIN CUTTING BLOCK) ;
G84 Z-12.7 R12.7 F1.25 (Lead = 1.25) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

```
%
o60843 (IMPERIAL TAP, SETTING 9 = IN) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part)
(T1 is a 1/4-20 Tap) ;
G20 (ALARM if setting 9 is not INCH) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G00 G54 X0 Z0.5 (Rapid to 1st position) ;
M08 (Coolant on) ;
S800 (RPM OF TAP CYCLE) ;
(BEGIN CUTTING BLOCK) ;
G84 Z-0.5 R0.5 F0.05 (Begin G84) ;
(1/20 = .05) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

```
%
o60844 (METRIC TAP, SETTING 9 = IN) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part)
(T1 is an M8 x 1.25 Tap) ;
G20 (ALARM if setting 9 is not INCH) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G00 G54 X0 Z0.5 (Rapid to 1st position) ;
M08 (Coolant on) ;
S800 (RPM OF TAP CYCLE) ;
```

```
(BEGIN CUTTING BLOCK) ;
G84 Z-0.5 R0.5 F0.0492 (1.25/25.4 = .0492) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

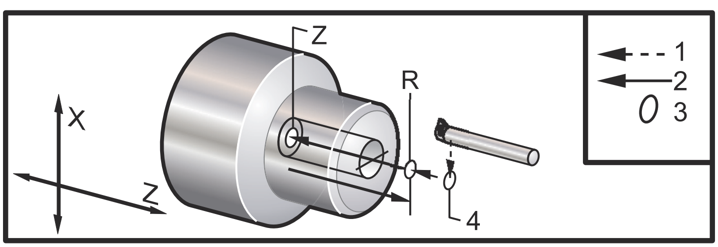
G85 Boring Canned Cycle (Group 09)

NOTE: *This cycle feeds in and feeds out.*

F - Feed Rate
*__L__ - Number of repeats
*__R__ - Position of the R plane
*__X__ - X-axis motion command
*__Y__ - Y-axis motion command
Z - Position of bottom of hole

* indicates optional

F7.41: G85 Boring Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.



G86 Bore and Stop Canned Cycle (Group 09)

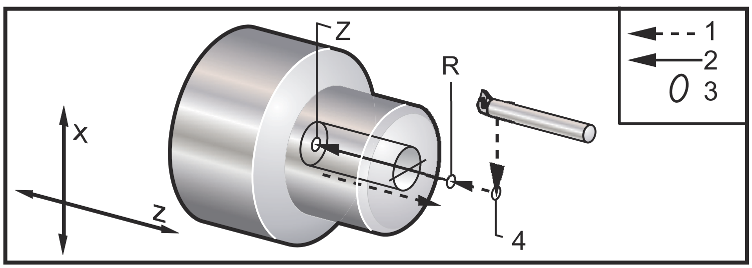
NOTE: *The spindle stops and it rapids out of the hole.*

F - Feed Rate
*__L__ - Number of repeats
*__R__ - Position of the R plane
*__X__ - X-axis motion command
*__Y__ - Y-axis motion command
Z - Position of bottom of hole

* indicates optional

This G code stops the spindle once the tool reaches the bottom of the hole. The tool retracts once the spindle has stopped.

F7.42: `G86` Bore and Stop Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.



G89 Bore and Dwell Canned Cycle (Group 09)

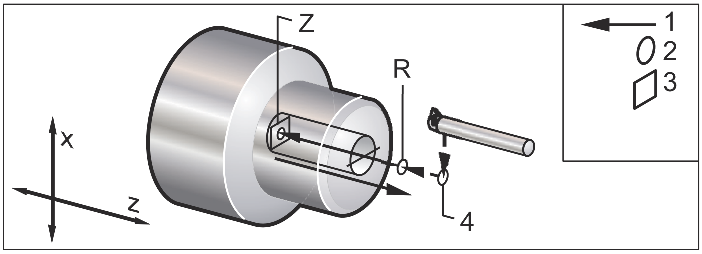
NOTE: *This cycle feeds in and feeds out.*

F - Feed Rate
*__L__ - Number of repeats
*__P__ - The dwell time at the bottom of the hole
*__R__ - Position of the R plane
*__X__ - X-axis motion command
*__Y__ - Y-axis motion command
Z - Position of bottom of hole

* indicates optional

NOTE: *The P values are modal. This means if you are in the middle of a canned cycle and a `G04 Pnn` or an `M97 Pnn` is used the P value will be used for the dwell / subprogram as well as the canned cycle.*

F7.43: `G89` Bore and Dwell Canned Cycle: [1] Feed, [2] Start or end of stroke, [3] Dwell, [4] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.

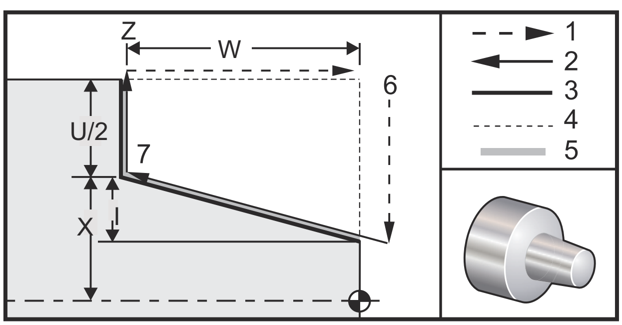


G90 O.D./I.D. Turning Cycle (Group 01)

F(**E**) - Feed rate
***I** - Optional distance and direction of X Axis taper, radius
***U** - X-axis incremental distance to target, diameter
***W** - Z-axis incremental distance to target
X - X-axis absolute location of target
Z - Z-axis absolute location of target

*indicates optional

F7.44: `G90` O.D./I.D. Turning Cycle: [1] Rapid, [2] Feed, [3] Programmed path, [4] Cut allowance, [5] Finish allowance, [6] Start position, [7] Target.

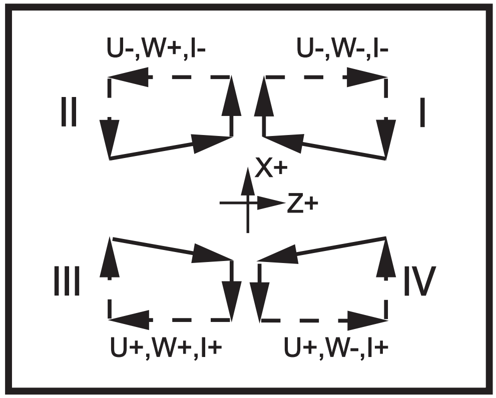


`G90` is used for simple turning, however, multiple passes are possible by specifying the `X` locations of additional passes.

Straight turning cuts are made by specifying `X`, `Z` and `F`. By adding an `I` value, a taper cut is made. The amount of taper is referenced from the target. That is, `I` is added to the value of `X` at the target.

Any of the four ZX quadrants can be programmed using U, W, X, and Z; the taper is positive or negative. The following figure gives a few examples of the values required for machining in each of the four quadrants.

F7.45: G90-G92 Address Relationships



G92 Threading Cycle (Group 01)

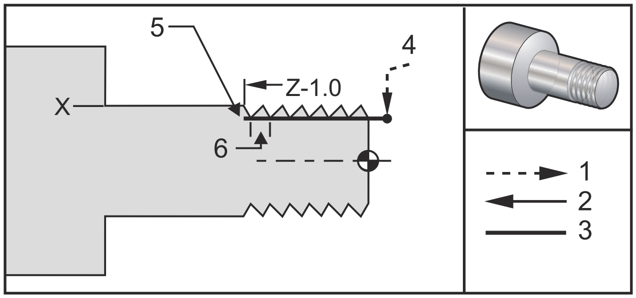
F(**E**) - Feed rate, the lead of the thread
*__I__ - Optional distance and direction of X Axis taper, radius
*__Q__ - Start Thread Angle
*__U__ - X-axis incremental distance to target, diameter
*__W__ - Z-axis incremental distance to target
X - X-axis absolute location of target
Z - Z-axis absolute location of target

* indicates optional

Programming Notes:

- Setting 95/Setting 96 determine chamfer size/angle. M23/M24 turn chamfering on/off.
- G92 is used for simple threading, however, multiple passes for threading are possible by specifying the X locations of additional passes. Straight threads are made by specifying X, Z, and F. By adding an I value, a pipe or taper thread is cut. The amount of taper is referenced from the target. That is, I is added to the value of X at the target. At the end of the thread, an automatic chamfer is cut before reaching the target; default for this chamfer is one thread at 45 degrees. These values can be changed with Setting 95 and Setting 96.
- During incremental programming, the sign of the number following the U and W variables depends on the direction of the tool path. For example, if the direction of a path along the X-axis is negative, the value of U is negative.

F7.46: `G92` Threading Cycle: [1] Rapid, [2] Feed, [3] Programmed path, [4] Start position, [5] Minor diameter, [6] 1/Threads per inch = Feed per revolution (Inch formula; F = lead of thread) .



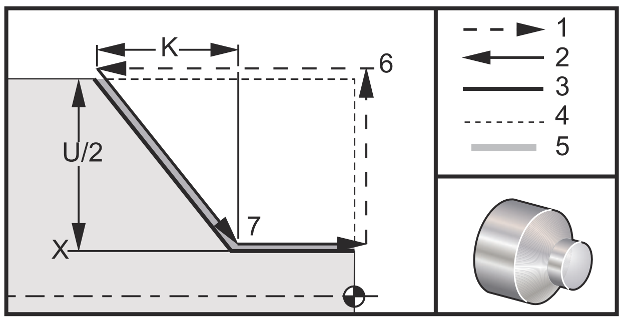

```
%
O60921 (G92 THREADING CYCLE) ;
(G54 X0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is an OD thread tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G50 S1000 (Limit spindle to 1000 RPM) ;
G97 S500 M03 (CSS off, Spindle on CW) ;
G00 G54 X0 Z0.25 (Rapid to 1st position) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
X1.2 Z.2 (Rapid to clear position) ;
G92 X.980 Z-1.0 F0.0833 (Begin Thread Cycle) ;
X.965 (2nd pass) ;
X.955 (3rd pass) ;
X.945 (4th pass) ;
X.935 (5th pass) ;
X.925 (6th pass) ;
X.917 (7th pass) ;
X.910 (8th pass) ;
X.905 (9th pass) ;
X.901 (10th pass) ;
X.899 (11th pass) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 M05 (Z home, spindle off) ;
M30 (End program) ;
%
```

G94 End Facing Cycle (Group 01)

F(**E**) - Feed rate
***K** - Optional distance and direction of Z Axis coning
***U** - X-axis incremental distance to target, diameter
***W** - Z-axis incremental distance to target
X - X-axis absolute location of target
Z - Z-axis absolute location of target

*indicates optional

F7.47: `G94` End Facing Cycle: [1] Rapid, [2] Feed, [3] Programmed path, [4] Cut allowance, [5] Finish allowance, [6] Start position, [7] Target.

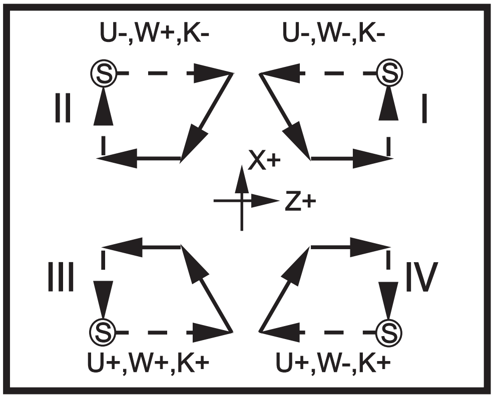


Straight end facing cuts can be made by specifying `X`, `Z` and `F`. By adding `K` a cone-shaped face is cut. The amount of coning is referenced from the target. That is `K` is added to the value of `X` at the target.

Any of the four ZX quadrants is programmed by varying `U`, `W`, `X`, and `Z`. The coning is positive or negative. The following figure gives a few examples of the values required for machining in each of the four quadrants.

During incremental programming, the sign of the number following the `U` and `W` variables depends on the direction of the tool path. If the direction of a path along the X-axis is negative, the value of `U` is negative.

F7.48: G94 Address Relationships: [S] Start position.



G95 Live Tooling Rigid Tap (Face) (Group 09)

*__C__ - C-Axis absolute motion command (optional)
F - Feed Rate
R - Position of the R plane
S - RPM, called prior to G95
W - Z-axis incremental distance
X - Optional Part Diameter X-axis motion command
*__Y__ - Y-axis motion command
Z - Position of bottom of hole

* indicates optional

G95 Live Tooling Rigid Tapping is an axial tapping cycle similar to G84 Rigid Tapping in that it uses the F, R, X and Z addresses, however, it has the following differences:

- The control must be in G99 Feed per Revolution mode in order for tapping to work properly.
- An S (spindle speed) command must have been issued prior to the G95.
- The X Axis must be positioned between machine zero and the center of the main spindle, do not position beyond spindle center.

```
%
o60951 (G95 LIVE TOOLING RIGID TAP) ;
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is a 1/4-20 tap) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G00 G54 X1.5 C0. Z0.5 (Rapid to 1st position) ;
M08 (Coolant on) ;
(BEGIN CUTTING CYCLE) ;
```

```
S500 (Select tap RPM) ;
G95 C45. Z-0.5 R0.5 F0.05 (Tap to Z-0.5) ;
C135. (next position) ;
C225. (next position) ;
C315. (last position) ;
(BEGIN COMPLETION BLOCKS) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 (Z home) ;
M30 (End program) ;
%
```

G96 Constant Surface Speed ON (Group 13)

`G96` commands the control to maintain a constant cutting speed at the tip of the tool. The spindle RPM is based on the diameter of the part where the cut is taking place, and the commanded `S` value (RPM=3.82xSFM/DIA). This means the spindle speed increases as the tool gets closer to `X0`. When Setting 9 is set to **INCH**, the `S` value specifies Surface Feet Per Minute. When Setting 9 is set to **MM**, the `S` value specifies Surface Meters Per Minute.

WARNING: *It is safest to specify a maximum spindle speed for the Constant Surface Speed feature. Use `G50` to set a maximum spindle RPM. Not setting a limit allows the spindle speed to increase as the tool reaches the center of the part. The excessive speed can throw parts and damage tooling.*

G97 Constant Surface Speed OFF (Group 13)

This commands the control to NOT adjust the spindle speed based on the diameter of cut and cancels any `G96` command. When `G97` is in effect, any `S` command is revolutions per minute (RPM).

G98 Feed Per Minute (Group 10)

`G98` changes how the `F` address code is interpreted. The value of `F` indicates inches per minute when Setting 9 is set to **INCH**, and `F` indicates millimeters per minute when Setting 9 is set to **MM**.

G99 Feed Per Revolution (Group 10)

This command changes how the `F` address is interpreted. The value of `F` indicates inches per revolution of the spindle when Setting 9 is set to **INCH**, while `F` indicates millimeters per revolution of the spindle when Setting 9 is set to **MM**.

G100 Disable / G101 Enable Mirror Image (Group 00)

***X** - X-axis command
***Z** - Z-axis command

* indicates optional. At least one is required.

Programmable mirror image can be turned on or off individually for the X and/or Z Axis. The bottom of the screen indicates when an axis is mirrored. These G codes are used in a command block without any other G codes and do not cause any Axis motion. `G101` turns on mirror image for any Axis listed in that block. `G100` turns off mirror image for any Axis listed in the block. The actual value given for the `X` or `Z` code has no effect; `G100` or `G101` by itself has no effect. For example, `G101 X 0` turns on X-axis mirror.

NOTE: *Settings 45 and 47 may be used to manually select mirror image.*

G103 Limit Block Look-Ahead (Group 00)

G103 specifies the maximum number of blocks the control looks ahead (Range 0-15), for example:

```
G103 [P..] ;
```

During machine motions, the control prepares future blocks (lines of code) ahead of time. This is commonly called “Block Look-ahead.” While the control executes the current block, it has already interpreted and prepared the next block for continuous motion.

A program command of `G103 P0`, or simply `G103`, disables block limiting. A program command of `G103 Pn` limits look-ahead to `n` blocks.

`G103` is useful for debugging macro programs. The control interprets Macro expressions during look-ahead time. If you insert a `G103 P1` into the program, the control interprets macro expressions (1) block ahead of the currently executing block.

It is best to add several empty lines after a `G103 P1` is called. This ensures that no lines of code after the `G103 P1` are interpreted until they are reached.

`G103` affects cutter compensation and High Speed Machining.

NOTE: *The P values are modal. This means if you are in the middle of a canned cycle and a `G04 Pnn` or an `M97 Pnn` is used the P value will be used for the dwell / subprogram as well as the canned cycle.*

G105 Servo Bar Command

This is the G-code used to command a Bar Feeder.

```
G105 [In.nnnn] [Jn.nnnn] [Kn.nnnn] [Pnnnnn] [Rn.nnnn]
```

I - Optional Initial Push Length (macro variable #3101) Override (variable #3101 if I is not commanded)
J - Optional Part Length + Cutoff (macro variable #3100) Override (variable #3100 if J is not commanded)
K - Optional Min Clamping Length (macro variable #3102) Override (variable #3102 if K is not commanded)
P - Optional cutoff subprogram
R - Optional spindle orientation for new bar

I, J, K are overrides to macro variable values listed on the Current Commands Page. The control applies the override values only to the command line in which they are located. The values stored in Current Commands are not modified.

NOTE: *A G105with a J code will not increment the counter. The J code is intended for double push operation to make a long part.*

G107 Cylindrical Mapping (Group 00)

Y - Y-Axis command
C - C-Axis command
*__R__ - Radius of the rotary axis
*__Q__ - Diameter of the cylindrical surface

* indicates optional

- Enable G107 mode with Y-axis as linear and C-axis as rotary axis. If address code R or Q is not specified, an alarm is generated.
- G107 R/Q; change the radius of the mapping cylinder.
- G107; Disables the G107 mode.

The following program **O00555** use I,J and K for the arc center point for the 0.500” radius in the corners. Program **O00556** uses the R in place of the I,J and K for the arc center points.

```
%O00555 (G107 CYLINDRICAL MAPPING);
T1717;
M133 P500;
G00 X4.2 Z1.;
Z-3.;
```

```
G98;
M15;
G107 Y0 C0 Q4.;
G01 X3.5 F40.;
G01 G41 Y-1.;
Z-4.5;
G03 Z-5. Y-0.5 J0.5;
G01 Y0.5;
G03 Z-4.5 Y1. K0.5;
G01 Z-1.5;
G03 Y0.5 Z-1. J-0.5;
G01 Y-0.5;
G03 Y-1. Z-1.5 K-0.5;
G01 Z-3.;
G01 G40 Y0;
G00 X4.2;
G107;
Z1.;
G28;
M30;
%

%O00556 (G107 CYLINDRICAL MAPPING R);
T1717;
M133 P500;
G00 X4.2 Z1.;
Z-3.;
G98;
M15;
G107 Y0 C0 Q4.;
G01 X3.965 F40.;
G01 G41 Y-1.;Z-4.5;
G03 Z-5. Y-0.5 R0.5;
G01 Y0.5;
G03 Z-4.5 Y1. R0.5;
G01 Z-1.5;
G03 Y0.5 Z-1. R0.5;
G01 Y-0.5;
G03 Y-1. Z-1.5 R0.5;
G01 Z-3.;
G01 G40 Y0;
G00 X4.2;
G107;
Z1.;
G28;
M30;%
```

G110 / G111 Coordinate System #7/#8 (Group 12)

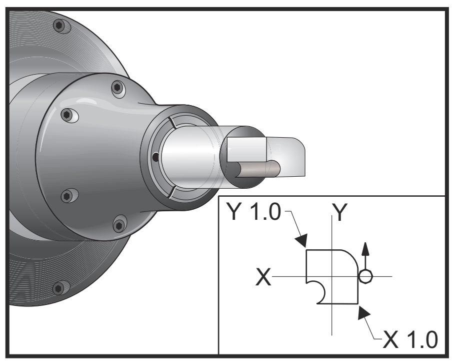
G110 selects #7 and G111 selects #8 additional work offset coordinates. All subsequent references to axes positions are interpreted in the new work offset coordinate system. Operation of `G110` and `G111` is the same as `G154 P1` and `G154 P2`.

G112 XY to XC Interpolation (Group 04)

The `G112` XY to XC coordinate interpolation feature lets you program subsequent blocks in Cartesian XY coordinates, which the control automatically converts to polar XC coordinates. While it is active, the control uses the `G17 XY` for `G01` linear XY strokes and `G02` and `G03` for circular motion. `G112` also converts `X`, `Y` position commands into rotary C-Axis and linear X-axis moves.

G112 Program Example

F7.49: `G112` XY to XC Interpolation




```
%
o61121 (G112 XY TO XC INTERPOLATION) ;
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is an end mill) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G17 (Call XY plane) ;
G98 (Feed per min) ;
P1500 M133 (Live tool CW at 1500 RPM) ;
G00 G54 X0.875 C0. Z0.1 (Rapid to 1st position) ;
G112 (XY to XC interpretation);
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G1 Z0. F15. (Feed towards face) ;
Y0.5 F5. (Linear feed) ;
G03 X.25 Y1.125 R0.625 (Feed CCW) ;
```

```
G01 X-0.75 (Linear feed) ;
G03 X-0.875 Y1. R0.125 (Feed CCW) ;
G01 Y-0.25 (Linear Feed) ;
G03 X-0.75 Y-0.375 R0.125 (Feed CCW) ;
G02 X-0.375 Y-0.75 R0.375 (Feed CW) ;
G01 Y-1. (Linear feed) ;
G03 X-0.25 Y-1.125 R0.125 (Feed CCW) ;
G01 X0.75 (Linear feed) ;
G03 X0.875 Y-1. R0.125 (Feed CCW) ;
G01 Y0. (Linear feed) ;
G00 Z0.1 (Rapid retract) ;
(BEGIN COMPLETION BLOCKS) ;
G113 (Cancel G112) ;
M135 (Live tool off) ;
G18 (Return to XZ plane) ;
G00 G53 X0 M09 (X home, coolant off) ;
G53 Z0 (Z home) ;
M30 (End program) ;
```

G113 Cancel XY to XC Interpolation (Group 04)

`G113` cancels the Cartesian to Polar coordinate conversion.

G114-G129 Coordinate System #9-#24 (Group 12)

`G114` - `G129` codes are user-settable coordinate systems, #9 - #24, for work offsets. All subsequent references to axes' positions are interpreted in the new coordinate system. Work coordinate system offsets are entered from the **`Active Work Offset`** display page. Operation of `G114` - `G129` codes is the same as `G154 P3` - `G154 P18`.

G154 Select Work Coordinates P1-P99 (Group 12)

This feature provides 99 additional work offsets. `G154` with a `P` value from 1 to 99 activates additional work offsets. For example `G154 P10` selects work offset 10 from the list of additional work offsets.

NOTE: *`G110` to `G129` refer to the same work offsets as `G154 P1` through `P20`; they can be selected by using either method.*

When a `G154` work offset is active, the heading in the upper right work offset will show the `G154 P` value.

NOTE: *The P values are modal. This means if you are in the middle of a canned cycle and a `G04 Pnn` or an `M97 Pnn` is used the P value will be used for the dwell / subprogram as well as the canned cycle.*

`G154` work offsets format

```
#14001-#14006 G154 P1 (also #7001-#7006 and G110)

#14021-#14026 G154 P2 (also #7021-#7026 and G111)

#14041-#14046 G154 P3 (also #7041-#7046 and G112)

#14061-#14066 G154 P4 (also #7061-#7066 and G113)

#14081-#14086 G154 P5 (also #7081-#7086 and G114)

#14101-#14106 G154 P6 (also #7101-#7106 and G115)

#14121-#14126 G154 P7 (also #7121-#7126 and G116)

#14141-#14146 G154 P8 (also #7141-#7146 and G117)

#14161-#14166 G154 P9 (also #7161-#7166 and G118)

#14181-#14186 G154 P10 (also #7181-#7186 and G119)

#14201-#14206 G154 P11 (also #7201-#7206 and G120)

#14221-#14221 G154 P12 (also #7221-#7226 and G121)

#14241-#14246 G154 P13 (also #7241-#7246 and G122)

#14261-#14266 G154 P14 (also #7261-#7266 and G123)

#14281-#14286 G154 P15 (also #7281-#7286 and G124)

#14301-#14306 G154 P16 (also #7301-#7306 and G125)

#14321-#14326 G154 P17 (also #7321-#7326 and G126)

#14341-#14346 G154 P18 (also #7341-#7346 and G127)

#14361-#14366 G154 P19 (also #7361-#7366 and G128)

#14381-#14386 G154 P20 (also #7381-#7386 and G129)

#14401-#14406 G154 P21

#14421-#14426 G154 P22

#14441-#14446 G154 P23

#14461-#14466 G154 P24
```

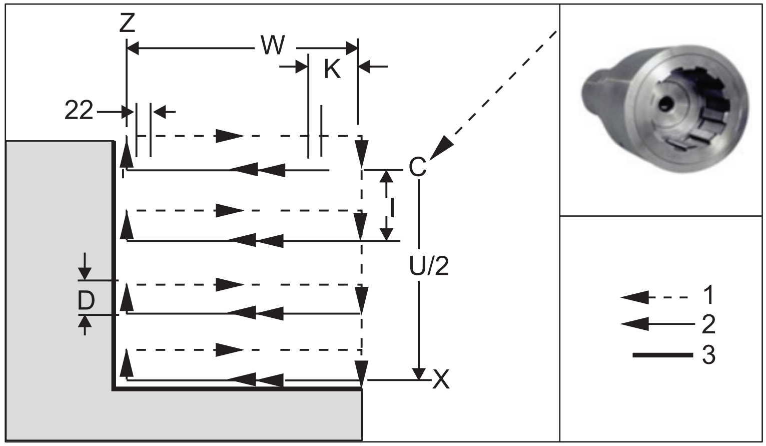
```
#14481-#14486 G154 P25
#14501-#14506 G154 P26
#14521-#14526 G154 P27
#14541-#14546 G154 P28
#14561-#14566 G154 P29
#14581-#14586 G154 P30
#14781-#14786 G154 P40
#14981-#14986 G154 P50
#15181-#15186 G154 P60
#15381-#15386 G154 P70
#15581-#15586 G154 P80
#15781-#15786 G154 P90
#15881-#15886 G154 P95
#15901-#15906 G154 P96
#15921-#15926 G154 P97
#15941-#15946 G154 P98
#15961-#15966 G154 P99
```

G156 Broaching Canned Cycle (Group 09)

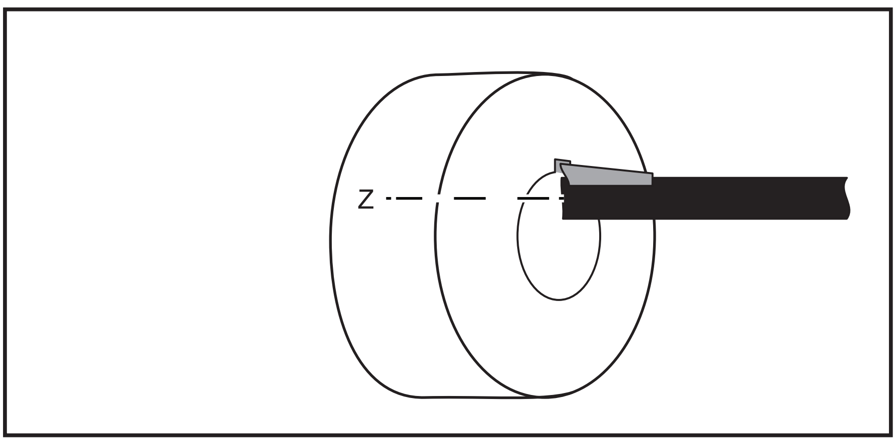
Z - Z-axis absolute location total pecking depth.
W - Z-axis incremental distance to total pecking depth.
* **Y** - Y-axis absolute location of furthest slice cycle (diameter).
* **V** - Y-axis incremental distance to furthest slice (diameter).
X - X-axis absolute location of furthest slice cycle (diameter)
* **U** - X-axis incremental distance to furthest slice (diameter).
* **I** - X-axis size of increment between slice cycles.
* **K** - Z-axis size of increment between pecks in a cycle.
* **F** - Feed rate per minute.
* **C** - C-axis position.
* **D** - Tool clearance when returning to starting plane.

*indicates optional

F7.50: G156 Broaching Canned Cycle: [1] Rapid, [2] Feed, [3] Programmed Path, [C] Clearance Point, [K] Peck ,[22] Peck retraction (Setting 22).



F7.51: Broaching operations canned cycle can be done using a single tool path (with adjustable programmable Pecks) when using a special broaching shape tool.



Spindle orient command M19 P/R code which used to position the spindle before broaching must be used first before executing G156 broaching canned cycle.

NOTE: *M19, P, or R should not be on the same line as G156. M19, P, or R value should not be negative. See*

M19

for more information.

When K is added to a G156 block, then pecking will be performed at each interval specified by K.

The D code can be used for grooving and turning to provide material clearance when returning to starting plane S.

The D code command is used to provide a tool clearance shift, in the X-axis before returning in the Z-axis to the "C" clearance point.

When an X/U, or Y/V code is added to a G156 block and the current (i.e. actual) X/U, Y/V axes machine position not at the position defined by G156 X/U, Y/V, then a minimum of two pecking canned cycles will occur, one at the current (actual) location of the machine X/U, Y/V axes, and another one at the X/U, Y/V location set by G156.

The I code is the incremental distance (slices cycle) between the start position S toward the target X/U, Y/V Axes positions (i.e. the thickness of each slice).

Adding an I will performs multiple, evenly spaced, slice cycles between the starting position S and X/U, Y/V Axes target position.

When the distance between S and target X/U, Y/V is not evenly divisible by I, then the last interval is less than I.

The spindles shall be positioned first (Main/Sup spindle) by either having the C-axis engaged (For Main spindle) or spindle to be oriented (Main/Sub) before staring the broach cycle.

NOTE: *An alarm will be generated when using Cutting feed rate as unit/revolution with Lathe broaching canned cycle.*

G112 (Cartesian to Polar coordinate transformation feature) is not allowed to be used in lathe broaching canned cycle and the control shall alarm out.

The tool clearance shift direction (i.e. shift in + X or - X, ZX plane,) will be depend on the sign of the D value (Positive/Negative).

When the D has a positive value, the tool moves in the positive direction of the X-axis.

When the D has a negative value, the tool moves further in the negative direction of the X-axis.

When D has a zero value or it not specified with the G156 lathe broaching canned cycle, then there shall be no tool clearance shift in the X-axis before returning in the Z-axis to the "C" clearance point.I.e. the tool returning in the Z-axis to the "C" clearance point will use the same peck path in the opposite direction.

NOTE: *An alarm will be generated if any spindle commanded to turn (other than spindle orient/ c axis position) within broaching canned cycle, or the broaching canned cycle is starting the Pecks cycle and the spindle is still turning.*

`Cxx` (position the C axis) could be use before or on the same line of `G156` broaching canned cycle.

Using the spindle brake before starting the broach cycle is recommended, but is not required.

NOTE: *The broaching canned cycle is a modal, and it shall be repeated on each line within the canned cycle uses any broaching address code. Refer to examples below:*

T7.1: Lathe Broaching: Example 1

<table>
<tr><th>G-Code</th><th>Program Sequence</th></tr>
<tr><td><pre>%
O41322 (BROACH SAMPLE
PROGRAM);
G00 G54 G18 G40 G80 G97
G98;
G0 T101;
X1. Z.25;
C0.;
G156 X1.05 Y0. Z-1. I.0025
D0.01 F150.;
C45.;
C90.;
C135.;
C180.;
G80 G0 Z.25;
G28 X0. Z0.;
M30;
%</pre></td><td>Safe Start Line
Tool 1 Offset 1
Rapid to Position
Position to C0.
Broach operations canned cycle start.
C-axis position.
End of Canned Cycle.</td></tr>
</table>

T7.2: Lathe Broaching: Example 2

G-Code	Program Sequence

`%`
`G156 X-10. Y-2.0 Z-12.0 I0.5 K0.5 F10.;`
`X-5. Y-1.5 Z-10.0 I2.0 K2.0;`
`I1.0 K1.0;`
`C90.;`
`C45.0 X-5. Z-10.0 I2.0 K2.0;`
`G00 G80;`
`%`
	• Lathe broaching operations canned cycle start.
	• Perform another broach with these new values at the current angle.
	• Perform another broach with these new values at the current angle.
	• Perform another broach at the new angle using the last specified address codes.
	• Rotate to new angle then broach using the new address codes.
	• End of canned cycle.

T7.3: Lathe Broaching: Example 3

G-Code	Program Sequence

`%`
`G156 X-10. Z-12.0 I0.5 K0.5 F10. C45.;`
`%`
	• Position the C-axis first, then start the Lathe broaching operations canned cycle start (Spindle should not be turning).

T7.4: Lathe Broaching: Example 4

G-Code	Program Sequence

`%`
`M19 P45 (or R45);`
`G156 X-10. Z-12.0 I0.5`
`K0.5 F10.;`
`M19 P90;`
`M19 P135;`
`C180.;`
`G00 G80;`
`%`
	• Position the spindle (Using Spindle Orient).
	• Lathe broaching operations canned cycle start.
	• Repeat the canned cycle at this position.
	• Repeat the canned cycle at this position.
	• Position the C axis first (Using C-axis), then start the Lathe broaching operations canned cycle start.
	• End of canned cycle.
	.

T7.5: Lathe Broaching: Example 5

G-Code	Program Sequence

`%`
`C45.;`
`G156 X-10. Z-12.0 I0.5`
`K0.5 F10.;`
`C90.;`
`C135.;`
`M19 R180.;`
`M03 S2000;`
`M19 P200.;`
`G00 G80;`
`%`
	• Position the spindle (Using C axis)
	• Lathe broaching operations canned cycle start
	• Repeat the canned cycle at this position
	• Repeat the canned cycle at this position
	• Using a different method (not C axis) to Position the spindle first, then start the Lathe broaching operations canned cycle start.
	• M03 S2000; Alarm: The spindle is running.
	• End of canned cycle
	.

T7.6: Lathe Broaching: Example 6

G-Code	Program Sequence
% M03 S2000; G156 X-10. Z-12.0 I0.5 K0.5 F10.; %	• Run the spindle • Alarm: The spindle is running. .

G167 - Modify Setting (Group 00)

P - This code specifies the setting number.
Q - specifies the setting value. This can be a decimal numeric value or an enumerated representation for unit-less settings.
K - the number after the `K` code specifies a guard code for permanent changes.

This G-code allows the user permanently make changes to settings during program execution.

The control will generate an Alarm when:

- the `P` or `Q` code is missing.
- the `P` code is not a valid setting number.
- the `Q` code is not a valid for the setting number.

Examples of G167:

Example #1

```
G167 P250 Q1 K10755;
```

Turns on setting 250 Mirror Image C Axis.

Example #2

```
G167 P84 Q3 K10755;
```

Sets setting 84 Tool Overload Action to Autofeed.

Example #3

```
G167 P142 Q1.25 K10755;
```

Sets setting 142 Offset Chng Tolerance to 1.25.

G184 Reverse Tapping Canned Cycle For Left Hand Threads (Group 09)

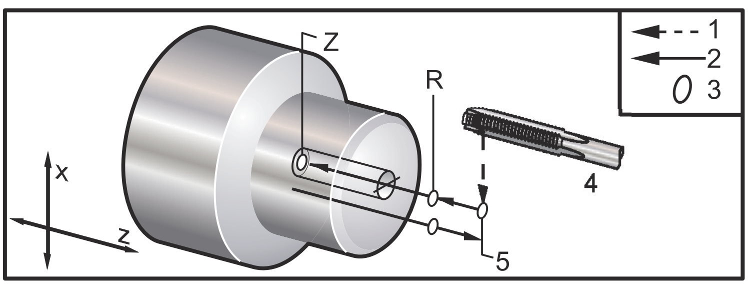
F - Feed Rate in inches (mm) per minute
R - Position of the R plane
S - RPM, called prior to `G184` is necessary
*__W__ - Z-axis incremental distance
*__X__ - X-axis motion command
Z - Position of bottom of hole

* indicates optional

Programming Notes: When tapping, the feedrate is the lead of the thread. See example of `G84`, when programmed in `G99` Feed per Revolution.

It is not necessary to start the spindle CCW before this canned cycle; the control does this automatically.

F7.52: `G184` Reverse Tapping Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Left-hand tap, [5] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.

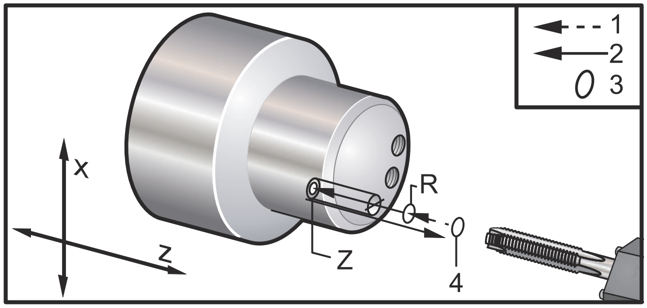


G186 Reverse Live Tool Rigid Tap (For Left Hand Threads) (Group 09)

F - Feed Rate
C - C-Axis position
R - Position of the R plane
S - RPM, called prior to `G186` is necessary
W - Z-axis incremental distance
*__X__ - Part Diameter X-axis motion command
*__Y__ - Y-axis motion command
Z - Position of bottom of hole

* indicates optional

F7.53: G95, G186 Live Tooling Rigid Tapping: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting plane, [R] R plane, [Z] Position of the bottom of the hole.



It is not necessary to start the spindle CW before this canned cycle; the control does this automatically. See G84.

G187 Accuracy Control (Group 00)

G187 is an accuracy command that can set and control both the smoothness and max corner rounding value when cutting a part. The format for using G187 is G187 Pn Ennnn.

P - Controls the smoothness level, P1(rough), P2(medium), or P3(finish). Temporarily overrides Setting 191.
E - Sets the max corner rounding value. Temporarily overrides Setting 85.

Setting 191 sets the default smoothness to the user specified **ROUGH**, **MEDIUM**, or **FINISH** when G187 is not active. The **Medium** setting is the factory default setting.

NOTE: *Changing Setting 85 to a low value may make the machine operate as if it is in exact stop mode.*

NOTE: *Changing setting 191 to **FINISH** will take longer to machine a part. Use this setting only when needed for the best finish.*

G187 Pm Ennnn sets both the smoothness and max corner rounding value. G187 Pm sets the smoothness but leaves max corner rounding value at its current value. G187 Ennnn sets the max corner rounding but leaves smoothness at its current value. G187 by itself cancels the E value and sets smoothness to the default smoothness specified by Setting 191. G187 will be canceled whenever **[RESET]** is pressed, M30 or M02 is executed, the end of program is reached, or **[EMERGENCY STOP]** is pressed.

G195 Forward Live Tool Radial Tapping (Diameter) / G196 Reverse Live Tool Radial Tapping (Diameter) (Group 09)

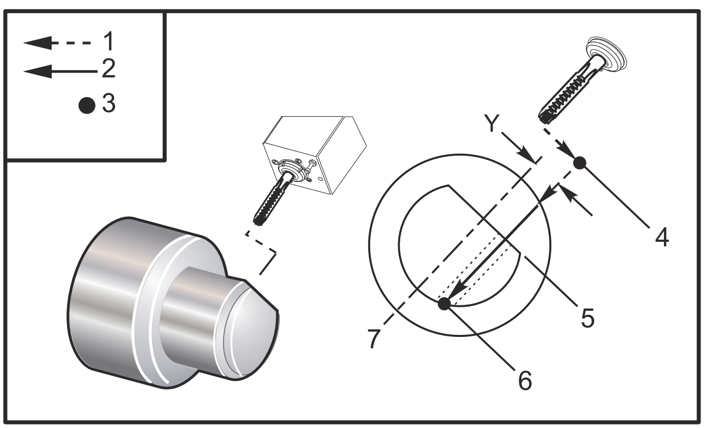
F - Feed Rate per revolution (G99)
***U** - X-Axis incremental distance to the bottom of hole
S - RPM, called prior to `G195`
X - X- Axis absolute position at the bottom of hole
***Z** - Z-Axis absolute position motion command
R - Position of the R plane
***C** - C-Axis absolute motion command
***Y** - Y-Axis absolute motion command
***W** - Z-Axis incremental motion command
***E** - Chip-clean RPM (Spindle reverses to remove chips after each hole)

* indicates optional

This G-code is modal in that it activates the canned cycle until it is canceled or another canned cycle is selected. The cycle begins from the current position, tapping to the X-Axis depth specified. An R plane can be used.

`S` RPM should be called out as a positive number. It is not necessary to start the spindle in the correct direction; the control does this automatically.

F7.54: `G195`/`G196` Live Tooling Rigid Tapping: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting point, [5] Part surface, [6] Bottom of the hole, [7] Centerline.




```
o61951 (G195 LIVE RADIAL TAPPING) ;
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is a tap) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G00 G54 X3.25 Z-0.75 C0. (Start Point) ;
```

```
M08 (coolant on) ;
(BEGIN CUTTING BLOCK) ;
S500 (Select tap RPM) ;
G195 X2. F0.05 (Taps to X2., bottom of hole) ;
G00 C180. (Index C-Axis) ;
G00 C270. Y-1. Z-1. (Index C-Axis, YZ-axis positioning) ;
G80 (Cancel Canned Cycle);
(BEGIN COMPLETION BLOCKS) ;
G00 Z0.25 M09 (Rapid retract, coolant off) ;
G53 X0 Y0 (X & Y home) ;
G53 Z0 (Z home) ;
M30 (End program) ;
```

G198 Disengage Synchronous Spindle Control (Group 00)

`G198` disengages synchronous spindle control and allows independent control of the main spindle and the secondary spindle.

G199 Engage Synchronous Spindle Control (Group 00)

***R** - Degrees, phase relationship of following spindle to commanded spindle

* indicates optional

This G code synchronizes the RPM of the two spindles. Position or speed commands to the following spindle, usually the secondary spindle, are ignored when spindles are in synchronous control. However, M codes on the two spindles are controlled independently.

The spindles remain synchronized until synchronous mode is disengaged using `G198`. This is the case even if power is cycled.

An `R` value on the `G199` block positions the following spindle to a specified number of degrees, relative to the 0 mark on the commanded spindle. Examples of `R` values in `G199` blocks:

```
G199 R0.0 (The following spindle's origin, 0-mark, matches the
commanded spindle's origin, 0-mark) ;
G199 R30.0 (The following spindle's origin, 0-mark, is
positioned +30 degrees from the commanded spindle's origin,
0-mark) ;
G199 R-30.0 (The following spindle's origin, 0-mark, is
positioned -30 degrees from the commanded spindle's origin,
0-mark) ;
```

When an R value is specified on the G199 block, the control first matches the velocity on the following spindle to that of the commanded spindle, then adjusts the orientation (R value in the G199 block). Once the specified R orientation is achieved the spindles are locked in synchronous mode until disengaged with a G198 command. This can also be achieved at zero RPM. Refer also to the G199 portion of the Synchronized Spindle Control Display on **228**.

```
%
o61991 (G199 SYNC SPINDLES) ;
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on the face of the part) ;

(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G20 G40 G80 G99 (Safe startup) ;

G00 G54 X2.1 Z0.5 ;
G98 M08 (Feed per min, turn coolant on) ;


(BEGIN CUTTING BLOCKS) ;
G01 Z-2.935 F60. (Linear feed) ;
M12 (Air blast on) ;
M110 (Secondary spindle chuck clamp) ;
M143 P500 (Secondary spindle to 500 RPM) ;
G97 M04 S500 (Main spindle to 500 RPM) ;
G99 (Feed per rev) ;
M111 (Secondary spindle chuck unclamp) ;
M13 (Air blast off) ;
M05 (main spindle off) ;
M145 (Secondary spindle off) ;
G199 (Synch spindles) ;

G00 B-28. (Rapid secondary spindle to face of part) ;
G04 P0.5 (Dwell for .5 sec) ;
G00 B-29.25 (Feed secondary spindle onto part) ;
M110 (secondary spindle chuck clamp) ;
G04 P0.3 (Dwell for .3 sec) ;
M08 (Turn coolant on) ;
G97 S500 M03 (Turn spindle on at 500 RPM, CSS off) ;
G96 S400 (CSS on, RPM is 400) ;
G01 X1.35 F0.0045 (Linear feed) ;
X-.05 (Linear feed) ;
G00 X2.1 M09 (Rapid retract) ;
G00 B-28. (Rapid secondary spindle to face of part) ;
G198 (Synch spindle off) ;
```

```
M05 (Turn off main spindle) ;
G00 G53 B-13.0 (Secondary spindle to cut position);
G00 G53 X-1. Y0 Z-11. (Rapid to 1st position) ;
(*******second side of part********)
G55 G99 (G55 for secondary spindle work offset) ;
G00 G53 B-13.0 ;
G53 G00 X-1. Y0 Z-11. ;
G14 ;
T101 (Select tool and offset 1) ;
G50 S2000 (limit spindle to 1000 RPM);
G97 S1300 M03 ( ;
G00 X2.1 Z0.5 ;
Z0.1 M08 ;
G96 S900 ;
G01 Z0 F0.01 ;
X-0.06 F0.005 ;
G00 X1.8 Z0.03 ;
G01 Z0.005 F0.01 ;
X1.8587 Z0 F0.005 ;
G03 X1.93 Z-0.0356 K-0.0356 ;
G01 X1.935 Z-0.35 ;
G00 X2.1 Z0.5 M09 ;
G97 S500 ;
G15 ;
G53 G00 X-1. Y0 Z-11. ;

(BEGIN COMPLETION BLOCKS) ;
G00 G53 X0 M09 (X home) ;
G53 Z0 (Z home) ;
G28 H0. (Unwind C-Axis) ;
M30 (End program) ;
%
```

G200 Index on the Fly (Group 00)

U - Optional relative move in X to tool change position
W - Optional relative move in Z to tool change position
X - Optional final X position
Z - Optional final Z position
T - Required tool number and offset number in standard form

`G200` Index on the Fly causes the lathe to perform a move away, change tools, and move back to the part, to save time.

CAUTION: *The G200 does speed things up, but it also requires you to be more careful. Make sure you proof the program well, at 5% rapid, and be very cautious if you are starting from the middle of the program.*

Normally, your tool change line consists of a few lines of code, like:

```
G53 G00 X0. (BRING TURRET TO SAFE X TC POS) ;
G53 G00 Z-10. (BRING TURRET TO SAFE Z TC POS) ;
T202 ;
```

Using `G200`, changes this code to:

```
G200 T202 U.5 W.5 X8. Z2. ;
```

If `T101` just finished turning the O.D. of the part, you don't need to go back to a safe tool change position, when using a G200. Instead (as in the example) the moment the G200 line is called the turret:

1. Unclamps, in its current position.
2. Moves incrementally in the X and Z axes by the values stated in `U` and `W` (`U.5 W.5`)
3. Completes the tool change at this position.
4. Using the new tool and work offsets, it rapids to the XZ position called out on the `G200` line (`X8. Z2.`).

This all happens very quickly, and nearly all at the same time, so try it out a few times, away from the chuck.

When the turret unclamps, it moves towards the spindle a tiny amount (perhaps .1-.2"), so you do not want the tool directly up against your jaws or collet when the `G200` is commanded.

Because the `U` and `W` moves are incremental distances from where the tool is currently, if you hand jog away and start your program in a new position, the turret moves up and to the right of that new position. In other words, if you manually jogged back within .5" of your tailstock, and then commanded `G200 T202 U.5 W1. X1. Z1.`, the turret would hit your tailstock - moving an incremental W1. (1" to right). For this reason, you may want to setup your Setting 93 and Setting 94, Tailstock Restricted Zone.

Information on this can be found on page **145**.

G211 Manual Tool Setting / G212 Auto Tool Setting

T - Tool number. May be entered as `Tnn` or `Tnnnn`.
H - Tool tip direction. `H-5` will approach the probe from the X (-) side and `H5` from the X (+) side.
***K** - Indicates a calibration cycle. (Values 1 or 2)
***M** - Tool breakage tolerance value.
***C** - Drill diameter value. Only valid with tip directions 5-8. Offset will be adjusted by half this amount (i.e. the program assumes a 90-degree drill point).
***X** - Adjust the approach and start points of a probing cycle.
***Z** - Adjust the approach and start points of a probing cycle.
***B** - Allows the user to use a different amount to move the tool the tool in X or Z while probing (from the start point to in position over the probe). Default value is 6mm.
***U** - Adjust the X start point on `H1 - 4`.
***W** - Adjust the Z start point on `H1 - 4`.

*indicates optional

NOTE: *The `G211` code requires a `Tnnn` code, either directly before the `G211`line, or on the same line. `The G211` code also requires an `Hnnn` code. The `G212` code only requires an `Hnnn` code on the same line but a `Tnnn` code tool call is required prior.*

Using G211 Manual Tool Setting

IMPORTANT: *The Automatic Tool Probe must be calibrated before using `G211 / G212`.*

The `G211` code is used to set an initial tool offset (X, Z or both). To use the probe arm must be lowered. Then the tool tip jogged into place about 0.25 in from the corner of the problem that corresponds to the desired tip direction. The code will either use the current tool offset if one has been called previously or the tool offset may be chosen using a `T` code. The cycle will probe the tool, enter the offset and return the tool to the start position.

Using G212 Auto Tool Setting

The `G212` code is used to re-probe a tool that already has an offset set, such after an insert is changed. It can be also be used to check for tool breakage. The tool will be moved from any location into proper orientation to the probe by the `G212` command. This path is determined by the tool tip direction variable `H`, this variable must be correct or the tool may crash.

IMPORTANT: *Care must be used for touching off any back working tools, to keep from hitting the spindle or the back wall of the machine. A tool and offset must be called **Tnnn** before running `G212`, or an alarm will be generated.*

`G212` code is used to re-probe a tool that already has an offset set, such after an insert is changed. It can be also be used to check for tool breakage. The tool will be moved from any location into proper orientation to the probe by the `G212` command. This path is determined by the tool tip direction variable `H` and it must be correct or the tool may crash.

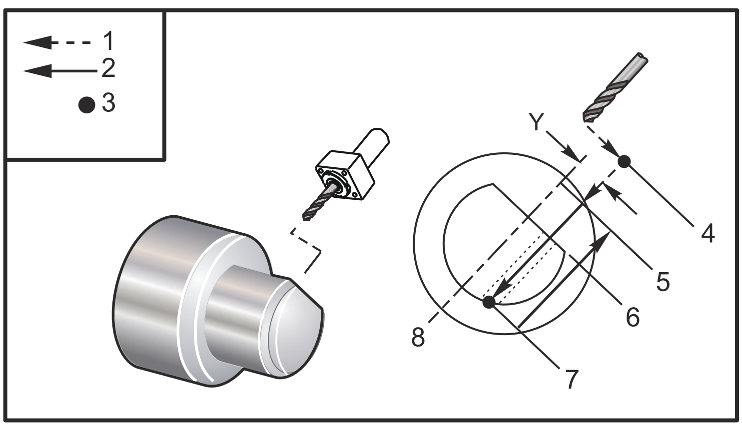
IMPORTANT: *Care must be used for touching off any back working tools, to keep from hitting the spindle or the back wall of the machine. A tool and offset must be called `Tnnn` before running `G212`, or an alarm will be generated.*

G241 Radial Drill Canned Cycle (Group 09)

C - C-Axis absolute motion command
F - Feed Rate
R - Position of the R plane (diameter)
X - Position of bottom of hole (diameter)
***Y** - Y-axis absolute motion command
***Z** - Z-axis absolute motion command

* indicates optional

F7.55: `G241` Radial Drill Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting point, [5] R plane, [6] Part surface, [Z] Bottom of the hole, [8] Centerline.




```
%
o62411 (G241 RADIAL DRILLING) ;
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is a drill) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G98 (Feed per min) ;
G00 G54 X5. Z-0.75 (Rapid to 1st position) ;
```

```
P1500 M133 (Live tool CW at 1500 RPM) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G241 X2.1 Y0.125 Z-1.3 C35. R4. F20. (Begin G241) ;
X1.85 Y-0.255 Z-0.865 C-75. (next position) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, coolant off) ;
M135 (Live tool off) ;
G53 X0 Y0 (X & Y Home) ;
G53 Z0 (Z Home) ;
M30 (End program) ;
%
```

G242 Radial Spot Drill Canned Cycle (Group 09)

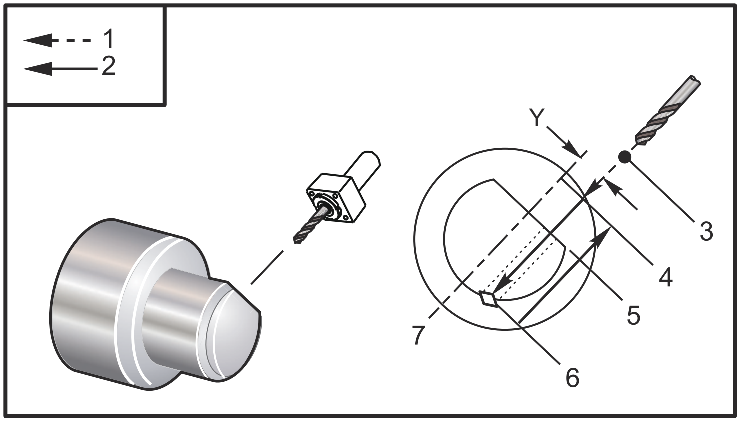
C - C-Axis absolute motion command
F - Feed Rate
P - The dwell time at the bottom of the hole
R - Position of the R plane (Diameter)
X - Position of bottom of hole (Diameter)
***Y** - Y-axis motion command
***Z** - Z-axis motion command

* indicates optional

This G code is modal. It remains active until it is canceled (`G80`) or another canned cycle is selected. Once activated, every motion of `Y` and/or `Z` executes this canned cycle.

NOTE: *The P values are modal. This means if you are in the middle of a canned cycle and a `G04 Pnn` or an `M97 Pnn` is used the P value will be used for the dwell / subprogram as well as the canned cycle.*

F7.56: G242 Radial Spot Drill Canned Cycle: [1] Rapid, [2] Feed, [3] Starting point, [4] R plane, [5] Part surface, [6] Dwell at the bottom of the hole, [7] Centerline.




```
%
o62421 (G242 RADIAL SPOT DRILL) ;
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is a spot drill) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G98 (Feed per min) ;
G00 G54 X5. Y0.125 Z-1.3 (Rapid to 1st position) ;
P1500 M133 (Live tool CW at 1500 RPM) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G241 X2.1 Y0.125 Z-1.3 C35. R4. P0.5 F20. ;
(Drill to X2.1) ;
X1.85 Y-0.255 Z-0.865 C-75. P0.7 (next position) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, coolant off) ;
M135 (Live tool off) ;
G53 X0 Y0 (X & Y Home) ;
G53 Z0 (Z Home) ;
M30 (End program) ;
%
```

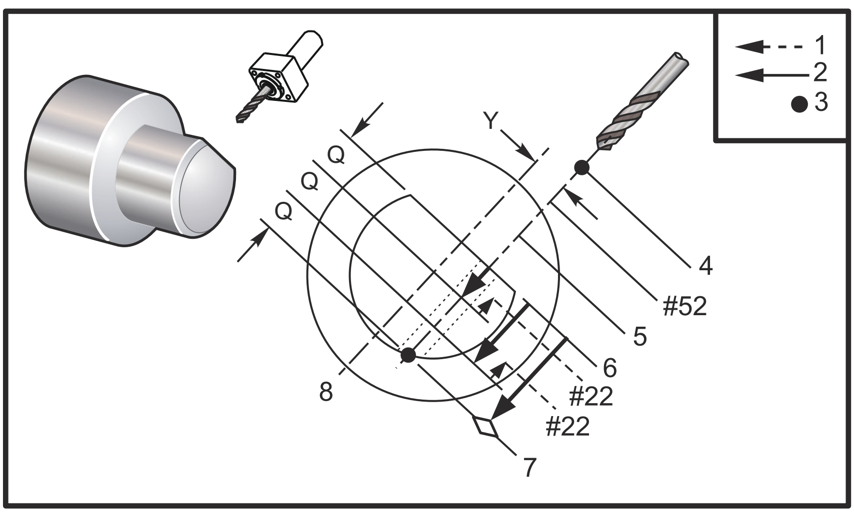
G243 Radial Normal Peck Drilling Canned Cycle (Group 09)

C - C-Axis absolute motion command
F - Feed Rate
*__I__ - Size of first cutting depth
*__J__ - Amount to reduce cutting depth each pass
*__K__ - Minimum depth of cut
*__P__ - The dwell time at the bottom of the hole
*__Q__ - The cut-in value, always incremental
R - Position of the R plane (Diameter)
X - Position of bottom of hole (Diameter)
*__Y__ - Y-axis absolute motion command
*__Z__ - Z-axis absolute motion command

* indicates optional

NOTE: *The P values are modal. This means if you are in the middle of a canned cycle and a `G04 Pnn` or an `M97 Pnn` is used the P value will be used for the dwell / subprogram as well as the canned cycle.*

F7.57: `G243` Radial Normal Peck Drilling Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] R plane, [#52] Setting 52, [5] R plane, [6] Part surface, [#22] Setting 22, [7] Dwell at the bottom of the hole, [8] Centerline.



Programming Notes: If `I`, `J`, and `K` are specified, a different operating mode is selected.The first pass will cut in the value of `I`, each succeeding cut will be reduced by amount J, and the minimum cutting depth is `K`. Do not use a `Q` value when programming with `I`,`J`, and `K`.

Setting 52 changes the way `G243` works when it returns to the `R` plane. Usually the `R` plane is set well outside the cut to insure that the chip clearing motion allows the chips to clear the hole. However, this is wasted motion when first drilling through this empty space. If Setting 52 is set to the distance required to clear chips, the `R` plane can be put much closer to the part being drilled. When the clear move to `R` occurs, the `Z` will be moved past `R` by this value in setting 52. Setting 22 is the amount to feed in `X` to get back the same point at which the retraction occurred.

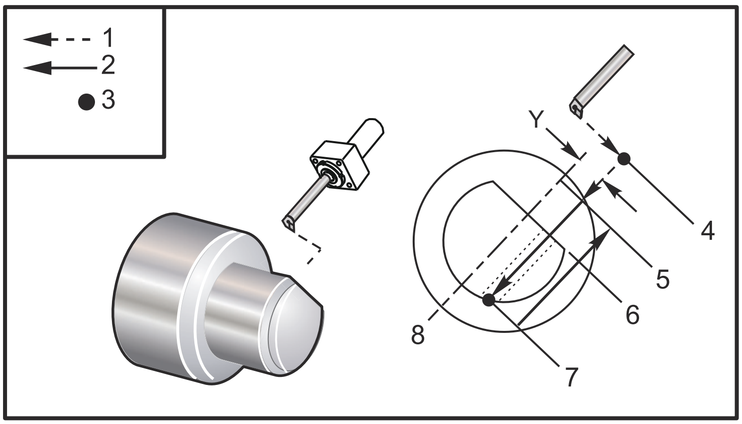
```
%
o62431 (G243 RADIAL PECK DRILL CYCLE) ;
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is a drill) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G98 (Feed per min) ;
G00 G54 X5. Y0.125 Z-1.3 (Rapid to 1st position) ;
P1500 M133 (Live tool CW at 1500 RPM) ;
M08 (Coolant on) ;
G243 X2.1 Y0.125 Z-1.3 C35. R4. Q0.25 F20. ;
(Drill to X2.1) ;
X1.85 Y-0.255 Z-0.865 C-75. Q0.25 (Next position);
G00 Z1. (Rapid retract) ;
M135 (Live tool off) ;
G00 G53 X0 M09(X home, coolant off)  ;
G53 Z0 ;
M00 ;
(G243 - RADIAL WITH I,J,K PECK DRILLING) ;
G00 G54 X5. Y0.125 Z-1.3 (Rapid to 1st position) ;
P1500 M133 (Live tool CW - 1500 RPM) ;
M08 (Coolant on) ;
G243 X2.1 Y0.125 Z-1.3 I0.25 J0.05 K0.1 C35. R4. F5. ;
(Drill to X2.1) ;
X1.85 Y-0.255 Z-0.865 I0.25 J0.05 K0.1 C-75. ;
(next position) ;
(BEGIN COMPLETION BLOCKS) ;
M135 (Turn live tool off) ;
G00 G53 X0 Y0 M09 (X & Y home, coolant off) ;
G53 Z0 (Z home) ;
M30 (End program) ;
%
```

G245 Radial Boring Canned Cycle (Group 09)

C - C-Axis absolute motion command
F - Feed Rate
R - Position of the **R** plane (Diameter)
X - Position of bottom of hole (Diameter)
***Y** - Y-axis absolute motion command
***Z** - Z-axis absolute motion command

* indicates optional

F7.58: `G245` Radial Boring Canned Cycle: [1] Rapid, [2] Feed, [3] Start or end of stroke, [4] Starting point, [5] R plane, [6] Part surface, [Z] Bottom of the hole, [8] Centerline.




```
%
o62451 (G245 RADIAL BORING) ;
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is a boring tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G98 (Feed per min) ;
G00 G54 X5. Y0.125 Z-1.3 (Rapid to 1st position) ;
P500 M133 (Live tool CW at 500 RPM) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G245 X2.1 Y0.125 Z-1.3 C35. R4. F20. ;
(Bore to X2.1) ;
X1.85 Y-0.255 Z-0.865 C-75. (next position) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, coolant off) ;
M135 (live tool off) ;
G53 X0 Y0 (X & Y home) ;
```

```
G53 Z0 (Z home) ;
M30 (End program) ;
%
```

G246 Radial Bore and Stop Canned Cycle (Group 09)

C - C-Axis absolute motion command
F - Feed Rate
R - Position of the R plane (Diameter)
X - Position of bottom of hole (Diameter)
*__Y__ - Y-axis absolute motion command
*__Z__ - Z-axis absolute motion command

*indicates optional

This G code stops the spindle once the tool reaches the bottom of the hole. The tool is retracted once the spindle has stopped.

```
%
o62461 (G246 RADIAL BORE AND STOP) ;
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is a boring tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G98 (Feed per min) ;
G00 G54 X5. Y0.125 Z-1.3 (Rapid to 1st position) ;
P500 M133 (Live tool CW at 500 RPM) ;
M08 (Coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G246 X2.1 Y0.125 Z-1.3 C35. R4. F20. ;
(Bore to X2.1) ;
X1.85 Y-0.255 Z-0.865 C-75. (next position) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, coolant off) ;
M135 (Live tool off) ;
G53 X0 Y0 (X & Y Home) ;
G53 Z0 (Z Home) ;
M30 (End program) ;
%
```

G247 Radial Bore and Manual Retract Canned Cycle (Group 09)

C - C-Axis absolute motion command
F - Feed Rate
R - Position of the R plane (Diameter)
*__X__ - Position of bottom of hole (Diameter)
*__Y__ - Y-axis absolute motion command
*__Z__ - Z-axis absolute motion command

* indicates optional

This G code stops the spindle at the bottom of the hole. At this point the tool is manually jogged out of the hole. The program continues when **[CYCLE START]** is pressed.

```
%
o62471 (G247 RADIAL BORE AND MANUAL RETRACT) ;
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is a boring tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G98 (Feed per minute) ;
G00 G54 X5. Y0.125 Z-1.3 (Rapid to 1st position) ;
P500 M133 (Live tool CW at 500 RPM) ;
M08 (coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G247 X2.1 Y0.125 Z-1.3 C35. R4. F20. ;
(Bore to X2.1) ;
X1.85 Y-0.255 Z-0.865 C-75. (next position) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, Coolant off) ;
M135 (Live tool off) ;
G53 X0 Y0 (X & Y Home) ;
G53 Z0 (Z Home) ;
M30 (End program) ;
%
```

G248 Radial Bore and Dwell and Manual Retract Canned Cycle (Group 09)

C - C-Axis absolute motion command
F - Feed Rate
P - The dwell time at the bottom of the hole
R - Position of the R plane (Diameter)
***X** - Position of bottom of hole (Diameter)
***Y** - Y-axis absolute motion command
***Z** - Z-axis absolute motion command

* indicates optional

This G code stops the tool at the bottom of the hole, and dwells with the tool turning for the time designated with the P value. At this point the tool is manually jogged out of the hole. The program continues when **[CYCLE START]** is pressed.

```
%
o62481 (G248 RADIAL BORE, DWELL, MANUAL RETRACT) ;
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is a boring tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G98 (Feed per minute) ;
G00 G54 X5. Y0.125 Z-1.3 (Rapid to 1st position) ;
P500 M133 (Live tool CW at 500 RPM) ;
M08 (coolant on) ;
(BEGIN CUTTING BLOCKS) ;
G248 X2.1 Y0.125 Z-1.3 C35. R4. P1. F20. ;
(Bore to X2.1) ;
X1.85 Y-0.255 Z-0.865 C-75. (next position) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, coolant off) ;
M135 (Live tool off) ;
G53 X0 Y0 (X & Y Home) ;
G53 Z0 (Z Home) ;
M30 (End program) ;
%
```

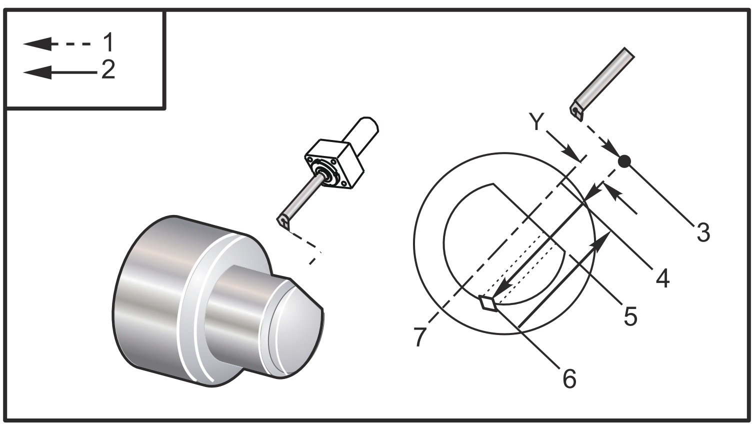
G249 Radial Bore and Dwell Canned Cycle (Group 09)

C - C-Axis absolute motion command
F - Feed Rate
P - The dwell time at the bottom of the hole
R - Position of the R plane
X - Position of bottom of hole
***Y** - Y-axis motion command
*Z - Z-axis motion command

* indicates optional

NOTE: *The P values are modal. This means if you are in the middle of a canned cycle and a `G04 Pnn` or an `M97 Pnn` is used the P value will be used for the dwell / subprogram as well as the canned cycle.*

F7.59: `G249` Radial Bore and Dwell Canned Cycle: [1] Rapid, [2] Feed, [3] Starting point, [4] R plane, [5] Part surface, [6] Dwell at the bottom of the hole, [7] Centerline.




```
%
o62491 (G249 RADIAL BORE AND DWELL) ;
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(T1 is a boring tool) ;
(BEGIN PREPARATION BLOCKS) ;
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
G98 (Feed per minute) ;
G00 G54 X5. Y0.125 Z-1.3 (Rapid to 1st position) ;
P500 M133 (Live tool CW at 500 RPM) ;
M08 (coolant on) ;
(BEGIN CUTTING BLOCKS) ;
```

```
G249 X2.1 Y0.125 Z-1.3 C35. R4. P1.35 F20. ;
(Bore to X2.1) ;
X1.85 Y-0.255 Z-0.865 C-75. P1.65 (next position) ;
(BEGIN COMPLETION BLOCKS) ;
G00 Z1. M09 (Rapid retract, Coolant off) ;
M135 (Live tool off) ;
G53 X0 Y0 (X & Y home) ;
G53 Z0 (Z home) ;
M30 (End program) ;
%
```

G266 Visible Axes Linear Rapid %Motion (Group 00)

E - Rapid rate.
P - Axis parameter number. Example P1 = X, P2 = Y, P3 = Z.
I - Machine coordinate position command.

The below example commands the X-axis to move to X-1. at 10% rapid rate.

```
%
G266 E10. P1 I-1
%
```

To use the bar feeder pushrod as a stop. The example below is commanding the bar feeder axis to move to -10. From home (left side) @ 10% rapid rate.

```
%
G266 E10. P13 I-10.
%
```

To load the pushrod, select **[RECOVER]** then there is an option to load the pushrod.

NOTE: *Make sure to retract the pushrod before machining.*

7.2 More Information Online

For updated and supplemental information, including tips, tricks, maintenance procedures, and more, visit the Haas Service page at www.HaasCNC.com. You can also scan the code below with your mobile device to go directly to the Haas Service page:

Chapter 8: M-codes

8.1 Introduction

This chapter gives detailed descriptions of the M-codes that you use to program your machine.

8.1.1 List of M-codes

CAUTION: *The sample programs in this manual have been tested for accuracy, but they are for illustrative purposes only. The programs do not define tools, offsets, or materials. They do not describe workholding or other fixturing. If you choose to run a sample program on your machine, do so in Graphics mode. Always follow safe machining practices when you run an unfamiliar program.*

NOTE: *The sample programs in this manual represent a very conservative programming style. The samples are intended to demonstrate safe and reliable programs, and they are not necessarily the fastest or most efficient way to operate a machine. The sample programs use G-codes that you might choose not to use in more efficient programs.*

M-codes are miscellaneous machine commands that do not command axis motion. The format for an M-code is the letter M followed by two to three digits; for example `M03`.

Only one M-code is allowed per line of code. All M-codes take effect at the end of the block.

Code	Description	Page
`M00`	Stop Program	**394**
`M01`	Stop Program	**394**
`M02`	Program End	**394**
`M03`	Spindle On Fwd	**395**
`M04`	Spindle On Rev	**395**

Code	Description	Page
M05	Spindle Stop	**395**
M08 / M09	Coolant On / Off	**395**
M10 / M11	Chuck Clamp / Unclamp	**395**
M12 / M13	Auto Jet Air Blast On / Off (Optional)	**395**
M14 / M15	Main Spindle Brake On /Off (Optional C-Axis)	**396**
M17	Turret Rotation Fwd	**396**
M18	Turret Rotation Rev	**396**
M19	Orient Spindle (Optional)	**396**
M21	Tailstock Advance (Optional)	**397**
M22	Tailstock Retract (Optional)	**397**
M23	Chamfer Out of Thread On	**397**
M24	Chamfer Out of Thread Off	**397**
M30	End of Program and Reset	**397**
M31	Chip Auger Forward (Optional)	**398**
M33	Chip Auger Stop (Optional)	**398**
M35	Parts Catcher Part-Off Position	**398**
M36	Parts Catcher On (Optional)	**398**
M37	Parts Catcher Off (Optional)	**398**
M38 / M39	Spindle Speed Variation On / Off	**398**
M41 / M42	Low / High Gear (Optional)	**399**
M43	Turret Unlock (Service Use Only)	**399**
M44	Turret Lock (Service Use Only)	**399**
M51 - M56	Turn On Built-In M-Code Relay	**399**

Code	Description	Page
M133	Live Tool Fwd (Optional)	**410**
M134	Live Tool Rev (Optional)	**410**
M135	Live Tool Stop (Optional)	**410**
M138	Spindle Speed Variation On	**411**
M139	Spindle Speed Variation Off	**411**
M143	Secondary Spindle Forward (Optional)	**411**
M144	Secondary Spindle Reverse (Optional)	**411**
M145	Secondary Spindle Stop (Optional)	**411**
M146 / M147	Steady Rest Clamp / Unclamp (Optional)	**411**
M158 / M159	Mist Condenser On/Off	**411**
M219	Live Tool Orient (Optional)	**412**

M00 Stop Program

The `M00` code stops a program. It stops the axes, spindle, and turns off the coolant (including optional Through Spindle Coolant, Through Tool Air Blast, and Auto Air Gun / Minimum Quantity Lubrication). The next block after the `M00` is highlighted when viewed in the program editor. Press **[CYCLE START]** to continue program operation from the highlighted block.

M01 Optional Program Stop

`M01` works the same as `M00`, except the optional stop feature must be on. Press **[OPTION STOP]** to toggle the feature on and off.

M02 Program End

`M02` ends a program.

NOTE: *The most common way of ending a program is with an `M30`.*

M03 / M04 / M05 Spindle On Fwd/On Rev/Stop

`M03` turns spindle on in the forward direction. `M04` turns spindle on in the reverse direction. `M05` stops the spindle. For spindle speed, refer to `G96`/`G97`/`G50`.

M08 Coolant On / M09 Coolant Off

P - `M08 Pn`

`M08` turns on the optional coolant supply and `M09` turns it off. For High Pressure Coolant, see `M88`/`M89`.

An optional P-Code can now be specified along with an `M08`.

NOTE: *The machine is equipped with a Variable Frequency Drive for the coolant pump*

As long as no other G-Codes are in the same block, and t, this P-Code can be used to specify the desired pressure level of the coolant pump: `P0` = Low Pressure `P1` = Normal Pressure `P2` = High Pressure

NOTE: *If no P-Code is specified or the specified P-Code is out of range, then normal pressure will be used.*

NOTE: *If the machine is not equipped with a Variable Frequency Drive for the coolant pump, then the P-Code will have no effect.*

M10 Chuck Clamp / M11 Unclamp

`M10` clamps the chuck and `M11` unclamps it.

The direction of clamping is controlled by Setting 282 (refer to page**451** for more information).

M12 / M13 Auto Jet Air Blast On/Off (Optional)

`M12` and `M13` activate the optional Auto Air Jet. `M12` turns the air blast on and `M13` turns the air blast off. `M12 Srrr Pnnn` (`rrr` in RPM and `nnn` is in milliseconds) turns the air blast on for the specified time, rotates the spindle at the specified speed while the air blast is on, then turns off both the spindle and the air blast automatically. The air blast command for the secondary spindle is `M112`/`M113`.

M14 / M15 Main Spindle Brake On/Off (Optional C-Axis)

These M Codes are used for machines equipped with the optional C-Axis. `M14` applies a caliper-style brake to hold the main spindle, while `M15` releases the brake.

M17 / M18 Turret Rotation Fwd/Rev

`M17` and `M18` rotate the turret in the forward (`M17`) or reverse (`M18`) direction when a tool change is made. The following `M17` program code causes the tool turret to move forward to tool 1 or reverse to tool 1 if an `M18` is commanded.

```
N1 T0101 M17 (Forward) ;
```

```
N1 T0101 M18 (Reverse) ;
```

An `M17` or `M18` stays in effect for the remainder of the program.

NOTE: *Setting 97, Tool Change Direction, must be set to `M17/M18`.*

M19 Orient Spindle (Optional)

`M19` adjusts the spindle to a fixed position. The spindle only orients to the zero position without the optional `M19` orient spindle feature.

The orient spindle function allows `P` and `R` address codes. For example, `M19 P270.` orients the spindle to 270 degrees. The `R` value allows the programmer to specify up to two decimal places; for example, `M19 R123.45`. View the angle in the **`Current Commands Tool Load`** screen.

`M119` positions the secondary spindle (DS lathes) the same way.

NOTE: *The range for `M19` or `M119` is 0 to 360 degrees. If a negative value if given, it will be ignored and spindle will orient to 0 degrees.*

Spindle orientation is dependent on the mass, diameter, and length of the workpiece and/or the workholding (chuck). Contact the Haas Applications Department if any unusually heavy, large diameter, or long configuration is used.

M21 / M22 Tailstock Advance/Retract (Optional)

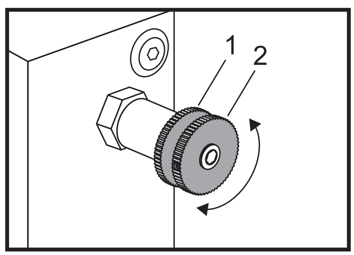
`M21` and `M22` position the tailstock. `M21` uses Settings 341and 342 to move to the Tailstock Advance Distance`M22` uses Setting 105 to move the tailstock to the Retract Point.

NOTE: *ST10 does not use any settings (105, 341, 342).*

Adjust pressure using the valves on the HPU (except ST-40, which uses Setting 241 to define hold pressure). For information on proper ST tailstock pressure, refer to pages **142** and **143**.

CAUTION: *Do not use an `M21` in the program if the tailstock is positioned manually. If this is done, the tailstock backs away from the workpiece and then repositions against the workpiece, which may cause the workpiece to drop.*

F8.1: Set Screw Hold Pressure Valve: [1] Locking knob, [2] Adjustment knob.



M23 / M24 Chamfer Out of Thread On/Off

`M23` commands the control to execute a chamfer at the end of a thread executed by `G76` or `G92`. `M24` commands the control not to perform chamfering at the end of the threading cycles (`G76` or `G92`). An `M23` remains in effect until changed by `M24`, likewise for `M24`. Refer to Settings 95 and 96 to control the chamfer size and angle. `M23` is the default at power-up and when the control is reset.

M30 Program End and Reset

`M30` stops a program. It stops the spindle and turns off the coolant and the program cursor returns to the start of the program.

NOTE: *`M30` no longer cancels tool length offsets.*

M31 / M33 Chip Auger Forward/Stop (Optional)

`M31` starts the optional chip auger motor in the forward direction (the direction that moves the chips out of the machine). The auger does not turn if the door is open. It is recommended that the chip auger be used intermittently. Continuous operation causes the motor to overheat. Settings 114 and 115 control the auger duty cycle times.

`M33` stops auger motion.

M35 Parts Catcher Part-Off Position

The `M35` code allows for cycle time savings instead of fully extend/retract the parts catcher for every part, you can command `M35` to position the parts catcher to the part-off position. Then when the part is finished command a `M36` to catch the part. Then command a `M37` to retract the parts catcher to its home position.

This function has been added to the Parts Catcher device page. To access the page press the **[CURRENT COMMANDS]** button then go to the **`Devices`** tab.

M36 / M37 Parts Catcher On/Off (Optional)

`M36` rotates the parts catcher into position to catch a part. `M37` rotates the parts catcher out of the work envelope.

NOTE: *The sub-spindle part ejector is a spring loaded manual device that ejects the part when the jaws are opened. The part ejector is not actuated by use of an M-code.*

M38 / M39 Spindle Speed Variation On/Off

Spindle Speed Variation (SSV) allows the operator to specify a range within which the spindle speed continuously varies. This is helpful in suppressing tool chatter, which can lead to an undesirable part finish and/or damage to the cutting tool. The control varies the spindle speed based on Settings 165 and 166. For example, in order to vary spindle speed +/- 50 RPM from its current commanded speed with a duty cycle of 3 seconds, set Setting 165 to 50 and Setting 166 to 30. Using these settings, the following program varies the spindle speed between 950 and 1050 RPM after the `M38` command.

M38/39 Program Example

```
%
o60381 (M38/39-SSV-SPINDLE SPEED VARIATION) ;
(G54 X0 Y0 is at the center of rotation) ;
(Z0 is on the face of the part) ;
(BEGIN PREPARATION BLOCKS) ;
```

```
T101 (Select tool and offset 1) ;
G00 G18 G20 G40 G80 G99 (Safe startup) ;
S1000 M3 (Turn spindle CW at 1000 RPM) ;
G04 P3. (Dwell for 3 seconds) ;
M38 (SSV ON) ;
G04 P60. (Dwell for 60 seconds) ;
M39 (SSV OFF) ;
G04 P5. (Dwell for 5 seconds) ;
G00 G53 X0 (X home) ;
G53 Z0 (Z home & C unwind) ;
M30 (End program) ;
%
```

The spindle speed continuously varies with a duty cycle of 3 seconds until an `M39` command is found. At that point the machine comes back to its commanded speed and the SSV mode is turned off.

A program stop command such as `M30` or pressing **[RESET]** also turns SSV Off. If the RPM swing is larger than the commanded speed value, any negative RPM values (below zero) translates into an equivalent positive value. The spindle, however, is not allowed to go below 10 RPM when SSV mode is active.

Constant Surface Speed: When Constant Surface Speed (`G96`) is activated (which calculates spindle speed) the `M38` command alters that value using Settings 165 and 166.

Threading Operations: `G92`, `G76` and `G32` allow the spindle speed to vary in SSV mode. This is not recommended due to possible thread lead errors caused by mismatched acceleration of the spindle and the Z-axis.

Tapping cycles: `G84`, `G184`, `G194`, `G195`, and `G196` are executed at their commanded speed and SSV is not applied.

M41 / M42 Low/High Gear (Optional)

On machines with a transmission, `M41` selects low gear and `M42` selects high gear.

M43 / M44 Turret Unlock/Lock (Service Use Only)

For Service use only.

M51-M56 Turn On Built-In M-Code Relay

`M51` through `M56` are used to control M-code relays. Each M-code turns on one relay and leaves it active. Use`M61` through `M66` to turn these off. **[RESET]** turns off all of these relays.

Refer to `M121` through `M126` on page **408** for details on the M-code relays.

M59 Turn On Output Relay

P - Discrete output relay number.

`M59` turns on a discrete output relay. An example of its usage is `M59 Pnnn`, where `nnn` is the relay number being turned on.

When using Macros, `M59 P90` does the same thing as using the optional macro command `#12090=1`, except that it is processed at the end of the line of code.

Built-In M-Code Relays	8M PCB Relay Bank 1 (JP1)	8M PCB Relay Bank 2 (JP2)	8M PCB Relay Bank 3 (JP3)
P114 (M121)	P90	P103	P79
P115 (M122)	P91	P104	P80
P116 (M123)	P92	P105	P81
P113 (M124)	P93	P106	P82
P112 (M125)	P94	P107	P83
P4 (M126)	P95	P108	P84
-	P96	P109	P85
-	P97	P110	P86

M61-M66 Turn Off Built-In M-Code Relay

The `M61` through `M66` codes are optional for user interfaces. They turn off one of the relays. Use `M51`-`M56` to turn these on. **[RESET]** turns off all of these relays.

See `M121`-`M126` for details on the M-code relays.

M69 Turn Off Output Relay

P - Discrete output relay number from 0 to 255.

`M69` turns off a relay. An example of its usage is `M69 P12nnn`, where `nnn` is the number of the relay being turned off.

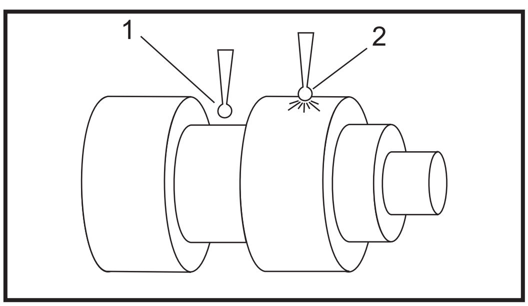
When using Macros, `M69 P12003` does the same thing as using the optional macro command `#12003=0`, except that it is processed in the same order as axis motion.

Built-In M-Code Relays	8M PCB Relay Bank 1 (JP1)	8M PCB Relay Bank 2 (JP2)	8M PCB Relay Bank 3 (JP3)
P114 (M121)	P90	P103	P79
P115 (M122)	P91	P104	P80
P116 (M123)	P92	P105	P81
P113 (M124)	P93	P106	P82
P112 (M125)	P94	P107	P83
P4 (M126)	P95	P108	P84
-	P96	P109	P85
-	P97	P110	P86

M78 / M79 Alarm if Skip Signal Found/Not Found

This M-code is used with a probe. `M78` generates an alarm if a programmed skip function (`G31`) receives a signal from the probe. This is used when a skip signal is not expected, and may indicate a probe crash. `M79` generates an alarm if a programmed skip function (`G31`) did not receive a signal from the probe. This is used when the lack of the skip signal means a probe positioning error. These codes can be placed on the same line as the skip G-code or in any block after.

F8.2: `M78`/`M79` Alarm if Skip Signal Found/Not Found: [1] Signal not found, [2] Signal found.



M85 / M86 Automatic Door Open/Close (Optional)

`M85` opens the Auto Door and `M86` closes it. The control pendant beeps when the door is in motion.

M90 Fixture Clamp Input ON / M91 Fixture Clamp Input OFF

The `M90` M-code enables fixture clamp input monitoring when setting 276 has a valid input number greater than 0. If variable #709 or #10709 = 1 and the spindle is commanded on, the machine will generate alarm: 973 Fixture Clamp Incomplete.

The `M91` M-code disables the fixture clamp input monitoring.

M88 / M89 High Pressure Coolant On/Off (Optional)

`M88` turns on the high pressure coolant option, and `M89` turns the coolant off. Use `M89` to turn off High Pressure coolant during program execution before rotating the tool turret.

DANGER: *Turn off High Pressure Coolant before performing a tool change.*

M95 Sleep Mode

Sleep mode is a long dwell. The format of the `M95` command is: `M95 (hh:mm)`.

The comment immediately following `M95` must contain the duration, in hours and minutes, that you want the machine to sleep. For example, if the current time were 6 p.m. and you want the machine to sleep until 6:30 a.m. the next day, command `M95 (12:30)`. The line(s) after `M95` should be axis moves and spindle warm-up commands.

M96 Jump If No Signal

P - Program block to go to when conditional test is met
Q - Discrete input variable to test (0 to 63)

This code tests a discrete input for 0 (off) status. This is useful for checking the status of automatic work holding or other accessories that generate a signal for the control. The `Q` value must be in the range 0 to 63, which corresponds to the inputs on the diagnostic display (The upper left input is 0 and the lower right is input 63. When this program block is executed and the input signal specified by `Q` has a value of 0, the program block `Pnnnn` is performed (the `Pnnnn` line must be in the same program).

```
N05 M96 P10 Q8 (Test input #8, Door Switch, until closed) ;
N10 (Start of program loop) ;
. ;
. (Program that machines part) ;
. ;
N85 M21 (Execute an external user function) ;
N90 M96 P10 Q27 (Loop to N10 if spare input [#27] is 0) ;
N95 M30 (If spare input is 1 then end program) ;
```

M97 Local Subprogram Call

This code calls a subprogram (subprogram) referenced by a line number (`N`) within the same program. A `Pnn` code is required and must match a line number within the same program. This is useful for subprograms within a program as it does not require a separate program. The subprogram must end with an `M99`. An `Lnn` code in the `M97` block will repeat the subprogram call `nn` times.

```
%
O69701 (M97 LOCAL SUBPROGRAM CALL) ;
M97 P1000 L2 (L2 will run the N1000 line twice) ;
M30 ;
N1000 G00 G55 X0 Z0 (N line that will run after M97 P1000 is
run) ;
S500 M03 ;
G00 Z-.5 ;
G01 X.5 F100. ;
G03 ZI-.5 ;
G01 X0 ;
Z1. F50. ;
G28 U0 ;
G28 W0 ;
M99 ;
%
```

M98 Subprogram Call

P - The subprogram number to run
L - Repeats the subprogram call (1-99) times.
(<PATH>) - The Subprogram’s directory path

`M98` calls a subprogram in the format `M98 Pnnnn`, where `Pnnnn` is the number of the program to call, or `M98 (<path>/Onnnnn)`, where <path> is the device path that leads to the subprogram.

The subprogram must contain an `M99` to return to the main program. You can add an `Lnn` count to the `M98` block `M98` to call the subprogram `nn` times before continuing to the next block.

When your program calls an `M98` subprogram, the control looks for the subprogram in the main program’s directory. If the control cannot find the subprogram, it then looks in the location specified in Setting 251. Refer to page **208** for more information. An alarm occurs if the control cannot find the subprogram.

`M98` Example:

The subprogram is a separate program (`O00100`) from the main program (`O00002`).

```
%
O00002 (PROGRAM NUMBER CALL);
M98 P100 L4 (CALLS O00100 SUB 4 TIMES) ;
M30 ;
%
%
O00100 (SUBPROGRAM);
M00 ;
M99 (RETURN TO MAIN PROGRAM) ;
%
```

```
%
O00002 (PATH CALL);
M98 (USB0/O00001.nc) L4 (CALLS O00100 SUB 4 TIMES) ;
M30 ;
%
%
O00100 (SUBPROGRAM);
M00 ;
M99 (RETURN TO MAIN PROGRAM) ;
%
```

M99 Subprogram Return or Loop

This code has three main uses:

1. An `M99` is used at the end of a subprogram, local subprogram, or macro to return back to the main program.
2. An `M99 Pnn` jumps the program to the corresponding `Nnn` in the program.
3. An `M99` in the main program causes the program to loop back to the beginning and run until **[RESET]** is pressed.

Programming Notes - You can simulate Fanuc behavior by using the following code:

	Haas	Fanuc
Calling program:	O0001	O0001
	...	...
	N50 M98 P2	N50 M98 P2

	Haas	Fanuc
	N51 M99 P100	...
	...	N100 (continue here)
	N100 (continue here)	...
	...	M30
	M30	
Subprogram:	O0002	O0002
	M99	M99 P100

`M99` With Macros - If the machine is equipped with the optional macros, you can use a global variable and specify a block to jump to by adding `#nnnnn = dddd` in the subprogram and then using `M99 P#nnnnn` after the subprogram call.

M104 / M105 Probe Arm Extend/Retract (Optional)

The optional tool setting probe arm is extended and retracted using these M-codes.

M109 Interactive User Input

P - A number in the range (500-549) representing the macro variable of the same name.

This M code allows a G-code program to place a short prompt (message) on the screen. A macro variable in the range 500 through 549 must be specified by a `P` code. The program can check for any character that can be entered from the keyboard by comparing with the decimal equivalent of the ASCII character.

T8.1: Values for ASCII Characters

32		space	59	;	semicolon
33	!	exclamation mark	60	<	less than
34	"	double quotation mark	61	=	equals
35	#	number sign	62	>	greater than
36	$	dollar sign	63	?	question mark
37	%	percent sign	64	@	at sign

38	&	ampersand	65-90	A-Z	capitol letters
39	’	closed single quote	91	[	open square bracket
40	(	open parenthesis	92	\	backslash
41	)	close parenthesis	93	]	closed square bracket
42	*	asterisk	94	^	carrot
43	+	plus sign	95	_	underscore
44	,	comma	96	‘	open single quote
45	-	minus sign	97-122	a-z	lowercase letters
46	.	period	123	{	open curly bracket
47	/	slash	124	|	vertical bar
48-57	0-9	numbers	125	}	closed curly bracket
58	:	colon	126	~	tilde

The following sample program asks the user a Yes or No question, then wait for either a `Y` or an `N` to be entered. All other characters are ignored.

```
%
o61091 (57 M109_01 Interactive User Input) ;
N1 #501= 0. (Clear the variable) ;
N5 M109 P501 (Sleep 1 min?) ;
IF [ #501 EQ 0. ] GOTO5 (Wait for a key) ;
IF [ #501 EQ 89. ] GOTO10 (Y) ;
IF [ #501 EQ 78. ] GOTO20 (N) ;
GOTO1 (Keep checking) ;
N10 (A Y was entered) ;
M95 (00:01) ;
GOTO30 ;
N20 (An N was entered) ;
G04 P1. (Do nothing for 1 second) ;
N30 (Stop) ;
M30 ;
%
```

The following sample program asks the user to select a number, then wait for a `1`, `2`, `3`, `4` or a `5` to be entered; all other characters are ignored.

```
%
O61092 (58 M109_02 Interactive User Input) ;
N1 #501= 0 (Clear Variable #501) ;
(Variable #501 will be checked) ;
(Operator enters one of the following selections) ;
N5 M109 P501 (1,2,3,4,5) ;
IF [ #501 EQ 0 ] GOTO5 ;
(Wait for keyboard entry loop until entry) ;
(Decimal equivalent from 49-53 represent 1-5) ;
IF [ #501 EQ 49 ] GOTO10 (1 was entered go to N10) ;
IF [ #501 EQ 50 ] GOTO20 (2 was entered go to N20) ;
IF [ #501 EQ 51 ] GOTO30 (3 was entered go to N30) ;
IF [ #501 EQ 52 ] GOTO40 (4 was entered go to N40) ;
IF [ #501 EQ 53 ] GOTO50 (5 was entered go to N50) ;
GOTO1 (Keep checking for user input loop until found) ;
N10 ;
(If 1 was entered run this sub-routine) ;
(Go to sleep for 10 minutes) ;
#3006= 25 (Cycle start sleeps for 10 minutes) ;
M95 (00:10) ;
GOTO100 ;
N20 ;
(If 2 was entered run this sub routine) ;
(Programmed message) ;
#3006= 25 (Programmed message cycle start) ;
GOTO100 ;
N30 ;
(If 3 was entered run this sub routine) ;
(Run sub program 20) ;
#3006= 25 (Cycle start program 20 will run) ;
G65 P20 (Call sub-program 20) ;
GOTO100 ;
N40 ;
(If 4 was entered run this sub routine) ;
(Run sub program 22) ;
#3006= 25 (Cycle start program 22 will be run) ;
M98 P22 (Call sub program 22) ;
GOTO100 ;
N50 ;
(If 5 was entered run this sub-routine) ;
(Programmed message) ;
#3006= 25 (Reset or cycle start will turn power off) ;
#1106= 1 ;
```

```
N100 ;
M30 ;
%
```

M110 / M111 Secondary Spindle Chuck Clamp/Unclamp (Optional)

These M codes will clamp and unclamp the secondary spindle chuck. OD / ID clamping is set with Setting 122.

NOTE: *The sub-spindle part ejector is a spring loaded manual device that ejects the part when the jaws are opened. The part ejector is not actuated by use of an M-code.*

M112 / M113 Secondary Spindle Air Blast On/Off (Optional)

`M112` turns on the secondary spindle air blast. `M113` turns the secondary spindle air blast off. `M112 Srrr Pnnn` (`rrr` is in RPM and `nnn` is in milliseconds) turns the air blast on for the specified time, rotates the spindle at the specified speed while the air blast is on, then turns off both the spindle and the air blast automatically.

M114 / M115 Secondary Spindle Brake On/Off (Optional)

`M114` applies a caliper-style brake to hold the secondary spindle, while `M115` releases the brake.

M119 Secondary Spindle Orient (Optional)

This command orients the secondary spindle (DS lathes) to the zero position. A `P` or `R` value is added to position the spindle to a specific position. A `P` value positions the spindle to that whole degree (e.g. `P120` is 120°). An `R` value positions the spindle to a fraction of a degree (e.g. `R12.25` is 12.25°). The format is: `M119 Pxxx`/`M119 Rxx.x`. The spindle angle is viewed in the Current Commands Tool Load screen.

M121-M126 Built-In M-codes Relays with M-Fin

The `M121` through `M126` codes are built-in M-code relays. They turns on a relay, pauses the program, and waits for an external M-Fin signal.

When the control receives the M-Fin signal, the relay turns off and the program continues.**[RESET]** terminates any operation that is hung-up waiting for M-fin.

M129 Turn On M-Code Relay with M-Fin

P - Discrete output relay number.

`M129` turns on a relay, pauses the program, and waits for an external M-Fin signal. An example of its usage is `M129 Pnnn`, where `nnn` is the relay number being turned on.

Built-In M-Code Relays	8M PCB Relay Bank 1 (JP1)	8M PCB Relay Bank 2 (JP2)	8M PCB Relay Bank 3 (JP3)
P114 (M121)	P90	P103	P79
P115 (M122)	P91	P104	P80
P116 (M123)	P92	P105	P81
P113 (M124)	P93	P106	P82
P112 (M125)	P94	P107	P83
P4 (M126)	P95	P108	P84
-	P96	P109	P85
-	P97	P110	P86

When the control receives the M-Fin signal, the relay turns off and the program continues. **[RESET]** stops any operation waiting for a relay-activated accessory to finish.

M130 Display Media / M131 Cancel Display Media

M130 Lets you display video and still images during program execution. Some examples of how you can use this feature are:

- Providing visual cues or work instructions during program operation
- Providing images to aid part inspection at certain points in a program
- Demonstrating procedures with video

The correct command format is `M130(file.xxx)`, where `file.xxx` is the name of the file, plus the path, if necessary. You can also add a second comment in parentheses to appear as a comment at the top of the media window.

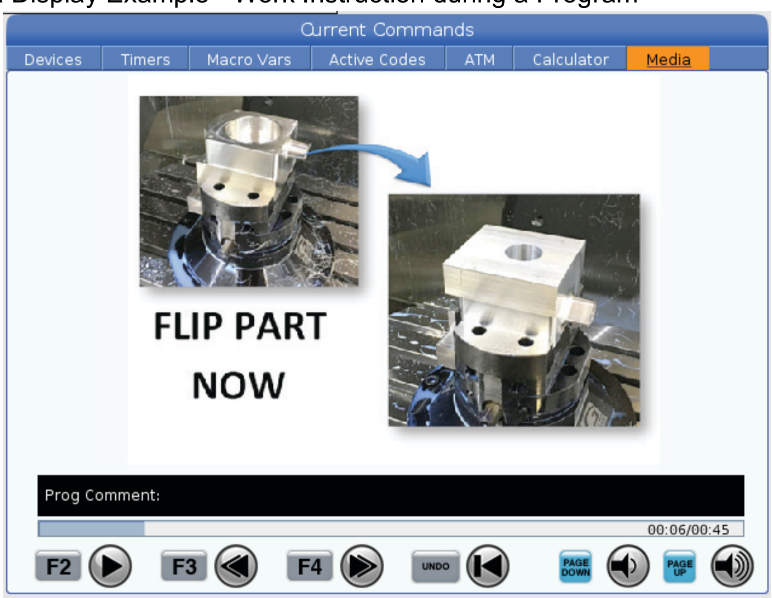
NOTE: *`M130` uses the subprogram search settings, Settings 251 and 252 in the same way that `M98` does. You can also use the `Insert Media File` command in the editor to easily insert an `M130` code that includes the filepath. Refer to page **160** for more information.*

Permitted file formats are MP4, MOV, PNG, and JPEG.

NOTE: *For the fastest loading times, use files with pixel dimensions divisible by 8 (most unedited digital images have these dimensions by default), and a maximum pixel size of 1920 x 1080.*

Your media appears in the Media tab under Current Commands. The media displays until the next **`M130`** displays a different file, or **`M131`** clears the media tab contents.

F8.3: Media Display Example - Work Instruction during a Program



M133 / M134 / M135 Live Tool Fwd/Rev/Stop (Optional)

`M133` turns the live tool spindle in the forward direction. `M134` turns the live tool spindle in the reverse direction. `M135` stops the live tool spindle.

Spindle speed is controlled with a `P` address code. For example, `P1200` would command a spindle speed of 1200 RPM.

M138 / M139 Spindle Speed Variation On/Off

Spindle Speed Variation (SSV) lets you specify a range within which the spindle speed continuously varies. This is helpful in suppressing tool chatter, which can lead to an undesirable part finish and/or damage to the cutting tool. The control varies the spindle speed based on Settings 165 and 166. For example, to vary spindle speed +/- 100 RPM from its current commanded speed with a duty cycle of 1 seconds, set Setting 165 to 100 and Setting 166 to 1.

The variation you use depends on the material, tooling, and the characteristics of you application, but 100 RPM over 1 second is a good starting point.

You can override the values of settings 165 and 166 using P and E address codes when used with M138. Where P is SSV Variation (RPM) and E is the SSV Cycle (Sec). See example below:

```
M138 P500 E1.5 (Turn SSV On, vary the speed by 500 RPM, cycle
every 1.5 seconds);

M138 P500(Turn SSV on, vary the speed by 500, cycle based on
setting 166);

M138 E1.5 (Turn SSV on, vary the speed by setting 165, cycle
every 1.5 seconds);
```

M138 is independent of spindle commands; once commanded, it is active even when the spindle is not turning. Also, M138 remains active until canceled with M139, or at M30, Reset, or Emergency Stop.

M143 / M144 / M145 Secondary Spindle Fwd/Rev/Stop (Optional)

`M143` turns the secondary spindle in the forward direction. `M144` turns the secondary spindle in the reverse direction. `M145` stops the secondary spindle

The subspindle speed is controlled with an `P` address code, for example, `P1200` commands a spindle speed of 1200 RPM.

M146 Steady Rest Clamp / M147 Steady Rest Unclamp

`M146` clamps the steady rest and `M147` unclamps it.

M158 Mist Condenser On / M159 Mist Condenser Off

`M158` turns on the Mist Condenser, and `M159` turns off the Mist Condenser.

NOTE: *The mist condenser will run continuously if a program is running with axis motion.*

NOTE: *An `M158` in MDI will activate the mist condenser for 10 seconds, after this the mist condenser will turn OFF. If continuous operation is required the program will require feed motion of at least one axis.*

NOTE: *If you want the mist condenser to remain ON, then go to CURRENT COMMANDS>DEVICES>MECHANISMS>MIST CONDENSER and press* **[F2]** *to turn it on*

M219 Live Tool Orient (Optional)

P - Number of degrees (0 - 360)
R - Number of degrees with two decimal places (0.00 - 360.00).

`M219` adjusts the live tool to a fixed position. An `M219` orients the spindle to the zero position. The orient spindle function allows P and R address codes. For example:

```
M219 P270. (orients the live tool to 270 degrees) ;
```

The `R`-value allows the programmer to specify up to two decimal places; for example:

```
M219 R123.45 (orients the live tool to 123.45 degrees) ;
```

8.2 More Information Online

For updated and supplemental information, including tips, tricks, maintenance procedures, and more, visit the Haas Service page at www.HaasCNC.com. You can also scan the code below with your mobile device to go directly to the Haas Service page: