

MACHINING

Level - III

Learning Guide 4

**Unit of Competence: Perform Advanced Milling
CNC Operations**

**Module Title: Performing Advanced Milling CNC
Operations**

LG Code: IND MAC3 04 0217

<i>Ethiopian TVET Program</i>	STEP-GIZ	<i>CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations</i>	<i>July 2020</i>	<i>Page 1 of 73</i>
-----------------------------------	-----------------	--	------------------	---------------------

Instruction Sheet	Learning Guide #1
--------------------------	--------------------------

This learning guide is developed to provide you the necessary information regarding the following **content coverage** and topics:

1. Determine job requirements
2. Write or load CNC milling machine program
3. Perform milling operations

This guide will also assist you to attain the learning outcome stated in the cover page. Specifically, **upon completion of this Learning Guide, you will be able to:**

- Cutting speed and feed rate are calculated based on work piece and cutting tool material standard
- written in standard CNC milling operation code format and in accordance with standard operating 2D and 3D.procedures
- CNC milling operations are performed to produce component according to drawing specifications

Learning Instructions:

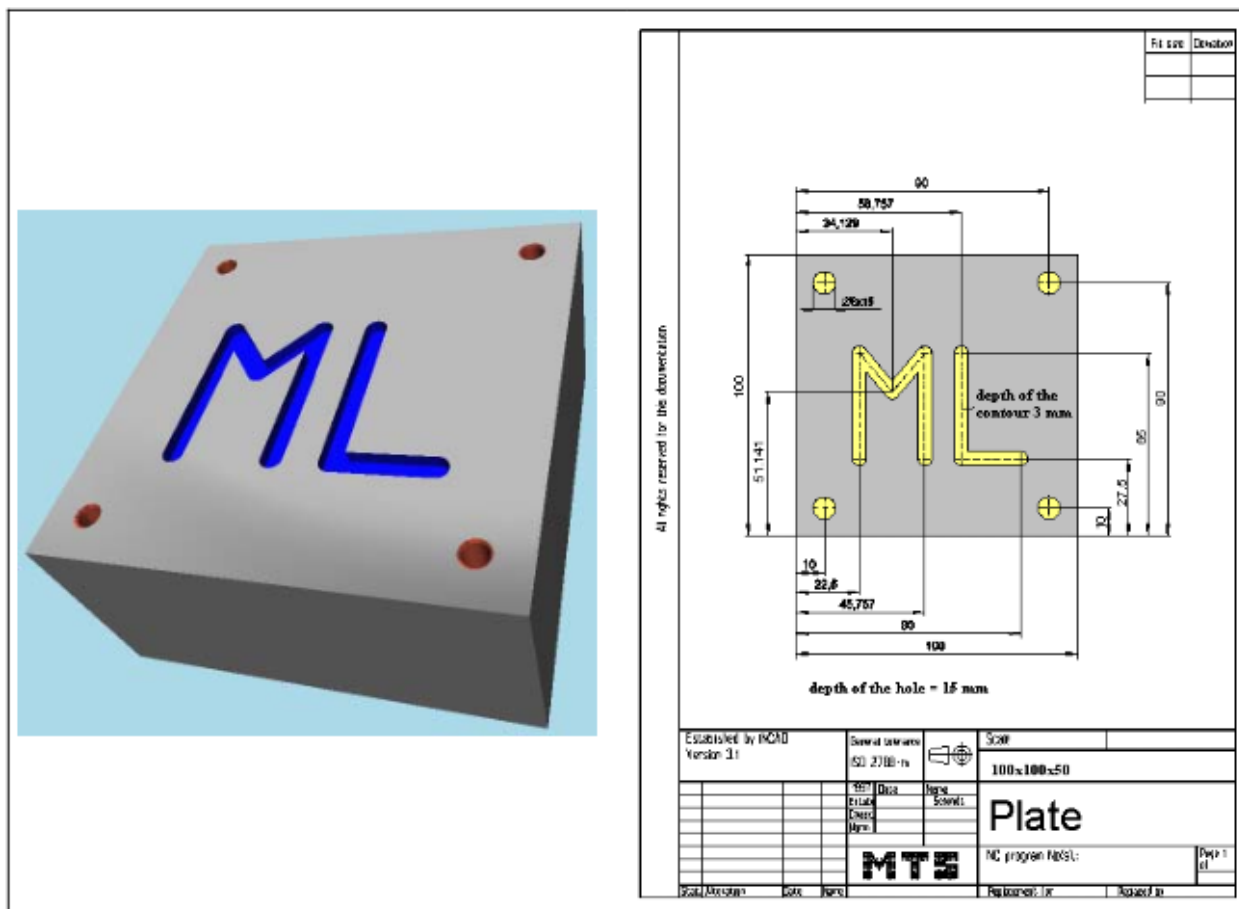
1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below 3 to 73.
3. Read the information written in the information “Sheet.
4. Accomplish the “Self-check test.
5. Do the “LAP test.

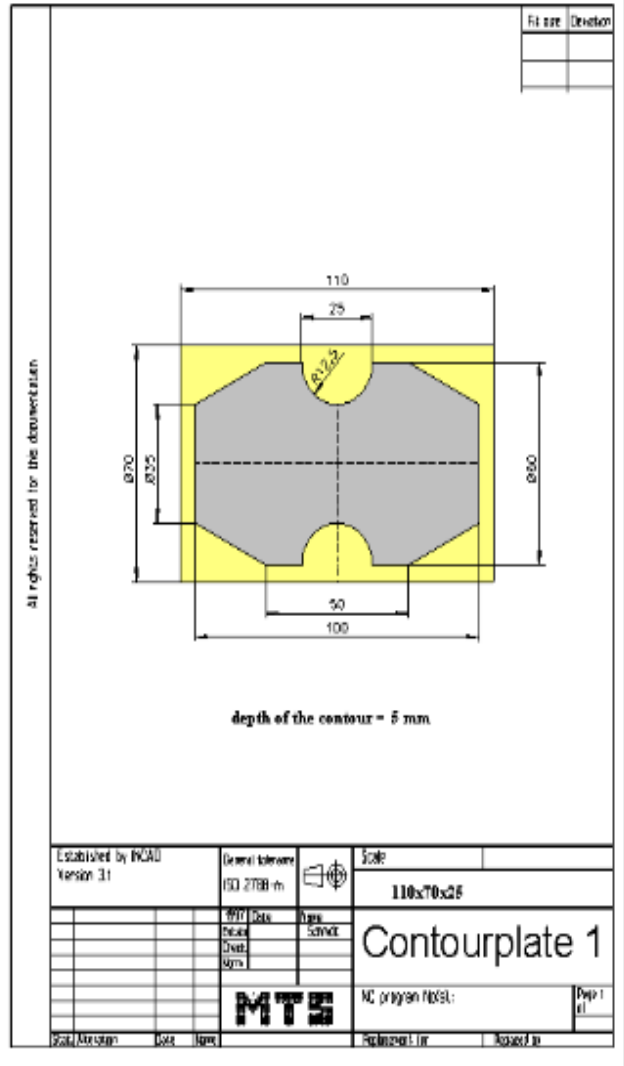
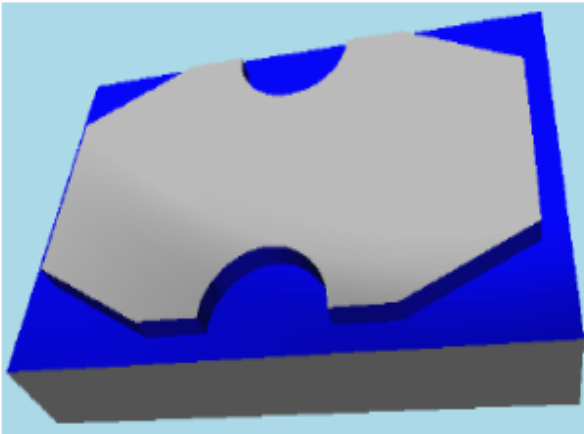
Determine job requirements

Introduction:

In CNC Milling Machining, operator or machinist must have to understand the drawing, interpret and analyze it as a reference to produce program manual script prior to manufacture the part. The following are guide before machining a part:

1. Drawings need to be interpreted to produce component as per specifications.
2. The sequence of operation should be established as well to determine what steps in producing the component according to specification.
3. Cutting tools are selected according to the requirements of the operation.
4. Cutting speed and feed rate calculated based on work- piece and cutting tool material.
5. Process / job adjustment sheets are filled up with relevant machine, tool and raw material data.





Coordinate System

Reference Point

Reference point is a fixed position on CNC machine tool, which is determined by cams and measuring system. Generally, it is used when the tool is required to exchange or the coordinate system is required to set.

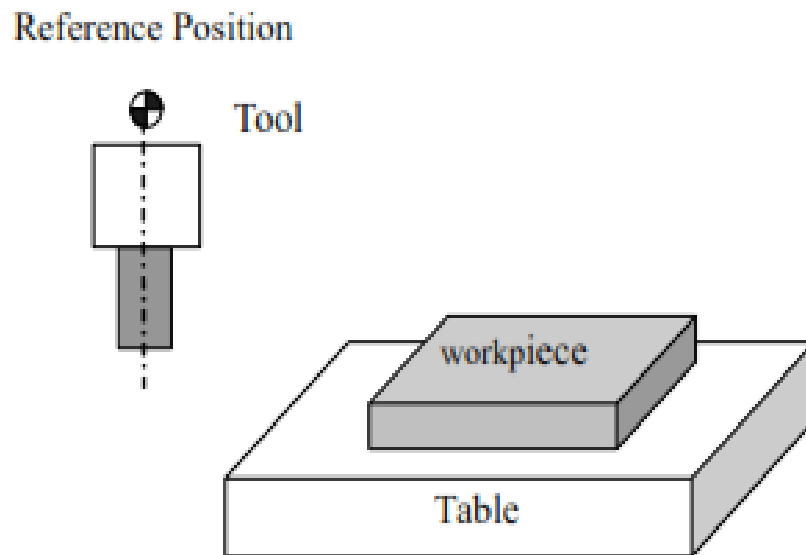


Figure 1.6 Reference Point

There are two ways to move to the reference point:

1. Manual reference position return: The tool is moved to the reference point by operating the button on the machine control panel. It is only used when the machine is turned on.
2. Automatic reference position return: It is used after the manual reference position return has been used. In this manual, this would be introduced.

Machine Coordinate System

The coordinate system is set on a CNC machine tool. Figure 1.7 is a machine coordinate system of milling machine, and shows the direction of axes:

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 5 of 73
---------------------------	-----------------	---	-----------	--------------

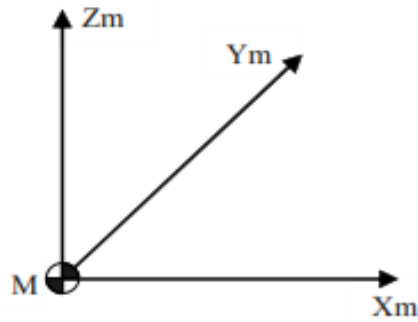


Figure 1.7 Machine Coordinate System

In general, three basic linear coordinate axes of motion are X, Y, Z. Moreover, X, Y, Z axis of rotation is named as A, B, C correspondently. Due to different types of milling machine, the axis direction can be decided by following the rule – “three finger rule” of the right hand.

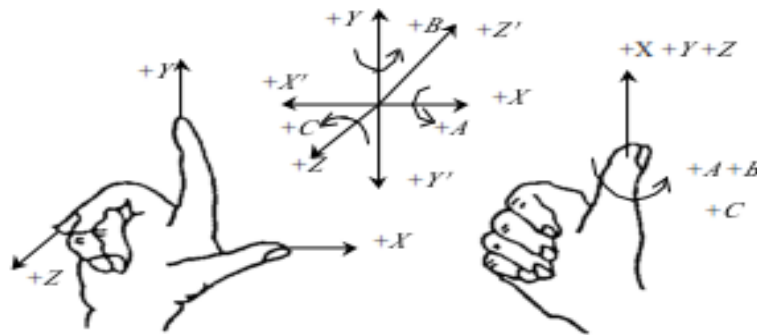


Figure 1.8 “three finger rule”

- The thumb points the X axis. X axis controls the cross motion of the cutting tool. “+X” means that the tool is away from the spindle centerline
- The index points the Y axis. Y axis is usually a virtual axis.
- The middle finger points the Z axis. Z axis controls the motion of the cutting tool. “+Z” means that the tool is away from the spindle.

Workpiece Coordinate System

The coordinate system is set on a workpiece. The data in the NC program is from the workpiece coordinate system.

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 6 of 73
---------------------------	-----------------	---	-----------	--------------

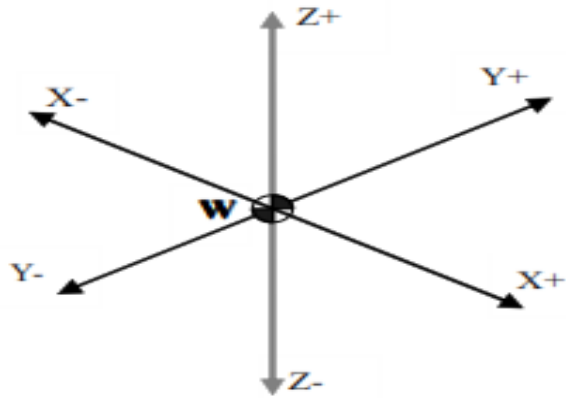


Figure 1.9 Workpiece Coordinate System

Example: Those three points can be defined on workpiece coordinate system:

P1 corresponds to X20 Y35

P2 corresponds to X50 Y60

P3 corresponds to X70 Y20

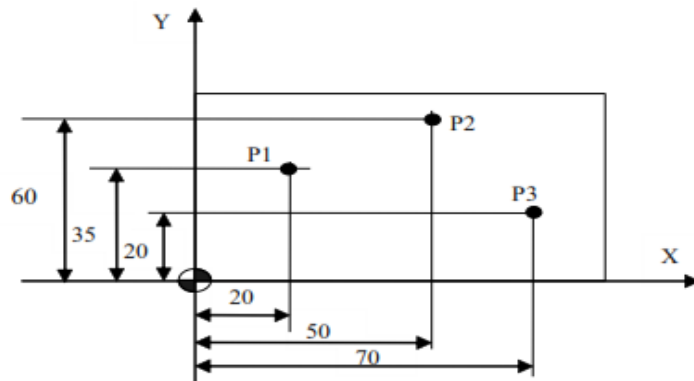


Figure 1.10 Example of defining points on workpiece coordinate system

Setting Two Coordinate Systems at the Same Position

When a workpiece is set on the table, the positional relation between machine coordinate system and workpiece coordinate system are set.

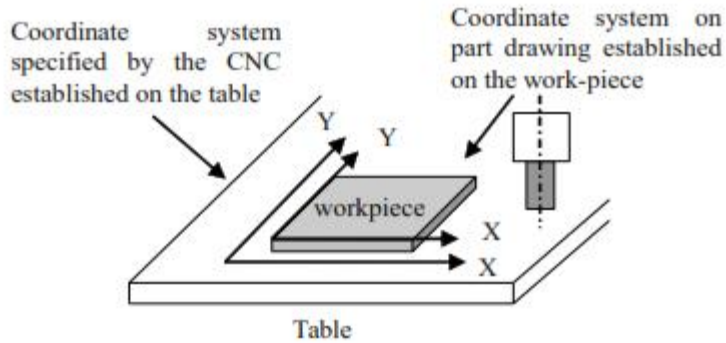


Figure 1.11 Setting two coordinate systems at the same position

According to the command program based on the workpiece coordinate system, the tool moves on the coordinate system specified by CNC, and cuts a workpiece.

Absolute Commands

The absolute dimension describes a point at “the distance from zero point of the coordinate system”.

Example: These three point in absolute dimensions are the following:
P1 corresponds to

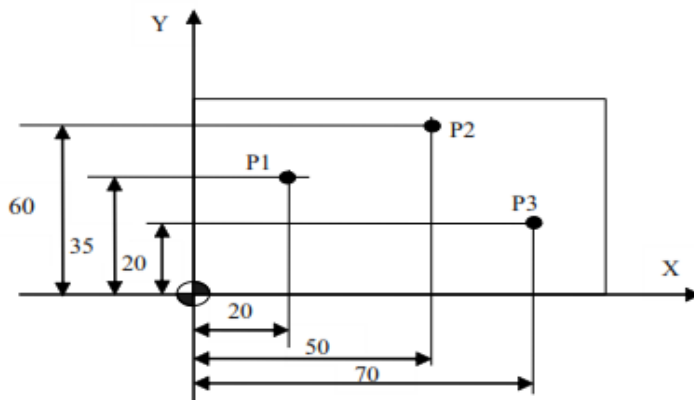


Figure 1.12 Absolute Dimension

Incremental Commands

The incremental dimension describes a distance from the previous tool position to the next tool position.

Example: These three point in incremental dimensions are the following:

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 8 of 73
------------------------	-----------------	---	-----------	--------------

P1 corresponds to X20 Y35 //with reference to the zero point
P2 corresponds to X30 Y20 //with reference to P1
P3 corresponds to X20 Y-35 //with reference to P2

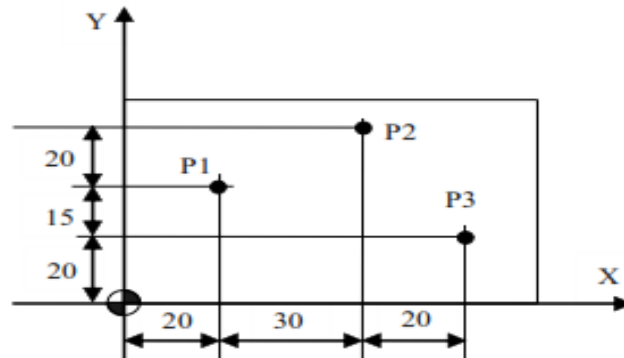


Figure 1.13 Incremental Dimension

Polar Coordinates

Beside the “Cartesian coordinate system”, another way to specify coordinates is “polar coordinates”. The polar coordinate method is useful only if there is radius and angle measurements on a workpiece.

Example: Two points P1 and P2 with reference to the pole are described as follows.

P1 corresponds to radius=100 plus angle=30°

P2 corresponds to radius=60 plus angle=75°

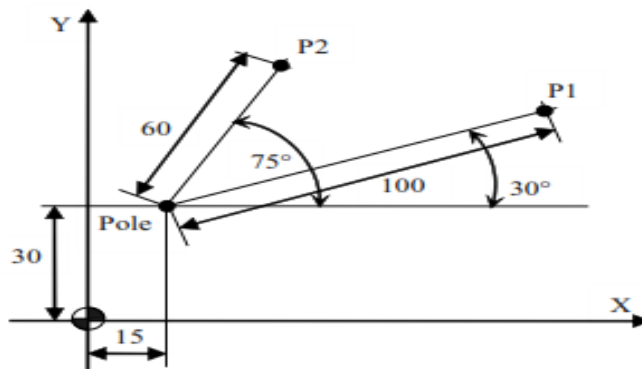


Figure 1.14 Polar Coordinates

Spindle Speed Function

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 9 of 73
---------------------------	-----------------	---	-----------	--------------

The cutting speed (v) refers to the speed of the tool with respect to the workpiece when the workpiece is cut. The unit of the cutting speed is m/min. As for the CNC, the cutting speed can be specified by the spindle speed (N) in min^{-1}

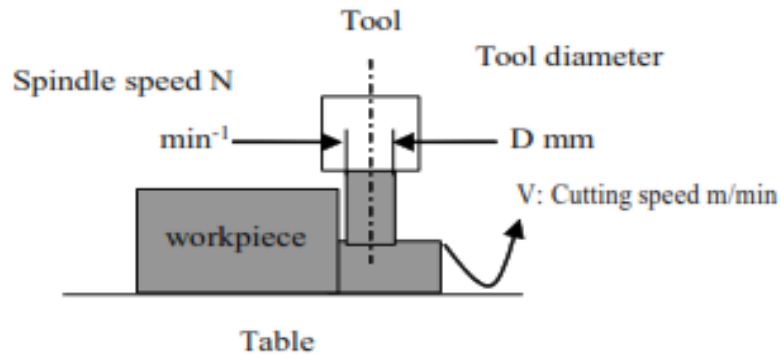


Figure 1.15 Cutting Speed and Spindle Speed

The formula to get the spindle speed is:

$$N = \frac{1000 * v}{\pi D}$$

N: the spindle speed

v: cutting speed

D: diameter value of the workpiece

Example: When the diameter of workpiece is 100mm, and the cutting speed is 80m/min, then the spindle speed:

$$N = \frac{1000 * v}{\pi D} = \frac{1000 * 80}{\pi * 100} \approx 250 r / m$$

The constant surface speed refers to the speed even when the workpiece diameter is changed, and the CNC changes the spindle speed. At this time, the spindle speed is the cutting speed.

Tool Function

Tool Selection

It is necessary to select a suitable tool when drilling, tapping, boring or the like is performed. As it is shown in Figure 1.16, a number is assigned to each tool. Then this number is used in the program to specify that the corresponding tool is selected.

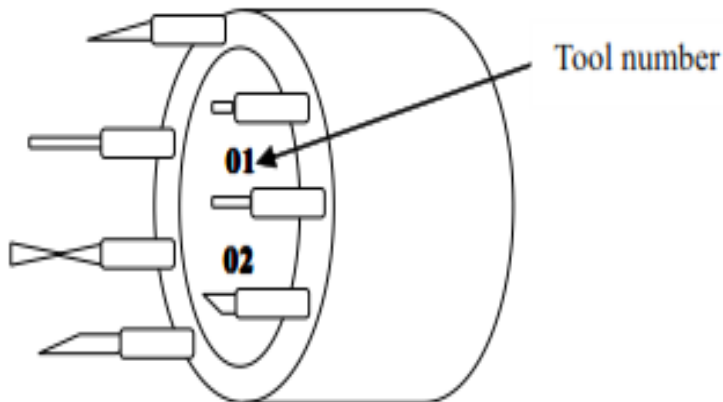


Figure 1.16 Tool Selection

Tool Offset

When writing a program, the operator just use the workpiece dimensions according to the dimensions in the part drawing. The tool nose radius center and the tool length are not taken into account. However, when machining a workpiece, the tool path is affected by the tool geometry. There are two kinds of tool offset: tool length compensation and tool radius compensation.

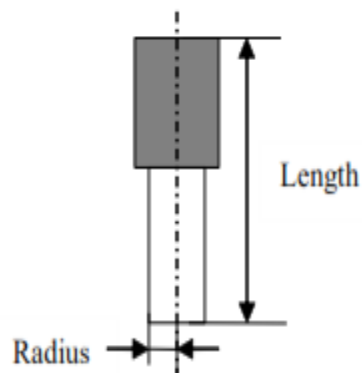


Figure 1.17 Length compensation and Radius compensation

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 11 of 73
------------------------	-----------------	---	-----------	---------------

➤ Tool Length Compensation

There are two kind of ways to specify the value of tool length compensation.

- ✓ Absolute value of tool compensation (the distance between tool tip and machine reference point)
- ✓ Incremental value of tool compensation (the distance between tool tip and the standard tool)

➤ Tool Radius Compensation

Figure 1.18 shows the difference between the programmed contour and the corrected tool path.

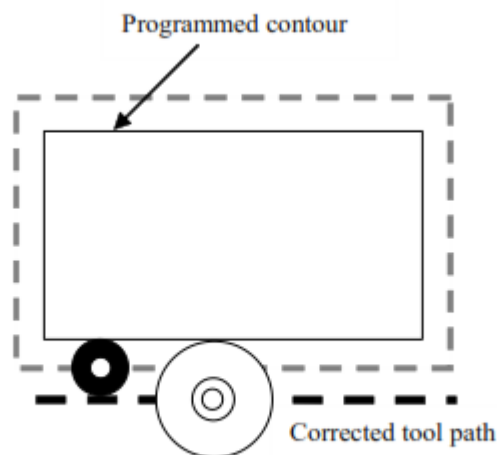


Figure 1.18 Difference between programmed contour and corrected tool path

Miscellaneous Function

Miscellaneous function refers to the operation to control the spindle, feed, and coolant. In general, it is specified by an M code.

When a move command and M code are specified in the same block, there are two ways to execute these commands:

1) Pre-M function

M command is executed before the completion of move command

2) Post-M function

M command is executed after the completion of move command.

The sequence of the execution depends on the specification of the machine tool builder.

Program Configuration

Structure of an NC Program

As it is shown in Figure 1.19, an NC program consists of a sequence of NC **blocks**. Each block is one of machining steps. **Commands** in each block are the instruction.

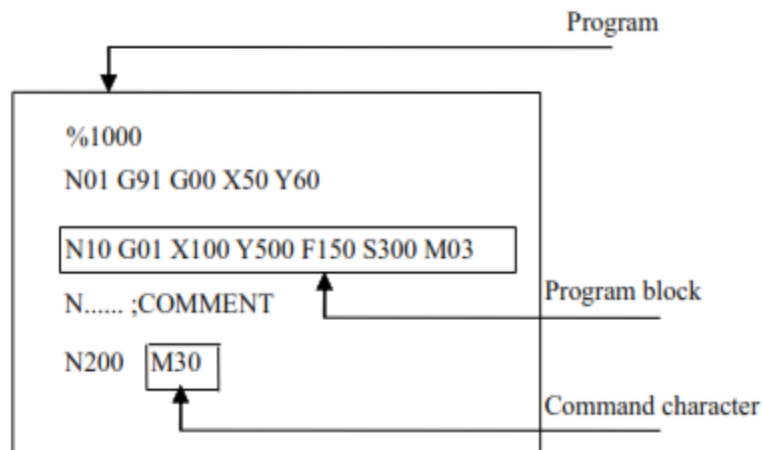


Figure 1.19 Structure of an NC Program

➤ Format of **program name**

The program name must be specified in the format OXXXX (X could be letters or numbers).

➤ Format of **program number**

The program number should be started with %XXXX or OXXXX (X could be numbers only)

➤ Format of **blocks**

A block starts with the program block number.

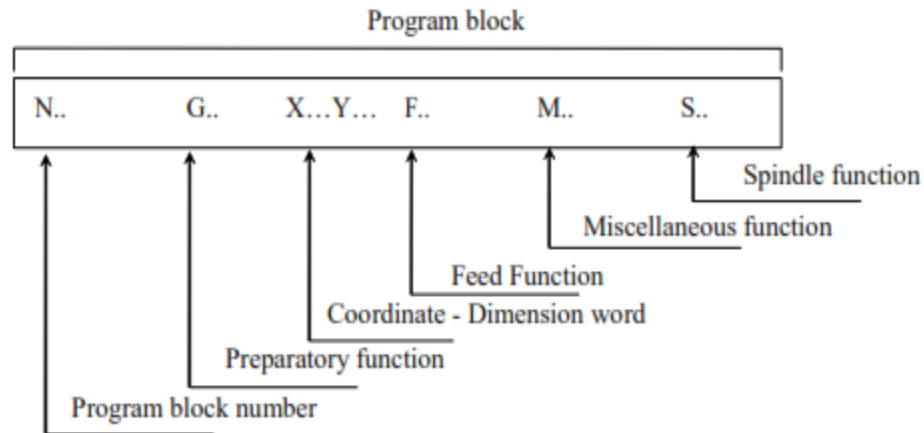


Figure 1.20 Structure of Block

➤ Format of **end of program**

The last block should contain M02 or M03 to indicate the end of program.

➤ Format of **Comments**

All information after the “;” is regarded as comments.

All information between “()” is regarded as comments.

Main Program and Subprogram

There are two type of program: main program and subprogram. The CNC operates according to the main program. When a execution command of subprogram is at the execution line of the main program, the subprogram is called. When the execution of subprogram is finished, the system returns control to the main program.

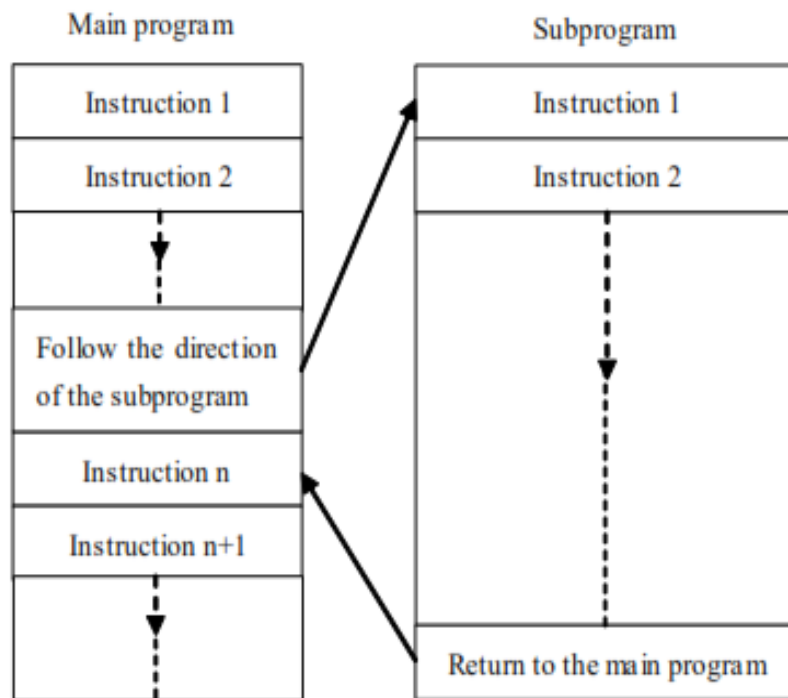


Figure 1.21 Main program and subprogram

Note:

Main program and its subprogram must be written in a same file with a different program codes.

Preparatory Function (G code)

There are two types of G code: one-shot G code, and modal G code.

Table 2 1 Type of G code

Type	Meaning
One-shot G code	The G code is only effective in the block in which it is specified
Modal G code	The G code is effective until another G code is specified.

Example: G01 and G00 are modal G codes.

```
N10 G01 X 100;  
N20 Y200 X200;  
N30 X300;  
N40 G00 Y100;
```

} G01 is effective from N10 to N30

G code List

The following table is the list of G code in HNC system

Table 2 2 G code list

G code	Group	function
G00	01	Rapid positioning
▲G01		Linear interpolation
G02		Circular interpolation/Helical interpolation CW
G03		Circular interpolation/Helical interpolation CCW
G04	00	Dwell
G07	00	Virtual axis
G09	00	Exact stop
▲G17	02	XY plane selection
G18		ZX plane selection
G19		YZ plane selection
G20	08	Input in inches
▲G21		Input in metrics
G22		Input in impulses equivalent weight
G24	03	Programmable mirror image
▲G25		Programmable mirror image cancel
G28	00	Return to reference point
G29		Return from reference point
G34	00	Thread tapping
G38	00	Polar Coordinates
▲G40	09	Cutter compensation cancel
G41		Cutter compensation left
G42		Cutter compensation right
G43	10	Tool length compensation +direction
G44		Tool length compensation - direction
▲G49		Tool length compensation cancel
▲G50	04	Scaling cancel
G51		Scaling

G53	00	Machine coordinate system selection
-----	----	-------------------------------------

G54		Workpiece coordinate system 1
G55		Workpiece coordinate system 2
G56	11	Workpiece coordinate system 3
G57		Workpiece coordinate system 4
G58		Workpiece coordinate system 5
G59		Workpiece coordinate system 6
G60	00	Single direction positioning
▲G61	12	Exact stop mode
G64		Cutting mode
G68	05	Coordinate rotation
▲G69		Coordinate rotation cancel
G73		High-speed drilling cycle
G74		Left-hand tapping cycle
G76		Fine boring cycle
▲G80		Canned cycle cancel
G81		Drilling cycle, Spot drilling
G82		Drilling cycle, Counter boring cycle
G83	06	Peck drilling cycle
G84		Tapping cycle
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Manual Boring cycle
G89		Boring cycle
▲G90	13	Absolute command
G91		Increment command
G92	00	Setting for work coordinate system
▲G94	14	Feed per minute
G95		Feed per rotation
▲G98	15	Return to initial point in canned cycle
G99		Return to R point in canned cycle

Explanation:

- 1) G codes in 00 group are one-shot G code, while the other groups are modal G code.
- 2) **▲** means that it is default setting.
- 3) Multiple G codes from different groups can be specified in the same block. If multiple G codes from the same group are specified in the same block, only the last G code specified is valid.

Interpolation Functions

This chapter would introduce:

- 1) Positioning Command (G00)
- 2) Single Direction Positioning (G60)
- 3) Linear Interpolation (G01)
- 4) Circular Interpolation (G02, G03)
- 5) Helical Interpolation (G02, G03)
- 6) Thread Tapping (G34)

1) Positioning Command (G00)

Programming

G00 X_Y_Z_A_

Explanation of the parameters

X, Y, Z, A Coordinate value of the end point in the absolute command or incremental command

Function

The tool is moved at the highest possible speed (rapid traverse). If the rapid traverse movement is required to execute simultaneously on several axes, the rapid traverse speed is decided by the axis which takes the most time. Thus, the tool path is nonlinear. The operator can use this function to position the tool rapidly, to travel around the workpiece, or to approach the tool change position.

<i>Ethiopian TVET Program</i>	STEP-GIZ	<i>CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations</i>	<i>July 2020</i>	<i>Page 19 of 73</i>
-----------------------------------	-----------------	--	------------------	----------------------

Example

Move tool from A (20, 15) to B (90, 45) at the rapid traverse speed.

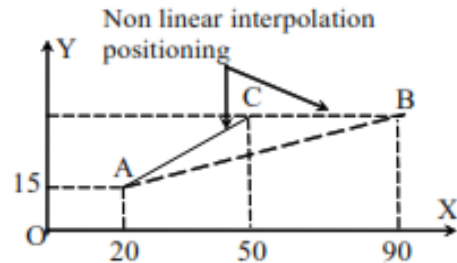


Figure 3.1 Positioning (Rapid Traverse)

Absolute programming:

G00 X90 Z45

Incremental programming:

G00 X70 Y30

2) Single Direction Positioning (G60)

Programming

G60 X_ Y_ Z_ A_

Explanation of the parameters

X, Y, Z, A Coordinate value of the end point in the absolute command or incremental command

Function

At first, move the tool from the start point to the intermediate point at the rapid traverse speed. Then, tool is moved from the intermediate point to the end point at the specified feed rate.

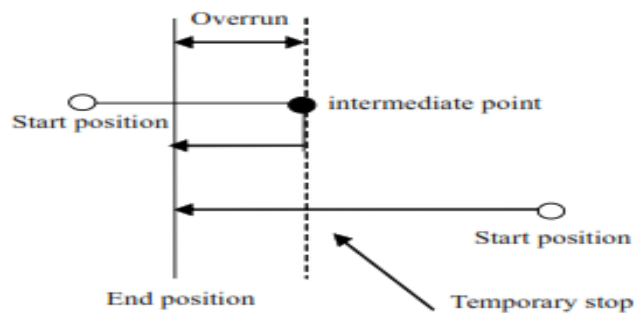


Figure 3.2 Single Direction Positioning (G60)

Note:

The direction and distance from the intermediate point to the end point are set by machine parameter – single direction positioning offset. When the value of the parameter is less than 0, the direction is negative. When the value of the parameter is more than 0, the direction is positive.

3) Linear Interpolation (G01)

Programming

G01 X_ Y_ Z_ A_ F_

Explanation of the parameters

X, Y, Z, A Coordinate value of the end point in the absolute command or incremental command

Function

The tool is moved along the straight line at the specified feed rate.

Example 1

Move tool from A (20, 15) to B (90, 45) at the rapid traverse speed.

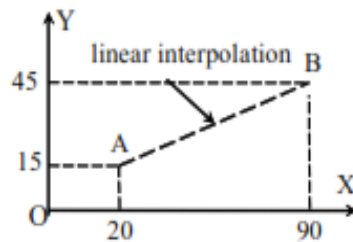


Figure 3.3 Linear Interpolation – Example 1

Absolute programming

G01 X90 Y45 F800

Incremental programming

G01 X70 Y30 F800

Example 2

Use the tool (8) to machine a groove (3mm) on a workpiece.

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 21 of 73
---------------------------	-----------------	---	-----------	---------------

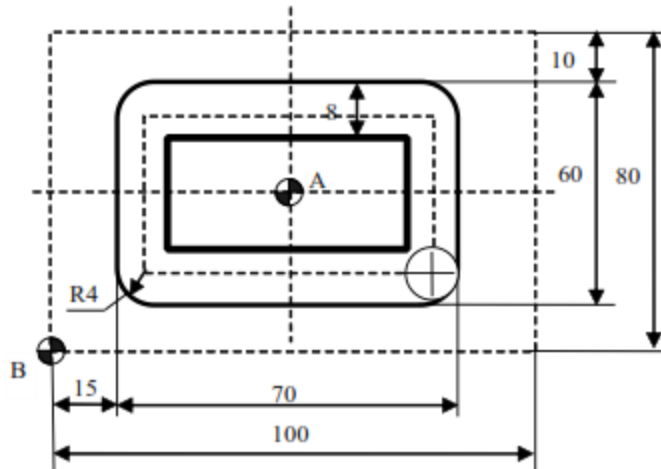


Figure 3.4 Linear Interpolation – Example 2

%3308 (the origin is on A)

N1 G92 X0 Y0 Z50

N2 M03 S500

N3 G00 X-31 Y-26

N4 Z5

N5 G01 Z-3 F40

N6 Y26 F100

N7 X31

N8 Y-26

N9 X-31

N10 G00 Z50

N11 X0 Y0

N12 M05

N13 M30

%3309 (the origin is on B)

N1 G92 X0 Y0 Z50

N2 M03 S500

N3 G00 X19 Y14

N4 Z5

N5 G01 Z-3 F40

N6 Y66 F100

N7 X81

N8 Y14

N9 X19

N10 G00 Z50

N11 X0 Y0

N12 M05

N13 M30

4) Circular Interpolation (G02, G03)

Programming

$$G17 \begin{cases} G02 \\ G03 \end{cases} X_Y_ \begin{cases} I_J_ \\ R_ \end{cases} F_$$

$$G18 \begin{cases} G02 \\ G03 \end{cases} X_Z_ \begin{cases} I_K_ \\ R_ \end{cases} F_$$

$$G19 \begin{cases} G02 \\ G03 \end{cases} Y_Z_ \begin{cases} J_K_ \\ R_ \end{cases} F_$$

Explanation of the parameters

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 22 of 73
---------------------------	-----------------	---	-----------	---------------

G17 The working plane is XY, and the infeed direction is Z
 G18 The working plane is XZ, and the infeed direction is Y
 G19 The working plane is YZ, and the infeed direction is X
 G02 a circular path in clockwise direction (CW) (Figure 3.5)
 G03 a circular path in counterclockwise direction (CCW)
 G02 and G03 are defined when the working plane is specified. Figure 3.5 shows the direction of circular interpolation.

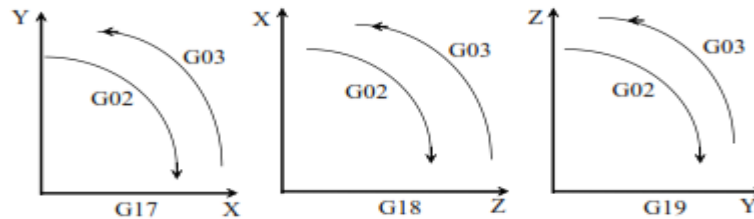


Figure 3.5 Direction of circular interpolation

X, Y/X, Z/Y, Z For an absolute command, the coordinate values of the circle end point in the specific working plane. For an incremental command, the coordinate values of the circle end point with reference to the circle starting point in the specific working plane.

I, J/I, K/J, K Coordinate values of the circle center point with reference to the circle starting point in incremental command. (Figure 3.6)

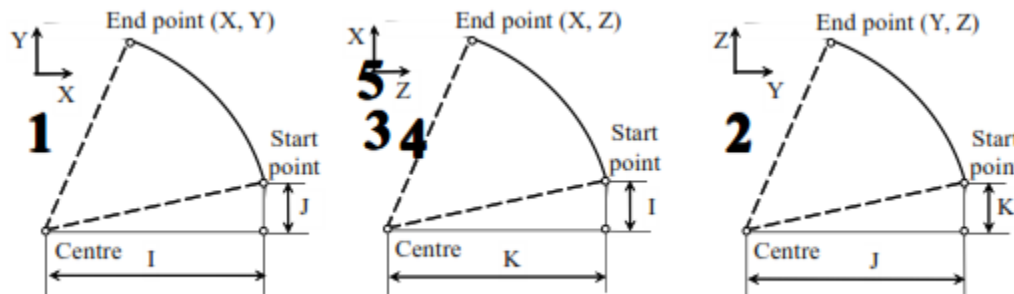


Figure 3.6 Distance from the start point to the circle centre point

R Circle radius. When the arc is less than 180° (minor arc), R is positive. If the arc is more than 180° (major arc), R is negative.

F Feed rate along the circle

Function

The tool is moved along a full circle or arcs.

Note:

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 23 of 73
------------------------	-----------------	---	-----------	---------------

- 1) When it is full circle programming, R can not be used in the program. I, J, K can only be used in this case.
- 2) When it is not full circle programming, the operator can select R or I, J, K to program. If I, J, K, and R addresses are all specified in the program, R takes precedence and the other are ignored.

Example 2

Use G02/G03 to program the full circle.

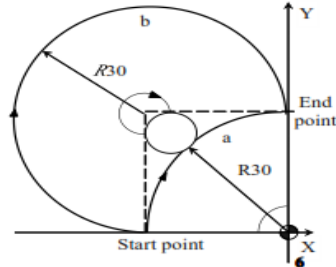


Figure 3.7 Circular Interpolation – Example 1

(i) Arc a

```
G91 G02 X30 Y30 R30 F300
G91 G02 X30 Y30 I30 J0 F300
G90 G02 X0 Y30 R30 F300
G90 G02 X0 Y30 I30 J0 F300
```

(ii) Arc b

```
G91 G02 X30 Y30 R-30 F300
G91 G02 X30 Y30 I0 J30 F300
G90 G02 X0 Y30 R-30 F300
G90 G02 X0 Y30 I0 J30 F300
```

Example 2

Use G02/G03 to program the full circle.

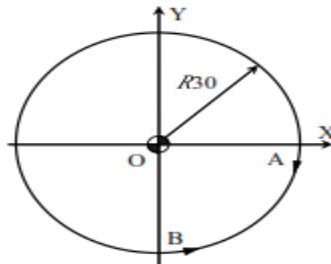


Figure 3.8 Circular Interpolation – Example 2

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 24 of 73
---------------------------	-----------------	---	-----------	---------------

i) Clockwise circle from A to A
 G90 G02 X30 Y0 I-30 J0 F300
 G91 G02 X0 Y0 I-30 J0 F300

(ii) Counterclockwise circle from B to B
 G90 G03 X0 Y-30 I0 J30 F300
 G91 G03 X0 Y0 I0 J30 F300

Example 3

Use the tool (8 dia.) to machine a groove (3mm) on a workpiece.

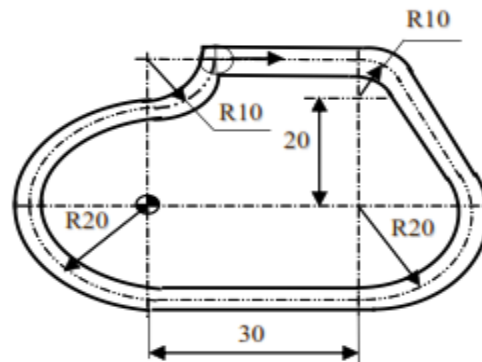


Figure 3.9 Circular Interpolation – Example 3

```
%3314
N1 G92 X0 Y0 Z50
N2 M03 S500
N3 G00 X10 Y30
N4 Z5
N5 G01 Z-3 F40
N6 X30
N7 G02 X38.66 Y25 R10
(N7 G02 X38.66 Y25 J-10)
N8 G01 X47.32 Y10
N9 G02 X30 Y-20 R20
(N9 G02 X30 Y-20 J-10 I-17.32)
N10 G01 X0
N11 G02 X0 Y20 R20
(N11 G02 X0 Y20 J20)
N12 G03 X10 Y30 R10
(N13 G03 X10 Y30 J10)
N14 G00 Z50
N15 X0 Y0
N16 M30
```

5) Helical Interpolation (G02, G03)

Programming

$$G17 \left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} X_Y_ \left\{ \begin{array}{l} I_J_ \\ R_ \end{array} \right\} Z_F_L$$

$$G18 \left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} X_Z_ \left\{ \begin{array}{l} I_K_ \\ R_ \end{array} \right\} Y_F_L$$

$$G19 \left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} Y_Z_ \left\{ \begin{array}{l} J_K_ \\ R_ \end{array} \right\} X_F_L$$

Explanation of the parameters

G17 The working plane is XY, and the infeed direction is Z

G18 The working plane is XZ, and the infeed direction is Y

G19 The working plane is YZ, and the infeed direction is X

G02 a circular path in clockwise direction (CW) (Figure 3.5)

G03 a circular path in counterclockwise direction (CCW)

X, Y/X, Z/Y, Z For an absolute command, the coordinate values of the circle end point in the specific working plane. For an incremental command, the coordinate values of the circle end point with reference to the circle starting point in the specific working plane.

I, J/I, K/J, K Coordinate values of the circle center point with reference to the circle starting point in incremental command.

R Circle radius. When the arc is less than 180° (minor arc), R is positive. If the arc is more than 180° (major arc), R is negative.

Z, Y, X The coordinate value of the end point with reference to the starting point on the third axis in the incremental command.

F Feed rate along the circle

L Number of circles on a workpiece

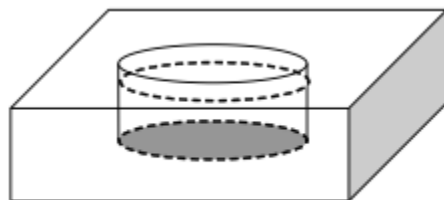


Figure 3.11 Helical Interpolation (G02, G03)

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 26 of 73
---------------------------	-----------------	---	-----------	---------------

Function

Helical interpolation can be used to manufacture threads on the workpiece.

Example 1

Use G03 to program.

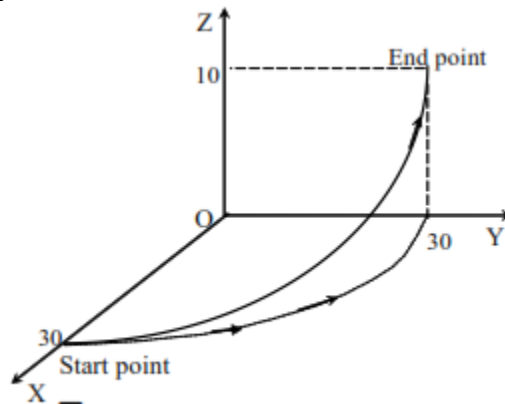


Figure 3.12 Helical Interpolation – Example 1

- Absolute programming
G90 G17 F300
G03 X0 Y30 R30 Z10
- Incremental programming
G91 G17 F300
G03 X-30 Y30 R30 Z10

Example 2

Use the tool (10mm dia.) to machine a hole (the diameter is 50mm, and the height is 10mm) on a workpiece.

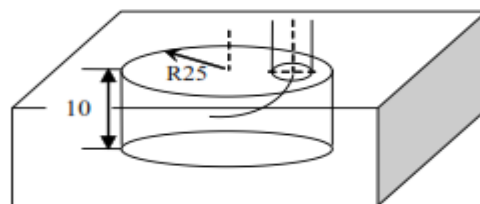


Figure 3.13 Helical Interpolation – Example 2

```

%3317
N1 G92 X0 Y0 Z30
N2 G01 Z11 X20 F200
N3 G91 G03 I-20 Z-1 L11
N4 G03 I-20
N5 G90 G01 X0
N6 G00 Z30
N7 X30 Y-50
N8 M30

```

6) Thread Tapping (G34)

Programming

G34 K_ F_ P_

Explanation of the parameters

K The distance from the starting point to the bottom of the hole

F Thread lead. If it is positive, the spindle turns clockwise during tapping. If it is negative, the spindle turns counterclockwise during tapping.

P Dwell time at the bottom of a hole. (The unit is seconds.)

Function

With this command, the operator can rigid tap a thread.

Note

1) When the spindle turns clockwise during tapping, the spindle would turn counterclockwise during retraction.

2) When the spindle turns counterclockwise during tapping, the spindle would turn clockwise during retraction.

In general, there is overshoot of the tap at the bottom of the thread during the spindle-braking portion of the tapping cycle. It can be set by PMC parameters (Table 3-1) to eliminate the overshoot errors.

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 28 of 73
---------------------------	-----------------	---	-----------	---------------

Table 3 1 PMC parameters

CNC system	PMC parameters	
HNC 18/19i	#0062	Maximum spindle speed during tapping
	#0063	Minimum spindle speed during tapping
	#0064	Dwell unit for tapping
	#0065	Optional dwell unit for tapping
HNC 21/22	#0017	Maximum spindle speed during tapping
	#0018	Minimum spindle speed during tapping
	#0019	Dwell unit for tapping
	#0030	Optional dwell unit for tapping

Optional dwell unit for tapping is only effective when “dwell unit for tapping” is assigned to “0”. Moreover, it is not necessary to restart the system.

The following formula is to calculate the dwelled unit (X):

$$D = (S * S / C) * X / 10000 = L * 360 / F$$

D dwell amount

S spindle speed

C Transmission gear ratio

X dwell unit

L overshoot error

F thread lead

Example

Use G34 to program.

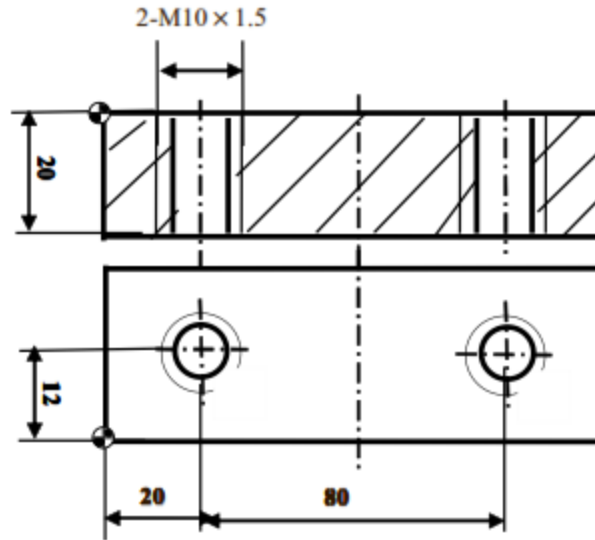


Figure 3.16 Tapping - Example

```

%0002
G92 X-20 Y-20 Z50
M03 S200
G00 X20 Y12
Z5
G34 K-27 F1.5
G00 X100
G34 K-27 F1.5
G00 Z50
X-20 Y-20
M05
M30

```

Feed Function

This chapter would introduce:

- 1) Rapid Traverse
The tool is moved at the rapid traverse speed set in CNC.
- 2) Cutting Feed
The tool is moved at the programmed cutting feed rate.
- 3) Dwell
- 4) Exact Stop
- 5) Cutting Mode

Rapid Traverse (G00)

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 30 of 73
---------------------------	-----------------	---	-----------	---------------

Positioning command (G00) is to move the tool at the rapid traverse speed (the highest possible speed).

This rapid traverse speed can be controlled by the machine control panel. For more detailed information, please refer to turning operation manual.

Cutting Feed (G94, G95)

Programming

G94 [F_]

G95 [F_]

Explanation of the parameters

G94 feed rate per minute.

On linear axis, the unit of feed rate is mm/min, or in/min.

On rotational axis, the unit of feed rate is degree/min.

G95 feed rate per revolution

The unit of feed rate is mm/rev, or in/rev.

Note:

- 1) G94 is the default setting
- 2) G95 is only used when there is spindle encoder.

Function

The feed rate can be set by G94 or G95.

Dwell (G04)

Programming

G04 P_

Explanation of the parameters

P dwell time (specified in seconds)

Function

It can be used to interrupt machining to get the smooth surface. It can be used to control the groove cutting, drilling, and turning path.

Example

Use G04 to get the smooth surface.

<i>Ethiopian TVET Program</i>	STEP-GIZ	<i>CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations</i>	<i>July 2020</i>	<i>Page 31 of 73</i>
-----------------------------------	-----------------	--	------------------	----------------------

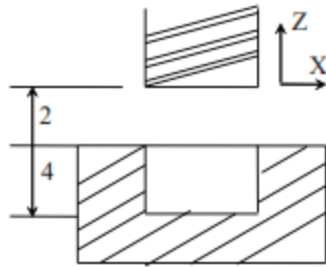


Figure 4.1 Dwell – Example

```

%0004
G92 X0 Y0 Z0
G91 F200 M03 S500
G43 G01 Z-6 H01
G04 P5
G49 G00 Z6 M05 M30

```

Exact Stop (G09, G61)

Programming

G09
G61

Explanations of the parameters

The tool is moved to the end point of a block, then the position of the end point is checked. Then, the next block is proceeded.

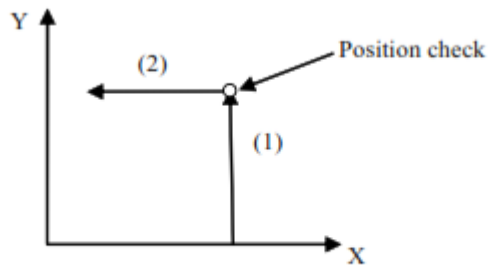


Figure 4.2 Exact Stop (G09/G61) – tool path from block (1) to block (2)

The difference between G09 and G61 is that G09 is one-shot G code. And G61 is modal G code.

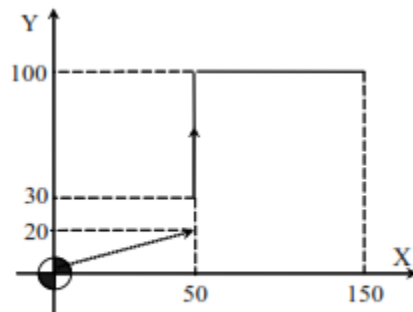
Function

G09 or G61 can be used to machine a sharp edge.

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 32 of 73
---------------------------	-----------------	---	-----------	---------------

Example

Use G61 to program.



8 Figure 4.3 Exact Stop - Example

```
%0061  
G92 X0 Y0 Z0  
G91 G00 G43 Z-10 H01  
G41 X50 Y20 D01  
G01 G61 Y80 F300  
X100  
...
```

Cutting Mode (G64)

Programming

G64

Explanation of the parameters

The tool is moved to the end point of a block. Then, the next block is proceeded. The tool path is shown in the following figure.

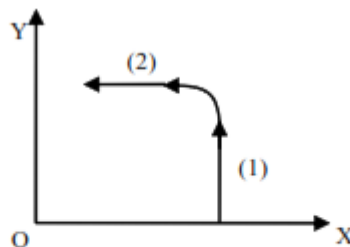


Figure 4.4 Cutting Mode (G64) – tool path from block (1) to block (2)

Function

G64 command can make the tool move smoothly between two blocks.

Example

Use G64 to program.

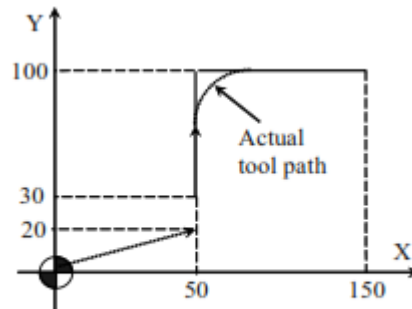


Figure 4.5 Cutting Mode – Example

```
%0064  
G92 X0 Y0 Z0  
G91 G00 G43 Z-10 H01  
G41 X50 Y20 D01  
G01 G64 Y80 F300  
X100  
...
```

Coordinate System

This chapter would introduce:

- 1) Reference Position Return (G28)
- 2) Auto Return from Reference Position (G29)
- 3) Setting a Workpiece Coordinate System (G92)
- 4) Selecting a Machine Coordinate System (G53)
- 5) Selecting a Workpiece Coordinate System (G54~G59)
- 6) Plane Selection (G17, G18, G19)
- 7) Absolute and Incremental Programming (G90, G91)
- 8) Dimension Selection (G20, G21, G22)
- 9) Polar Coordinates (G38)

1) Reference Position Return (G28)

Programming

G28 X_ Y_ Z_ A_

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 34 of 73
---------------------------	-----------------	---	-----------	---------------

Explanation of the parameters

X, Y, Z, A Coordinate values of the intermediate point in absolute command/incremental command

Function

The tool is moved to the intermediate point rapidly, and then returned to the reference point.

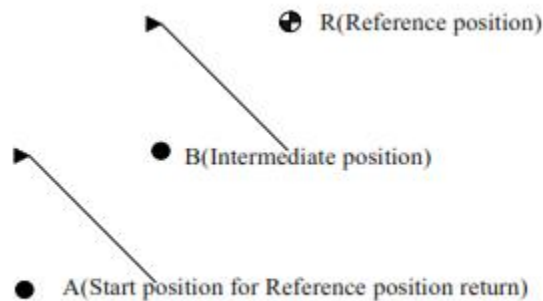


Figure 5.1 Reference Position Return (G28)

Note:

- 1) In general, G28 is used to change tools or cancel the mechanical error. Tool radius compensation and tool length compensation should be cancelled when G28 is executed.
- 2) G28 can not only make the tool move to the reference point, but also can save the intermediate position to be used in G29.
- 3) When the power is on and manual reference position return is not available, G28 is same as the manual reference position return. The direction of this reference position return (G28) is set by the axis parameter – reference approach direction.
- 4) G28 is one-shot G code.

2) Auto Return from Reference Position (G29)

Programming

G29 X_ Y_ Z_ A_

Explanation of the parameters

X, Y, Z, A Coordinate value of the end point in absolute command/incremental command

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 35 of 73
---------------------------	-----------------	---	-----------	---------------

Function

The tool is moved rapidly from the intermediate point defined in G28 to the end point. Thus, G29 is generally used after G28 is defined.

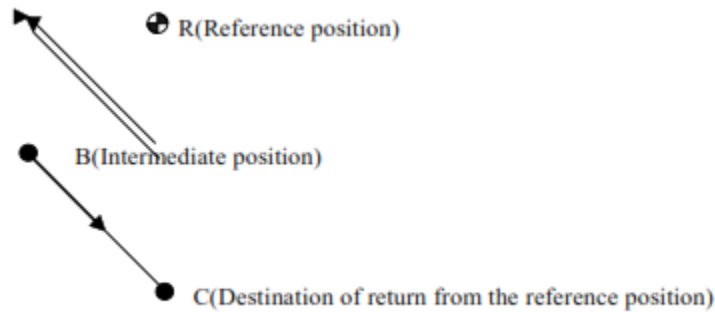


Figure 5.2 Auto Return from Reference Position (G29)

Note:

G29 is one-shot G code.

Example

Use G28, G29 command to program the track shown in. It moves from the starting point A to the intermediate point B, and then returns to the reference point R. At last, it moves from the reference point R to the end point C through the intermediate point B.

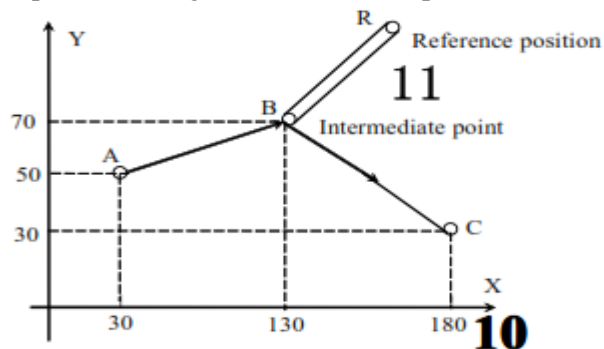


Figure 5.3 Reference Position – Example

```
...  
G91 G28 X100 Y20 ;A→B→R  
M06 T02 ;Changing the tool  
G29 X50 Y-40 ;R→B→C  
...
```

3) Setting a Workpiece Coordinate System (G92) Programming

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 36 of 73
------------------------	----------	---	-----------	---------------

G92 X_ Y_ Z_ A_

Explanation of the parameters

X, Y, Z, A Coordinate values of the tool position in the workpiece coordinate system.

Functions

G92 can set a workpiece coordinate system based on the current tool position (X_ Y_ Z_ A_).

Example

Use G92 to set a workpiece coordinate system.

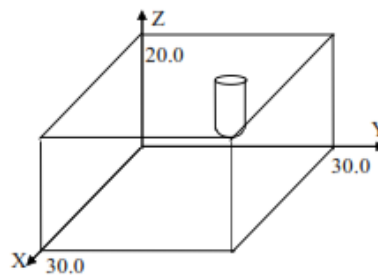


Figure 5.4 Setting a Workpiece Coordinate System – Example

G92 X30.0 Y30.0 Z20.0

4) Selecting a Machine Coordinate System (G53)

Programming

G53 X_ Y_ Z_ A_

Explanation of the parameters

X, Y, Z, A Absolute coordinate values of a point in the machine coordinate system.

Function

A machine coordinate system is selected, and the tool moves to the position at the rapid traverse speed.

Note:

1) Absolute values must be specified in G53. The incremental values would be ignored by G53.

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 37 of 73
---------------------------	-----------------	---	-----------	---------------

2) G53 is one-shot G code.

5) Selecting a Workpiece Coordinate System (G54~G59)

Programming

$\left. \begin{array}{l} G54 \\ G55 \\ G56 \\ G57 \\ G58 \\ G59 \end{array} \right\} X_Y_Z_A_$

Explanation of the parameters

X, Y, Z, A Coordinate values of the point with reference to the origin of machine in absolute command

Function

There are six workpiece coordinate system to be selected. If one coordinate system is selected, the tool is moved to a specified point.

Note:

- 1) The workpiece coordinate system must be set before using these commands (G54~G59). The workpiece coordinate system can be set by using the MDI panel. For detailed information, please refer to the milling operation manual.
- 2) Reference position must be returned before these commands (G54~G59) are executed.
- 3) G54 is the default setting.

Example

Example

Select one of workpiece coordinate system, and the tool path is Current point → A→B.

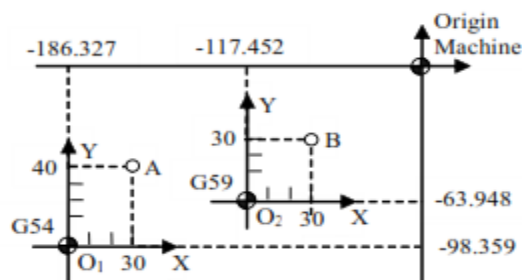


Figure 5.5 Workpiece Coordinate System – Example

```

%1000
N01 G54 G00 G90 X30 Y40
N02 G59
N03 G00 X30 Y30
N04 G54
N05 X0 Y0
N06 M30

```

6) Plane Selection (G17, G18, G19)

Programming

G17

G18

G19

Explanation of the parameters

G17 working plane is XY, infeed direction is Z

G18 working plane is ZX, infeed direction is Y

G19 working plane is YZ, infeed direction is X

Function

The working plane is specified and used for tool radius compensation and circular interpolation.

Note:

Move command is not related with the plane selection. For example, in the command G17

G01 Z10, Z axis does still move.

7) Absolute and Incremental Programming (G90, G91)

Programming

G90 X_ Y_ Z_ A_

G91 X_ Y_ Z_ A_

Explanation of the parameters

G90 Absolute programming

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 39 of 73
---------------------------	-----------------	---	-----------	---------------

X, Y, Z, A Coordinate values of the point with reference to the origin of programming

G91 Incremental programming

X, Y, Z, A Coordinate values of the point with reference to the previous position

Function

The tool is moved to the specified position.

Example

Move the tool from point 1 to point 2 through point 3, and then return to the current point.

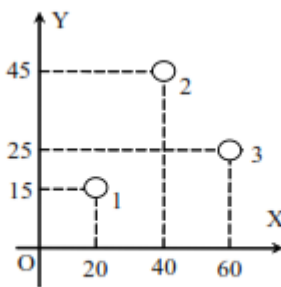


Figure 5.6 Absolute and Incremental Programming – Example

G90 programming
%0001
M03 S500
N01 G92 X0 Y0 Z10
N02 G01 X20 Y15
N03 X40 Y45
N04 X60 Y25
N05 X0 Y0 Z10
N06 M30

G91 programming
%0001
M03 S500
N01 G92 X0 Y0 Z10
N02 G91 G01 X20 Y15
N03 X20 Y30
N04 X20 Y-20
N05 G90 X0 Y0
N06 M30

8) Dimension Selection (G20, G21, G22)

Programming

G20
G21
G22

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 40 of 73
---------------------------	-----------------	---	-----------	---------------

Explanation of the parameters

G20: Inch input

G21: Metric input

G22: Impulses equivalent weight input

The units of linear axis and circular axis are shown in the following table

Function

Depending on the part drawing, the workpiece geometries can be programmed in metric measures, inches, or impulses equivalent weight.

Table 5 1. Unit of Linear axis and Circular axis

	Linear axis	Circular axis
Inch system (G20)	Inch	Degree
Metric system (G21)	Mm	Degree
Pulse system (G22)	Impulses equivalent weight	Impulses equivalent weight

9) Polar Coordinates (G38)

Programming

G38 X_ Y_

G01 AP=_ RP=_

G02/G03 AP=_ RP=_ R_

Explanation of the parameters

G38 Setting a polar coordinate system

X, Y Coordinate value of the pole in the workpiece coordinate system

AP Polar angle

RP Polar radius

R Circle radius

Function

The polar coordinate method is useful only if there is radius and angle measurements on a workpiece.

Note

These commands can be used with commands of workpiece coordinate system.

Example 1

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 41 of 73
---------------------------	-----------------	---	-----------	---------------

Use polar coordinates command to program.

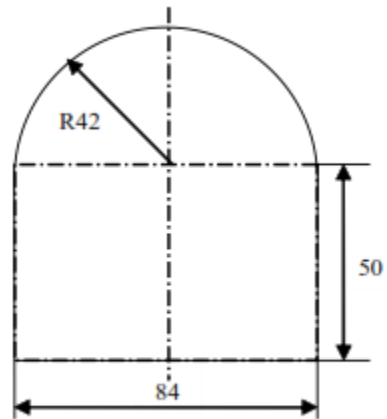


Figure 5.7 Polar Coordinates – Example 1

```

%3326
G92 X0 Y0 Z10
G00 X-50 Y-60
G00 Z-3
G01 G41 X-42 D01 F1000
Y0
G38 X0 Y0
G02 AP=0 RP=42 R42
G01 Y-50
X-50
G00 G40 Y-60
Z10
G00 X0 Y0
M30
    
```

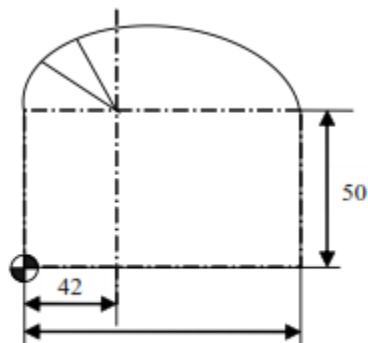


Figure 5.8 Polar Coordinate – Example 2

```

%0001
G54 G00 X-15 Y-15 Z10
G00 Z-3
G01 G41 X0 D01 F1000
Y50
G38 X42 Y50
#0=180
#1=42
while #0 gt 0
G01 AP=[#0] RP=[#1]
#0=#0-10
#1=#1+2
Endw
G01 AP=0 RP=78
Y0
X-15
G00 G40 Y-15
Z10
M30

```

6. Spindle Speed Function

Spindle function controls the spindle speed (S), the unit of spindle speed is r/min. S is modal G code command; it is only available when the spindle is adjustable. Spindle speed programmed by S code can be adjusted by overrides on the machine control panel.

Tool Function

This chapter would introduce:

- 1) Tool selection and Tool offset (T code)
- 2) Tool radius compensation (G40, G41, G42)

Tool Selection and Tool Offset (T code)

Programming

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 43 of 73
---------------------------	-----------------	---	-----------	---------------

T XX XX

Explanation of the parameters

XX Tool number (two digits). The number of tool depends on manufacture's configuration.

XX Tool offset number (two digits). It corresponds to the specific compensation value.

Functions

To select the desired tool, T command makes the turret turn, selects a cutter, and calls the compensation value.

Note:

- 1) T command is only effective when it is used with tool move command, such as G00.
- 2) When T command and tool move command are in the same program block, T command is executed at first.
- 3) The same tool can have different compensation values. For example, T0101, T0102, T0103 are possible.
- 4) Different tool can have same compensation values. For example, T0101, T0201, and T0301 are possible.

Tool Radius Compensation (G40, G41, G42)

Programming

$$\left\{ \begin{matrix} G17 \\ G18 \\ G19 \end{matrix} \right\} \left\{ \begin{matrix} G40 \\ G41 \\ G42 \end{matrix} \right\} \left\{ \begin{matrix} G00 \\ G01 \end{matrix} \right\} X _ Y _ Z _ D _$$

Explanation of the parameters

G17 Tool radius compensation on plane XY

G18 Tool radius compensation on plane ZX

G19 Tool radius compensation on plane YZ

G40 Deactivate tool radius compensation

G41 Activate tool radius compensation, tool operates in machining operation to the left of the contour.

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 44 of 73
---------------------------	-----------------	---	-----------	---------------

G42 Activate tool radius compensation, tool operates in machining operation to the right of the contour.

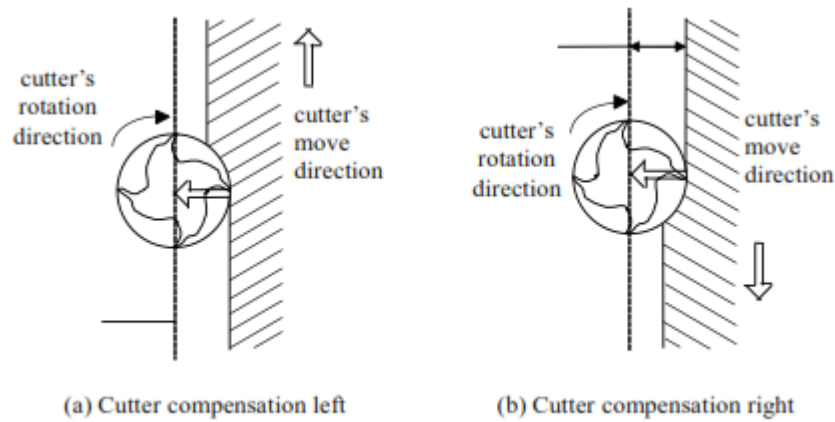


Figure 7.1 Tool Radius Compensation

X, Y, Z Coordinate values of the end point. It is the point where the tool radius compensation is activated or deactivated.

D There are two ways to specify the value of D.

- D01~D99 Each code corresponds to the different values of the tool radius compensation.
- #100~#199 Variable of radius compensation

Function

These commands can control the tool radius compensation to get the equidistant tool paths for different tools.

Note:

- 1) G40, G41, and G42 must be used with G00 or G01.
- 2) Changing the plane of tool radius compensation can only be done when there is no compensation.

Example 1

Use the tool radius compensation, and program for the part shown in Figure 7.2. The dashed line stands for the actual tool path.

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 45 of 73
------------------------	-----------------	---	-----------	---------------

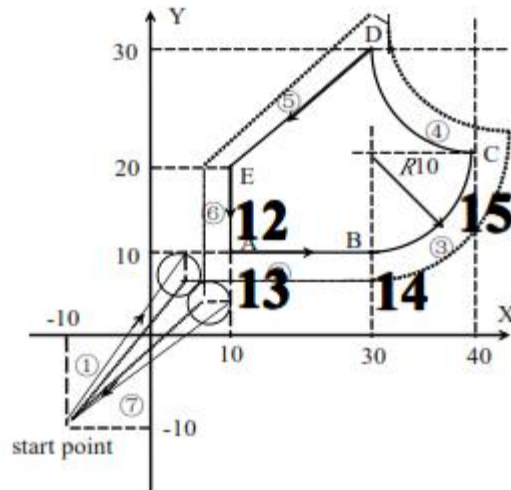


Figure 7.2 Tool Radius Compensation –Example 1

```

%3322
G92 X-10 Y-10 Z50
G90 G17
G42 G00 X4 Y10 D01
Z2 M03 S900
G01 Z-10 F800
X30
G03 X40 Y20 I0 J10
G02 X30 Y30 I0 J10
G01 X10 Y20
Y5
G00 Z50 M05
G40 X-10 Y-10
M02

```

Example 2

Use the tool (diameter is 8). The depth of cutting is 3mm.

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 46 of 73
---------------------------	-----------------	---	-----------	---------------

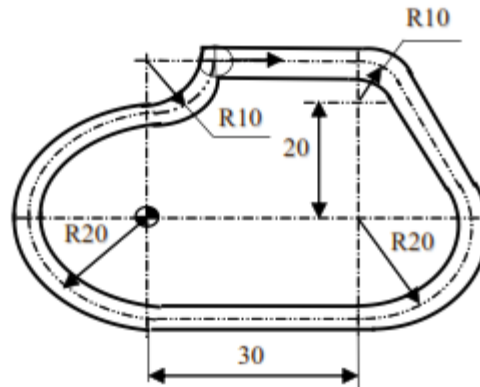


Figure 7.3 Tool Radius Compensation – Example 2

```

%3323
N1 G92 X-40 Y50 Z50
N2 M03 S500
N4 G01 Z-3 F400
N5 G01 G41 X5 Y30 D01 F40
N6 X30
N7 G02 X38.66 Y25 R10
(N7 G02 X38.66 Y25 J-10)
N8 G01 X47.32 Y10
N9 G02 X30 Y-20 R20
(N9 G02 X30 Y-20 I-17.32 J-10)
N10 G01 X0
N11 G02 X0 Y20 R20
(N11 G02 X0 Y20 J20)
N12 G03 Y40 R10
(N12 G03 Y40 J10)
N13 G00 G90 G40 X-40 Y50
N14 G00 Z50
N15 M30

```

Example 3

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 47 of 73
------------------------	-----------------	---	-----------	---------------

Use the tool (diameter is 8). The depth of cutting is 3mm.

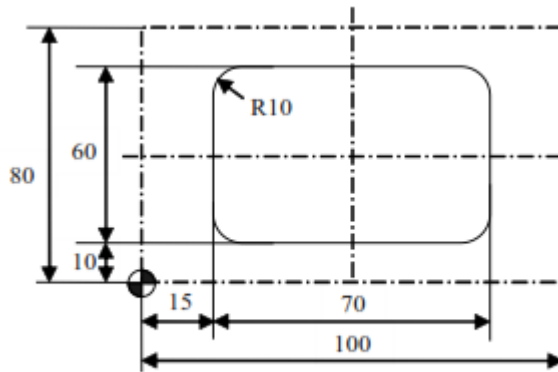


Figure 7.4 Tool Radius Compensation – Example 3

%3322 (female die)

N1 G92 X-10 Y-10 Z50

N2 M03 S500

N3 Z5

N4 G00 X25 Y20

N5 G01 Z-3 F40

N6 G41 Y30 D01 f100

N7 G03 Y10 R10

N8 G01 X75

N9 G03 X85 Y20 R10

N10 G01 Y60

N11 G03 X75 Y70 R10

N12 G01 X25

N13 G03 X15 Y60 R10

N14 G01 Y20

N15 G03 X23 Y12 R8

N16 G01 Z10

N17 G00 G40 X25 Y20

N18 G0 Z50

N19 M30

%3323 (male die)

N1 #101=4

N2 G92 X-10 Y-10 Z50

N3 M03 S500

N4 Z5

N5 G01 Z-3 F40

N6 G41 X15 D101 f100

N7 Y60

N8 G02 X25 Y70 R10

N9 G01 X75

N10 G02 X85 Y60 R10

N11 G01 Y20

N12 G02 X75 Y10 R10

N13 G01 X25

N14 G02 X15 Y20 R10

N15 G01 Z10

N16 G00 G40 X0 Y0

N17 G0 Z50

N18 M30

Tool Length Compensation (G43, G44, G49)

Programming

$$\left\{ \begin{array}{l} G17 \\ G18 \\ G19 \end{array} \right\} \left\{ \begin{array}{l} G43 \\ G44 \\ G49 \end{array} \right\} \left\{ \begin{array}{l} G00 \\ G01 \end{array} \right\} X_Y_Z_H_$$

Explanation of the parameters

- G17 XY plane selection (compensate for the difference in tool length along Z axis)
- G18 ZX plane selection (compensate for the difference in tool length along Y axis)
- G19 YZ plane selection (compensate for the difference in tool length along X axis)
- G43 Positive offset
- G44 Negative offset
- G49 Deactivate the tool length compensation
- X, Y, Z Coordinate value of the end point
- H H00~H99: Each code corresponds to the different values of the tool length compensation.

Function

These command can compensate the difference between the assumed tool length in the programming and the actual tool length.

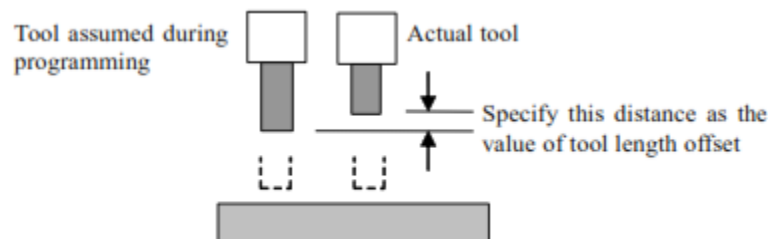


Figure 7.5 Tool Length Compensation (G43, G44, G49)

Example

Use the tool length compensation function to program.

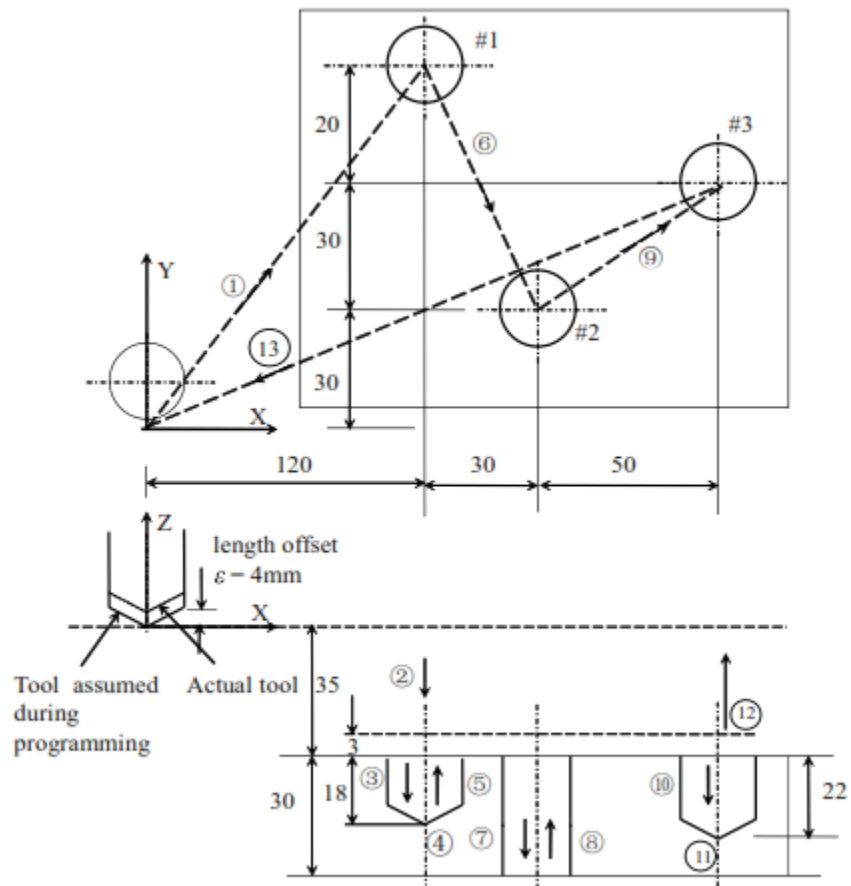


Figure 7.6 Tool Length Compensation - Example

```

%1050
G92 X0 Y0 Z0
G91 G00 X120 Y80 M03 S600
G43 Z-32 H01
G01 Z-21 F300
G04 P2
G00 Z21
X30 Y-50
G01 Z-41
G00 Z41
X50 Y30
G01 Z-25
G04 P2
G00 G49 Z57
X-200 Y-60
M05 M30

```

RTCP (Rotation Tool Center Point Programming)

RTCP (Rotation Tool Center Point Programming) refers to the auto tool length compensation when the spatial orientation of the tool changes.

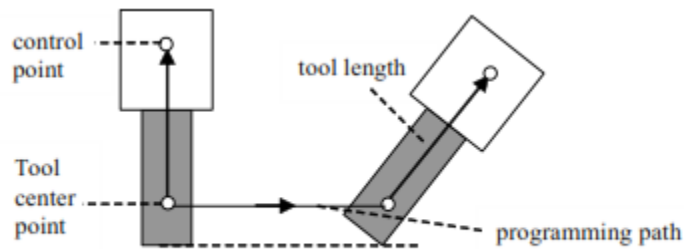


Figure 7.7 Rotation Tool Center Point Programming

G01 (linear interpolation), G00 (rapid positioning), and G02/G03 (circular interpolation) can be used in the rotation tool center point programming. G43, G44, G49 can also be used for the tool length compensation.

Miscellaneous Function

As it is mentioned in Chapter 1.8, there are two ways of execution when a move command and M code are specified in the same block.

1) Pre-M function

M command is executed before the completion of move command.

2) Post-M function

M command is executed after the completion of move command

There are two types of M code: one-shot M code, and modal M code.

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 51 of 73
---------------------------	-----------------	---	-----------	---------------

Table 8 1 Type of M code

Type	Meaning
One-shot M code	The M code is only effective in the block in which it is specified
Modal M code	The M code is effective until another M code is specified.

M code List

The following is a list of M command.

Table 8 2 M code List

CNC M-function	Type of Mode	Function	Pre/Post-M function
M00	One-shot	Program stop	Post-M function
M01	One-shot	Optional stop	Post-M function
M02	One-shot	End of program	Post-M function
M30	One-shot	End of program with return to the beginning of program	Post-M function
M98	One-shot	Calling of subprogram	Post-M function
M99	One-shot	End of subprogram	Post-M function
PLC M-function	Type of Mode	Function	Pre/Post-M function
M03	Modal	Spindle forward rotation	Pre-M function
M04	Modal	Spindle reverse rotation	Pre-M function
M05	Modal	▲ Spindle stop	Post-M function
M06	One-shot	Tool Selection	Post-M function
M07	Modal	Number1 Coolant on	Pre-M function
M08	Modal	Number2 Coolant on	Pre-M function
M09	Modal	▲ Coolant off	Post-M function

▲: default setting

CNC M-Function

Program Stop (M00)

M00 is one-shot M function, and it is post-M function.


The program can be stopped, so that the operator could measure the tool and the part, adjust part and change speed manually, and so on.

When the program is stopped, the spindle is stopped and the coolant is off. All of the current modal information remains unchanged. Resuming program could be executed by pushing “Cycle Run” button on the machine control panel.

Optional Stop (M01)

M01 is one-shot M function, and it is post-M function.

Similarly to M00, M01 can also stop the program. All of the modal information is maintained. The difference between M00 and M01 is that the operator must press

M01 button () on the machine control panel. Otherwise, the program would not be stopped even if there is M01 code in the program.

End of Program (M02)

M02 is one-shot M function, and it is post-M function.

When M02 is executed, spindle, feed and coolant are all stopped. It is usually at the end of the last program block. To restart the program, press “Cycle Run” button on the operational panel.

End of Program with return to the beginning Program (M30)

M30 is one-shot M function, and it is post-M function.

Similarly to M02, M30 can also stop the program. The difference is that M30 returns control to the beginning of program. To restart the program, press “Cycle Run” button on the operational panel.

Subprogram Control (M98, M99)

➤ End of Subprogram (M99)

M99 indicates the end of subprogram and returns control to the main program. It is one-shot M function, and it is post-M function.

➤ Calling a Subprogram (M98)

M98 P_ L_

P program number of the subprogram

L repeated times of subprogram

M98 is used to call a subprogram. It is one-shot M function. Moreover, it is post-M function.

PLC M Function

Spindle Control (M03, M04, M05)

M03 starts spindle to rotate CW at the set speed set in the program.

M04 starts spindle to rotate CCW at the set speed in the program.

<i>Ethiopian TVET Program</i>	STEP-GIZ	<i>CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations</i>	<i>July 2020</i>	<i>Page 53 of 73</i>
-----------------------------------	-----------------	--	------------------	----------------------

M05 stops spindle.

M03, M04 are modal M code, and they are pre-M function. M05 is modal M code, and it is post-M function. M05 is the default setting.

Tool Selection (M06)

M06 can select a desired tool to set on the spindle.

For example, M06 T01; the tool No.01 is selected.

M06 is one-shot M code, and it is post-M function.

Coolant Control (M07, M08, M09)

M07, M08 can turn on the coolant.

M09 can turn off the coolant.

M07 and M08 are modal M code, and they are pre-M function. M09 is one-shot M code, and it is post-M function. Moreover, M09 is the default setting.

Canned Cycles

To simplify programming, the canned cycle command can execute the specific operation using one G code, instead of several separated G commands in the program.

In general, a canned cycle consists of six operations.

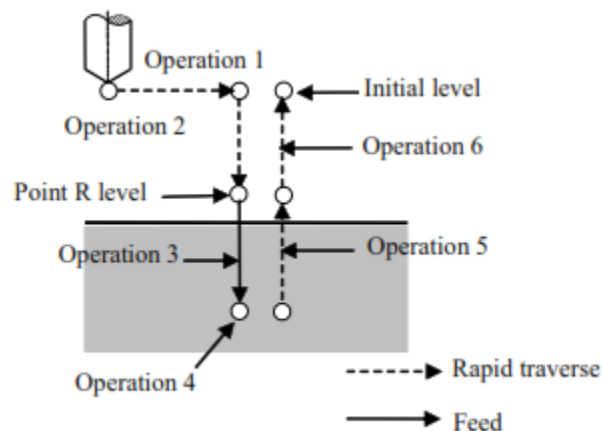


Figure 9.6 Sequence of canned cycle operation

- 1) Positioning to the initial point
- 2) Rapid traverse to point R

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 54 of 73
------------------------	----------	---	-----------	---------------

- 3) Hole machining
- 4) Operation at the bottom of hole
- 5) Retraction to point R
- 6) Retraction to the initial point

Return to the Initial Point/R point

Programming

$$\left. \begin{matrix} \{G98\} \\ \{G99\} \end{matrix} \right\} G_X_Y_Z_R_Q_P_I_J_K_F_L_$$

Explanation of the parameters

G98 Return to the initial point

G99 Return to point R level

The other parameters would be explain in the specific canned cycle.

Function

Generally, G99 is used for the first drilling operation. G98 is used for the last drilling operation.

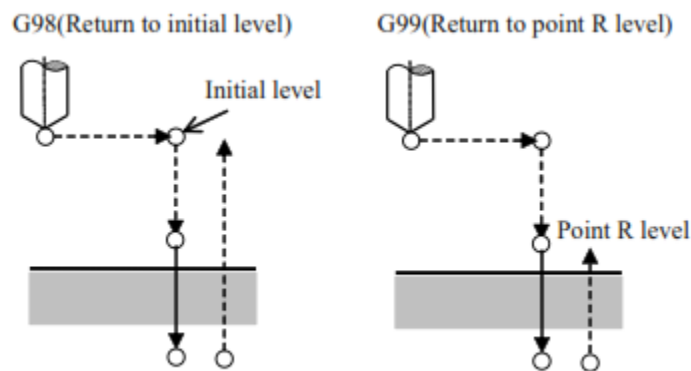


Figure 9.7 Return to the initial point/R point level

High-speed Peck Drilling Cycle (G73)

Programming

$$\left. \begin{matrix} \{G98\} \\ \{G99\} \end{matrix} \right\} G73 X_Y_Z_R_Q_P_K_F_L_$$

Explanation of the parameters

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 55 of 73
---------------------------	-----------------	---	-----------	---------------

- X,Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command
- Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command
- R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command
- Q Depth of cutting for each cutting feed in the incremental command
- P Dwell time at the bottom of a hole
- K Retraction amount at each time in the incremental command
- F Cutting feed rate
- L Number of repeats

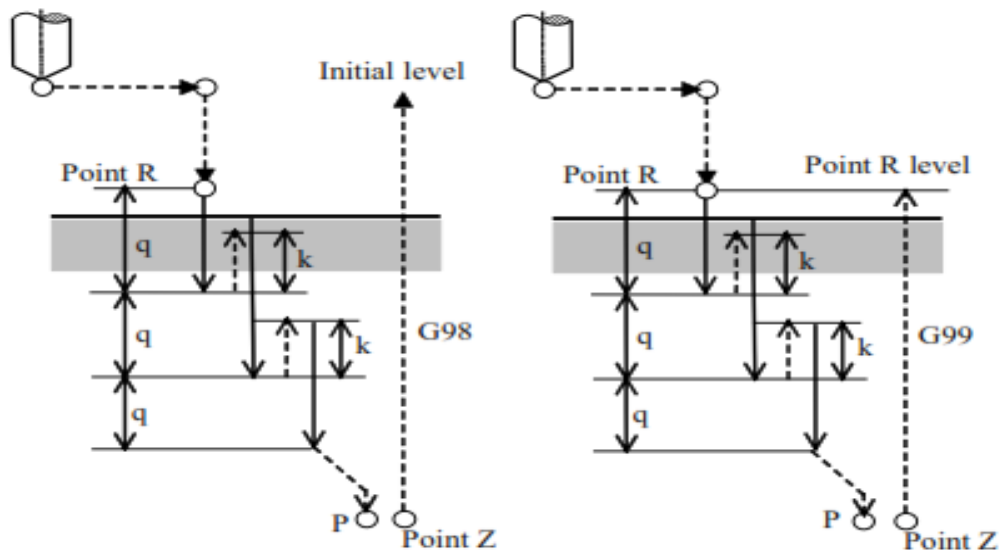


Figure 9.8 High-speed Peck Drilling Cycle

Function

This command can be used to drill a hole intermittently, so that the operator can remove the chips during machining.

Note

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 56 of 73
------------------------	-----------------	---	-----------	---------------

- 1) If the value of Z/K/Q is zero, G73 would not be performed.
- 2) $|Q| > |K|$

Example

Use a tool (10) to drill a hole.

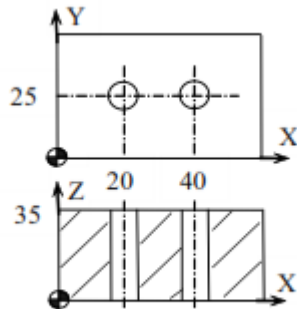


Figure 9.9 High-speed Peck Drilling – Example

```

%3337
N10 G92 X0 Y0 Z80
N15 M03 S700
N20 G00 Y25
N30 G98G73G91X20G90R40P2Q-10K2Z-3L2F80
N40 G00 X0 Y0 Z80
N45 M30

```

Left-hand Tapping Cycle (G74)

Programming

$$\left. \begin{matrix} \{G98\} \\ \{G99\} \end{matrix} \right\} G74 X_ Y_ Z_ R_ P_ F_ L_$$

Explanation of the parameters

- X,Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command
- Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command
- R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command
- P Dwell time at the bottom of a hole

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 57 of 73
---------------------------	-----------------	---	-----------	---------------

- F Thread lead
- L Number of repeats

Function

G74 command can create a reverse thread. Tapping is performed by turning the spindle counterclockwise. Then, the spindle turns clockwise for retraction when the tool reaches the bottom of the hole.

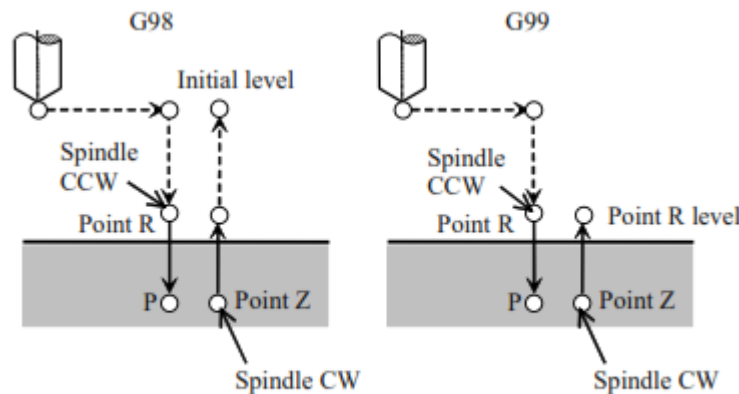


Figure 9.10 Left-hand Tapping Cycle (G74)

Note

If the value of Z is zero, G74 would not be performed.

Rigid Tapping Mode

There are two ways for the rigid tapping:

- 1) C-axis tapping: the tapping is performed along C-axis.
- 2) Z-axis tapping: the tapping is performed along Z-axis.

The default setting is Z-axis tapping. To set the C-axis tapping, M29 is used (M29 is modal M code). The format of setting the C-axis tapping is as follows:

M29 ; C-axis tapping is set as rigid tapping mode

G74 xx xxxxx ; C-axis tapping is performed

Example

Use the tool (M10 × 1) for the left-hand tapping.

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 58 of 73
---------------------------	-----------------	---	-----------	---------------

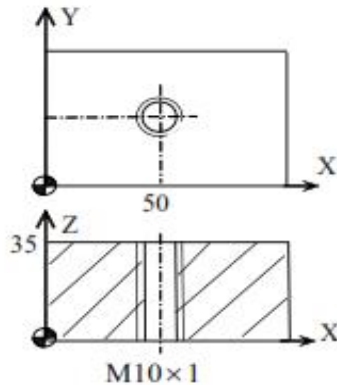


Figure 9.11 Left-hand Tapping – Example

```

%3339
N10 G92 X0 Y0 Z80 F200
N15 M04 S300
N20 G98G74X50Y40R40P10G90Z-5F1
N30 G00 X0 Y0 Z80
N40 M30

```

Fine Boring Cycle (G76)

Programming

```

{G98}
{G99} } G76 X_Y_Z_R_P_I_J_F_L_

```

Explanation of the parameters

- X,Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command
- Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command
- R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command
- P Dwell time at the bottom of a hole
- I Shift amount along X-axis at the bottom of a hole
- J Shift amount along Y-axis at the bottom of a hole
- F Cutting feed rate
- L Number of repeats

Function

G76 command would bore a hole precisely. When the tool reaches the bottom of the hole, the spindle stops, and the tool is retracted to the direction opposite to the tool tip.

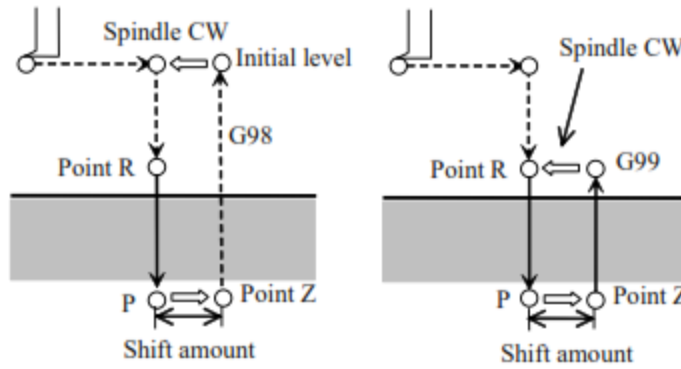


Figure 9.12 Fine Boring Cycle (G76)

Note

If the value of Z, G76 would not be performed.

Example

Use the tool to bore a hole on a workpiece.

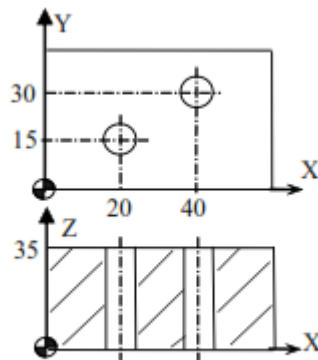


Figure 9.13 Fine Boring Cycle – Example

```

%3341
N10 G54
N12 M03 S600
N15 G00 X0 Y0 Z80
N20 G98G76X20Y15R40P2I-5Z-4F100
N25 X40Y30
N30 G00 G90 X0 Y0 Z80
N40 M30

```

Drilling Cycle, Spot Drilling (G81)

Programming

$$\left\{ \begin{array}{l} G98 \\ G99 \end{array} \right\} G81 X_Y_Z_R_F_L_$$

Explanation of the parameters

- X,Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command
- Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command

- R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command
- F Cutting feed rate
- L Number of repeats

Function

G81 command can be used to drill a hole. The tool is moved at the cutting feed rate when it is drilling a hole. Then, the tool is moved at the rapid traverse speed when it is retracting from the bottom of the hole.

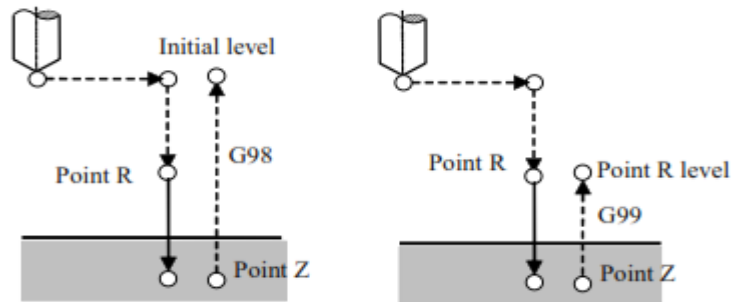


Figure 9.14 Drilling Cycle, Spot Drilling (G81)

Note

If the value of Z is zero, G81 would not be performed.

Example

Use the tool (10) to drill a hole.

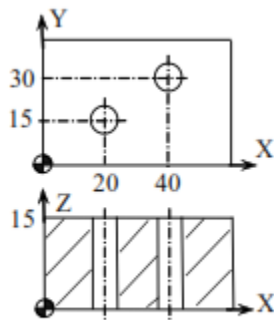


Figure 9.15 Drilling Cycle, Spot Drilling - Example

```
%3343
N10 G92 X0 Y0 Z80
N15 M03 S600
N20 G98G81G91X20Y15G90R20Z-3P2L2F200
N30 G00 X0 Y0 Z80
N40 M30
```

Drilling Cycle, Counter Boring (G82)

Programming

$$\left\{ \begin{array}{l} \text{G98} \\ \text{G99} \end{array} \right\} \text{G82 X_Y_Z_R_P_F_L_}$$

Explanation of the parameters

- X,Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command
- Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command
- R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command
- P Dwell time at the bottom of the hole
- F Cutting feed rate
- L Number of repeats

Function

G82 command can be used to drill a counter bore or a blind bore.

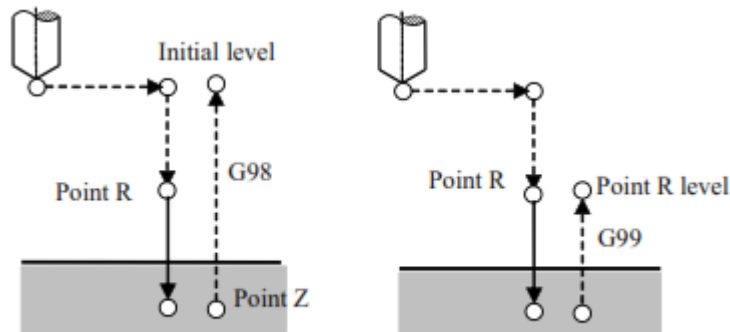


Figure 9.16 Drilling Cycle, Counter Boring Cycle (G82)

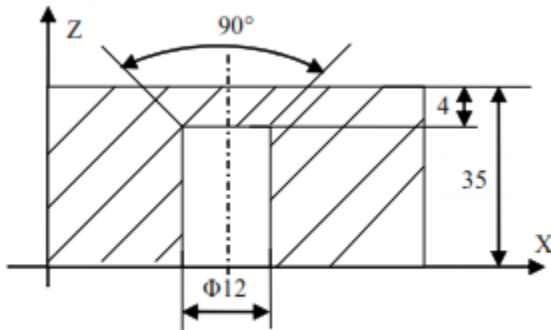
Note

If the value of Z is zero, G82 would not be performed.

Example

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 62 of 73
---------------------------	-----------------	---	-----------	---------------

Use the tool to drill a counter bore.



```
%3345
N10 G92 X0 Y0 Z80
N15 M03 S600
N20 G98G82G90X25Y30R40P2Z25F200
N30 G00 X0 Y0 Z80
N40 M30
```

Figure 9.17 Drilling Cycle, Counter Boring Cycle – Example

Peck Drilling Cycle (G83)

Programming

$$\left\{ \begin{array}{l} G98 \\ G99 \end{array} \right\} G83 X_Y_Z_R_Q_P_K_F_L_$$

Explanation of the parameters

- X,Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command
- Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command
- R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command
- Q Depth of cutting for each cutting feed in the incremental command
- P Dwell time at the bottom of the hole
- K Retraction amount at each time in the incremental command
- F Cutting feed rate
- L Number of repeats

Function

G83 command can perform a peck drilling. The intermittent cutting feed allows the operator to remove the chips from the hole.

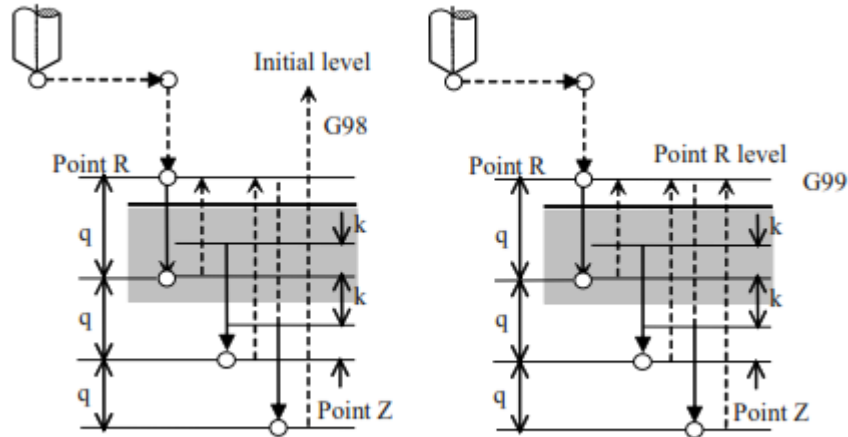


Figure 9.18 Peck Drilling Cycle (G83)

Note

If the value of Z/Q/K is zero, G83 would not be performed.

Example

Use the tool (F10) to drill a hole.

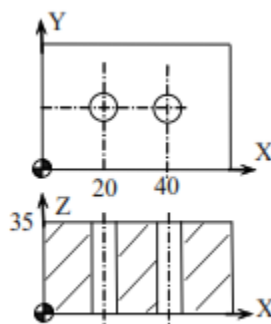


Figure 9.19 Peck Drilling Cycle – Example

```
%3347
N10 G55 G00 X0 Y0 Z80
N15 Y25
N20 G98G83G91X20G90R40P2Q-10K5G91Z-43F100L2
N30 G90 G00 X0 Y0 Z80
N40 M30
```

Tapping Cycle (G84)

Programming

$$\begin{cases} G98 \\ G99 \end{cases} G84 X_Y_Z_R_P_F_L_$$

Explanation of the parameters

- X,Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command
- Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command
- R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command
- P Dwell time at the bottom of the hole
- F Thread lead
- L Number of repeats

Function

G84 command is used for tapping. When the tool has reached the bottom of the hole, it is retracted in the reverse direction.

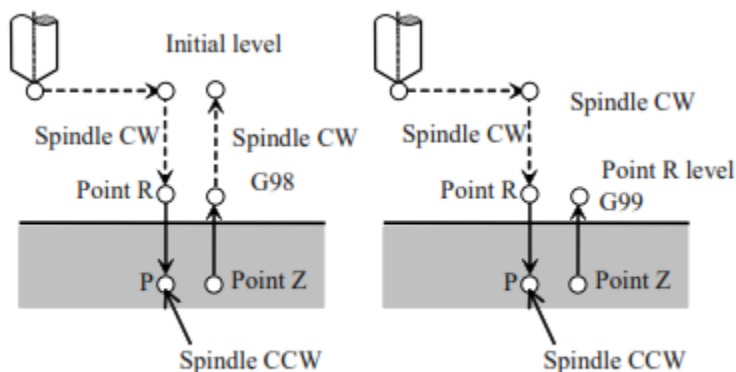


Figure 9.20 Tapping Cycle (G84)

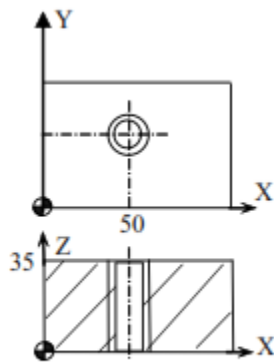
Note

If the value of Z is zero, G84 would not be performed.

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 65 of 73
------------------------	----------	---	-----------	---------------

Example

Use the tool (M10 * 1) to drill a hole.



```
%3349
N10 G92 X0 Y0 Z80
N15 M03 S300
N20 G98G74G91X50Y40G90R38P3G91Z-40F1
N30 G90 G0 X0 Y0 Z80
N40 M30
```

Figure 9.21 Peck Drilling Cycle – Example

Boring Cycle (G85)

Programming

$$\left\{ \begin{array}{l} G98 \\ G99 \end{array} \right\} G85 X_Y_Z_R_F_L_$$

Explanation of the parameters

- X,Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command
- Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command
- R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command
- P Dwell time at the bottom of the hole
- F Cutting feed rate
- L Number of repeats

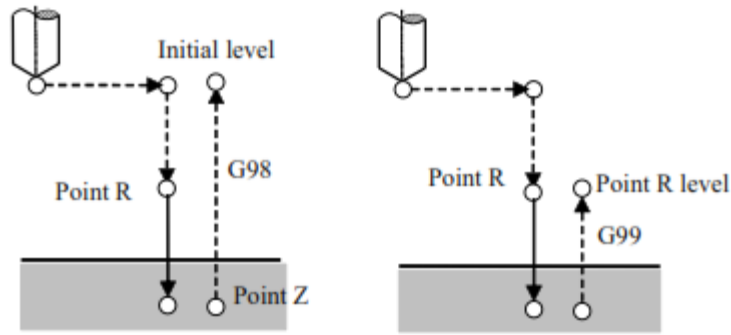


Figure 9.22 Boring Cycle (G85)

Function

G85 command is used to bore a hole, which is not required the precise boring.

Note

If the value of Z is zero, G85 would not be performed.

Example

Use the boring tool to bore a hole.

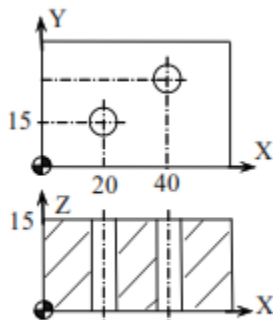


Figure 9.23 Boring Cycle – Example

```
%3351
N10 G92 X0 Y0 Z80
N15 M03 S600
N20 G98G85G91X20Y15R-42P2Z-40L2F100
N30 G90 G00 X0 Y0 Z80
N40 M30
```

Boring Cycle (G86)

Programming

$$\left\{ \begin{array}{l} G98 \\ G99 \end{array} \right\} G86 X_Y_Z_R_F_L_$$

Explanation of the parameters

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 67 of 73
---------------------------	-----------------	---	-----------	---------------

- X,Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command
- Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command
- R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command
- F Cutting feed rate
- L Number of repeats

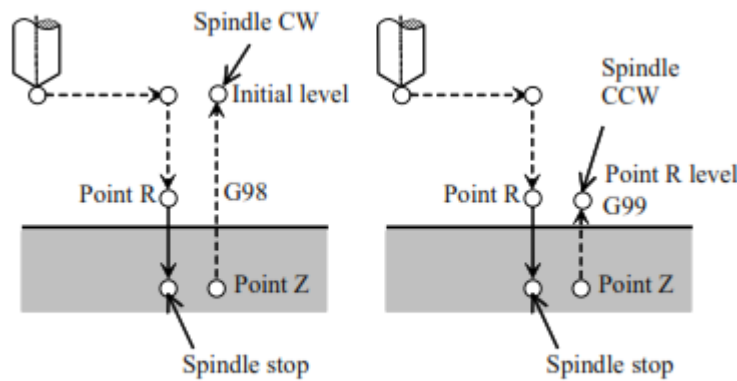


Figure 9.24 Boring Cycle (G86)

Function

G86 command is used to bore a hole, which is not required the precise boring.

Note

If the value of Z is zero, G86 would not be performed.

Example

Use the reamer to bore a hole.

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 68 of 73
---------------------------	-----------------	---	-----------	---------------

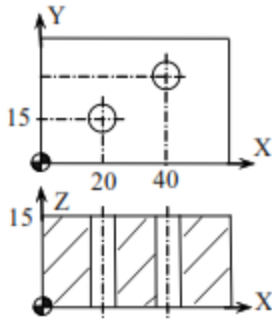


Figure 9.25 Boring Cycle – Example

```

%3353
N10 G92 X0 Y0 Z80
N15 G98G86G90X20Y15R38Q-10K5P2Z-2F200
N20 X40 Y30
N30 G90 G00 X0 Y0 Z80
N40 M30

```

Back Boring Cycle (G87)

Programming

G98 G87 X_ Y_ Z_ R_ P_ I_ J_ F_ L_

Explanation of the parameters

- X,Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command
- Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command
- R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command
- P Dwell time at the bottom of the hole
- I Shift amount along X-axis at the bottom of a hole
- J Shift amount along Y-axis at the bottom of a hole
- F Cutting feed rate
- L Number of repeats

Function

G87 command is used to bore a hole.

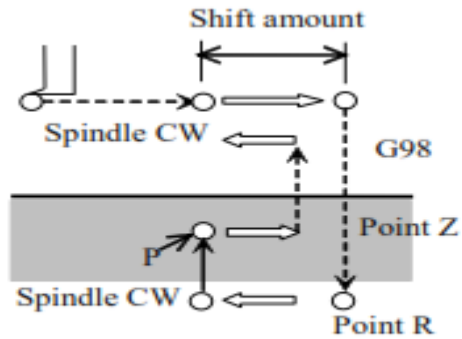


Figure 9.26 Boring Cycle (G87)

Note

- 1) If the value of Z is zero, G87 would not be performed.
- 2) G87 cannot be used with G99.

Example

Use the reamer to bore a hole (F28).

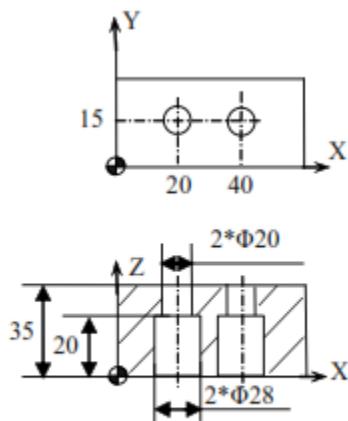


Figure 9.27 Back Boring Cycle – Example

```

%3355
N10G92 X0 Y0 Z80
N15M03 S600
N20G00 Y15 F200
N25G98G87G91X20I-5R-83P2Z23L2
N30G90 G00 X0 Y0 Z80 M05
N40M30

```

Manual Boring Cycle (G88)

Programming

$$\left. \begin{matrix} \{G98\} \\ \{G99\} \end{matrix} \right\} G88 X_Y_Z_R_P_F_L_$$

Explanation of the parameters

- X,Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command
- Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command
- R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command
- P Dwell time at the bottom of the hole
- F Cutting feed rate
- L Number of repeats

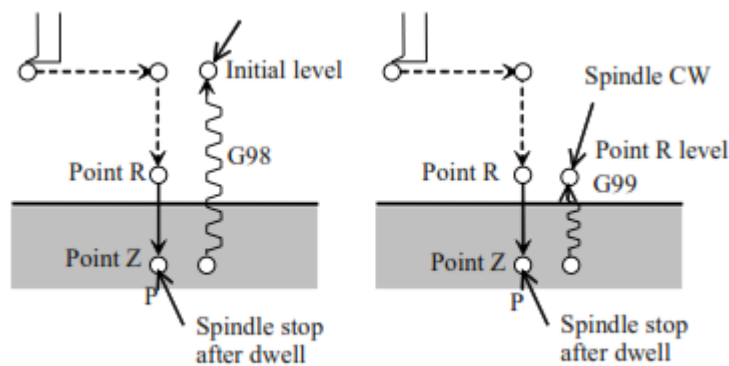


Figure 9.28 Manual Boring Cycle (G88)

Function

G88 command can be used to bore a hole precisely. The main difference is that the tool is returned to the initial point or the point R in the manual mode key (on the machine control panel).

Ethiopian TVET Program	STEP-GIZ	CT program for Remote Teaching Title: Machining L-3 Perform Advanced CNC Milling Operations	July 2020	Page 71 of 73
---------------------------	-----------------	---	-----------	---------------

Note

- 1) If the value of Z is zero, G88 would not be performed.
- 2) When manually moving the tool for retraction, the position of tool must be higher

Example

than the initial point (G98) or the point R (G99).

Use the reamer to bore a hole.

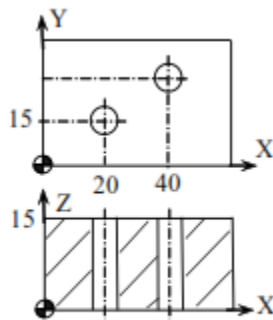


Figure 9.29 Manual Boring Cycle – Example

```
%3357  
N10 G54  
N12 M03 S600  
N15 G00 X0 Y0 Z80  
N20 G98G88G91X20Y15R-42P2I-5Z-40L2F100  
N30 G00 G90 X0 Y0 Z80  
N40 M30
```

Boring Cycle (G89)

Programming

$$\left\{ \begin{array}{l} G98 \\ G99 \end{array} \right\} G89 X_Y_Z_R_P_F_L_$$

Explanation of the parameters

- X,Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command
- Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command

- R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command
- P Dwell time at the bottom of the hole
- F Cutting feed rate
- L Number of repeats

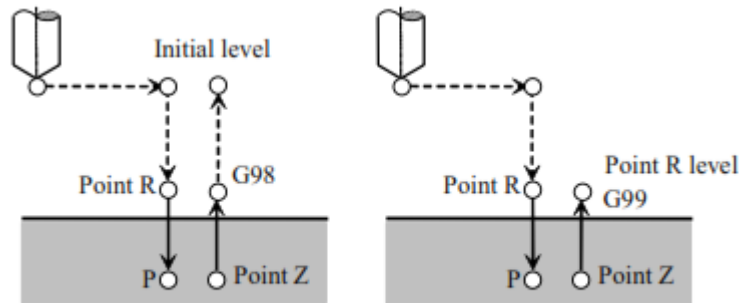


Figure 9.30 Boring Cycle (G89)

Function

G89 command is used to bore a hole.

Note

If the value of Z is zero, G89 would not be performed.

Canned Cycle Cancel (G80)

Programming

G80

Explanation the parameters

G80 a canned cycle is cancelled. Meanwhile, the point R and the point Z (the bottom of the hole) are all cancelled.

Function

G80 is used to cancel a canned cycle.