Milling machine programming manual

Programming section

Please read this manual thoroughly and fully understand its contents beforeusing Lynuc numerical control system.
Please designate a custodian to keep it safely in the designated location so thatit can be read at any time.

Shanghai Rhenac numerical control technology co., ltd

Shanghai Lynuc CNC Technology Co., Ltd.

Overview

About this manual

- Manual Name： Programming Manual for Milling Machine of Numerical Control System
- Document type:： Programming instructions for milling machine series numerical control system
- Version： Ver1.0

Readers of this manual

- Electrical Engineer/Product Technician/Technical Service Personnel/Product User

Operational premise

- Familiar with relevant concepts in this manual
- Trained in the operation of rhenium sodium control device

Symbolic description

- T Series: Lathe System (Turning)
- M Series: Milling Machine System (Milling)
- T/M: universal for lathe/milling machine systems
- Note: Supplementary explanation of narrative content

Manual version history

Version	Release date	Revision notes
Ver1.0	2021/2/18	1. Initial release

DIRECTORY

I. Chapter

chapter is an overview

1. Product Overview

Overview

This manual introduces the basic knowledge, machining instructions, programming examples and diagrams, high-speed contour control function and macro function programming and explanation of the system programming based on NC device for milling machine developed by Re NAC.

The actual selection function of the numerical control device of the machine tool should also refer to the instructions issued by various machine tool manufacturers. In addition, the specifications and usage methods of the machine tool operation panel may be different. Please refer to the instructions issued by the machine tool manufacturer.

2. General Operation of CNC Machine Tool

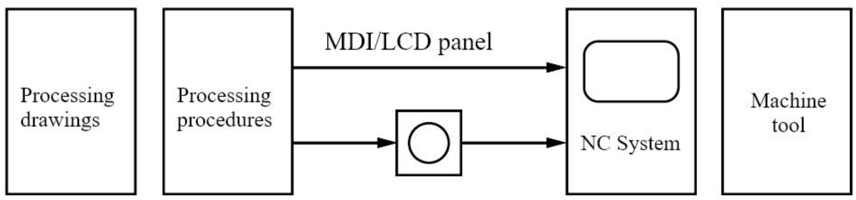
Overview

When machining parts with CNC machine tools, we should first use NC language to compile machining programs, and then use the programs to control CNC Machine tools.

【Steps】:

(1) Firstly, according to the machining drawings, the parts machining program is compiled. In this manual, "Chapter 2 Programming" introduces the programming method of rhenium sodium numerical control device in detail, and lists typical programming examples, illustrations and matters needing attention.

(2) (2)CNC read into the program, the parts and cutter installed on the machine tool, the cutter according to the program movement, machining the actual parts. For actual operation, please refer to "Rhenium Sodium Numerical Control System Manual-Operation Chapter".



3. Basic matters of safe operation

Overview

This manual contains safety precautions to ensure the safety of programmers and prevent damage to controllers, and according to their importance in safety, they are described as "dangers", "warnings" and "precautions", and relevant supplementary instructions are described as "instructions".

Before use, you must thoroughly read the items described in these "dangers", "warnings", "precautions" and "instructions".

Danger

Indicates that if this danger cannot be avoided, the result is likely to lead to serious injury or death.

Warning

Indicates that if this danger cannot be avoided, there is a potential danger of serious injury or death.

Attention

Indicates that if this precaution is violated, the equipment may be damaged or its life may be shortened.

Description

Point out additional explanations besides dangers, warnings and precautions

Safe job matters related to programming

The following describes the safety operation matters in programming. For the safe use of this equipment, please read carefully and observe the following items.

Warning

1. On the setting of coordinate system:

   The setting of warning coordinate system is very important. If t he setting is wrong, even if the program operation instruction is correct, it will probably lead to abnormal operation of the machine tool. Thereby damaging machine tools and other devices, and even hurting equipment operators.

2. About the input of data units:

   This system supports input modes of inches and millimeters. The two can be converted to each other. However, during mutual conversion, various parameters, current position, etc. are not changed. Therefore, before running the machine tool, please fully confirm the correctness of such data. If the wrong data is used for operation, it may also damage machine tools and other objects, and even hurt equipment operators.

3. About the constant control of weekly speed:

   In the constant weekly speed control, the closer the workpiece coordinate system of the control shaft is to zero, the more block the speed is. If you get too close, you will get too fast. Therefore, before operating this function, please specify the maximum speed of the spindle correctly, so as to avoid damaging the machine tool and even causing personal injury.

4. About the speed of the rotating shaft:

   When interpolating polar coordinates, please pay attention to controlling the speed of the rotating shaft. If the speed is set too fast, or the workpiece installation method is wrong, the workpiece will fall off, damage tools, machine tools, etc., and even hurt equipment operators.

Attention

1. About Absolute and Incremental Features:

   Note that when entering absolute instructions and incremental instructions, the corresponding programs must be corresponding. If the program written with absolute value is executed in incremental mode, or the program written with incremental value is executed in absolute mode, it may lead to abnormal operation of the machine tool.

2. About plane selection function:

   If the correct plane is not selected during circular interpolation spiral interpolation and fixed cycle it will also lead to abnormal operation of the machine tool.

3. Regarding the compensation function:

   This function cannot be performed at the same time as the function of mechanical coordinate system and reference point return. Otherwise, the compensation instruction will be temporarily cancelled, resulting in abnormal operation of the machine tool. Therefore, before executing the above instructions, please cancel the compensation function mode first.

Description

1. Indicates that if this precaution is violated, the equipment may be dam aged or its life may be shortened.

2. Point out additional explanations besides dangers, warnings and precautions.

II. Chapter Programming section

1. Preparation function (G function)

GG instruction is also called preparation function. According to the instruction class G and the following values, the NC device makes relevant preparations for what machining method is used in the designated block or how the shaft moves.

Indicates the meaning of the block command according to the value after the instruction category G. There are two kinds of G instructions.

Table 1-1 G Code Types

Species	Meaning
Primary G instruction	The G instruction is valid only in the specified block. Block: the minimum unit required for equipment action. Sometimes equivalent to a 1-line concept.
Modal G instruction	The G instruction is valid until other G instructions of the same set are executed.

For example, G01 and G00 are modal G instructions (G instructions other than 00 groups).

```
G01 X_ Z_
X_          } In the meantime, G01 is valid
Z_
G00 X_ Z_
```

Attention

(1) When the groups of G instructions are different and there is no mutual exclusionbetween the instructions, several of them can be executed in the same block G Instruction.

(2) When two or more G instructions in the same group are executed in the same block, or there is mutual exclusion between instructions, When the system is set, a warning will appear on the screen.

(3) Once a G instruction that is not in the G instruction list is executed, a warning will appear on the screen.

(4) The G instruction with the VA mark in the G instruction list is used by default when the power is turned on.

Table 1- 2 List of G InstructionG

G instruction	Group	Function
G00	01	Positioning (fast movement)
The G01		Linear interpolation (cutting feed)
G02		Clockwise arc interpolation (CW)
G03		Counterclockwise circular interpolation (CCW)
G02.4		Spatial circular interpolation
G03.4		Spatial circular interpolation
G04	0	Delay
G05		High Speed Contour Control Function (G-ACC)
G05.1		High speed and high precision parameter selection
G09		Accurate stop
G10	20	Programmable data input
G11		Programmable data input cancellation
TheG15	22	Cancel polar coordinate command
G16		Turn on polar coordinate command
The G17	2	Select XY plane
G18		Select ZX plane
G19		Select YZ plane
G20	6	Inch input
The G21		Millimeter input
G27	20	Reference point reversion detection
G28		Automatic return to reference point
G29		Reverse from reference point
G30		Return to Reference Points 2, 3, 4
G32		Coordinate reading function
G31	0	Uniaxial high-speed measurement
G31.2		Multi-axis linkage measurement
The G40		Cancel tool radius compensation

G41	7	Left compensation of tool radius
G42		Right compensation of tool radius
The G40.1	23	Cancel the radius compensation of five-axis tool
G41.1		Left compensation of five-axis tool radius
G42.1		Five-axis tool radius right compensation
G43	8	Tool length compensation (positive direction)
G44		Tool length compensation (negative direction)
G43.1		Five-axis fixed-axis machining
G43.4		Five-axis linkage machining
The G49		Cancel tool length compensation
The G50	11	Unzoom
G51		Zoom
The G50.1	18	Unmirroring
G51.1		Mirror image
G52	0	Local coordinate system setting
G53		Select mechanical coordinate system
G53.1		Tool axis direction control
G54.1	14	Select supplementary workpiece coordinate system (P1 ~ P54)
The G54 G55 G56 G57 G58G59		Select workpiece coordinate system
G61	15	Accurate stop
The G64		Cutting mode
G65	0	Macro program call
G66	12	Macro modal call
The G67		Macro modal call cancellation
G68	16	Coordinate rotation
◤G69		Cancel coordinate rotation

G68.2	19	Inclined surface machining
G68.3		Inclined surface machining
◤G69.2		Cancel the machining of inclined surface
G70	30	Circumferential mode
G71		Arc mode
G72		Linear mode
G73	9	High-speed deep drilling circulation
G74		Reverse rigid tapping cycle
G76		Fine boring cycle
◤G80		Fixed cycle cancellation for drilling
G81		Drilling cycle
G82		Drilling cycle
G83		Wood pecking deep hole drilling cycle
G84		Rigid tapping cycle
G85		Fine boring cycle
G86		Boring cycle
G87		Back boring cycle
G80.1		Cancel the high-speed drilling cycle
G81.1		High speed drilling cycle
G73.4		Inclined high-speed deep drilling cycle
G74.4		Inclined reverse rigid tapping cycle
G81.4		Inclined drilling cycle
G82.4		Inclined drilling cycle
G83.4		Inclined wood pecking deep hole drilling cycle
G84.4		Inclined rigid tapping cycle
G85.4		Inclined fine boring cycle
G86.4		Inclined boring cycle
◤G90	3	Absolute instruction
G91		Incremental instruction
G90.1	4	Absolute modal input of circular arc center coordinates

◤G91.1		Incremental modal input of circular arc center coordinates
G92	0	Set the workpiece coordinate system
G92.1		Workpiece coordinate system preset function
◤G94	5	Feed per minute
G95		Per turn feed
◤G98	10	Fixed loop back to the initial plane
G99		Fixed loop returns to R point plane
G110	20	Automatic length measurement (Option)
G150		Emergency stop rollback function
The G150.1	25	Allow handwheel simulation switching
G151.1		Handwheel analog switching is prohibited
G154…G954 … G159…G959	14	Select supplementary workpiece coordinate system
G160.1		Two-way milling of circular plane
G160.2		Two-way milling of rectangular plane
G160.3		Co-directional milling of rectangular plane
G161.1	20	Two-way milling of circular cavity
G161.2		Two-way milling of rectangular cavity
G162.1		Milling inner circle
G162.2		Cylinder milling
G162.3		Milling inner rectangle
G162.4		Milling inner rectangle (fillet)
G162.5		Milling outer rectangle
G162.6		Milling inner circle (helix)
G162.7		Milling excircle (helix)
G163.1		Rectangular frame drilling
G163.2		Rectangular mesh drilling
G163.3		Straight drilling

G164.1		Rectangular frame tapping
G164.2		Rectangular mesh tapping
G164.3		Straight tapping

2.Interpolation function

2.1Summary

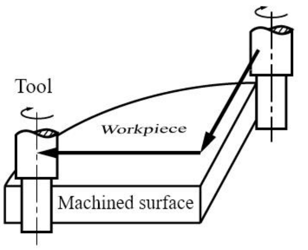
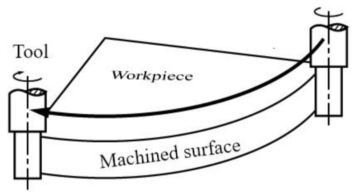
【Functions】：

The cutter moves along the workpiece in a straight line or circular arc.

【Classification】：

Linear interpolation and circular interpolation.

Table 2- 1 Interpolation Function

Linear interpolation	Arc interpolation
	
G01 X_ Y ; X_ ;	G02 X_ Y_ R_ ;

2.2Quick Location (G00)

Overview

【Functions】：

In the non-cutting state, the tool moves at a fast-forward speed to the position in the workpiece coordinate system specified by the absolute value instruction or the incremental value instruction.

【Instruction Format】：

G00 X_ Y_ Z_ ;

Instruction description

(1) The fast forward speed can be adjusted in the range of 0 ~ 100% through the feed ratio switch of the operation panel. If it exceeds the range of 100%,it will be processed according to the magnification of 100%.

(2) Fast forward speed cannot be specified by F instruction. Move according to The setting of [System-Parameters-Path-Fast Forward Speed] and the maximum motor speed of each shaft.

2.3Linear interpolation (G01)

Overview

【Functions】：

The cutter moves in a straight line to the specified position.

【Instruction Format】：

G01 X_ Y_ Z_ F_ ;

Instruction description

(1) The tool moves in a straight line to the specified position at the feed speed specified by F. The specified feed speed is valid until a new value is specifie Therefore, there is no need to specify F for each program segment.

(2) The feed speed specified with F instruction is measured along a straight track. If F instruction is not specified, the feed speed is the last specified speed.

(3) 机 The default feed speed is used when starting the machine tool, and the default feed speed is set in "System-Parameters-Common Use" (# 32961). The feed speed in each axis direction is as follows:

G01 Xx Yy Zz Cc Ff;

Feed speed in X axis： $Fx = \frac{Lx}{L} \times f$

Feed speed in the direction of Y axis： $Fy = \frac{Ly}{L} \times f$

Feed speed in Z axis： $Fz = \frac{Lz}{L} \times f$

Feed speed in the direction of axis ： $Fc = \frac{Lc}{L} \times f$

$$L = \sqrt{Lx^2 + Ly^2 + Lz^2 + Lc^2}$$

Where L denotes the combined moving distance, and Lx, Ly, Lz, and Lc denote components of the moving distance in each axis direction, respectivelyL

(4) The feed speed of the rotating shaft, in degrees/minutes.

(5) When linear interpolation is performed on a linear axis (e.g. X) and a rotation axis (e.g. C), the speed specified by F (mm/min) is a tangent feed speed in the X and C rectangular coordinate systems.

(6) Acquisition of C-axis feed speed: First calculate the speed required for distribution using the above formula, and then convert the speed unit into degrees/minutes.

Program legend

【Example1】：Linear interpolation

(G90) G01 X150. Y100. F500

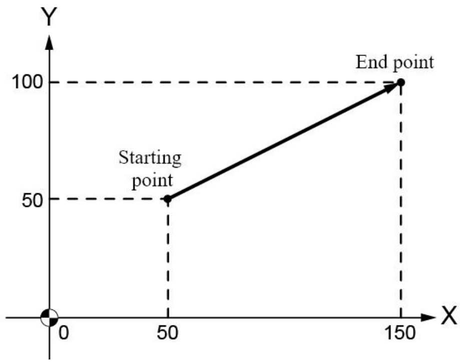


Figure2- 1 G01 Line Interpolation

【Example2】：Rotary axis interpolation

G90 G01 C-90. F300

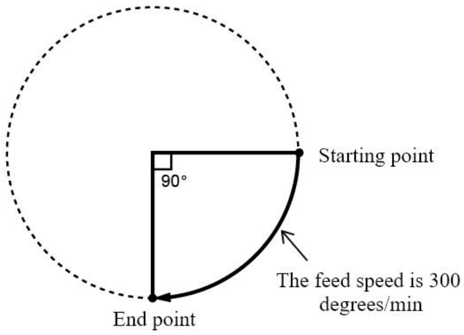


Figure 2- 2 Rotary axis interpolation

2.4Plane selection (G17, G18, G19)

Overview

【Functions】:

The plane to be used must be specified when selecting the plane to be used, using arc interpolation function (G02, G03), tool radius compensation (G40, G41, G42), coordinate rotation function (G68, G69) and drilling cycle.

【Instruction Format】:

G17 ;	Select XY plane
G18 ;	Choose ZX Plane
G19 ;	Choose YZ Plane

Instruction description

(1) When the power supply is turned on, one of G17 (XY plane), G18 (ZX plane) and G19 (YZ plane) is selected by parameters.

(2) When powered on, the default is G17 (XY plane).

(3) The plane remains unchanged in the program segment where G17, G18 and G19 are not specified. Move command has nothing to do with plane selection.

2.5Arc interpolation (G02, G03)

Overview

【Functions】:

The tool machines the specified arc at a feed rate (F) in either a clockwise direction (G02) or a counterclockwise direction (G03).

【Instruction Format】:

G17 G○○ X_ Y_ I_ J_ F_ ;	G17 Specifies the arc of the XY plane
G17 G○○ X_ Y_ R_ F_ ;	
G18 G○○ X_ Z_ I_ K_ F_ ;	G18 Specifies the arc of the ZX plane
G18 G○○ X_ Z_ R_ F_ ;	
G19 G○○ Y_ Z_ J_ K_ F_ ;	G19 Specifies the arc of the YZ plane
G19 G○○ Y_ Z_ R_ F_ ;	

G○○	:	G02 corresponds to clockwise direction G03 corresponds to counterclockwise direction
I_	:	((G90.1 Absolute) Center of Arc on X Axis (G91.1 increment instruction) The distance from the starting point of the X axis to the center of the arc
J_	:	(G90.1 Absolute) Center of Arc on Y Axis (G91.1 increment instruction) The distance from th e starting point of the Y axis to the center of the arc
K_	:	(G90.1 Absolute) Center of the arc on the Z axis (G91.1 increment instruction) The distance from the starting point of the Z axis to the center of the arc
R_	:	The radius value of an arc when R is used to specif y the arc
F_	:	Feed speed when performing circular interpolation

Instruction description

(1) Direction of arc interpolation: In rectangular coordinate system, when viewed from the positive to negative direction of Z axis (Y or X axis)When XY plane (ZX or YZ plane), determine the "clockwise" of XY plane (ZX or YZ plane)(G02) and "counterclockwise" (G03). As shown in the following figure.

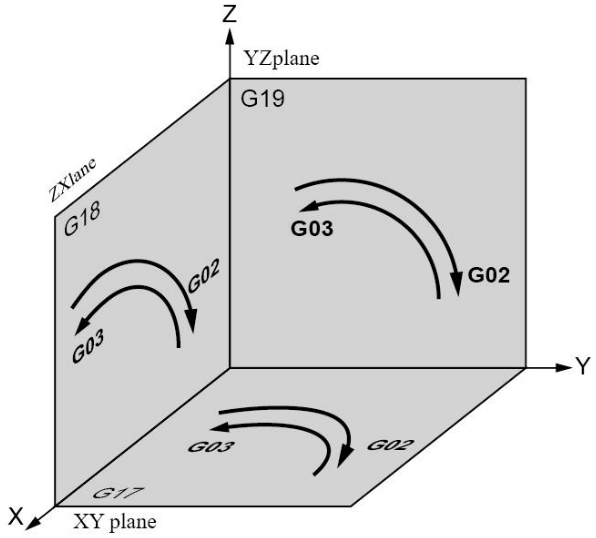


Fig. 2- 3 Direction of Arc Interpolation

(2) The arc instruction executed by the R instruction. When the arc is less than 180, R is positive; R is negative when the arc exceeds 180.

(3) When machining the whole circle, please use I and J instructions. If the R command is used to process a whole circle, an alarm will be displayed.

(4) If the address I, J, K and R instructions are specified at the same time, the R instruction is used.

(5) When the command is close to an arc with a central angle of 180, the calculated center coordinates may have a large error, resulting in a shape Shape size out of tolerance. In this case, it is recommended that the user use the I, J, K instructions to specify the center of the arc.

Program legend

【Example1】：

R is positive when the arc is less than 180.

G91 G01 X0 Y0 F350

G02 X30. Y70. R80. F300

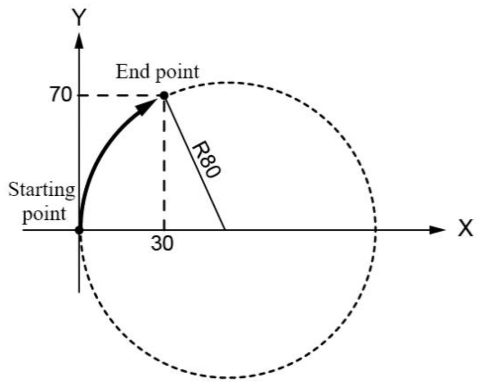


Fig. 2-4 2-4 The R command executes the arc command less than 180

【Example2】：

R is negative when the arc exceeds 180.

G91 G01 X0 Y0 F350

G02 X30. Y70. R-80. F300

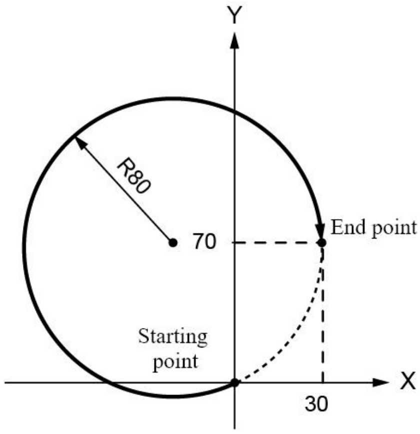
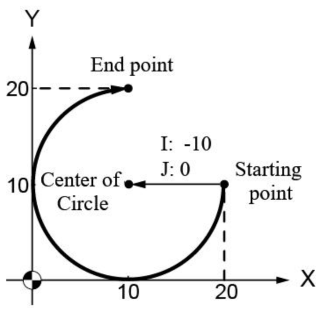
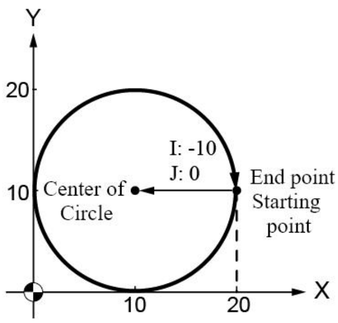


图 Fig. 2- 5 2-5 The R command executes a circular arc command greater than 180

【Example3】：

Machining circular arc and whole circle with I and J instructions

Table2- 2 Machining Arc and Whole Circle with I and J Instructions

Circular arc	Whole circle
	
G90 G00 X20. Y10. G02 X10. Y20. I-10. J0 F350	G90 G00 X20. Y10. G02 X20. Y10. I-10. J0 F350

2.6Spiral interpolation (G02, G03)

Overview

【Functions】：

Synchronized with the action of arc interpolation, linear interpolation can be carried out on the axis that does not belong to the arc plane, and the motion of spiral rotating cutter is called spiral interpolation.

【Instruction Format】：

G17 G○○ X_ Y_ R_ α_ (β_) F_ ;	G17 Specifies XY Plane
G17 G○○ X_ Y_ I_ J_ α_ (β_) F_ ;	
G18 G○○ Z_ X_ R_ α_ (β_) F_ ;	G18 Specifies ZX Plane
G18 G○○ Z_ X_ K_ I_ α_ (β_) F_ ;	
G19 G○○ Y_ Z_ R_ α_ (β_) F_ ;	G19 Specifies the YZ plane
G19 G○○ Y_ Z_ J_ K_ α_ (β_) F_ ;	

G○○	：	G02 corresponds to clockwise direction; G03 corresponds to counterclockwise direction
α_ (β_)	：	Alpha and beta generation refer to any a xis not in the selected plane

Instruction description

(1) When specifying the arc feed speed including the straight axis:

$$\text{Tangent velocity of circular arc} = F * \frac{\text{Arc length of circular arc}}{\sqrt{(\text{Arc length of circular arc})^2 + (\text{Axial length of straight line})^2}}$$

$$\text{Linear axis velocity} = F * \frac{\text{Axial length of straight line}}{\sqrt{(\text{Arc length of circular arc})^2 + (\text{Axial length of straight line})^2}}$$

(2) Tool length compensation cannot be specified in the program section where spiral interpolation is specified.

(3) Tool radius compensation is only applied to circular arcs.

2.7 Space Arc (G02.4, G03.4)

Overview

【Functions】：

By using the method of three-point circle determination, the spatial arc can be generated only by specifying the non-collinear midpoint and end point in NC.

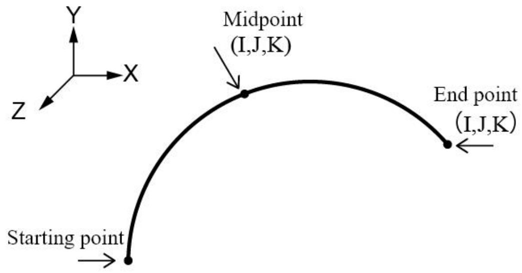


Fig. 2-6 Schematic diagram of spatial circular arc

【Instruction Format】:

G02.4 / G03.4 X_ Y_ Z_ A_ B_ C_ I_ J_ K_ L_ P_ Q_ R_ ;		
X_ Y_ Z_ A_ B_ C_	:	Coordinates of the end point
I_ J_ K_ L_P_ Q_	:	Midpoint coordinates corresponding to XYZABC
R_	:	Whole circle radius

Instruction description

(1) Three-point circle does not need to judge clockwise and counterclockwise problems, so G03.4 and G02.4 have the same function.

(2) Conditions to be met for starting point-midpoint-end point:

a) Three points are not collinear;

b) There is no coincidence point between the starting point-midpoint-end point (also belonging to the collinear category).

(3) In G91 mode: The specified midpoint increment is based on the starting point and the specified end point increment is based on the midpoint.

(4) R instruction description: When R is specified, it means that it is a whole circle in space at present, but the arc radius is determined by the circle radius of the three-point fixed circle, which has nothing to do with the specified value after R. When R is not specified, the current instruction is a spatial arc, not a whole circle.

(5) Spatial arcs do not support the following situations:

a) Lathe system;

b) In radius compensation mode;

c) In the fixed cyclic mode;

d) In GACC0 mode;
e) In GACC1 mode, there are axis shifts except XYZ;
f) Single-step pause in three-dimensional circular arc movement;
g) In the mirror rotation scaling mode.

(6) Spatial arc correlation error reporting:
a) Error ID 3058: NC program error, space arc command and the current mode command conflict. Example: lathe system; In radius compensation mode; In the fixed cyclic mode; mirror scaling rotation mode; GACC0 mode.
b) Error ID 5034: GACC program error, in current GACC mode, the spatial arc instruction specifies an unsupported axis shift.
Example: There is axial motion other than XYZ in GACC1 mode.
c) Error ID 3057: NC program error, in the spatial arc instruction, the starting point, midpoint and end point are collinear or coincident.
d) Error ID 0049: Single step is not allowed in spatial arc instruction.

Program legend

【Example1】：

G02.4 / G03.4 X100 Y100 Z100 I10 J10 K10 (all XYZ midpoints and endpoints specified)

G02.4 / G03.4 X100 Y100 Z100 A100 B100 I10 J10 K10 R10 P10 (XYZAB midpoint and terminalPoint all specified)

G02.4 / G03.4 X100 A100 B100 I10 J10 K10 R10 P10 (default at YZ endpoint)

G02.4 / G03.4 X100 A100 B100 I10 R10 P10 (default at YZ midpoint)

G02.4 / G03.4 Z100 R10 P10 A100 B100 I10 X100 J10 Y100 K10 (the coordinates of end point and midpoint are chaotic Order Specification)

【Example2】：

Three-dimensional arc drawing plane arc:

starting point: X0 Y0 Z0

Analog G02 instruction: G17 G02 X100 Y0 Z0 R50

Space arc instruction: G02.4 X100 Y0 Z0 I50 J50 K0

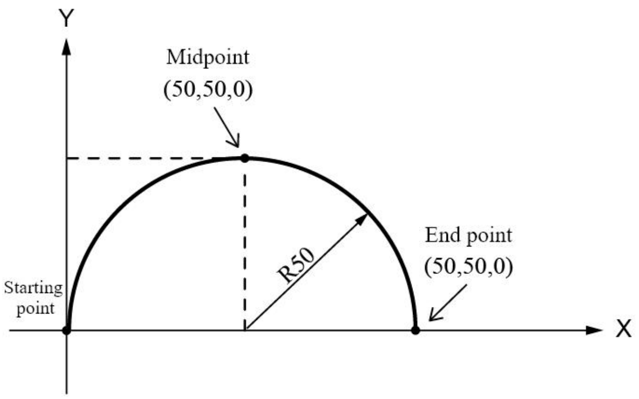


Figure 2- 7 Three-dimensional circular arc drawing plane circular arc

【Example3】：

Three-dimensional arc drawing space arc:

starting point X0 Y0 Z0

Space arc instruction: G02.4 X70.721 Y70.721 Z0 I35.36 J35.36 K50

【Example4】：

Spatial arc command is greater than 3 axes:

Example NC: G02.4 X100 Y0 Z0 A10 I50 J50 K0 R80

This is the midpoint (50, 50), the end point (100, 0), the starting point of the arc of the center (50, 0) to the midpoint: the A axis

moves from 0 to 80

Midpoint to end point: Axis A moves from 80 to 10

【Example5】：

Error instruction format:

G02.4/G03.4 I10 J10 K10 (only specify midpoint coordinates)

G02.4/G03.4 X100 Y100 Z100 (only specify end point coordinates)

Attention

At least one axis must be specified for the midpoint coordinate and the end point coordinate respectively, and only specifying the midpoint coordinate or theend point coordinate is an error command.

2.8Cylindrical Interpolation (G07.1)

Overview

【Functions】：

The movement of the rotation axis specified by the angle is converted into the movement along the circumference, and the linear interpolation and circular interpolation are carried out between the rotation axis and other axes, which is convenient for the interpolation movement of programming the unfolded diagram of the cylindrical surface directly.

【Instruction Format】：

G07.1 C（r）；

C ： Workpiece radius

Instruction description

(1) Specify Cylindrical Interpolation Start and Cylindrical Interpolation Cancel in a single program segment.

【Example】：

G07.1 C（r） Cylinder Interpolation Begins

G07.1 C0 Cylindrical interpolation cancellation

(2) When interpolating a cylinder, only one rotation axis can be specified. When G17 ~ G19 (plane selection) is specified, the rotation axis is regarded as a straight line axis.

【Example】：

When the axis of rotation C is parallel to the axis of X, you can select X and Y by specifying G17 and axis addresses C and Y at the same time Plane of axis (Xp-Yp)

(3) When cylindrical interpolation, the feed speed F is specified for the circumference.

(4) Circular interpolation can be performed between a rotating shaft for cylindrical interpolation and another linear axis. The format is the same as that of circular interpol-ation performed with R instruction. Note: I, J, K directives are not available for radius designation.

【Example】：

Carry out circular interpolation between C axis and Z axis, and the circular interpolation instruction at this time is:

G18 Z_ C_

G02(03) Z_ C_ R_

(5) Cylindrical interpolation and tool radius compensation or tip radius compensation cannot be synchronized. The tool radius compensation or tip radius compensation in progress must be cancelled before cylindrical interpolation. After that, the radius compensation is restarted or terminated in this mode.

(6) Positioning and circulating instructions cannot be carried out during cylindrical interpolation. Before executing the above instruction, please cancel the cylinder interpolation instruction first, and this instruction (G07.1) is invalid in the positioning instruction (G00).

(7) During cylindrical interpolation, the drilling fixed cycle instruction cannot be specified. (G73, G74, G81 ~ 87).

(8) During cylindrical interpolation, the setting of workpiece coordinate system and local coordinate system cannot be specified. Workpiece coordinate system (G92,G54 ~ 59) Local coordinate system (G52).

(9) Tool position offset shall be specified before cylindrical interpolation. The tool position offset cannot be changed during cylindrical interpolation.

Program legend

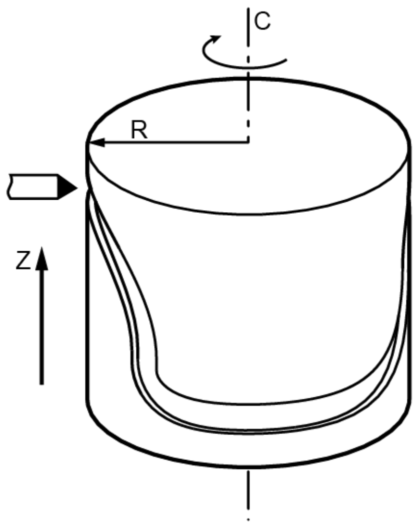


Figure2- 8 Example of Cylindrical Interpolation Program-1

```
N01 G00 G90 Z100.0 C0;
N02 G01 G91 G18 Z0. C0.;
N03 G07.1 C57.299;
N04 G90 G01 G42 Z130.0 D01 F250.;
N05 C75;
N06 G02 Z74.0 C131.0 R56.0;
N07 G01 Z70.0;
N08 G03 Z42.0 C159.0 R28.0;
N09 G01 C220.0;
N10 G03 Z114.0 C284.0 R64.0;
N11 G02 Z130.0 C310.0 R26.0;
N12 G01 C360.0;
N13 G40 Z100.0;
N14 G07.1 C0.0;
N15 M30;
```

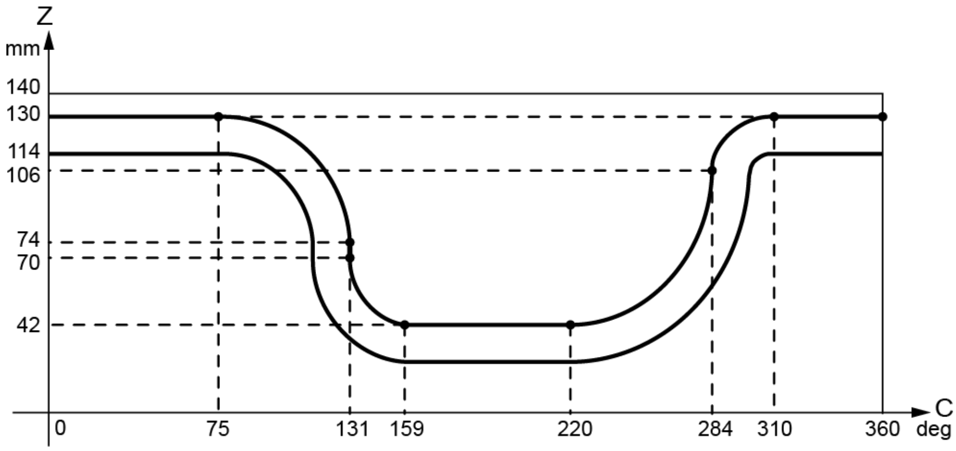


Figure 2- 9 Example of Cylindrical Interpolation Program-2

2.9Polar Coordinate Command (G15, G16)

Overview

【Functions】:

End point coordinate values are supported in polar coordinates of radius and angle. From the positive direction of the first axis of the specified polar coordinate command plane, the angle in the counterclockwise direction is positive and the angle in the clockwise direction is negative.

Radius and angle can be specified under absolute command/incremental command

【Instruction Format】：

G□□ G○○ G16 ;	Polar command (polar mode) start
G00 IP_ ;	Polar coordinate command
G15 ;	Polar command (polar mode) cancellation

G16	：	Polar command start
G15	：	Polar command cancellation
G□□	：	Plane selection of polar coordinates instructions (G17, G18, or G19)
G○○	：	Center selection of polar coordinates command (G90 or G91)
IP_	：	Axis address and instruction value of the plane constituting polar coordinates instruction

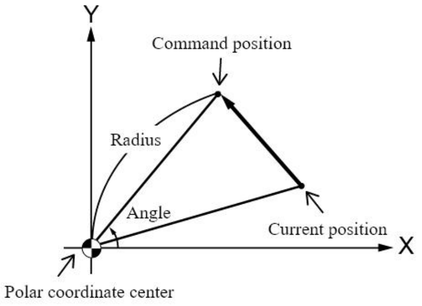
---

Instruction description

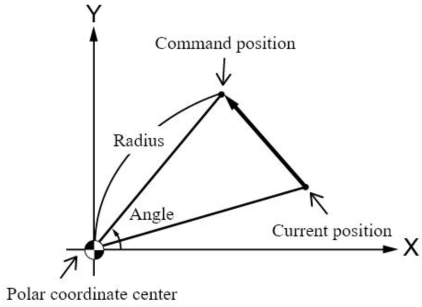
(1) When the polar coordinate command center is selected, the G90 polar coordinate center is the origin of the workpiece coordinate system, and the G91 polar coordinate center is the current position.

(2) When the plane axis address is selected, the first axis of the plane specifies the radius of polar coordinates, and the second axis of the plane specifies polar coordinatesThe angle of.

(3) When the origin of the workpiece coordinate system is set to the center of polar coordinates, the radius value is specified in absolute value. The origin of the workpiece coordinate system becomes the center of polar coordinates. However, when the local coordinate system (G52) is used, the origin of the local coordinate system becomes the center of the polar coordinates.

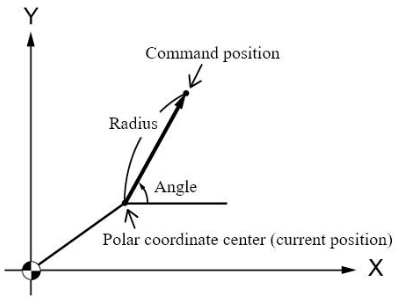


When the angle is absolute command

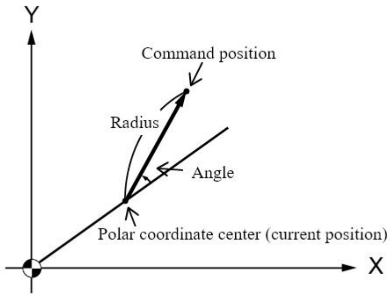


When the angle is an incremental command

(4) When the current position is set to the center of polar coordinates, the radius value is specified in increments. The current position is set to the center of polar coordinates.



When the angle is absolute command



When the angle is an incremental command

(5) Radius programming in polar coordinate mode: In polar coordinate mode, The radius of circular interpolation and spiral interpolation (G02, G03) are specified by R instruction

(6) Chamfer/corner R at any angle: In polar coordinate mode, chamfer/corner R at any angle cannot be specified.

(7) Axis instructions that will not be regarded as polar instructions in polar mode. Axial instructions containing the following instructions are regarded as non-polarinstructions:

- Pause (G04)
- Programmable data input (G10)
- Local coordinate system setting (G52)
- Workpiece coordinate system change (G92)
- Selection of mechanical coordinate system (G53)
- Storage stroke detection (G22)
- Coordinate rotation (G68)
- Scale scaling (G51)

Program legend

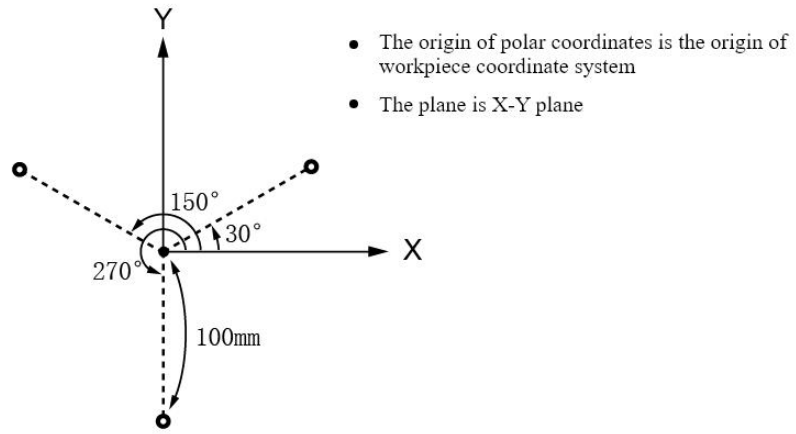


Fig. 2- 10 Bolt hole circulation

【Example1】：

When the radius value and angle are absolute commands:

N1 G17 G90 G16	Polar coordinate instruction, X-Y plane selection The origin of polar coordinates is the origin of workpiece coordinate system
N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0	Radius 100mm, angle 30deg
N3 Y150.0	Radius 100mm, angle 150deg
N4 Y270.0	Radius 100mm, angle 270deg
N5 G15 G80	Polar coordinate command cancellation

【Example2】：

When the radius value is absolute Instruction and the angle is incremental instruction:

N1 G17 G90 G16	Polar coordinate instruction, X-Y plane Selection The origin of polar coordinates is the origin of workpiece coordinate system
N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0	Radius 100mm, angle 30deg
N3 G91 Y120.0	Radius 100mm, angle 120deg
N4 Y120.0	Radius 100mm, angle 120deg
N5 G15 G80	Polar coordinate command cancellation

2.10Pause instruction (G04)

Overview

【Functions】：

An instruction to execute the next action at a specified time delay.

【Instruction Format】：

G04 X_ ;	Decimal points can be used (unit: seconds)
G04 P_ ;	You cannot use decimal points (unit: milliseconds)

Instruction description

(1) Under the "System-Parameters-Common" menu, you can set "Decimal Point Automatic Judgment" (# 32955) to ON or OFF. When using G04 pause instruction, even if the instructions are the same, if the decimal point automatic judgment state is different, the stop time may be different.

(2) Please refer to the Parameter Manual of Lynuc numerical Control System for the specific description of decimal point function.

Program legend

【Example1】：

G04 X1.5 Pause for 1.5 seconds

G04 P5000 Pause for 1.5 seconds

【Example2】：

Table2- 3 Examples of automatic decimal point judgment of G04 pause instruction

Instruction	Automatic decimal point judgment ON	Automatic decimal point judgment OFF	Remarks
G04 X5.	Pause for 5 seconds	Pause for 5 seconds	It is suggested that X and U should be programmed with decimal points, otherwise, when the decimal point is automatically judged as ON, it will be used as this value Calculated by one thousandth of.
G04 X5	Pause 0.005 seconds	Pause for 5 seconds	

G04 P5.	Pause 0.005 seconds	Pause 0.005 seconds	
G04 P5	Pause 0.005 seconds	Pause 0.005 seconds	

Attention

(1) If X (U) and P appear at the same time in the same block, an alarm is displayed.

(2) If the delay time is negative, an alarm is displayed

(3) This function can be used in the cutting instruction (G64).

(4) You can also specify G04 when you want to make an accurate inspection in G64 (cutting mode)

3.Coordinate values and instructions

3.1Absolute value instruction and incremental value instruction (G90, G91)

Overview

【Functions】:

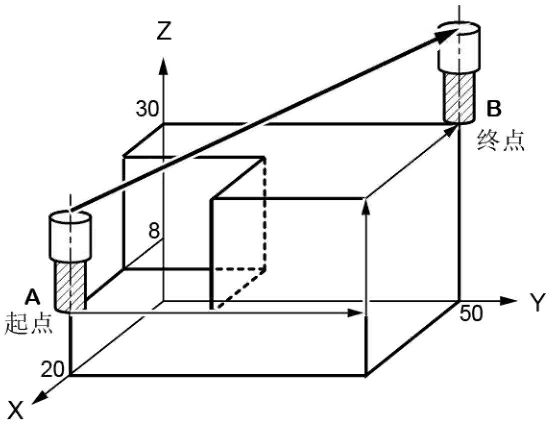
As the method of instruction axis movement, there are two methods: absolute value instruction and incremental value instruction. Absolute value instruction is a method of programming with the coordinate value of the end position of axis movement; Incremental value instruction is a method of direct programming with shaft movement.

【Instruction Format】:

G90 ;	Absolute instruction
G91 ;	Incremental instruction

Instruction description

G90 and G91 are modal G instructions. When the power is turned on, the G90 command (absolute command) is used by default.



Absolute instruction:

G90 X0 Y50 Z30 (coordinates of point B)

Incremental instruction:

G91 X-20 Y50 Z22 (Movement of Axes Direction and Distance)

Figure 3- 1Absolute and Incremental Instructions

3.2Absolute and incremental instruction of center coord (G90.1, G91.1)

3.2.1Absolute command of center coordinates (G90.1)

Overview

【Functions】:

Specifies the coordinate value of the center of the arc.

【Instruction Format】:

G90.1 ;

Instruction description

(1) G90.1 and G91.1 are modal G instructions. When the power is turned on, 91.1

(2) instruction (incremental instruction) is used by default.

(3) G90.1 and G91.1 are used only when machining arcs, and are used to specify the center of arcs.

Program legend

【Example】:

An arc

G90 G00 X20. Y10.

G90.1 G02 X10. Y20. I10. J10. F350

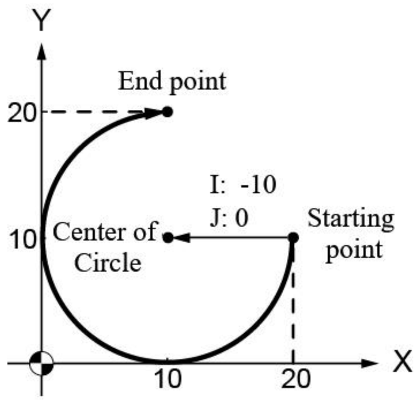


Figure 3- 2Arc

3.2.2Increment instruction of center coordinate (G91.1)

Overview

【Functions】：

Specifies the distance from the starting point to the center of the arc.

Specifies the direction from the current position with the symbol (+ or-) before the numeric value. The symbol "+" can be omitted.

【Instruction Format】：

G91.1 ;

Instruction description

(1) G91.1 and G90.1 are modal G instructions. When the power is turned on, G91.1 instruction (incremental instruction) is used by default.

(2) G90.1 and G91.1 are used only when machining arcs, and are used to specify the center of arcs.

Program legend

【Example1】：

A circle

G90 G00 X20. Y10.

G91.1 G02 X20. Y10. I-10. J0 F3

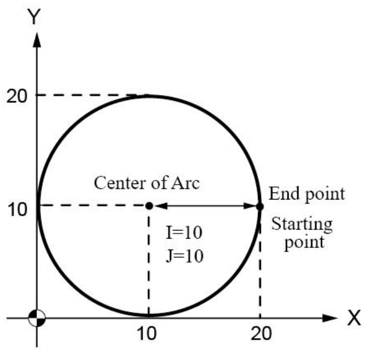


Figure 3- 3 Circumference

3.3Imperial units and the Metric system inputs (G20, G21)

Overview

【Functions】:

Enter whether the unit is imperial or metric, and select it with G code G20 and G21.

【Instruction Format】:

G20 ;	Imperial units (minimum set unit: 0.0001 inch)
G21 ;	The Metric system (minimum set unit: 0.001 mm)

Attention

(1) Metric-English switching G code should be at the beginning of the program, before the coordinate system is set, with a separate program segment instruction.

(2) At present, the system does not support inch specification. If G20 is specified, the system will give an alarm.

4.Feed function

4.1Summary

Overview

【Functions】:

The rapid movement of a tool at a specified speed or the movement of cutting a workpiece can be divided into two types: rapid movement and cutting feed.

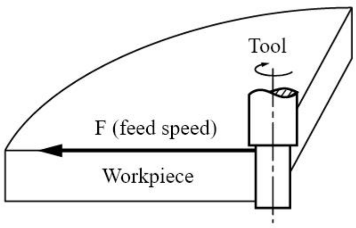


Figure 4- 1Feed function

4.2Fast feed (G00)

Overview

【Functions】:

The tool moves at a fast-forward speed to a position in the specified workpiece coordinate system.

【Instruction Format】:

G00 X_ Y_ Z_ ;

Instruction description

(1) By specifying the G00 (position) command, you can quickly move to the specified position.

(2) The fast moving speed is set by parameters. The magnification of speed can be set to 0% ~ 100% through the operation panel.

4.3Cutting Feed (G94, G95)

Overview

【Functions】:

The tool cuts the workpiece at the specified speed. It is divided into two types: feed per minute and feed per turn.

【Instruction Format】:

G94 F_ ;	Feed per minute (mm/min or inch/min)
G95 F_ ;	Feed per revolution (mm/rev or inch/rev)

Instruction description

(1) The feeding mode when the power supply is turned on is controlled by parameters.

(2) Using the operation panel switch, the rate of 0% ~ 200% can be applied to feed per minute and feed per revolution.

(3) The cutting feed speed is expressed by F instruction and its subsequent value

- Once an F instruction is executed, it remains valid until the next F instruction is executed.
- The F instruction is executed in principle in the same block as the cutting feed instruction (G01) or in the block preceding the execution of the G01 instruction.

(4) Calculation method of feed speed F:

F (mm/min) = S ×F;

S: Spindle rotation speed (rev/min)

f: Tool feed (mm/rev) when the spindle is rotated once

(5) In per-revolution feed mode, once the F per-revolution feed is specified, the actual feed rate varies with the spindle command speed S.

(6) The default is feed per minute. When using F to specify pitch directly in rigid tapping, it is necessary to use G95 instruction per turn.

Program legend

【Example1】：

Feed per minute: The amount of tool feed per minute

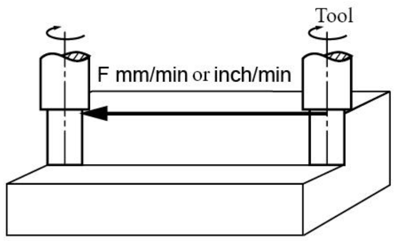


Figure 4- 2 Feed per minute

【Example2】：

Feed per revolution: The feed of the tool per revolution of the spindle.

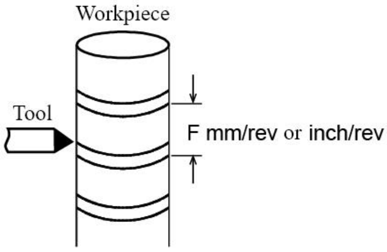


Figure 4- 3 Feed per turn

【Example3】：

G90 G54 G00 X0 Y0 ;	
G43 H1 Z5. ;	
M08 ;	
S3500 M03 ;	The spindle rotates forward at a speed of 3500 rpm
G01 X10. Y10. F200 ;	The cutter moves to position X10. Y10. at a speed of 200 mm/min
M30 ;	

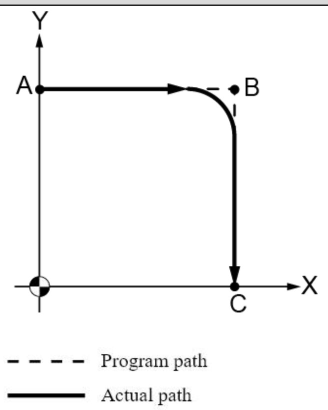
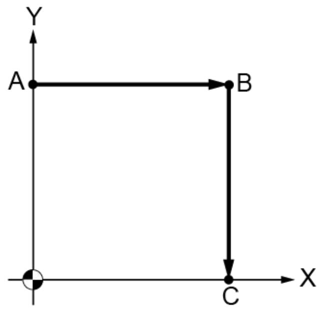
4.4Accurate positioning function (G09, G61)

Overview

【Functions】：

When the shaft is moved at high speed, in the usual cutting mode (G64), the actual machined workpiece rotation angle is not as sharp as that on the drawing,but becomes rounded. Accurate positioning functions (G09, G61) can be used if precise machining of corner parts is required. When G09 and G61 instructions are issued, the feed speed will gradually decrease until it decreases to 0 at the end. After confirming that the specified point has been reached, start to execute the next program instruction.

Table 4- 1 Difference of actual path rotation angle between cutting mode and accurate positioning mode

Cutting mode (G64)	**Accurate positioning mode (G01, G61)**
	
The actual path angle becomes rounded	The actual path angle is sharp

【Instruction Format】：

G09 ;	Accurate positioning
G61 ;	Accurate positioning mode
G64 ;	Cutting mode (G61 canceled)

Instruction description

(1) G09 is valid only in the specified block (once).

(2) G61 is valid until G64 instruction is executed (modal instruction).

(3) When the power is turned on, it defaults to G64 (cutting mode).

Program legend

【Example1】：

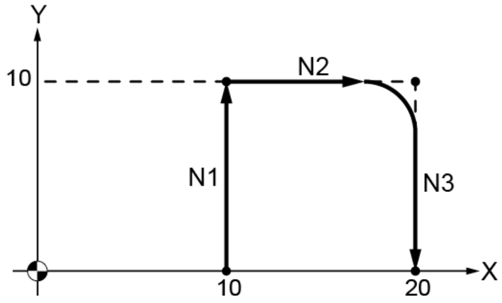


Figure4- 4Accurate Positioning Command G09

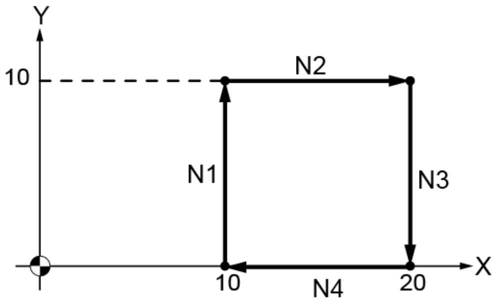
N1 G91 G09 G01 Y10.

N2 X10.

N3 Y-10.

N4 M30

【Example2】：



G61Figure 4- 5 Accurate Positioning Modal Command G61

N1 G91 G61 G01 Y10.

N2 X10.

N3 Y-10.

N4 X-10.

G64

5.Reference point

5.1Summary

Overview

【Functions】：

Reference point refers to the fixed reference position point established in the machine tool coordinate system based on mechanical zero point on CNC machinetool. Reference points can be used as automatic tool change (ATC), automatic tray exchange (APC), etc. Usually, the origin of the machine tool and the reference point of the machine tool coincide on the CNC milling machine. The second, third and fourth reference points need to refer to the position of the origin of the machine tool, and set coordinate values in "System-Parameters".

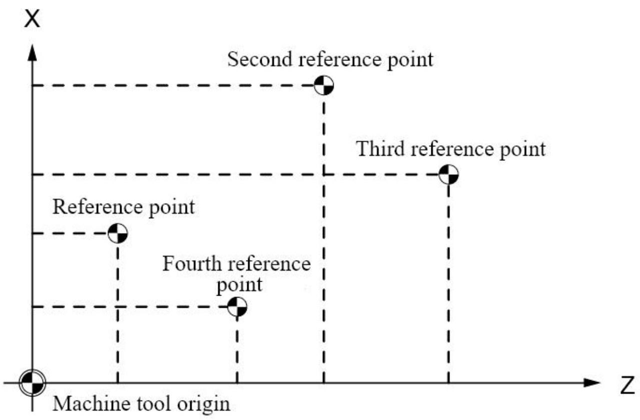


Figure 5- 1 Reference point

5.2Reference point reversion detection (G27)

Overview

【Functions】：

The G27 instruction checks whether the specified shaft is correctly returned to the reference point. If the tool does not reach the reference point, a warning message will be displayed and the equipment will be put into a suspended state.

【Instruction Format】：

G27 X_ Y_ Z_ ;		check instruction that returns to the reference point
X_ Y_ Z_	：	Specifies the coordinates of the reference point (absolute/relative value spesification)

Program legend

【Example1】： In G90 (Absolute Command) mode, the coordinate value of the reference point is specified by the coordinate value of the workpiece coordinate system.

G40（G49）	Cancel tool radius compensation (cancel tool length compensation)
G90	Absolute instruction
G27 Z-100.	Departing from Z axis in advance to prevent interference
G27 X140. Y140.	Reference point R coordinates (X140, Y140, Z-100)

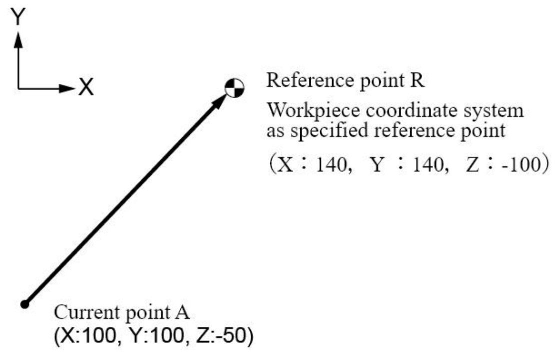


Fig.5- 2 Designation of G90 Modal Reference Point Coordinate Values

【Example2】： In G91 (incremental instruction) mode, the reference point coordinate value specifies the distance to move from the current position.

G40（G49）	Cancel tool radius compensation (cancel tool length compensation)
G91	Incremental instruction
G27 Z40.	Departing from Z axis in advance to prevent interference
G27 X-40. Y-40.	Reference point R coordinates (X-40, Y-40, Z40)

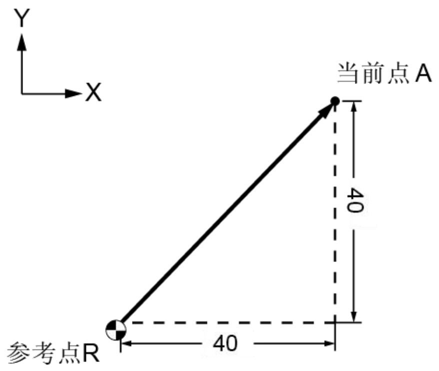


Fig. 5- 3Designation of coordinate values of G91 modal reference points

5.3Automatic return to reference point (G28)

Overview

【Functions】:

Reset the specified axis to the machine tool reference point (the first reference point). If this instruction is carried out, fast-forward positioning to the specified intermediate point, and then automatically return to the machine tool reference point (the first reference point). There is no need to calculate the movement from the intermediate point to the reference point of the machine tool. The specified intermediate point coordinates are automatically saved for the intermediate point of the G29 (Automatic Reset to Reference Point) instruction that is subsequently executed.

【Instruction Format】:

G28 X_ Y_ Z_ ;		Automatic return reference point instruction
X_ Y_ Z_	:	Specifies the intermediate point (absolute/relative value specified) that passes during the return of the referencepoint

Instruction description

(1) The intermediate point coordinate value specifies the value of the workpiece coordinate system in G90 (absolute instruction) mode and the distance from the current position in G91 (incremental instruction) mode.

(2) If the G28 instruction is executed during tool radius compensation, it will be positioned at the position where radius compensation has been cancelled.

(3) Only the coordinate values specified in G28 block are stored in the intermediate point coordinate values.

(4) When G28 is specified, the tool is quickly positioned to the specified IP, and then returned from the IP point to the reference point.

Motion process

- The machine tool is positioned from the current position to the intermediate point position (point A → point B) at a fast speed of returning to zero
- The machine tool is positioned from the intermediate point to the reference point (point B → point R) at a fast speed of returning to zero

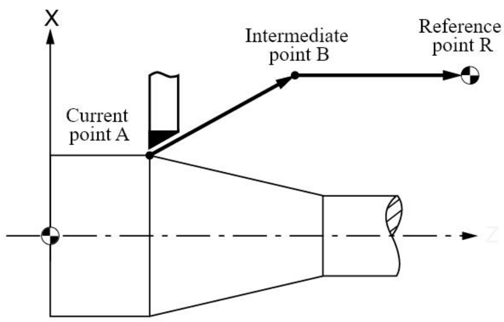


Figure5- 4 Action to return to a reference point

Program legend

【Example1】： In G90 (Absolute Command) mode, the coordinate value of the intermediate point is specified by the coordinate value of the workpiece coordinate system.

G40（G49）	Cancel tool diameter compensation (Cancel tool length compensation)
G91 G28 Z0.	The Z axis are raised in advance to prevent the coordi nate interference of the intermediate points.
G90 G28 X-40. Y-40.	Move from point A through intermediate point B to reference point R

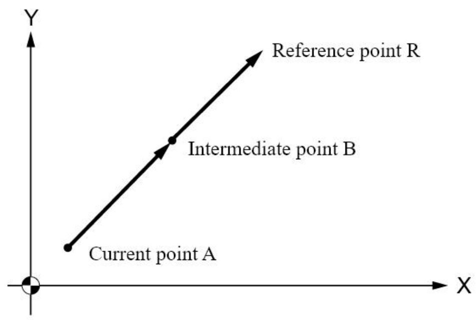


Fig. 5- 5 Coordinate values of intermediate points in absolute command mode

【Example2】: In G91 (incremental instruction) mode, the intermediate point coordinate value pecifies the distance to move from the current position.

G40（G49）	Cancel tool diameter compensation (Cancel tool length compensation)
G91	Incremental instruction
G28 Z0.	Raise the Z axis in advanceto prevent interference of intermediate point coordinates
G28 X-40. Y-40.	Move from point A through intermediate point B to reference point R

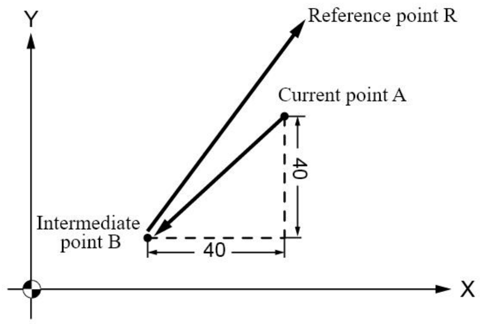
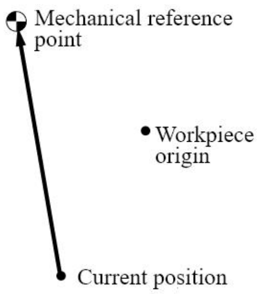
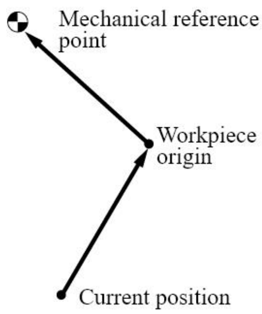


Fig. 5- 6 Coordinate values of intermediate points in incremental instruction mode

【Example3】: The difference between the commonly used instructions G91G28Z0 and G90G28Z0 during knife withdrawal.

Table 5- 1Tool withdrawal difference between G91G28Z0 and G90G28Z0

G91G28Z0 (incremental instruction mode)	G90G28Z0 (Absolute instruction Mode)
	

5.4Automatic revert from reference point (G29)

Overview

【Functions】:

Position the specified axis from the reference point (1st, 2nd, 3rd, 4th reference point) to the specified position via the intermediate point. If the automatic reference point return (G28, G30) command is executed after the command is executed, the intermediate point through which the automatic reference point return (G28, G30) command is executed is fast-forwarded to a designated position. There is no need to calculate the amount of movement from the intermediate point to the reference point.

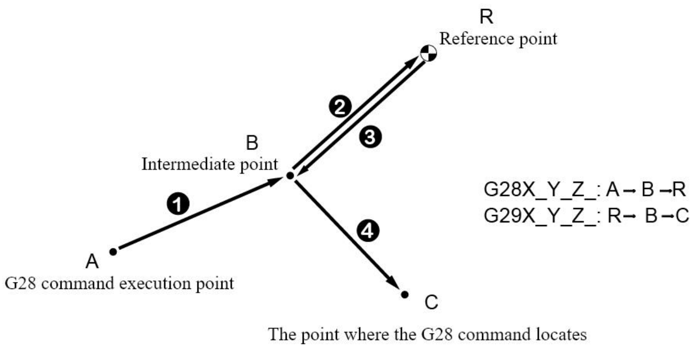


Figure 5- 7 Automatic reversion from reference point

【Instruction Format】:

G29 X_ Y_ Z_ ;	Instruction to automatically return from reference point

X_ Y_ Z_	：	Position point coordinate value (workpiece coordinate value)

Instruction description

(1) Be sure to execute the G29 instruction in the next block of the G28 instruction. Please ensure that the position of the specified shaft is the same as that of the G28 command to return to the reference point, otherwise an alarm will appear.

(2) If G29 instruction is executed in tool radius compensation, it will be positioned at the position where radius compensation has been cancelled.

Program legend

【Example1】： In G90 (Absolute Command) mode, the coordinate value of the positioning point is specified by the coordinate value of the workpiece coordinate system.

G40（G49）	Cancel tool diameter compensation (Cancel tool length compensation)
G91 G28 Z0.	The Z axis G90 is raised in advance to prevent interference of intermediate point coordinates
G90	Absolute instruction
G28 X40. Y40.	Move from point A through intermediate point B to reference point R
G29 X60. Y0.	Move from reference point R to specified point C through intermediate point B

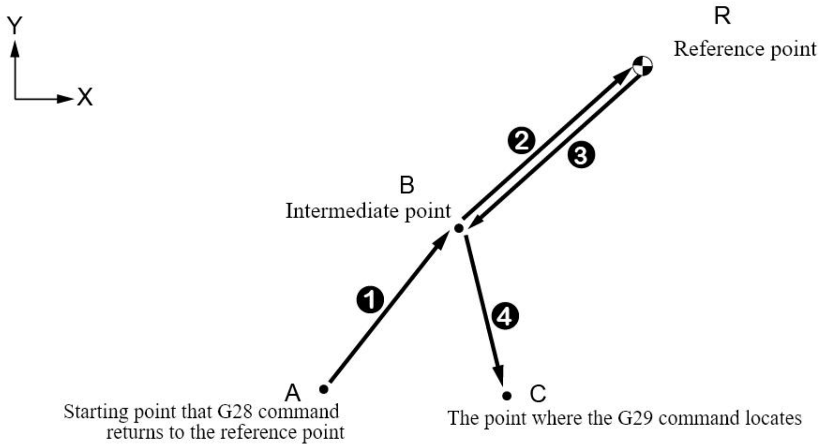


Fig. 5- 8 Automatic reversal diagram of absolute instruction from reference point

【Example2】： In G91 (incremental instruction) mode, the coordinate value of the positioning point is specified by the coordinate value of the workpiece coordinate system.

G40（G49）	Cancel tool diameter compensation (Cancel tool length compensation)
G91 G28 Z0.	The Z axis G90 is raised in advance to prevent interfeence of intermediate point coordinates
G90	Absolute instruction
G28 X40. Y40.	Move from point A through intermediate point B to reference point R
G29 X60. Y0.	Move from reference point R to specified point C through intermediate point B

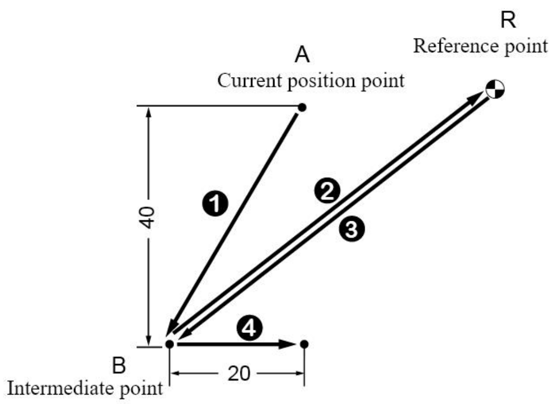


Fig. 5- 9 Automatic reversal of incremental instructions from reference points

5.5Return to Reference Points 2, 3, 4 (G30)

Overview

【Functions】：

With multiple reference points set (up to 4), return the specified axis to the 2, 3, 4 reference points. Usually, it is used at the same time when the automatic tool change position and the reference point position are different. Or when the mechanical reference point is different from the reference point that needs to be returned quickly.

【Instruction Format】：

G30 P2 X_ Y_ Z_ ;	Return to the second test center (P2 can be omitted)
G30 P3 X_ Y_ Z_ ;	Return to Reference Point 3
G30 P4 X_ Y_ Z_ ;	Return to Reference Point 4

6.Coordinate system

6.1Summary

Overview

【Functions】：

When the machine tool is working, the tool moves to the corresponding position according to the coordinates specified by the program, and the basis of the program coordinates is the machine tool coordinate system. Coordinate values can be specified using one of machine tool coordinate system, workpiece coordinate system and local coordinate system. The relationship of each coordinate system is as follows:

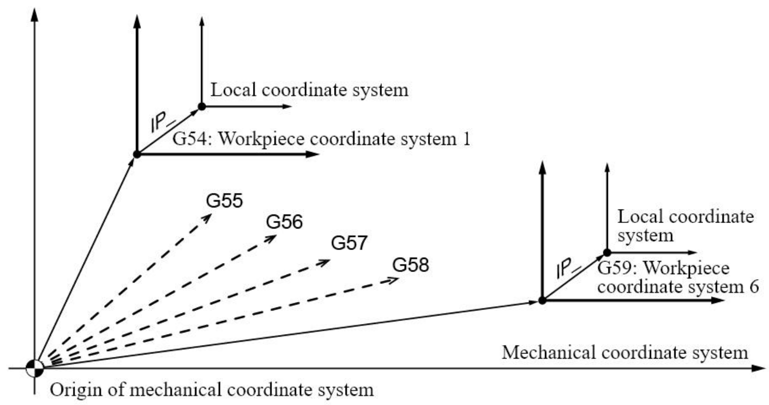


Fig. 6- 1 Relationship between coordinate systems of machine tools

Description

(1) **Machine tool coordinate system:** The fixed point on the machine tool is taken as the origin. Once the machine tool returns to zero, CNC establishes the machine tool coordinate system according to the origin of the machine tool.

(2) **Workpiece coordinate system:** The coordinate system set at the end face of the workpiece to be processed or the center of the chuck after considering the processing technology. Its setting principle is mainly to consider whether the processing sequence and programming are convenient.

Workpiece coordinate system is a sub-coordinate system in machine tool coordinate system. They can be moved in the machine tool coordinate system by reset or command offset. During use, once the workpiece coordinate system is determined, it is not allowed to change the workpiece coordinate system at will unless the workpiece variety or process requirements are changed.

(3) **Local coordinate system:** Considering the processing technology requirements, in order to facilitate the setting of reference points in the workpiece coordinate system. Generally, the local coordinate system can be set at the independent machining unit of the workpiece. The local coordinate system may be an offset adjustment to the workpiece coordinate system.

6.2Mechanical coordinate system (G53)

Overview

【Functions】：

he coordinate system with the zero point of the machine tool as the origin of the coordinate is called the machine tool coordinate system. Machine tool manufacturers install fixed machine tool zeros for machine tools. After the machine tool is powered on and returns to the reference point, the machine tool coordinatesystem is established and remains unchanged before power failure.

【Instruction Format】：

G53 X_ Y_ Z_ ;		Machine tool coordinate system instruction
X_ Y_ Z_	：	Absolute coordinate value specifies

Instruction description

(1) Before executing G53 instruction, the machine tool coordinate system must be established. After switching on the power supply, please establish the machine tool coordinate system through the origin reversion operation.

(2) G53 is valid only in the specified block (once).

(3) The coordinate value (X_Y_Z_) is an absolute coordinate value, which is specified as an absolute coordinate value even in the incremental instruction mode.

(4) Machine tool coordinate system can also be selected by G54. 1P54 or G959.

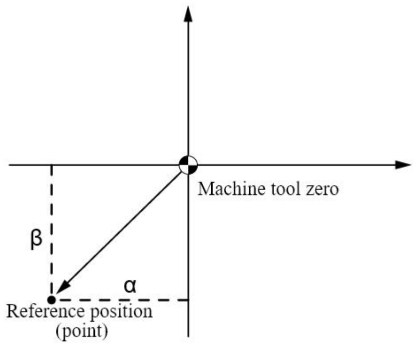


Fig.6- 2 Determination of machine tool coordinate system

When the CNC system is powered on, a machine tool coordinate system can be established immediately by manually returning to the reference point. The coordinate values of the reference point are (alpha, beta, …) is set by automatically setting coordinate system parameters (XXX) after returning to zero. When α and β are both 0, the machine tool coordinate system coincides with the zero point of the machine tool.

Program legend

【Example1】：

```
G54 G90 X0
Y0
G00 X10. Y10.          ①
G53 X30. Y30.          ②
X0. Y0.                ③
(G54: X, Y, Z = 100., 50., -100.)
```

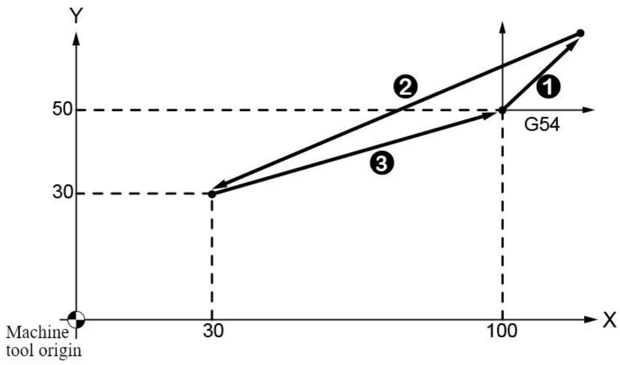


Fig. 6- 3 Program Legend of Machine Tool Coordinate System and Workpiece Coordinate

System-1

【Example2】：

Absolute mode		Incremental mode
G54 G90 G00 X0 Y0		G54 G90 G00 X0 Y0
X-10. Y10.	①	G91 X-10. Y10.
G01 Y50. F500	②	G01 Y40. F500
X-50.	③	X-40.
Y10.	④	Y-40
X-10	⑤	X40
G53 X30. Y30.	⑥	G53 X30. Y30.
X10. Y10.	⑦	X80. Y30.
G01 Y50.	⑧	G01 Y40.
X50.	⑨	X40.
Y10.	⑩	Y-40.
X10.	⑪	X-40
G00 X0 Y0	⑫	G00 X-10. Y-10.
M30		M30
(G54: X, Y, Z = 100., 50., -100.)		(G54: X, Y, Z = 100., 50., -100.)

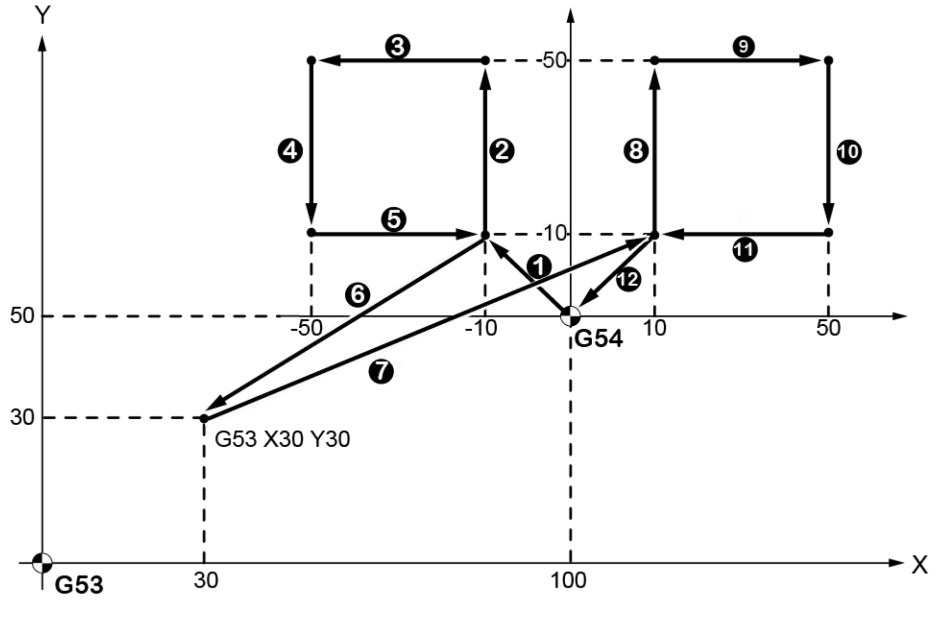


Fig. 6- 4 Program Legend of Machine Tool Coordinate System and Workpiece Coordinate System

Attention:

Whether it is absolute mode or incremental mode, the mechanical coordinate system selection instruction is the same

6.3Workpiece coordinate system

6.3.1Change the workpiece coordinate system (G10)

Overview

【Functions】:

Change the origin offset of the workpiece coordinate system.

【Instruction Format】:

G10 L2 P_ X_Y_ ;	Setting G54 ~ G59 coordinate system does not change the offset value of coordinate system in the system
G10 P_ X_ Y_ ;	The same as above.
G10 L10001 P_ ;	Set the origin value of G54 ~ G59 coordinate system, and change the offset value of coordinate system accordingly
G10 L10002 P_ ;	Set the offset value of G54 ~ G59 coordinate system, and change the system settings accordingly
G10 L10002 P0 ;	Set the offset value relative to the current coordinate system without changing the offset setting of the system

L_	:	Corresponding to different functions
P_	:	Specifies the coordinate system(G54/…/G59)
X _ Y_ Z_	:	Corresponding to different axes (X YZABCUVW)

Instruction description

(1) The workpiece coordinate system set by G10L2 will automatically return to the set value in the "Offset" module after M30, reset and restart.

(2) If you want the workpiece coordinate system set by G10 to remain, you need to use the G10L10002P_X_Y_Z_ format.

(3) P1 ~ P6 represent coordinate systems G54 ~ G59, and P7 ~ P53 represent subsequent coordinate systems.

(4) XYZABCUVW parameters set in all instructions will be affected by G90 absolute mode and G91 incremental mode.

Program legend

【Example1】：

G90

G10 L2 P1 X1 Y1 Z1

X1 Y1 Z1 represents the new G54 coordinate system temporary workpiece compensation value, but after setting the corresponding G54 in the system coordinate system setting workpiece compensation has not changed.

【Example2】：

G90 G54

G10 L10001 P1 X1 Y1 Z1

X1Y1Z1 represents the new workpiece coordinate value of the corresponding G54 coordinate system, and the set value of the workpiece compensation system in G54 coordinate system will change according to its set origin coordinate value.

In G91 mode, the set coordinate point is the current coordinate accumulation input coordinate value.

【Example3】：

G90

G10 L10002 P1 X1 Y1 Z1

X1 Y1 Z1 represents the offset value of G54 coordinate system relative to mechanical coordinates, and the offset value set by the system will change. After setting, the origin coordinate value of absolute coordinate of coordinate system will be calculated according to the offset value.

The difference with instruction G10 L2 P1 X1 Y1 Z1 is that it changes the system offset setting.

【Example4】:

G90 G56

G10 L10002 P0 X1 Y1 Z1

X1 Y1 Z1 represents the offset compensation value for all coordinate systems or the offset compensation value for external coordinates. This instruction also changes all coordinate system offset compensation or external coordinate offset compensation settings in the system.

In G91 mode, a new offset compensation value is set after the input value is accumulated on the basis of the original offset compensation value.

6.3.2Change the supplementary workpiece coordinate system (G10)

Overview

【Functions】:

Change the origin offset of the supplementary workpiece coordinate system.

【Instruction Format】:

G10 L20 P_ X_ Y_ ;	Set G154 ~ G958 coordinate system without changing the offset value set by the system
G10 L10010 P_ ;	Set the origin value of G154 ~ G958 coordinates, and the offset value of coordinate system will be changed accordingly
G10 L10020 P_ ;	Set the offset value of G154 ~ G958 coordinate system, and change the system settings accordingly

L_	:	Corresponding to different functions
P_	:	Specifies the coordinate system (G154/…/G159)X_
X _ Y_ Z_	:	对 Corresponding to different axes (X YZABCUVW)

Instruction description

(1) The supplementary workpiece coordinate system set by G10 L20 will automatically return to the set value in the "Correction" module after M30, reset and restart.

(2) The G10L10020P_X_Y_Z_ format is required if you want the supplementary workpiece coordinate system set by G10 to remain.

(3) P1 ~ P6 represent coordinate systems G54 ~ G59, and P7 ~ P53 represent subsequent

coordinate systems.

(4) XYZABCUVW parameters set in all instructions will be affected by G90 absolute mode and G91 incremental mode.

6.3.3Select workpiece coordinate system (G54 ~ G59, G154 ~ G159... G954 ~ G959)

Overview

【Functions】:

To select the workpiece coordinate system to be used, it is necessary to log in the origin positions of 60 workpiece coordinate systems in the workpiece compensation screen in advance, and select the workpiece coordinate system through G54 ~ G59, G154 ~ G159... G954 ~ G959 instructions. After the workpiece coordinate system selection instruction, its absolute instruction value becomes the coordinate value of the selected workpiece coordinate system.

【Instruction Format】:

G54 ;	lect workpiece coordinate system 1
~	~
G59 ;	Select workpiece coordinate system 6
G154 ;	Select Supplementary Workpiece Coordinate System 1
~	~
G159 ;	Select Supplementary Workpiece Coordinate System 6
G254 ;	Select Supplementary Workpiece Coordinate System 7
~	~
G259 ;	Select Supplementary Workpiece Coordinate System 12
~	~
G954 ;	Select Supplementary Workpiece Coordinate System 49
~	~
G959 ;	Select Supplementary Workpiece Coordinate System 54

Instruction description

(1) When the power is turned on, G54 is used by default (select workpiece coordinate system 1).

(2) Please do not use G54 ~ G59, G154 ~ G159 ⋯ G954 ~ G959 and G92 in the same program. Otherwise, the coordinate system may be confused and the equipment may be

damaged.

(3) Setting the offset of G959 is not allowed.

(4) The coordinate origin of G959 is consistent with the mechanical origin.

Program legend

【Example1】：

G54 G90 X0 Y0

G01 X10. Y10.

(G54: X, Y, Z = 100., 50., -100.)

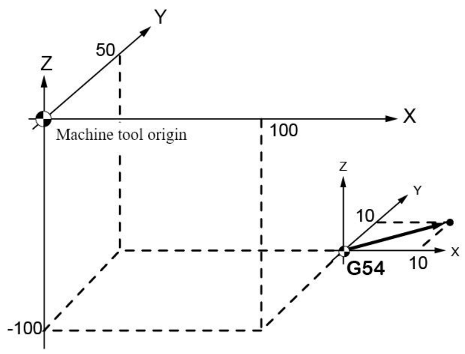


Figure 6- 5Selecting workpiece coordinate system

6.3.4 Select Supplementary Workpiece Coordinate System (G54.1)

Overview

【Functions】：

选择补充工件坐标系，可以选择 54 个补充工件坐标系（P1～P54）。P54 为机床坐标系。

【Instruction Format】：

G54.1 P_ ;		Select supplementary workpiece coordinate system
P1 / P2 / … / P6	：	Supplementary workpiece coordinate system 1 (G154) /2 (G155)/…/6(G159)
P49 / P50 / … / P54	：	Supplementary workpiece coordinate system 49 (G954)/50 (G955)/…/54(G959)

Program legend

【Example】：

G54.1 P1	Select the Supplementary Workpiece Coordinate System 1
G01 X10.	
Y10.	
G54.1 P54	Select Supplementary Workpiece Coordinate System 54 (Mechanical Coordinate System)
X Y Z	Move to the origin of mechanical coordinate system
G54	Select workpiece coordinate system 1

6.3.5Set the workpiece coordinate system (G92)

Overview

【Functions】：

Establish a workpiece coordinate system so that the current tool position becomes the specified coordinate value of the currently selected workpiece coordinate system. The workpiece coordinate system created by this instruction will remain valid until a new workpiece coordinate system is reassigned with G92. When power failure and restart, the workpiece coordinate system established by this instruction will be automatically eliminated. It can be specified by parameters, and the workpiece coordinate system can be automatically eliminated when reset. The workpiece coordinate system established by G92 and the currently selected workpiece coordinate system (G54, for example), there is an offset. When another workpiece coordinate system is selected (such as G55), this offset will be automatically carried into the new coordinate system, affecting the origin coordinate of the coordinate system.

【Instruction Format】:

G92 X_ Y_ Z_ ;

Instruction description

Set the coordinate system through G92. When the cutter length is compensated, the position before offset is determined by G92 as the absolute coordinate value (X_Y_Z_). Even in the incremental instruction mode, the coordinate value is specified as an absolute coordinate value.

Program legend

【Example1】：

The workpiece coordinate system is set with the instruction of G92 X52 Y35 Z48. As shown in the figure, the tip of the tool is the starting point of the program.

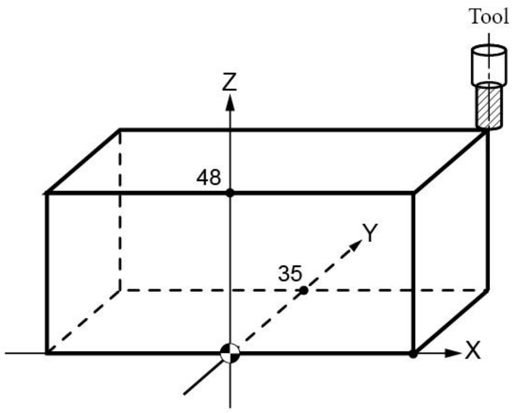


Figure 6- 6 G92 Setting Workpiece Coordinate System

Attention

G92 cannot be specified at the same time as the program segment where the tool length compensation vector changes, otherwise abnormal coordinate system setting will occur.

6.3.6Workpiece coordinate system preset (G92.1)

Overview

【Functions】:

Workpiece coordinate system preset function, which makes the newly established workpiece coordinate system and the originally selected workpiece coordinate system (such asG54).

【Instruction Format】:

G92.1 X_ Y_ Z_ ;	Preset the workpiece coordinate system of the specified axis
G92.1 X0 Y0 Z0 ;	Cancel the workpiece coordinate system of the specified axis

Attention

Unspecified axes do nothing.

6.4Local coordinate system (G52)

Overview

【Functions】:

When programming in the workpiece coordinate system, the sub-coordinate system of the workpiece coordinate system can be set for the convenience of programming. Sub-coordinate system is called local coordinate system, which can be used to set the offset of workpiece coordinate system.

【Instruction Format】:

G52 X_ Y_ Z_ ;	Set offset coordinate system
G52 X0 Y0 Z0 ;	Cancel offset coordinate system

Instruction description

(1) The setting of offset coordinate system is valid for all workpiece coordinate systems.

(2) When the G92 instruction is specified, the original G52 offset coordinate system will be automatically cleared.

(3) After M02, M30 and system reset, the offset coordinate system specified by G52 is cancelled.

(4) When the offset coordinate system is set, the set coordinate value is the value of the absolute value mode (G90) instruction.

Program legend

【Example】:

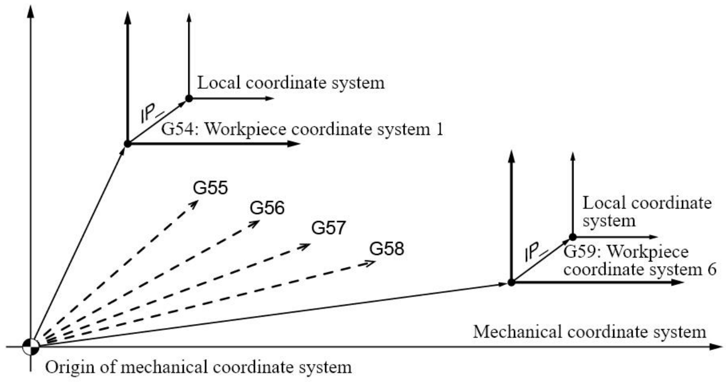


Fig. 6- 7 Relationship between local coordinate system and workpiece coordinate system

6.5Coordinate reading function (G32)

Overview

【Functions】:

Use coordinates to read into G32 instruction, and set the coordinates at that time (mechanical, workpiece or relative coordinates) in the specified common variables.

【Instruction Format】:

G32 P1 Xx Yy Zz ;	he mechanical coordinates of each axis (X/Y/Z) are stored in common variables respectively.
G32 P2 Xx Yy Zz ;	The workpiece coordinates of each axis (X/Y/Z) are stored in common variables respectively.
G32 P3 Xx Yy Zz ;	The relative coordinates of each axis (X/Y/Z) are stored in common variables respect

Xx	:	Save x axis coordinates to public variable # x
Yy	:	Save y axis coordinates to public variable # y
Zz	:	Save z axis coordinates to public variable # z

Instruction description

Both # 100 and # 1699 can be used as public variables. An error will occur when others are used as public variables.

Program legend

【Example】:

```
G32 P1 X100 Y101 Z102
…… (Other procedures)
G53 X#100 Y#101 Z#102
M30
```

7.Zoom, Mirroring, and Rotation

7.1Zoom function (G50, G51)

Overview

【Functions】:

G51 Reduces or enlarges edited shapes around any point. You can specify the same magnification for all axes, or you can specify magnification for each axis separately. G50 eliminates the zoom function.

【Instruction Format】:

G51 X_ Y_ Z_ P_ ;		Zoom command (all axes have the same magnification)
G50 ;		Unzoom instruction
X_ Y_ Z_	:	Absolute coordinates of zoom center
P_	:	Zoom magnification

G51 X_ Y_ Z_ I_ J_ K_ ;		Zoom command (each axis specifies the mag i fication)
G50 ;		Unzoom instruction
X_ Y_ Z_	:	Absolute coordinates of zoom center
I_	:	Zoom ratio of X axis Zoom ratio of Y axis
J_	:	Zoom ratio of Z axis
K_	:	Zoom command (each axis specifies the magn ification)

Instruction description

(1) When the magnification parameter is not specified, it is not scaled by default. The scaling factor cannot be less than 0.0005.

(2) In the zoom function, you cannot specify arcs with different zoom ratios.

(3) If X, Y, and Z are omitted, the current coordinates are used as the scaling center.

(4) Please do not specify zoom, mirror and rotate in Z direction for fixed loop program; Please do not work with boring function G76 and G87 instructions specify the scaling, mirroring and rotation functions of XY plane.

(5) Scale is invalid for tool radius compensation value and tool length compensation value.

(6) Avoid specifying scaling parameters (P and I, J, K) at the same time, otherwise an alarm will be displayed.

(7) When executing coordinate rotation, zoom, and mirror commands on a shape at the same time, follow the order of mirror, zoom, and coordinate rotation commands.

(8) You cannot change the current workpiece coordinate system in coordinate rotation, scaling and mirroring.

(9) Even in incremental instruction programming, the zoom center is still an absolute coordinate value.

Program legend

【Example1】： The scaling center is within the shape (all axes are scaled at a uniform magnification).

G54 G90 X0 Y0	
G51 X20. Y20. P2	Zoom instruction (zoom center and nmagnification)
(1) G01 X10. Y10. F500	
(2) Y30.	
(3) X30.	Unscaled shape
(4) Y10.	
(5) X10.	
G50	Unzoom
M30	

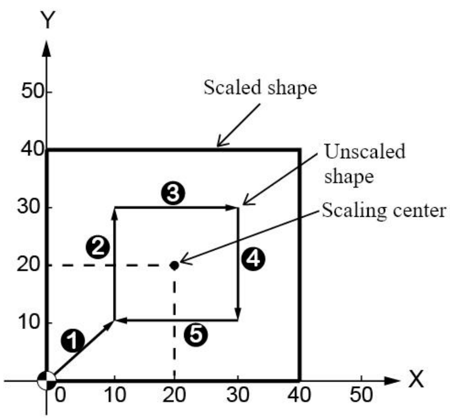


Figure 7- 1 Zoom Center in Shape

【Example2】： The scaling center is outside the shape (all axes are scaled at a unified magnification).

```
G54 G90 G00 X0 Y0
G51 X-10. Y25. P2.
G01 X10. Y10. F500
Y30.
X50.
Y10.
X10.
G50
M30
```

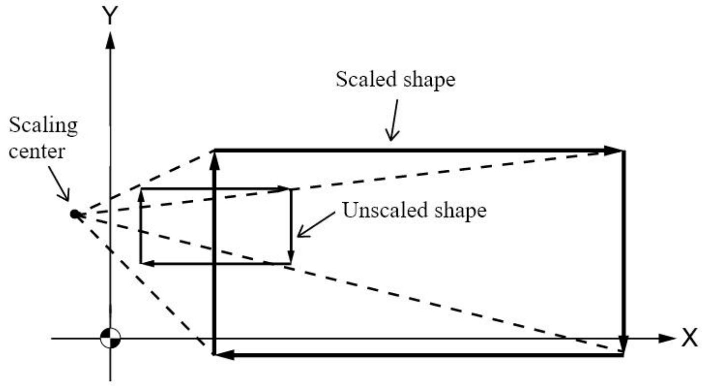


Figure 7- 2 Zoom Center Outside Shape

【Example3】： Each axis specifies the scaling of magnification.

```
G54 G90 G00 X0 Y0
G51 X25.Y25.I2. J3.
G01 X10. Y10. F500
X20. Y45.
X65. Y30.
X45. Y5.
X10. Y10.
G50
M30
```

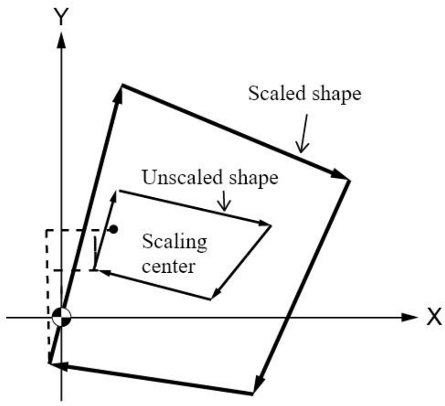


Figure 7- 3 Scaling at different magnifications

【Example4】： Even in G91 incremental mode, the specified zoom center is stillan absolute coordinate.

```
G54 G90 G00 X0 Y0
G51 G91 X30. Y20. P2.
G01 X10. Y10. F500
Y20.
X40.
Y-20.
X-40.
G50
```

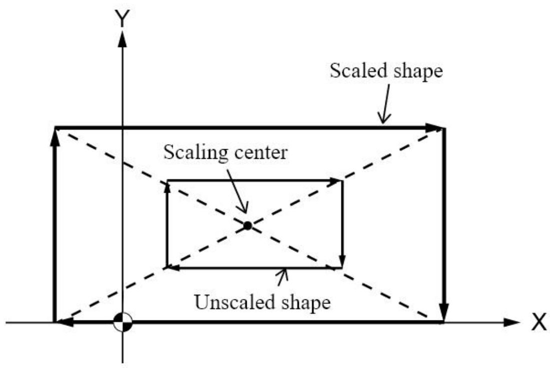


Figure 7- 4 Zoom centers are always absolute coordinates

7.2 Mirroring Function (G50.1, G51.1)

Overview

【Functions】：

G51. 1 mirrors edited shapes with arbitrary axes as symmetry lines.

G50.1 cancels the mirroring function.

【Instruction Format】：

G51.1 X_ Y_ Z_ ;		Mirror instruction
G50.1 ;		Cancel mirroring instruction
X_ Y_ Z_	:	Absolute coordinates of mirror reference poin

Instruction description

(1) When executing coordinate rotation, zoom, and mirror commands on a shape at the same time, follow the order of mirror, zoom, and coordinate rotation commands. Mirroring function cannot be specified in scaling or coordinate rotation mode.

(2) You cannot change the current workpiece coordinate system in coordinate rotation,scaling and mirroring.

(3) The circular interpolation instruction (G02, G03) and tool diameter compensation instruction (G41, G42) are reversed (G02 becomes G03, G41 becomes G42) when only one axis is mirrored on the same plane. There is no change when two axes are mirrored at the same time.

Program legend

【Example1】：

```
G54 G90 X0 Y0
G51.1 X-10.              Mirror instruction (Mirror base: X-10.)
(1) G01 X10. Y10. F500
(2) Y40.X40
(3) X10. Y40.            Shapes that are not mirrored
(4) Y10.
(5) G00 X0 Y0
```

G50.1 Unmirroring

M30

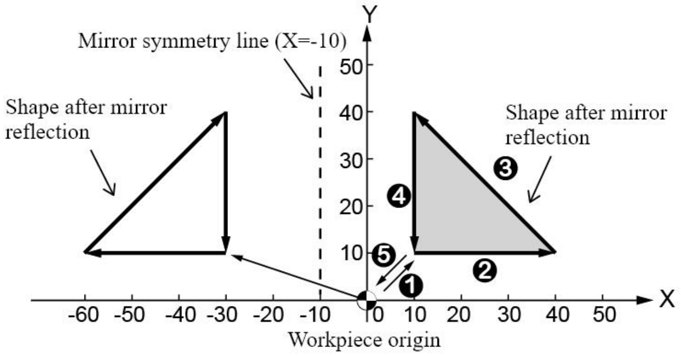


Figure 7- 5 Mirror instruction

【Example2】：

Shape (1)	Shape (2)	Shape (3)
G54 G90 G00 X0 Y0	G54 G90 G00 X0 Y0	G54 G90 G00 X0 Y0
G51.1 X0	G51.1 Y0	G51.1 X0 Y0
G01 X20. Y40. F500	G01 X20. Y40. F500	G01 X20. Y40. F500
X40. Y50.	X40. Y50.	X40. Y50.
X60. Y20.	X60. Y20.	X60. Y20.
X40. Y10.	X40. Y10.	X40. Y10.
X20. Y40.	X20. Y40.	X20. Y40.
G00 X0 Y0	G00 X0 Y0	G00 X0 Y0
G50.1	G50.1	G50.1
M30	M30	M30

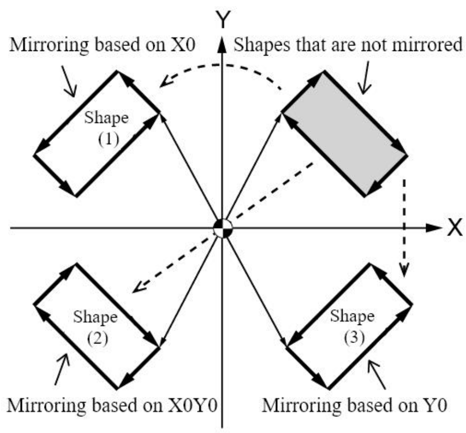


Fig. 7- 6 Mirror images of different baselines

【Example3】：

Even in G91 incremental mode, the specified mirror reference point coordinates are still absolute instruction values.

```
G54 G90 G0 X10. Y10.
G91
G51.1                          Mirror instruction (Mirror base: X20.)
X20
(1) G01 X20. Y40. F500
(2) X20. Y10.
(3) X20. Y-30.                 Shapes for mirror reflection
(4) X-20. Y-10.
(5) X-20. Y30.
G00 X-20. Y-40.
G50.1                          Unmirroring
M30
```

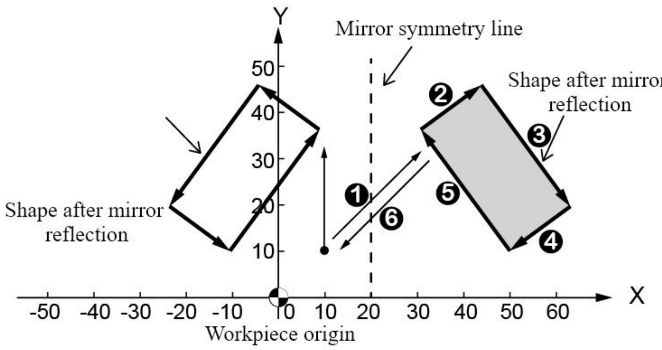


Figure 7- 7 G91 Mode Mirror Reflection

【Example4】:

Mirror image of circular interpolation

```
G54 G90 G00 X0 Y0
G51.1 X-10.
(1) G01 X10. Y10.
(2) Y70.
(3) X60.                       Shapes that are not mirrored
(4) G02 X70. Y60. R10.
(5) G01 Y10.
(6) X10.
G00 X0 Y0
G50.1
```

M30

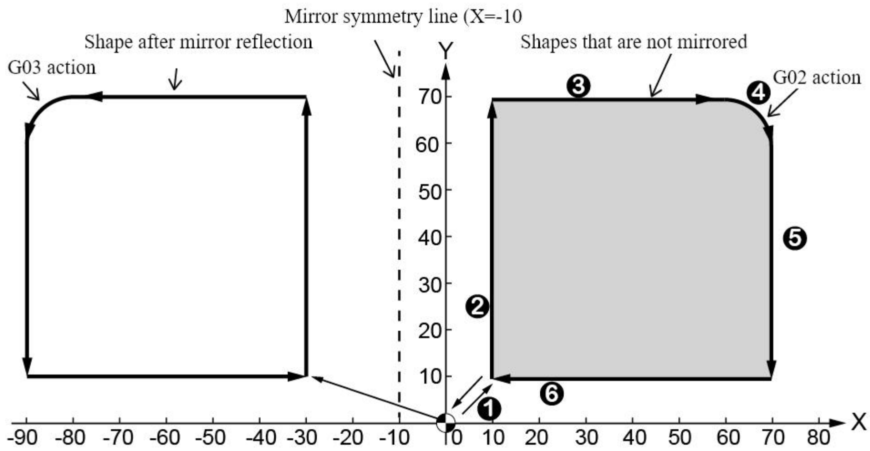


Fig. 7- 8Arc Interpolation Image

【Example5】：Simultaneous implementation of mirroring instruction and scaling instruction

G54 G90 G00 X0 Y0	
G51.1 X0	Mirror instruction (mirror symmetry line: X=0)
G51 X30. Y20. P1.5	Zoom instruction (zoom basis: X=30., Y=20.) (1)
(1)G01 X10. Y10.	
F500	
(2)Y30.	The original shape
(3)X50.	
(4)Y10.	
(5)X10.	
G50	Unzoom
G50.1	Unmirroring
M30	

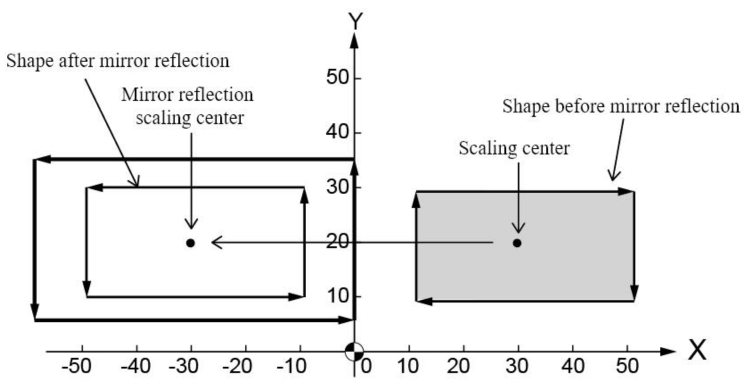


Figure7- 9 Shape of implementing mirroring and scaling instructions

7.3Coordinate rotation function (G68, G69)

Overview

【Functions】：

G68 allows edited shapes to be rotated around any point.

G69 cancels coordinate rotation function.

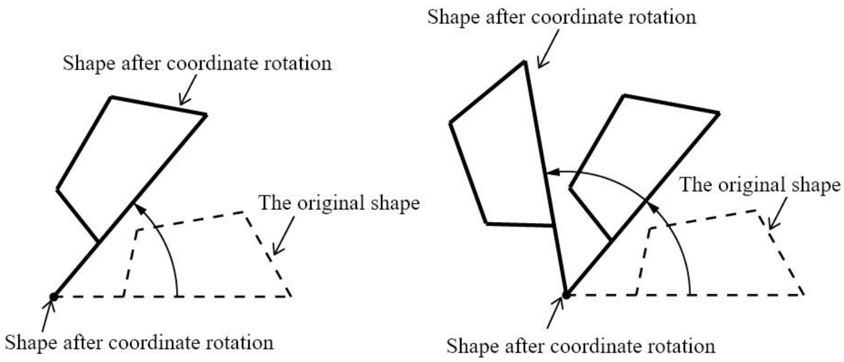


Figure 7- 10 Coordinate Rotation

【Instruction Format】：

G68 X_ Y_ Z_ R_ ;		Coordinate rotation instruction
G69 ;		Cancel coordinate rotation
X_ Y_ Z_	：	Absolute coordinates of rotation center
R_	：	Rotation angle (+: counterclockwise,-: clockwise)

Instruction description

(1) In G91 mode, R is superimposed as an increment to the previously set rotation angle.

(2) Before G68 instruction (coordinate rotation instruction) is implemented, the plane selection instruction (G17, G18, G19) must be used to select the plane to be rotated. The plane selection instructions (G17, G18, G19) are executed in the preceding block of the G68 instruction block. The default is G17 (XY plane) when the power is turned on.

(3) If the rotation angle R is not specified, an alarm will be displayed in operation.

(4) When executing coordinate rotation, zoom, and mirror directives on a shape at the same time, specify the directives in the order of mirror, zoom, and coordinate rotation.

(5) You cannot change the current workpiece coordinate system in coordinate rotation, scaling and mirroring.

(6) When X/Y/Z does not specify a value, the tool position of G68 program segment is the center of rotation.

(7) After the G68 instruction and before the absolute value instruction, the center of rotation of the increment value is the tool position.

(8) The first movement instruction after the coordinate system rotation cancellation instruction (G69) must use the absolute value instruction. If incremental value instruction is used, it is possible to move incorrectly.

Program legend

【Example1】:

Even in G91 incremental mode, the center of rotation specifies an absolute command.

G54 G17 G90 G00 X0 Y0	XY plane selection
G91 G68 X0 Y0 R90	Coordinate rotation instruction
(1)G90 G01 X40. Y20. F500	
(2)X80.	
(3)Y50.	Shapes without coordinate rotation
(4)X40.	
(5)Y20.	
G69	Coordinate rotation cancellation
M30	

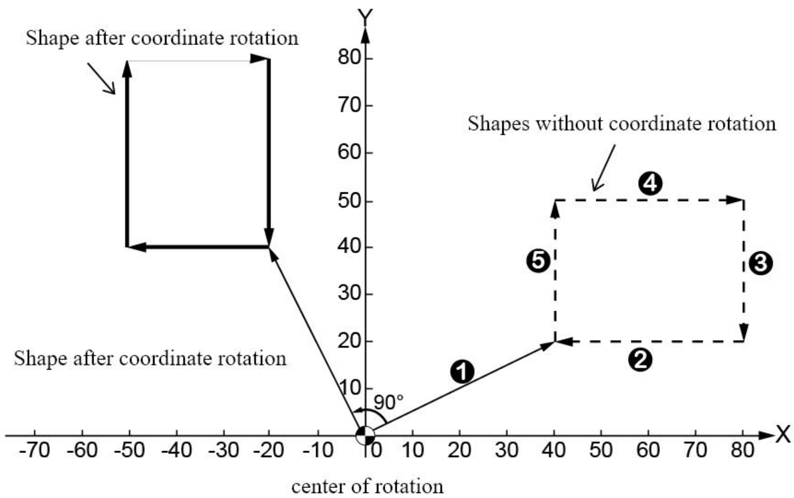


Figure7- 11 G91 Mode Coordinate Rotation

【Example2】：

After the G68 instruction, before the absolute value instruction is specified, the rotation center of the incremental instruction value is the tool position, and after the absolute value instruction is specified, the rotation center is the absolute instruction coordinate specified by the G68 instruction.

Shape (1)	**Shape (2)**
G54 G90 G00 X10. Y10. G68 X20.Y20.R60. G90 G01 X20.Y20. F500 G91 Y40. X20. G03 X20. Y-20. R20. G02 X-20. Y-20. R20. G01 X-20. G69 M30	G54 G17 G90 G00 X10. Y10. G68 X20. Y20. R180. G91 G01 X10. Y10. F500 Y40. X20. G03 X20. Y-20. R20. G02 X-20. Y-20. R20. G01 X-20. G69 M30

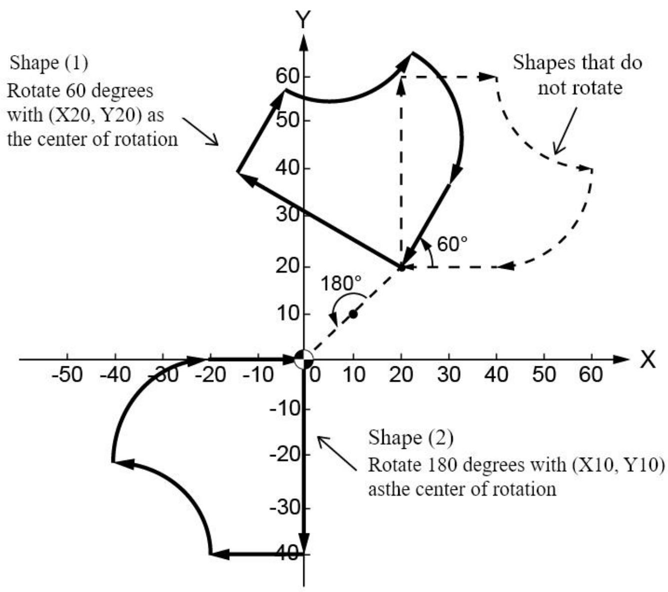


Figure 7- 12 G68 Command Coordinate Rotation

【Example3】：

At the same time, coordinate rotation instruction, scaling instruction and mirror instruction are executed

G54 G90 G00 X0 Y0	
G51.1 X0	Mirror instruction
G51 X30. Y20. P1.5	Zoom instruction
G68 X10. Y10. R60.	Coordinate rotation instruction
(1)G01 X10. Y10. F500	
(2)Y30.	The original shape
(3)X50.	
(4)Y10.	
(5)X10.	
G69	Coordinate rotation cancellation
G50	Zoom cancellation
G50.1	Mirror cancellation
M30	

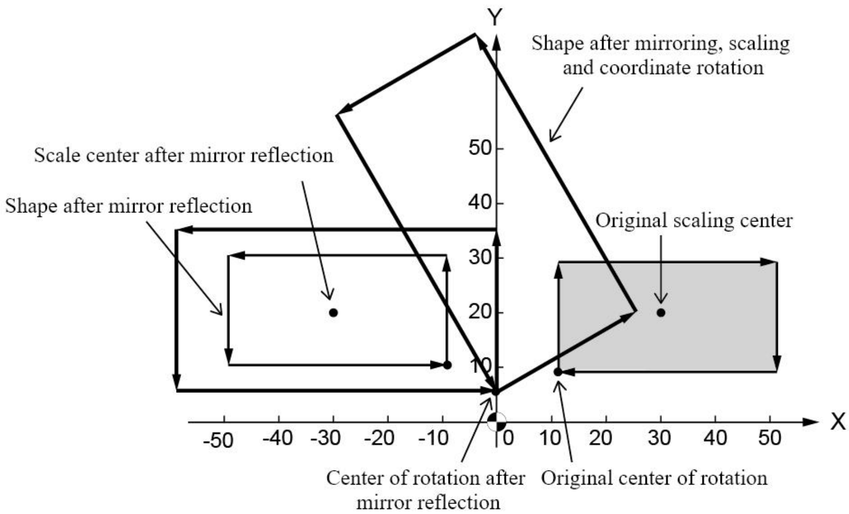


Figure 7- 13 Coordinate rotation for simultaneous image scaling

【Example4】：

In G91 mode, the rotation angle specified by the R command will be accumulated to the previously specified rotation angle.

G92 X0 Y0 G69 G17

G01 F200

M98 P2100

M98 P2200

G00 G90 X0 Y0

M30

```
O2200 G68 X0 Y0 G91 R45.
 G90 M98 P2100
M99

O2100 G90 G01 X0 Y-10.
X4.142
X7.071. Y-7.071
M99
```

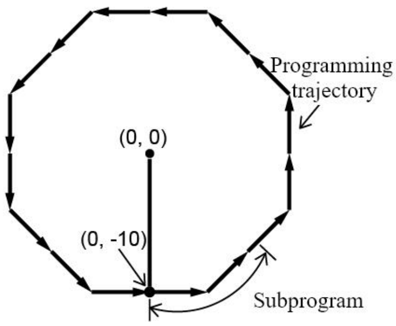


Figure 7- 14 G91 Mode R Command Coordinate Rotation

8.Tool length compensation (G43, G44)

8.1Summary

Overview

【Functions】：

The difference between the tool length when programming and the tool length actually used is set as the compensation of the tool length. Using tool length compensation instruction, it is unnecessary to consider the actual length of tools and the different length dimensions of each tool when programming. When the tool length changes due to tool wear, tool replacement and other reasons, only the tool length compensation needs to be corrected without adjusting the program or tool.

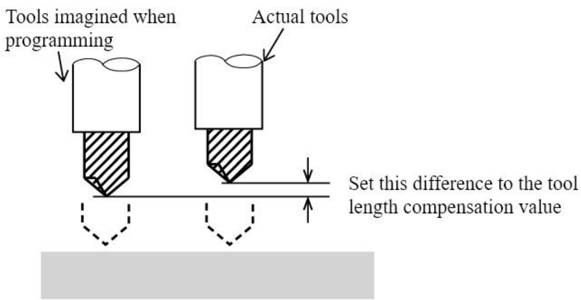


Figure 8- 1 Tool Length Compensation

【Instruction Format】：

G43 Z_ H_ ;		Tool length compensation (in positive direction)
G44 Z_ H_ ;		Tool length compensation (in negative direction)
G49		tool length compensation is taken Elimination
Z_	：	Tool length compensation start coordinates
H_	：	Code for tool length compensation value

Instruction description

(1) The tool length compensation functions G43, G44 and G49 are modal G instructions. Once G43 and G44 instructions are executed, they are valid until G49 cancellation is used. When the power supply is turned on, it defaults to G49 instruction (tool length compensation is taken Elimination).

(2) In the process of cutting, please do not implement the tool length compensation cancellation instruction (G49).

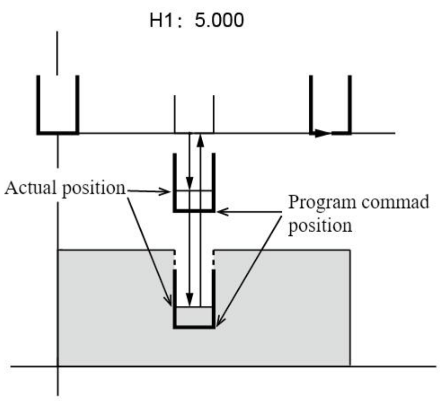
(3) The code H for tool length compensation value is as follows:

- A list of code H and corresponding tool length compensation values can be seen in the system's Info-Tool Info. The tool length compensation value is edited in [OFFSET-Tool Compensation].
- When the code H of the tool length compensation value is omitted, the system defaults to the code H of the last tool length compensation value.
- When the power supply is turned on, omit the tool long compensation code H, the system default selection code H0 (compensation value is 0).
- When the compensation value is set to a negative value, the tool length compensation direction is opposite to that when the compensation value is set to a positive value. (Eg. Legend 1)

Program legend

【Example1】：

G00 X30. Z60	
G43 Z40. H1	Tool length compensation starts
G01 Z10. F50	
G49 G00 Z60.	Tool length compensation cancellation
X60.	



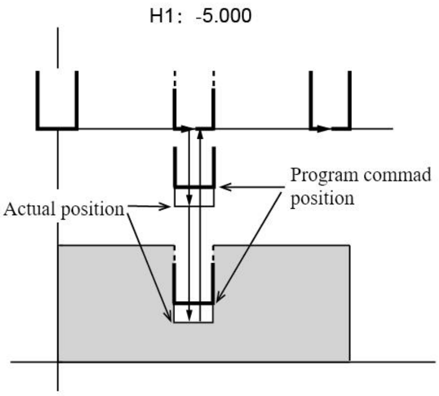


Figure8- 2 Positive and negative compensation values correspond to a length compensation

【Example2】:

G54 G90 G00 X Y Z60.	
X30. Y40.	
G43 Z35. H1	Tool length compensation starts
G01 Z10. F500	
G00 Z35.	
X50. Y20.	
G01 Z15.	
G00 Z35.	
X70. Y40.	
G01 Z5.	
G49 G00 Z60.	Tool length compensation cancellation
X0 Y0	
M30	
(H1=5.000)	

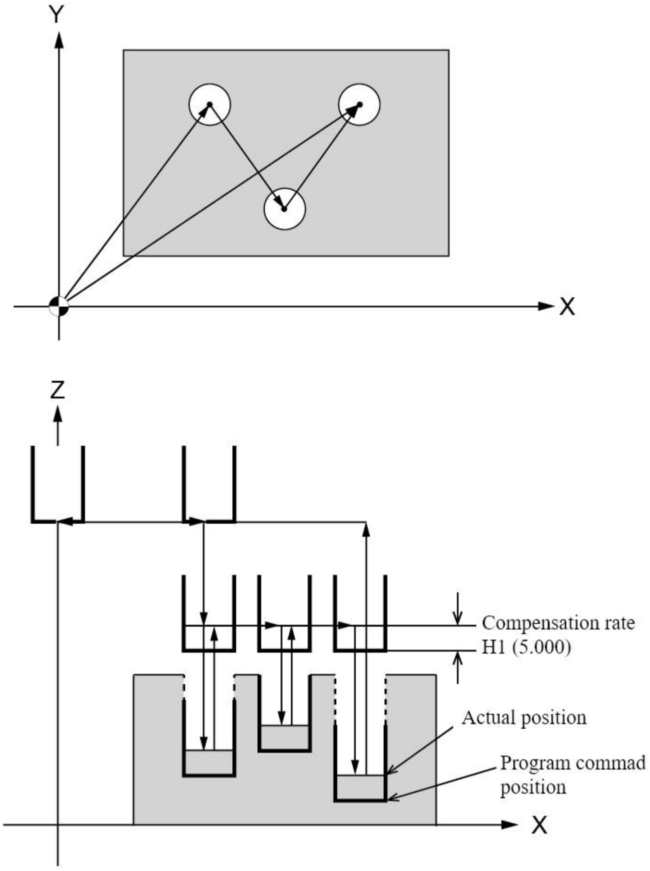


Figure 8- 3 Tool Length Compensation under Variable Z Value

8.2 Action

Overview

In the basic operation of the tool length compensation function, the tool length compensation amount specified by the code H is added or subtracted from the Z coordinate value relative to the end point of the axis movement command, and the resulting coordinate value becomes the actual end point. (G43 plus, G44 minus).

(1) Actions on startup

a) Axis movement command is limited to Z axis:

【Example】：

G90 G00 Z50.

G43 Z10. H1 (H1=5.000)

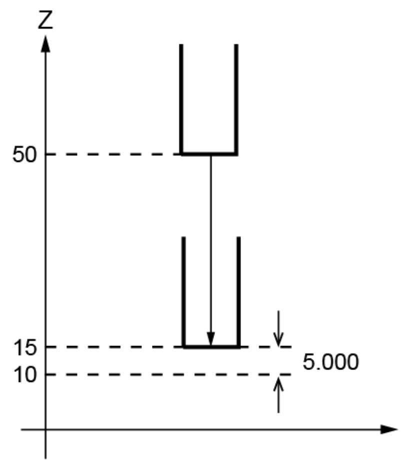


Figure8- 4 Single Z Axis Moving Length Compensation

b) When the axis movement command includes Z axis and other axes:

【Example】：

G90 G00 X20. Z50.

G43 X50. Z10. H1 (H1=5.000)

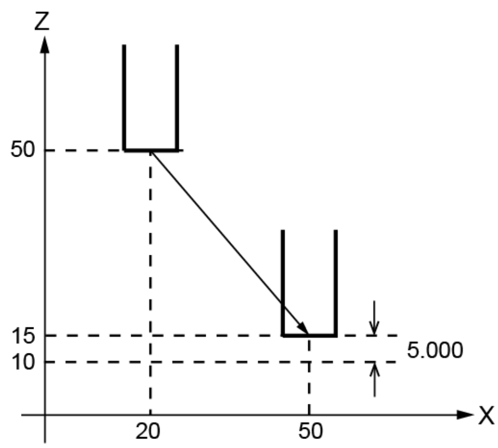


Fig. 8- 5 Multi-axis Moving Length Compensation

(2) Action when tool length compensation is cancelled

a) Axis movement command is limited to Z axis:

【Example】：

(G43 mode, compensation 5.000)：

G00 Z10.

G49 Z50.

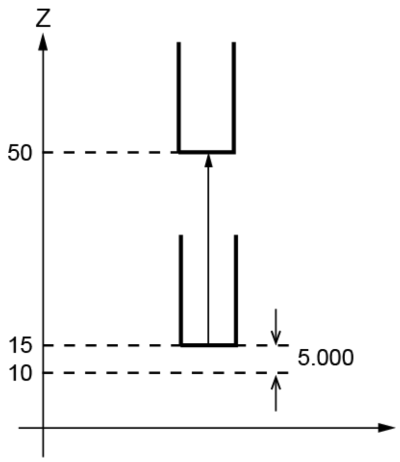


图 8-6 单 Z 轴移动长补偿取消

b) When the axis movement command includes Z axis and other axes:

【Example】：

(G43 mode, compensation 5.000))：

G00 X50. Z10.

G49 X20. Z50.

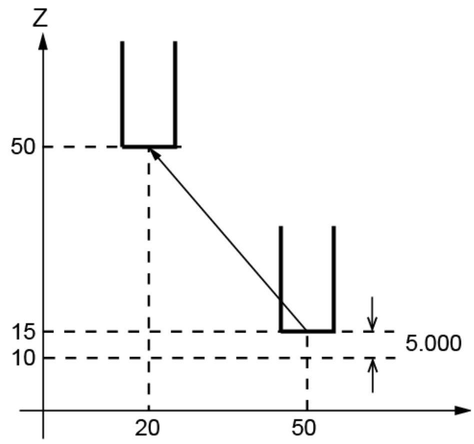


Fig. 8-7 Cancellation of Multi-axis Moving Length Compensation

8.3Change of tool length compensation amount

Overview

By executing the tool length compensation instruction and the code H of the new tool length compensation value, machining can be performed according to the new tool length compensation amount.

【Example】：

G90 G54 G00 X0 Y0 Z40.	(1)	
G00 X20. Y40.	(2)	change the tool lengt compensation
G43 Z0. H1 (H1 = 5.000)	(3)	
G01 Z-30. F200	(4)	
G00 Z0	(5)	

X40.	(6)
G01 Z-30.	(7)
G00 Z0	(8)
G43 X60. Y25. H2 (H2 = 10.000)	(9)
G01 Z-30.	(10)
G00 Z0	(11)
X80.	(12)
G01 Z-30.	(13)
G49 G00 Z40.	(14)
X0 Y0	
M30	

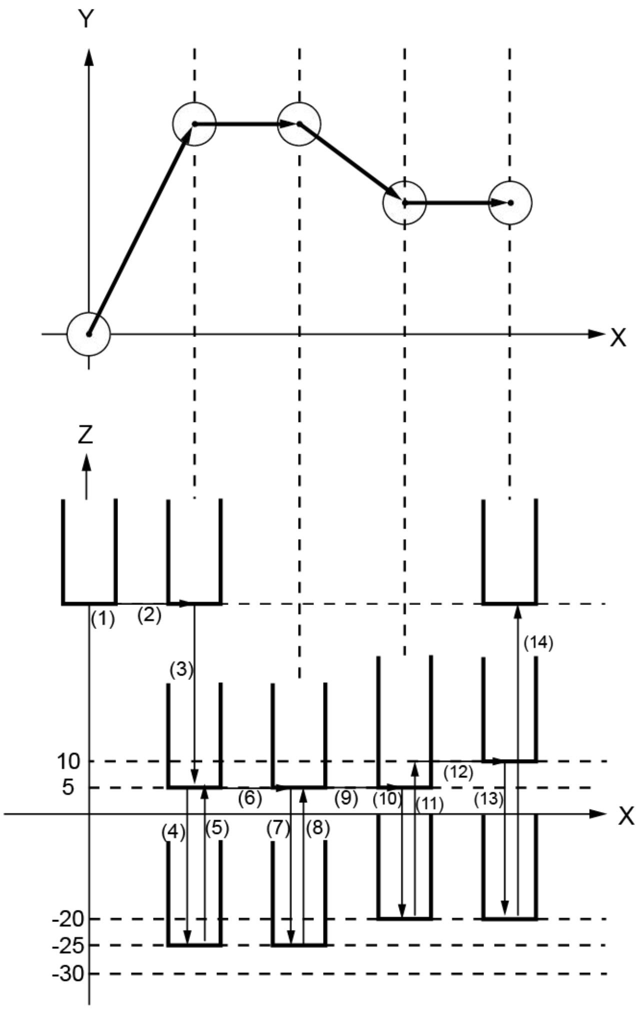


Figure8- 8 Changing Tool Length Compensation Amount

9.Tool radius compensation (G41, G42)

9.1Summary

Overview

【Functions】：

Because of the influence of tool radius, the center trajectory of tool is often inconsistent with the contour of workpiece when milling the contour of workpiece. In actual machining, if the toolcenter cuts along the outline of the workpiece in the design drawing, it will lead to the phenomenon that the tool radius is over-cut in every machining path, and the correct shape cannot be machined. In order to avoid calculating the tool center trajectory, the system provides the tool radius compensation function. Through this function, even if the tool radius changes, the programmer can directly program according to the contour size on the workpiece drawing.

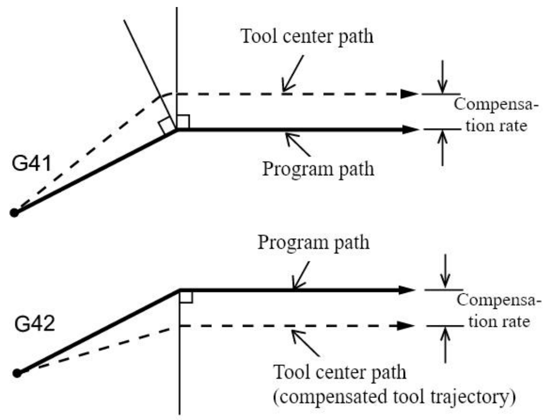


Figure 9- 1 Tool radius compensation

【Instruction Format】：

G41 X_ Y_ D_ F_ ;		Tool radius compensation (when left compe nsation)
G42 X_ Y_ D_ F_ ;		Tool radius compensation (right compensation)
G40 X_ Y_ ;		取消工具半径补偿
X_ Y_	：	Cancel tool radius compensation
D_	：	Tool radius compensation start coordinate value

Instruction description

(1) The tool radius compensation functions G40, G41 and G42 are modal G instructions.

Once executed, G41 and G42 remain valid until conditions such as cancellation using G40 or "error occurred" and "program stopped being pressed" occur. When the power is turned on, the default command is G40 (cancel tool radius compensation).

(2) G41 is the tool radius left compensation instruction, that is, look along the tool forward direction (assuming that the workpiece is not moving and the tool is located at the tool The left side of the outline of the piece; G42 is the tool radius right compensation instruction, that is, when viewed along the direction of tool advancement (assuming the workpiece is not moving), the tool is located to the right of the workpiece profile.

(3) Do not execute the Cancel Tool Radius Compensation command while cutting.

(4) The radius compensation function can only be started or cancelled in the straight line mode of G00 or G01.

(5) Regarding the selection of compensation plane, it is explained as follows:

- When executing the tool radius compensation instruction, the compensation plane (the plane where tool compensation is performed) must be selected. Instructions are executed through plane selection functions G17, G18 and G19. The default is G17 (XY plane) when the power is turned on.
- The compensation plane cannot be switched in the tool radius compensation mode.
- In the tool radius compensation mode, the compensation direction and compensation amount cannot be switched.
- In this item, the XY plane (G17) is described as an object. Other planes (ZX plane G18, YZ plane G19) and so on.

(6) Code D for tool radius compensation value is as follows:

- the tool length compensation Code D and a list of the corresponding tool radius compensation values can be viewed in the system's Info-Tool Info. The tool radius compensation value is edited in [OFFSET-Tool Compensation].
- When code D for the tool radius compensation value is omitted, the system defaults to code D for the last compensation value.
- When the power is turned on, the system selects the code D0 by default (the compensation value is 0).
- When the compensation value is set to a negative value, the tool radius

- compensation direction is opposite when the compensation value is set to a positive value (as shown in Figure 9-2).

Program legend

【Example1】：

```
G54 G90 G00 X Y
G41 G01 X40. D1 F500
X60. Y40.
X80.
X100. Y0
G40 G00 X140.
M30
```

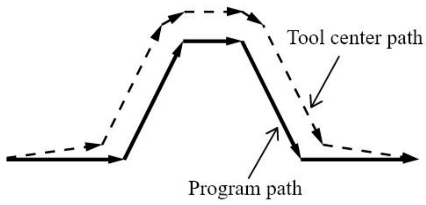
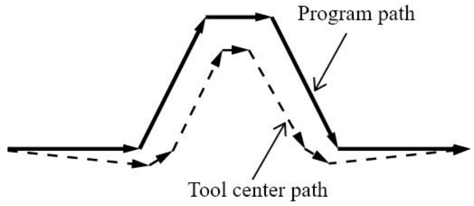
Set value for D: 10.0000	Setting value for D:-10.0000
	

Figure 9- 2 Positive and negative compensation values tool radius compensation direction is opposite

【Example2】：

G17 G90 G00 X Y Z	Select XY plane
G41 X-40. Y20. D1 F200	Tool radius compensation starts
G01 Y40.	
G02 X-20. Y60. R20.	
G01 X20.	
G02 X40. Y40. R20.	
G01 Y20.	
G40 G00 X Y	Cancel tool radius compensation
(D1 = 5.000)	

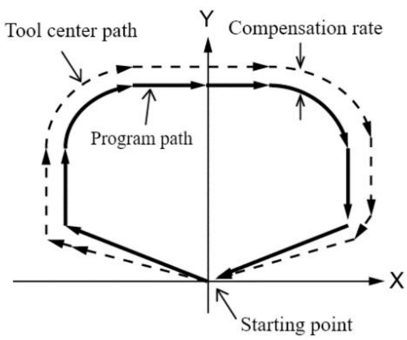


Fig. 9- 3 with arc radius compensation

Attention

When the radius compensation of machining arc is cancelled, if the angle is less than 90 degrees, it is easy to produce over-cutting phenomenon. It is suggested that The cutting angle should not be less than 90 degrees.

9.2 Start action

Overview

【Definition】:

The action when moving from the tool radius compensation cancellation mode to the tool radius compensation mode is called the start action. When in the tool radius compensation cancellation mode, the tool radius compensation instruction (G41, G42) is executed and then started.

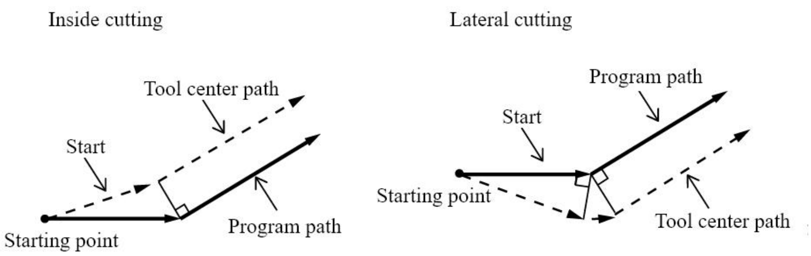


Figure 9- 4 Tool radius compensation start action

Description

(1) The axis movement command at startup must be G00 (positioning) or G01 (linear interpolation). G02 cannot be implemented, G03 (circular interpolation) instruction.

(2) If there is no axis movement instruction during startup, the startup action is executed in the axis movement block where the instruction is executed next time.

Program legend

【Example1】:

```
G54 G90 G00 X0 Y0
G41 G01 X40. D1 F500
X60. Y40.
X80.
```

X100. Y0

G40 G00 X140.

M30

(D = 10.0000)

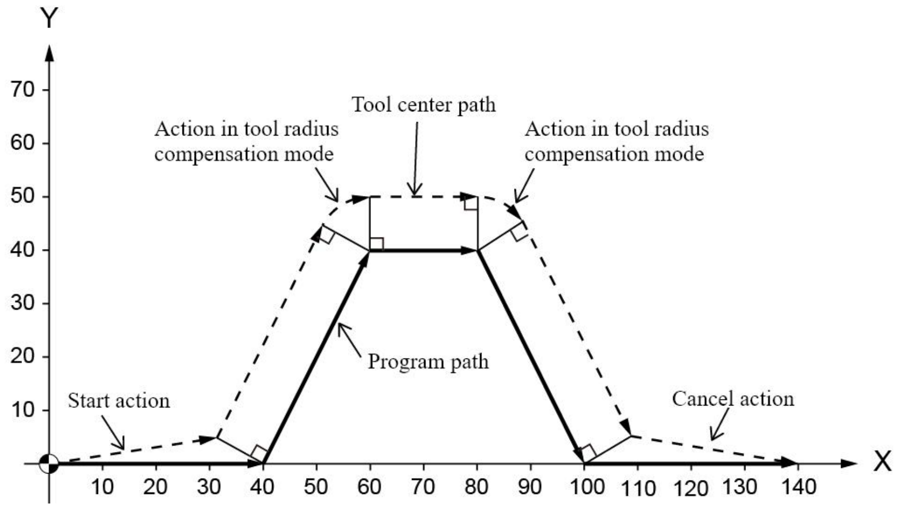
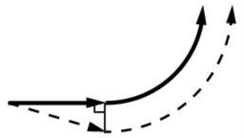
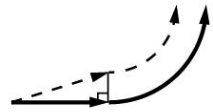
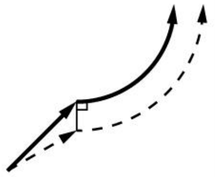
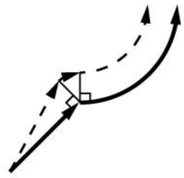
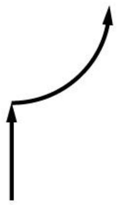
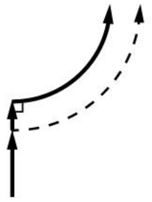
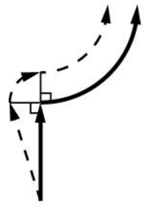
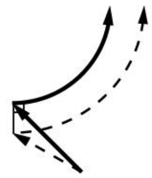
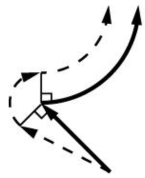
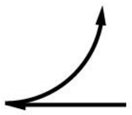
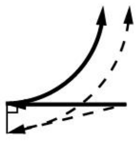
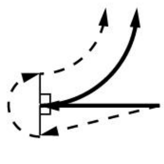
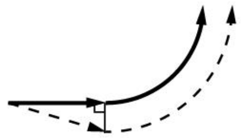
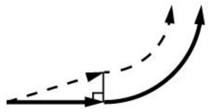
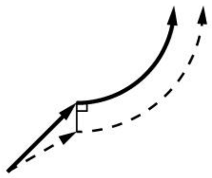
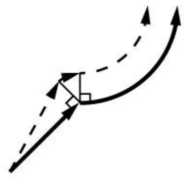
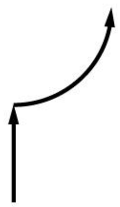
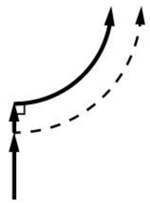
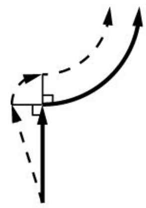
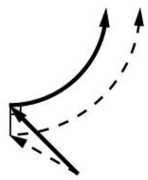
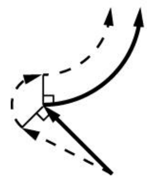
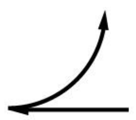
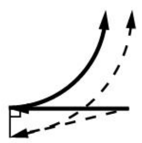
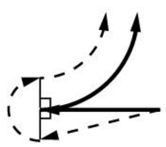


Figure 9- 5 Tool radius compensation mode

(1) Straight line → straight line

θ	Program path	Inside	Lateral
θ = 180°			
90°< θ <180°			
θ = 90°			
0°< θ <90°			
θ = 0°			

(2) Straight line → circular arc

θ	Program path	Inside	Lateral
θ = 180°			
90°< θ <180°			
θ = 90°			
0°< θ <90°			
θ = 0°			

9.3 Action in tool radius compensation mode

Overview

【Functions】：

In the tool radius compensation mode, G00 (positioning), G01 (linear interpolation), G02, G03 (circular interpolation) are used to compensate the tool radius to prevent the tool from feeding the workpiece excessively.

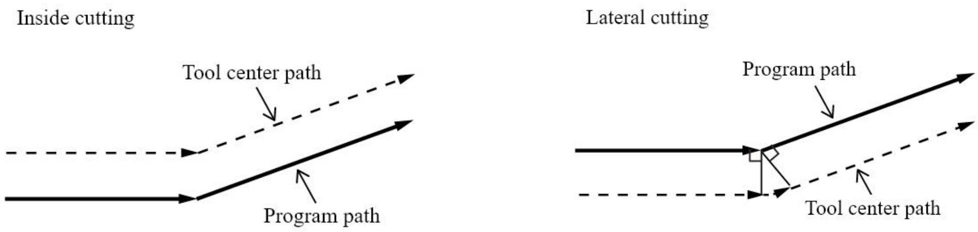


Figure 9- 6 Actions in Tool Radius Compensation Mode

Description

(1) The compensation plane cannot be switched in the tool radius compensation mode.

(2) When there are more than 20 consecutive blocks without axis movement instructions, the following message will be displayed:

- More than 20 non-interpolation instructions;
- Non-mobile commands can act normally when there are less than 19 commands.

Program legend

【Example1】：

```
G54 G90 G00 X0 Y0
G41 G01 X40. D1 F500
X60. Y40.
X80.
X100. Y0
G40 G00 X140.
M30
(D = 10.0000)
```

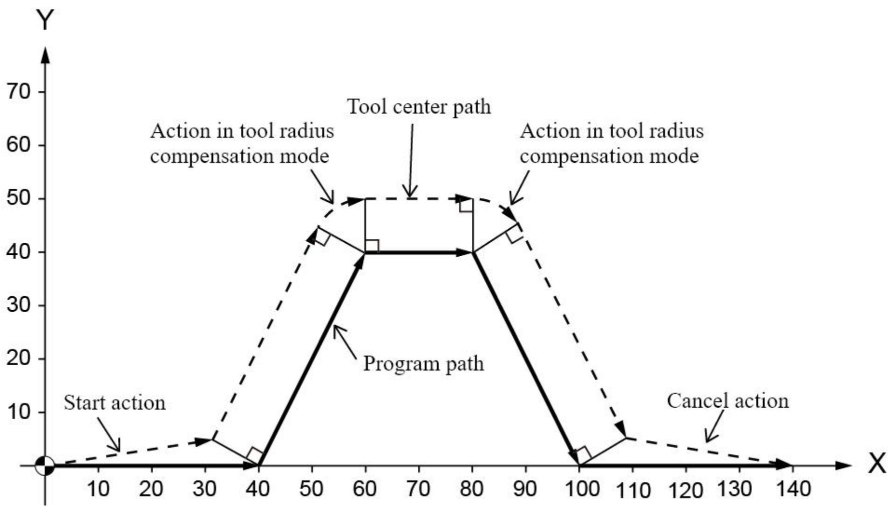
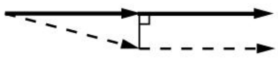
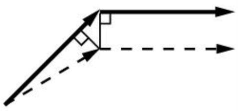
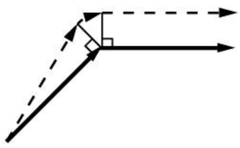
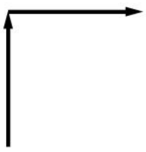
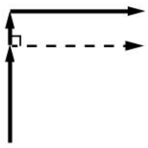
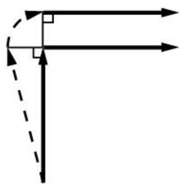
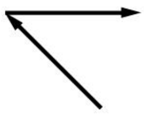
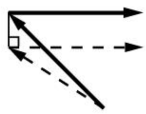
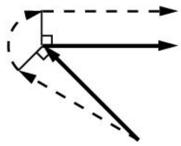
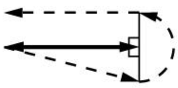
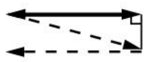
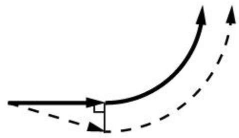
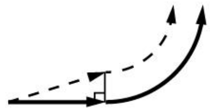
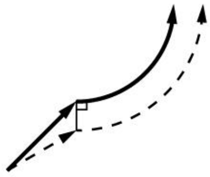
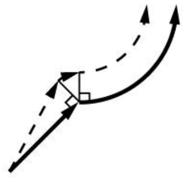
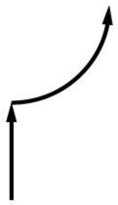
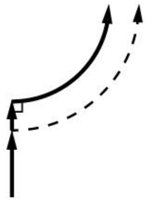
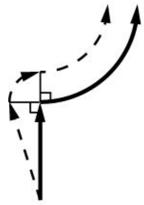
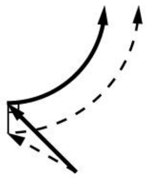
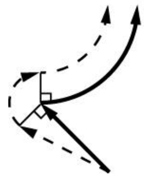
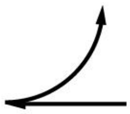
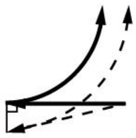
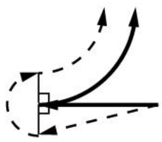


Figure9- 7 Tool radius compensation mode

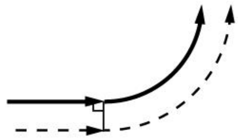
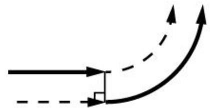
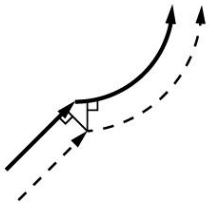
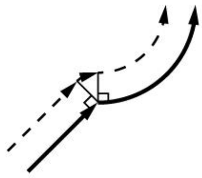
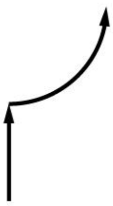
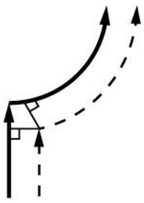
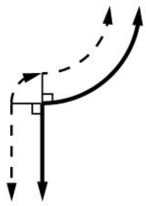
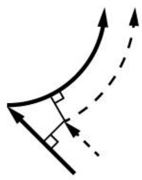
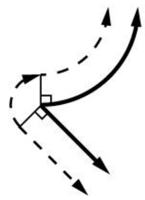
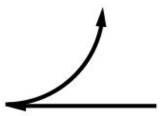
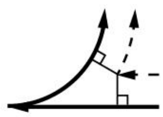
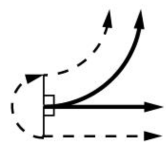
(1) Straight line → straight line

θ	Program path	Inside	Lateral
θ = 180°			
90°< θ <180°			
θ = 90°			
0°< θ <90°			
θ = 0°			

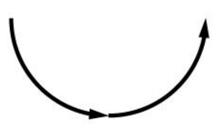
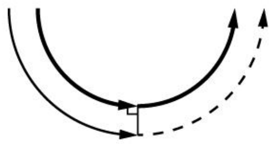
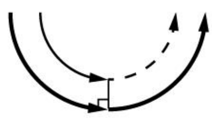
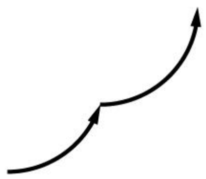
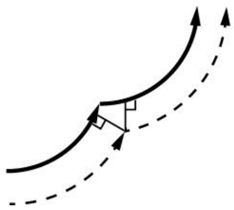
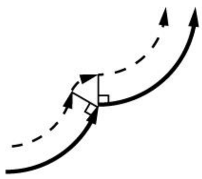
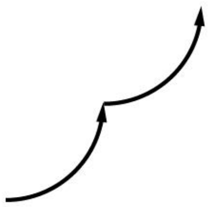
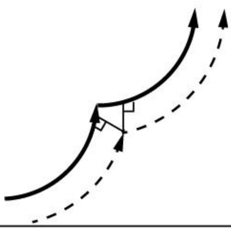
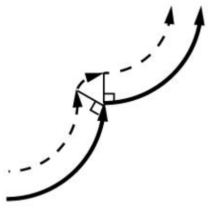
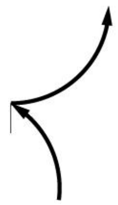
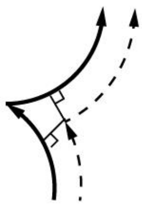
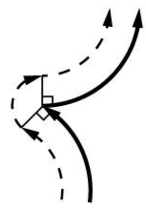
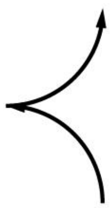
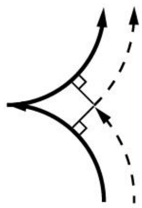
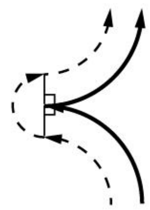
(2) Straight line → circular arc

θ	Program path	Inside	Lateral
θ = 180°			
90°< θ <180°			
θ = 90°			
0°< θ <90°			
θ = 0°			

(3) Circular arc → straight line

θ	Program path	Inside	Lateral
θ = 180°			
90°< θ <180°			
θ = 90°			
0°< θ <90°			
θ = 0°			

(4) Circular arc → circular arc

θ	Program path	Inside	Lateral
θ = 180°			
90°< θ <180°			
θ = 90°			
0°< θ <90°			
θ = 0°			

9.4Cancel action

Overview

【Definition】:

The action of moving from tool radius compensation mode to tool radius compensation cancellation mode is called cancellation action. In the tool radius compensation mode, the cancel action is performed when the cancel tool radius compensation instruction (G40) is executed.

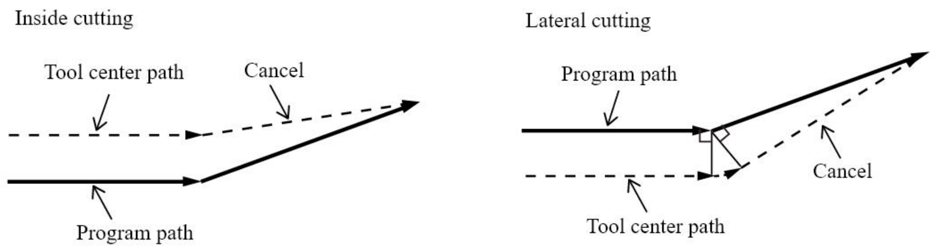


Figure 9- 8 Tool Radius Compensation Cancel Action

Program legend

【Example1】：

```
G54 G90 G00 X0 Y0
G41 G01 X40. D1 F500
X60. Y40.
X80.
X100. Y0
G40 G00 X140.
M30
(D = 10.0000)
```

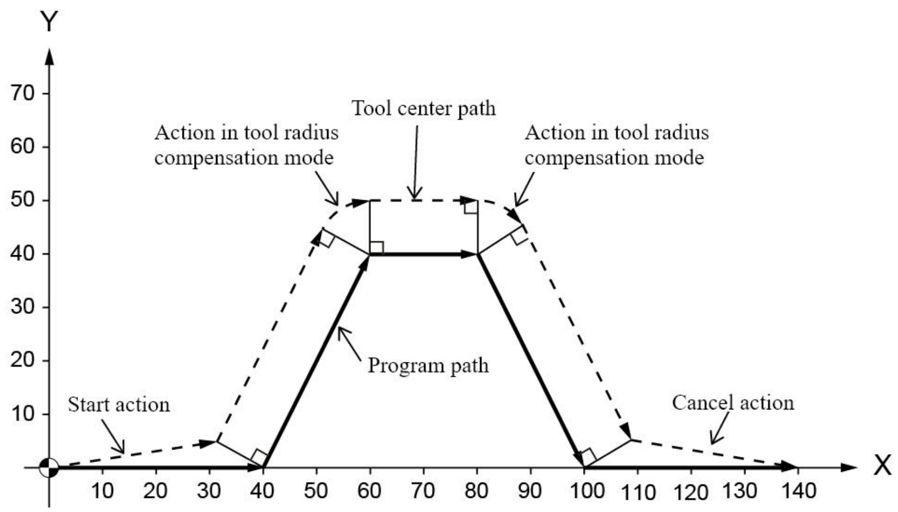
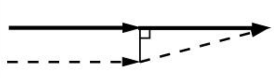
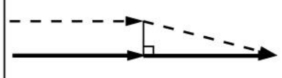
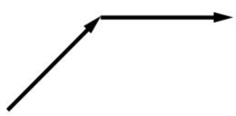
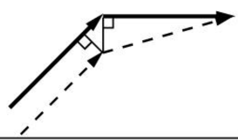
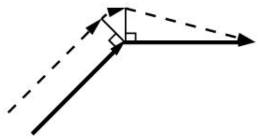
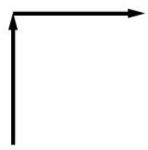
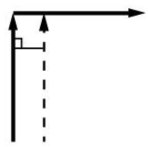
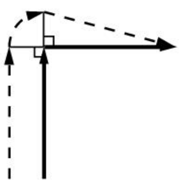
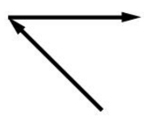
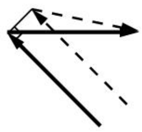
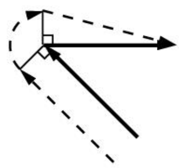
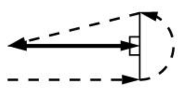
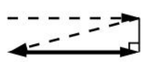
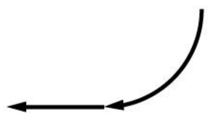
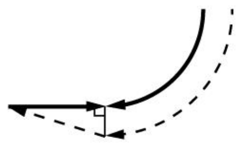
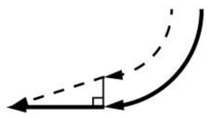
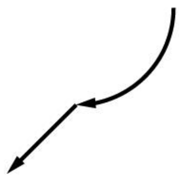
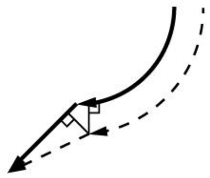
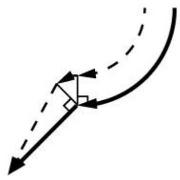
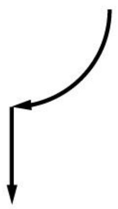
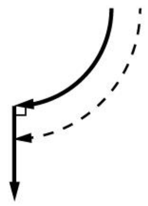
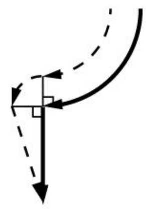
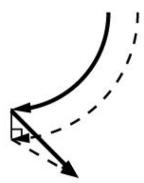
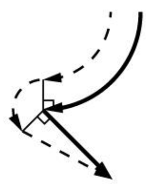
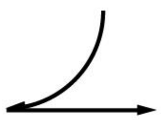
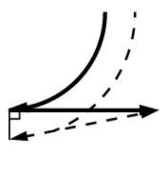
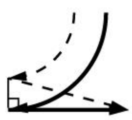


Figure 9- 8 Tool radius compensation mode

(1) Straight line → straight line

θ	Program path	Inside	Lateral
θ = 180°			
90°< θ <180°			
θ = 90°			
0°< θ <90°			
θ = 0°			

(2) Circular arc → straight line

θ	Program path	Inside	Lateral
θ = 180°			
90°< θ <180°			
θ = 90°			
0°< θ <90°			
θ = 0°			

9.5 Implementation of NC instruction in tool radius compensation

Overview

【Functions】:

NC instructions in tool radius compensation can be divided into the following two categories:

- NC instruction that can be directly implemented in tool radius compensation;

- NC instructions that can be implemented after automatically temporarily canceling the tool radius compensation function.

In tool radius compensation mode Directly executable NC instruction	NC instructions that can be implemented after automatically temporarily canceling tool radius compensation mode (tool withdrawal)
G00, G01, G02, G03, G04, G05, G17, G18, G19, G20, G21, G40, G41, G42, G43, G44, G49, G9, G61, G64, G65, G52, G51, G50, G51.1, G50.1, G68, G69, G54-G59, G154-G159, G254-G259, G354-G359, G454-G459, G554-G559, G654-G659, G754-G759, G854-G859, G954-G959, G90, G91, G92, M98, M99	G10, G27, G28, G29, G32, G53, 固定循环指令, M02, M30 M00, M01, M03, M05, M06, M07, M08, M09

Program legend

【Example1】：

No automatic tool withdrawal (NC instruction that can be directly implemented in tool radius compensation mode)

```
G54 G90 G00 X Y Z
G41 G01 X40. D1 F500
X60. Y40.
X80.
X100. Y0
G40 G00 X140.
M30
(D1 = 10.0000)
```

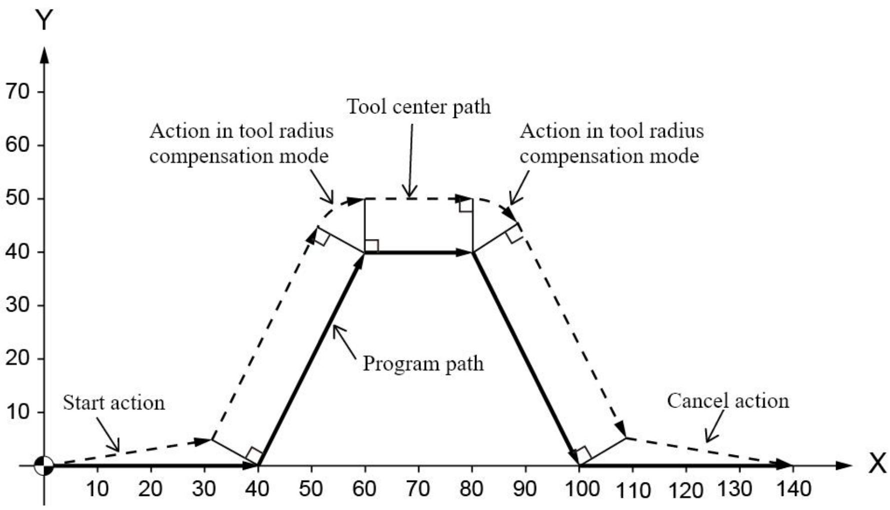


Figure 9- 9Tool radius compensation mode

【Example2】：

Move command automatic tool withdrawal (NC command that can be implemented after automatically temporarily canceling tool radius compensation mode)

G54 G90 G00 X Y Z
G41 G01 X40. D1 F500
X60. Y40.
G53 X80. ((Move Command Automatic Temporary Tool Withdrawal))
X100. Y0.
G40 G00 X140.
M30
(G54origin of workpiece coordinate system:X，Y，Z = 10.，10.，20.；D1 = 10.0000)

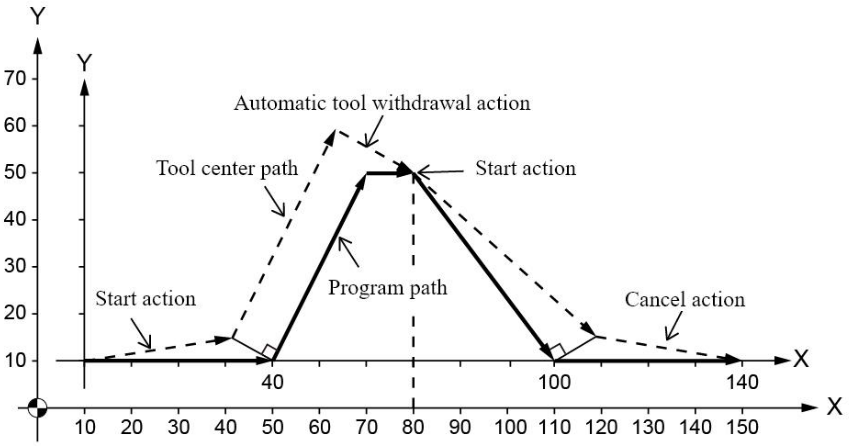


Fig.9- 10Automatic Tool Withdrawing with Moving Command

【Example3】：

Non-moving command automatic tool withdrawal (NC command that can be implemented after automatically temporarily canceling tool radius compensation mode).

G54 G90 G00 X Y Z

G41 G01 X40. D1 F500

X60. Y40.

G32 P1 X100. Y101. (Non-moving command automatic temporary tool withdrawa)

X80.

X100. Y0.

G40 G00 X140.

M30

(D1 = 10.0000)

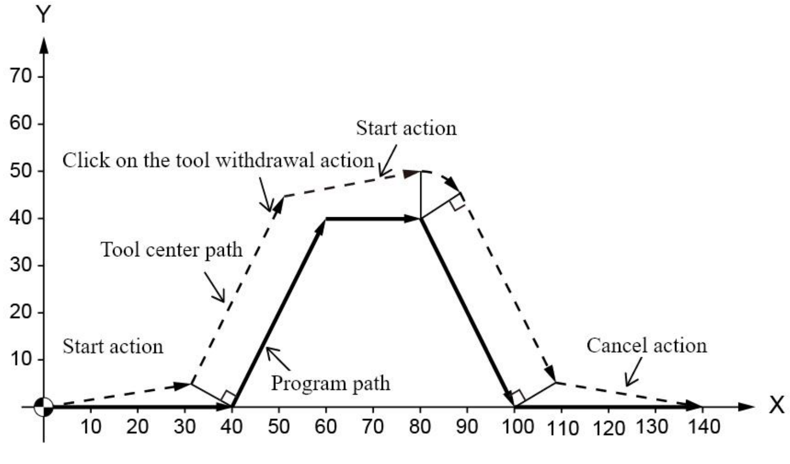


Fig.9- 11 Automatic cutter withdrawal with non-moving command

10.Fixed cycle

Overview

【Functions】：

It is not necessary to specify a plurality of machining actions which are frequently used in a plurality of blocks, but may be specified in a block containing the G function. In this way, program editing becomes simple, and at the same time, it can shorten the program and make effective use of memory.

Drilling and boring data can be specified in a block after the following G instruction.

Table 10- 1 Fixed cycle Instruction List

Instruction	Function
G70	Circumferential mode
G71	Arc mode
G72	Linear mode
G73	High-speed deep drilling circulation
G76	Fine boring cycle
G80	Fixed cycle cancellation
G81	Fixed-point drilling cycle
G82	Cycle delay of fixed-point drilling
G83	Wood pecking drilling cycle
G84	Rigid tapping fixing cycle
G85	Boring cycle
G86	Boring cycle
G87	Back boring
G98	Return to the starting point
G99	Return to R point

Instruction description

The fixed cycle consists of the following six actions.

(1) Axis positioning

(2) Fast forward to R point

(3) Hole machining

(4) Action at the bottom of the hole

(5) Return to R Point

(6) Return to the starting point

Program legend

【Example1】：

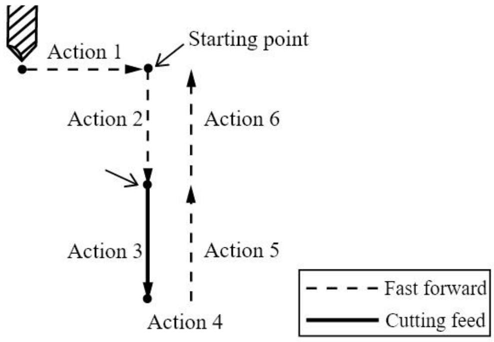


Fig. 10- 1 Fixed cycle action

Attention

(1) In the fixed cycle, the general positioning plane is the XY plane (G17), and the Z axis is used as the drilling axis.

(2) When the program specifies ZX plane (G18) and YZ plane (G19), the corresponding Y axis and X axis become drilling axes, while other axes become positioning axes.

(3) When executing G98 instruction, return to the starting point after machining to the bottom of the hole; When executing G99 instruction, return to R point after machining to the bottom of hole.

【Example2】：

Different planes correspond to drilling shafts

G17 G81……Z_	The drilling axis is Z axis
G18 G81……Y_	The drilling axis is Y axis
G19 G81……X_	The drilling axis is X axis

10.1Circumferential mode (G70)

Overview

【Functions】：

The tool is positioned equidistantly on the circumference.

【Instruction Format】：

G70 I_ J_ L_ ;

I_	：	The radius value of the circle (I > 0)
J_	：	Angle from the center of the starting circle
L_	：	Number of holes in an arc

Instruction description

(1) The G70 instruction must be executed together with the G instruction (G73, G81 ~G83) specified in the previous fixed loop.

(2) The center coordinates of the circumference pattern specified by G70 are the X, Y coordinate values in the fixed loop specified in the previous block.

Program legend

【Example1】：

```
G90 G54 G00 Z100. S2500 M03
G99 G81 X90. Y90. Z-20. R3. F200 L0
G70 I40. J45. L8
G80
```

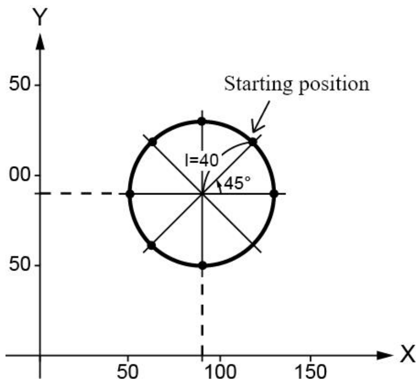


Fig. 10- 2 Circumferential Mode

10.2Arc mode (G71)

Overview

【Functions】：

The tool is positioned equidistantly on the arc.

【Instruction Format】：

G71 I_ J_ K_ L_ ;

I_	：	The radius value of the circle (I > 0)
J_	：	Angle from the center of the arc at the starting point
K_	：	Angle between holes
L_	：	Number of holes in an arc

Instruction description

(1) The G71 instruction must be executed together with the G instruction (G73, G81 ~ G83) specified in the previous fixed loop.

(2) The center coordinates of the circumference pattern specified by G71 are the X, Y coordinate values in the fixed loop specified in the previous block.

Program legend

【Example1】：

G90 G54 G00 Z100. S2500 M03

G99 G81 X70. Y60. Z-20. R3. F200 L0

G71 I50. J0 K30. L4

G80

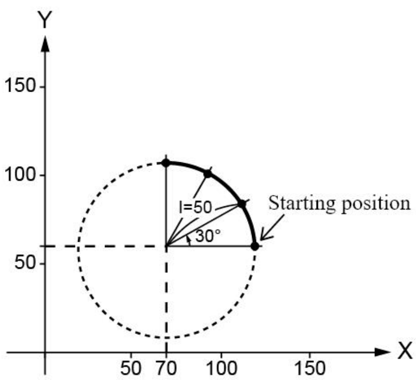


Fig. 10- 3 Arc Mode

10.3Linear mode (G72)

Overview

【Functions】:

The tool is positioned equidistantly on a straight line.

【Instruction Format】:

G72 I_ J_ L_ ;		
I_	:	Distance between holes (I > 0)
J_	:	Angle of line to X axis
L_	:	Number of holes in a line

Instruction description

(1) The G72 instruction must be executed together with the G instruction (G73, G81 ~ G83) specified in the previous fixed loop.

(2) The center coordinates of the line pattern specified by G72 are the X, Y coordinate values in the fixed loop specified in the previous block.

(3)

Program legend

【Example1】:

```
G90 G54 G00 Z100. S2500 M03
G99 G81 X40. Y40. Z-18. R3. F200 L0
G72 I20. J45. L5
```

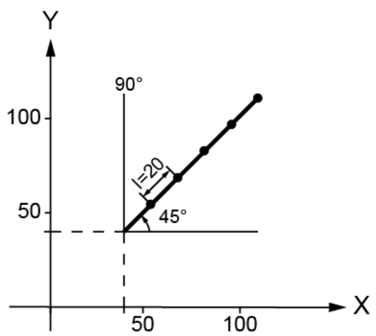


Figure 10- 4 Linear Mode

10.4High-speed deep drilling cycle (G73)

Overview

【Functions】：

After fast-forward positioning at the specified positions on the X and Y axes, drill holes at the specified feed speed from point R until the depth specified by Q. Then, fast forward back to the distance set in [G73 space left]. Then, continue drilling until the distance of the next Q specified position [G73 space left] is reached. This movement mode is repeated until the Z specified position is reached, and then fast forward back from Z point.

【Instruction Format】：

G73 X_ Y_ Z_ R_ Q_ F_ (K_) ;		
X_	：	X coordinate of hole position
Y_	：	Y coordinate of hole position
Z_	：	Z coordinate of hole bottom position
R_	：	Z coordinate of point R
Q_	：	Feed per feed
F_	：	Cutting feed speed
K_	：	Number of repetitions (Default value is 1, pecified as needed.)

Instruction description

(1) G73 is valid until G80 (Fixed Loop Cancel) is specified and is implemented in all blocks containing X-axis and Y-axis movement commands.
(2) [G73 Space left] is set in [System-Parameters-Common].
(3) The intermittent cutting feed of high-speed deep drilling cycle is beneficial to discharge machining chips from the drilling hole.

Program legend

【Example1】：

```
G54 G90
M03 S1000
G98 (G99)
G73 X10. Y10. Z-30. R3. Q10. F100
G80
```

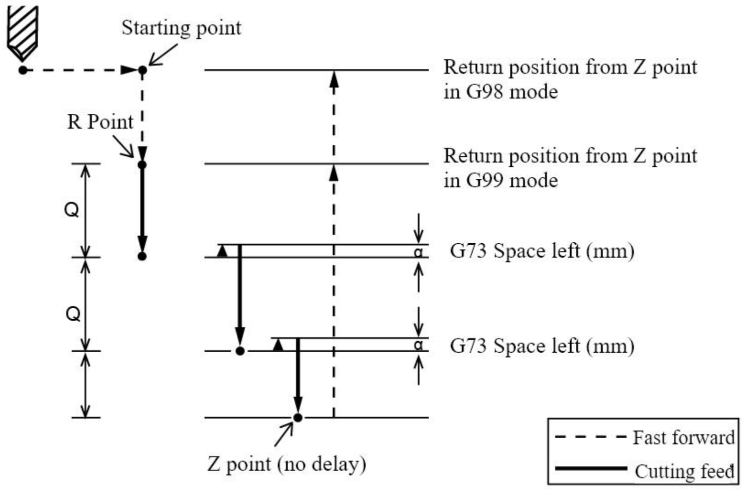


Fig. 10- 5 High Speed Deep Drilling Circulation

10.5Fixed-point drilling cycle (G81)

Overview

【Functions】:

After fast-forward positioning at the specified position on the X and Y axes, drill holes on the Z axis from point R at the specified feed speed until the specified position on Z is reached. Then fast forward and return immediately from the bottom of the hole.

【Instruction Format】:

G81 X_ Y_ Z_ R_ F_ (K_) ;		
X_	:	X coordinate of hole position
Y_	:	Y coordinate of hole position
Z_	:	Z coordinate of hole bottom position
R_	:	Z coordinate of point R
F_	:	Cutting feed speed
K_	:	Number of repetitions (Default value is 1, speci fied as needed.)

Instruction description

G81 is valid until G80 (Fixed Loop Cancel) is specified and is implemented in all blocks containing X-axis and Y-axis movement commands.

Program legend

【Example1】:

```
G81 X10. Y10. Z-10. R3. F100
X20. Y20.
X30.
X40. Y10.
G80
```

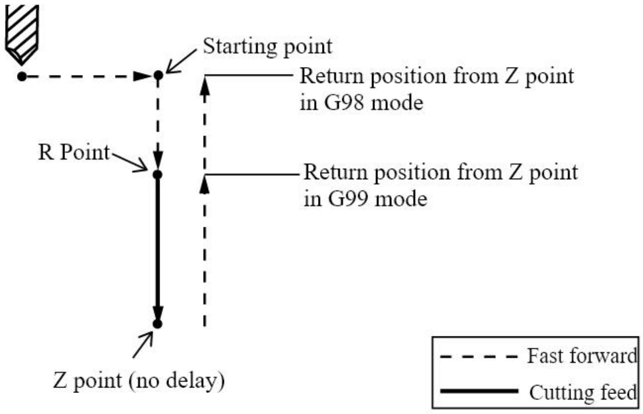


Fig. 10- 6 Fixed Point Drilling Cycle

10.6 Fine boring cycle (G76)

Overview

【Functions】:

Boring precision holes. After fast-forward positioning at the specified position on the X and Y axes, drill holes at the specified feed speed from point R, rotate the spindle clockwise, and bore holes to point Z. After the spindle stops P milliseconds at point Z, fast forward back to the distance set in [G76/G87 (finish/back boring) blank], then the spindle stops directionally at this position, and the tool moves an offset Q (or I and J) in the direction in which the tool head moves away from the inner surface of the machined workpiece. Then fast forward at fast feed speed to return to the return point. The tool moves back to offset Q in the direction of the tip, and then the spindle begins to rotate clockwise.

【Instruction Format】：

G76 X_ Y_ Z_ R_ Q_ (I_ J_) (P_) F_ (K_) ;

X_	：	X coordinate of hole position
Y_	：	Y coordinate of hole positio
Z_	：	Z coordinate of hole bottom position
R_	：	Z coordinate of point R
Q_	：	Offset at the bottom of the hole
I_ J_	：	Offset of X and Y to the bottom of the hole
P_	：	Pause time at the bottom of the hole (in millisecon ds,default value is 0, specified as needed)
F_	：	Cutting feed speed
K_	：	Number of repetitions (Default value is 1, specified as needed.)

Instruction description

(1) [G76/G87 (fine/reverse boring) blank] is set in [system-parameters-common].

(2) If the Z parameter is greater than the R parameter, the system will generate an alarm.

(3) When the plane chooses to use G17, G18 or G19, only I and J of G17 plane are supported. For addresses I and J, all values should be set to incremental values. The positive and negative values of I and J determine the direction of migration. The values of I, J: $I=|Q|\sin\theta$， $J=|Q|\cos\theta$ (Q is the normal offset, θ is the angle of the normal to the Y axis)

(4) When the Q parameter is used to specify the offset, the direction is determined by [the avoidance direction of G76/G87 parameter Q], where 0 ~ 3 corresponding to X +, X-, Y + and Y-respectively. If the Q parameter is less than zero, the Q parameter will be automatically positive.

(5) If the offset is not specified by Q or I and J, an alarm signal will be generated.

(6) The K parameter can be specified as needed (the default is 1).

(7) The P parameter can be specified as needed (the default is 0).

(8) When Q, I and J parameters are specified, the system gives priority to Q and ignores I and J.

Program legend

【Example】：

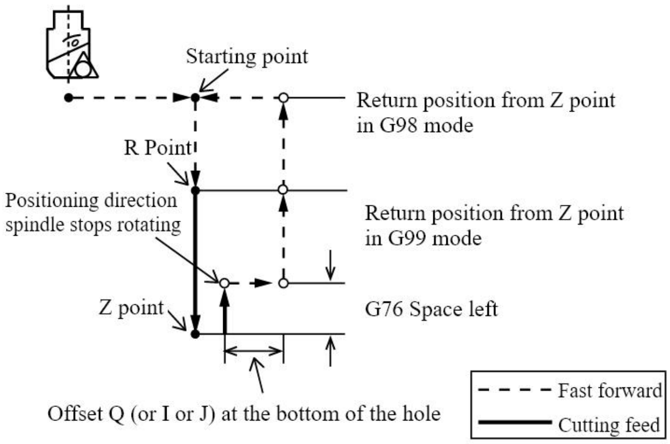


Fig.10- 7 Fine Boring Cycle

Attention

Please rotate the spindle before executing this instruction, otherwise an alarm signal will be generated.

10.7Fixed-point drilling cycle delay (G82)

Overview

【Functions】：

After fast-forward positioning at the specified position on the X and Y axes, drill holes on the Z axis from point R at the specified feed speed until the specified position on Z is reached. Then, wait for the time specified by P from the bottom of the hole and fast forward back.

【Instruction Format】：

G82 X_ Y_ Z_ R_ (P_) F_ (K_) ;

X_	：	X coordinate of hole position
Y_	：	Y coordinate of hole position
Z_	：	Z coordinate of hole bottom position
R_	：	Z coordinate of point R

P_	:	Delay time at the bottom of the hole (in milliseconds, default value is 0, specified as needed)
F_	:	Cutting feed speed
K_	:	Number of repetitions (Default value is 1, specified as needed.)

Instruction description

G82 is valid until the execution of the G80 (Fixed Loop Cancel) instruction andis implemented in all blocks containing the X-axis and Y-axis move instructions.

Program legend

【Example】：

```
G82 X10. Y10. Z-10. R3. P100 F100
X20. Y20.
X30.
X40. Y10.
G80
```

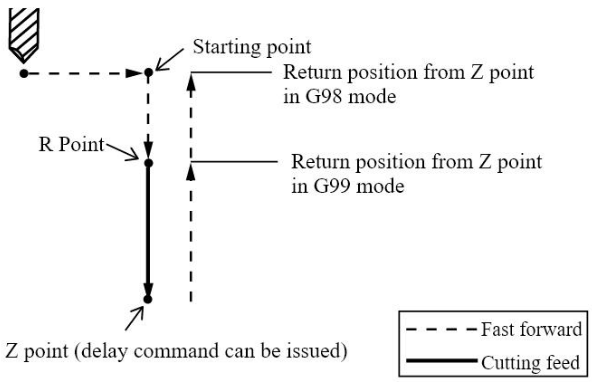


Fig. 10- 8 Cycle Delay of Fixed Point Drilling

10.8Wood pecking drilling cycle (G83)

Overview

【Functions】：

After fast-forward positioning at the specified positions on the X and Y axes, drill holes at the specified feed speed from the R point, and the depth is specified by the Class I instruction. Then, first fast forward back to the R-point position, and then fast forward to the position specified in [G83 blank amount]. Then, the increment of Q-n × J (n is 0, 1, 2, 3...) and K parameter are compared, and the larger one is taken as the feed depth of the next drilling. After reaching the position specified by Z, the waiting time at the bottom of the hole can be specified by P instruction.

【Instruction Format】：

G83 X_ Y_ Z_ R_ I_ Q_ (J_ L_) (P_) F_ (K_) ;

X_	：	X coordinate of hole position
Y_	：	Y coordinate of hole position
Z_	：	Z coordinate of hole bottom position
R_	：	Z coordinate of point R
I_	：	Initial feed
Q_	：	Standard feed for the second start
J_	：	The amount of reduction each time from the third ti me
L_	：	Minimum feed
P_	：	Delay time at the bottom of the hole (unit: milliseco nds, default value is 0, specified as required)
F_	：	Cutting feed speed
K_	：	Number of repetitions (Default value is 1, specified as needed.)

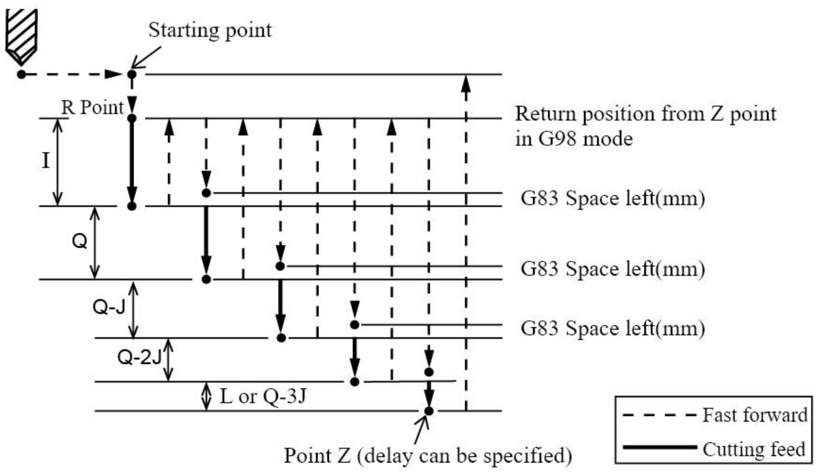


Fig. 10- 9 Wood pecking drilling cycle

Instruction description

(1) G83 is valid until G80 (Fixed Loop Cancel) is specified and is implemented in all blocks

containing X-axis and Y-axis movement commands.

(2) [G83 Allowance] is set in [System-Parameters-Common].

(3) The intermittent cutting feed of wood pecking drilling cycle is beneficial to discharge machining chips from drilling holes.

Program legend

【Example】:

```
G54 G90
M03 S1000
G98 (G99)
G83 X10. Y10. Z-30. R3. I6. Q5. J1. L3. F100
G80
M30
```

10.9Rigid tapping fixing cycle (G84)

【Functions】:

After fast-forward positioning at the specified position of X and Y axes, tapping at the specified feed speed starts from R point, and the depth of each cutting feed is specified by Q instruction. After tapping reaches the specified depth of Q, the feed returns to the distance set in [G84 blank]. This moving mode is repeated until it reaches the specified position of Z, and the waiting time at the bottom of the hole can be specified by P instruction.

In G99 mode, return to R point according to cutting feed speed. In G98 mode, after returning to R point according to cutting feed speed, fast forward to returnto the starting point position surface.

【Instruction Format】:

G84 X_ Y_ Z_ R_ (P_) Q_ F_ (K_) (J_) ;

X_	:	X coordinate of hole position
Y_	:	Y coordinate of hole position
Z_	:	Z coordinate of hole bottom position
R_	:	Z coordinate of point R
P_	:	Time to pause at the bottom of the hole. (Units: mill isecond, default value is 0, specified as needed)
Q_	:	Cutting depth per cutting feed

F_	:	Cutting feed speed
K_	:	Number of repetitions (Default value is 1, specified as ne eded.)
J_	:	Reverse speed. When not specified or the specified speed is lower than the feed speed, the default is feed speed

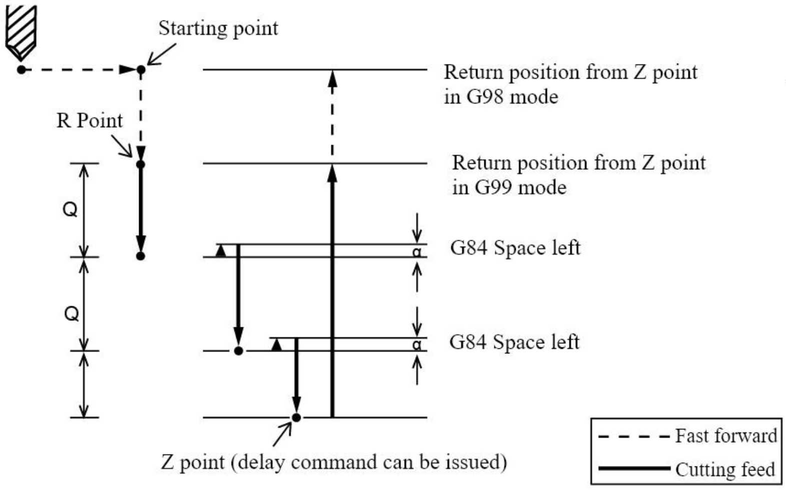


Fig. 10- 10 Rigid tapping fixing cycle

10.10Reverse rigid tapping fixing cycle (G74)

【Functions】：

After fast-forward positioning at the specified position of X and Y axes, reverse tapping at the specified feed speed starts from R point, and the depth of each cutting feed is specified by Q instruction. After tapping reaches the specified depth of Q, the feed returns to the distance set in [G84 blank]. This moving mode is repeated until it reaches the specified position of Z, and the waiting time at the bottom of the hole can be specified by P instruction.

In G99 mode, return to R point according to cutting feed speed. In G98 mode, after returning to R point according to cutting feed speed, fast forward to returnto the starting point position surface.

【Instruction Format】：

G74 X_ Y_ Z_ R_ (P_) Q_ F_ (K_) (J_) ;

X_	:	X coordinate of hole position
Y_	:	Y coordinate of hole position
Z_	:	Z coordinate of hole bottom position

R_	:	Z coordinate of point R
P_	:	Time to pause at the bottom of the hole. (Units: millisecond, default value is 0, specified as needed)
Q_	:	Cutting depth per cutting feed
F_	:	Cutting feed speed
K_	:	Number of repetitions (Default value is 1, specified as needed.)
J_	:	Reverse speed. When not specified or the specified speed is lower than the feed speed, the default is feed speed

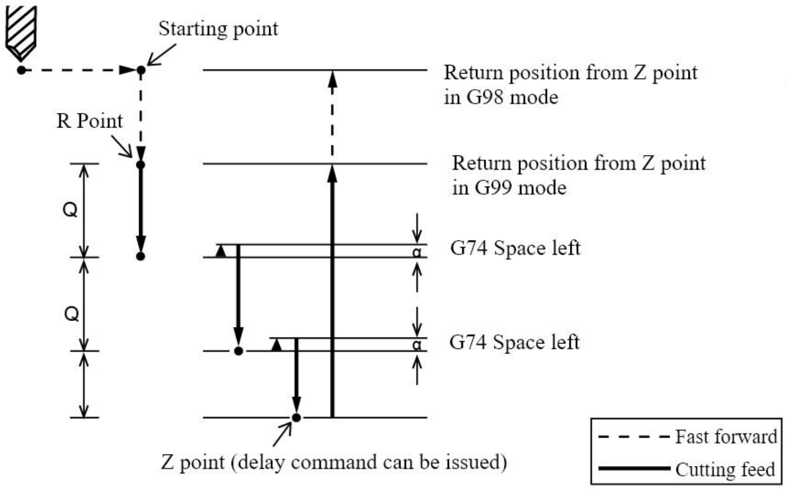


Fig. 10- 11 Reverse Rigid Tapping Fixing Cycle

10.11Boring cycle (G85)

Overview

【Functions】:

Cycle of boring holes. After fast-forward positioning at the specified position on the X and Y axes, bore the hole at the specified feed speed from point R, rotate the spindle clockwise, and bore the hole to point Z. After the spindle stops at Z point for P seconds, it returns to R point according to the cutting feed speed (G99 mode). In G98 mode, the spindle rotates back to the R point at cutting feed speed, and then rotates back to the position surface of the starting point at fast forward speed.

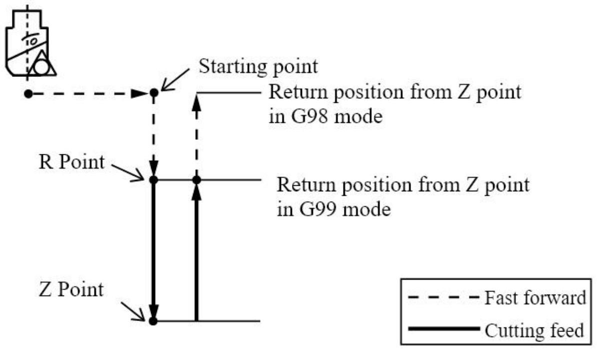


Fig. 10- 12 Boring cycle

【Instruction Format】:

G85 X_ Y_ Z_ R_ (P_) F_ (K_) ;

X_	:	X coordinate of hole position
Y_	:	Y coordinate of hole position
Z_	:	Z coordinate of hole bottom position
R_	:	Z coordinate of point R
P_	:	Time to pause at the bottom of the hole. (Unit: millisecond, the default value is 0, specified as needed)
F_	:	Cutting feed speed
K_	:	Number of repetitions (Default value is 1, specified as needed.)

Instruction description

(1) If the Z parameter is greater than the R parameter, the system will generate an alarm.

(2) The K parameter can be specified as needed (the default is 1).

(3) The P parameter can be specified as needed (the default is 0).

Attention

Please rotate the spindle before executing this instruction, otherwise an alarm signal will be generated.

10.12Boring cycle (G86)

Overview

【Functions】:

Cycle of boring holes. After fast-forward positioning at the specified position on the X and Y axes, bore the hole at the specified feed speed from point R, rotate the spindle clockwise, and bore the hole to point Z. After the spindle stops at Z point for P seconds, the spindle stops rotating and then returns to the return point at fast forward speed. After returning to the return point, the spindle starts to rotate clockwise again.

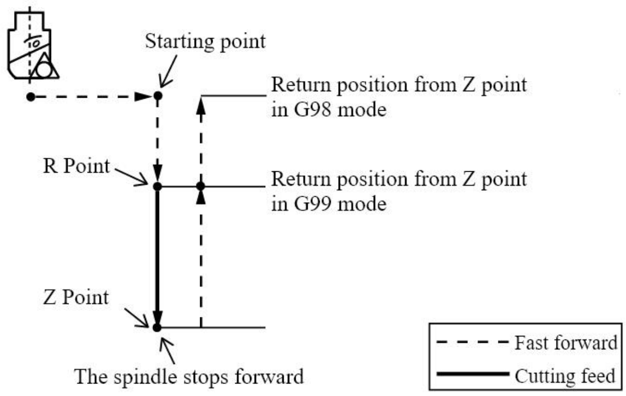


Fig. 10- 13 Boring cycle

【Instruction Format】:

G86 X_ Y_ Z_ R_ (P_) F_ (K_) ;		
X_	:	X coordinate of hole position
Y_	:	Y coordinate of hole position
Z_	:	Z coordinate of hole bottom position
R_	:	Z coordinate of point R
P_	:	Pause time at the bottom of the hole (unit: millisec onds, default value is 0, specified as required)
F_	:	Cutting feed speed
K_	:	Number of repetitions (Default value is 1, specifie d as needed.)

Instruction description

(1) If the Z parameter is greater than the R parameter, the system will generate an alarm

(2) The K parameter can be specified as needed (default is 1)

(3) The P parameter can be specified as needed (default is 0)

Attention

Please rotate the spindle before executing this instruction, otherwise an alarm signal will be generated.

10.13Back boring (G87)

Overview

【Functions】:

Cycle of precision boring. After positioning at the specified position on the X and Y axes at fast forward speed, the spindle stops (spindle orientation). Then, according to the setting of offset Q (or I and J), it is offset in the opposite direction to the tip of the tool, and then positioned to the R point at a fast forward speed. At point R, the tool returns an offset Q (or I and J) and the spindle begins to rotate clockwise to bore upwards. Bore the hole at the specified cutting feed speed to point Z, and stop at point Z for P seconds. After that, the Z axis returns according to the setting of [G76/G87 (fine/back boring) blank amount] and the spindle stops rotating (spindle orientation). The tool compensates for an offset Q (or I and J) in the opposite direction to the tool tip and returns to the return point at a fast feed speed. Then return to the starting point according to the offsets Q (or I and J).

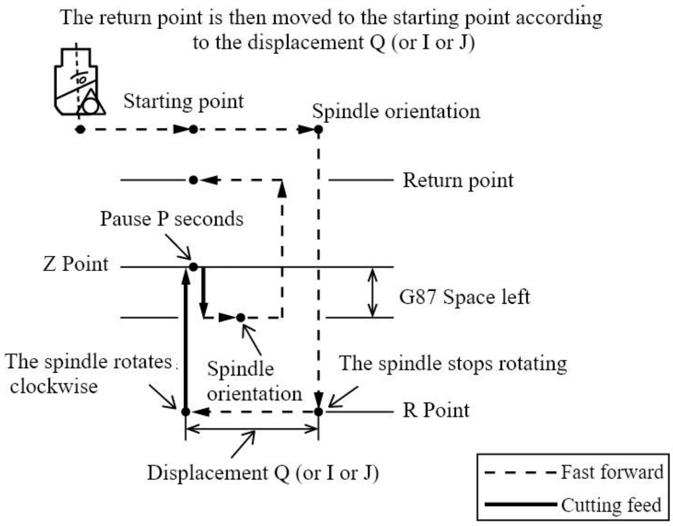


Figure 10- 14 Back Boring

【Instruction Format】：

G87 X_ Y_ Z_ R_ Q_ (I_ J_) (P_) F_ (K_) ;		
X_	：	X coordinate of hole position
Y_	：	Y coordinate of hole position
Z_	：	Z coordinate of hole bottom position
R_	：	Z coordinate of point R
Q_	：	Offset at the bottom of the hole
I_ J_	：	Hole bottom offset in X and Y directions
P_	：	X coordinate of hole position
F_	：	Y coordinate of hole position
K_	：	Z coordinate of hole bottom position

Instruction description

(1) [G87 space left] is set in [System-Parameters-Common].

(2) If the R parameter is greater than the Z parameter, the system will generate an alarm.

(3) When the plane chooses to use G17, G18 or G19, only I and J of G17 plane are supported. For addresses I and J, all values should be set to incremental values. The compensation direction is always defined in the machine tool coordinate system. The values of I and J are I = Qsin θ, J = Qcos θ (Q is the normal offset, and θ is the angle between the normal and the Y axis).

(4) If the offset is not specified by Q or I and J, an alarm signal will be generated.

(5) When the Q parameter is used to specify the offset, the direction is determined by [the avoidance direction of G76/G87 parameter Q], where 0 ~ 3 corresponding to X+, X-, Y+ and Y-respectively. If the Q parameter is less than zero, the Q parameter will be automatically positive.

(6) The K parameter can be specified as needed (the default is 1).

(7) The P parameter can be specified as needed (the default is 0).

(8) When Q, I and J parameters are specified, the system gives priority to Q and ignores I and J.

Attention

Please rotate the spindle before executing this instruction, otherwise an alarm signal will be generated.

10.14Fixed cycle cancellation (G80)

Overview

【Functions】:

Cancel all fixed cycle.

【Instruction Format】:

G80 ;

Instruction description

If the G00/G01/G02/G03 instruction is executed while the fixed loop is active, the fixed loop is canceled.

10.15Revert to the starting point (G98)

Overview

【Functions】:

Take the starting point in the fixed loop as the return point.

【Instruction Format】：

G98 ;

Program legend

【Example】：

G98

G81 X10. Y10. Z-10. R3. F100

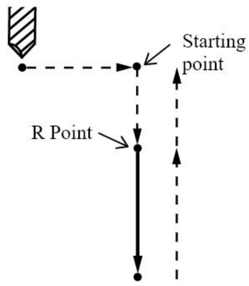


Figure 10- 15Revert to the starting

10.16Revert to R point (G99)

Overview

【Functions】：

Take the R point in the fixed loop as the returned point.

【Instruction Format】：

G99 ;

Program legend

【Example】：

G99

G81 X10. Y10. Z-10. R3. F100

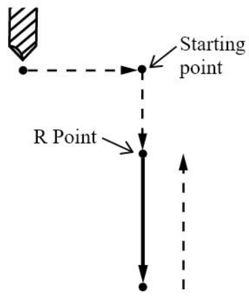


Figure 10- 16 Reverts to R Point

10.17High-speed drilling cycle (G81.1)

Overview

【Functions】：

When several holes with small spacing are punched at one time, the positioning action in XY direction is completed at the same time in the process of lifting the tool from the previous hole to R point and the next hole from R point, so that the punching efficiency is optimized.

【Instruction Format】：

G81.1 X_ Y_ Z_ R_ F_ (I_) (K_) ;

X_	：	X coordinate of hole position
Y_	：	Y coordinate of hole position
Z_	：	Z coordinate of hole bottom position
R_	：	Z coordinate of point R
F_	：	Cutting feed speed
I	：	Safe height for positioning between holes (default 1.0 mm, specified as required)
K_	：	Number of repetitions (Default value is 1, speci fied as needed.)

Instruction description

G81.1 is valid until G80.1 (Fixed Loop Cancel) is specified and is implemented in all blocks containing X-axis and Y-axis movement instructions.

Program legend

【Example】：

G81.1 X10. Y10. Z-10. R3. F100
X13.
X16.
X19.
G80.1

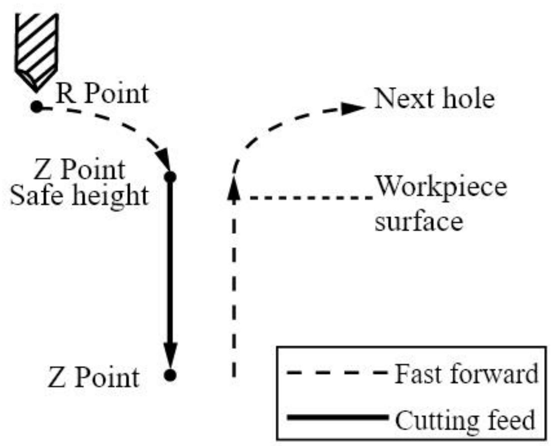


Figure 10- 17Revert to R Point

10.18Cancel the high-speed drilling cycle (G80.1)

Overview

【Functions】:

Cancel the high-speed drilling cycle.

【Instruction Format】:

G80.1 ;

Instruction description

G80.1 must be used in pairs with G81.1.

11.Sfunction (spindle function)

Overview

【Functions】：

Send the code signal to the machine tool by specifying the address S and the value 1 ~ 5 after it, which is used for the spindle control of the machine tool and specifies the rotation speed (revolution/minute) of the spindle. It is executed together with M03 (spindle forward rotation). For details of M03, please refer to the instructions provided by the machine tool manufacturer.

【Instructions description:】：

The spindle rotation speed can be obtained from the following formula:

$$S\ (\mathrm{min}^{-1}) = \frac{1000 \times V}{\pi \times D}$$

V：Cutting speed (m/min)

D： Tool diameter (mm)

Program legend

【Example】：

```
G90 G54 G00 X0 Y0
G43 H1 Z5.
M08
The S3500 M03 spindle rotates forward at a speed of 3500 rpm
:
:
M30
```

Attention

Once an S instruction is executed, it remains valid until the next time another S instruc tion is executed.

12.Tool function

12.1 Tool selection function

Overview

【Functions】:

The tool on the machine tool is selected by instruction category T and the following two digits. In a program segment, you can instruct T codes. When the move instruction and the T code are instructed in the same program segment, the move instruction and the T code start at the same time.

【Instruction Format】:

T_ ;

【Instructions description】:

(1) T instruction and M code are used together to realize tool change action; Through H instruction and G43/G44 code, tool length compensation is provided; By using D instruction and G41/G42 code, it provides tool radius compensation function.

(2) It is represented by T instruction and its 2-digit value, specifying the tool number to use. Execute with M06 (Automatic Tool Exchange Instruction). Reference: For details of M06, please refer to [14.1.7].

Program legend

【Example】:

Return the tool in the spindle to the tool holder, and install the tool with tool number 1 on the spindle.

T01 M06 or T01 M06

Attention

The T instruction is executed in principle in the block preceding or in the same block as the automatic switching tool instruction (M06).

12.2Tool compensation value setting

【Functions】：

The tool on the machine tool is selected by instruction category T and the following two digits. In a program segment, you can instruct T codes. When the move instruction and the T code are instructed in the same program segment, the move instruction and the T code start at the same time.

【Instruction Format】：

G10 L_ P_ R_ ;		
L_	：	L10: Long compensation; L11: Wear length compensation; L12: Radius compensation; L13: Wear radius compensation
P_	：	Tool No. (1 ~ 99)
R_	：	Compensation value

13.F Feed Speed Specification

Overview

【Functions】:

It is expressed by instruction class F and the following 1 ~ 5 digits, specifying the feed speed.

【Instruction Format】:

F_ ;	Mm/min

【Instructions description】:

(1) Once an F instruction is issued, it remains valid until the next F instruction is issued.

(2) The F instruction is issued in principle in the same block as the cutting feed instruction (G01) or in the block prior to the G01 instruction.

(3) The feed speed can be obtained by the following formula:
F (mm/min) = S×f
S: Spindle rotation speed（min-1）
f: Feed for one spindle rotation（mm/rev）

Program legend

【Example】:

G90 G54 G00 X0 Y0	
G43 H1 Z5.	
M08	
S3500 M03	The spindle rotates forward at a speed of 3500 min-1
G01 X10. Y10. F200	Move to the position X10. Y10 at a speed of 200 mm/min
:	
:	
M30	

14.Auxiliary function

14.1List of M Instructions

Overview

【Definition】：

M instruction is also called auxiliary function. As an auxiliary function of G instruction, it can carry out mechanical control such as program stop, coolant discharge, stop, spindle rotation and stop. It controls the ON/OFF of the machine tool through the last two digits of the M instruction.

M instruction	FUNCTIONS
M00	Program stop
M01	Optional stop
M02	End of program
M03	Forward rotation of spindle
M05	Spindle stop
M06	Automatic tool exchange
M07	Spray coolant start-up
M08	Nozzle coolant start-up
M09	Spray and nozzle coolan t stop
M30	End of program
M98	Call subprogram
M99	Subprogram return

Attention

(1) Only 3 M codes are valid in a program segment. When the limit is exceeded, the system only uses the first 3 codes, while the following M codes are ignored.

(2) Here only listed the system provided by default M code, the specific M code of the

machine tool, please refer to the machine tool manual provided by the machine tool plant.

(3) The T instruction is executed in principle in the block preceding or in the same block as the automatic switching tool instruction (M06).

14.1.1Program stop (M00)

Overview

【Functions】:

Unconditional suspension of automatic processing. Once the M00 in the program is read, the feed of each shaft will be suspended.

【Instruction Format】:

```
M00 ;
```

Instruction description

(1) Press the "Cycle Start" switch, and the automatic operation will be resumed, and the following procedures will continue to be executed.

(2) When executing the M00 command, the automatic operation will stop regardless of whether the "optional stop" switch of the operation panel is ON or OFF, which is different from the M01 command.

14.1.2Optional Stop（M01）

Overview

【Functions】:

When the "optional stop" button is set to ON in the operation panel, if M01 in the program is read in, the feed of each axis will be paused. This instruction is usually executed in the final segment of each process, which is used to check dimensions, remove chips, disassemble workpieces, etc.

【Instruction Format】:

```
M01 ;
```

Instruction description

When the "optional stop" switch of the operation panel is set to OFF, the M01 command will be ignored and the automatic operation will not stop. Pressing the cycle start switch will restart the automatic operation and continue to execute the following procedures.

14.1.3End of program（M02）

【Functions】：

End automatic operation.

If you read M02 in the program, all actions will stop, the NC device will become ready, and the cursor will return to the beginning of the program. This instruction is usually executed in the last program segment of each operation.

【Instruction Format】：

```
M02 ;
```

14.1.4End of program（M30）

Overview

【Functions】：

Stop the automatic operation, restart the program to reset it, and return to the beginning of the main program. Once the M30 in the program is read, all actions will stop, the NC device will become ready, and the cursor will return to the beginning of the program. This instruction is usually executed in the last program segment.

【Instruction Format】：

```
M30 ;
```

Program legend

【Example】：

```
G90 G54 X0 Y0
:                      Machining program
:
M30                    The last block
```

14.1.5Subprogram call, end（M98、M99）

Overview

【Functions】：

M98 specifies the invocation of subprogram.

M99 ends the subprogram and returns to the main program.

【Instruction Format】：

M98 P_ L_ ;	Subprogram call (when instruction category is O)
M98 O_ L_ ;	Subprogram call (when instruction category is O)
M98 H_ L_ ;	Is the same as subprogram call in file
M98 (_) L_ ;	Subprogram call (when called by file name)
M99 ;	End the call of subprogram and reset to main program

P_	:	Subprogram number (4 digits after instruction class letter O)
O_	:	Subprogram number (4 digits after instruction class letter O)
H_	:	Subprogram N label in the same file (the last 4 digits)
(_)	:	Subprogram file name (specify an extension, if any)
L_	:	Number of repetitions (1 when omitted)

Instruction description

(1) In a program, if there are multiple identical processing modes at the same time, only the program of this processing mode is written, which is called subroutine. Compared with subroutines, the original program is called the main program.

(2) M98H can call subroutines in the same file.

(3) The "(…)" immediately following M98 is not a comment statement, but a subroutine name.

(4) Calling a subroutine in a subroutine is the same as calling a subroutine in a main program, and further subroutines can be called in a subroutine. You can call up to 10 layers.

(5) Calling a subroutine in a subroutine is the same as calling a subroutine in a main program, and further subroutines can be called in a subroutine. You can call up to 10 layers.

(6) All O, P subroutines that specify a 4-digit number must have a 4-digit name beginning with the letter O and must have a ". NC" extension.

(7) The set range of repetition times is 1 ~ 1000.

(8) Subroutine M30, according to the set parameters: path-M30 return to the main program, set to OFF, will immediately end the program processing, will not return to the main program. When set to ON, the main program is returned.

Program legend

【Example 1】： The same file subprogram is called

Main program **Legend**

```
G90 G54 G00 XY
G91 G01 X10. F500
M98 H1000
G01 X10.
M98 H2000
G01 X10.
M98 H3000
G01 X10.
M98 H4000
M30
N1000
G01 Y30.
G01 X15.
G03 Y-20. R10.
M99
N2000
G01 Y30.
G03 X20. R10.
G01 Y-30.
M99
N3000
G01 Y30.
G02 X20. R10.
G01 Y-30.
M99
N4000
G03 Y20. R10.
G01 X15.
G01 Y-30.
M99
```

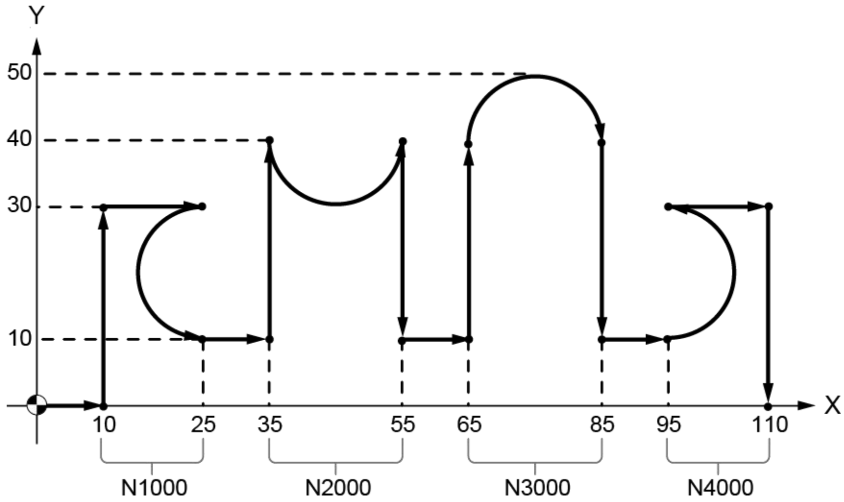


Fig. 14-1 Legend of the same file subprogram call

【Example 2】： Subprogram call

Main program	Subprogram （SUB1.NC）	Subprogram （SUB2.NC）
G90 G54 G00 X Y G91 G01 X10. F500 M98 (SUB1.NC) G01 X10. M98 (SUB2.NC) G01 X10. M98 (SUB1.NC) G01 X10. M98 (SUB2.NC) M30	G01 Y30. G03 X20. R10. G01 Y-30. M99	G01 Y30. G02 X20. R10. G01 Y-30. M99

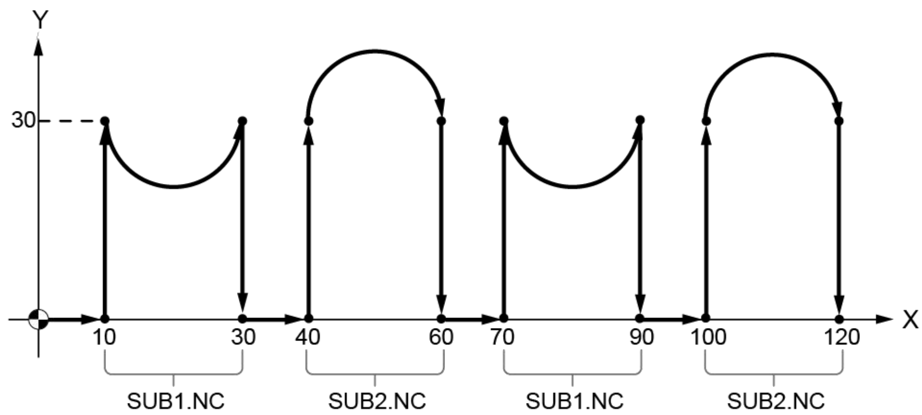


Figure 14-2 Legend of subprogram invocation

【Example3】： Subprogram call nesting

Main program	Subprogram （SUB1.NC）	Subprogram （SUB2.NC）	Subprogram （SUB3.NC）
G90 G54 G00 X Y G91 G01 X5. F500 M98 (SUB1.NC) G01 X5. M98 (SUB1.NC) M30	G01 Y10. X5. M98 (SUB2.NC) G01 X5. M98 (SUB3.NC) G01 X5. M98 (SUB2.NC) G01 X5. Y-10. M99	G01 Y15. G03 X10. R5. G01 Y-15. M99	G01 Y15. G02 X10. R5. G01 Y-15. M99

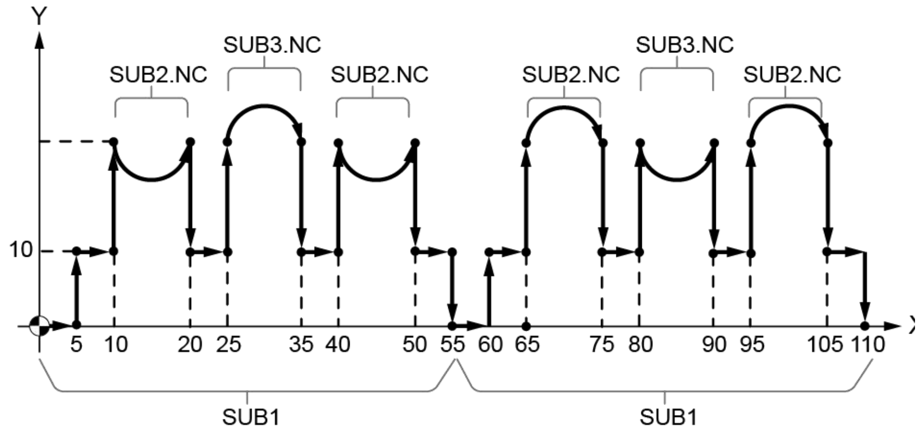


Figure 14-3 Subprogram invocation nesting legend

14.1.6Spindle rotation, stop（M03、 M05）

Overview

【Functions】：

Specifies spindle rotation, stop. Executed with the S instruction.

【Instruction description】：

M03 ;	The spindle rotates forward (clockwise)
M05 ;	Spindle stop

Instruction description

M03, M05 can be executed together with other instructions in the same block. At this time, M03 is unconditionally executed first in the block and M05 is unconditionally executed last in the block.

Program legend

【Example】：

G00 X100. Y100. S2000 M03

The spindle rotates at 2000 rpm and is positioned at X100. Y100.

14.1.7Automatic tool exchange（M06）

Overview

【Functions】：

Use ATC device to exchange tool instructions. Executes at the same time as the T instruction.

Specifically, it includes the following functions：

- Local tion of tool exchange position;
- Stop of spindle rotation;
- Spindle positioning;
- Exchange of tools in spindle and tool holder;
- Switch of gate;

【Instruction description】：

M06 ;

Instruction description

T01 and M06 instructions can be executed in the same block.

T01 M06

Program legend

【Example】：

T01
M06
Return the tool in the spindle to the tool holder, and install the tool with tool number 1 on the spindle.

14.1.8Spray Coolant Start, Nozzle Coolant Start, Stop（M07、M08、M09）

Overview

【Functions】：

Specify the start and stop of coolant.

【Instruction description】：

M07 ;	Spray coolant start-up
M08 ;	Nozzle coolant start-up
M09 ;	Spray and nozzle coolant stop

Program legend

【Example1】：

```
G90 G54 G00 X0 Y0
G43 H1 Z5.
M08                      Nozzle coolant start-up
S3500 M03:

M09                      Nozzle coolant stop
```

【Example2】：

```
G90 G54 G00 X0 Y0
G43 H1 Z5.
M07                      Spray coolant start-up
S3500 M03

M09                      Spray coolant stop
```

14.1.9Spindle orientation, orientation release（M18、M19）

【Functions】：

Implement the shrinkage and release of the spindle.

【Instruction description】：

M19 ;	Axis orientation
M18 ;	Spindle orientation release

14.1.10Rigid tapping, rigid tapping cancellation（M28、M29）

Overview

【Functions】：

The machine tool enters the rigid tapping mode. Under the rigid tapping mode, the Z-axis feed and the spindle speed establish a strict positional relationship, so that the machining of threaded holes can be carried out very conveniently.

【Instruction description】：

M29 ;	Rigid tapping

M28 ;	Rigid tapping cancellation

Program legend

【Example】：

G91 G28 Z0

G90 G54 X Y

M29 S2700 Rigid tapping

G84 X5. Y5. Z-5. R6. P2000 Q1. F1400 L1

M28 Rigid tapping

M30

14.2G10 Parameter Setting and Saving Function

Overview

【Functions】：

Used to set any system setting parameters.

【Instruction description】：

G10 L3 P_ Q_ ;		Set system setting parameters
G10 L30 ;		Save system parameters
P_	：	Macro variable number
Q_	：	Value

Program legend

【Example】：Change the RTCP type to 1 and save it. (Note: The macro variable index corresponding to RTCP type is 41150).

G10L3P41150Q1.
G10L30

14.3Emergency stop rollback function（G150）

Overview

【Functions】：

When the machine tool is in the process of machining, if there is an abnormal alarm, the machine tool suddenly stops on the workpiece, which may cause damage or damage to the workpiece. It is hoped that the system can automatically retreat for a certain distance before alarming, away from the workpiece, and will not cause damage or damage to the workpiece.

【Instruction description】：

G150 P1 X_ Y_ Z_ A_ B_ C_ U_ V _ W_ ;	Turn on rollback
...	NC program for normal machining
G150 P0 ;	Close rollback
M30 ;	
X_ Y_ Z_ A_ B_ C_ U_ V _ W_ ：	Backoff vector

Instruction description

(1) G150 macro program (O0150. NC (673B)) is needed in system MOTION. Or temporary use, you need to put G150 macro program in user MOTION.

(2) System-Parameters-Change the setting in the path: the emergency stop fallback function is effective = ON.

Program legend

【Example】：

G150 P1 X1. Y1. Z1. G91 G01 X20. F1000 X-20. G43.4 H100 X1. Y1. G150 P0	Execute the left NC under MDI. When G43. 4H100 gives an alarm, the Y axis and Z axis will automatically roll back 1mm.

Attention

(1) When an error of power failure occurs, the shaft will not automatically retreat. Because the strong current drops too fast, the shaft does not retreat in a hurry. Temporary scheme is to modify the power-down delay of PLC. (This method has certain risks and is not recommended as a formal solution.)

(2) There is no limit to the back-off amount of the shaft, and it is found that a large number

can be input during the test, which leads to the risk of collision, and there is no soft limit protection.

14.4Disable handwheel analog switching function（G150.1、G151.1）

Overview

【Functions】：

Start or stop handwheel simulation quickly when changing tools, so as to control the handwheel simulation function when using the fast movement command G00.

【Instruction description】：

G151.1 ;	Turn on the state of prohibiting handwheel simulation switching
G150.1 ;	Turn off and prohibit switching handwheel simulation state

Instruction description

(1) In G151.1 mode, it is forbidden to switch handwheel simulation, but the status indicator will change accordingly;

(2) G151.1/G150.1 comes with CMD [STOP] function, which can effectively block hand rotation regression function;

(3) When M30 or reset, the state will be automatically restored to G150.1 state.

Program legend

【Example】：Tapping MOTION

```
......
G90 (G90 MODE)
G00 Y#30962 (MOVE R POINT)
G151.1 (Handwheel analog switching is prohibited)
#30040=1 (OPEN RATE CONTROL)
#30039=1 (OPEN DELAY STOP PRO)
#30038=1 (OPEN KEEP)
IF[#34609 EQ 1] M98(MOTIONSINGLE.NC)
IF[[#52 EQ 0] AND [#34610 EQ 1]] G65 P9284 A1
IF[[#52 NE 0] AND [#34610 EQ 1]] G65 P9384 A1
IF[#34610 EQ 0] G65 P9184 A1
#70=#30962-#67 (TEMP VAR R-Q)
#71=ABS[#30964-#30962] (TEMP VAR |Y-R|)
```

```
IF[#30965 EQ 0] GOTO 1830
WHILE[[ABS[#71]] GT [ABS[#30965]]] DO1
G01 Y#70 (MOVE NEXT POINT)
IF[#34604 EQ 1] Y#30962 (MOVE R POINT)
IF[#34604 NE 1] Y[#70+#66] (RETURN #D)
#70=#70-#67 (NEXT LOOP VALUE)
#71=#71-ABS[#30965] (NEXT LOOP VALUE)
END1
N1830
G01 Y#30964 (MOVE Y POINT)
IF[#30967 GT 1] G04 P#30967 (SLEEP)
IF[[#35605 EQ 94] AND [#30968 GT #51]] S[#30968] F[#30968*#35997]
IF[[#35605 EQ 95] AND [#30968 GT #51]] S[#30968]
G01 Y#30962 (MOVE R POINT)
G150.1 (Cancel the prohibition of handwheel simulation switching state)
#30040=0 (CLOSE RATE CONTROL)
#30039=0 (CLOSE DELAY STOP PRO)
#30038=0 (CLOSE KEEP)
IF[[#52 EQ 0] AND [#34610 EQ 1]] G65 P9284 A-1
IF[[#52 NE 0] AND [#34610 EQ 1]] G65 P9384 A-1
IF[#34610 EQ 0] G65 P9184 A-1
IF[#35610 EQ 98] G00 Y#30963 (GOTO Y AXIS START POINT)
IF[#87 EQ 91] G91 (G91 MODE)
IF[[#35605 EQ 94] AND [#30968 GT #51]] S[#51] F[#51*#35997]
IF[[#35605 EQ 95] AND [#30968 GT #51]] S[#51]
GOTO 3000
......
```

15.High Speed Contour Control Function（GACC）

15.1Summary

Overview

【Functions】：

The high-speed contour control function eliminates the machining error caused by acceleration and deceleration after interpolation by reading multiple blocks in advance, and considers the factors such as shape and speed changes, mechanical allowable acceleration and so on, thus realizing smoother acceleration and deceleration.

【Instruction description】：

G05 P10000 / G05 P20000 ;	Enable high-speed contour control
G05 P0 ;	Disable high-speed contour control

Instruction description

(1) The high-speed contour control function corresponds to three straight axes X, Y and Z and two rotation axes.

(2) By executing the G05P10000 or G05P20000 command, the high-speed contour control function is set to ON, and the high-speed contour control function continues to be effective until the high-speed contour control function is cancelled.

(3) When the high-speed contour control function is implemented, the effective range of feed ratio is only 0 ~ 100%. If it exceeds the range of 100%, it will be processed according to the magnification of 100%.

(4) Adding instructions G05P10000/G05P20000 and G05P0 to NC can set ON and OFF of high-speed contour control function. The relationship between it and the parameter "Default GACC Mode One" in "System-Parameters-Path" is shown in the following table.

Table 15-1 Relationship between System Settings and G05P Instructions

System setting	Program instruction	Is the high-speed contour control function effective

默认 GACC1 模式 ON	无	High-speed contour control function ON
默认 GACC1 模式 OFF	无	High-speed contour control function ON
默认 GACC1 模式 ON	G05P10000	High-speed contour control function ON
默认 GACC1 模式 ON	G05P20000	High-speed contour control function ON
默认 GACC1 模式 ON	G05P0	High-speed contour control function ON
默认 GACC1 模式 OFF	G05P10000	High-speed contour control function ON
默认 GACC1 模式 OFF	G05P20000	High-speed contour control function ON
默认 GACC1 模式 OFF	G05P0	High-speed contour control function ON

(5) After M30 and M02 are executed, the setting of high-speed contour control function is restored to the system setting after the system is reset the state of ([Set-Common-Default GACC1 Mode]).

Program legend

【Example】：

O0001
G90G10P1X***Y***Z***
T01
M06
G40G49
G90G54
G00G43X0Y0Z50.0H01M01
M08
G05P10000（或 G05P20000） High Speed Contour Control Function ON
S35000M03
G00X-15. Y0.
Z3.
M98P101000 Subprogram call (machining program)
G05P0 High-speed contour control function OFF
G00Z50.0
G49
M05

M09
M01
G91G28Z0
M30

15.2Parameter setting

【Functions】：

Fast forward instruction G00 and feed instruction (G01, G02, G03) in the high-speed contour control function refer to the high-speed contour control function parameters for action.

High-speed contour control function parameters include G00 parameters and feed instruction parameters. The parameters of G00 can only be specified by setting screen, and the parameters of feed instruction can be set in NC program or in [System-Parameters-Path].

Attention

(1) The parameters set on the setting screen are modal parameters. The settings in the NC program will be restored to the parameters set in the setting screen after the NC program end instruction is implemented or the system is reset.

(2) When the power is started, it defaults to the value set in [System-Parameters-Path].

15.2.1Setting in system parameters

Overview

【Functions】：

In order to make GACC parameter switching more convenient, several sets of GACC parameters can be set at one time before machining, and these GACC parameters can be freely switched and used for machining during machining. The interface adds several sets of GACC parameters setting and management, and provides G05.1 instructions to switch between different GACC parameters and modes in machining.

Description

(1) A total of 10 sets of GACC parameters are provided in the interface, and each set of GACC parameters includes the following 5 GACC parameters:

a) Maximum allowable error

b) Maximum acceleration

c) Corner velocity

d) Interpolation filtering time

e) Minimum acceleration time

(2) 10-Level GACC parameter macro variable mapping：

a) # 36450: Default processing condition index (0 ~ 10)

b) #36451~#36455：The first set of processing conditions

c) #36456~#36460：The second set of processing conditions

d) ……

e) #36496~#36500：The 10th set of processing conditions

(3) Enter the interface system module > > configuration > > function parameters to manage these 10 sets of GACC parameters and set which set of GACC parameters are currently used by default.

15.2.2Setting in NC Program

Overview

【Instruction description】：

G05 A_ E_ T_ ;

A_	：	Maximum acceleration (unit: g)
E_	：	Maximum permissible error (unit: millimeters)
T_	：	Minimum acceleration time (unit: seconds)

Instruction description

①. Maximum acceleration, which is described as follows：

- Set the maximum acceleration (MaxAl) using the above command. This maximum acceleration is a composite acceleration, and each axis cannot be set separately. The unit is g (g=9. 8 m/sec2).

- Effective range of maximum acceleration: 0.001 g ≤ xxx ≤ 1g
- No matter whether [decimal point automatic judgment] is ON or OFF, the input without decimal point judgment is adopted.

②. Maximum allowable error, which is described as follows：

- E Specifies the unit of error in millimeters.
- Effective range of MaxError: 0 mm < xxx 1.0 mm.
- No matter whether [decimal point automatic judgment] is ON or OFF, the input without decimal point judgment is adopted.

③. Minimum acceleration time, which is explained as follows：

- Set the minimum acceleration time (MinT) by using the above instructions. The unit is seconds.
- Effective range of Minimum Acceleration Time (MinT): 0.002 sec ≤ xxx ≤ 1.0 sec.
- No matter whether [decimal point automatic judgment] is ON or OFF, the input without decimal point judgment is adopted.

Attention

Be sure to specify it in a separate block.

Program legend

【Example1】： Maximum acceleration

N0 G05A0.1ExxxTxxx MaxAl=0.1g=0.98 m/sec2

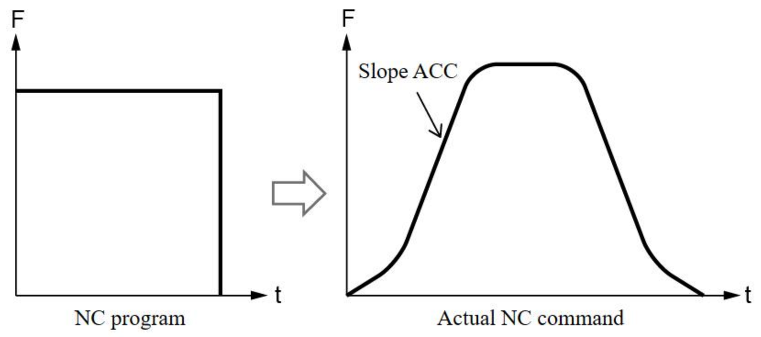


Figure 15-1 Maximum Acceleration (|Acc|≦MaxAl)

【Example2】： maximum allowable error

N0 G05AxxxE0.05Txxx MaxError=0.05mm=50µm

N1 G01 G91 X1.0F3000

N2 Y1.0

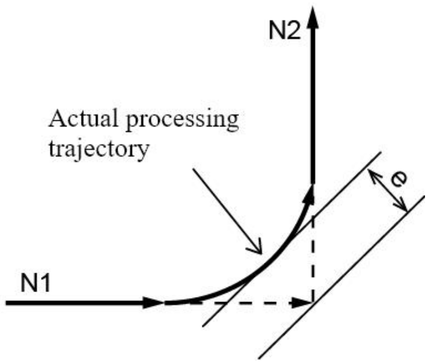


Figure 15-2 Maximum Acceleration (e ≤ MaxError)

【Example3】： Figure 15-2 Maximum Acceleration (e ≤ MaxError)

N0 G05AxxxExxxT0.02 MinT=0.02 sec=20 msec

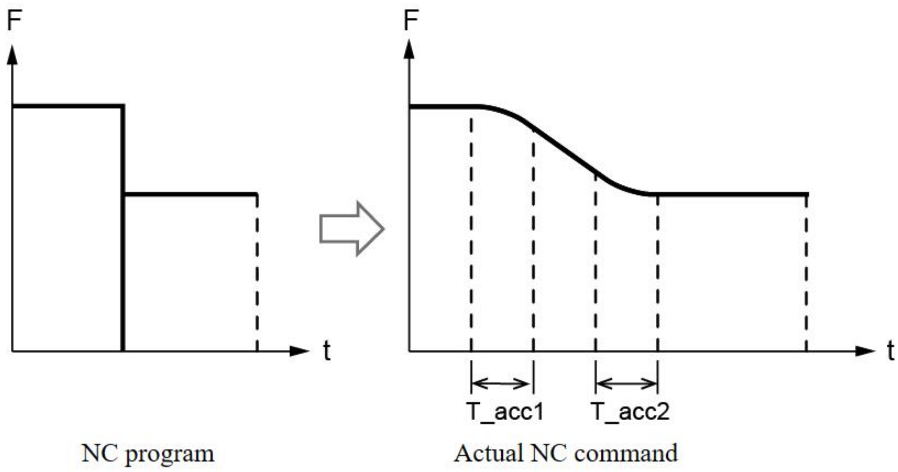


Figure 15- 3 Minimum Acceleration Time(T_acc1=MinT T_acc2=MinT)

Attention

Rapid changes in acceleration can lead to vibration, so it is important to set an appropriate minimum acceleration time (MinT).

15.3NC instructions that can be realized in high-speed contour control function

Overview

NC instructions in the high-speed contour control function can be divided into two categories：

- Direct executable NC instruction in high speed contour control function
- NC command that can be executed after automatically temporarily canceling the high-speed contour control function.

In the high-speed contour control function, Directly executable NC instructions	After automatically temporarily canceling the high-speed contour control function, Executable NC instruction
G00, G01, G02, G03, G05 G17, G18, G19 G20,G21, G40, G41, G42 G43, G44, G49 G90, G91 G9, G61, G64 G54-G59, G154-G159 G254-G259, G354-G359, G454-G459, G554-G559, G654-G659, G754-G759, G854-G859, G954-G959 D, F, H, N， O， S, T, G04,G51,G50 G51.1,G50.1,G68,G69 G52,G92	G00 G10 G27, G28, G29, G53, G65, M00, M01, M02, M03 ,M05, M06, M08, M09, M30, M98, M99 Fixed cycle instruction

Attention

When G00 is set to OFF in [system-parameter-path-GACC supports G00 interpolation mode], it becomes an automatic temporary cancellation instruction; When set to ON, it becomes a directly executable instruction.

Program legend

The difference between the execution state of the command that can be executed directly in the high-speed contour control function and the NC command that can be executed after automatically temporarily canceling the high-speed contour control function is shown in Figure 15-4 below.

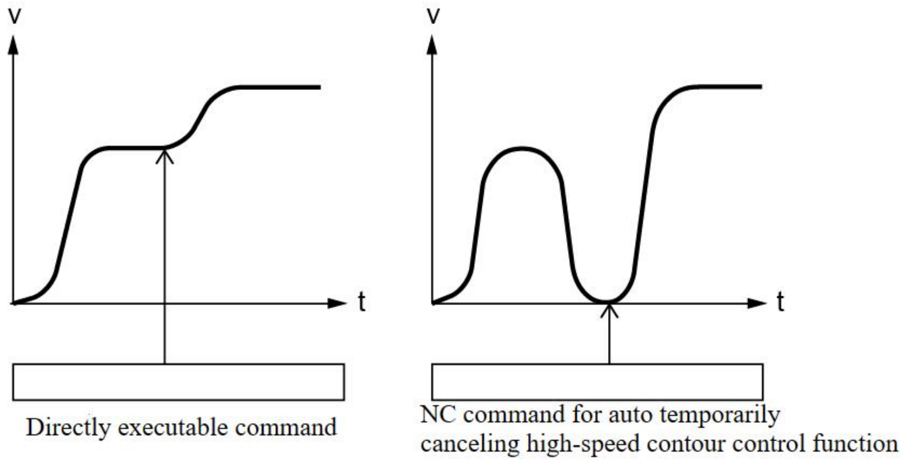


Figure 15-4 NC Command in High Speed Contour Control Function

15.4High speed and high precision parameter selection（G05.1）

Overview

【Functions】：

Switch between different GACC parameters and modes in machining. The high- speed profile control function eliminates the machining error caused by acceleration and deceleration after interpolation by reading multiple blocks in advance, and considers the shape and speed changes. Speed and other factors, thus realizing smoother acceleration and deceleration.

【Instruction description】：

G05.1 Q_ R_ ;

Q_	：	GACC Level
R_	：	Processing parameter Level

Instruction description

(1) Q Value range: 0/1/2/3. If the value exceeds the range, 3309 error will be reported.

a) When it is 1, GACC7 is turned on and 5 GACC processing parameters in the processing conditions specified by R are used;

b) When 2, use normal GACC2 and use 5 GACCs in the processing conditions specified by processing parameters;

c) When 3, turn on GACC1 or GACC3 and use 5 of the processing conditions specified by R GACC processing parameters;

d) When 0, it works the same as G05P0. Turn off GACC1/2 and use GACC0. At the same time, the corresponding five GACC machining parameters are restored to the machining condition items corresponding to the default GACC machining condition index. (If the default machining condition index number is 0, the machining parameters set by the system are used; if the setting item G05P0 restores the default parameter setting to ON, the machining parameters set by the system are used).

(2) Rvalue range: 0 ~ 10. If the value exceeds the range, 3309 error will be reported.

a) When R value is not specified, use a set of machining conditions corresponding to the set default value # 36450 (if the default machining condition index number is 0, use the machining parameters set by the system);

b) When R is 0, the processing parameters set by the system are used.

(3) G05.1Q_R_ is prohibited in the G43.4/G43.6/G43.1 five-axis mode. Permits execution in the following modes：

- GACC1/GACC2/GACC0 mode;
- G66 cycle, fixed cycle mode;
- G41/G42/G43/G44 corrected mode;
- Mirror/rotation/scaling mode;

(4) Switching and Recovery of GACC Modes：

a) G05.1Q1R_ turns on GACC7, which has the same function as G05P20000 instruction;

b) G05.1Q2R_ turns on GACC2, which has the same function as G05P20000 instruction;

c) G05.1Q3R_ turns on GACC1, which has the same command function as G05P10000;

d) G05.1Q0R_ turns off GACC, which has the same function as G05P0 instruction.

e) After reset, the GACC parameters are restored to the GACC parameters corresponding to the default Level in the configuration. When the default Level is set to 0, the GACC parameter set in "System-Parameters-Path" is restored; The GACC mode is restored to the "default GACC mode" set in "System-Parameters-Path".

(5) G05.1Q1R_ The GACC processing conditions used will affect the following GACC calculations：

- GACC1(G05P10000，G43.1)；
- GACC2 (G05P20000)；
- GACC5 (G43.4，G43.6)。

(6) Switch control, G05.1 function can be opened in the following two ways：

- Modify G05.1 function parameter Level in the system interface;Add instruction G05.1Q_R_ in NC;
- Add instruction G05.1Q_R_ in NC;

(7) G05.1 Functional parameter Level：

When the default Level is set to 0, "feed allowable error", "feed acceleration", "corner reference speed", "filter time after interpolation" and "feed acceleration time" in "system-parameter-path" are used by default at the beginning of processing.

When the default Level is set to 1 ~ 10, the set of GACC parameters in the configuration settings is used by default at the beginning of processing count.

16.Inclined surface machining

16.1Inclined surface machining function（G68.2、G69.2）

Overview

After the workpiece is tilted, the datum plane of the previous workpiece changes correspondingly, which is called the tilted plane. Programmed in the coordinate system fixed to the sloping surface (called "characteristic coordinate system"), shapes such as machined holes or pits in the sloping surface can be sloped. Support inclined surface machining and inclined surface rotating machining.

16.1.1Inclined surface machining

Overview

【Functions】：

Establish a "characteristic coordinate system" on the inclined plane, and program with this plane coordinate system.

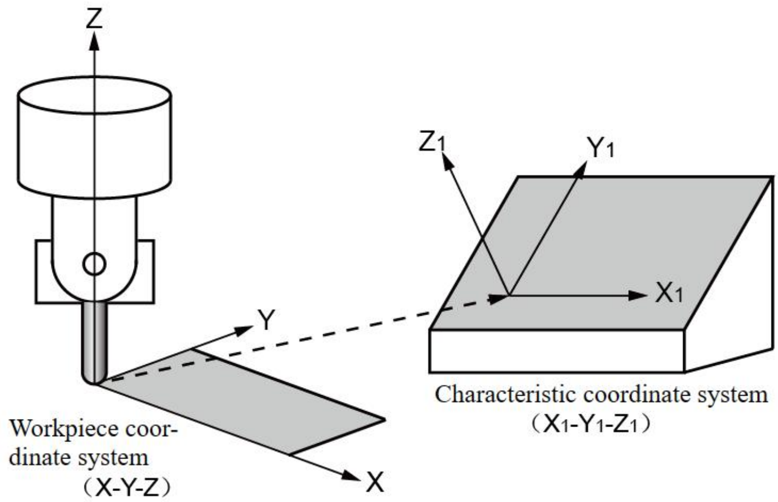


Figure 16-1 G68.2 instruction establishes characteristic coordinate system

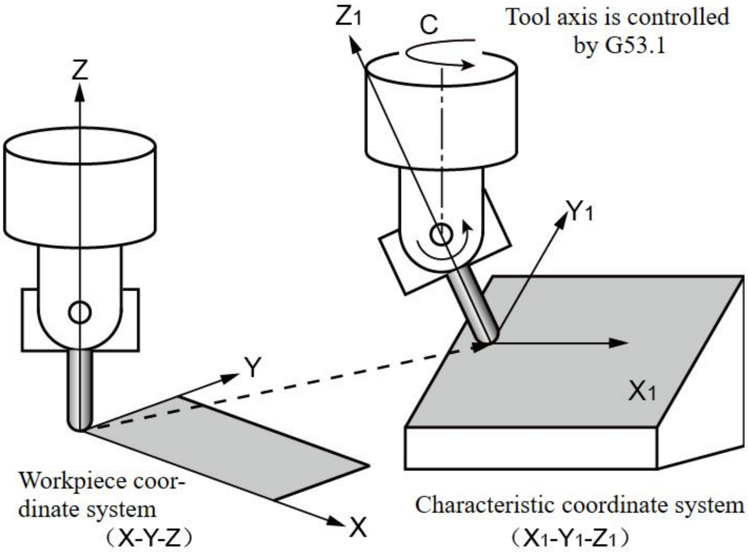


Figure 16-2 The G53.1 instruction makes the Z axis of the tool perpendicular to the XY plane of the characteristic coordinate system

【Instruction description】：

G68.2 X x0 Y y0 Z z0 Iα Jβ Kγ ;		Characteristic coordinate system setting
G53.1 ;		Tool axis direction control
G69.2 ;		Unset the characteristic coordinates
X_ Y_ Z_	：	The origin of the characteristic coordinate system
I_ J_ K_	：	Euler Angle that determines the orientation of the characteristic coordinate system

Instruction description

(1) The offset between the origin of the characteristic coordinate system and the origin of the current coordinate system is x0y0z0. If the input XYZ is omitted, the offset between the origin of the characteristic coordinate system of the corresponding axis and the origin of the current coordinate system is 0.

(2) The Euler angle in the direction of the characteristic coordinate system is α β γ. If the input IJK is omitted, the Euler angle corresponding to the direction of the characteristic coordinate system is 0.

(3) Follow the Z-X-Z axis rotation rules:

- Taking Z axis of characteristic coordinate system as rotation axis, rotating α angle;
- Taking X axis of characteristic coordinate system as rotation axis, rotating β angle;
- Taking Z axis of characteristic coordinate system as rotation axis, rotate γ angle.

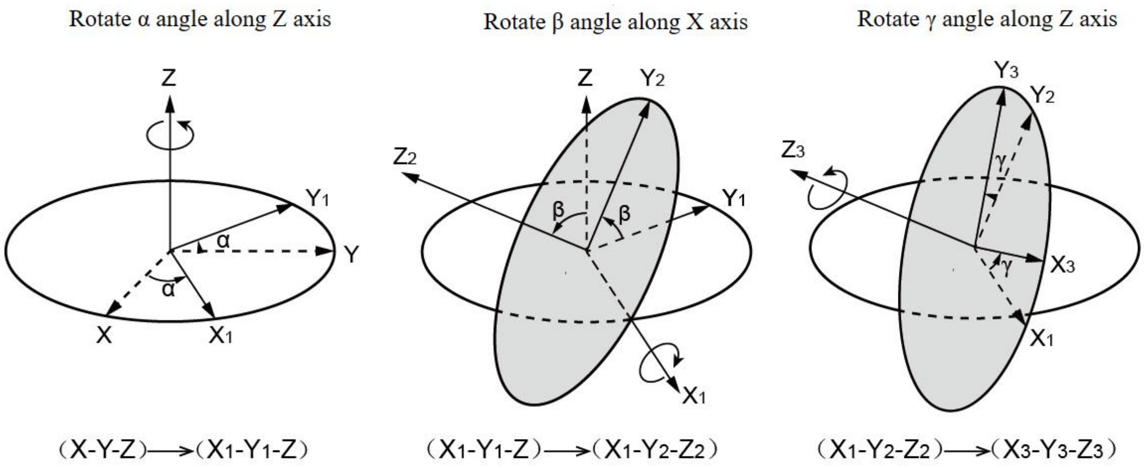


Figure 16-3 Coordinate Transformation Using Euler Angle

Attention

(1) G68.2 Peer only allows the following directives: XYZIJK, G17/G18/G19, G90/G91, F, comments.

(2) In G68.2 mode, the following instructions cannot be specified: G12 polar coordinates, tool change instruction T, G43.4/G43.6 (not supported for the time being), G53 positioning, G52/G92, and the movement instructions of the first rotating shaft and the second rotating shaft set in the five-axis machine tool structure.

(3) G68.2 cannot be specified in the following modes: G02/G03, G12 polar coordinates, G41/G42, G43/G44,G66, G81 and other fixed cyclic modes, G68/G51/G51.1 rotation/mirror/zoom.

(4) When G54 ~ G59, G54.1 coordinate system is switched under G68.2 mode and G10 coordinate offset is set, whether to report error is decided according to the setting item. When it is set to ON, error will be reported, and when it is set to OFF, the origin of the workpiece in the characteristic coordinate system will be shifted again relative to the new coordinate system.

(5) The G69.2 cancel instruction cannot be specified in the following modes: G41/G42, G43.4/G43.6, G66, G81, etc. Fixed cycle modes, G68/G51/G51.1 rotation/mirror/scale.

(6) G69.2 Peers are not allowed to have any instructions, and peers will make errors when they have mobile instructions.

(7) In the tapping action of G68.2 mode, the determination of spindle following relationship and proportion needs to use the tool axis direction vector to calculate, and needs to modify the MOTION extension instruction.

16.1.2Rotary machining of inclined surface

Overview

【Functions】:

After G68.2 determines the basic tilt plane (that is, in G68.2 mode), rotate the tilt plane by C angle to the new tilt plane, and rotate the attitude to the new tilt plan perpendicular to the tool vector (at this time, the new tilt plane transformation matrix and matrix offset will be calculated), and then the machining NC will be programmed on the tilt plane after rotation.

【Instruction description】：

G68.2 X x0 Y y0 Z z0 Iα Jβ Kγ ;		Characteristic coordinate system setting
G53.1 ;		ool axis direction control
C_ ;		Recalculate the inclined surface and
G69.2 ;		Cancel the setting of characteristic
X_ Y_ Z_	：	Origin of Characteristic Coordinate System
I_ J_ K_	：	Euler Angle Determining the Direction of Characteristic Coordinate System

Instruction description

(1) When the rotary machining setting item is turned on, the turntable is not rotated in line G53.1;

(2) C axis plays the role of G53.1, and the current transformation matrix needs to be changed at the same time;

(3) C axis can't walk with any instruction, similar to G53.1;

(4) Multiple C-axis commands are allowed in G68.2 mode.

Program legend

【Example】：Take AC structure as an example：（multiple rotation machining of the same inclined surface）

G01A90C90（AC angle is the rotation axis angle corresponding to vector IJK）

G68.2X_Y_Z_I_J_K_

G53.1

X_Y_Z

...

G69.2

G68.2X_Y_Z_I_J_K_

G53.1

C_ (this line recalculates the tilt surface and rotates the current attitude go to the new bevel plane perpendicular to the tool vector)

X_Y_Z

...

G69.2

...

M30

Attention

Whether the G68.2 inclined surface rotary machining function is turned on or not is controlled by the setting item "G68.2 rotary machining function is effective", which is effective by default.

16.2G68.3

Overview

【Functions】:

The G68.3 instruction automatically specifies the characteristic coordinate system with the tool axis as the Z + direction, as shown in Figure 16-4. Using this characteristic coordinate system, the process of machining holes and cavities on the inclined surface relative to the tool coordinate system will become simple. The cancel command of G68.3 is also G69.2.

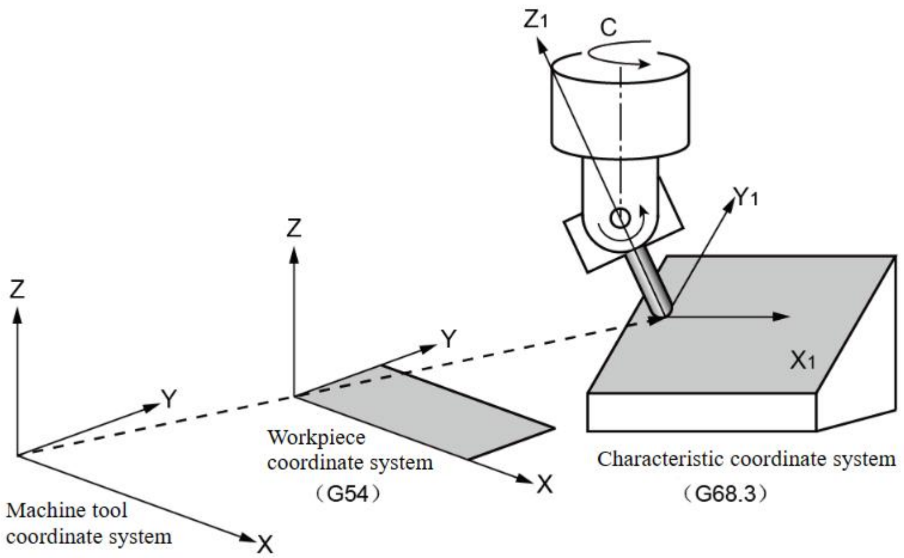


Figure 16-4 G68.3 Specified Characteristic Coordinate System

【Instruction description】:

G68.3 X x0 Y y0 Z z0 Rα ;		Characteristic coordinate system setting
G69.2 ;		Disable the characteristic coordinate system
X_ Y_ Z_	:	The origin of the characteristic coordinate system in absolute coordinates
R_	:	The Angle of rotation centered on the Z-axis of the characteristic coordinate system

Instruction description

(1) X，Y，Z：When omitted, the current position is the origin of the characteristic coordinate system.

(2) R：The clockwise rotation is positive from the Z axis direction of the characteristic coordinate system. The range is [0.0, 360], and it is zero when omitted.

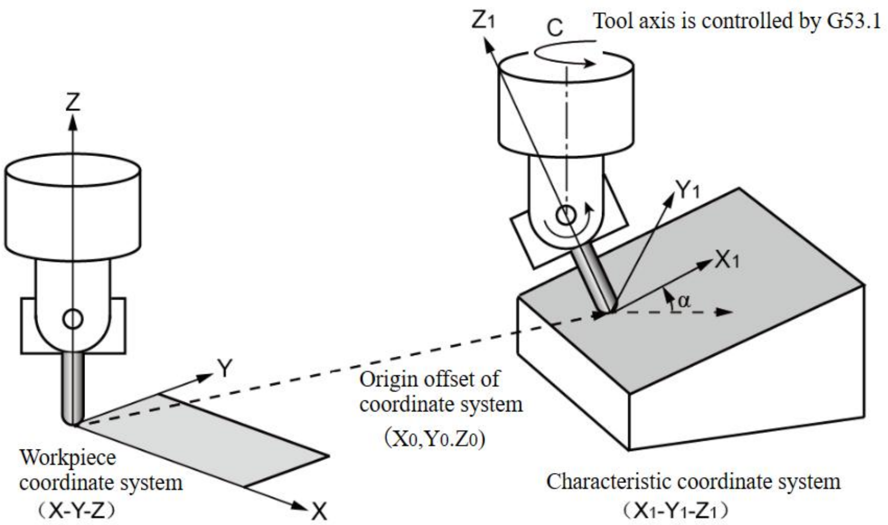


Figure 16-5 G68.3 instruction

Program legend

【Example1】：Taking BC double swing head as an example, the parameter information of tool and rotating shaft is as follows：

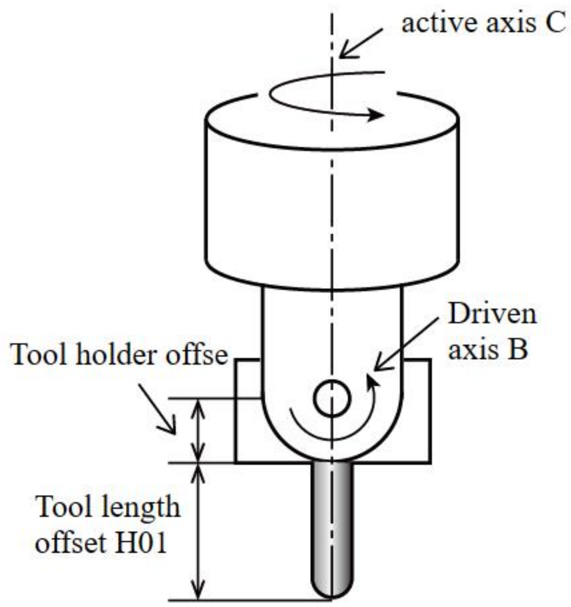


Figure 16-6 Parameter information of BC double swing head

【Example2】：Taking BC double swing head as an example, explain the use process of G68.3 instruction.

Example program O0100. NC：

N1 G55	
N2 G90 G01 X0Y0Z50.0F1000	
N3 G43 H01 X0 Y0 Z0	Tool length compensation is carried out in workpiece coordinate system. The front point of the tool moves to the origin of the workpiece coordinate system.
N4 B-45.0	Tilt the tool
N5 G68.3 …	Switch the tool axis to Z axis, and set the characteristic coordinate system with the front end of the tool as the origin.
N6 G69.2 …	Cancel the characteristic coordinate system and switch back to the workpiece coordinate system.

The sample program O0100. NC executes as follows：

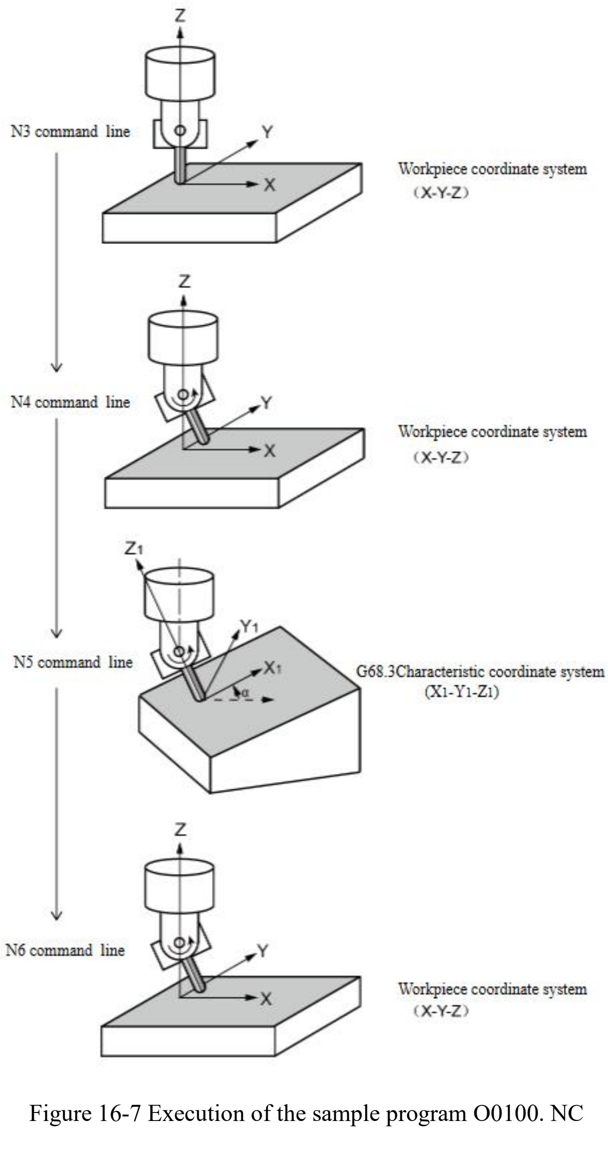


Figure 16-7 Execution of the sample program O0100. NC

16.3Tool Axis Direction Control（G53.1）

Overview

【Functions】：

The direction of tool axis is controlled in the machining of G68.2 mode inclined surface.

【Instruction description】：

G53.1 ;	Tool axis direction control

Instruction description

(1) The G53.1 instruction can only be executed in G68.2 mode.

(2) When G53.1 instruction controls the tool axis, only the rotating axis moves, and the moving axis X/Y/Z does not move, so that the tool axis direction becomes the Z + direction of the characteristic coordinate system.

(3) In general, there are two solutions for G53.1 tool axis orientation, and peer P instruction is used to select which solution is used for tool axis orientation. P0/1: Adopt the positive solution value of the second rotating axis, and P2: Adopt the negative solution value of the second rotating axis. When the P instruction has no input, P0/1 is used by default, as shown in Figure 16-8.

(4) When the G53.1 tool is positioned in the direction of axis, the rotating axis of the machine tool may move, so it is necessary to lift the tool to a safe height before G53.1 action to avoid workpiece interference.

(5) G53.1 cannot be specified in the following modes: G02/G03, G12 polar coordinates, G41/G42,G43.4/G43.6, G66, G81 and other fixed cyclic modes, G68/G51/G51.1 rotation/mirror/zoom.

Program legend

【Example】： Action in G53.1

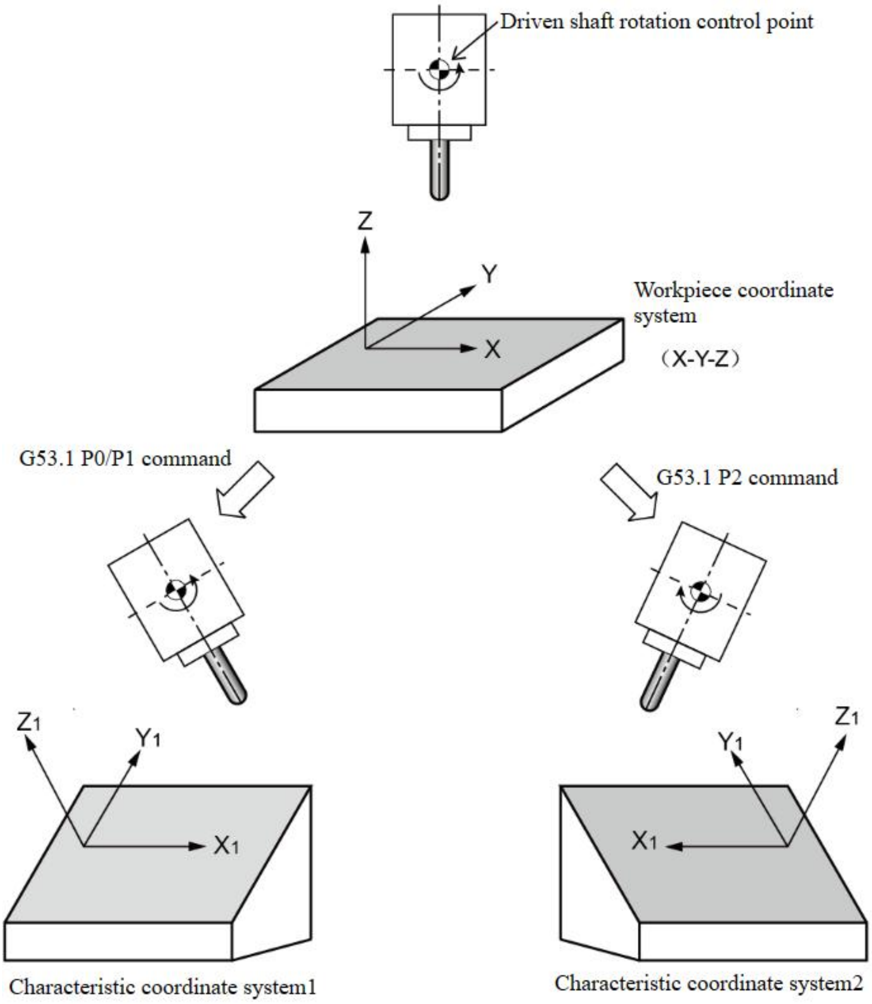


Figure 16-8 Motion Direction Selection

17.Five-axis machining

Overview

In addition to the five-axis machine tool has X, Y, Z three directions of linearaxes, but also has two rotary axes. According to the installation position of the two rotating axes, five-axis machine tools can be divided into double turntable type, double swing head type and hybrid (one swing head and one turntable) type.

17.1Five-axis fixed-axis machining（G43.1）

Overview

【Functions】：

Open the five-axis (3+2-axis) fixed-axis machining function. Used in five-axis indexing axis machining mode, the five-axis coordinates are converted into three-dimensional XYZ coordinates, and machined with GACC7.

【Instruction description】：

G43.1 H_ ;	
G49 ;	Close RTCP and cancel the tool length offset

Instruction description

(1) G43.1 must be used with the tool number;

(2) G43.1 instruction can not be followed by any M instruction, otherwise the system will report an error prompt;

(3) A NC can not have more than G43.1, otherwise the system will report "RTCP mode, specified unsupported instructions" error;

(4) G43.1 and G49 instructions should be used strictly together (when the next G43.1 instruction is opened, it needs to be closed with G49 instruction before it can be opened).

Program legend

【Example1】：

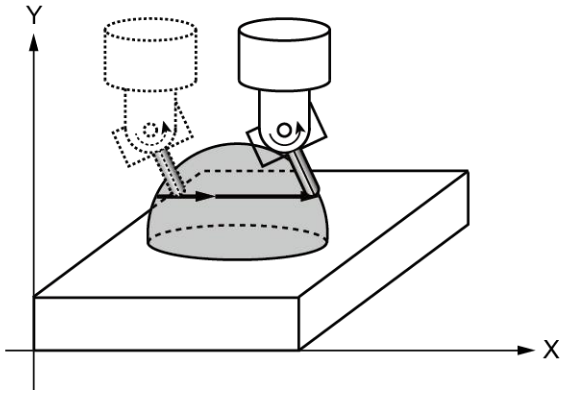


Fig. 17-1 Five-axis indexing machining bevel

【Example2】The standard usage format is as follows:

```
G91 G28 Z0.0
T02 M6
G00 G90 G54
G05A0.1E0.002T0.06
M3 S5500
M8
G43.1 G0 X-66.003 Y-32.6458 Z-48.3542 A54.73561 C315. H02
G0 X-61.53 Y-28.1728 Z-52.8272
G1 G90 X-59.798 Y-26.4408 Z-54.5592 F820.
G1 X-54.899 Y-28.8903 Z-52.1097
G1 X-54.899 Y-7.7805 Z-31.
…
…
…
G1 X-40.202 Y-40.202 Z-48.7246
G1 X-61.53 Y-28.1728 Z-52.8272
G0 X-66.003 Y-32.6458 Z-48.3542
G49
M5 M9
G91 G28 Z0.0
M30
```

17.2Five-axis linkage machining（G43.4）

Overview

【Functions】：

It is also commonly called tip following function.

【Instruction description】：

G43.4 H_ ;

G49 ;	Close RTCP and cancel the tool length offset

Instruction description

(1) G43.4 Hxx must be used in conjunction with tool number

(2) Five-axis linkage does not support arc command function;

(3) G43.4 can't have any M instruction behind it, otherwise the system will report an error prompt;

(4) A NC can not have more than G43.4 appear, otherwise the system will report RTCP mode, specified unsupported instructions;

(5) G43.4 and G49 instructions should be used strictly together (when the next G43.4 instruction is opened, it needs to be closed with G49 instruction before it can be opened).

Program legend

【Example1】： The hemisphere is processed by five-axis linkage machining

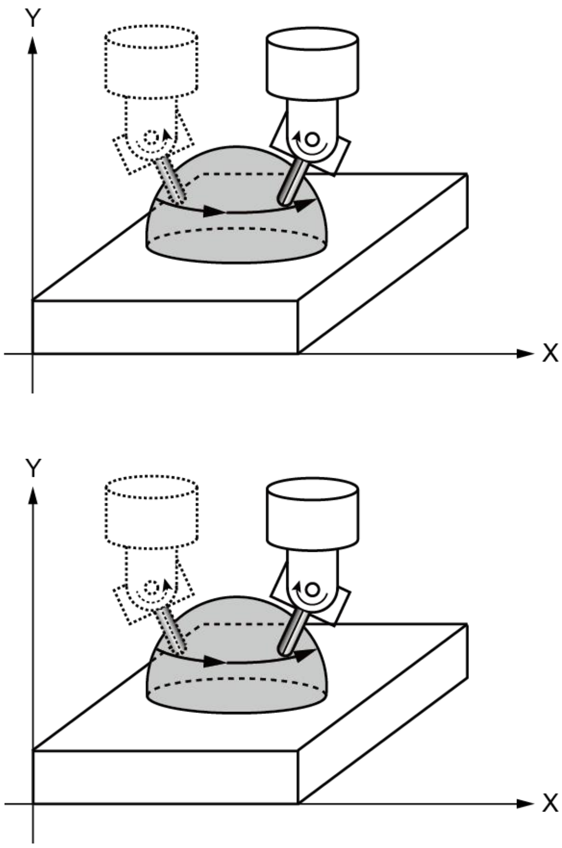


Fig. 17-2 Five-axis linkage machining

【Example2】The standard usage format is as follows：

```
G91 G28 Z0.0
T02 M6
G00 G90 G54
G05A0.1E0.002T0.06
M3 S1800
M8
G43.4 G0 X-2.9999 Y0.0 Z82. A-.00009 C270. H02
G0 X-3. Z2. A-.00009 C270.
G1 G90 X-2.9695 Y0.0 Z1.5731 A-.00009 C270. F1250.
G1 X.0756 Y-.0313 Z-1.0001 A-.1875 C247.5
G1 X.1157 Y-.1157 Z-1.0005 A-.375 C225.
G1 X.0939 Y-.2268 Z-1.0012 A-.5625 C202.5
G1 X0.0 Y-.3272 Z-1.0021 A-.75 C180.
G1 X-.25 Y-.3741 Z-1.004 A-1.03125 C146.25
……
G0 X31.008 A-90. C270.
G49
M5 M9
G91 G28 Z0.0
M30
```

17.3Five-axis tool radius compensation（G40.1、G41.1、G42.1）

Overview

【Functions】：

In five-axis machining, the direction of the tool vector can be calculated according to the position of the rotation axis, and the machining trajectory can be compensated by the three-dimensional tool radius in the plane perpendicular to the tool direction vector (compensation plane).

【Instruction description】：

G41.1 D_ ;	Five-axis tool radius compensation (left radius compensation)
G42.1 D_ ;	Five-axis tool radius compensation (right radius compensation)
G40.1 ;	Disable the five-axis tool radius compensation
D_ ：	Tool radius compensation value code

Instruction description

It is mainly suitable for machining some five-axis contour models, but can not be used for compensation of five-axis machining on 3D curved surfaces.

Program legend

【Example1】：

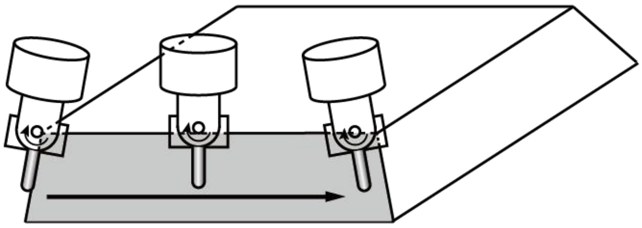


Fig. 17-3 Five-axis tool radius compensation

【Example2】The standard usage format is as follows：

```
G91 G28 Z0.0
G40 G17 G49 G01 G90 G54
G00 A0 C0
T02 M6
M3 S22000
M8
G43.4 G41.1 D02  G00 X-38.5 Y80.7022  Z18.1  A-14.34 C0.0 H02
Y78.2467 Z8.4948
G1 X-38.4961 Y78.1742 Z8.2111 F1000.
X-38.4844  Y78.1017  Z7.9276
…
…
…
X-38.4649  Y78.0293  Z7.6445
X-38.4377  Y77.9571  Z7.362
X-38.4027  Y77.8851  Z7.0803
G40.1 X-38.36  Y77.8134  Z6.7997
G49
M30
```

Attention

Five-axis tool radius compensation is a three-dimensional tool radius compensation in space, and the compensation value cannot be set too large.

18.Macro function

18.1User-specific macro program specification

Overview

User macro program is a programming method similar to high-Level language, which allows users to use variables, arithmetic and logic operation instructions and condition transfer, which makes it more convenient to compile the same machining program than traditional methods, and can also create universal programs such as groove machining and independent fixed cycles more simply.

In addition, like subroutines, the machining program can call user-specific macro programs through simple commands.

【Example】：

Machining program	User-specific macros
O0001	O9010
…	#1=#18/2
…	G01 G42 X#1 Y#1 F300
…	G02 X#1 Y-#1 R#1
G65 P9010 R50.0 L2	…
…	…
…	…
…	…
M30	M99

18.1.1Variable

Overview

Add the specified number after "#" to correspond to different types of variables. Variable numbers can be specified directly by numbers or by expressions. When you specify an expression, you need to put square brackets on the expression.

【Example1】：#3

【Example2】：#[#2+#1-12]

The state where the variable value is not defined is called [null], at which time only reading is supported, not writing.

【Variable type】： According to the variable number, variables can be divided into the following categories：

Table 18-1 Variable types

Number	Variable type	Remarks
#0	Constant	Null value, mainly used to judge the validity of parameters, and # 0 means no parameter is passed in.
#1-#99	Local l variable	Each program (main program, subprogram) has its own and does not affect each other's variables, which are only valid in the current NC program. Once the program is loaded or executed to M99 or M30, it will be automatically cleared.
#100-#299	Common variable	After the system is powered down and restarted, it will be automatically cleared.
#300-#1699	Common variable	The system is still restarted after power failure.
#1700~#1999	System reserved variables	The restart of the system after power failure is still retained.
#2000 以上	System reserved variables	It is suggested that machine tool users should use it carefully, and assign the variable space value operation, which may cause system errors.

【Range of variables】： The values of Local l and public variables can be used in the following ranges：

$-10^{47}\sim -10^{-29}$

0.0

$10^{-29}\sim 10^{47}$

If this range is exceeded during the operation, an alarm will appear.

【Omission of decimal point】： When defining variable values in programs, you can omit decimal points.

【Example】： When # 1 = 123, # 1 is 123.000.

【Use of variables】：

- To use variable values in programs, you need to specify variable numbers after #;
- In addition, when specifying with an equation, please add square brackets before and after the equation;

- When you need to change the value of a variable, put "-" before # and put square brackets before and after the variable.
- [Example]: G00X-[# 1]
- When a variable is defined or used by an expression, the value after the decimal point in the calculation result of the expression is removed.

  【Example】： #[300 + 0.6]equals#300。

【Undefined variable】：

A variable whose value is undefined is called an undefined variable. # 0 is anormally empty variable that can be read, but cannot write.

The description of undefined variables is as follows：

(1) Use of variables: When undefined variables are used, undefined variables will be ignored.

【Example】： When # 10 is empty, once G92 X-# 10Z-5 is implemented, the X axis will not move.

(2) Calculation formula: Except for the case of directly substituting < empty >, the rest is the same as the variable value of 0.

Table 18-2 Undefined Variable Formula

When # 1= < empty >	**When # 1=0**
#2=#1 ↓ #2=< empty >	#2=#1 ↓ #2=0
#2=#1∗5 ↓ #2=0	#2=#1∗5 ↓ #2=0
#2=#1+#1 ↓ #2=0	#2=#1+#1 ↓ #2=0

(3) Condition expressions: < empty > and 0 are different only in the case of EQ and NE.

Table 18-3 Undefined variable conditional formula

When # 1= < empty >	**When # 1=0**
#1 EQ #0 ↓	#1 EQ #0 ↓

Establish	Fail
#1 NE 0 ↓ Establish	#1 NE 0 ↓ Fail
#1 GE #0 ↓ Establish	#1 GE #0 ↓ Establish
#1 GT 0 ↓ Fail	#1 GT 0 ↓ Fail

【Display of Variable Values】：

Displays the overview information of macro variables in Information-Macro Variables-Macro View of the system. Macro variables can only be changed through Macro Login in the Correction module when the program is not implemented.

Attention

(1) Program number and sequence number cannot be used in variables.

(2) Program number (O # 1): When the program number is interpreted as an implementation block, an error will be reported [format error of NC program].

(3) Serial number (N # 1): When interpreted as an implementation block, it will report an error [format error of NC program], but if it is after GOTO statement, macro variables or formulas can be used. Serial number was used as the search object, but it was not found.

【Example】：

```
#3=100
GOTO #3
……
N 100
```

18.1.2Operation instruction

Overview

In the macro statement can flexibly use arithmetic operators, functions and other operations, it is very convenient to achieve complex programming requirements. # j, # K in the format can be replaced with constants. The formula can also be used in the variable number in the left formula.

As shown in the following table：

Table 18-4 Operation Instructions

Operation type	Operation instruction	Meaning
Substitution	#i=#j	
Arithmetic operation	#i=#j + #k	Addition operation
	#i=#j - #k	Subtraction operation
	#i=#j *#k	Multiplication
	#i=#j / #k	Division operation
	#i=#J MOD #k	Modular operation
Function	#i=SIN[#j]	Sine (in radians)
	#i=ASIN[#j]	Arcsine
	#i=COS[#j]	Cosine (unit: radian)
	#i=ACOS[#j]	Inverse cosine
	#i=TAN[#j]	Tangent (in radians)
	#i=ATAN[#j] / [#k]	Arctangent
Symbolic operation	#i=SQRT[#j]	Square root
	#i=ABS[#j]	Absolute value
	#i=ROUND[#j]	Integer after rounding
	#i=FIX[#j]	After discarding the decimal point
	#i=FUP[#j]	Carry after decimal point
	#i=LN[#j]	Natural logarithm
	#i=EXP[#j]	Natural index
Logical operation	#i=#j OR #k	Logic and operation
	#i=#j XOR #k	Bitwise addition operation
	#i=#j AND #k	Logical product operation

①. ASIN function

【Form】：#i = ASIN[#j]

【Format Description】

i represents the angle (range [-90, 90] in angles), and # j represents the sine corresponding to the angle of # i.

Range of # j [-1, 1], exceeding error reported. S function

②. ACOS function

【Form】：#i = ACOS[#j]

【Format Description】

i denotes the angle (range [0,180] in angles), and # j denotes the cosine corresponding to the angle of # i.

Range of # j [-1, 1], exceeding error reported.

③. ATAN function

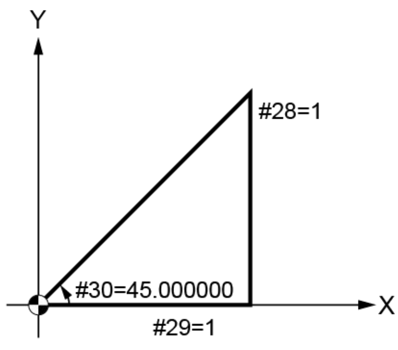
【Form】：#i = ATAN[#j]/[#k]

【Format Description】：

i denotes the angle (range [0,360] in angles), and # j and # k denote the oppositeand adjacent sides of the angle # i in a right triangle, respectively, using constants.

The range of # j, # k is real.

【Example1】：



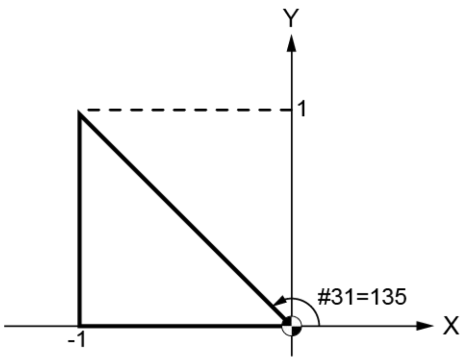
#28 = 1

#29 = 1

#30 = ATAN[#28] / [#29]

Figure 18-1 Example of ATAN Function-1

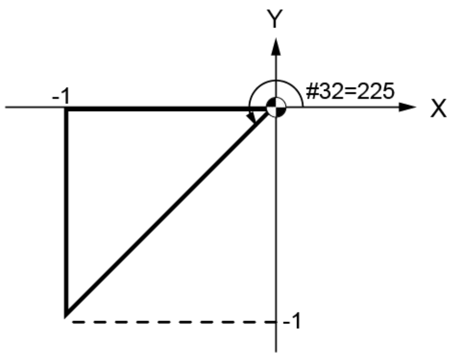
【Example2】：



#31 = ATAN[1]/[-1]

Figure 18-2 Example of ATAN Function-2

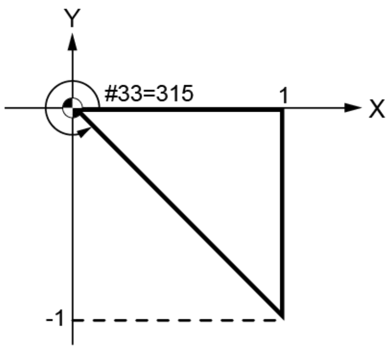
【Example3】：



#32 = ATAN[-1]/[-1]

Figure 18-3 Example of ATAN Function-3

【Example4】：



#33 = ATAN[-1]/[1]

Figure 18-4 Example of ATAN Function-4

④. ROUND function

a) ROUND function in operation.

#1=1.23456789

#2=ROUND[#1] At this time, the first decimal place in # 2 is rounded and 1.0 is entered

b) ROUND function in axis action instruction.

G01 X[ROUND[3.456789]] X is rounded to the first decimal place to become X3

1=5.556789

G01 Y[ROUND[#1]] At this time, Y is rounded to the first decimal place to become Y6.

⑤. Carry and give up

In the NC device, when the absolute value of the integer value is larger than the absolute value of the original value, it is called carry, and when it becomes smaller, it is called scrap.

Attention

Pay special attention when dealing with negative numbers.

【Example】：

When # 1=1. 2, # 2=-1. 2

When # 3 = FUP [# 1]， #3 becomes2.0。

when # 3 = FIX [# 1]， #3 becomes1.0。

When # 3 = FUP [# 2]， #3 becomes-2.0。

when # 3 = FIX [# 2]， #3 becomes-1.0。

⑥. Priority order of operation

a) High priority, as shown below:

b) Brackets [,]

c) Variable #

d) Symbols +,-

e) Functions SIN, COS, TAN, ASIN, ACOS, ATAN, SQRT, ROUND, LN, EXP, ABS, FIX, FUP

f) Multiplication, division* ,/, MOD, AND

g) Addition, Subtraction +,-, OR, XOR

h) Related Operators EQ, NE, LT, LE, GT, GE

i) Assignment =

Note: When operators with the same priority are together, they are executed from left to right.

⑦. Logical operators of bits

The operand of the bit operator is a 32-bit signed integer. A real number with a decimal part is changed into an integer after the decimal part is discarded, and then the operation is carried out. Note that real numbers greater than 2147483647 or less than-2147483648 cannot be changed. When the operand is # 0 (NULL), an error message is displayed.

【Example：】 #1 = #0

Table 18-5-bit logic operations

Formula	Results	Description
IF[#1 AND 255 EQ 1]	False	# 1 takes part in the operation with 0, first bit operation, and then logical comparison [#1 AND 255] EQ 1
#101=12.8 #100=#101 OR 4	12.0	Rounding # 101 first, then doing bit operation [FIX[#101]] OR 4
#110=2200000000 #111=#110 XOR 12	Report an error	#110 数值太大，越界
#120=-1234567 #121=#120 AND 7654321 #122=#120 XOR 7654321 #123=#120 OR 7654321	 6555953 -6692152 -136199	-1234567 = 0XFFED2979 7654321 = 0X0074CBB1 6555953 = 0X00640931 -6692152 = 0XFF99E2C8 -136199 = 0XFFFDEBF9
#130=3 #131=127 IF[2 GT # 130 AND #131]	 False	127 = 0X7F 3 AND 0X7F = 3 2 GT [#130 AND #131]
#141=4 IF[#130+#141 AND 127 EQ 7]	 True	[[#130+#141] AND 127] EQ 7
#150=-11.6 #151=220.3 #152=#150 AND #151 #153=#150 OR #151 #154=#150 XOR #151	 212 -4 -216	Rounded ROUND # 150, # 151 ROUND[-11.6]=-12 ROUND[220.3]=220 -12 = 0XFFFFFFF4 220 = 0X000000DC 212 = 0X000000D4 -4 = 0XFFFFFFFC -216 = 0XFFFFFF28

Attention:

(1) Brackets: The brackets used in the formula are square brackets []. Parentheses () are used for annotation.

(2) Operation error: There will be errors in every calculation

(3) Divisor: In division, an error occurs when the divisor is "0" or TAN [90].

Table 18-6 Relationship between Macro Statement and NC Statement and Setting of "Automatic Decimal Point Judgment"

<table>
<tr><th>Block of macrostatement</th><th>NC statements (blocks other than macro statements)</th></tr>
<tr><td>Block containing operation instructions (=,etc.)</td><td rowspan="7">(1) The substitution and operation of macro statements have nothing to do with the automatic judgment of decimal points. As opposed to macrovariables, the setting of "Setting-Channel-Decimal Point Automatic Judgment" is often carried out in OFF state.

(2) When adjustment is needed (G0T0 statement, 234 of # [# 234], etc.), automatic variables that automatically discard decimal functions and parentheses can also perform real number operation although they are not variables.</td></tr>
<tr><td>Block containing control instructions (G0T0, IF, WHILE, END)</td></tr>
<tr><td>Block containing macro invocation instruction (G65)</td></tr>
<tr><td>Blocks containing certain functional instructions (G32, G27, G28, G29, fixed cycle instructions, etc.)</td></tr>
<tr><td>Using blocks of macro variables through the # symbol</td></tr>
<tr><td>Block containing user-defined extension instructions</td></tr>
<tr><td>Block containing extended instructions called by GMT</td></tr>
</table>

【Example: 】Internal processing when [decimal point automatic judgment] = ON

Table 18-7 Instructions and Internal Processing Methods

Instruction	Internal treatment method
G65 P1 X3I4 E5.	#24 =3.0 #4 = 4.0 #8 = 5.0
#100=7 G65 X3I#100 E5 P1	(#24 = 3.0 #4 = 7.0 #8 =5.0)
G00 X300 Y100	=G00 X0.3 Y0.1
#3=300 G0 X#3 Y100	=G00 X300. Y0. 1
X[100+100]	X200.0
#4=110.3 #3=11 IF [#3 EQ 3] GOTO #4	IF[11.0 EQ 3.0]GOTO 110
G32 X101 Y102	=G32 X101. Y102.

ABS[-50.]	=50.0

18.1.3Branching and repetition

Overview

GOTO statement and IF statement can be used in the program to change the operation order of the program.

(1) Branches consist of the following two categories：

- GOTO statement (unconditional branch);
- IF statement (conditional branch, IF ~then）；

(2) Repeat contains 1 category:

WHILE statement (repeated, in-process);

18.1.3.1Branching

Overview

(1) GOTO statement (unconditional branch)

Serial number n unconditional branch, range 1 ~ 99999999, n can be constant or formula

GOTO n　　n：Serial number（1~99999999）

【Example】：

GOTO 1

GOTO [#10+2]

(2) IF statement (conditional branch)

IF is followed by a conditional formula. IF the conditional formula holds, it branches to the serial number n, and IF it does not hold, it executes the next block.

【Example】：If # 1 is greater than 10 in the following cases, it branches to sequence number N2.

IF[#1 GT 10] GOTO 2 : : :	(1) The conditional formula writes comparison operators between two variables for comparison or between variables and constants, all

Handle : : : N2 G00 G91 X10.0	in square brackets [] boxes. (2) You can write formulas instead of variables.

【Comparison Operator】

Comparison operators consist of two-character letters, which determine whether they are large, small or equal.

Attention

Symbols cannot be used.

Table 18-8 Comparison Operator Meaning

Operator	Meaning
EQ	Equal to (=)
NE	Not equal to（≠）
GT	Greater than（>）
GE	Greater than or equal to（≥）
LT	Less than （<）
LE	Less than or equal to （≤）

【Example】：

Find the sum from 1 to 10:

Initial value of # 1=0 answer

Initial value of # 2=1 addend

N1 IF[#2 GT 10] GOTO 2 The addend exceeds 10 and branches to N2

#1=#1+#2 Calculate the answer

#2=#2+1 Next addend

GOTO 1 Branch to N1

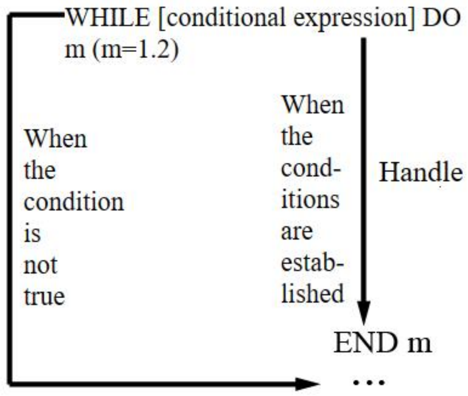
N2 M30 Branch to N1

18.1.3.2Repeat (WHILE statement)

Overview

Specify a conditional expression after WHILE, and when the specified conditional expression is satisfied, execute from DO to END. Between programs. When the specified condition is not met, it goes to the next block of END.

The invocation format is as follows:



Description

When the condition holds, the block from DO to END is implemented after WHILE. If the condition does not hold, enter the next block of END corresponding to DO. The conditions and operators are the same as the IF statement. DO and END after the value for the designated implementation range of the identification number, can use 1 ~ 10, if the use of other codes, will lead to errors.

【Nested】

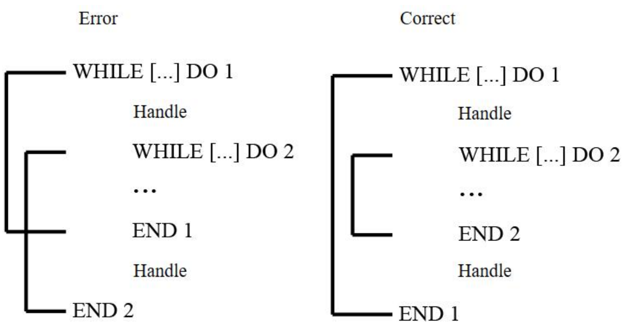
The identification numbers (1 ~ 10) used in DO ~ END can be reused many times. However, when the program has cross-repeating loops (overlapping of DO ranges), the program will make errors.

①. The identification numbers (1 ~ 10) used in DO ~ END can be reused many times.
WHILE […] DO 1
Handle
END 1
……
WHILE […] DO 1

Handle

END

②. The range of DO cannot be crossed.



③. DO can reach 10 layers at most. Note: A maximum of 10 layers are nested, and must start with 1 and contiguous 2,3, 4, …, cannot be marked like 1, 3, 7.

WHILE […] DO 1

……

WHILE […] DO 2

……

WHILE […] DO 3

Handle

END 3

……

END 2

……

END 1

④. Can jump out of the cycle .

WHILE […] DO 1

IF […] GOTO n

END 1

Nn;

⑤. You can't jump into the branch of the cycle.

IF […] GOTO n

……

WHILE […] DO 1

Nn…

END 1

Attention

(1) Infinite cycle

When the WHILE statement is omitted and only DO m is specified, an infinite loop between DO and END is formed.

(2) Processing time

When branching to the sequence specified by the GOTO statement, you need to find the sequence number. When jumping to the sequence number specified in front of the program segment by using GOTO statement, it can also constitute duplication, but the search time is long. In order to shorten the processing time, please use WHILE statement as duplicate instruction.

(3) Undefined variable

In the conditional formula, < null > and 0 (zero) are different only when EQ and NE, and < null > and 0 (zero) can be regarded as the same under other conditions.

【Example】： Find the total from 1 to 10

```
O0001
#1=0
#2=1
WHILE [#2 LE 10] DO 1
#1=#1+#2
#2=#2+1
END 1
M30
```

18.2Macro program call

Overview

The system supports the following ways to call macro programs:

- Modeless call: G65
- Call with macro program modality: G66, G67
- GMT macro program call: G code, M code, T code

18.2.1Independent variable specification rule

Overview

Argument assignments fall into two broad categories:

- The first category: Specify letters other than G, L, M, N, O and P once each.
- The second category: A, B, C once and 10 groups of I, J, K are used for designation, and automatic judgment is made according to the combination of designated letters.

An error occurs when the argument specifies a mixture of types I and II.

■ The first category

Table 18-9 Argument Specification Type I

Argument name	Macro variable	Argument name	Macro variable	Argument name	Macro variable
A	#1	I	#4	T	#20
B	#2	J	#5	U	#21
C	#3	K	#6	V	#22
D	#7	M	#13	W	#23
E	#8	Q	#17	X	#24
F	#9	R	#18	Y	#25
H	#11	S	#19	Z	#26

Attention:

(1) G, L, M, N, O, P cannot be used as independent variables.

(2) Usually M cannot be used as an independent variable. When and only when used as a GMT invocation format, the M instruction is taken as an argument and the value is passed into # 13.

(3) Arguments that do not need to be specified can be omitted. The macro variable corresponding to the omitted argument is empty.

■ The second category

Table 18-10 Argument Specification Type II

Argument name	Macro variable	Argument name	Macro variable	Argument name	Macro variable
A	#1	A	#1	J7	#23
B	#2	B	#2	K7	#24

C	#3
I1	#4
J1	#5
K1	#6
I2	#7
J2	#8
K2	#9
I3	#10
J3	#11

C	#3
I1	#4
J1	#5
K1	#6
I2	#7
J2	#8
K2	#9
I3	#10
J3	#11

I8	#25
J8	#26
K8	#27
I9	#28
J9	#29
K9	#30
I10	#31
J10	#32
K10	#33

Attention：

(1) The subscripts of I, J, K representing the specified order of arguments are not written in the actual program.

(2) The arguments A, B, C are used only once, and I, J, K are specified as a set of methods that can be reused up to 10 times. Typically, the Class II designation method is used when the value of three-dimensional coordinates is taken as an independent variable.

(3) The G65 block needs to be formatted as a separate block.

Level of Local l macro variable

Corresponding to nesting, there are 11 Level s of Local l macro variables from 0 to 10. The Level of the main program is 0. Every time the macro program is called, the Level of the Local l macro variable increases by one Level , and the Local l macro variable of the previous Level is saved

Inside the NC device. After implementing M99, return to the calling program. At this point, the Level of the Local l variable is reduced by 1, and the Local l variable value saved when the subroutine was called is restored.

Description

When M98 calls subroutines, the Level of Local l macro variables does not increase by one Level . The Local l macro variable Level increases only when G65, MOTION, GMT are called.

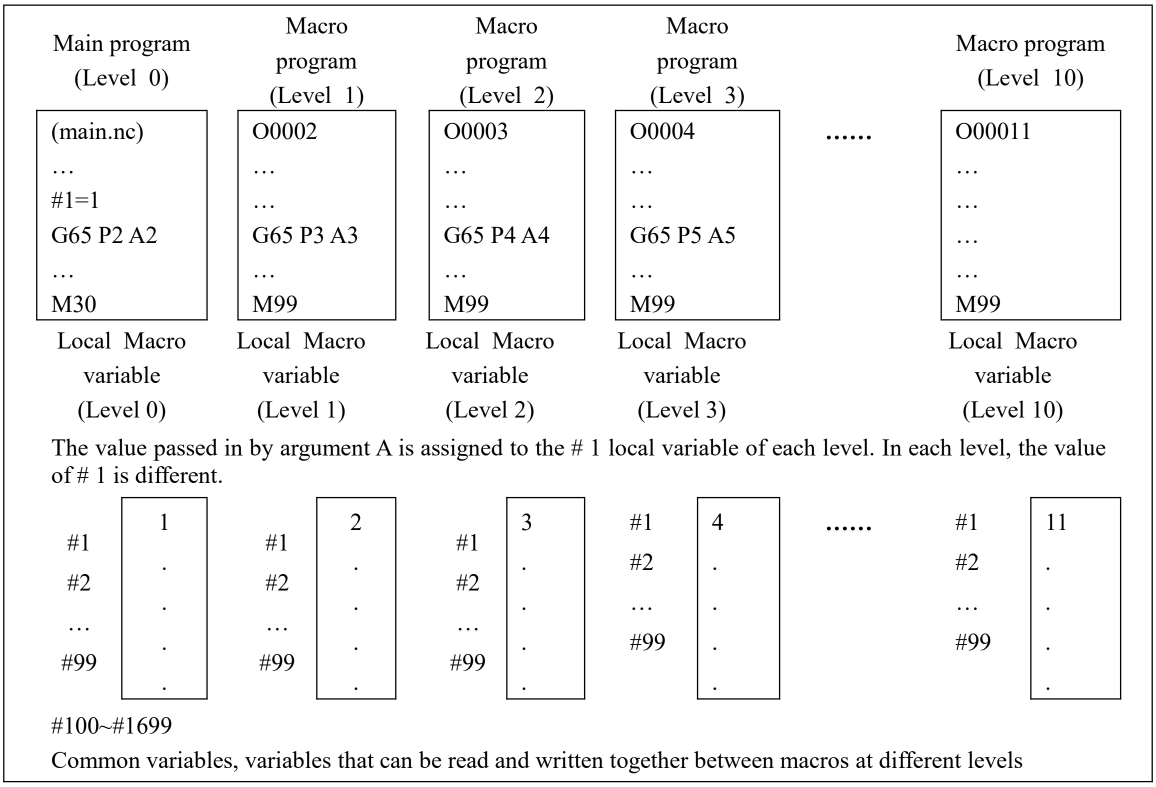


Figure 18-5 Relationship between local macro variables and nested calls of subroutines

18.2.2Modeless call（G65）

Overview

When G65 is specified, the user macro program specified with parameter P is called, and the independent variables and variables needed by the user macro program are passed to the user macro program.

【Instruction description】：

G65 P_ L_ [independent variable address word] ;

P_	：	The program number to call
L_	：	Number of repeated calls
[_]	：	Data that the user needs to pass to the macro program

【Example】：

```
O0001
……
G65 P100 L2 A1.0 B2.0
……
M30
```

```
O0100
#3 = #1 + #2
IF [#3 GT 360] GOTO 9
G00 G91 X#3
N9 M99
```

Instruction description

(1) G65 is a modeless instruction and needs to be specified in this line every time a macro program is called.

(2) Specify the program number of the macro program with P after G65. The file name of macro program must be specified by adding 4 digits (integer) after O. The suffix is ". NC".

(3) If you need to repeat the specification more than once, specify the L parameter, and the duplication range is between 1 and 10000. When L is omitted, the number of repetitions is regarded as one.

(4) When an argument is specified, the argument value is passed into the corresponding macro variable. See Figure 18-5 for details.

18.2.3Macro program modal call （G66、G67）

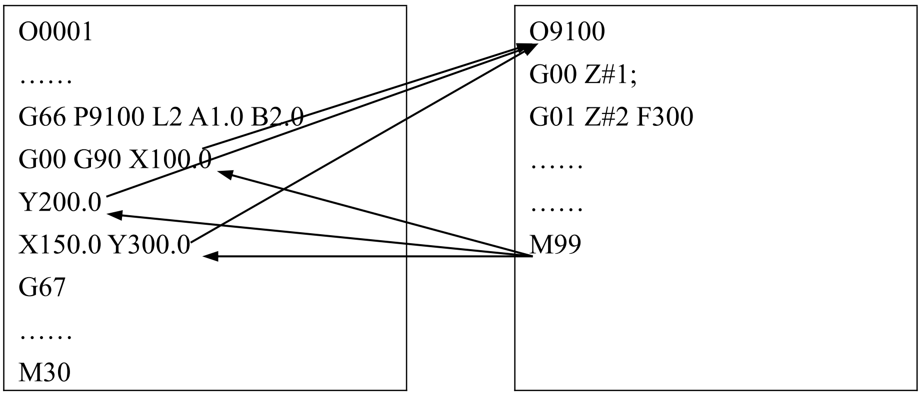
Overview

After the modal call is specified by G66, the specified subroutine is automatically called after the program segment of axis movement is executed every time until the modal call is cancelled by G67.

【Instruction description】：

G66 P_ L_ [Independent variable address word] ;		
P_	：	The program number to call
L_	：	Number of repeated calls (1 when omitted)
[_]	：	Data that the user needs to pass into the macro program (up to 4-bit integers)

【Example】：



Instruction description

- Call

(1) After G66, use P to specify the program number for modal call. The file name of the macro program must be followed by an 4 digits (integers). The suffix is ". NC".

(2) If you need to repeat the specification more than once, specify the L parameter, and the duplication range is between 1 and 10000. When L is omitted, the number of repetitions is regarded as one.

(3) Like G65 calls, arguments can be used to specify the data passed to macro instructions.

(4) In G66 mode, after each execution of the program segment of the mobile instruction, the macro program is called.

- Nesting

(1) Subroutine calls are nested in 10 layers, including M98 calls, G65 calls and G66 modal calls.

(2) Nested use of G66 modal calls is not allowed. When G66 modal call instructions are specified continuously, the modal calls specified later are valid.

Attention:

(1) G66 and G67 are generally used in pairs. However, G67 can also be specified if it is not in G66 mode.

(2) In the G66 program section, no macro program calls are made, but local variables

(arguments) have been set.

(3) In the program segment of auxiliary functions, G27, G28, G29, G30 and other instructions, macro program calls are not made.

(4) When the auxiliary function and the mobile instruction are specified, the mobile instruction and the auxiliary instruction are executed first, and then the macro program is called.

(5) In modal calling mode, it is not allowed to specify fixed loop instructions.

Program legend

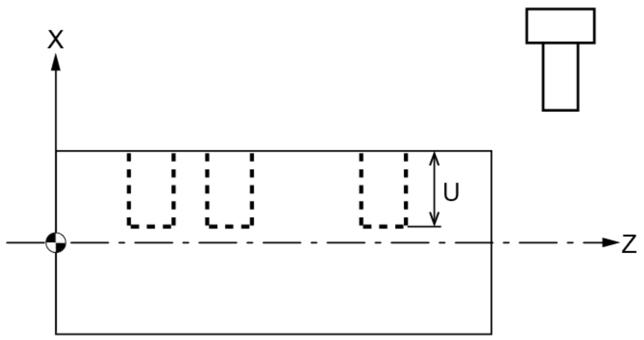


Fig. 18-5 Legend of slotting process

【Instruction description】:

G66 P9110 U_ F_ ;

U_	:	Hole depth (incremental command value)
F_	:	Drilling speed

User program	Called program
O0001	O9110
G50 X100.0 Z200.0	G01 U-[#21] F#9
S1000 M03	G00 U#21
G66 P9110 U5.0 F0.5	M99
G00 X60.0 Z80.0	
Z50.0	
Z30.0	
G67	
G00 X0 Z200.0 M05	

M30

18.2.4 GMT macro program call

18.2.4.1 G code calls Macro program

Overview

In addition to modeless (G65) callers, users can also call macro programs in the form of G code, which is called in the same way as G65. By setting a G code number used to call macro instructions in the parameters in advance, the macro program can be called through the G code. It invokes the same method as G65.

Instruction description

(1) By setting a G code number in "Fixed G Instruction Number 1" ~ "Fixed G Instruction Number 10" (# 32920 ~ # 32929), the user macro program starting with "G Instruction Subroutine Number Offset" (# 32950) can be called.

(2) Normally set "G instruction subroutine number offset" (# 32950) to 9010. Therefore, the corresponding relationship between parameters and program numbers is shown in the following table:

Table 18-11 G Instruction Subroutine Offset-Parameter and Program Number Correspondence

Parameter number	Parameter name	Program number
#32920	Fixed G instruction number 1	O9010.NC
#32921	Fixed G instruction number 2	O9011.NC
#32922	Fixed G instruction number 3	O9012.NC
#32923	Fixed G instruction number 4	O9013.NC
#32924	Fixed G instruction number 5	O9014.NC
#32925	Fixed G instruction number 6	O9015.NC
#32926	Fixed G instruction number 7	O9016.NC
#32927	Fixed G instruction number 8	O9017.NC
#32928	Fixed G instruction number 9	O9018.NC
#32929	Fixed G instruction number 10	O9019.NC

(3) You can specify a number of repetitions from 1 to 10000 by address L.

(4) When a negative G code is set, it becomes a modal call (equivalent to G66).

Program example

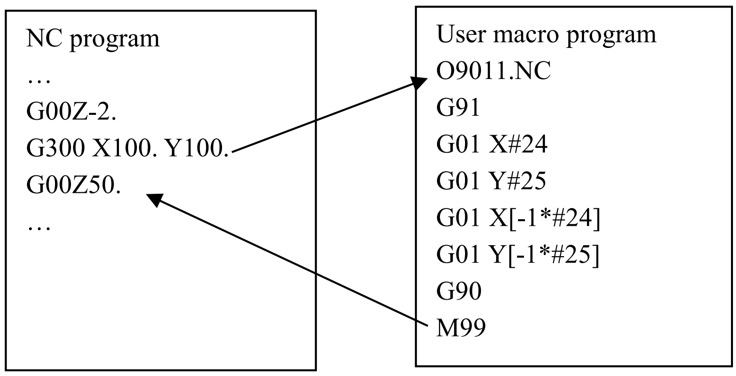
【Example】：

The macro program is called by G code, and the G300 instruction is used to call the user macro program, which means to execute a cutting from the current position and along the rectangular path defined by X and Y.

【Operational Steps】

①. Modify the parameter "Fixed G Instruction Number 2" (# 32921) to 300, and store the user macro program written in the user disk with the file name O9011. NC.

②. After opening "GMT Quick Call Effective" (# 32957), write G300 instruction in NC program, and then call O9011. NC. You can specify the arguments to pass into the user macro program according to the argument specification method of G65.



18.2.4.2 G code calls macro program (multiple specifications)

Overview

By setting the initial G code number used in macro instruction calling, the initial number of called subroutines and the number of called codes in advance, a plurality of continuous G codes for macro program calling can be defined.

Instruction description

(1) The G code can be set in the parameter "G area call start code" (# 32971), and the

number of G codes in the parameter "G area call code number" (# 32972) can be set, and the user macro program with the program number and the number of G codes set in the parameter "G area call code number" (# 32972) can be called from the program number set in the parameter "G area call subroutine start number" (# 32973). To invalidate this call, set 0 in the parameter "Number of code called in the G area" (# 32972).

(2) Repeat, the argument specification is the same as the macro program call using G code.

Program example

【Example】：

Set "G Area Call Start Code" (# 32971) = 900, "Number of G Area Call Codes" When (# 32972) = 100 and "G Region Calling Subroutine Start Number" (# 32973) = 1000, the relationship between G code and calling subroutine is as follows:

Table 18-12 Relationship between G code and calling subroutine

G code	Call subroutine number
G900	O1000.NC
G901	O1001.NC
G902	O1002.NC
G903	O1003.NC
G904	O1004.NC
G905	O1005.NC
…	…
G999	O1099.NC

Attention:

(1) When the specified "fixed G instruction number 1" ~ "fixed G instruction number 10" (#32920~#32929) calling G code is repeated with multiple specified G codes for calling, the priority is to execute the call using "fixed G instruction number 1" ~ "fixed G instruction number 10" (#32920~#32929)

(2) The M code, G code and T code in the called macro program (subprogram) are treated

as ordinary M code, G code and T code.

18.2.4.3M code calls macro program

Overview

In addition to modeless (G65) calling programs, users can also call macro programs in the form of M code, and their calling methods are the same as those of G65 and M98 respectively.

Instruction description

(1) Through "Fixed M Instruction Number 1 (G65)" ~ "Fixed M Instruction Number 10 (G65)"(# 32930 ~ # 32939), set a M code number, can be called to "M instruction subprogram number offset(G65) "(# 32951).

(2) Normally set the "M Instruction Subprogram Number Offset (G65)" (# 32951) to 9020. Therefore, the corresponding relationship between parameters and program numbers is shown in the following table:

Table 18-13 M instruction subroutine number offset-corresponding relationship between parameters and program number

Parameter number	Parameter name	Program number
#32930	Fixed M instruction number 1(G65)	O9020.NC
#32931	Fixed M instruction number 2(G65)	O9021.NC
#32932	Fixed M instruction number 3(G65)	O9022.NC
#32933	Fixed M instruction number 4(G65)	O9023.NC
#32934	Fixed M instruction number 5(G65)	O9024.NC
#32935	Fixed M instruction number 6(G65)	O9025.NC
#32936	Fixed M instruction number 7(G65)	O9026.NC
#32937	Fixed M instruction number 8(G65)	O9027.NC
#32938	Fixed M instruction number 9(G65)	O9028.NC
#32939	Fixed M instruction number 10(G65)	O9029.NC

Attention

(1) You can specify a number of repetitions from 1 to 10000 by address L.

(2) When the macro program is called, the M code is substituted into the argument # 13.

(3) M code, G code and T code in the called macro program (subprogram) are treated as ordinary M code, G code and T code.

18.2.4.4M code calls macro program (multiple specifications)

【Functions】：

By setting the range of M code number used in the call of macro instruction and the number of called subprograms in advance, the macro program call using multiple M codes can be defined. The calling method is the same as G65.

Description

(1) By setting the upper and lower limits of M code number in "lower limit of M area calling code" ~ "upper limit of M area calling code" (# 32968 ~ # 32969), the user macro program specified by "M area calling subrprogram number" (# 32970) can be called.

(2) You can specify a number of repetitions from 1 to 10000 by address L.

(3) When the macro program is called, the M code is substituted into the argument # 13.

Attention

(1) GMT macro program calls are only available if "GMT shortcut call valid" (# 32957) is turned on.

(2) M code, G code and T code in the called macro program (subroutine) are regarded as ordinary M code, G code, T code processing.

18.2.4.5M code subroutine call

【Functions】：

By setting an M code number used to call the subroutine in the parameter in advance, the subroutine can be called through the M code. The calling method is the same as M98.

【Description】：

(1) Through "Fixed M Instruction Number 1 (M98)" ~ "Fixed M Instruction Number 10 (M98)" (# 32940 ~ # 32949), set a M code number, can be called to "M instruction subprogram number offset(M98) "(# 32952).

(2) Normally set the "M Instruction Subprogram Number Offset (M98)" (# 32952) to 9001. Therefore, the corresponding relationship between parameters and program numbers is shown in Table 18-14.

Table 18-14 Correspondence between parameters and programs

Parameter number	Parameter name	Program number
#32940	Fixed M instruction number 1(M98)	O9001.NC
#32941	Fixed M instruction number 2(M98)	O9002.NC
#32942	Fixed M instruction number 3(M98)	O9003.NC
#32943	Fixed M instruction number 4(M98)	O9004.NC
#32944	Fixed M instruction number 5(M98)	O9005.NC
#32945	Fixed M instruction number 6(M98)	O9006.NC
#32946	Fixed M instruction number 7(M98)	O9007.NC
#32947	Fixed M instruction number 8(M98)	O9008.NC
#32948	Fixed M instruction number 9(M98)	O9009.NC
#32949	Fixed M instruction number 10(M98)	O9010.NC

(3) You can specify a number of repetitions from 1 to 10000 by address L. Argument specification is not allowed.

Attention

(1) GMT macro program calls are only available if "GMT shortcut call valid" (# 32957) is turned on.

(2) M code, G code and T code in the called macro program (subprogram) are treated as ordinary M code, G code and T code.

18.2.4.6T code calls macro program

Overview

By setting "T instruction directly calls subprogram" as ON in the parameter in advance, the macro program can be called through T code. The calling method is the same as G65. Specific code is detailed in the chapter of M code calling macro program in the user programming manual of Lynuc.

Instruction description

(1) By setting "T instruction direct call subroutine" (# 32953) to ON, the user macro program specified by "T instruction subroutine number" (# 32954) can be called.

(2) Usually, the "T instruction subroutine number" (# 32954) is set to 9000. That is, whenever T code is specified in the machining program, subroutine O9000. NC can be called. The T code specified in the machining program is brought into the argument # 20.

(3) You can specify a number of repetitions from 1 to 10000 by address L.

(4) If the T instruction is specified to be treated as a normal T code after the G code of the macro program call, the T code is substituted into the argument # 20.

19.Common instructions for model machining

19.1Two-way milling of circular plane（G160.1）

【Functions】：

Two-way milling is carried out on the cylindrical surface of XY plane.

【Instruction description】：

G160.1 X_ Y_ R_ W_ Z_ I_ B_ E_ C_ Q_ A_ F_ ;

X_	：	Workpiece coordinate value of X axis of cylinder center
Y_	：	The workpiece coordinate value of the Y axis in the center of the cylinder
R_	：	Safety plane Z axis of the workpiece coordinate value
W_	：	The workpiece coordinate value of the starting plane Z axis
Z_	：	Workpiece coordinate value of Z axis of starting plane
I_	：	The contour radius of the cylindrical part
B_	：	Milling tool radius
E_	：	The ratio of the amount of step movement in the radius direction to the effective milling tool
C_	：	Workpiece avoidance clearance amount
Q_	：	Z-axis each drop depth
A_	：	Machining allowance relative to the plane in place
F_	：	Cutting feed speed

Instruction description

(1) Please refer to the parameter range given by the instruction in [Intelligent Editing] to specify parameters;

(2) The tool used in program processing is flat-bottomed tool. If you choose a flat-bottomed tool with rounded corners, you need to consider rounded corners;

(3) Pay attention to the relationship among R, W and Z to ensure processing safety;

(4) The following procedures are only compatible with G90 format;

(5) Temporarily incompatible with scaling;

19.2Two-way milling of rectangular plane（G160.2）

【Functions】：

The rectangular surface of XY plane is milled in two directions.

【Instruction description】：

G160.2 X_ Y_ R_ W_ Z_ U_ V_ B_ E_ C_ Q_ A_ F_ ;		
X_	：	Workpiece coordinate value of X axis in the lower left corner of rectangle
Y_	：	Workpiece coordinate value of Y axis in lower left corner of rectangle
R_	：	Workpiece coordinate value of Z axis of safety plane
W_	：	Workpiece coordinate value of Z axis of starting plane
Z_	：	Workpiece coordinate value of Z axis of position plane
U_	：	Transverse (X) length of workpiece
V_	：	Longitudinal (Y) length of workpiece
B_	：	Milling tool radius
E_	：	Effective ratio of radial step movement to milling tool
C_	：	Workpiece avoidance clearance
Q_	：	Depth of each Z-axis descent
A_	：	Machining allowance relative to the in-place plane
F_	：	Cutting feed speed

Instruction description

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters

(2) The cutter used in program machining is flat-bottomed tool. If flat-bottomed tool with fillet is selected, fillet should be considered

(3) Pay attention to the relationship among R, W and Z to ensure the processing safety

(4) The following program is only compatible with G90 format

(5) Temporarily incompatible with scaling

19.3Co-directional milling of rectangular plane（G160.3）

【Functions】:

The rectangular surface of XY plane is milled in the same direction.

【Instruction description】:

G160.3 X_ Y_ R_ W_ Z_ U_ V_ B_ E_ C_ Q_ A_ F_ ;		
X_	:	Workpiece coordinate value of X axis in the lower left corner of rectangle
Y_	:	Workpiece coordinate value of Y axis in lower left corner of rectangle
R_	:	Workpiece coordinate value of Z axis of safety plane
W_	:	Workpiece coordinate value of Z axis of safety plane
Z_	:	Workpiece coordinate value of Z axis of position plane
U_	:	Transverse (X) length of workpiece
V_	:	Longitudinal (Y) length of workpiece
B_	:	Milling tool radius
E_	:	Effective ratio of radial step movement to milling tool
C_	:	Workpiece avoidance clearance
Q_	:	Depth of each Z-axis descent
A_	:	Machining allowance relative to the in-place plane
F_	:	Cutting feed speed

Instruction description

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters.

(2) The cutter used in program machining is flat-bottomed tool. If flat-bottomed tool with fillet is selected, fillet should be considered.

(3) Pay attention to the relationship among R, W and Z to ensure processing safety.

(4) The following program is only compatible with G90 format.

(5) Scaling is temporarily incompatible.

19.4Two-way milling of circular cavity（G161.1）

【Functions】：

Two-way milling of cavity for XY plane cylinder is carried out.

【Instruction description】：

G161.1 X_ Y_ R_ W_ Z_ I_ U_ V_ B_ E_ C_ Q_ A_ F_ ;		
X_	：	Absolute coordinate value of arc center X of circular part
Y_	：	Absolute coordinate value of circular arc center Y of circular part
R_	：	Workpiece coordinate value of Z axis of safety plane
W_	：	Workpiece coordinate value of Z axis of starting plane
Z_	：	Workpiece coordinate value of Z axis of position plane
I_	：	Cavity radius of round parts
U_	：	Height of spiral lower cutting
V_	：	Angle of spiral lower cutting
B_	：	Milling tool radius
E_	：	Effective ratio of radial step movement to milling tool
C_	：	Machining allowance of workpiece side wall
Q_	：	Depth of each Z-axis descent
A_	：	Machining allowance relative to the in-place plane
F_	：	Cutting feed speed

Instruction description

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters.

(2) The cutter used in program machining is flat-bottomed tool. If flat-bottomed tool with fillet is selected, fillet should be considered.

(3) Pay attention to the relationship among R, W and Z to ensure processing safety.

(4) The following program is only compatible with G90 format.

(5) Scaling is temporarily incompatible.

(6) The procedure is spiral cutting, pay attention to the speed of cutting.

19.5Two-way milling of rectangular cavity（G161.2）

【Functions】：

The rectangle of XY plane is milled in two directions.

【Instruction description】：

G161.2 X_ Y_ R_ W_ Z_ U_ V_ J_ I_ B_ E_ C_ Q_ A_ F_ ;

X_	：	Workpiece coordinate value of X axis in the lower left corner of rectangle
Y_	：	Workpiece coordinate value of Y axis in lower left corner of rectangle
R_	：	Workpiece coordinate value of Z axis of safety plane
W_	：	Workpiece coordinate value of Z axis of starting plane
Z_	：	Workpiece coordinate value of Z axis of position plane
U_	：	Transverse (X) length of workpiece
V_	：	Longitudinal (Y) length of workpiece
J_	：	Height of slash cutting
I_	：	Angle of slash cutting
B_	：	Milling tool radius
E_	：	Effective ratio of radial step movement to milling tool
C_	：	Machining allowance of workpiece side wall
Q_	：	Depth of each Z-axis descent
A_	：	Machining allowance relative to the in-place plane
F_	：	Cutting feed speed

Instruction description

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters.

(2) The tool used in program processing is flat-bottomed tool. If you choose a flat-bottomed tool with rounded corners, you should consider rounded corners.

(3) Pay attention to the relationship among R, W and Z to ensure processing safety.

(4) The following program is only compatible with G90 format.

(5) Scaling is temporarily incompatible.

(6) The procedure is slash cutting, pay attention to the speed of cutting.

19.6Milling Inner Circle（G162.1）

【Functions】：

Milling of circular inner contour.

【Instruction description】：

G162.1 X_ Y_ R_ W_ Z_ I_ B_ C_ Q_ F_ ;

X_	：	Workpiece coordinate value of circular groove center X axis
Y_	：	Workpiece coordinate value of Y axis of circular groove center
R_	：	Workpiece coordinate value of Z axis of safety plane
W_	：	Workpiece coordinate value of Z axis of starting plane
Z_	：	Workpiece coordinate value of Z axis of position plane
I_	：	Circular groove profile radius
B_	：	Milling tool radius
C_	：	Cut-in circle radius value of tool contacting workpiece
Q_	：	Depth of each Z-axis descent
F_	：	Cutting feed speed

Instruction description

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters.

(2) The cut-in circle radius must be greater than the tool radius.

(3) Pay attention to the relationship among R, W and Z to ensure processing safety.

(4) The following program is only compatible with G90 format.

(5) Scaling is temporarily incompatible.

19.7Milling Cylinder（G162.2）

【Functions】：

Round contour milling.

【Instruction description】：

G162.2 X_ Y_ R_ W_ Z_ I_ B_ C_ Q_ A_ F_ ;

X_	：	Workpiece coordinate value of X axis of circle center
Y_	：	Workpiece coordinate value of Y axis of circle center
R_	：	Workpiece coordinate value of Z axis of safety plane
W_	：	Workpiece coordinate value of Z axis of starting plane
Z_	：	Workpiece coordinate value of Z axis of position plane
I_	：	Circular outer contour radius
B_	：	Milling tool radius
C_	：	Cut-in circle radius value of tool contacting workpiece
Q_	：	Depth of each Z-axis descent
F_	：	Cutting feed speed

Instruction description

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters.

(2) The cut-in circle radius must be greater than the tool radius.

(3) Pay attention to the relationship among R, W and Z to ensure processing safety.

(4) The following program is only compatible with G90 format.

(5) Scaling is temporarily incompatible.

19.8Milling Inner Rectangle（G162.3）

【Functions】：

Milling of rectangular inner contour.

【Instruction description】：

G162.3 X_ Y_ R_ W_ Z_ U_ V_ B_ C_ Q_ F_ ;		
X_	：	Workpiece coordinate value of X axis in the lower left corner of rectangle
Y_	：	Workpiece coordinate value of Y axis in lower left corner of rectangle
R_	：	Workpiece coordinate value of Z axis of safety plane
W_	：	Workpiece coordinate value of Z axis of starting plane
Z_	：	Workpiece coordinate value of Z axis of position plane
U_	：	Transverse (X) length of workpiece
V_	：	Longitudinal (Y) length of workpiece
B_	：	Milling tool radius
C_	：	Cut-in circle radius value of tool contacting workpiece
Q_	：	Depth of each Z-axis descent
F_	：	Cutting feed speed

Instruction description

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters.

(2) The cut-in circle radius must be greater than the tool radius.

(3) Pay attention to the relationship among R, W and Z to ensure processing safety.

(4) The following program is only compatible with G90 format.

(5) Scaling is temporarily incompatible.

19.9Inner Milling Rectangle (Round Corner)（G162.4）

【Functions】：

Milling of rectangular inner contour.

【Instruction description】：

G162.4 X_ Y_ R_ W_ Z_ U_ V_ I_ B_ C_ Q_ F_ ;

X_	：	Workpiece coordinate value of X axis in the lower left corner of rectangle
Y_	：	Workpiece coordinate value of Y axis in lower left corner of rectangle
R_	：	Workpiece coordinate value of Z axis of safety plane
W_	：	Workpiece coordinate value of Z axis of starting plane
Z_	：	Workpiece coordinate value of Z axis of position plane
U_	：	Transverse (X) length of workpiece groove
V_	：	Longitudinal (Y) length of workpiece groove
I_	：	Corner radius value of workpiece groove
B_	：	Milling tool radius
C_	：	Cut-in circle radius value of tool contacting workpiece
Q_	：	Depth of each Z-axis descent
F_	：	Cutting feed speed

Instruction description

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters.

(2) The cut-in circle radius must be greater than the tool radius.

(3) Pay attention to the relationship among R, W and Z to ensure processing safety.

(4) The following program is only compatible with G90 format.

(5) Scaling is temporarily incompatible.

19.10Milled Outer Rectangle（G162.5）

【Functions】：

Rectangular contour milling.

【Instruction description】：

G162.5 X_ Y_ R_ W_ Z_ U_ V_ I_ B_ C_ Q_ F_ ;		
X_	：	Workpiece coordinate value of square workpiece center Xaxis
Y_	：	Workpiece coordinate value of Y axis of square workpiece center
R_	：	Workpiece coordinate value of Z axis of safety plane
W_	：	Workpiece coordinate value of Z axis of starting plane
Z_	：	Workpiece coordinate value of Z axis of position plane
U_	：	Transverse (X) length of workpiece boss
V_	：	Longitudinal (Y) length of workpiece boss
I_	：	Corner radius value of workpiece boss
B_	：	Milling tool radius
C_	：	Cut-in circle radius value of tool contacting workpiece
Q_	：	Depth of each Z-axis descent
F_	：	Cutting feed speed

【Description】：

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters.

(2) The cut-in circle radius must be greater than the tool radius.

(3) Pay attention to the relationship among R, W and Z to ensure processing safety.

(4) The following program is only compatible with G90 format.

(5) Scaling is temporarily incompatible.

19.11Milling Inner Circle (Helix)（G162.6）

【Functions】：

Spiral milling of circular inner profile.

【Instruction description】：

G162.6 X_ Y_ R_ W_ Z_ I_ U_ B_ E_ F_ ;		
X_	:	Workpiece coordinate value of circular groove center X axis
Y_	:	Workpiece coordinate value of Y axis of circular groove center
R_	:	Workpiece coordinate value of Z axis of safety plane
W_	:	Workpiece coordinate value of Z axis of starting plane
Z_	:	Workpiece coordinate value of Z axis of position plane
I_	:	Circular groove profile radius
U_	:	Height of spiral lower cutting
B_	:	Milling tool radius
E_	:	Z-axis spiral trajectory step
F_	:	Cutting feed speed

Instruction description

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters.

(2) Pay attention to the relationship among R, W and Z to ensure processing safety.

(3) The following program is only compatible with G90 format.

(4) Scaling is temporarily incompatible.

19.12Milling Cylinder (Helix) （G162.7）

【Functions】：

Spiral milling of circular outer profile.

【Instruction description】：

G162.7 X_ Y_ R_ W_ Z_ I_ U_ B_ E_ F_ ;

X_	：	Workpiece coordinate value of circular boss center X axis
Y_	：	Workpiece coordinate value of Y axis of circular boss center
R_	：	Workpiece coordinate value of Z axis of safety plane
W_	：	Workpiece coordinate value of Z axis of starting plane
Z_	：	Workpiece coordinate value of Z axis of position plane
I_	：	Profile radius of round boss
U_	：	Height of spiral lower cutting
B_	：	Milling tool radius
E_	：	Z-axis spiral trajectory step
F_	：	Cutting feed speed

Instruction description

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters.

(2) Pay attention to the relationship among R, W and Z to ensure processing safety.

(3) The following program is only compatible with G90 format.

(4) Scaling is temporarily incompatible.

19.13Rectangular frame drilling（G163.1）

【Functions】：

The hole group is drilled by means of counterclockwise displacement drilling along the center line of rectangular frame hole group of square parts.

【Instruction description】：

G163.1 X_ Y_ R_ Z_ A_ I_ J_ U_ V_ Q_ F_ ;

X_	：	Workpiece coordinate value of X axis of starting hole position in lower left corner
Y_	：	Workpiece coordinate value of Y axis of starting hole position in lower left corner
R_	：	Workpiece coordinate value of Z axis of drilling safety plane
Z_	：	Workpiece coordinate value of Z axis of drilling position plane
A	：	Rotation angle between rectangular frame integral hole position and X axis
I_	：	Number of Horizontal Axis Hole Rows
J	:	Number of rows of longitudinal axis holes
U_	：	Spacing between horizontal axis holes
V_	：	Longitudinal shaft hole spacing
Q_	：	Borehole depth per feed
F_	：	Cutting feed speed

Instruction description

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters.

(2) In G99 mode, return to R point according to cutting feed speed. In G98 mode, after returning to R point according to cutting feed speed, fast forward to return to the starting point position surface.

(3) Drilling mode is G73.

(4) Pay attention to the relationship among R, W and Z to ensure processing safety.

(5) The following program is only compatible with G90 format.

(6) Scaling is temporarily incompatible.

19.14Rectangular mesh drilling（G163.2）

【Functions】：

The rectangular mesh hole group is drilled by mesh path diagram.

【Instruction description】：

G163.2 X_ Y_ R_ Z_ A_ I_ J_ U_ V_ Q_ F_ ;		
X_	：	Workpiece coordinate value of X axis of starting hole position in lower left corner
Y_	：	Workpiece coordinate value of Y axis of starting hole position in lower left corner
R_	：	Workpiece coordinate value of Z axis of drilling safety plane
Z_	：	Workpiece coordinate value of Z axis of drilling position plane
A	：	Rotation angle between rectangular mesh integral hole position and X axis
I_	：	Number of Horizontal Axis Hole Rows
J	：	Number of rows of longitudinal axis holes
U_	：	Spacing between horizontal axis holes
V_	：	Longitudinal shaft hole spacing
Q_	：	Borehole depth per feed
F_	：	Cutting feed speed

Instruction description

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters.

(2) In G99 mode, return to R point according to cutting feed speed. In G98 mode, after returning to R point according to cutting feed speed, fast forward to return to the starting point position surface.

(3) Drilling mode is G73.

(4) Pay attention to the relationship among R, W and Z to ensure processing safety.

(5) The following program is only compatible with G90 format.

(6) Scaling is temporarily incompatible.

19.15Straight Borehole（G163.3）

【Functions】：

The co-directional displacement drilling method is adopted to drill the row of holes.

【Instruction description】：

G163.3 X_ Y_ R_ Z_ A_ I_ U_ Q_ F_ ;

X_	：	Workpiece coordinate value of X axis of starting hole position in lower left corner
Y_	：	Workpiece coordinate value of Y axis of starting hole position in lower left corner
R_	：	Workpiece coordinate value of Z axis of drilling safety plane
Z_	：	Workpiece coordinate value of Z axis of drilling position plane
A	：	Rotation angle between rectangular mesh integral hole position and X axis
I_	：	Number of horizontal axis Hole Rows
U_	：	Spacing between horizontal axis holes
Q_	：	Borehole depth per feed
F_	：	Borehole cutting feed speed

Instruction description

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters.

(2) In G99 mode, return to R point according to cutting feed speed. In G98 mode, after returning to R point according to cutting feed speed, fast forward to return to the starting point position surface.

(3) Drilling mode is G73.

(4) Pay attention to the relationship among R, W and Z to ensure processing safety.

(5) The following program is only compatible with G90 format.

(6) Scaling is temporarily incompatible.

19.16Rectangular frame tapping（G164.1）

【Functions】：

The hole group is tapped by counter-clockwise displacement tapping along the center line of rectangular frame hole group of square parts.

【Instruction description】：

G164.1 X_ Y_ R_ Z_ A_ I_ J_ U_ V_ Q_ F_ ;

X_	:	Workpiece coordinate value of X axis of starting hole position in lower left corner
Y_	:	Workpiece coordinate value of Y axis of starting hole position in lower left corner
R_	:	Workpiece coordinate value of Z axis of drilling safety plane
Z_	:	Workpiece coordinate value of Z axis of drilling position plane
A	:	Rotation angle between rectangular frame integral hole position and X axis
I_	:	Number of horizontal axis hole rows
U_	:	Spacing between horizontal axis holes
Q_	:	Tapping depth per feed
F_	:	Feed speed of tapping and cutting

Instruction description

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters.

(2) In G99 mode, return to R point according to cutting feed speed. In G98 mode, after returning to R point according to cutting feed speed, fast forward to return to the starting point position surface.

(3) The tapping method is G84.

(4) Pay attention to the relationship among R, W and Z to ensure processing safety.

(5) The following program is only compatible with G90 format.

(6) Scaling is temporarily incompatible.

19.17Rectangular mesh tapping（G164.2）

【Functions】：

The rectangular mesh hole group is tapped by mesh path diagram.

【Instruction description】：

G164.2 X_ Y_ R_ Z_ A_ I_ J_ U_ V_ Q_ F_ ;		
X_	：	X-axis workpiece coordinate value of the starting hole position in the lower left corner
Y_	：	Y-axis workpiece coordinate value of the starting hole position in the lower left corner
R_	：	Workpiece coordinate value of Z axis of tapping safety plane
Z_	：	Workpiece coordinate value of Z axis of tapping position plane
A	：	Rotation angle between rectangular mesh integral holeposition and X axis
I_	：	Number of horizontal axis hole rows
J	：	Number of rows of longitudinal axis holes
U_	：	Spacing between horizontal axis holes
V_	：	Longitudinal shaft hole spacing
Q_	：	Tapping depth per feed
F_	：	Feed speed of tapping and cutting

Instruction description

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters.

(2) M29 and M28 instructions are required before and after the program, and the program is in rigid tapping mode.

(3) In G99 mode, return to R point according to cutting feed speed. In G98 mode, after returning to R point according to cutting feed speed, fast forward to return to the starting point position surface.

(4) The tapping method is G84.

(5) Pay attention to the relationship among R, W and Z to ensure processing safety.

(6) The following program is only compatible with G90 format.

(7) Scaling is temporarily incompatible. Please refer to the parameter range given by the instruction in Smart Editing to specify parameters.

19.18Straight Tapping（G164.3）

【Functions】：

Tapping of row holes by displacement tapping in the same direction.

【Instruction description】：

G164.3 X_ Y_ R_ Z_ A_ I_ U_ Q_ F_ ;		
X_	：	X-axis workpiece coordinate value of the starting hole positionin the lower left corner
Y_	：	Y-axis workpiece coordinate value of the starting hole position in the lower left corner
R_	：	Workpiece coordinate value of Z axis of tapping safety plane
Z_	：	Workpiece coordinate value of Z axis of tapping position plane
A	：	Rotation angle between rectangular mesh integral hole position and X axis
I_	：	Number of horizontal axis hole rows
U_	：	Spacing between horizontal axis holes
Q_	：	Tapping depth per feed
F_	：	Feed speed of tapping and cutting

Instruction description

(1) Please refer to the parameter range given by the instruction in Smart Editing to specify the parameters.

(2) M29 and M28 instructions are required before and after the program, and the program is in rigid tapping mode.

(3) In G99 mode, return to R point according to cutting feed speed. In G98 mode, after returning to R point according to cutting feed speed, fast forward to return to the starting point position surface.

(4) The tapping method is G84.

(5) Pay attention to the relationship among R, W and Z to ensure processing safety.

(6) The following program is only compatible with G90 format.

(7) Scaling is temporarily incompatible. Please refer to the parameter range given by the instruction in Smart Editing to specify parameters.

20.Automatic Tool Length Measurement（OPTION）

20.1Instruction format and parameter meaning

【Instruction description】：

G110 I_[R _][X _ W_][Z_][Q1.0][K1.0] ;

I_	：	Number of measuring tool
R_	：	Tool number of reference tool
X_	：	Offset of the center of the spindle in the X direction relative to the sensor
W_	：	Offset of the center of the spindle in the Y direction relative to the sensor
Z_	：	Allowable value of tool long offset, rewrite compensation value within allowable value (rewrite measured value when Z is omitted)
Q1	：	Rewriting of compensation is prohibited (rewriting when Q is omitted)
K1	：	Quick measurement, no repetition accuracy

Measure many times to eliminate the influence of chips on the tip of the tool and ensure that the measurement accuracy has no deviation;

Parameters I and R are mutually exclusive, that is, I and R cannot be specified at the same time;

Attention

LYNUC Automatic Tool Length Measurement System does not support multipletools.

20.2UIinterface setting

Automatically measure the mechanical coordinate value of sensor X (# 1993);

Automatically measure the mechanical coordinate value of sensor Y (# 1994);

Automatic measurement of mechanical coordinate value of sensor Z (# 1995);

Automatic measurement of maximum tool length (# 1946).

20.3Compensation type

(1) If the length of the standard tool is not 0, the non-standard tool length will be measured when the non-standard tool is automatically measured.

(2) If the length of the standard cutter is entered as 0 in the standard tool, the measured value is the difference from the standard cutter when the non-standard tool is automatically measured.

20.4Use example

T1M06 --------------Change the No.1 cutter to the spindle;

G110R1------------- Take No.1 cutter as standard tool for automatic measurement;

T2M06-------------- Change the No.2 cutter to the spindle;

G110I2--------------Take No.2 cutter as a non-standard tool for automatic measurement;

M30

Shanghai Lynuc CNC technology co., ltd.

Address: No.30-31, Lane 2338, Duhui Road, Minhang District, Shanghai, China
Zip Code: 201108
Tel: +86 21 618 37766
Fax: +86 21 607 20487
Website: http://www.lynuc.cn