In this user manual, we will try to describe various matters related to the operation of the system. Due to space limitations and specific use of the product, it is not possible to describe in detail all the operations that do not have to be done and/or cannot be done in the system. Therefore, matters not specifically specified in this user manual are considered "impossible" or "impermissible" to be done.

preface

Dear Customer:

For the products you choose, the company is deeply honored and grateful!

This user manual details the programming, operation and installation of the machining center's CNC system. In order to ensure the safe, normal and effective operation of the product, please be sure to carefully read this user manual before installing and using the product.

Security warnings

 Improper operation will cause accidents and must be qualified to operate the system.

Special note: The system power supply installed on the chassis (inside) is a special power supply provided only for the CNC system manufactured by the company. Users are prohibited from using this power supply for other purposes. Otherwise, there will be great danger!

Security warnings

■ Transportation and storage

1. The product packaging box stacking shall not exceed six layers;
2. Do not climb, stand or place heavy objects on the product box;
3. Do not use the cable connected to the product to drag or handle the product;
4. It is strictly forbidden to collide and scratch the panel and display;
5. Product packaging should be protected from moisture, exposure to sunlight and rain.

■ Check out of the box

1. After opening the package, please confirm whether it is the product you have purchased;
2. Check whether the product is damaged in transit;
3. Check whether the parts are complete and whether there is damage against the list;
4. If there is a product model inconsistency, missing accessories or shipping damage, please contact the company in time.

■ Wiring

1. The personnel participating in the wiring and inspection must be professionals with corresponding capabilities;
2. The product must be reliably grounded, the grounding resistance should be less than 0.1Ω, and the neutral wire (zero wire) cannot be used instead of the ground wire;
3. The wiring must be correct and firm, so as not to lead to product failure or unexpected consequences;
4. The surge absorption diode connected to the product must be connected in the specified direction, otherwise the product will be damaged;
5. The power supply to the product must be cut off before plugging or opening the product chassis.

■ Overhaul

1. The power supply must be cut off before overhauling or replacing components;
2. When a short circuit or overload occurs, the fault should be checked, and the troubleshooting can be restarted;
3. Do not frequently power off the product, if you must re-power after the

power outage, the interval time is at least 1min.

statement

The manual describes as many different things as possible, but because of the possibilities involved, it is not possible to explain all the actions that can or cannot be done, so what is not specifically stated in this manual can be considered unusable.

warn

Before the installation connection, programming and operation of this product, it is necessary to read the product manual and the instruction manual of the machine tool manufacturing plant in detail, and strictly follow the requirements of the manual and the instructions for related operations, otherwise it may lead to damage to the product, the machine tool, the scrap of the workpiece and even personal injury.

The product functions and technical indicators described in this manual (such as accuracy, speed, etc.) are only for this product, the CNC

machine tool of this product is installed, the actual functional configuration and technical performance are determined by the design of the machine tool manufacturer, and the functional configuration and technical indicators of the CNC machine tool are subject to the instructions of the machine tool manufacturer; Although the system is equipped with a standard machine tool operation panel, the function of each button of the standard machine tool panel is defined by the PLC program (ladder diagram). Please note that the function of the keys in the machine panel of this manual is described for the standard PLC program!

*** The contents of this manual are subject to change without notice.**

The safety responsibility of the manufacturer

- The manufacturer shall be responsible for the hazards that have been eliminated and/ or controlled in the design and structure of the CNC system provided and the accessories supplied with the accompanying supply.

——The manufacturer shall be responsible for the safety of the CNC system provided and the accessories supplied with the supply.

- The manufacturer shall be responsible for the use information and suggestions provided to the user.

User's responsibility for safety

——The user should learn and train the safe operation of the numerical control system, and be familiar with and master the content of the safe operation.

——The user shall be responsible for the safety and dangers caused by adding, changing or modifying the original CNC system and accessories.

- The user shall be responsible for the hazards caused by the failure to operate, adjust, maintain, install and store the product in accordance with the provisions of the manual.

*** This manual is for the end-user collection. Sincerely thank you for your friendly support to our company when our products!**

The first article is programming instructions

directory

The first one

Programming instructions

Chapter 1 Programming Basics

1.1 Introduction to milling machines

1.1.1 Product Brief

The system uses a 32-bit high-performance CPU and ultra-large-scale programmable device FPGA, real-time control and hardware interpolation technology to ensure high efficiency μm-level accuracy of the system, editable PLC makes the logic control function more flexible and powerful.

The system 1ms high-speed interpolation, 0.1μm control accuracy, significantly improve the efficiency, accuracy and surface quality of part processing.

Product features

- ✧ 1ms interpolation cycle, 0.1um control accuracy
- ✧ Straight, exponential and S types are available in a variety of acceleration and deceleration modes
- ✧ Built-in multi-PLC program, the currently running PLC program can be selected
- ✧ PLC program online display, real-time monitoring, signal real-time tracking
- ✧ Support for statement macro code programming, support for macro program calls with parameters
- ✧ Rigid and flexible tapping can be set by parameters
- ✧ It has rotation, scale, polar coordinates, fixed cycle and a variety of milling and slot composite cycle functions
- ✧ With historical alarm and operation history function, convenient for user operation and maintenance management
- ✧ Provides multi-level password protection for easy device management
- ✧ Support standard RS232 and USB interface, support U disk file operation, system configuration and software upgrade, can realize file transfer, serial port DNC processing and USB online processing functions

1.1.2 Technical Specifications

Feed axis function

- ✧ Position command range:

Metric input (G21): -9999.9999mm ~ 9999.9999mm, minimum command unit: 0.0001mm

Imperial input (G20): -999.9999inch ~ 999.9999inch, minimum command unit: 0.0001inch

- ✧ Electronic gear: command multiplier coefficient 1 ~ 65536, instruction divide factor 1 ~ 65536
- ✧ Fast movement speed: up to 60m/min
- ✧ Fast magnification: F0, 25%, 50%, 100% four-level real-time adjustment
- ✧ Cutting feed speed: up to 15 m/min (G94) or 500.00 mm/r (G95).
- ✧ Feed magnification: 0 ~ 150% 16 levels of real-time adjustment
- ✧ Manual feed rate: 0 ~ 150% sixteen-level real-time adjustment
- ✧ Handwheel feed: 0.001mm, 0.01mm, 0.1mm, 1mm four gears
- ✧ Single stepping feed: 0.001mm, 0.01mm, 0.1mm, 1mm four gears
- ✧ Interpolation methods: linear interpolation, circular interpolation, spiral interpolation and rigid tapping
- ✧ Automatic chamfering function

Acceleration and deceleration function

- ✧ Cutting feed: front acceleration and deceleration linear type, front acceleration and deceleration S type, post-acceleration and deceleration linear type, rear acceleration and deceleration index type
- ✧ Fast movement: front acceleration and deceleration linear type, front acceleration and deceleration S type, rear acceleration and deceleration linear type, rear acceleration and deceleration index type
- ✧ The system has a forward-looking function, up to 15 NC programs can be read in advance, so that the interpolation of small line segments is high-speed smooth, suitable for part processing mold processing
- ✧ The start speed, end speed and acceleration and deceleration time of acceleration and deceleration are set by parameters
- ✧ The manual mode and handwheel mode are post-acceleration and deceleration control, and the handwheel mode can choose the stop-and-go mode or the full operation mode.
- ✧ Quick Position selects straight or polyline positioning

Spindle function

- ✧ Spindle encoder: The number of encoder lines can be set (100 p/r ~ 5000p/r).
- ✧ The transmission ratio of the encoder to the spindle: (1 ~ 255):(1 ~ 255).
- ✧ Spindle magnification: 50% ~ 120% A total of 8 levels of real-time trimming
- ✧ Spindle constant linear speed control
- ✧ Tapping cycle, rigid tapping

Tool function

- ✧ Tool length compensation
- ✧ Tip radius compensation (type C).

Precision compensation

- ✧ Pitch error compensation: the number of compensation points, compensation interval, compensation origin can be set
- ✧ Reverse clearance compensation: The amount of reverse clearance of the machine can be compensated by fixed frequency or by way of lifting and decreasing speed

PLC function

- ✧ Two-stage PLC program with a processing speed of 1.5μs/step basic instructions; Up to 4700 steps, level 1 program cycle of 8ms
- ✧ PLC warnings and PLC alarms are supported
- ✧ Support for multi-PLC programs (up to 20) and selectable PLC programs currently running
- ✧ Number of instructions: 45 (including 10 basic instructions and 35 functional instructions).

Hmi

- ✧ Chinese, English and other languages display
- ✧ 2D tool trajectory display
- ✧ Real-time clock

Operations management

- ✧ Operation mode: editing, automatic, input, machine back to zero, handwheel / single step, manual, DNC
- ✧ Multi-level operation permission management
- ✧ Alarm logs

Program editing

- ✧ Program capacity: 56MB, can store up to 400 programs (including subroutines, macro programs).
- ✧ Editing function: program / program segment / word retrieval, modification, deletion, copy, paste
- ✧ Program format: ISO code, support statement macro code programming, support relative coordinate, absolute coordinate, polar coordinate programming
- ✧ Program calls: Macro program calls with parameters are supported, and level 4 subroutine nesting is supported

Communication capabilities

- ✧ RS232: Two-way transfer of parts programs, parameters and other files, support PLC

programs, system software serial port upgrades

- ✧ USB: U disk file operation, U disk file direct processing, support PLC program, system software U disk upgrade
- ✧ Support serial port DNC processing function and USB online processing function

Reliability and safety features

- ✧ Emergency stop
- ✧ Hardware stroke limit
- ✧ Software trip check
- ✧ Data backup and recovery

G Code Table

code	function	code	function
G00	Positioning (Fast Moving)	G51	Scale scaling
G01	Linear interpolation (cutting feed)	G53	Select the machine coordinate system
G02	Arc interpolation CW (clockwise)	G54	Select the workpiece coordinate system 1
G03	Arc interpolation CCW	G55	Select the workpiece coordinate system 2
G04	Suspend, quasi-stop	G56	Select the workpiece coordinate system 3
G12	Storage stroke detection function is turned on	G57	Select the workpiece coordinate system 4
G13	The storage stroke detection function is disconnected	G58	Select the workpiece coordinate system 5
G15	The polar directive is canceled	G59	Select the workpiece coordinate system 6
G16	Polar coordinates instruction	G54.1～G54. 48	Select an additional workpiece coordinate system
G17	XY plane selection	G60	One-way positioning
G18	ZX plane selection	G61	Quasi-stop method
G19	YZ plane selection	G62	Automatic corner multiplier
G20	Imperial data entry	G63	Tapping method
G21	Metric data entry	G64	Cutting method
G22	Rough milling of recesses in a counterclockwise circle	G65	Macro program instructions
G23	Rough milling of recesses in the circle clockwise	G68	Coordinate rotation
G24	Counterclockwise intracircle fine milling cycle	G69	Coordinate rotation is canceled
G25	Clockwise intra-circle fine milling cycle	G73	High-speed deep hole machining cycle
G26	Counterclockwise outer circle fine milling cycle	G74	Left-handed tapping cycle
G27	Returns reference point detection	G76	Fine boring cycle

G28	Returns the reference point	G80	The fixed loop is canceled
G29	Returns from the reference point	G81	Drilling cycle (point drilling cycle)
G30	Returns 2, 3, and 4 reference points	G82	Drilling cycle, countersinking cycle
G31	Jump function	G83	Chip evacuation drilling cycle
G32	Clockwise outer circle fine milling cycle	G84	Right-hand tapping cycle
G33	Counterclockwise rectangular groove rough milling	G85	Bore cycle
G34	Rough milling clockwise rectangular grooves	G86	Bore cycle
G35	Counterclockwise rectangular groove fine milling cycle	G87	Back boring cycle
G36	Clockwise rectangular groove within the fine milling cycle	G88	Bore cycle
G37	Counterclockwise rectangular external fine milling cycle	G89	Bore cycle
G38	Clockwise rectangular external fine milling cycle	G90	Absolute value programming
G39	Corner offset arc interpolation	G91	Incremental value programming
G40	Tool radius compensation is canceled	G92	Floating coordinate system settings
G41	Left tool radius compensation	G94	Per cent feed
G42	Right tool radius compensation	G95	Per feed feed
G43	Positive tool length compensation	G96	Constant cycle speed control (cutting speed)).
G44	Negative tool length compensation	G97	Constant cycle speed control cancellation (cutting speed).
G49	Tool length compensation is canceled	G98	Returns the initial plane
G50	Scale zoom canceled	G99	Return to the R point plane

PLC Instruction Sheet

The instruction code	function	The instruction code	function	The instruction code	function
RD	Read normally open contacts	SPE	The subroutine ends	DIFU	Rising edge detection
ED. NOT	Read normally closed contacts	SET	Assertion	DIFD	Descending edge detection
WRT	Output coil	RST	reposition	COMP	Binary number comparison
WRT. NOT	The output coil is reversed	JMPB	Label jump	COIN	Consistency comparison
AND	Normally open contacts are connected in series	LBL	grade	MOVN	Data transfer
AND. NOT	Normally closed contacts are connected in series	TMR	timer	MOVB	One byte transfer
OR	Normally open contacts are connected in parallel	TMRB	Fixed timer	HOW	Two-byte transfer
OR. NOT	Normally closed contacts are connected in parallel	TMRC	Any total address timer	XMOV	Binary variable address number transfer
OR. STK	Paralleling of circuit blocks	CTR	Binary counter	DSCH	Binary data search
AND. STK	Series connection of circuit blocks	DEC	Binary decoding	ADD	Binary addition
END1	The first-level sequential procedure ends	COD	Binary code conversion	SUB	Binary subtraction
END2	The first-level sequential procedure ends	.COM	Common line control	ANDF	Logic with
CALL	Call the subroutine	COME	The public line control ends	ORF	Logical or
CALLU	Unconditionally call the subroutine	ROT	Binary rotation control	NOT	Logical non
SP	Subroutines	SFT	The register is shifted	EOR	XOR

1.1.3 Adaptability to climate and environment

The environmental conditions for storage, transportation, and working are as follows:

project	Working climatic	Storage and transportation
Ambient temperature	0℃～45℃	-40℃～+70℃
relative humidity	≤ 90% (non-condensing).	≤95%（40℃）
Atmospheric pressure	86 kPa～106 kPa	86 kPa～106 kPa
altitude	≤ 1000M	≤ 1000M

1.1.4 Power adaptability

The system operates normally under the following AC input power supply conditions. Voltage change: within the range of (0.85~1.1)× rated AC input voltage (AC220V); Frequency change: 49Hz ~ 51Hz continuous change.

1.1.5 Protection

Protection class not less than IP20.

1.2 Running of the program

1.2.1 The order in which the programs run

The currently open program must be in automatic mode, and the system cannot have 2 or more programs open at the same time, so the system can only run one program at any one time. When you open a program, the cursor is at the beginning of the line of the first program segment, and you can move the cursor under edit mode. In the operation stop state of the automatic operation mode, the operation of the program is started from the program segment where the current cursor is located with the cyclic start signal (the key of the machine panel or the 

external loop start signal) is used to start the operation of the program, usually in accordance with the order in which the program segment is written, and usually executed one by one in the order in which the program segment is written, until M02 or M02 is executed M30 code, the program stops. The cursor moves as the program runs. The order or state in which the program runs changes in the following cases:

- ✧ When the program runs, the key or the E-STOP button is pressed, and the program is terminated; 

- ✧ A CNC alarm or PLC alarm is generated when the program is running, and the program operation is terminated;

- ✧ The operation mode of the program running is switched to the input and editing operation mode, the program runs a single segment to stop (after running the current program segment, the program operation is suspended), switches to the automatic operation mode, and then presses the key or external loop to start the operation of the program when the signal is turned on, and the program is started from the program segment where the current cursor is located; 

- ✧ The operation mode of the program is switched to another mode of operation, and the program operation stops;
- ✧ When the program is running and the key is pressed or the external pause signal is disconnected, the program operation is suspended, and then the key or external loop start signal is turned on, the program continues to run from the stopped position; 

- ✧ When the single-segment switch is turned on, the program operation is suspended after the operation of each program segment, and when the key or external loop start signal is turned on, the operation continues from the next program segment; 

- ✧ The program section jump switch is turned on, and the program segment with "/" in front of the program segment is skipped and not executed;
- ✧ When executing the G65 jump code, go to the jump target program segment to run;
- ✧ When executing M98 code, the corresponding subroutine or macro program is called to run; The subprogram or macro program finishes running, and when the M99 code is executed, the next program segment of the calling program segment in the main program is returned (if the M99 code specifies the target program segment number to be returned, go to the target program segment to run);
- ✧ When M99 code is executed in the main program (the operation of which was not started by a call to another program), the first segment of the returning program continues to run, and the current program runs repeatedly in a loop.

1.2.2 The order in which code words are executed within a program segment

A program segment can have G, X, Z, F, R, M, S, T Most of the M, S, and T code words are interpreted by NC and sent to the PLC for processing, and other code words are directly processed NC processing. M98, M99, and the S-code words given the spindle speed in r/min and m/min are also directly derived from the NC Dispose.

NC finishes executing G when the G code is in the same program segment as M00, M01,

M02, M30 After the code, the M code is executed and the corresponding M signal is sent to the PLC for processing.

When the G code word is in the same program segment as the M98 and M99 code words, NC executes the G code before executing these M code words (do not send M signal to the PLC).

When the G code word is in the same program segment as other M, S, T code words processed by the PLC, the PLC program (ladder diagram) determines M , S, T code words and G code words are executed at the same time, or after the G code is executed , S, T code words, the order of execution of the relevant code words should be subject to the instructions of the machine tool manufacturer.

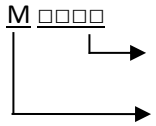
M00, M01, M02, M30 execute after the execution of other code in the current program segment.

Note: Standard PLC executes M ST order, with priority execution of T and then to M code and finally to S code, and will execute both G code and M ST code at the same line as G code.

Chapter 2 MST Codes

2.1 M Code (Accessibility).

The M code consists of the code address M and the 1 to 2 digits or 4 digits following it, which is used to control the process of program execution or to output the M code to the PLC 。



M89、M90、M91、M92、M93、M98、M99 、M9000 ~M9999 is handled independently by the NC and does not output M code to the PLC.

M02, M30 have been defined by NC as the program end code, but also output M code to the PLC, can be used by the PLC program for input and output control (off spindle, off cooling, etc.).

M98 and M99 as program call codes, M02 and M30 as program end codes, PLC programs cannot change the meaning of the above codes. Other M codes are output to the PLC, the PLC program defines the code function, please refer to the machine manufacturer's instructions.

There can be only one M code in a program segment, and the CNC appears with an alarm when two or more M codes appear in the program segment.

Table 2-1 List of **process M codes that control program execution**

code	function
M02	The program ends
M30	The program ends
M89	Customize the output (refer to the "Debug Use Documentation" for details).
M90	The program segment ends repeatedly (refer to the "Debug Use Document" for details).
M91	The program segment starts repeatedly (refer to the "Debug Use Document" for details).
M92	Conditional redirects (refer to the Debug Usage Document for details).
M93	Unconditional jump (refer to the "Debug Use Documentation" for details).
M98	A subroutine call

M99	Returns from subroutine; If M99 is used for the end of the main program (i.e. the current program is not called by another program), the program is executed repeatedly

2.1.1 Program ends M02

Code format: M02 or M2

Code function: In the automatic mode, the execution of M02 code, the other code of the current program segment is completed, the automatic run ends, the number of machining parts plus 1, cancel the tip radius compensation, the cursor returns to the beginning of the program (whether to return the beginning of the program is determined by the parameter).

2.1.2 Program runs end of M30

Code format: M30

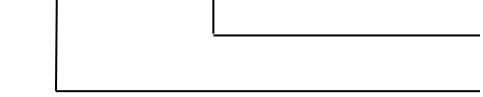
Code function: In the automatic mode, the execution of M30 code, the other code of the current program segment is completed, the automatic run ends, the number of processed parts plus 1, cancel the tip radius compensation, the cursor returns to the beginning of the program (whether to return the beginning of the program is determined by the parameter).

When the CNC status parameter NO.005 is set to 0 BIT4, the cursor does not return to the beginning of the program; When BIT4 for CNC status parameter NO.005 is set to 1, the program finishes executing and the cursor immediately returns to the beginning of the program.

2.1.3 The subroutine calls M98

Code format:

M98 Poooo L□□□□



Number of repeated calls

The subroutine number of the invoked

Code function: In automatic mode, when executing M98 code, after the execution of other code in the current program segment is completed, the CNC calls to execute the subprogram specified by P, and the subroutine can execute up to 9999 times. M98 code does not run under MDI.

2.1.4 Returns M99 from a subroutine

Code format:

M99 Poooo

Returns the program segment number (0000~9999) where the main program will be executed
Leading 0 can be omitted.

Code function: After the execution of other code in the current program segment is completed, the program segment specified by P in the main program continues execution, and when P is not entered, the next program segment in the main program that calls the current subprogram M98 code continues execution. If M99 is used for the end of the main program (that is, the current program is not executed by another program call), the current program is executed repeatedly. M99 code runs invalidally under MDI.

The system can call the quadruple subprogram, that is, other subroutines can be called in the subprogram (Figure 2-3).

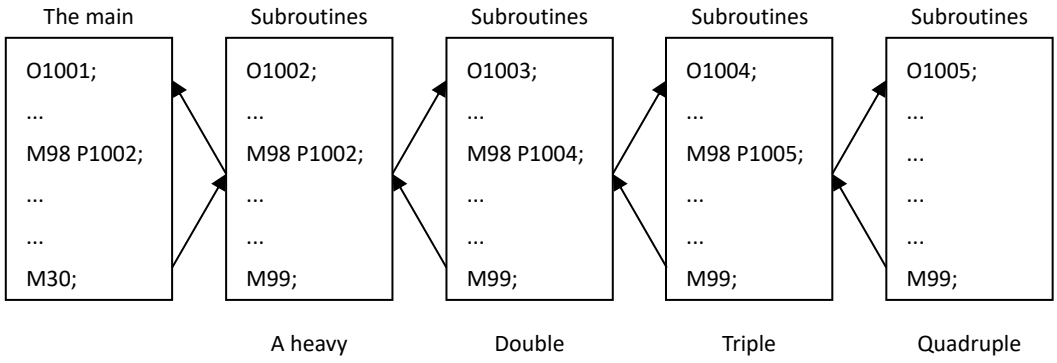


Figure 2-3 Subprogram nesting

Note 1: M99 uses loops (infinite loops) that can implement the main program when the main program is used;

Note 2: When M99 P_ is used in the main program, the main program jumps to the P_ specifies line number N;

For example:

N10
N20
N30 M99 P50
N40
N50
N60

N10
N20
N30
N40
N50 M99 P30
N60

;

2.1.5 M codes defined by standard PLC ladders

Except for the above codes (M02, M30, M89, M 90, M 91, M 92, M 93 , M98, M99), other M codes are defined by the PLC. The following is the M code defined for the standard PLC, milling machine CNC for machine tool control, the function, significance, control timing and logic of the M code please refer to the description of the machine tool manufacturer.

The M code defined by the standard **PLC** ladder diagram

Code M	definition	remark
M00	Unconditional suspension	The program pauses when it reaches M00
M01	Select Pause to stop	The program pauses when it runs to M01 and the system has the opt-out function turned on, otherwise it runs down
M02	The program ends	
M03	The spindle is spinning	Interlock, boot default M05
M04	The spindle is reversed	
*M05	The spindle stops	
M06	Automatic tool change	
M07	Process blowing	M07 and M 09 interlock, M 08 and M 09 interlock, boot default M 09
M08	Coolant	
*M09	Turn off the processing blowing/cooling	
M10	Table clamping	Interlock
M11	The workbench is released	
M16	The spindle tool is released	Interlock, boot default M17
*M17	Spindle tool clamping	
*M18	Spindle positioning is canceled	Interlock
M19	Spindle positioning output	
M20	retain	Reserved M codes are invalid function codes
M21	The magazine is forward/the holder is vertical	Can not be used alone, need to be used with the tool magazine, written in the tool change program, to be after the M50 can be used normally.
M22	Tool magazine looking for knives	

M23	Knife arm buckle knife	
M24	Tool magazine rewind/sleeve level	
M25	The knife arm rotates	
M26	The knife arm returns to the original point	
M27	retain	Reserved M codes are invalid function codes
*M28	Rigid tapping is canceled	Interlock
M29	Rigid tapping opens	
M30	The program ends	
M31	The number of machined parts plus 1	
M32	Lubrication output	Interlock
*M33	Lubrication shuts off	
M34	Protective door output	Interlock
*M35	The protective door is closed	
M36	Chip evacuator output	Interlock
*M37	The chip evacuator is turned off	
M38	Work light output	Interlock
*M39	The work light is off	
M40	retain	Reserved M codes are invalid function codes
M41	Spindle shift 1 gear	Interlock
M42	Spindle shift 2nd gear	
M43	Spindle shift in 3 gears	
M44	Spindle shift 4 gears	
M45	retain	Reserved M codes are invalid function codes
M46		
M47		
M48		
M49		
M50	The knife change begins	Interlock
*M51	End of tool change	
M52	retain	Reserved M codes are invalid function codes
M53		

M54		
M55		
M56		
M57		
M58		
M59		
M60		
M61		
M62		
M63	The second spindle is spinning	Interlock, boot default M65
M64	The second spindle is reversed	
*M65	The second spindle stops	
M66	retain	Reserved M codes are invalid function codes
M67		
M68		
M69		
M70	Blow-out to knife	Interlock
*M71	Blow off the knife	
M72	Customize theK2 output	Interlock, lead screw clamping
*M73	Custom K2 off	
M74	Customize theK3 output	Interlock
*M75	Custom K3 off	
M76	retain	Reserved M codes are invalid function codes
M77		
M78	Recovery feed magnification 100%.	Interlock
M79	Restore the original feed rate	
M80		
M81		
M82		
M83		
M84		
M85		
M86		
M87		
M88		
M89	Customize the output	Multi-format programming

M90	The program segment ends repeatedly	Interlock, using M91 C_ needs to end with M90
M91 C	The program segment repeats to begin	
M92	Conditional jump	
M93	Unconditional jump	
M94	retain	Reserved M codes are invalid function codes
M95		
M96		
M97		
M98	Call the subroutine	When the L number of times is not written, it is called 1 time by default
M99	The subroutine ends	When you write the main program, the program is an infinite loop
M9000—M9999		A single call to the program O9000-O9999

NOTE: The code with the standard PLC defined with the "*" is valid at power-up.

2.1.6 Program stops M00

Code format: M00 or M0

Code function: After executing the M00 code, the program stops running, the word "Pause" is displayed, and the program continues to run after pressing the cyclic start key.

2.1.7 Program Opt Out M01

Code format: M01 or M1

Code function: in the automatic, entry mode is effective, press the select stop key to make the select stop 

button indicator light, it means that the select stop state, at this time after the execution of the M01 code, the program runs stopped, the word "pause" is displayed, press the cycle start key, the program continues to run. If the program opt-in switch is not turned on, the program does not pause even if you run the M01 code.

2.1.8 Counterclockwise, clockwise and spindle stop control M03, M04 and M05

Code format: M03 or M3;

M04 or M4;

M05 or M5;

Code function: M03: Counterclockwise;

M04: clockwise;

M05: Spindle stops.

Note: The control timing and logic of M03, M04 and **M05 defined** by the standard **PLC are detailed in the "Debug Documentation".**

2.1.9 Cooling pump/processing blow control M07, M08, M09

Code format: M07 or M7;

M08 or M8;

M09 or M9;

Code function: M07: Processing blow air open

M08: cooling pump open;

M09: Cooling pump/processing blow off.

Note: The control timing and logic of M07, M08 and M09 defined by the standard **PLC are detailed in the "Debug Use Document".**

2.1.10 Workbench control M 10, M11

Code format: M10;

M11；

Code function: M10: Table clamping

M11: Table loosened;

Note: The control timing and logic of M10 and M11 as defined by the standard **PLC are detailed in the "Debug Use Documentation".**

2.1.11 Tool control M16, M17

Code format: M16;

M17；

Code function: M16: spindle tool release;

M17: Spindle tool clamping.

Note: The control timing and logic of M16 and M17 as defined by the standard **PLC are detailed in the "Debug Use Documentation".**

2.1.12 Spindle orientation M18, M19

Code format: M18;

M19；

Code function: M18: spindle orientation cancellation;

M19: Spindle orientation.

Note: The control timing and logic of M 18 and M19 defined by the standard **PLC are detailed in the "Debugging Documentation".**

2.1.13 Rigid tapping M28, M29

Code format: M28;

M29；

Code function: M28: Rigid tapping canceled;

M29: Rigid tapping.

Note: The control timing and logic of M28 and M29 defined by the standard **PLC are detailed in the "Debug Use Documentation".**

2.1.14 Lubricating fluid control M32, M33

Code format: M32;

M33；

Code function: M32: lubrication pump open;

M33: Lubrication pump off.

Note: The control timing and logic of M28 and M29 defined by the standard **PLC are detailed in the "Debug Use Documentation".**

2.1.1 5 Spindle automatic shift M41, M42, M43, M44

Code format: M4n; (n=1、2、3、4)

Code function: When M4n is executed, the spindle is changed to the nth gear

Note: The control timing and logic of M 4n as defined by the standard **PLC are detailed in the "Debug Use Documentation".**

2.1.16 Protective door control M34, M35

Code format: M34;

M35；

Code function: M34: protective door open;

M35: Protective door closed.

Note: The control timing and logic of M34 and M35 as defined by the standard **PLC are detailed in the "Debug Use Documentation".**

2.1.17 Chip evacuation control M36, M37

Code format: M36;

M37；

Code function: M36: chip removal open;

M37: Chip Off.

Note: The control timing and logic of M36 and M37 defined by the standard **PLC are detailed in the "Debug Use Documentation".**

2.1.18 Work light control M38, M39

Code format: M38;

M39；

Code function: M38: work light on;

M39: The working light is off.

Note: The control timing and logic of M38 and M39 defined by the standard **PLC are detailed in the "Debug Use Documentation".**

2.1.19 The second spindle controls M63, M 64, M65

Code format: M63;

M64；

M65；

Code function: M63: 2nd spindle forward rotation;

M64: 2nd spindle reversal;

M65: 2nd spindle stops

Note: The control timing and logic of M 63, M64 and **M65 defined** by the standard **PLC are**

detailed in the "Debug Use Documentation".

2.1.20 Custom output control M70-M77

Note: The control timing and logic of M 70-M77 as defined by the standard **PLC are detailed in the "Debug Use Documentation".**

2.2 Spindle function

S code is used to control the spindle speed, the system controls the spindle speed in two ways: spindle speed switching control mode: S □□ (2-digit code value) code is processed by the PLC , PLC Output the switching signal to the machine tool to achieve a stepwise change in the spindle speed. Spindle speed analog voltage control mode: S □□□□ (4-digit code value) specifies the actual spindle speed, NC output 0 ~ 10V The analog voltage signal is given to the spindle servo device or frequency converter to achieve stepless speed regulation of the spindle speed.

2.2.1 Spindle speed switching control

When the status parameter NO.001 BIT4 is set to 0, the spindle speed is controlled by the switching quantity. A program segment can have only one S code, and when two or more S codes appear in the program segment, the CNC appears with an alarm.

When the S code is co-segmented with the code word for performing the mobile function, the order of execution is defined by the PLC program, please refer to the instructions of the machine manufacturer.

When the spindle speed switch is controlled, the system is used for machine tool control, and the timing and logic of S code execution should be subject to the instructions of the machine tool manufacturer. The S codes defined below for the standard PLC for milling machine systems are for reference only.

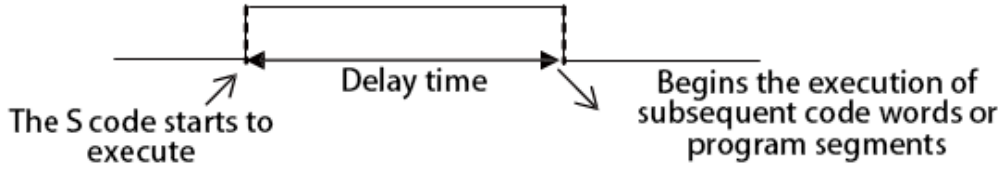
Code format:

S □□

00 ~ 04 (leading zero can be omitted): 1 ~ 4 spindle speed switch control.

In the spindle speed switching control mode, after the code signal of the S code is sent to the PLC, the FIN signal is returned after delaying the time set by the PLC parameter T006, which is

called at this time The execution time of the S code.



When the CNC is reset, the output status of S01, S02, S03, and S04 remains unchanged.

The S1 ~ S4 output is invalid when the CNC is powered up. Execute any of the code S01, S02, S03, S04, and the corresponding S signal output is valid and maintained, while the remaining 3 are canceled Output of an S signal. When executing the S0 5-S08 code, the output of S1 ~ S4 is canceled, and only one valid S1 ~ S4 is at the same time.

2.2.2 Spindle speed analog voltage control

When the status parameter NO.001 BIT4 is set to 1, the spindle speed is controlled by an analog voltage.

Code format:

S □□□□

0000 ~ 9999 (leading 0 can be omitted): Spindle speed analog voltage control

Code function: set the spindle speed, CNC output 0V ~ 10V analog voltage control spindle servo or inverter, to achieve the spindle stepless variable speed, S code value power loss does not remember, power-up time 0。

When the spindle speed simulation voltage control function is effective, there are 2 ways to input the spindle speed: the S code sets the fixed speed of the spindle (r/min), and the spindle speed is constant when the S code value does not change, which is called constant speed control (G97 mode); S code sets the tangent speed (m/min) of the tool relative to the outer circle of the workpiece, which is called constant linear speed control (G96 mode), under the constant linear speed control mode, The spindle speed at the time of cutting feed changes with the absolute value of the X-axis absolute coordinate value of the programmed trajectory. See section 2.2.3 of this chapter for details.

CNC has a four-speed spindle mechanical gear function, when executing the S code, according to the current spindle gear of the highest spindle speed (output analog voltage is 10V) set value (corresponding to the data parameter NO.210 ~ NO.213) to calculate the corresponding analog voltage value for a given speed , and then output to the spindle servo or inverter, the

actual speed of the control spindle is consistent with the required speed.

When the CNC is powered up, the analog voltage output is 0V, and after the S code is executed, the output analog voltage value remains unchanged (unless it is in the cutting feed state controlled by constant linear speed and the absolute value of the X-axis absolute coordinate value changes). After S0, the analog voltage output is 0V. When the CNC is reset or stopped, the analog voltage output remains unchanged.

2.2.3 Spindle magnification

When the spindle speed simulation voltage control mode is effective, the actual speed of the spindle can be modified with the spindle magnification, and the actual speed after the spindle magnification is adjusted is limited by the highest speed of the current gear of the spindle, and the minimum spindle speed limit value and the maximum spindle speed limit value are also limited by the constant linear speed control mode.

NC provides 8 stages of spindle magnification (50% ~ 120%, each stage change of 10%), the actual number of spindle magnification, trimming method, etc. are defined by the PLC ladder diagram, and the use should be subject to the instructions of the machine tool manufacturer. The following is a functional description of the standard PLC ladder diagram of a milling machine system and is for reference only.

The spindle magnification defined by the standard PLC ladder diagram of the milling machine system has a total of 8 stages, and the actual speed of the spindle can be adjusted in real time with the spindle magnification adjustment key in the range of 50% to 120% of the command speed, and the spindle magnification is remembered for power-down. The spindle magnification adjustment operation is detailed in the user manual "Operating Instructions Manual".

2.2.4 Constant surface speed control G96/G97

Code format: Constant surface speed control G96 S_ surface speed m m/min

Constant surface speed control eliminates G97 S_ spindle speed r/min

Constant surface speed control controlled axis code G96 P_ P1:X axis P2:Y axis P3:Z axis P4:4TH axis

Maximum speed clamping G92 S_ S specifies the maximum speed r/min

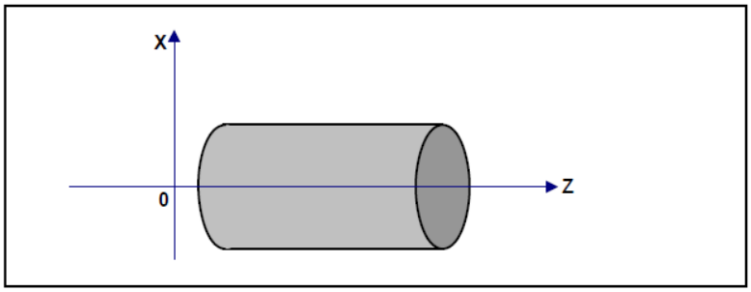
Function: Specifies the surface speed (the relative speed between the tool and the workpiece) after S, and the spindle rotates so that the surface cutting speed remains constant, regardless of the tool position.

1, G96 is a modal code, after the instruction G96, the program enters the constant speed

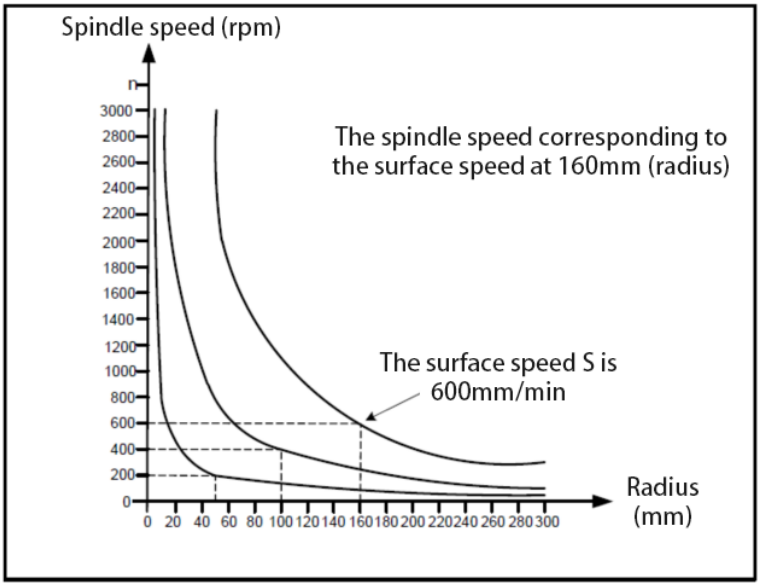
control mode, S value is the surface speed.

2. G96 code must specify the axis, along which constant speed control is used. G97 code cancels the G96 way.

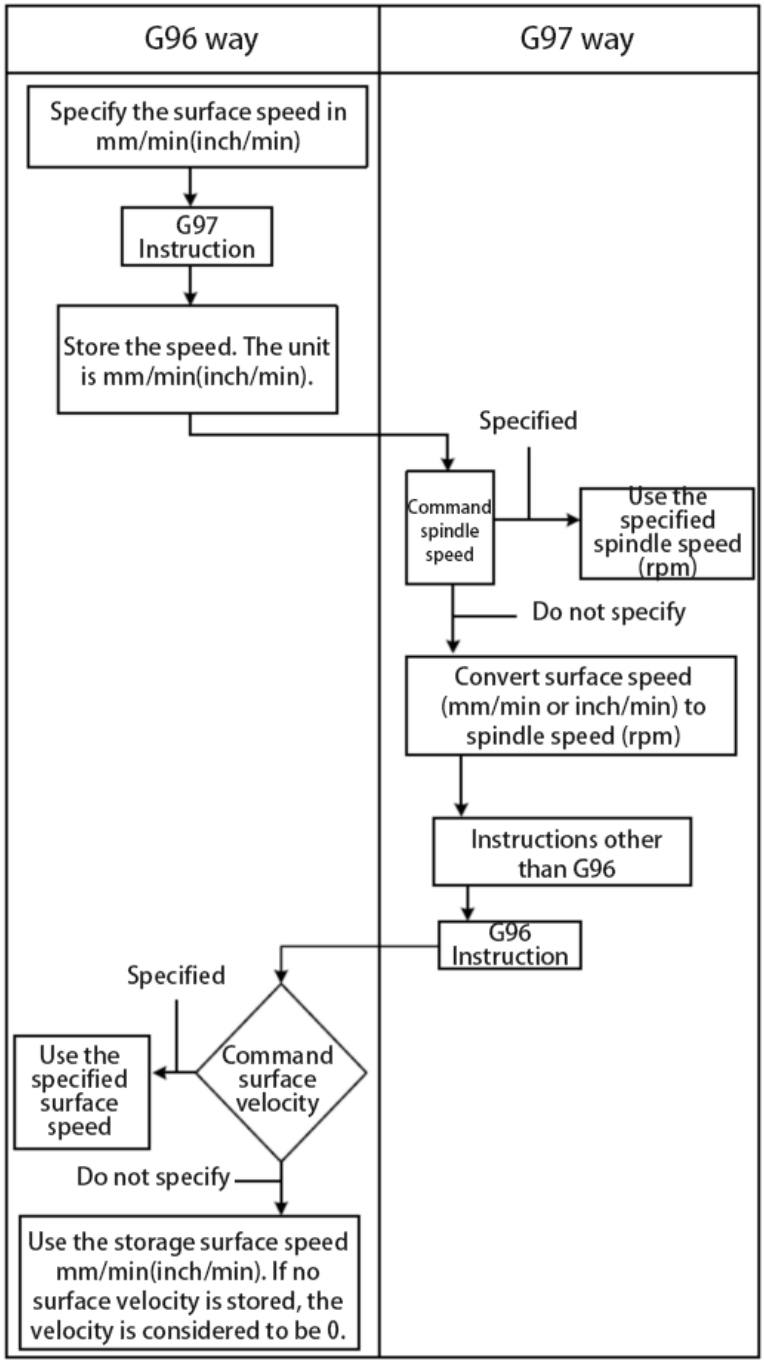
3. In order to perform constant surface cutting speed control, it is necessary to set the coordinate system of the workpiece so that the central coordinate of the rotation axis becomes zero.



4. When using constant surface cutting speed control, when it is higher than the G92 S_ setting, it is clamped on the highest spindle speed. When the maximum spindle speed has not been set when the power is turned on, S in the G96 code is treated as S=0 until M3 or M4 appears in the program.



5. Specify the surface cutting speed in the G96 mode:



6, G96 related parameter settings: bit parameter 5 7.4 set G0 fast positioning to calculate the reference coordinates of G96 spindle speed (0: end point, 1: current point), bit parameter 5 7.5 set G96 spindle speed clamping (0: before spindle magnification, 1: after spindle magnification), bit parameter 42.0 Set whether to use constant cycle speed control.

Restrictions:

1. Because when the spindle speed changes, the response problem in the servo system is not considered, and during the thread cutting, the constant surface cutting speed control is also effective. Therefore, the constant surface cutting speed is eliminated with G97 before threading.

2. In the fast moving program section specified by G00, the constant surface speed control is not calculated according to the instantaneous change of the tool position, but the surface speed calculated according to the end point of the segment, because the fast movement is not cut, therefore, the constant surface cutting speed is not used.

3. When processing such as flexible tapping, rigid tapping or deep hole rigid tapping, it is necessary to use G97 to cancel the constant surface cutting speed, otherwise there will be chaotic teeth or broken taps.

2.3 Tool function

Specify a numeric value (up to 8 digits) after address T to select the tool on the machine.

In principle, you cannot command more than two T codes in the same program segment, and if the same set of codes is set in the same section, no alarms are raised.

The T code that appears later shall prevail, and the number of digits that can be specified for the address T and the corresponding machine action of the T code can be found in the instruction manual of the machine tool factory.

When mobile code and T code are specified in the same program segment, there are two ways to execute the code:

1. Mobile code and T code are executed at the same time.

2. Execute the T code after the mobile code is executed.

When the T code and the tool change code M06 are the same paragraph, the T code is executed first and then the tool change code is executed. When the T code and the tool change code M06 are different segments, M06 executes the T code specified by the previous program.

The following example procedure:

O00010；

N10 T2M6； The knife on the spindle is a T2 knife

N20 M6T3； The knife on the spindle is T3 knife

N30 T4； The knife on the spindle is T3 knife

N40 M6； The knife on the spindle is T4 knife

N50 T5； The knife on the spindle is T4 knife

N60 M30

%

After performing the tool change procedure, the knife on the spindle is T4 knife.

Chapter 3 G Code

3.1 Prepare the types of function G codes

The preparation function is indicated by the G code and followed by a number, which specifies the meaning of the program segment in which it is located. There are two types of G codes:

Table 3-1-1

class	significance
Non-modal G code	Valid only in the program segment being ordered
Modal G code	Valid until other G-code instructions in the same group

(Example) G01 and G00 are modal G codes of the same group

G01 X __;

Z _____; G01 is valid

X _____; G01 is valid

G00 Z ____; G00 is valid

Note: For specific system parameters, please refer to the system parameter table

Table 3-1-2 G Code and Function Sheet

Code G	Constituencies	Instruction form	function
*G00	01	G00 X_ Y_ Z_	Positioning (fast moving)
G01		G01 X_ Y_ Z_ F_	Linear interpolation (cutting feed)
G02		G02/G03 X_ Y_ { R_ / F_ / I_ J_ }	Arc interpolation CW (clockwise).
G03			Arc interpolation CCW (counterclockwise).
G04	00	G 04P_ or G04 X/U_	Suspend, quasi-stop
G10		G10 L_ N_ P_ R_	Programmable data entry
*G11		G11	Programmable data entry cancellation
G12	16	G12 X_ Y_ Z_ I_ J_ K_	Storage stroke detection function is turned on
*G13		G13	The storage stroke

				detection function is disconnected
*G15	11	G15		The polar directive is canceled
G16		G16		Polar coordinates instruction
*G17	02	Written in the program segment for arc interpolation and tool radius compensation.		XY plane
G18				ZX plane
G19				Y-Z plane
G20	06	Must be specified in a separate program segment at the beginning of the program, before the coordinate system is set.		Imperial data entry
*G21				Metric data entry
G22	09	G22 X_Y_Z_R_I_L_W_Q_V_D_F_K_		Rough milling of recesses in a counterclockwise circle
G23		G23 X_Y_Z_R_I_L_W_Q_V_D_F_K_		Rough milling of recesses in the circle clockwise
G24		G24 X_Y_Z_R_I_J_D_F_K_		Counterclockwise full circle internal fine milling cycle
G25		G25 X_Y_Z_R_I_J_D_F_K_		Clockwise full circle internal fine milling cycle
G26		G26 X_Y_Z_R_I_J_D_F_K_		Counterclockwise outer circle fine milling cycle
G27	00	G27	X_Y_Z_	Returns reference point detection
G28		G28		Returns the reference point
G29		G29		Returns from the reference point
G30		G30Pn		Returns 2, 3, and 4 reference points
G31		G31		Jump function
G32	09	G32 X_Y_Z_R_I_J_D_F_K_		Clockwise outer circle fine milling cycle
G33		G33 X_Y_Z_R_I_J_L_W_Q_V_U_D_F_K_		Counterclockwise rectangular groove rough milling
G34		G34 X_Y_Z_R_I_J_L_W_Q_V_U_D_F_K_		Rough milling clockwise rectangular grooves
G35		G35 X_Y_Z_R_I_J_L_U_D_F_K_		Counterclockwise rectangular groove fine milling cycle

G36		G36 X_Y_Z_R_I_J_L_U_D_F_K_			Clockwise rectangular groove within the fine milling cycle
G37		G37 X_Y_Z_R_I_J_L_U_D_F_K_			Counterclockwise rectangular external fine milling cycle
G38		G38 X_Y_Z_R_I_J_L_U_D_F_K_			Clockwise rectangular external fine milling cycle
G39	00	G39 I_J_； I_K_； J_K_； or G39			Corner offset arc interpolation
*G40	07	G17	G40 G41 G42	X_Y_	Tool radius compensation is canceled
G41		G18		Z_X_	Left tool radius compensation
G42		G19		Y_Z_	Right tool radius compensation
G43	08	G17	G43 G44 G49	H_Z_	Positive tool length compensation
G44		G18		H_Y_	Negative tool length compensation
*G49		G19		H_X_	Tool length compensation is canceled
*G50	12	G50			Scale zoom canceled
G51		G51 X_Y_Z_P_			Scale scaling
G53	00	Write in the program			Select the machine coordinate system
*G54	05	Write in the program, usually at the beginning of the program			Workpiece coordinate system 1
G55					Workpiece coordinate system 2
G56					Workpiece coordinate system 3
G57					Workpiece coordinate system 4
G58					Workpiece coordinate system 5
G59					Workpiece coordinate system 6
G60	00	G60 X_Y_Z_			One-way positioning
G61	14	G61			Quasi-stop method
G62		G62			Automatic corner multiplier
G63		G63			Tapping method

*G64		G64	Cutting method
G65	00	G65 H_P#i Q#j R#k	Macro program instructions
G68	13	G68 X_Y_R_	Coordinate rotation
*G69		G69	Coordinate rotation is canceled
G73	09	G73 X_Y_Z_R_Q_F_	High-speed deep hole machining cycle
G74		G74 X_Y_Z_R_P_F_	Left-handed tapping is broken
G76		G76 X_Y_Z_R_P_Q_F_K_	Fine boring cycle
*G80		Writes to other programs in a program segment	The fixed loop is canceled
G81		G81 X_Y_Z_R_F_	Drilling cycle (point drilling cycle)
G82		G82 X_Y_Z_R_P_F_	Drilling cycle (countersinking cycle).
G83		G83 X_Y_Z_R_Q_F_	Chip removal processing cycle is bad
G84		G84 X_Y_Z_R_P_F_	Right-hand tapping cycle
G85		G85 X_Y_Z_R_F_	Bore cycle
G86		G86 X_Y_Z_R_F_	Bore cycle
G87		G87 X_Y_Z_R_Q_P_F_	Back boring cycle
G88		G88 X_Y_Z_R_P_F_	Bore cycle
G89		G89 X_Y_Z_R_P_F_	Bore cycle
*G90	03	Write in a program segment	Absolutely programming
G91			Incremental programming
G92	00	G92 X_Y_Z_	Floating coordinate system settings
*G94	04	G94	Per cent feed
G95		G95	Per feed feed
G96	15	G96 S_	Constant cycle speed control (cutting speed).
*G97		G97 S_	Constant cycle speed control cancellation (cutting speed).
*G98	10	Write in a program segment	Returns the starting plane in a fixed loop
G99			Returns the R plane in a fixed loop

Note: 1. If the modal instruction is in the same **segment as the non-modal instruction, the non-modal**

instruction takes precedence, and the corresponding modality is changed according to other modal instructions in the same paragraph, but they are not executed.

2, with the * mark G code, when the power is turned on, the system is in the state of this G code. The G codes of the 3 and 00 groups are non-modal G codes except G10, G11, and G92 .

4. If the G code not listed in the G code list is used, the alarm appears, or the G code that does not have the selection function is commanded, and the alarm is also indicated.

5, in the same program segment can be instructed several different groups of G code, in principle, can not be in the same program segment instructions more than two of the same group of G code, if the same set of instructions in the same section does not alarm, then the G code that appears later shall prevail.

6. When Group 01 and Group 09 G instructions are the same, Group 01 shall prevail. In fixed loop mode, if the G code of group 01 is commanded, the fixed loop is automatically canceled and becomes the G80 state.

7. G codes are represented by each group number according to different types. Set by the bit parameters NO: 46#1 ~ 7 and NO: 47#0 ~ 7 to set whether to clear each group G during reset or emergency stop Code.

8. When the rotation scale instruction and the 01 group or 09 group instruction are the same paragraph, the rotation scale instruction will prevail, and the modality of the 01 or 09 group will be changed at the same time. The system will alarm when the rotation scale command and the 00 group command are in the same section.

3.2 Simple G Code

3.2.1 Quickly locate the G00

Instruction format: G00 X_Y_Z_

Function: G00 command, the tool moves at fast speed to a position in the specified workpiece coordinate system with an absolute or incremental value command.

Set with the bit parameter N0:14#0, select one of the following two tool trajectories (as shown in Figure 3-2-1-1).

1. Linear interpolation positioning: The tool trajectory is the same as that of linear interpolation (G01), and the tool is positioned in the shortest time at a fast moving speed of no more than each axis.

2. Non-linear interpolation positioning: the tool is positioned at the fast moving speed of each axis, and the tool trajectory is generally not straight (positioning efficiency is higher).

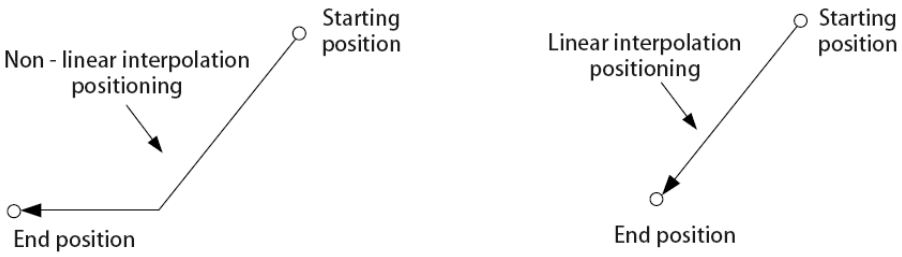


Fig. 3-2-1-1

Illustrate:

1. After the implementation of G00, the system changes the modality of the current tool movement mode to the G00 mode. By changing the value of the system bit parameter N0:48#0, you can set whether the default mode of the system when the power is powered on is G00 (when the parameter value is 0) or G01 (When the parameter value is 1).

2. The tool does not move without specifying the positioning parameters, and the system only changes the modality of the current tool movement mode to G00.

3. G00 and G0 are equivalent formats.

4. X, Y, Z axis G0 speed is set by data parameters P90 ~ P92 .

Limit:

The fast movement speed is set by parameters, such as setting the F speed in the G0 command, which is the cutting feed speed of the subsequent machining section.

For example:

G0 X0 Y10 F800； Fast feed at a speed set by system parameters

G1 X20 Y50； Feeds at F800

Fast feed speed with the buttons on the operator panel to adjust (Figure 3-2-1-2) F0, 25, 50, 100%; The corresponding speed of F0 is set by the data parameter P85 and is common to each axis.

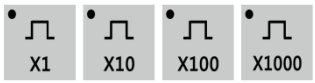


Fig. 3-2-1-2 Fast feed magnification button

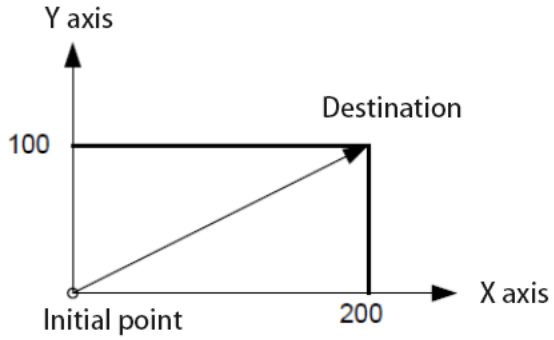
Note: Pay attention to the table and workpiece position when programming to prevent the impact of the knife.

3.2.2 Line interpolation G01

Instruction format: G01 X_ Y_ Z_ F_

Command function: The tool moves in a straight line to the specified position at the feed rate (mm/ min) specified in parameter F.

Instruction Description:



G01 X200 Y100 F200 ;

Note: The speed of each axis direction is as follows:

G01 Xα Yβ Zγ Ff;

In this program segment:

Fig. 3-2-2-1

Speed in the direction of the X axis:$\mathrm{Fy} = \frac{\beta}{\mathrm{L}} \times \mathrm{f}$

Speed in the direction of the Y axis:$\mathrm{Fz} = \frac{\gamma}{\mathrm{L}} \times \mathrm{f}$

Speed in the direction of the Z:$\mathrm{Fx} = \frac{\alpha}{\mathrm{L}} \times \mathrm{f}$

$$\mathrm{L} = \sqrt{\alpha^2 + \beta^2 + \gamma^2}$$

1. X_ Y_ Z_ is the endpoint coordinate value.

2. The feed speed specified by F is valid until the new F value is specified. The feed velocity of the F code instruction is calculated by interpolating along the straight line trajectory, and if the F code is not ordered in the program, the feed speed takes the default F value feed when the system is powered up. (See Data parameter P83 for settings).

Notes:

1. Except for F, the command parameters are all positioning parameters. The upper limit of the cutting feed velocity F can be set with the data parameter P88. The actual cutting speed (feed rate after using the multiplier) is limited to the upper limit if the upper limit value is exceeded. The unit is mm/min. The lower limit of the cutting feed velocity F can be set with the data parameter P89. The actual cutting speed (feed rate after using the multiplier) is limited to the lower limit if it is below the limit. The unit is mm/min.

2. When the tool does not move when the positioning parameters are not specified after G01, the system only changes the modality of the current tool movement mode to G01. By changing the value of the system position parameter N0:48# 0, you can set whether the default mode of the system when the power is switched on is G00 (when the parameter value is 0). G01 (when the parameter value is 1).

3.2.3 Arc (helix) interpolation G02/G03

A. Arc interpolation G02/G03

G02 and G03 regulations:

In-plane arc interpolation is the completion of the arc trajectory from the start point to the end point in the specified direction and radius (or center) in the specified plane.

Since the start and end points are known, the trajectory of the arc cannot be completely determined, so it is necessary to give:

* Rotation direction of the arc (G02, G03).

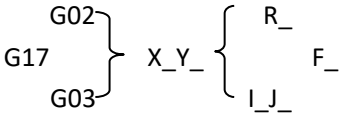
* The plane of arc interpolation (G17, G18, G19).

* Center coordinates or radius, which leads to two code instruction formats, center coordinates I, J, K or radius R programming.

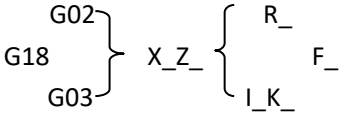
Only when all three of the above points are confirmed can the interpolation operation be performed in the coordinate system.

With the following command, arc interpolation can be performed, and the tool can move along the arc as follows:

The arc of the XY plane



The arc of the ZX plane



Thearc of the Y-Z plane

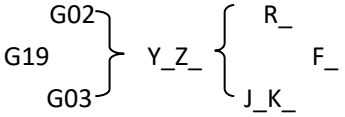


Table 3-2-3-1

project	Specify the content	command	significance
1	Flat setting	G17	XY plane
		G18	ZX plane
		G19	Y-Z plane
2	Turn direction	G02	Clockwise CW
		G03	CounterclockwiseC CW
3	G90 mode The destination location G91 way	Two axes in X, Y, Z	The coordinates of the end location of the workpiece coordinate system
		Two axes in X, Y, Z	The coordinates of the end point relative to the starting point
4	The distance from the starting point to the center of the circle	Two axes in I, J, and K	The coordinates of the center of the circle relative to the starting point
	The radius of the arc	R	The radius of the arc
5	Feed speed	F	Arc cutting speed

Clockwise and counterclockwise refers to the XY plane (ZX plane, YZ plane) in the right-hand rectangular coordinate system, from the Z axis (Y axis, X). axis) in the positive direction of the negative direction (as shown in Figure 3-2-3-1).

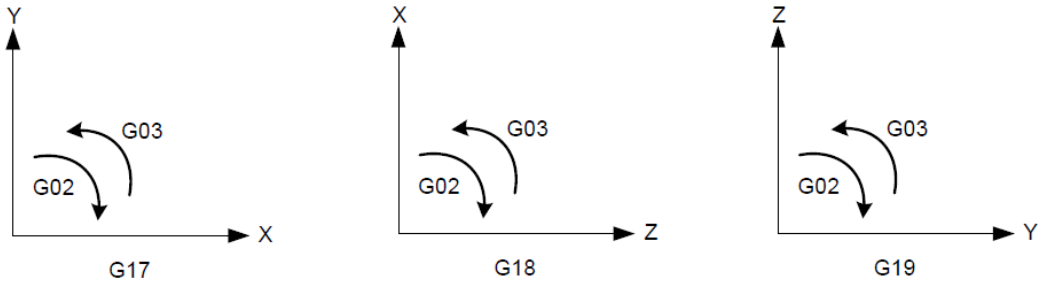


Figure 3-2-3-1

Set bit parameters N0: 48 #1, #2, #3 to specify the default plane modal information at boot.

Specify the end point of the arc with the parameter words X, Y, or Z. The G90 instruction corresponds to an absolute value, and the G91 instruction is expressed in increments, which are the coordinates of the end point relative to the start point. The center of the circle is specified by the parameter words I, J, and K, which correspond to X, Y, and Z, respectively. The I, J, and K parameter values are either in absolute mode G90 or relative mode G91 The coordinates of the center of the circle relative to the beginning of the arc (simple can be understood as the coordinate of the center of the circle that is temporarily the origin of the

coordinates and the coordinates of the center of the circle), which are the incremental values with symbols. (Figure 3-2-3-2):

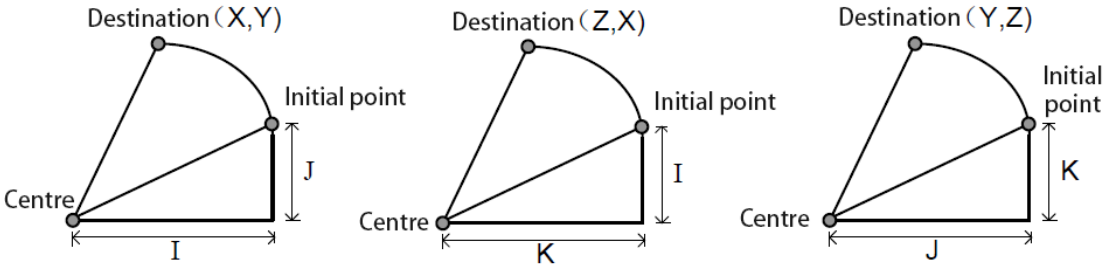


Fig. 3-2-3-2

I, J, K are symbolized according to the center of the circle relative to the starting point. In addition to being specified by I, J, K, the center of the arc can also be specified by the radius R. As follows:

G02 X_ Y_ R_ ；
G03 X_ Y_ R_ ；

1. At this time, the following two arcs can be drawn, a circle greater than 180 ° and a circle less than 180 °. For arcs greater than 180 °, the radius is specified with a negative value.

(e.g. Fig. 3-2-3-3).

(1) When the arc is less than 180°

G91 G02 X60 Y20 R50 F300 ；

(2) When the arc is greater than 180°

G91 G02 X60 Y20 R-50 F300 ；

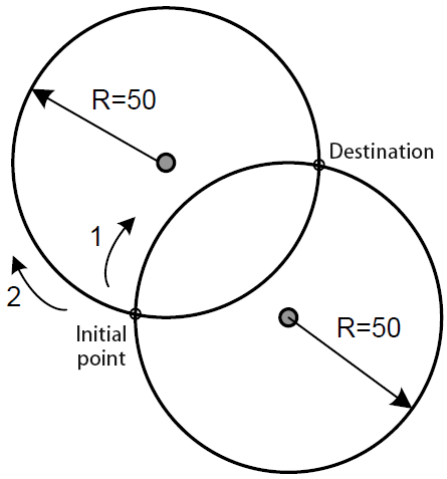


Fig. 3-2-3-3

2. For arcs equal to 180°, I , J, K can also be programmed with R:

Example: G90 G0 X0 Y0; G2 X20 I10 F100；

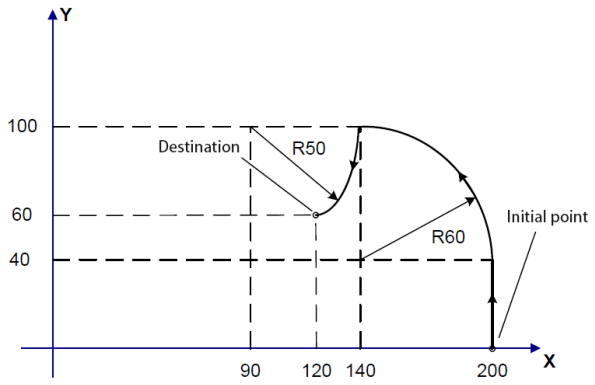
equivalent to G90 G0 X0 Y0; G2 X20 R10 F100

or G90 G0 X0 Y0; G2 X20 R-10 F100

Note: The positive and negative values for 180° arc R do not affect the trajectory of the arc.

3. For arcs equal to 360°, only I, J, K can be programmed.

(Instance of the program) ：



The tool trajectory of Fig. 3-2-3-4 is programmed as follows

1. Absolute value programming

G90 G0 X200 Y40 Z0；

G3 X140 Y100 R60 F300；

G2 X120 Y60 R50；

or

G0 X200 Y40 Z0；

G90 G3 X140 Y100 I-60 F300；

G2 X120 Y60 I-50；

2. Incremental value programming

G0 G90 X200 Y40 Z0；

G91 G3 X-60 Y60 R60 F3000；

G2 X-20 Y-40 R50；

or

G0 G90 X200 Y40 Z0；

G91 G3 X-60 Y60 I-60 F300；

G2 X-20 Y-40 I-50；

Limit:

1. If the program specifies addresses I, J, K and R at the same time, the arc specified by address R takes precedence, and the others are ignored.

2. If the arc radius parameter and the parameter from the starting point to the center of the arc are not specified, the system will alarm.

3. If you want to interpolate the whole circle, you can only specify the parameters I, J, K from the starting point to the center of the arc, and cannot take the form of specifying R.

4. Pay attention to the selection settings of the coordinate plane when interpolating the arc.

5. If X, Y, and Z are all omitted, that is, the starting and end locations are the same, and R is specified (such as: G02R50;), the tool does not move.

B. Spiral interpolation

Instruction format: G02/G03

The arc of the XY plane

G17 { G02 / G03 } X_Y_Z_ { R_ / I_J_ } F_

The arc of the ZX plane

G18 { G02 / G03 } X_Z_Y_ { R_ / I_K_ } F_

Thearc of the Y-Z plane

G19 { G02 / G03 } Y_Z_X_ { R_ / J_K_ } F_

Fig. 3-2-3-5

Function: Moves the tool from the current point to the specified position at the feed speed specified in parameter F from the trajectory of the spiral at the current point.

Illustrate:

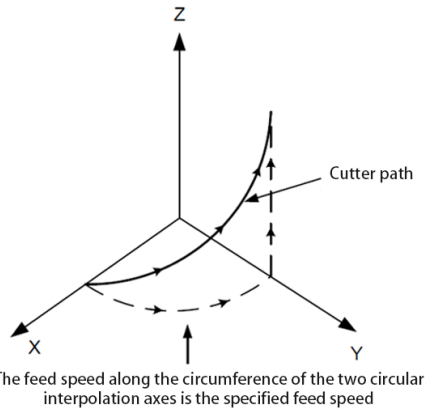


Fig. 3-2-3-6

The first two digits of the directive parameters are the positioning parameters. The

parameter word is the axis number (X, Y or Z) of the two axes in the current plane. These two positioning parameters specify where the tool should be moved within the current plane. The parameter word in the third position of the command parameter is a straight line axis except for the arc interpolation axis. Its parameter value is the height of the spiral. The specific meanings and limitations of other instruction parameters are the same as those of circular arc interpolation.

If the system cannot machine a circle based on the given instruction parameters, the system returns an error message. After execution, the system changes the modality of the current tool movement mode to the G02/G03 mode.

The feed velocity along the circumference of the interpolation axis of the two arcs is specified

The command method simply adds a moving axis that is not an arc interpolation axis, and the F instruction specifies the feed rate along the arc. Therefore, the feed velocity of the linear axis is as follows:

$$\mathrm{Fc} = \mathrm{F} * \frac{The\ length\ of\ the\ straight\ axis}{Length\ of\ arc}$$

Determine the feed velocity so that the feed velocity of the linear axis does not exceed any limit value.

Limitations: Note the selection settings for the coordinate plane when performing spiral interpolation.

3.2.4 Absolute/ incremental programming G90/G91

Instruction format: G90/G91

Function: As a method of indicating the amount of movement of the axis, there are two methods: absolute value instruction and incremental value instruction. The absolute value directive is a method of programming with the coordinate values of the end point locations where the axis is moving. The incremental value directive is a method of programming directly with the relative movement of the axis. The incremental value has nothing to do with the coordinate system in which it is located, just give the direction and distance of the end location relative to the start location. The absolute and incremental instructions are G90 and G91, respectively.

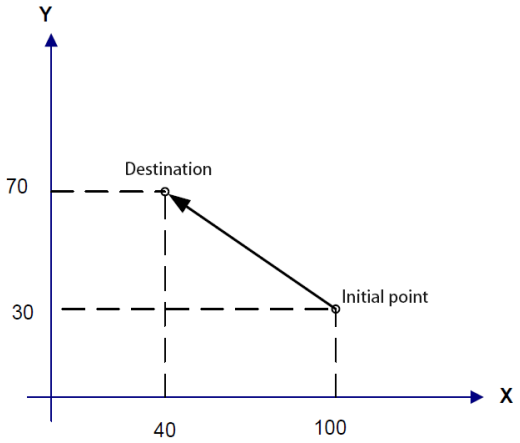


Fig. 3-2-4-1

The movement from the start point to the end point in Fig. 3-2-4-1 is programmed with absolute instruction G90 and incremental instruction G91 as follows:

G90 G0 X40 Y70 ；

or G91 G0 X-60 Y40;

Both methods can complete the same action, and the operator can use it flexibly as needed.

Illustrate:

* No instruction parameters. Sections can be written with other instructions.

* G90 and G91 are modal values of the same group, i.e. when specified as G90, they are both G90 squares (default mode) until G91 is specified, and for G91 , they are not specified All are valid until the G90 mode.

System parameters:

Setting the bit parameter N0:48#4 specifies whether the default positioning parameter at boot is G90 mode (when parameter is 0) or G91 mode (parameter 1). time).

3.2.5 Suspension (G04).

Instruction format: G04 X_ or P_

Function: G04 performs the pause operation and executes the next program segment according to the specified time delay. In addition, in the cutting mode, in the G64 mode, you can specify a pause for accurate stop inspection.

G04	X	0 ～ 9999.9	X corresponds to the second
	P	0 ～ 99999.9999	P corresponds to milliseconds

Table 3-2-5-1

Illustrate:

1. G04 is a non-modal instruction and is only valid in the current line.
2. When the X and P parameters appear at the same time, the X value is valid.
3. When the X and P values are set to negative values, the alarm will be raised.
4. When X and P are not specified, the system does not perform a pause.

3.2.6 Unidirectional positioning (G60).

Instruction format: G60 X_ Y_ Z_

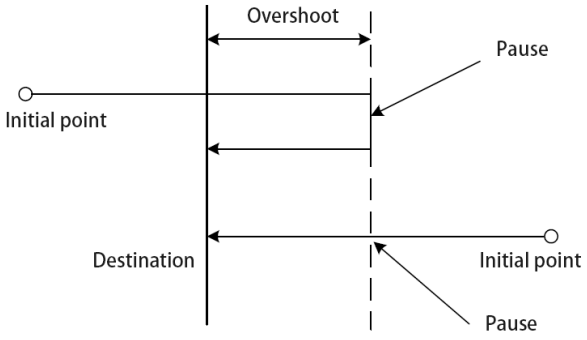


Fig. 3-2-6-1

Function: When precise positioning is required to eliminate machine backlash, the G60 can be used for accurate positioning from one direction.

Description:

G60 is a non-modal G code (can be set by bit parameter NO:52#2 whether it is a modal value) and is valid only in the specified program segment.

The parameters X, Y, and Z, when programmed with absolute values, represent the coordinate values of the end point and the distance the tool moves when programming the increment values. When using one-way positioning under tool offset, the trajectory of one-way positioning is the trajectory after tool compensation.

In the figure above, the marked overimpulse can be set by the system parameters P351, P352, P353, P354, P355, and the pause time can be set by P350 To set, the direction of positioning can be determined by the plus or minus of the set overimpulse, please refer to the system parameters for details.

Example 1:

G90 G00 X-10 Y10；

G60 X20 Y25； (1)

If the system parameters are P350=1, P351=-8, P352=5; In the case of the (1) statement, the trajectory of the tool is AB → pauses for 1s → BC

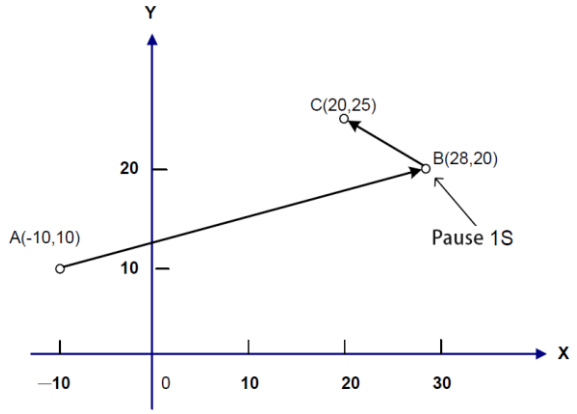


Fig. 3-2-6-2

System parameters:

Table 3-2-6-1

P350	Pause time in one-way positioning (unit: s)
P351	X-axis unidirectional positioning direction and overrange amount (in mm)
P352	Y-axis unidirectional positioning direction and overrange amount (unit: mm)
P353	Z axis unidirectional positioning direction and overrange (unit: mm)
P354	4TH axis unidirectional positioning direction and overrange amount (unit: mm)
P355	5TH axis unidirectional positioning direction and overrange amount (unit: mm)

Note: 1, the number parameter P351 ~ **P354** symbol represents the direction of one-way positioning, the value of the **parameter represents the overdrive amount.**

2. The overrange > 0, and the positioning direction is positive.

3. The overrange <0, and the positioning direction is negative.

4. Overdrive = 0, no one-way positioning.

3.2.7 Online changes to system parameters (G10).

Function description: This function is used to set or modify the values of pitch error compensation, tool radius, length offset, external zero point offset, workpiece zero offset, additional workpiece zero offset, number parameter, bit parameter, etc. in the program.

Instruction format:

G10 L50 N_P_R_； Set or modify bit parameters

G10 L51 N_R_； Set or modify the number parameter

G11； Cancel the parameter input mode

Parameter definitions:

N: Parameter number. The ordinal number of the parameter to modify.

P: Parameter tag. The parameter tag to modify.

R: Modify the value. Lets you specify the modified value of the parameter.

You can also modify the specified values with the following instructions, as detailed in the relevant sections:

G10 L2 P_X_Y_Z_A_B_； Set or modify the external zero offset or workpiece zero offset

G10 L10 P_R_； Sets or modifies the length offset

G10 L11 P_R_； Set or modify the length wear value

G10 L12 P_R_； Sets or modifies the radius offset

G10 L13 P_R_； Set or modify the radius wear value

G10 L20 P_ X_Y_Z_A_B_； Sets or modifies the zero offset of an additional workpiece

Note: 1. In the parameter input mode, you cannot specify other NC statements except for annotative statements.

2, G10 program segment must be a separate instruction, otherwise there is an alarm, after using G10 **remember** to use **G11** to cancel the parameter input mode**, so as not to affect the normal use of the program.**

3. G10 modifies the parameter value, which must meet the range of system parameters, and will alarm if it is not met

4. Before running G10, the **modal command of the fixed cycle must be canceled, otherwise the system will alarm**

5. The parameters that need to be powered off and restarted to be effective cannot be modified with

G10.

3.2.8 Workpiece coordinate system G54 ~ G59

Instruction format: G54 ~ G59

Function: Specify the current workpiece coordinate system, select the workpiece coordinate system by specifying the workpiece coordinate system G code in the program.

Illustrate:

1. No instruction parameters.

2. The system itself can set six workpiece coordinate systems, and any one of the coordinates can be selected by the instruction G54 ~ G59

G54 ---------- Workpiece coordinate system 1

G55 ---------- Workpiece coordinate system 2

G56 ---------- Workpiece coordinate system 3

G57 ---------- Workpiece coordinate system 4

G58 ---------- Workpiece coordinate system 5

G59 ---------- Workpiece coordinate system 6

3. When booting up, the system displays the coordinate system of the workpiece G54 ~ G59 or additional workpiece coordinate system that was executed before the power failure.

4. When different workpiece coordinate systems are called in the program section, the axis that is instructed to move will be positioned to the coordinate point under the new workpiece coordinate system; Without an axis that is ordered to move, the coordinates will jump to the corresponding coordinate values in the new workpiece coordinate system, and the actual machine position will not change.

Example:

The coordinates corresponding to the origin of the G54 coordinate system are (10, 10, 10).

The coordinates of the G55 are (30, 30, 30) for the origin of the coordinate system

When executing the program sequentially, the absolute coordinates of the end points and the machine coordinates are displayed as follows:

Table 3-2-8-1

procedure	Absolute coordinates	Machine coordinates
G0 G54 X50 Y50 Z50	50，50，50	60，60，60
G55 X100 Y100	100，100，30	130，130，60
X120 Z80	120，100，80	150，130，110

5. You can use G10 to change the zero offset value of the external workpiece or the zero offset value of the workpiece. Here's how:

X_Y_Z_ with the instruction G10 L2 Pp

P=0 : The value of the external workpiece zero offset (base offset).

P=1 to 6 : The workpiece zero offset from the workpiece coordinate system 1 to 6.

X_Y_Z_ : For the absolute value instruction (G90), the zero offset value of the workpiece for each axis.

For the Increment Value Directive (G91), add the offset of each axis to the set workpiece zero (the result of the addition is new.)

workpiece zero offset).

With the G10 command, the coordinate system of each workpiece can be changed individually.

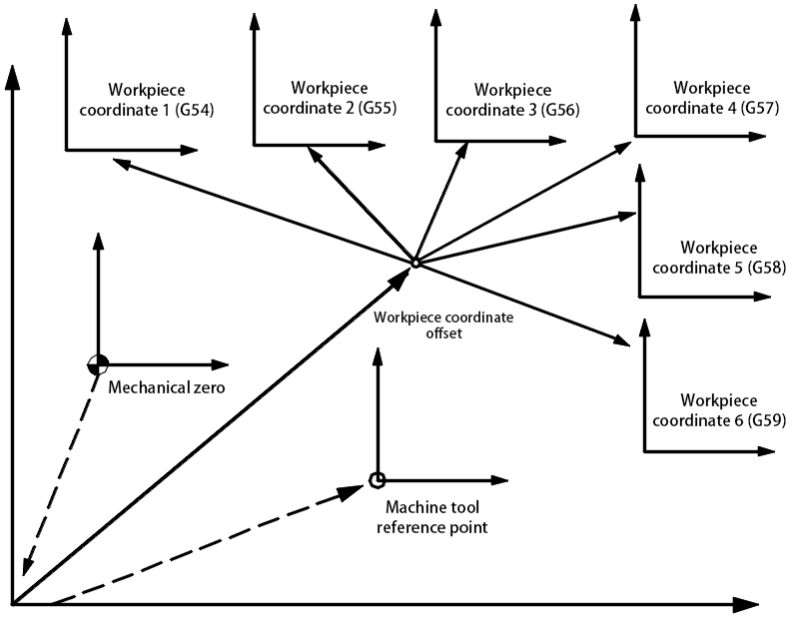


Fig. 3-2-8-1

As shown in Figure 3-2-8-1 above, the machine tool manually returns to the mechanical zero point after starting, and the machine tool coordinate system is established by the mechanical zero point, thereby generating the machine tool reference point and determining the workpiece coordinate system. The values corresponding to the workpiece coordinate system offset data parameters P260 ~ 264 are the overall offsets of the 6 workpiece coordinate systems. You can specify the origin of 6 workpiece coordinate systems by entering the input or setting data parameters P265 ~ P294, which are set according to the distance from the mechanical zero point to the zero point of the respective coordinate system.

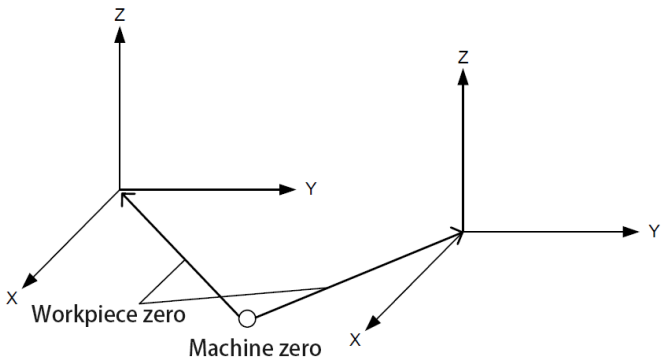


Fig. 3-2-8-2

Example:

N10 G55 G90 G00 X100 Y20；

N20 G56 X80.5 Z25.5；

In the example above, when the N10 program segment starts executing, it is quickly positioned to the position of the workpiece coordinate system G55 (X=100, Y=20). When the N20 program segment starts to execute, it is quickly positioned to the position of the workpiece coordinate system G56, and the absolute coordinate value automatically becomes the coordinate value under the G56 workpiece coordinate system (X=80.5, Z=25.5). ）。

3.2.9 Additional workpiece coordinate system

In addition to 6 workpiece coordinate systems (G54 to G59), the system can also use 48 additional workpiece coordinate systems.

Instruction format: G54 Pn

Pn: Specifies the code for the additional workpiece coordinate system. The range of Pn is 1 to 48.

The settings and limits of the additional workpiece coordinate system are consistent with the workpiece coordinate system G54 ~ G59 .

In the additional workpiece coordinate system, you can use G10 to set the workpiece zero offset value. Here's how:

Instruction: G10 L20 Pn X_Y_Z_;

n=1 to 50: Additional workpiece coordinate system code.

X_Y_Z_ : Set the axis address and offset value of the zero offset of the workpiece.

For the absolute value directive (G90), the specified value is the new offset value.

For the Increment Value Directive (G91), the specified value is added to the current offset value to obtain a new offset.

With the G10 command, the coordinate system of each workpiece can be changed individually.

When the P address of the attached workpiece coordinate system is the same as other instructions containing the P address, it shares a P address.

3.2.10 Select the machine coordinate system G53

Instruction format: G53 X_ Y_ Z_

Function: Quickly position the tool to the corresponding coordinates under the machine coordinate system.

Illustrate:

1. When G53 is used in the program, the subsequent command coordinates should be the coordinate values under the machine coordinates, and the machine tool will be quickly positioned to the specified position.

2. G53 is a non-modal instruction that is only valid in the current section. It does not affect the previously defined coordinate system.

Limit:

Select the machine coordinate system G53

When a position on the machine coordinate system is instructed, the tool moves quickly to that position. The G53 used to select the coordinate system of the machine is a non-modal G code; That is, it is only valid for the program segment of the coordinate system of the command machine. The absolute value G90 should be specified for G53, G53 should be specified in increment mode (G91), and G91 should be ignored (G53 is still in G90 mode, but does not change the modality of G91 mode). The tool can be instructed to move to a special location of the machine, for example, the tool change position can be positioned to this point with the G53 programming moving program.

Note: When G53 is specified**, tool radius compensation and tool length offsets are temporarily removed and restored in the next program segment.**

3.2.11 Floating coordinate system G92

Instruction format: G92 X_ Y_ Z_

Function: Set the floating workpiece coordinate system. The 3 command parameters specify the absolute coordinate value of the current tool in the new floating workpiece coordinate system. This command does not produce a movement of the motion axis.

Illustrate:

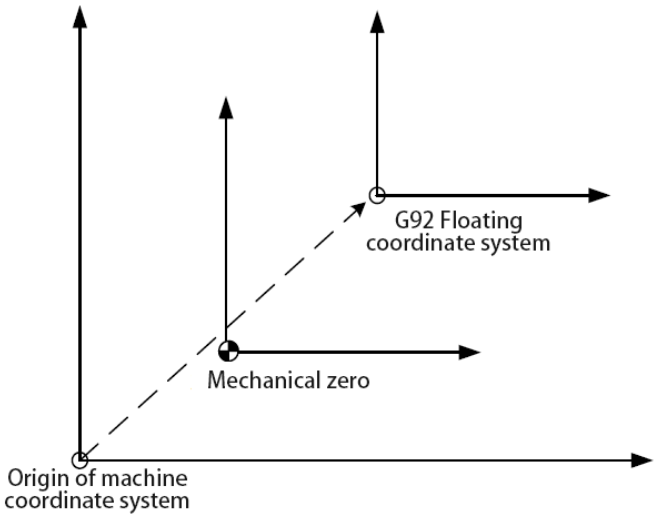


Fig. 3-2-11-1

1. As shown in Figure 3-2-11-1, the origin corresponding to the G92 floating coordinate system is the value under the machine tool coordinate system, which has no relationship with the workpiece coordinate system and can only be established after the machine tool returns to the mechanical zero point.

The validity after setting for G92 is valid before the following cases:

1) Before the system is powered off

2) Before calling the workpiece coordinate system

3) Before the machine is returned to zero

The G92 floating coordinate system is usually used for straightening out temporary workpiece machining and is lost after a power failure. Usually run at the beginning of the program or autorun the program before the MDI mode under the instruction G92.

2. There are two ways to determine the floating coordinate system:

1) Determine the coordinate system with the tip of the knife:

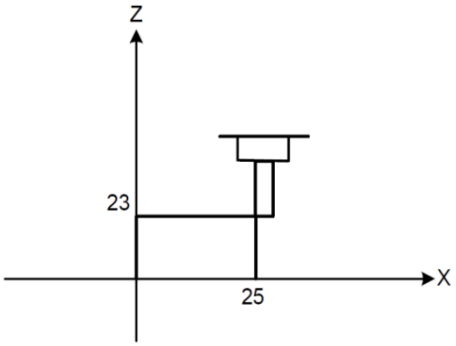


Fig. 3-2-11-2

As shown in Fig. 3-2-11-2, the G92 X25 Z23 takes the position of the tip as the point under the floating coordinate system (X25, Z23).

2) Determine the coordinate system with a fixed point on the tool holder as the datum:

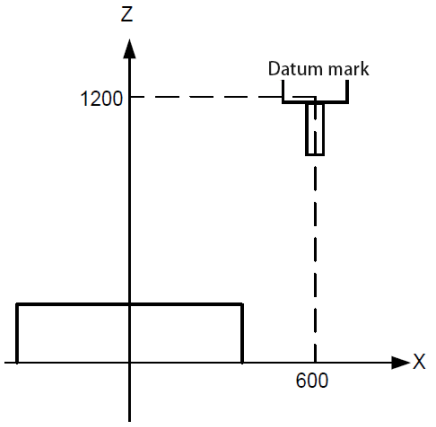


Fig. 3-2-11-3

As shown in Fig. 3-2-11-3, using the G92 X600 Z1200; Command to set the coordinate system (when a datum point on the tool holder is the starting point). If the reference point on the tool holder is used as the starting point, if the motion is instructed according to the absolute value in the program, the reference point is moved to the position that is instructed, and the tool length compensation must be added, and its value is the difference between the reference point and the tool tip.

Note: 1. If the coordinate system is set with G92 **in the tool bias, the coordinate system set by G92 is not added to the tool length compensation .**

2. For tool radius compensation, the tool deviation should be cancelled when using **the G92 command.**

Limit:

After setting the floating coordinate system, the first fixed loop instruction must appear in full format, otherwise the pass will appear incorrect.

3.2.12 Plane selection G17/G18/G19

Instruction format: G17/G18/G19

Function: Plane selection is required for arc interpolation, tool radius compensation or drilling and boring. The plane is selected through G17/G18/G19.

Note: There is no command parameter, and the system defaults to the G17 plane at boot. You can also set the bit parameters N0: 48#1, #2, #3 to determine the default plane of the system after boot. Correspondence of instructions to planes:

G17----------XY plane

G18----------ZX plane

G19----------YZ plane

G17, G18, G19 In unordered program segments, the plane does not change.

Example:

G18 X_ Z_； ZX plane

G0 X_ Y_； The plane is invariant (ZX plane).

In addition, the move instruction has nothing to do with plane selection. For example, in the case of the following command, the Y axis does not exist on the ZX plane, and the Y axis movement is independent of the ZX plane.

G18 Y_；

Tip: Currently, only fixed loops under the G17 plane are supported, and when programming, for specification or strict reasons, it is best to explicitly specify the plane in the corresponding program segment, especially if multiple people share the same system. This avoids accidents or exceptions caused by programming errors.

3.2.13 Polar coordinates start/cancel G16/G15

Instruction format: G16/G15

Function:

G16 specifies the beginning of the polar representation of the positioning parameter.

G15 Specifies the cancellation of the polar representation of the positioning parameter.

Illustrate:

No command arguments.

Set G16 so that coordinate values can be entered with polar radii and angle. The positive

direction of the angle is the counterclockwise turn of the 1st axis of the selected plane, while the negative direction is the clockwise turn. Both radius and angle can be used as absolute value instructions or increment value instructions (G90, G91).

With G16 introduced, the first axis of the positioning parameters of the tool movement command represents the polar diameter in the polar coordinate system and the second axis represents the polar angle in the polar coordinate system.

Set G15 to cancel the polar coordinate mode and return the coordinate value to the cartesian coordinate input.

For the polar coordinate origin:

1. In the G90 absolute mode, when the G16 method is indicated, the zero point of the workpiece coordinate system is the origin of the polar coordinates.

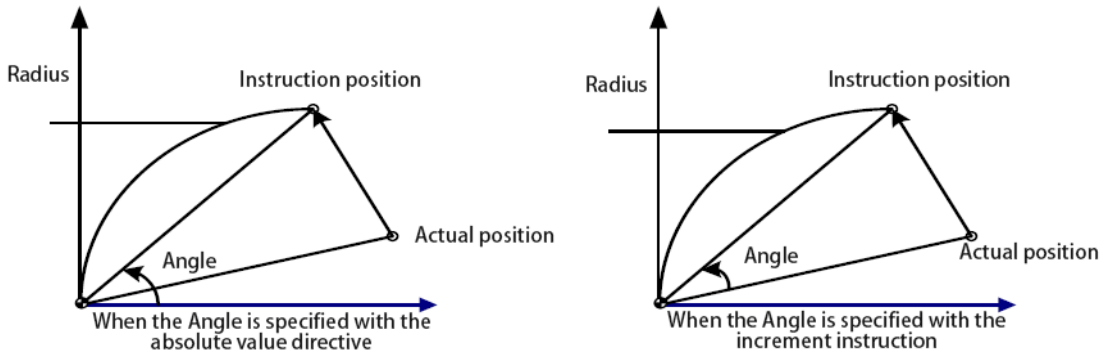


Fig. 3-2-13-1

2. In the G91 incremental mode, when the instruction is used in the G16 mode, the current point is used as the polar coordinate origin.

Example: Bolt hole circle (the zero point of the workpiece coordinate system is set as the origin of the polar coordinates, and the X-Y plane is selected).

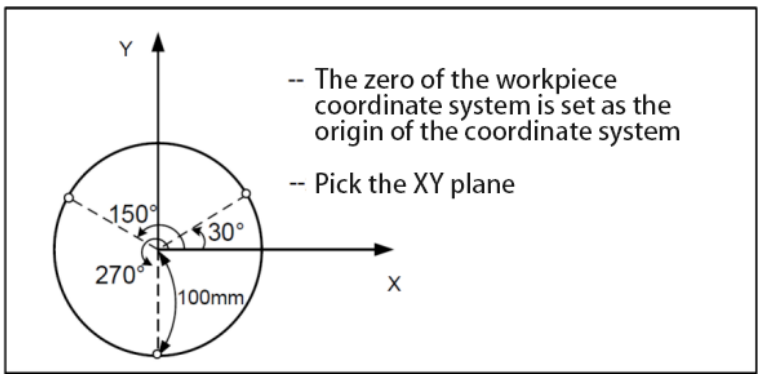


Fig. 3-2-13-2

Specify the angle and radius with absolute values

G17 G90 G16； Specify polar coordinates directives and select XY planes

Sets the zero point of the workpiece coordinate system as the origin of the polar coordinate system

G81 X100 Y30 Z-20 R -5 F200； Specify a distance of 100mm and an angle of 30 degrees

Y150； Specify a distance of 100mm and an angle of 150 degrees

Y270； Specify a distance of 100mm and an angle of 270 degrees

G15 G80； Cancels the polar coordinates instruction

Indicate the angle with the incremental value and the pole diameter with the absolute value

G17 G90 G16； Specify polar coordinates directives and select XY planes

Sets the zero point of the workpiece coordinate system as the origin of the polar coordinate system

G81 X100 Y30 Z-20 R -5 F200； Specify a distance of 100mm and an angle of 30 degrees

G91 Y120； Specify a distance of 100mm and an angle of 150 degrees

Y120； Specify a distance of 100mm and an angle of 270 degrees

G15 G80； Cancels the polar coordinates instruction

In addition, when programming with polar coordinates, attention should be paid to the setting of the current coordinate plane. The polar plane is related to the current coordinate plane, for example, under G91, if the current coordinate plane is G17, the X,Y axis component of the current tool position is the origin. If the current coordinate plane is G18, the Z,X axis components of the current tool position are taken as the origin.

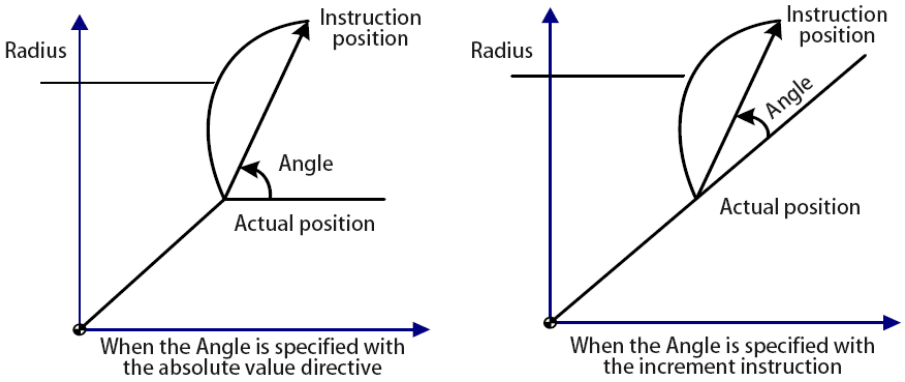


Fig. 3-2-13-3

If the positioning parameters of the first hole cycle command after G16 are not specified, the system takes the current position of the tool as the default positioning parameter for the hole cycle. The first fixed cycle instruction after the polar coordinates must be complete, otherwise, the pass is incorrect.

After G16, in addition to the hole loop, the parameter words of the tool movement command positioning parameters are related to the specific plane selection mode. When you cancel the polar coordinates using the G15 command, immediately following the move command, the default current tool location is the starting point of this move command.

3.2.14 Scale G51/G50 within the plane

Instruction format:

G51 X_ Y_ Z_ P_ (X.Y.Z: Absolute value instruction for scaling center coordinate values, P: Axes scaled at the same scale).

... The scaled machining program segment

G50 scaling canceled

Or G51 X_ Y_Z_ I_ J_ K_ (each axis scaled at a different scale (I, J, K).

... The scaled machining program segment

G50 scaling canceled

Function: The G51 enables the programmed shape to be centered on a specified position, zooming in and out at the same or different scales. It should be noted that the G51 needs to be specified as a separate program segment (otherwise unexpected conditions may occur, resulting in damage to the workpiece and injury to personnel) and cancelled with the G50 .

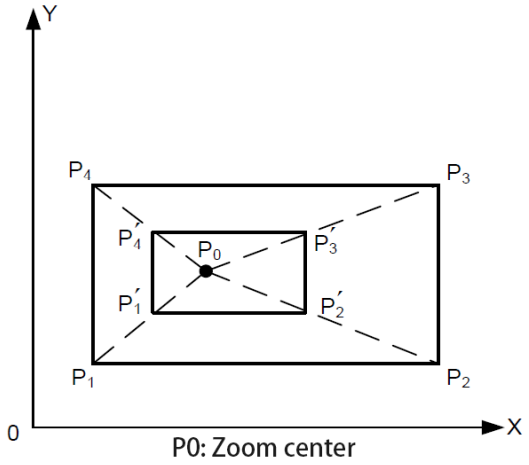


Fig. 3-2-14-1 Scaling (P1P2P3P4 → P1'P2'P3'P4').

Illustrate:

1. Zoom center: G51 can be X_Y_Z_ with 3 positioning parameters, which is an optional

parameter. The positioning parameter specifies the zoom center of G51. If you do not specify positioning parameters, the current position of the tool is set to the scale center. Regardless of whether the current positioning method is absolute or incremental, the zoom center is specified in absolute positioning. In addition, in the polar coordinates G16, the parameters in the G51 instruction are also represented in a Cartesian coordinate system.

Example:

G17 G91 G54 G0 X10 Y10；

G51 X40 Y40 P2； In incremental mode, the zoom center is still the absolute coordinates (40, 40) in the G54 coordinate system

G1 Y90； The parameter Y is still incremental

2. Zoom ratio: Whether it is currently G90 or G91, the scale ratio is always expressed in absolute terms.

In addition to the scale specified in the program, you can also set in the parameters, the data parameters P345 ~ 347 respectively correspond to X, Y, Z zoom magnification, such as no zoom magnification instructions with the data parameter P344 Set the value to scale.

If the parameter values of the specified parameters P or I, J, K are negative, the corresponding axes are mirrored.

3. Scale setting: bit parameter N0:43#2 Set whether X axis scaling is valid, bit parameter N0:43#3 Set whether Y axis scaling is valid, Bit parameter N0:43#4 sets whether the Z axis scaling is valid, bit parameter N0:43#7 sets the zoom magnification of each axis specified (0: P instruction for each axis; 1: I, J, K instructions for each axis).

4. Zoom cancellation: After using the G50 command to cancel the scale zoom, immediately following the move command, the default cancellation of coordinate scaling, the position of the tool is the starting point of the move command.

5. Zoom state, can not command the G code of the reference point (G27 ~ G30, etc.) and the G code of the command coordinate system (G53 ~ G59, G92, etc.). If these G codes must be specified, they should be specified after the zoom function is unscaled.

6. Even if different zoom ratios are specified for arc interpolation and each axis, the tool does not draw an elliptical trajectory.

When the scale ratios of each axis are different, and the arc interpolation is programmed with the radius R, the interpolation graph is shown in Fig. 3-2-14-2 (in the following example, the ratio of the X axis is 2 and the proportion of the Y axis is 1 ）。

G90 G0 X0 Y100;

G51 X0 Y0 Z0 I2 J1;

G02 X100 Y0 R100 F500;

The above directive is equivalent to the following directive:

G90 G0 X0 Y0 Z0；

G02 X200 Y0 R200 F500;

The ratio of the radius R is scaled by the larger of I or J.

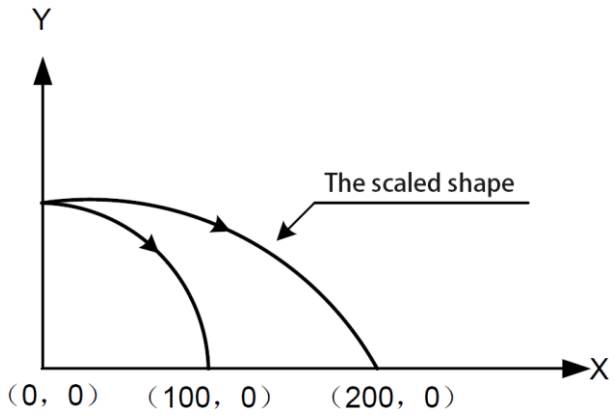


Fig. 3-2-14-2 Scaling of circular interpolation 1

When the scale ratio of each axis is different, and the arc interpolation is programmed with I, J, and K, the system will alarm if the arc is not established.

7. Proportional scaling is invalid for tool radius compensation value tool length compensation value and tool offset value, see Figure 3-2-14-3.

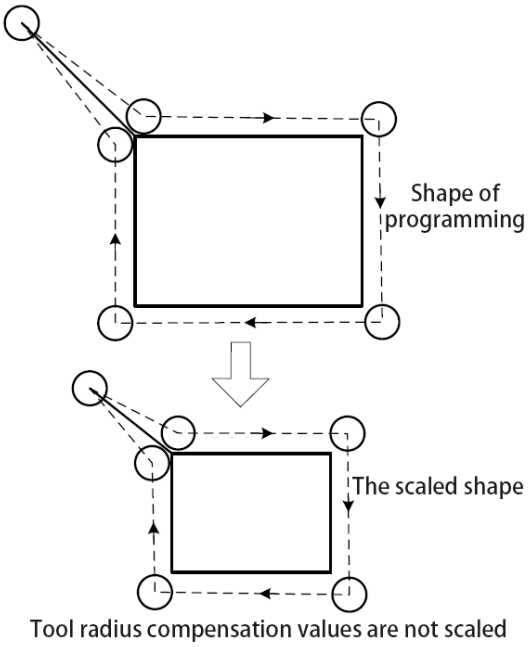


Fig. 3-2-14-3 Scale scaling when tool radius is compensated

Example of an image program:

The main program

G00 G90；

M98 P9000；

G51 X50.0 Y50.0 I-1 J1；

M98 P9000；

G51 X50.0 Y50.0 I-1 J-1；

M98 P9000；

G51 X50.0 Y50.0 I1 J-1；

M98 P9000；

G50；

M30；

subroutine

O9000；

G00 G90 X60.0 Y60.0；

G01 X100.0 F100；

G01 Y100；

G01 X60.0 Y60.0；

M99；

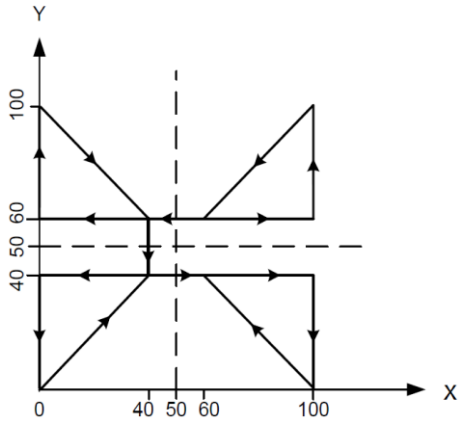


Fig. 3-2-14-4

Limit:

1. In the following fixed cycle, the moving scale of the Z axis is invalid:

1) The cut value Q and return value d of the deep hole drilling cycle (G83, G73).

2) Fine boring cycle (G76).

3) The offset value Q of the X and Y axes in the back boring cycle (G87).

2. When running manually, the moving distance cannot be increased or decreased by the

zoom function.

Note: 1. The position displays the coordinate value after the scale is scaled.

2. When the specified plane has an axis to perform mirroring, the result is as follows:

1) Arc instruction ... . The direction of rotation is reversed

2) Tool radius compensation C..... . The offset direction is reversed

3) The coordinate system rotates..... . The rotation angle is reversed

3.2.15 Coordinate system rotation G68/G69

For machining workpieces consisting of many shapes of the same shape, it can be programmed using the coordinate rotation function, which only needs to be subprogrammed for the graphics unit and then subroutine invoked by the rotation function.

Instruction format: G17 G68 X_ Y_ R_;

or G18 G68 X_ Z_ R_;

or G19 G68 Y_ Z_ R_ ;

G69；

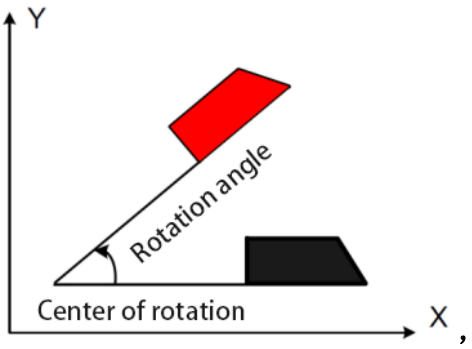


Fig. 3-2-15-1

Function: G68 rotates a programmed shape in a plane at a specified center of origin. G69 is used to cancel coordinate system rotation.

Illustrate:

1. G68 can have 2 positioning parameters, which are optional parameters. The Position parameter specifies the center of the rotation operation. If you do not specify a center of rotation, the system centers the rotation at the current tool position. The positioning parameters are related to the current coordinate plane, select X and Y under G17; Under G18, select Z, X; Under G19, Y, Z.

2. Regardless of whether the current positioning method is absolute or relative, the rotation center can only be specified in the absolute positioning mode of the Cartesian coordinate system. The G68 can also take a command parameter R with a parameter value of the angle at which the rotation is made, and a positive value indicates a counterclockwise rotation. The rotation angle is measured in degrees. The angle of rotation used when there is no rotation angle command in coordinate rotation is set by the data parameter P342.

3. In the G91 mode, the system takes the current tool position as the center of rotation; Whether the rotation angle performs increments is set by the bit parameter NO: 42#3 (G68 coordinate rotation angle, 0: absolute instruction, 1:G90/G91 instruction).

4. When the system is in the rotation mode, the plane selection operation cannot be performed, otherwise the alarm will appear. Care should be taken when programming procedures.

5. In the coordinate system rotation mode, the G code (G27 ~ G30, etc.) and the G code of the command coordinate system (G53 ~ G59) cannot be commanded , G92, etc.). If these G codes must be specified, they should be specified after the rotation function is canceled.

6. After the coordinate system is rotated, the tool radius compensation, tool length compensation, tool offset and other compensation operations are performed.

7, the scale of the zoom mode (G51) in the execution of the coordinate system rotation instructions, the coordinate value of the rotation center is also scaled, however, do not scale the angle of rotation, when the move command is issued, the scale scaling is executed first, and then the coordinates are rotated.

Example 1: Rotation:

```
G92 X-50 Y-50 G69 G17；
G68 X-50Y-50 R60；
G90 G01 X0 Y0 F200；
G91 X100；
G02 Y100 R100；
G3 X-100 I-50 J-50；
G01 Y-100；
G69；
M30；
```

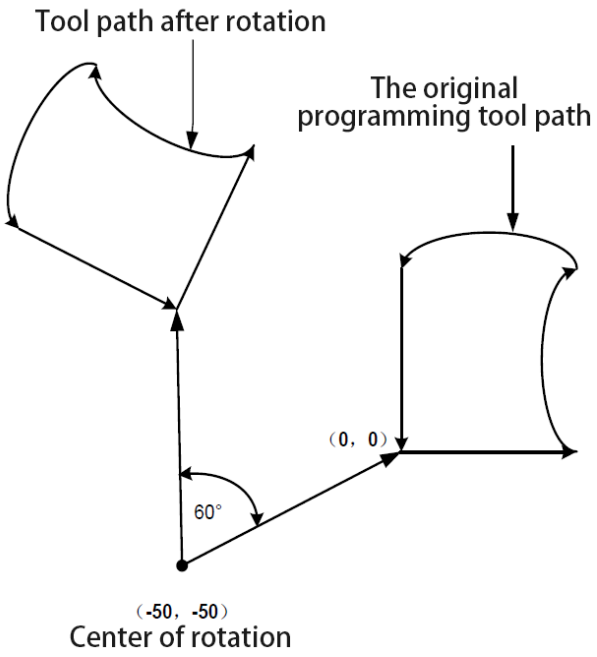


Fig. 3-2-15-2

Example 2: Scale plus rotation:

G51 X300 Y150 P0.5；

G68 X200 Y100 R45；

G01 G90 X400 Y100；

G91 Y100；

X-200；

Y-100；

X200；

G69 G50；

M30；

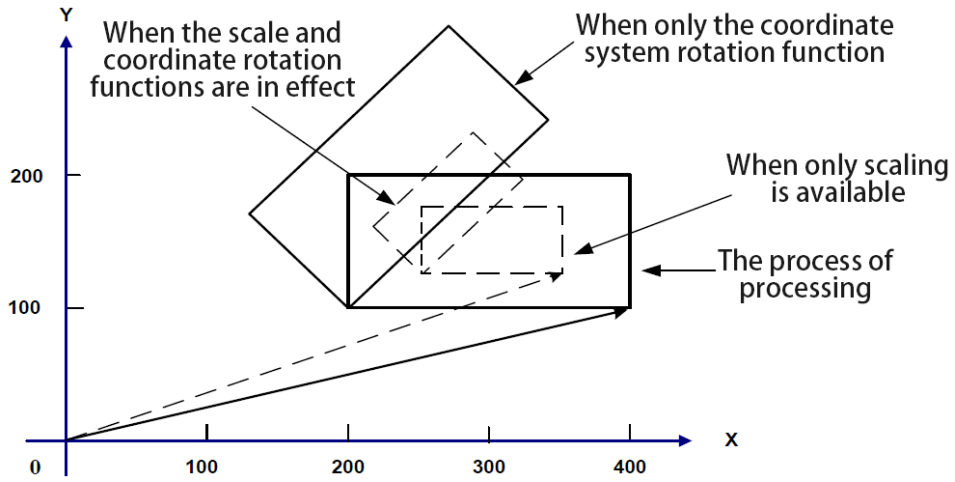


Fig. 3-2-15-3

Example 3: Reuse G68

According to the program (main program)

G92 X0 Y0 Z20 G69 G17；

M3 S1000；

G0 Z2；

G51 X0 Y0 I1.2 J1.2；

G42 D01； (Knife Bias Settings)

M98 P2100(P02100)； (subroutine call)

M98 P2200L7； (Called 7 times)

G40；

G50；

G0 G90 Z20；

X0Y0；

M30；

Subroutine 2200

O2200

G91

G68 X0 Y0 R45.0； (Relative Rotation Angle)

G90；

M98 P2100； (Subprogram O2200 calls subprogram O2100).

M99；

Subprogram 2100

O2100

G90 G0 X0 Y-20； (Right knife repair method is established).

G01 Z-2 F200；

X8.284；

X14.142 Y-14.142；

M99；

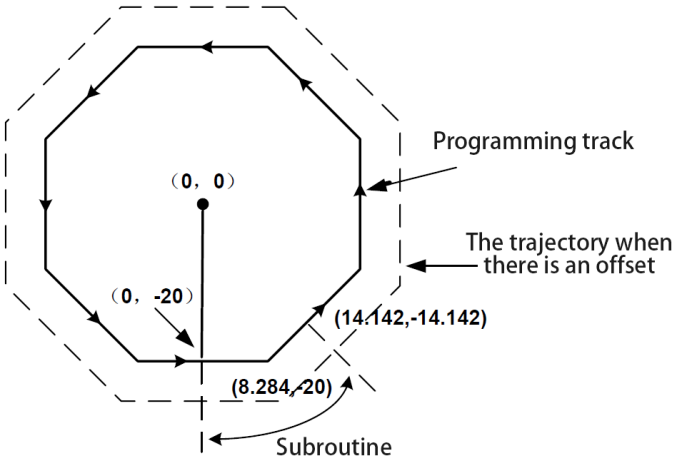


Fig. 3-2-15-4

3.2.16 Jump function G31

Instruction format: G31 X_Y_Z_

Function: After the G31 instruction, like G01, a straight line interpolation can be indicated, and during the execution of the instruction, if an external jump signal is entered, the execution of the instruction is interrupted and the next program segment is executed. The jump function is used when the machining end point is not programmed, but the machining end point is specified with a signal from the machine. For example for grinding. The jump function is also used to measure the dimensions of the workpiece.

Illustrate:

1. G31 is a non-modal G code and is valid only in the specified program segment.

2. When applying tool radius compensation, if the G31 command is issued, the alarm is displayed, and the tool radius compensation should be cancelled before the G31 command.

Example:

The next program segment of G31 is the single-axis movement of the increment value instruction, as shown in Fig. 3-2-16-1

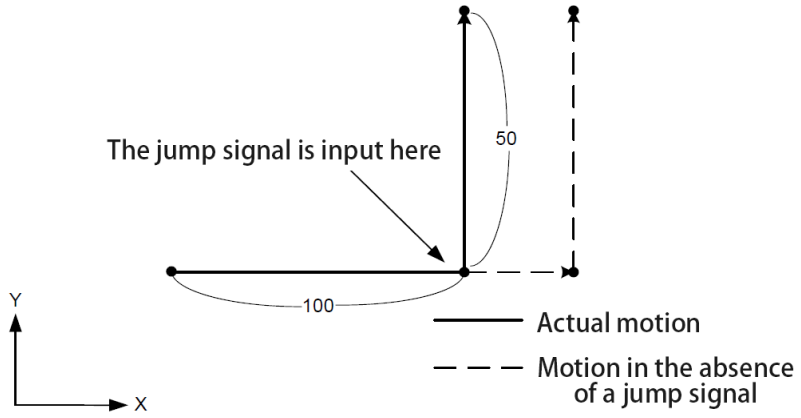


Fig. 3-2-16-1 The next program segment is the single-axis movement of the increment value instruction

The next program segment of G31 is an absolute value that commands a single-axis movement, as shown in Fig. 3-2-16-2

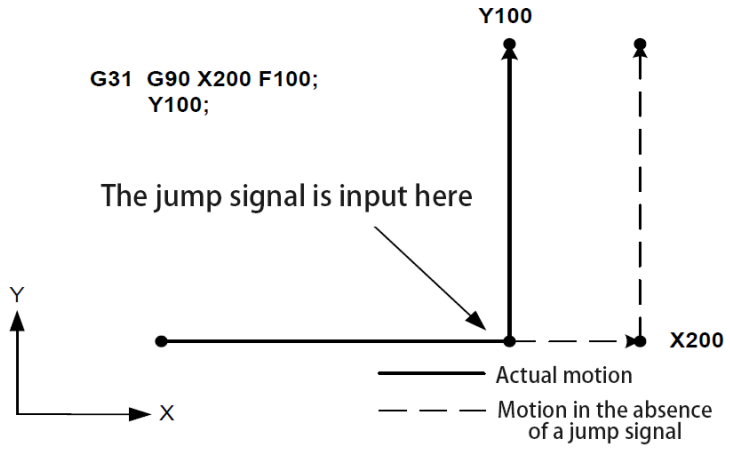


Fig. 3-2-16-2 The next program segment is the single-axis movement of the absolute value instruction

The next program segment of G31 is the 2-axis movement of the absolute value instruction, as shown in Fig. 3-2-16-3

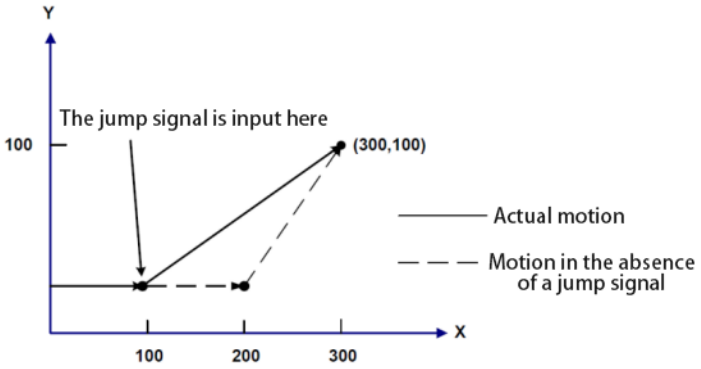


Fig. 3-2-16-3 The next program segment is the **2-axis movement** of the absolute value

instruction

Note: The DNC and MDI single-segment modes do not support program jumping.

3.2.17 Imperial/ Metric conversion G20/G21

Instruction format: G20: Imperial input

G21: Metric input

Function: Can realize program input imperial/ metric conversion.

Description: After the Imperial/ Metric conversion, change the units of the following values:

The feed speed commanded by the F code, the position command, the workpiece zero offset value, the tool compensation value, the scale unit of the hand pulse generator, the distance traveled in the incremental feed.

The G code is the same as it was before the power supply was disconnected when the power supply is turned on.

Note: 1. During the execution of the program, the imperial/metric conversion is not possible.

2. When the imperial system is switched to metric or the opposite, the tool compensation value must be pre-set according to the minimum input increment unit.

3. When the imperial system is switched to metric or vice versa, the operation of the first G28 command from the midpoint is the same as manually returning to the reference point.

4. If the minimum input incremental unit and the minimum command incremental unit are different, the maximum error is half of the minimum command unit, and the error does not accumulate.

5. You can set the program input imperial / metric system by bit parameter N0:01#0.

6. You can set the program output imperial/ metric system by bit parameter N0:04#7.

3.2.18 Any angle chamfer/ corner arc

Instruction format: ,L_: Chamfer

, R_: Corner arc transition

Function: When the above instructions are added to the end of the linear interpolation (G01) or arc interpolation (G02, G03) program segment, the chamfer or transition arc is automatically added to the corner during processing. The program segments of a chamfer or

corner arc transition can be specified consecutively.

Illustrate:

1, the chamfer is after L, specify the distance from the virtual inflection point to the corner start and end point, the virtual inflection point is the actual existence of the corner point if the chamfer is not performed. As shown in the following figure:

（1）G91 G01 X100 ,L10;
（2）X100 Y100;

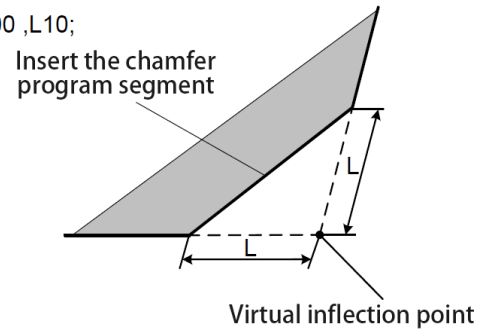


Fig. 3-2-18-1

2. After R, specify the radius of the corner arc, as shown in the following figure

（1）G91 G01 X100 ,R10;
（2）X100 Y100;

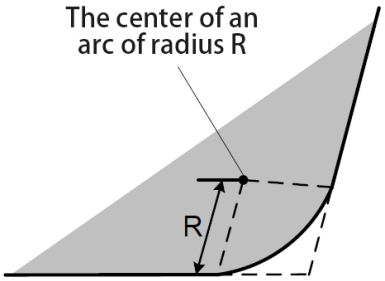


Fig. 3-2-18-2

Limit:

1. Chamfers and corner arcs can only be performed in the specified plane, and parallel axes cannot perform these functions.

2. If the inserted chamfer or arc transition program segment causes the tool to exceed the range of the original interpolation movement, the alarm is raised.

3. The corner arc transition cannot be specified in the thread processing program section.

4. When the instruction chamfer value and corner value are negative, the system takes its absolute value.

3.3 Reference point G code

The reference point is a fixed point on the machine tool to which the tool can be easily moved with the reference point return function.

For the reference point, there are three ways to operate the command, as shown in Figure 3-3-1 Through G28, the tool can pass through the middle point and automatically move to the reference point along the specified axis in the instruction; With the G29, the tool can be automatically moved from the reference point, through the middle point, along the specified axis in the command, to the specified point.

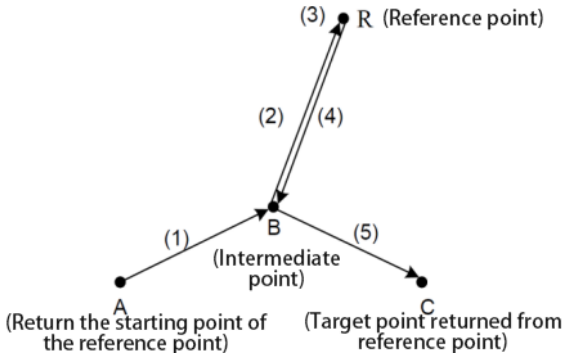


Fig. 3-3-1

3.3.1 Returns the reference point G28

Instruction format: G28 X_ Y_ Z_

Function: The G28 command is used to perform operations that return to a reference point (a specific location on the machine) through an intermediate point.

Illustrate:

Midpoint:

The midpoint is specified by the instruction parameter in G28 and can be represented by an absolute value instruction or an incremental value instruction. When executing this program segment, the coordinate values of the intermediate points of the instruction axis are also stored for use by the G29 (Return from Reference Point) instruction.

Note:

The coordinates of the midpoint are stored in the CNC, but only the coordinate values of the axes of the G28 instruction are stored at a time, and for other axes without instructions, the coordinate values of the previous G28 instructions are used. Therefore, when using G28 instructions, it is best to specify each axis if the default midpoint in the current system is not clear. Consider this in conjunction with the N5 program segment in Example 1 below.

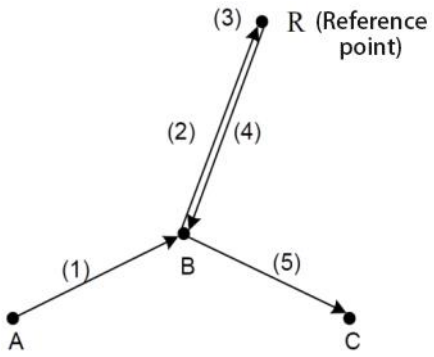


Fig. 3-3-1-1

1. The action of the G28 program segment can be decomposed as follows (see Fig. 3-3-1-1):

(1) Position from the current position to the middle point of the command axis (point A→ point B) at fast moving speed.

(2) Locate from the midpoint to the reference point (point B → R point) at fast moving speed.

2. G28 is a non-modal instruction that is only valid for the current segment.

3. Support the combination of single axis or multiple axis to return to the reference point, and the coordinates of the intermediate points saved in the system are saved in the system when the workpiece coordinate is transformed.

Example:

N1 G90 G54 X0 Y10；

N2 G28 X40； Set the middle point on the X axis to be X40 in the G54 workpiece coordinate system, and return to the reference point through the point (40, 10), that is, the X axis returns to the reference point separately.

N3 G29 X30； From the reference point through the point (40, 10) back to the point (30, 10), that is, the X axis returns to the target point alone.

N4 G01 X20；

N5 G28 Y60； The middle point (X40, Y60), since there are no instructions on the X axis, is replaced by X40 commanded in the previous G28

Note: The midpoint is not (20, 60).

N6 G55； When the workpiece coordinate system is transformed, the middle point is replaced by the point (40, 60) under the G54 workpiece coordinate system

is the point (40, 60) in the G55 workpiece coordinate system.

N7 G29 X60 Y20； From the reference point, pass through the midpoint (40, 60) in the G55 workpiece coordinate system and return to the point (60, 20).

G28 automatically cancels knife repair. However, this instruction is generally used in automatic tool change (that is, after returning to the

reference point, change the tool at the reference point), so when using this command, in principle, the tool radius compensation and tool length compensation should be cancelled first. The first reference point is set to the parameters P50 ~ P54.

3.3.2 Returns the 2, 3, 4 reference points G30

Four reference points are set in the machine coordinate system, but in systems without an absolute position detector, returns to 2, 3, 4 can only be used after an automatic return to reference (G28) or manual return of reference points has been performed Reference point function.

Instruction format:

G30 P2 X_ Y_ Z_； Returns the second reference point (P2 can be omitted).

G30 P3 X_ Y_ Z_； Return to the 3rd reference point

G30 P4 X_ Y_ Z_； Return to the 4th reference point

Function: G30 performs the operation of returning to the specified reference point through the intermediate point specified in G30 .

Illustrate:

1、X_ Y_ Z_； Instructions that specify the middle position (absolute/incremental/incremental).

2. The G30 instruction settings and limits are consistent with G28, and the reference points 2, 3 and 4 are set in the data parameter P55 ～ 69。

3. G30 instructions can also be used with G29 (return from reference point) instructions, and the settings and limits are consistent with G28.

3.3.3 Automatic return to G29 from reference points

Instruction format: G29 X_ Y_ Z_

Function: G29 performs the operation of returning from the reference point (or the current point) to the specified point through the intermediate point of the instruction in G28 or G30.

Illustrate:

1. The action of the G29 program segment can be decomposed into the following steps (see Fig. 3-3-1-1):

(1) Locate from the reference point (or current point) to the middle point defined in

G28, G30 (R point → B point) at a fast moving speed.

(2) Locate from the midpoint to the point of instruction at fast moving speed (point B→ point C).

2. G29 is non-modal information, which is only valid for the current segment. In general, after the G28 or G30 instructions, the return instruction from the reference point should be specified immediately.

3. The optional parameters XY and Z in the G29 instruction format are used to specify the target point returned from the reference point (that is, the C point in Figure 3-3-1-1), which can be represented by absolute or incremental value instructions. Programming increment values, instruction values specify increment values that leave the midpoint. When not specified for some axes, it means that there is no amount of movement for that axis relative to the intermediate point. G29 is followed by instructions with only one axis, and the remaining axes will not move.

Example:

G90 G0 X10 Y10；

G91 G28 X20 Y20； Returns to the reference point through the midpoint (30, 30).

G29 X30； Returning from the reference point (60, 30) through the midpoint (30, 30), note that it is in incremental programming mode, X

The axial component should be 60.

The midpoint of the G29 instruction is assigned by the G28 and G30 instructions. For the definition, specification, and system defaults of the intermediate point, see the G28 directive for details.

3.3.4 Return reference point detection G27

Instruction format: G27 X_ Y_ Z_

Function: G27 performs return reference point detection, X_ Y_ Z_ instructions specifying reference points (absolute/incremental value instructions).

Illustrate:

1, G27 command, the tool is positioned at a fast moving speed. If the tool reaches the reference point, the return reference point indicator is lit; However, if the tool arrives at a position other than the reference point, an alarm is displayed.

2, the machine is locked state, even if the G27 command is specified, the tool has automatically returned to the reference point, and the return completion indicator is not lit.

3. The position reached by the G27 command tool in the bias mode is the position

obtained by adding the bias position, so if the position of the offset value is not the reference position, the indicator light is not lit, and the alarm is displayed. Tool bias should generally be removed before using the G27 command.

4. The X, Y, and Z coordinate point positions specified by G27 are the positions under the coordinate system of the machine tool.

3.4 Fixed loop G code

The fixed loop is a program segment containing G function to realize the processing action that can be completed by multiple program segment instructions, so that the program can be simplified. Makes programming easy for programmers (the system only has a fixed loop of the G17 plane).

General process of fixed loops:

The fixed loop consists of 6 sequential actions, as shown in Fig. 3-4-1.

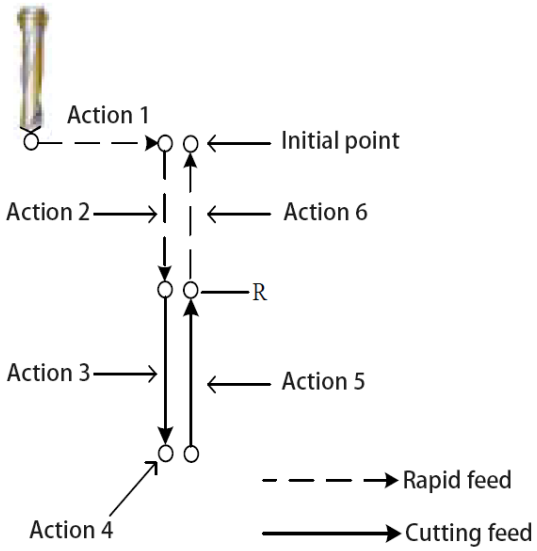


Fig. 3-4-1

Action 1: Positioning of the X and Y axes (can also include another axis)

Action 2: Quickly move to point R

Action 3: Hole making

Action 4: The action at the bottom of the hole

Action 5: Return to point R

Action 6: Quickly move to the initial point

Positioning in the XY plane, holemaking in the Z direction. Stipulating a fixed loop action is determined by three ways. They are each specified by the G code.

1) Data form

G90 absolute value mode; G91 increment value mode

2) Return to the point plane

G98 initial point plane; G99 R point plane

3) Groove processing method

G22、G23、G24、G25、G26、G32、G33、G34 、G35、G36、G37、G38。

4) Hole processing method

G73、G74、G76、G81 ～ G89

The initial point Z plane and the R point plane

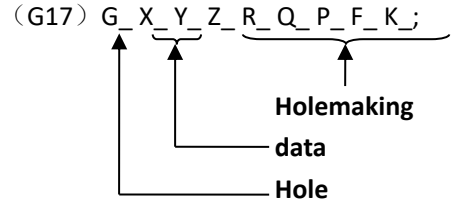
Initial Point Plan: Indicates the absolute position of the tool in the Z-axis direction before the fixed cycle state begins.

R point plane: also known as safety plane, is the position of the Z axis direction in the fixed cycle when the fast movement is converted to the cutting feed, generally set at a certain distance above the surface of the workpiece, to prevent the tool from hitting the workpiece, and to ensure that the acceleration process is completed at sufficient distance.

G73/G74/G76/G81 ~ G89 specify all the data of the fixed cycle (hole position data, hole machining data, number of repetitions) to form a program segment.

Z, R: When the first drilling is performed, the bottom parameter Z and parameter R are either missing, and the system only changes the modality and does not perform the Z-axis action.

The format of the hole machining method is as follows:



Among them, the basic meaning of hole position data and hole machining data is shown in Table 3-4-1

Table 3-4-1

Specify the content	Parameter word	illustrate
Hole machining	G_	Please refer to Table 3-4-3 to note the above limitations

Hole position data	X_,Y_	Specify the position of the hole with absolute or incremental values, with the same control as when G00 is positioned
Holemaking data	Z_	As shown in Fig. 3-4-2(A), specify the distance from point R to the bottom of the hole or with the increment value Absolute values indicate the coordinate values at the bottom of the hole. The feed speed in action 3 is the speed specified by F, and in action 5, depending on the way the hole is processed, it is a fast feed or the speed of the F code command.
	R_	Specify the distance from the initial point plane to the R point of Figure 3-4-2 (B) in increments, or in increments Specifies the coordinate value of the R point with absolute values. The feed speed is all fast feed in Action 2 and Action 6.
	Q_	Specify the amount of pan (increment) in G73,G83 or the amount of translation (increment value) in G76,G87
	P_	Specifies the pause time at the bottom of the hole. Fixed loop instructions can be P_ with one parameter Specify in the parameter value of the P_ when the pause operation is performed after the tool reaches the Z-plane. The unit is ms. The minimum parameter value is set by the data parameter P336, and the maximum parameter value is set by the data parameter P337
	F_	Specifies the cutting feed rate
	K_	Specify the number of repetitions in the parameter value of the K_, Kis valid only within the specified program segment. It can be omitted and not written, and the default is one time. The maximum number of drilling times is 99999 times, when a negative value is specified, it is executed according to its absolute value, and when it is zero, no drilling operation is performed , only change the modality

Limit:

The fixed loop G instruction is a modal instruction that remains valid until the G code that unloops is specified.

G codes for unfixed loops , with G codes for groups G80 and 01.

Machining data, once specified in a fixed cycle, is maintained until the fixed cycle is canceled, so at the beginning of the fixed cycle, all the necessary holemaking data is specified, and only the changed data needs to be specified in the subsequent fixed cycle.

Concentrate:

1. **The cutting speed of the F command is maintained even if the fixed cycle is canceled.**
2. **Note: When fixing the cycle, the Z** axis **(cutting axis) scaling function is invalid.**
3. **In the single-stage mode, the fixed cycle generally adopts the three-stage processing**

mode, and the positioning → R plane → the initial plane.

4. In the fixed cycle, if the position is reset, the hole processing data and the hole position data are eliminated. The above persisting and purging data is shown in the following table:

Table 3-4-2

order	The specification of the data	illustrate
1	G00 X_ M3	
2	G81 X_Y_Z_R_F_	Since it is the beginning, to Z, R, F to specify the required values
3	Y_	Because the hole processing method and processing data specified in Hole 2 are the same, G 81, Z _R_F_ can all be omitted. The position of the hole is moved Y, and the hole is machined once in the G 81 method
4	G82 X_P_	Moving only in the X-axis direction relative to the hole 3 position, it is machined in the manner of G 82, and the holemaking is performed with the Z_R_F_ specified in 2 and P_ specified in 4 for the holemaking data
5	G80 X_Y_	No hole making is performed, and all hole data is canceled
6	G85 X_Y_R_P_	Because all data is canceled in 5, Z_R_ needs to be specified again, and F_ is the same as specified in 2, so it can be omitted. P_ this program segment is not needed, just save it
7	X_Z_	Hole machining differs from 6 with only Z_ value, and the hole position is only shifted in the X-axis direction
8	G89 X_Y_	Hole processing in G89 mode is performed by using the Z_ specified in 7, the R_ specified in 6, P_ and F_ specified in 2 as holemaking data
9	G01 X_Y_	Eliminate holemaking methods and holemaking data

A. Absolute value instruction and increment value instruction of fixed loop G90/G91

The distance traveled along the drilling axis changes to G90 and G91 as shown in Figure 3-4-2. (Generally programming with G90, if programming with G91, Z and R are treated as negative values.)

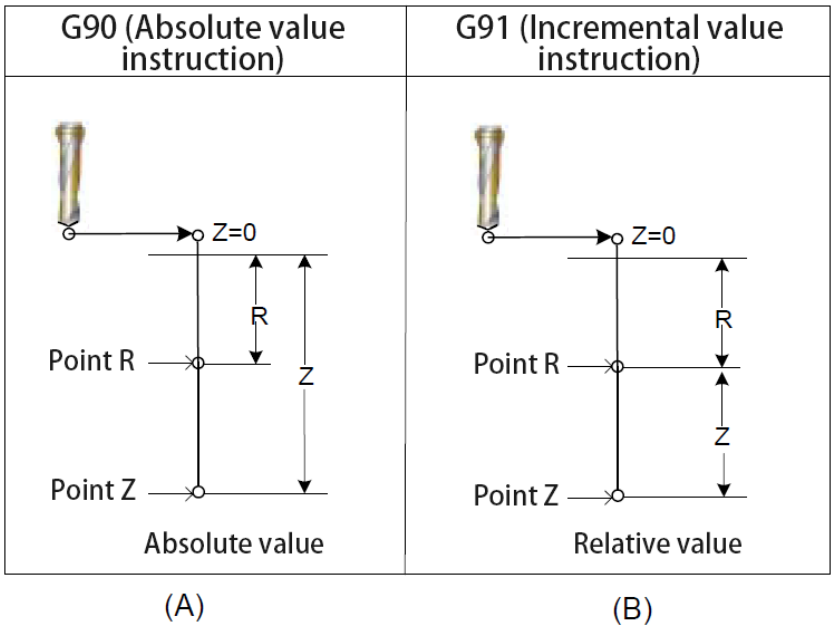


Fig. 3-4-2

B. The return plane instruction of the fixed loop G98/G99

When the tool reaches the bottom of the hole, the tool can return to the R point plane or the initial position plane. Depending on the G98 and G99, you can return the tool to the initial point plane or the R point plane.

In general, the G99 is used for the first drill and the G98 is used for the last drilling. Even when machining holes in the G99 state, the initial plane does not change. The actions of the instructions G98 and G99 are shown in the following figure.

The system defaults to G98.

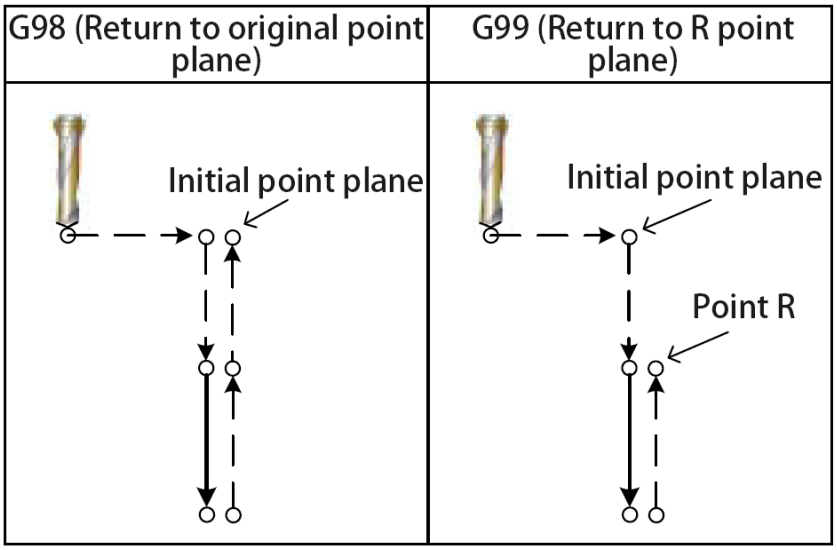


Fig. 3-4-3

Each fixed loop interprets the following symbols used in the diagram:

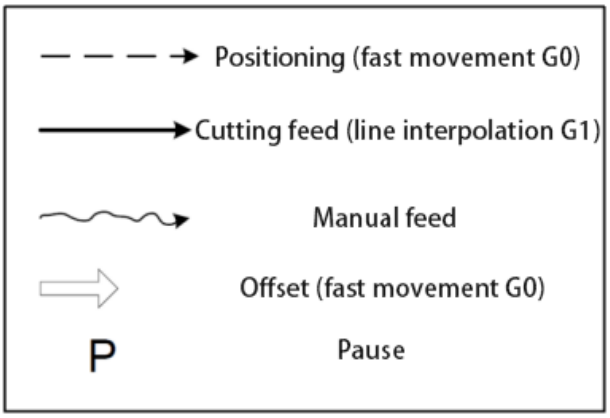


Fig. 3-4-4

Fixed cycle comparison table (G22 ~ G89).

Table 3-4-3

Code G	Drilling (-Z direction).	Hole bottom action	Retract action (+Z direction).	use
G22	Cutting feed		Fast feeding	Rough milling of recesses in a counterclockwise circle
G23	Cutting feed		Fast feeding	Rough milling of recesses in the circle clockwise
G24	Cutting feed		Fast feeding	Counterclockwise intracircle fine milling cycle
G25	Cutting feed		Fast feeding	Clockwise intra-circle fine milling cycle
G26	Cutting feed		Fast feeding	Counterclockwise outer circle fine milling cycle
G32	Cutting feed		Fast feeding	Clockwise outer circle fine milling cycle
G33	Cutting feed		Fast feeding	Counterclockwise rectangular groove rough milling
G34	Cutting feed		Fast feeding	Rough milling clockwise rectangular grooves
G35	Cutting feed		Fast feeding	Counterclockwise rectangular groove fine milling cycle
G36	Cutting feed		Fast feeding	Clockwise rectangular groove within the fine milling cycle
G37	Cutting feed		Fast feeding	Counterclockwise rectangular external fine milling cycle

G38	Cutting feed		Fast feeding	Clockwise rectangular external fine milling cycle
G73	Intermittent feeding		Fast feeding	High-speed deep hole machining
G74	Cutting feed	Pause spindle rotation	Cutting feed	Counter-attack wire loop
G76	Cutting feed	The spindle is stopped directionally	Fast feeding	Fine boring
G80				Cancel
G81	Cutting feed		Fast feeding	Drill, point drill
G82	Cutting feed	Stop the knife	Fast feeding	Drilling, boring stepped holes
G83	Intermittent feeding		Fast feeding	Deep hole machining cycle
G84	Cutting feed	Stop the tool → spindle forward rotation	Cutting feed	Tapping
G85	Cutting feed		Cutting feed	Boring
G86	Cutting feed	The spindle stops	Fast feeding	Boring
G87	Cutting feed	The spindle is spinning	Fast feeding	Boring
G88	Cutting feed	Stop the tool → spindle forward rotation	Manual	Boring
G89	Cutting feed	Time out	Cutting feed	Boring

Limit:

During fixed cyclic positioning, the tool radius offset (D) is ignored.

3.4.1 Circular groove coarse milling G22/G23

Instruction format:

G98/G99 { G22 / G23 } X_Y_Z_R_I_L_W_Q_V_D_F_K_

Function: Starting from the center of the circle, multiple arc interpolations are performed in the form of a helix until the circular groove of the programmed size is machined.

Illustrate:

G22: counterclockwise circular groove rough milling;

G23: Clockwise circular groove rough milling;

X, Y: X, Y plane starting point;

Z: Processing depth, absolute position at G90, relative to R datum at G91;

R:R datum position, absolute position at G90, position relative to the beginning of this program segment at G91 ;

I: Circle inner groove radius, I should be greater than the radius of the current tool;

L: The width increment of the cutting in the XY surface should be less than the tool diameter, greater than 0;

W: The first depth of cut in the direction of the Z axis, which is the distance from the R datum to the bottom, should be greater than 0 (if the first depth of cut exceeds the bottom position, it is directly machined in the bottom position of the groove);

Q: The cutting depth of each cutting feed;

V: When quickly cutting the knife, the distance from the raw surface is greater than 0;

D: Tool compensation number, value range is 0 ~ 256, D0 defaults to 0. Takes out the current tool diameter value according to the given serial number;

K: Number of repetitions.

Cyclic process:

1) Snap positioning to the position of the XY plane;
2) Quickly descend to the R point plane;
3) Cutting speed downward cutting W distance depth;
4) The center outwards each time the helix mills the circular face with a radius of I by increasing the L value;
5) The Z axis quickly returns to the R datum;
6) The X,Y axis is quickly positioned to the center of the circle;
7) The Z axis quickly descends to the distance from the surface V of the machined surface;
8) Depth of Z axis downward cutting (Q+V);
9) Cycle the action (4) ~ (8) until the round surface of the total cut depth is processed;
10) Depending on the specified G98 or G99, returns to the initial point plane or R point plane.

Instruction trajectory:

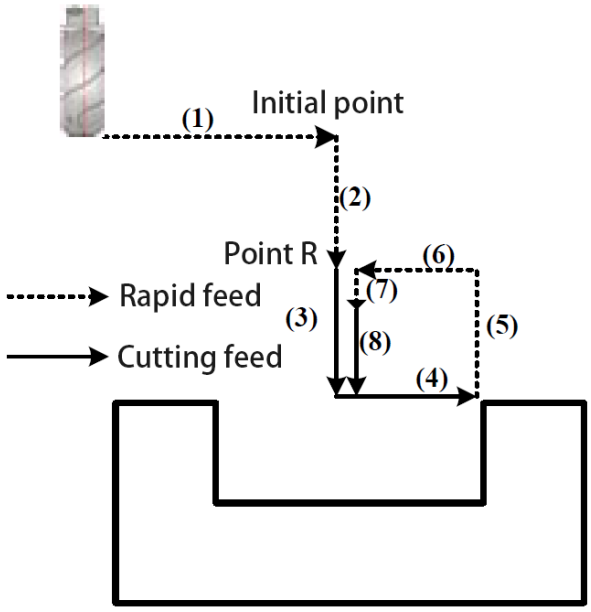


Fig. 3-4-1-1

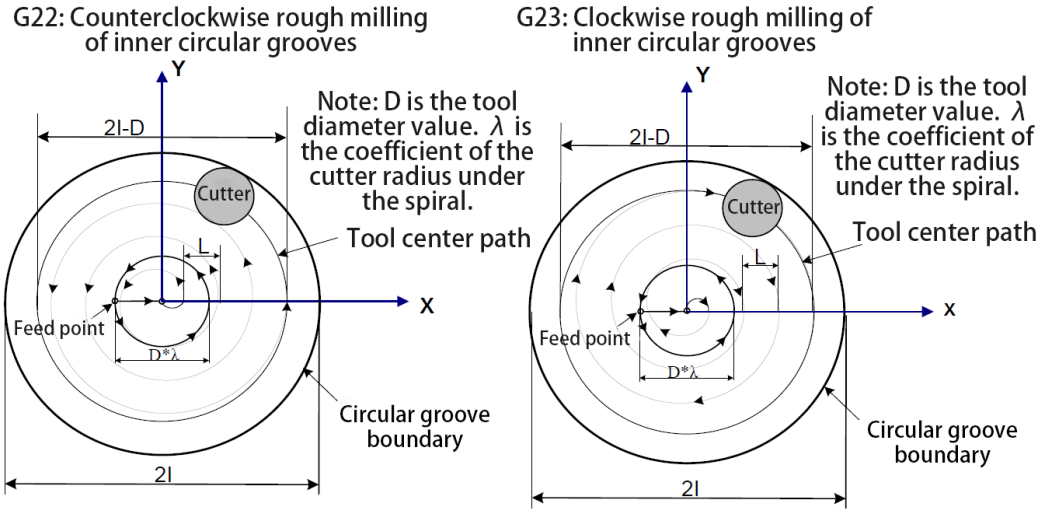


Fig. 3-4-1-2

Note:

1. When using this command, NO:14#1 must be changed to 1 (when moving fast, stop when fast magnification F0).

2. The coefficient setting of the radius of the screw knife in the groove cycle must be greater than 0, and the radius of the screw knife is set by the data parameter P333. When its speed is written to be greater than the set screw cutting speed, the Z-axis downward cutting speed is used to set the downward rotation speed. Parameter 329 of the knife speed setting under the helix.

Example: Coarse milling of a circular groove with the fixed-loop G22 instruction, as shown in the following figure:

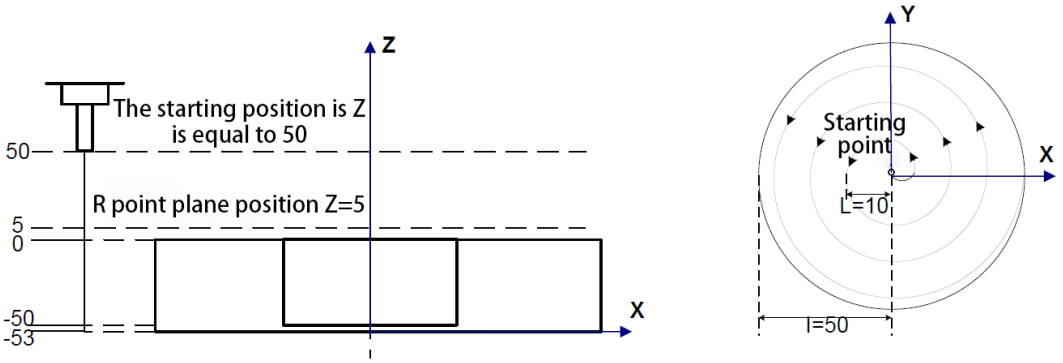


Fig. 3-4-1-3

G90 G00 X50 Y50 Z50； (G00 Quick Positioning).

G99 G22 X25 Y25 Z-50 R5 I50 L10 W20 Q10 V10 D1 F800； (Perform a rough milling cycle of grooves in a circle).

G80 X50 Y50 Z50； (Unpin the loop and return from the R point plane).

M30；

Cancel:

You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes (Bit parameters NO: 52#2 set to 1) and G22/G23, otherwise G22/G23 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.2 Full circle internal milling cycle G24/G25

Instruction format:

G98/G99 { G24 / G25 } X_ Y_ Z_ R_ I_ J_ D_ F_ K_

Function: The tool is finely milled inside the circle with the specified radius value I and direction, and returned after the fine milling is completed.

Illustrate:

G24: Counterclockwise intracircle milling cycle.

G25: Clockwise intracircle milling cycle.

X, Y: X, Y plane starting point;

Z: Processing depth, absolute position at G90, relative to R datum at G91;

R:R datum position, absolute position at G90, position relative to the beginning of this

program segment at G91 ;

I: Precision milling circle radius, value range 0.0001 ~ 99999.9999mm, when the value is negative, take its absolute value;

J: The distance between the starting point of the fine milling and the center of the fine milling circle, the value range is 0 ~ 99999.9999mm , and the absolute value is taken when it is negative;

D: Tool compensation number, value range is 0 ~ 256, D0 defaults to 0 . Takes out the current tool diameter value according to the given serial number;

K: Number of repetitions.

Cyclic process:

1) Quickly locate to the position of the XY plane;
2) Quickly descend to the R point plane;
3) Cutting feed to the bottom of the hole machining starting point;
4) Circular arc interpolation with transition arc 1 as the trajectory from the starting point;
5) Interpolate the whole circle with the fine milled inner circle as the trajectory;
6) Alternate arc interpolation with transition arc 4 as the trajectory to return to the starting point;
7) Depending on the specified G98 or G99, returns to the initial point plane or R point plane.

Instruction trajectory:

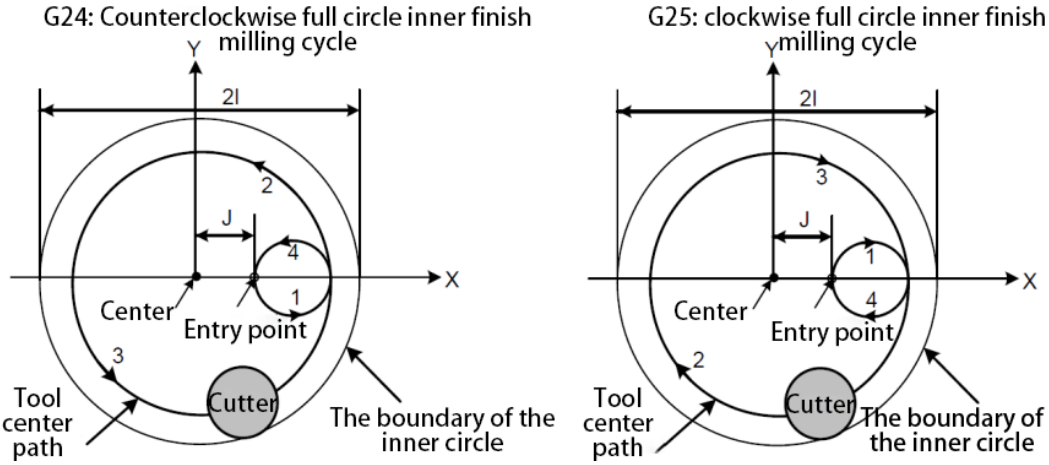


Fig. 3-4-2-1

Note: To use this directive, change NO:14#0 to 1. (The fast-moving G00 interpolation trajectory is straight-lined).

Example: With the fixed cycle G24 command, fine milling of a coarsely milled circular groove

as shown in the following figure.

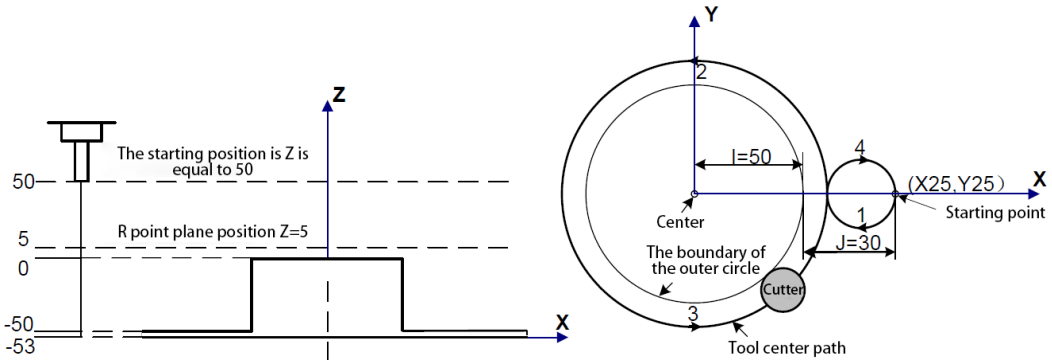


Fig. 3-4-2-2

G90 G00 X50 Y50 Z50； (G00 Quick Positioning).

G99 G24 X25 Y25 Z-50 R5 I50 J10 D1F800； (Start the fixed cycle and go down to the bottom of the hole for the circular fine milling cycle).

G80 X50 Y50 Z50； Unpin the loop and return from the R point plane).

M30；

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO: 52#2 set to 1) and G24/G25, otherwise G24/G25 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.3 Outer Circle Fine Milling Cycle G26/G32

Instruction format:

$$G98/G99 \begin{Bmatrix} G26 \\ \\ G32 \end{Bmatrix} X\_\ Y\_\ Z\_\ R\_\ I\_\ J\_\ D\_\ F\_\ K\_;$$

Function: The tool is finely milled outside the circle with the specified radius value and direction, and returned after the fine milling is completed.

Illustrate:

G26: Counterclockwise outer circle fine milling cycle.

G32: Clockwise outer circle milling cycle.

X, Y: X, Y plane starting point;

Z: Processing depth, absolute position at G90, relative to R datum at G91;

R:R datum position, absolute position at G90, position relative to the beginning of this

program segment at G91 ;

I: Precision milling circle radius, value range 0.0001 ~ 99999.9999mm, when the value is negative, take its absolute value;

J: The distance between the starting point of fine milling and the circular edge of fine milling, the value range is 0.0001 ~ 99999.9999mm, and its absolute value is taken when it is negative;

D: tool compensation number, value range is 0 ~ 256, D0 defaults to 0 , according to the given sequence number to take out the current tool radius value;

K: Number of repetitions.

Cyclic process:

1) Snap positioning to the position of the XY plane;
2) Quickly descend to the R point plane;
3) Cut feed to the bottom of the hole;
4) The starting point is to interpolate the arc with the transition arc 1 as the trajectory;
5) Arc 2, arc 3 interpolated the whole circle for the trajectory;
6) Transition arc 4 Circular arc interpolation for the trajectory back to the starting point;
7) Depending on the specified G98 or G99, returns to the initial point plane or R point plane.

Instruction trajectory:

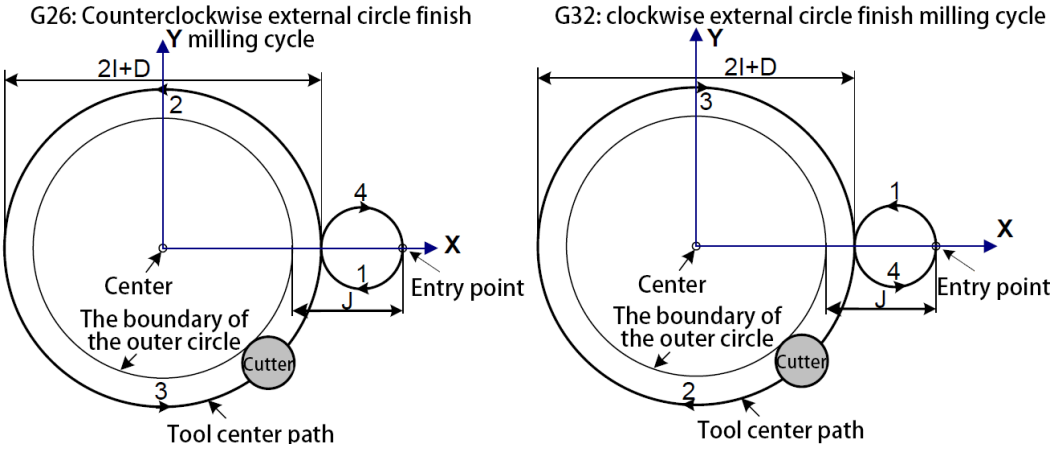


Fig. 3-4-3-1

Related Notes:

When the outer circle is milled, the interpolation direction of the transition arc and the fine milling arc is inconsistent, and the interpolation direction in the instruction description refers to the interpolation direction of the fine milling arc.

Example: With the fixed-cycle G26 command, fine milling of a coarsely milled circular

groove as shown in the following figure.

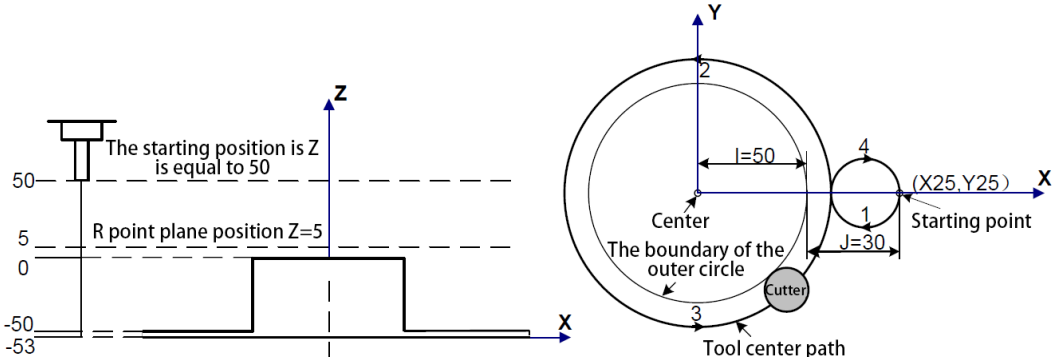


Fig. 3-4-3-2

G90 G00 X50 Y50 Z50； (G00 Quick Positioning).

G99 G26 X25 Y25 Z-50 R5 I50 J30 D1 F800； (Start the fixed cycle and go down to the bottom of the hole for the outer circle milling cycle).

G80 X50 Y50 Z50； (Unpin the loop and return from the R point plane).

M30；

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO:52#2 set to 1) and G26/G32, otherwise G26/G32 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.4 Rectangular groove rough milling G33/G34

Instruction format:

G98/G99 { G33 / G34 } X_Y_Z_ R_ I_ J_ L_ W_ Q_ V_ U_ D_ F_ K_

Function: Starting from the center of the rectangle, the straight-line cutting cycle is performed with the specified parameter data until the rectangular groove of the programmed size is machined.

Illustrate:

G33: Counterclockwise rectangular groove rough milling.

G34: Clockwise rectangular groove rough milling.

X, Y: X, Y plane starting point;

Z: Processing depth, absolute position at G90, relative to R datum at G91;

R:R datum position, absolute position at G90, position relative to the beginning of this program segment at G91 ;

I: The width of the rectangular groove in the X-axis direction. I should be greater than the tool diameter. The radius of the spiral knife should be less than half of I;

J: The width of the rectangular groove in the direction of the Y axis. J should be greater than the tool diameter. The radius of the screw down knife should be less than half of J;

L: The width increment of the cutting in the specified plane should be less than the tool diameter, greater than 0;

W: The first depth of cut in the direction of the Z axis, which is the distance from the R datum to the bottom, should be greater than 0 (if the first depth of cut exceeds the bottom position, it is directly machined in the bottom position of the groove);

Q: The cutting depth of each cutting feed;

V: When quickly cutting the knife, the distance from the raw surface is greater than 0;

U: Corner arc radius, omitted to indicate no corner arc transition. U should be greater than or equal to the tool radius;

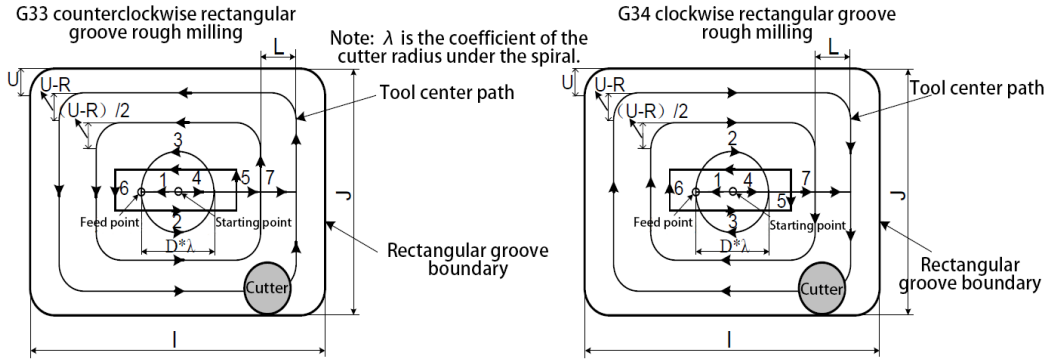
D: Tool compensation number, the value range is 0 ~ 256, D0 defaults to 0 Takes out the current tool diameter value according to the given serial number;

K: Number of repetitions.

Cyclic process:

1) Quickly locate to the starting point of the screw downblade in the XY plane;
2) Quickly descend to the R point plane;
3) Multiply the radius compensation value by the value of the data parameter No. 333 as the diameter screw under the knife W distance;
4) feed to the center of the rectangle X0Y0;
5) The center outwards mills the rectangular surface each time increments by the L value;
6) The Z axis quickly returns to the R datum;
7) Quickly locate to the starting point of the screw downblade in the XY plane;
8) The Z axis quickly descends to the distance from the surface V of the machined surface;
9) Depth of Z axis downward cutting (Q+V);
10) Cycle 4) ~ 9) until the rectangular surface of the total depth of cut is processed;
11) Depending on the specified G98 or G99, returns to the initial point plane or R point plane.

Instruction trajectory:



Note: λ is the coefficient of the radius of the spiral knife.

Fig. 3-4-4-1

Note: To use this directive, change NO:14#0 to 1.

Example: Coarse milling of a rectangular inner groove with the fixed-loop G33 instruction, as shown in the following figure:

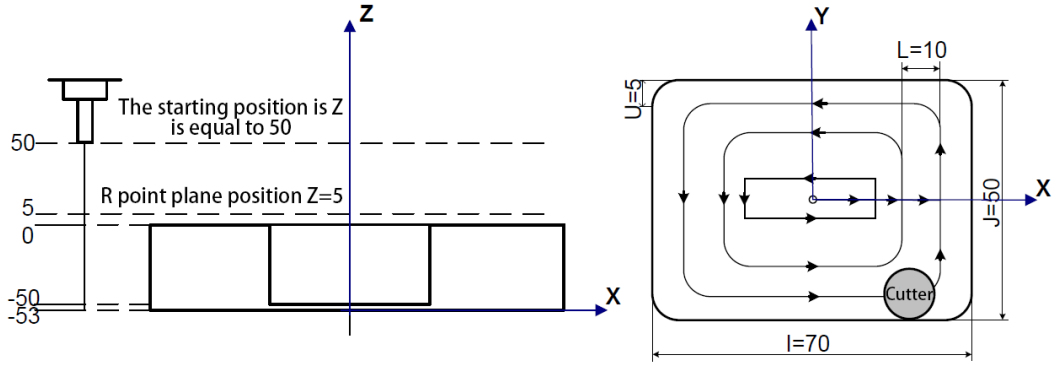


Fig. 3-4-4-2

G90 G00 X50 Y50 Z50； (G00 Quick Positioning).

G99 G33 X25 Y25 Z-50 R5 I70 J50 L10 W20 Q10 V10 U5 D1 F800； (Perform a rectangular inner groove rough milling cycle).

G80 X50 Y50 Z50； (Unpin the loop and return from the R point plane).

M30；

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO: 52#2 set to 1) and G33/G34, otherwise G33/G34 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.5 Precision milling cycle G35/G36 in rectangular grooves

Instruction format:

G98/G99 { G35 / G36 } X_ Y_ Z_ R_ I_ J_ L_ U_ D_ F_ K_;

Function: The tool is finely milled inside the rectangle at the specified width and direction, and returned after the milling is completed.

Illustrate:

G35: Counterclockwise rectangular groove fine milling cycle.

G36: Precision milling cycle in clockwise rectangular grooves.

X, Y: X, Y plane starting point;

Z: Processing depth, absolute position at G90, relative to R datum at G91;

R:R datum position, absolute position at G90, position relative to the beginning of this program segment at G91 ;

I: The width of the rectangular X axis direction, the value range tool diameter ~ 99999.9999mm ;

J: The width of the rectangular Y axis direction, the value range of the tool diameter ~ 99999.9999mm ;

L: The distance between the starting point of the fine milling and the X axis direction of the rectangular side, the tool diameter of the value range ~ 99999.9999mm ;

U: Corner arc radius, Omissions indicate no corner arc transitions. while0 ＜ U When the radius of the tool is <, the alarm is issued;

D: Tool compensation number, the value range is 0 ~ 256, D0 defaults to 0 Takes out the current tool diameter value according to the given serial number;

K: Number of repetitions.

Cyclic process:

1) Speed positioning to the starting point of the XY plane;
2) Speed down to the R point plane;
3) Cut feed to the bottom of the hole;
4) The starting point is to interpolate the arc with the transition arc 1 as the trajectory;
5) Take 2)-3)-4)-5)-6) as the trajectory for straight line and arc interpolation;
6) Transition arc 7 Circular arc interpolation for the trajectory back to the starting point;
7) Depending on the specified G98 or G99, returns to the initial point plane or R

point plane.

Instruction trajectory:

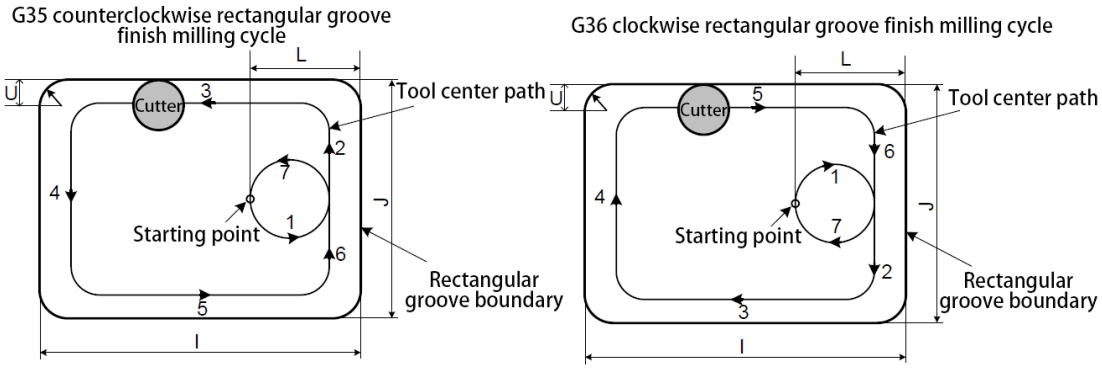


Fig. 3-4-5-1

Note: To use this directive, change NO:14#0 to 1.

Example: Using the fixed cycle G35 command, fine milling of a coarsely milled circular groove as shown in the following figure.

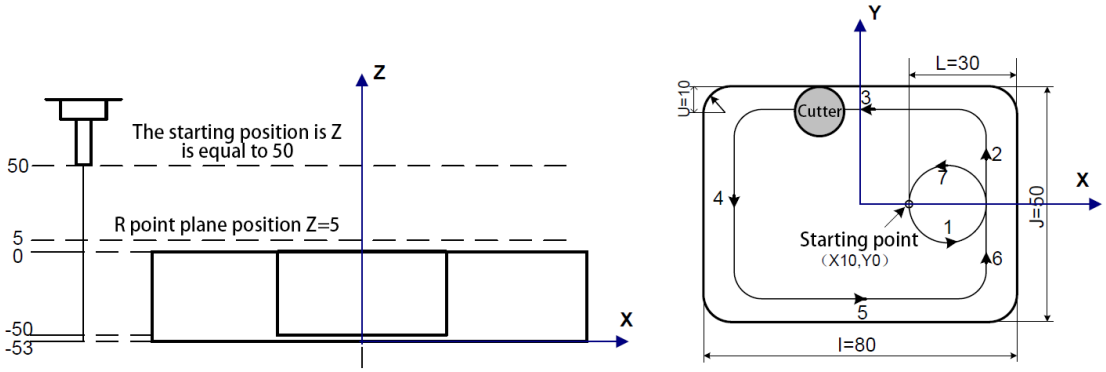


Fig. 3-4-5-2

G90 G00 X50 Y50 Z50； (G00 Quick Positioning).

G99 G35 X10 Y0 Z-50 R5 I80 J50 L30 U10 D1 F800； (Fixed cycle down to the bottom of the hole for rectangular groove internal milling).

G80 X50 Y50 Z50； (Unpin the loop and return from the R point plane).

M30；

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO: 52#2 set to 1) and G35/G36, otherwise G35/G36 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.6 Rectangular external fine milling cycle G37/G38

Instruction format:

G98/G99 { G37 / G38 } X_ Y_ Z_ R_ I_ J_ L_ U_ D_ F_ K_

Function: The tool is finely milled on the outside of the rectangle at the specified width and direction, and returned after the fine milling is completed.

Illustrate:

G37: Counterclockwise rectangular external fine milling cycle.

G38: Clockwise rectangular external fine milling cycle.

X, Y: X, Y plane starting point;

Z: Processing depth, absolute position at G90, relative to R datum at G91;

R:R datum position, absolute position at G90, position relative to the beginning of this program segment at G91 ;

I: The width of the rectangular X axis direction, the value range is 0mm ~ 99999.9999mm ;

J: The width of the rectangular Y axis, the value range is 0mm ~ 99999.9999mm ;

L: The distance between the starting point of fine milling and the direction of the X axis of the rectangular side, the value range is 0mm ~ 99999.9999mm ;

U: corner arc radius, omitted means no corner arc transition;

D: tool compensation number, value range is 0 ~ 256, D0 defaults to 0 , according to the given sequence number to take out the current tool diameter value;

K: Number of repetitions.

Cyclic process:

1) Speed positioning to the starting point of the XY plane;
2) Speed down to the R point plane;
3) Cut feed to the bottom of the hole;
4) The starting point is to interpolate the arc with the transition arc 1 as the trajectory;
5) 2)-3)-4)-5)-6) Straight-line and arc interpolation for the trajectory;
6) Transition arc 7 Circular arc interpolation for the trajectory back to the starting point;
7) Depending on the specified G98 or G99, returns to the initial point plane or R point plane.

Instruction trajectory:

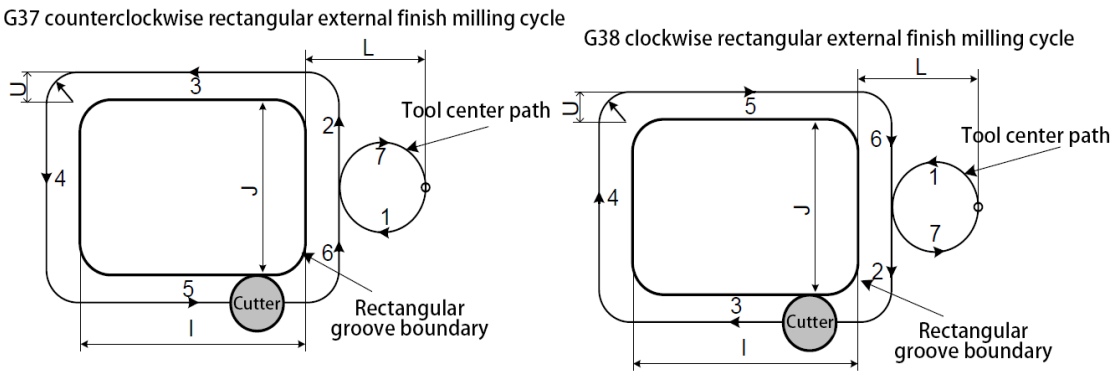


Fig. 3-4-6-1

Related Notes:

When rectangular external milling, the interpolation direction of the transition arc and the fine milling arc is inconsistent, and the interpolation direction in the instruction description refers to the interpolation direction of the fine milling arc.

Example: Rectangular external milling with fixed cycle G37 instructions.

G90 G00 X50 Y50 Z50； (G00 Quick Positioning).

G99 G37 X25 Y25 Z-50 R5 I80 J50 L30 U10 D1F800； (Fixed circulation hole bottom for rectangular external fine milling).

G80 X50 Y50 Z50； (Unpin the loop and return from the R point plane).

M30；

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO:52#2 set to 1) and G37/G38, otherwise G37/G38 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.7 High speed deep hole machining cycle G73

Instruction format: G73 X_Y_Z_R_Q_F_K_

Function: This cycle is specially set for performing high-speed deep hole drilling, which performs intermittent cutting feed up to the bottom of the hole, and quickly retracts from the hole while feeding, removing chips. The schematic diagram of the action is shown in Figure 3-4-7-1.

Illustrate:

X_Y_: hole positioning data;

Z_: incremental programming means specifying the distance from the R point to the bottom of the hole; Absolute programming represents the absolute coordinate value of the bottom of

the hole;

R_: incremental programming represents the distance from the initial point plane to the R point; Absolute programming represents the absolute coordinate value of an R point;

Q_: the depth of the cut feed for each cut;

F_: cutting feed speed;

K_: Number of repetitions.

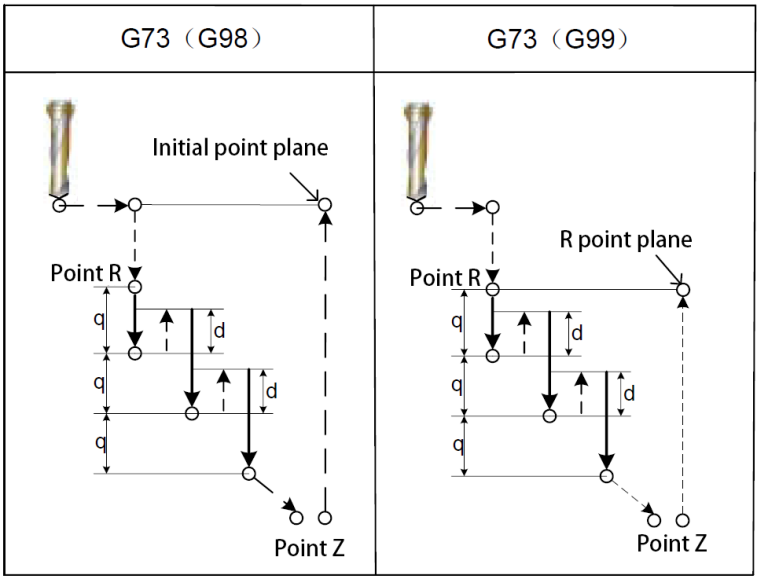


Figure 3-4-7-1

Z, R: When the first drilling is performed, the bottom parameter Z and parameter R are either missing, and the system only changes the modality and does not perform the Z-axis action.

Q: When specifying the instruction parameter Q, an intermittent feed will be made as shown in the figure above. In this case, the system will roll back with the amount of retract d set in the data parameter P334 (as shown in Fig. 3-4-7-1), and the tool will intermittently perform a fast movement return with a distance of d with each feed .

When the G73 instruction and the M instruction are specified in the same program segment, the M code is executed at the same time as the first hole positioning action, and then the system processes the next drilling action.

When you specify the number of repetitions K, the M code is executed only for the first hole, and no M code is executed for subsequent holes .

Note: 1. If you do not specify the command parameter Q, the system will give an alarm prompt: "Address Q is not found or the Q value is 0 (G73/G83)". If the Q value is specified as a negative value, the system will intermittently feed its absolute value.

2. When specifying the tool length bias (G43, G44 or G49) in a fixed cycle, the bias is added or unbiased while **positioning to** the **R point.**

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO:52#2 set to 1) and G73, otherwise G73 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

Example 1

M3 S1500； (The spindle starts to rotate.) 〉

G90 G99 G73 X0 Y0 Z-15 R-10 Q5 F120； (Locate, drill 1 hole, and return to point R).

Y-50； (Position, drill 2 holes, and return to point R).

Y-80； (Position, drill 3 holes, and return to point R).

X10； (Position, drill 4 holes, and return to point R).

Y10； (Position, drill 5 holes, and return to point R).

G98 Y75； (Position, drill 6 holes, and return to the initial position plane).

G80；

G28 G91 X0 Y0 Z0； (Return to reference point).

M5； (Spindle stops rotating).

M30；

Note: In the above example, when performing holes 2 to 6, although Q is omitted, the chip removal action is also performed.

3.4.8 Drilling cycle, point drilling cycle G81

Instruction format: G81 X_ Y_ Z_ R_ F_ K_

Function: This cycle is used as a normal drilling cutting feed, executed to the bottom of the hole, and then the tool moves quickly from the bottom of the hole to return.

Illustrate:

X_Y_: hole positioning data;

Z_: incremental programming means specifying the distance from the R point to the bottom of the hole; Absolute programming represents the absolute coordinate value of the bottom of the hole;

R_: incremental programming represents the distance from the initial point plane to the R point; Absolute programming represents the absolute coordinate value of an R point;

F_: cutting feed speed;

K_: Number of repetitions (if necessary).

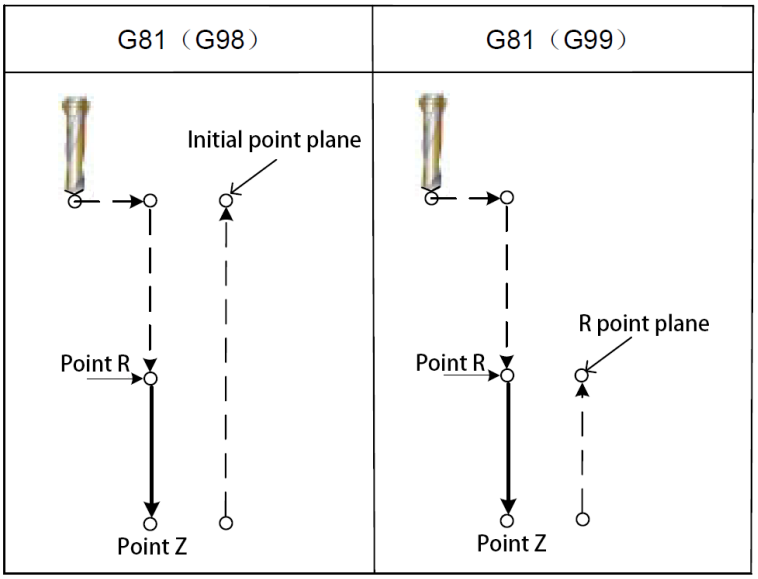


Fig. 3-4-8-1

Z, R: When the first drilling is performed, the bottom parameter Z and parameter R are either missing, and the system only changes the modality and does not perform the Z-axis action.

After positioning along the X and Y axes, quickly move to point R and perform drilling from point R to point Z Then the tool moves quickly back.

Rotate the spindle with the accessibility M code before specifying the G81.

When the G81 instruction and the M instruction are specified in the same program segment, the M code is executed at the same time as the first hole positioning action, and then the system processes the next drilling action.

When you specify the number of repetitions K, the M code is executed only for the first hole, and no M code is executed for subsequent holes .

When specifying a tool length offset G43 G44 or G49 in a fixed loop, the offset is added or unbiased while positioning to the R point.

Example:

M3 S2000 (spindle starts rotating).

G90 G99 G81 X300. Y-250. Z-150. R-10. F120 (positioning, drilling 1 hole, then returning to point R).

Y-550. ； (Position, drill 2 holes, and return to point R).

Y-750. ； (Position, drill 3 holes, and return to point R).

X1000. ； (Position, drill 4 holes, and return to point R).

Y-550. ； (Position, drill 5 holes, and return to point R).

G98 Y-750. ； (Position, drill 6 holes, and return to the initial position plane).

G80；

G28 G91 X0 Y0 Z0 ； (Return to reference point).

M5； (Spindle stops rotating).

M30；

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO: 52#2 set to 1) and G81, otherwise G81 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.9 Drilling cycle, countersinking cycle G82

Instruction format: G82 X_ Y_ Z_ R_ P_ F_ K_;

Function: This cycle is used as a normal drilling hole, the cutting feed is executed to the bottom of the hole, the execution is paused, and then the tool moves quickly from the bottom of the hole to return.

Illustrate:

X_Y_: hole positioning data;

Z_: incremental programming means specifying the distance from the R point to the bottom of the hole; Absolute programming represents the absolute coordinate value of the bottom of the hole;

R_: incremental programming represents the distance from the initial point plane to the R point; Absolute programming represents the absolute coordinate value of an R point;

F_: cutting feed speed;

P_: pause time;

K_: Number of repetitions.

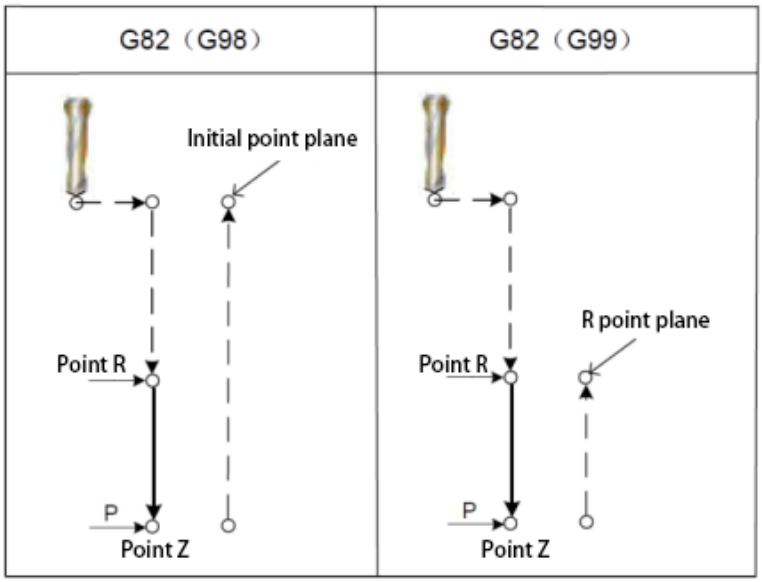


Fig. 3-4-9-1

After positioning along the X and Y axes, quickly move to point R and perform drilling from point R to point Z. When at the bottom of the hole, a pause is performed and then the tool moves quickly back.

Rotate the spindle with the accessibility M code before specifying the G82.

When the G82 instruction and the M instruction are specified in the same program segment, the M code is executed at the same time as the first hole positioning action , and then the system processes the next drilling action.

When you specify the number of repetitions K, the M code is executed only for the first hole, and no M code is executed for subsequent holes .

When specifying a tool length bias G43, G44 or G49 in a fixed cycle , the offset is added or unbiased while positioning to the R point.

P is a modal code instruction, the minimum parameter value is set by the data parameter P336, and the maximum parameter value is set by the data parameter P337 . The p-value is less than the P336 parameter setpoint and runs at the minimum value, and is greater than the P337 parameter setpoint and runs at the maximum. If specified in the program segment where drilling is not performed, P cannot be stored as modal data.

Example:

M3 S2000 (spindle starts rotating).

G90 G99 G82 X300 Y-250 Z-150 -100 P1000 F120 (positioning, drilling 1 hole, pause at the bottom of the hole for 1 second, then return to point R).

Y-550； (Position, drill 2 holes, pause at the bottom of the hole for 1 second, then return to point R).

Y-750； (Position, drill 3 holes, pause at the bottom of the hole for 1 second, then return

to point R).

X1000. ； (Position, drill 4 holes, pause at the bottom of the hole for 1 second, then return to point R).

Y-550； (Position, drill 5 holes, pause at the bottom of the hole for 1 second, then return to point R).

G98 Y-750； (Position, drill 6 holes, pause at the bottom of the hole for 1 second, then return to the initial position plane).

G80； (Unpin Loop).

G28 G91 X0 Y0 Z0 ； (Return to reference point).

M5； (Spindle stops rotating).

M30；

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO: 52#2 set to 1) and G82, otherwise G82 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.10 Chip removal drilling cycle G83

Instruction format: G83 X_ Y_ Z_ R_ Q_ F_ K_

Function: This cycle performs deep hole drilling Perform an intermittent cut feed to the bottom of the hole, which excludes chips from the hole during drilling.

Illustrate:

X_Y_: hole positioning data;

Z_: incremental programming means specifying the distance from the R point to the bottom of the hole; Absolute programming represents the absolute coordinate value of the bottom of the hole;

R_: incremental programming represents the distance from the initial point plane to the R point; Absolute programming represents the absolute coordinate value of an R point;

Q_: the depth of the cut feed for each cut;

F_: cutting feed speed;

K_: Number of repetitions.

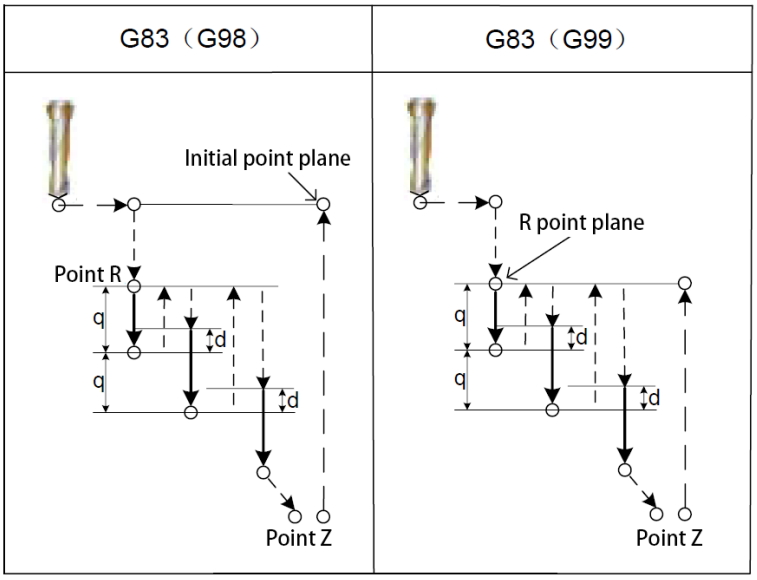


Fig. 3-4-10-1

Q: Indicates the depth of cut fed per cut, it must be expressed in increments. In the second and subsequent feed of the cut, perform a quick move to a point where the distance is d before the end of the last drill, and perform the feed again, with the value of d set by parameter P335. This is shown in Fig. 3-4-10-1.

A positive value must be specified in Q, the negative sign is ignored, and the system still treats it as a positive value.

Q is specified in the program segment where drilling is performed, and if it is specified in the program segment where drilling is not performed, Q cannot be stored as modal data.

Before specifying G83, rotate the spindle (M code) with accessibility.

When the G83 instruction and the M instruction are specified in the same program segment, the M code is executed at the same time as the first hole positioning action , and then the system processes the next drilling action.

When you specify the number of repetitions K, the M code is executed only for the first hole, and no M code is executed for subsequent holes .

When specifying the tool length offset (G43, G44, or G49) in a fixed cycle, the offset is added or unbiased while positioning to the R point.

Example:

M3 S2000； (The spindle starts to rotate).

G90 G99 G83 X300 -250 -150 -100 Q15 F120； (Position, drill 1 hole, and return to point R).

Y-550； (Position, drill 2 holes, and return to point R).

Y-750； (Position, drill 3 holes, and return to point R).

X1000； (Position, drill 4 holes, and return to point R).

```
Y-550；     (Position, drill 5 holes, and return to point R).
G98 Y-750；     (Position, drill 6 holes, and return to the initial position plane).
G80；
G28 G91 X0 Y0 Z0 ；                    (Return to reference point).
M5；                                    (Spindle stops rotating).
M30；
```

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO: 52#2 set to 1) and G83, otherwise G83 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.11 Right tapping cycle G84

Instruction format:

M29 S_

G84 X_ Y_ Z_ R_ P_ F_

Function: This loop performs tapping. In this tapping cycle, the spindle rotates in the opposite direction when it reaches the bottom of the hole.

Illustrate:

X_Y_: hole positioning data;

Z_: incremental programming means specifying the distance from the R point to the bottom of the hole; Absolute programming represents the absolute coordinate value of the bottom of the hole;

R_: incremental programming represents the distance from the initial point plane to the R point; Absolute programming represents the absolute coordinate value of an R point;

P_: pause time;

F_: cutting feed speed.

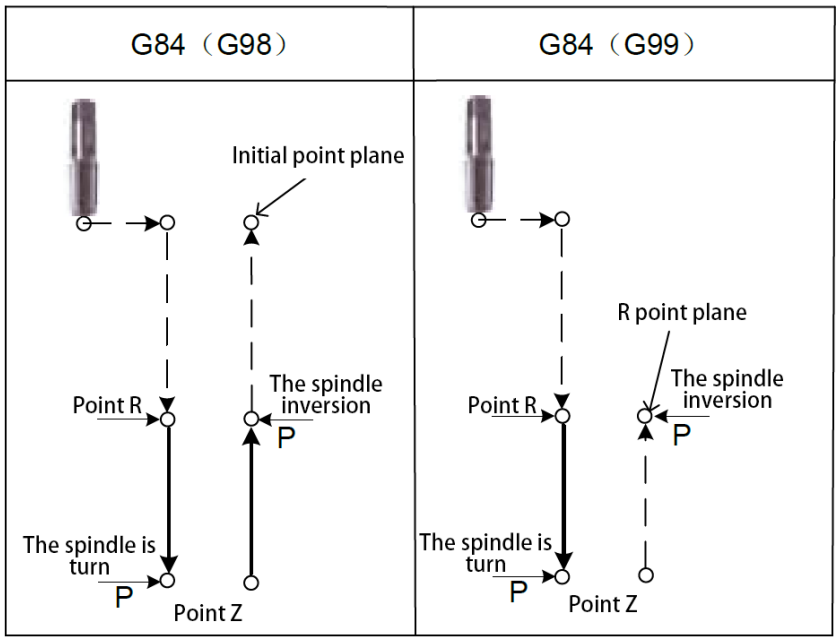


Fig. 3-4-11-1

The spindle rotates clockwise to perform tapping, and when it reaches the bottom of the hole, in order to rewind the spindle, rotating in the opposite direction, this process generates threads.

During tapping, the feed multiplier is ignored. The feed is paused, and the machine is not stopped until the return action is completed.

Switch the system to tapping mode with the auxiliary function M29 code before specifying the G84. If the spindle is not ordered to rotate clockwise, the system automatically adjusts to rotate clockwise according to the current spindle command speed in the R plane.

When the G84 instruction and the M instruction are specified in the same program segment, the M code is executed at the same time as the first hole positioning action , and then the system processes the next drilling action.

When you specify the number of repetitions K, the M code is executed only for the first hole, and no M code is executed for subsequent holes .

P is a modal instruction, the minimum parameter value is set by the data parameter P336, and the maximum parameter value is set by the data parameter P337 . The p-value is less than the P336 parameter setpoint and runs at the minimum value, and is greater than the P337 parameter setpoint and runs at the maximum. If specified in the program segment where drilling is not performed, P cannot be stored as modal data.

When specifying a tool length offset G43 G44 or G49 in a fixed loop, the offset is added or unbiased while the positioning to the R point is performed.

In the per-minute feed mode, the relationship between the thread lead and the feed speed and the spindle speed is as follows

Feed speed F = tap pitch × spindle speed S

Such as: tap the threaded hole of M12×1.5 on the part, and the parameters can be selected;

S500=500 r /min； F=1.5×500=750mm/min；

When threaded for multiple heads, multiply by the number of heads to get the F value.

Example:

M29 S100 ； (Enter tapping mode).

G90 G99 G84 X300 -250 Z-150 -120 P300 F120 (positioning, tapping 1 hole, then back to point R).

Y-550； (Position, tapping the 2 holes, then returning to point R).

Y-750； (Position, tapping 3 holes, then returning to point R).

X1000； (Position, tapping the 4 holes, then returning to point R).

Y-550； (Position, tapping 5 holes, then returning to point R).

G98 Y-750； (Position, tapping 6 holes, then returning to the initial position plane).

G80；

G28 G91 X0 Y0 Z0 ； (Return to reference point).

M5； (Spindle stops rotating).

M30；

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO:52#2 set to 1) and G84, otherwise G84 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.12 L-tapping loop G74

Instruction format:

M29 S_

G74 X_ Y_ Z_ R_ P_ F_

Function: This loop performs tapping. In this tapping cycle, the spindle rotates in the opposite direction when it reaches the bottom of the hole.

Illustrate:

X_Y_: hole positioning data;

Z_: incremental programming means specifying the distance from the R point to the bottom of the hole; Absolute programming represents the absolute coordinate value of the bottom of the hole;

R_: incremental programming represents the distance from the initial point plane to the R point; Absolute programming represents the absolute coordinate value of an R point;

F_: cutting feed speed.

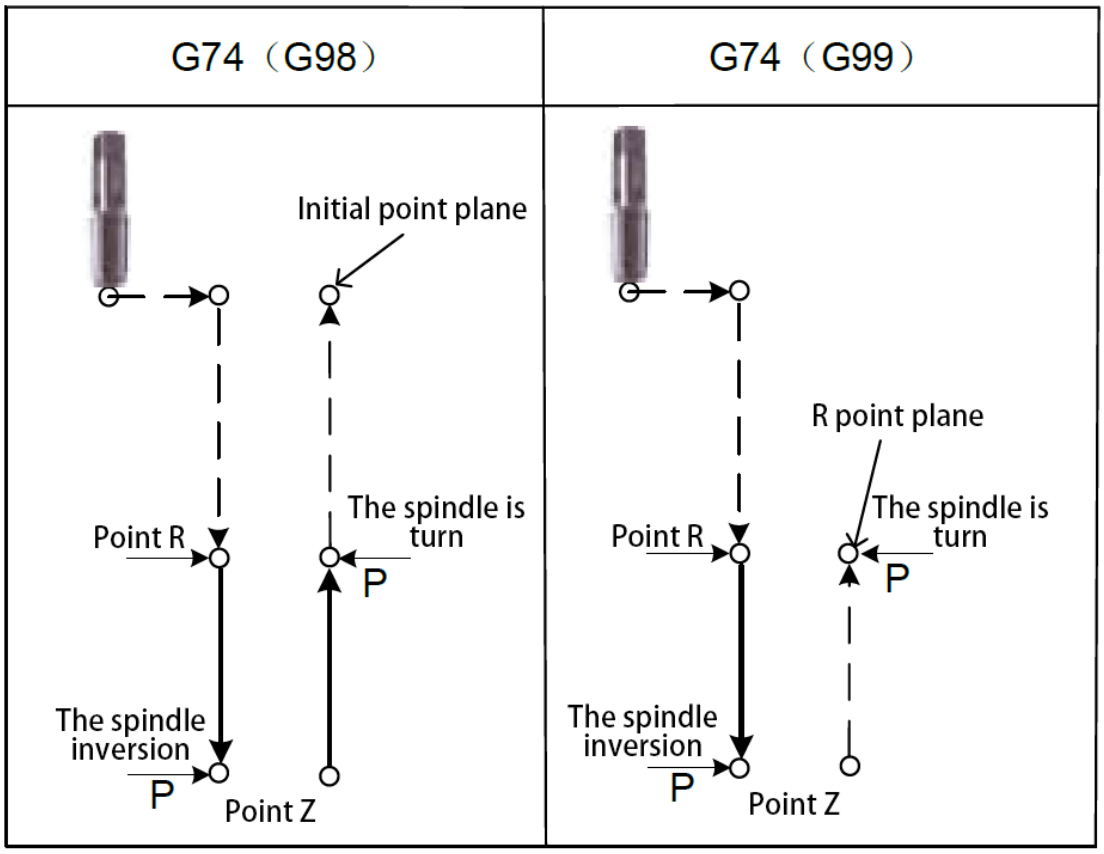


Fig. 3-4-12-1

The spindle rotates counterclockwise to perform tapping, and when it reaches the bottom of the hole, rotates in the opposite direction in order to rewind the spindle. This process generates threads.

During tapping, the feed magnification is ignored, the feed is paused, and the machine is not stopped until the return action is completed.

Before specifying the G74, use the accessibility M29 code to put the system into tapping mode. If the spindle is not instructed to rotate counterclockwise, the system automatically adjusts to rotate counterclockwise according to the current spindle command speed in the R plane.

When the G74 instruction and the M instruction are specified in the same program segment, the M code is executed at the same time as the first hole positioning action , and then the system processes the next drilling action.

When you specify the number of repetitions K, the M code is executed only for the first hole, and no M code is executed for subsequent holes .

P is a modal instruction, the minimum parameter value is set by the data parameter P336, and the maximum parameter value is set by the data parameter P337 . The p-value is less than the P336 parameter setpoint and runs at the minimum value, and is greater than the P337 parameter setpoint and runs at the maximum. If specified in the program segment where drilling

is not performed, P cannot be stored as modal data.

When specifying a tool length offset G43 G44 or G49 in a fixed loop, the offset is added or unbiased while the positioning to the R point is performed.

Example:

M29 S100； (Enter tapping mode).

G90 G99 G74 X300. Y-250. Z-150. R-120 P300 F120 (positioning, tapping 1 hole, then return to point R).

Y-550； (Position, tapping the 2 holes, then returning to point R).

Y-750； (Position, tapping 3 holes, then returning to point R).

X1000； (Position, tapping the 4 holes, then returning to point R).

Y-550； (Position, tapping 5 holes, then returning to point R).

G98 Y-750； (Position, tapping 6 holes, then returning to the initial position plane).

G80；

G28 G91 X0 Y0 Z0； (Return to reference point).

M5； (Spindle stops rotating).

M30；

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO:52#2 set to 1) and G74, otherwise G74 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.13 Fine boring cycle G76

Instruction format: G76 X_Y_Z_Q_R_P_F_K_

Function:

Fine boring cycle is suitable for fine boring of holes.

When the bottom of the hole is reached, the spindle stops and the cutting tool leaves the surface where the workpiece is machined and returns.

Prevent the appearance of retract marks during retraction, affect the finish of the machined surface, and avoid tool damage.

Illustrate:

X_Y_: hole positioning data;

Z_: incremental programming means specifying the distance from the R point to the bottom of the hole; Absolute programming represents the absolute coordinate value of the bottom of the hole;

R_: incremental programming represents the distance from the initial point plane to the R point; Absolute programming represents the absolute coordinate value of an R point;

Q_: the offset of the bottom of the hole;

P_: pause time;

F_: cutting feed speed;

K_: the number of fine boring.

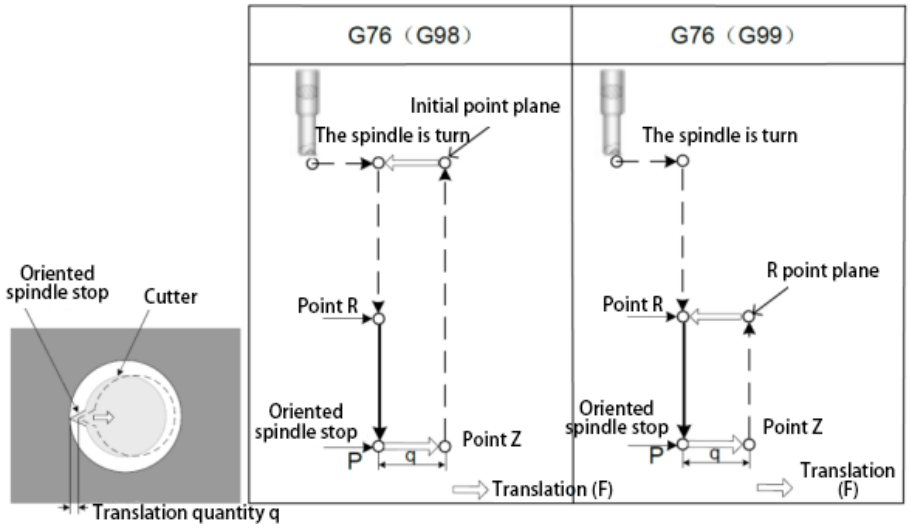


Fig. 3-4-13-1

When the tool reaches the bottom of the hole, the spindle stops at a fixed swivel position and the tool moves in the opposite direction of the tip to retract. This ensures that the machining surface is not destroyed and enables precise and efficient boring. Parameter Q specifies the distance to retract. The retract axis and direction are specified by the bit parameters N0:44#4 and N0:44#5, and the Q value must be is a positive value. Even with negative values, the symbol does not work. The offset of Q at the bottom of the hole is the modal value saved within the fixed cycle and must be specified carefully. Because it is also used as a cutting depth for G73 and G83.

Before specifying the G76, rotate the spindle with the accessibility M code.

When the G76 instruction and the M instruction are specified in the same program segment, the M code is executed at the same time as the first hole positioning action , and then the system processes the next action.

When you specify the number of repetitions K, the M code is executed only for the first hole, and no M code is executed for subsequent holes .

When specifying a tool length offset G43, G44, or G49 in a fixed cycle, the offset is added or unbiased while the positioning to the R point is performed.

Shaft switching: The fixed cycle must be unfixed before changing the drilling shaft.

Boring: No boring is performed in program segments that do not contain X, Y, Z, R or other

axes.

Example:

M3 S500 (spindle starts rotating).

G90 G99 G76 X300 Y-250 (positioning, boring 1 hole, then return to point R).

Z-150 R-100 Q5 (bottom of the hole is oriented and then moved 5 mm).

P1000 F120； (Stop at the bottom of the hole for 1 s).

Y-550； (Position, boring 2 holes, and return to point R).

Y-750； (Position, boring 3 holes, and return to point R).

X1000； (Position, boring 4 holes, then return to point R).

Y-550； (Position, boring 5 holes, then return to point R).

G98 Y-750； (Position, boring 6 holes, then return to the initial position plane).

G80 G28 G91 X0 Y0 Z0； (Return to reference point).

M5； (Spindle stops rotating).

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO:52#2 set to 1) and G76, otherwise G76 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.14 Boring cycle G85

Instruction format: G85 X_ Y_ Z_ R_ F_ K_

Function: This cycle is used for boring.

Illustrate:

X_Y_: hole positioning data;

Z_: incremental programming means specifying the distance from the R point to the bottom of the hole; Absolute programming represents the absolute coordinate value of the bottom of the hole;

R_: incremental programming represents the distance from the initial point plane to the R point; Absolute programming represents the absolute coordinate value of an R point;

F_: cutting feed speed;

K_: Number of repetitions.

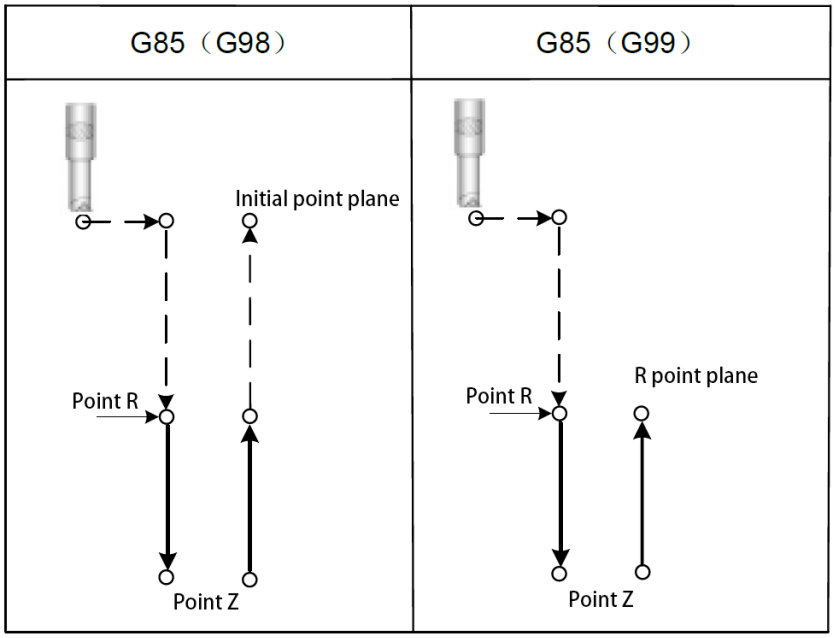


Fig. 3-4-14-1

After positioning along the X and Y axes, quickly move to point R, then perform boring from point R to point Z , and when you reach the bottom of the hole, perform the cutting feed and then return to R point.

Rotate the spindle with the accessibility M code before specifying the G85.

When the G85 instruction and the M instruction are specified in the same program segment, the M code is executed at the same time as the first hole positioning action , and then the system processes the next action.

When you specify the number of repetitions K, the M code is executed only for the first hole, and no M code is executed for subsequent holes .

When specifying tool length offsets G43, G44, and G49 in a fixed cycle, the offset is added while positioning to point R.

Shaft switching: The fixed cycle must be unfixed before changing the drilling shaft.

Boring: No boring is performed in a program section that does not contain X, Y, Z, R or other axes.

Example:

M3 S100 ； (The spindle starts to rotate).

G90 G99 G85 X300 Y-250 Z-150 R-120 F120 (positioning, boring 1 hole, then return to point R).

Y-550； (Position, boring 2 holes, and return to point R).

Y-750； (Position, boring 3 holes, and return to point R).

X1000； (Position, boring 4 holes, then return to point R).

Y-550； (Position, boring 5 holes, then return to point R).

G98 Y-750； (Position, boring 6 holes, and then return to the initial position plane).

G80；

G28 G91 X0 Y0 Z0 ； (Return to reference point).

M5； (Spindle stops rotating).

M30；

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO:52#2 set to 1) and G85, otherwise G85 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.15 Boring cycle G86

Instruction format: G86 X_ Y_ Z_ R_ F_ K_;

Function: This cycle instruction is used for boring cycles.

Illustrate:

X_Y_: hole positioning data;

Z_: incremental programming means specifying the distance from the R point to the bottom of the hole; Absolute programming represents the absolute coordinate value of the bottom of the hole;

R_: incremental programming represents the distance from the initial point plane to the R point; Absolute programming represents the absolute coordinate value of an R point;

F_: cutting feed speed;

K_: the number of times the machining is repeated.

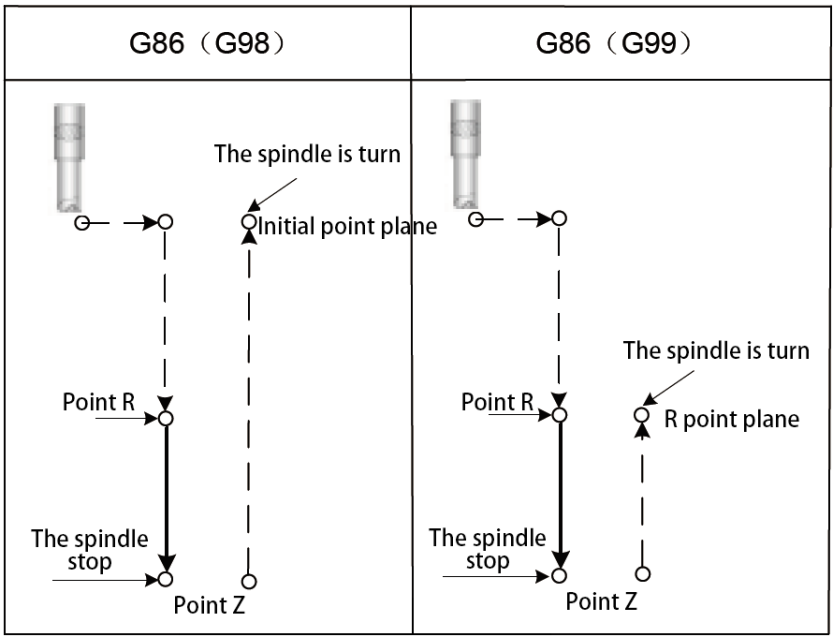


Fig. 3-4-15-1

Once positioned along the X and Y axes, quickly move to point R and perform boring from point R to point Z. When the spindle stops at the bottom of the hole, the tool rewinds with a fast movement.

Before specifying G86, rotate the spindle with the accessibility M code.

When the G86 instruction and the M instruction are specified in the same program segment, the M code is executed at the same time as the first hole positioning action , and then the system processes the next action. When you specify the number of repetitions K, the M code is executed only for the first hole, and no M code is executed for subsequent holes . When specifying tool length bias G43 G44 or G49 in a fixed cycle, offset or unbias while positioning to the R point.

Shaft switching: The fixed cycle must be unfixed before changing the drilling shaft.

Boring: No boring is performed in a program section that does not contain X, Y, Z, R or other axes.

Example:

M3 S2000； (The spindle starts to rotate).

G90 G99 G86 X300 Y-250 Z-150 R-100 F120 (positioning, boring 1 hole, then return to point R).

Y-550； (Position, boring 2 holes, and return to point R).

Y-750； (Position, boring 3 holes, and return to point R).

X1000； (Position, boring 4 holes, then return to point R).

Y-550； (Position, boring 5 holes, then return to point R).

G98 Y-750； (Position, boring 6 holes, and then return to the initial position plane).

G80；

G28 G91 X0 Y0 Z0 ； (Return to reference point).

M5； (Spindle stops rotating).

M30；

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO: 52#2 set to 1) and G86, otherwise G86 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.16 Boring cycle, back boring cycle G87

Instruction format: G87 X_Y_Z_R_Q_P_ F_;

Function: This cycle performs precision boring

Illustrate:

X_Y_: hole positioning data;

Z_: incremental programming means specifying the distance from R point to Z point; Absolute programming represents the absolute coordinate value of a Z point;

R_: incremental programming represents the distance from the initial point plane to the R point; Absolute programming represents the absolute coordinate value of an R point; (bottom of the hole).

Q_: the offset of the bottom of the hole;

P_: pause time;

F_: cutting feed speed;

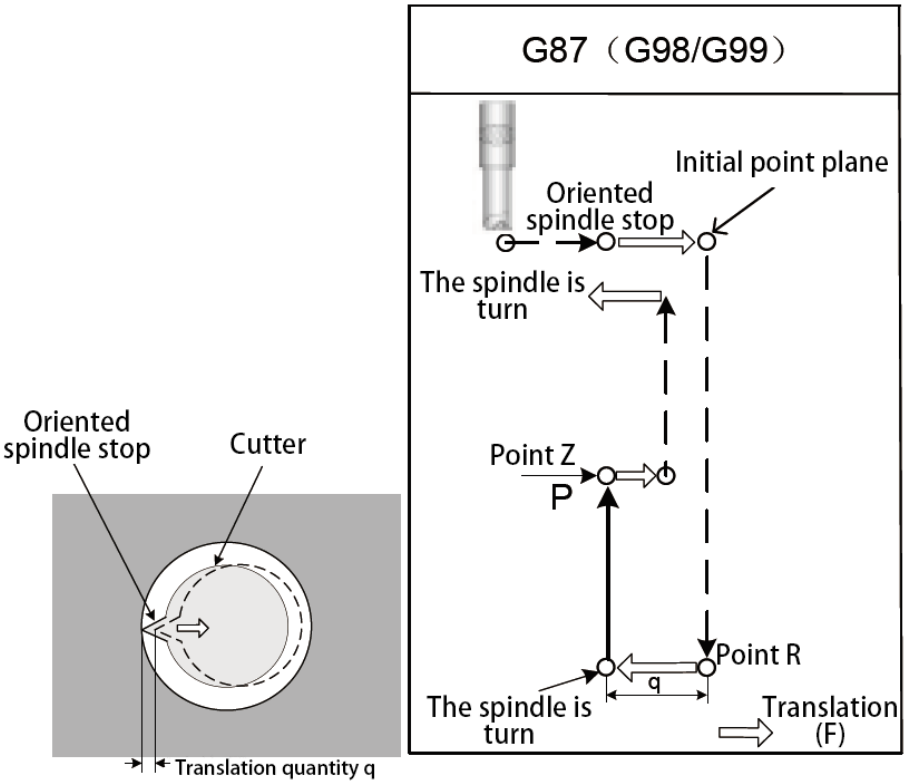


Fig. 3-4-16-1

After positioning along the X and Y axes, the tool is stopped after the spindle is oriented, moves in the opposite direction of the tip, and moves at the feed speed at the R point at the bottom of the hole, then the tool moves in the direction of the tip, and the spindle is rotated positively, along Z The positive boring of the shaft is until point Z, and after the axis at point Z is oriented again, the spindle stops at a fixed rotational position, and the tool moves in the opposite direction of the tip to retract the knife, and then the tool returns to the initial plane. The tool shifts the spindle forward in the direction of the tip of the tool and performs the machining of the next program section.

The parameter Q value specifies the distance to retract. The retract direction is specified by the system parameters N0:44#4 and N0:44#5, and the Q value must be positive.

Even with negative values, the symbol does not work. The offset of Q at the bottom of the hole is the modal value saved within the fixed cycle and must be specified carefully, as it is also used as the cutting depth for G73 and G83.

Rotate the spindle with the accessibility M code before specifying the G87.

When the G87 instruction and the M instruction are specified in the same program segment, the M code is executed at the same time as the first hole positioning action , and then the system processes the next drilling action.

When you specify the number of repetitions K, the M code is executed only for the first hole, and no M code is executed for subsequent holes .

When specifying tool length bias G43, G44, or G49 in a fixed cycle, the offset is added or unbiased while positioning to the R point.

Shaft switching: The fixed cycle must be untied before switching the boring shaft.

Boring: No boring is performed in a program section that does not contain X, Y, Z, R or other auxiliary axes.

Prompt:

When programming the back boring cycle, keep in mind that the Z value and R are worth specifying, and in general, the Z position here is above the R position. Otherwise, the system will alert.

Example:

M3 S500 ； (The spindle starts to rotate).

G90 G99 G87 X300 Y-250 Z-120 R-150 Q5 P1000 F120 ；

(Positioning, boring 1 hole, orienting at the initial position and then offsetting 5mm to stop at the Z point for 1 sec).

Y-550 ； (Position, boring 2 holes, then return to point R).

Y-750 ； (Position, boring 3 holes, then return to point R).

X1000 ； (Position, boring 4 holes, then return to point R).

Y-550 ； (Position, boring 5 holes, then return to point R).

G98 Y-750 ； (Position, boring 6 holes, then return to the initial position plane).

G80 G28 G91 X0 Y0 Z0 ； (Return to reference point).

M5 ； (Spindle stops rotating).

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO:52#2 set to 1) and G87, otherwise G87 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.17 Boring cycle G88

Instruction format: G88 X_Y_Z_R_ P_F_

Function: This cycle is used for boring

Illustrate:

X_Y_: hole positioning data;

Z_: incremental programming means specifying the distance from the R point to the bottom of the hole; Absolute programming represents the absolute coordinate value of the bottom of

the hole;

R_: incremental programming represents the distance from the initial point plane to the R point; Absolute programming represents the absolute coordinate value of an R point;

P_: pause time;

F_: cutting feed speed.

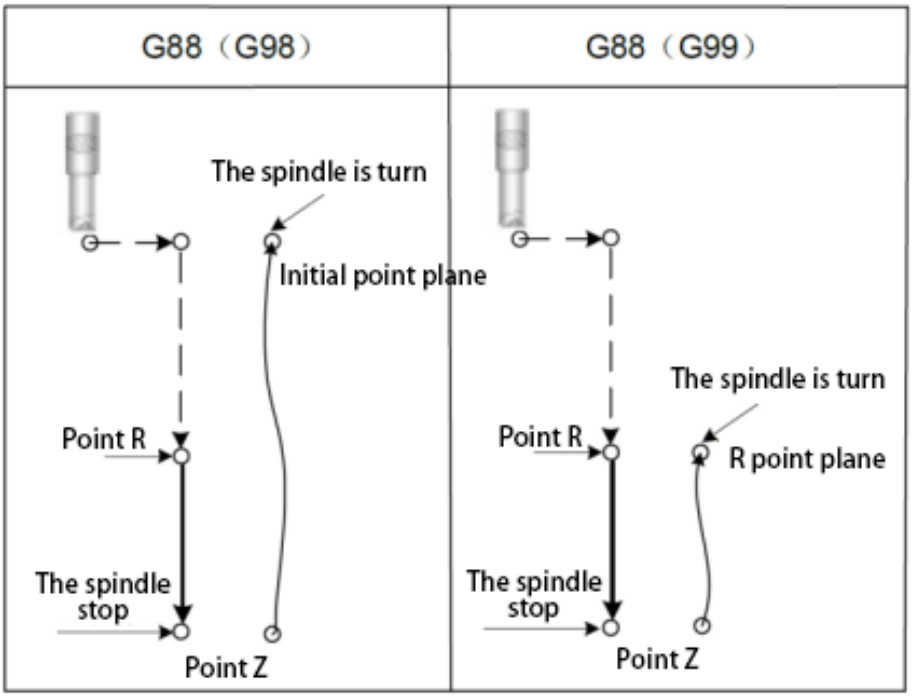


Fig. 3-4-17-1

After positioning along the X and Y axes, quickly move to point R, then perform boring from point R to point Z, when boring is complete, perform a pause, then the spindle stops, the tool is performed After manually returning from the bottom Z point to the R point (G99 case) or the initial point (G98 case), the spindle forward rotation begins.

Before specifying the G88, rotate the spindle with the accessibility M code.

When the G88 instruction and the M instruction are specified in the same program segment, the M code is executed at the same time as the first hole positioning action , and then the system processes the next drilling action.

When you specify the number of repetitions K, the M code is executed only for the first hole, and no M code is executed for subsequent holes .

P is a modal instruction, the minimum parameter value is set by the data parameter P336, and the maximum parameter value is set by the data parameter P337 . The p-value is less than the P336 parameter setpoint and runs at the minimum value, and is greater than the P337 parameter setpoint and runs at the maximum. If specified in the program segment where drilling is not performed, P cannot be stored as modal data.

When specifying tool length bias G43 G44 or G49 in a fixed cycle, offset or unbias while

positioning to point R.

Shaft switching: The fixed cycle must be untied before switching the boring shaft.

Boring: No boring is performed in a program section that does not contain X, Y, Z, R or other auxiliary axes.

Example:

M3 S2000； (The spindle starts to rotate).

G90 G99 G88 X300 Y-250 Z-150 R-100 P1000 F120 (positioning, boring 1 hole, then return to point R).

Y-550； (Position, boring 2 holes, then return to point R).

Y-750； (Position, boring 3 holes, then return to point R).

X1000； (Position, boring 4 holes, then return to point R).

Y-550； (Position, boring 5 holes, then return to point R).

G98 Y-750； (Position, boring 6 holes, then return to the initial position plane).

G80 G28 G91 X0 Y0 Z0； (Return to reference point).

M5； (Spindle stops rotating).

M30;

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO: 52#2 set to 1) and G88, otherwise G88 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.18 Well cycle G89

Instruction format: G89 X_ Y_ Z_ R_ P_ F_ K_

Function: This cycle is used for boring.

Illustrate:

X_Y_: hole positioning data;

Z_: incremental programming means specifying the distance from the R point to the bottom of the hole; Absolute programming represents the absolute coordinate value of the bottom of the hole;

R_: incremental programming represents the distance from the initial point plane to the R point; Absolute programming represents the absolute coordinate value of an R point;

P_: pause time;

F_: cutting feed speed;

K_: Number of repetitions.

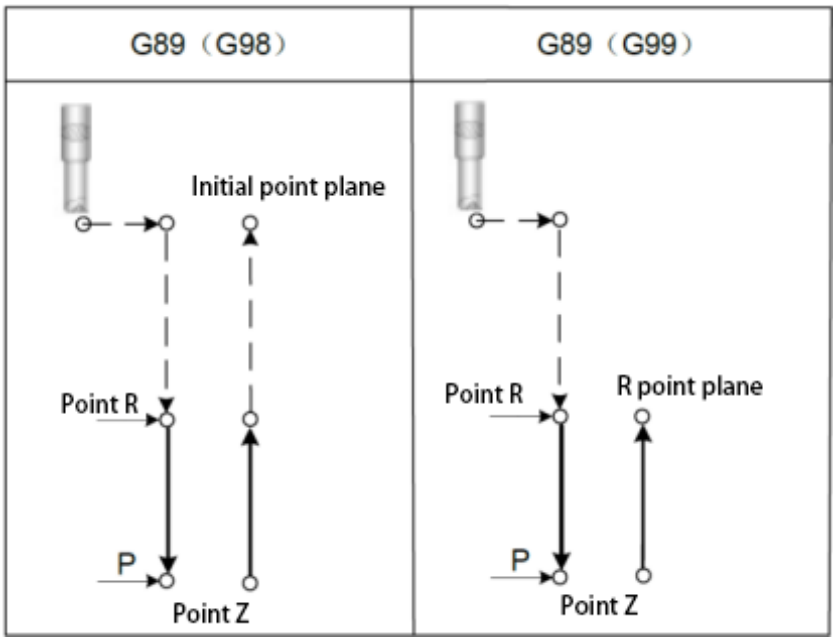


Fig. 3-4-18-1

The cycle is almost identical to the G85, except that the cycle is paused at the bottom of the hole.

Rotate the spindle with the accessibility M code before specifying the G89.

When the G89 instruction and the M instruction are specified in the same program segment, the M code is executed at the same time as the first hole positioning action , and then the system processes the next drilling action.

When you specify the number of repetitions K, the M code is executed only for the first hole, and no M code is executed for subsequent holes .

P is a modal instruction, the minimum parameter value is set by the data parameter P336, and the maximum parameter value is set by the data parameter P337 . The p-value is less than the P336 parameter setpoint and runs at the minimum value, and is greater than the P337 parameter setpoint and runs at the maximum. If specified in the program segment where drilling is not performed, P cannot be stored as modal data.

When specifying a tool length offset G43, G44, or G49 in a fixed cycle, the offset is added while positioning to point R.

Shaft switching: The fixed cycle must be untied before switching the boring shaft.

Boring: No boring is performed in a program section that does not contain X, Y, Z, R or other auxiliary axes.

Example:

M3 S100； (The spindle starts to rotate).

G90 G99 G89 X300 Y-250 Z-150 R-120 P1000 F120；

(Position, boring 1 well, then return to point R and stop at the bottom of the hole for 1

sec).

Y-550； (Position, boring 2 holes, and return to point R).

Y-750； (Position, boring 3 holes, and return to point R).

X1000； (Position, boring 4 holes, then return to point R).

Y-550； (Position, boring 5 holes, then return to point R).

G98 Y-750； (Position, boring 6 holes, and then return to the initial position plane).

G80；

G28 G91 X0 Y0 Z0 ； (Return to reference point).

M5； (Spindle stops rotating).

M30；

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO: 52#2 is set to 1) and G89, otherwise G89 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

3.4.19 L-Stiff Tapping G74

Instruction format:

M29 S_

G74 X_Y_Z_R_P_F_K_

Function: In the rigid mode, the spindle motor works as a servo motor, which can realize left-handed high-speed high-precision tapping.

Illustrate:

X_Y_: Hole positioning data

Z_: incremental programming means specifying the distance from the R point to the bottom of the hole; Absolute programming represents the absolute coordinate value of the bottom of the hole.

R_: incremental programming represents the distance from the initial point plane to the R point; Absolute programming represents the absolute coordinate value of an R point.

P_: The pause time at point R at the bottom of the hole or at the time of fallback.

F_: cutting feed speed.

K_: Number of repetitions.

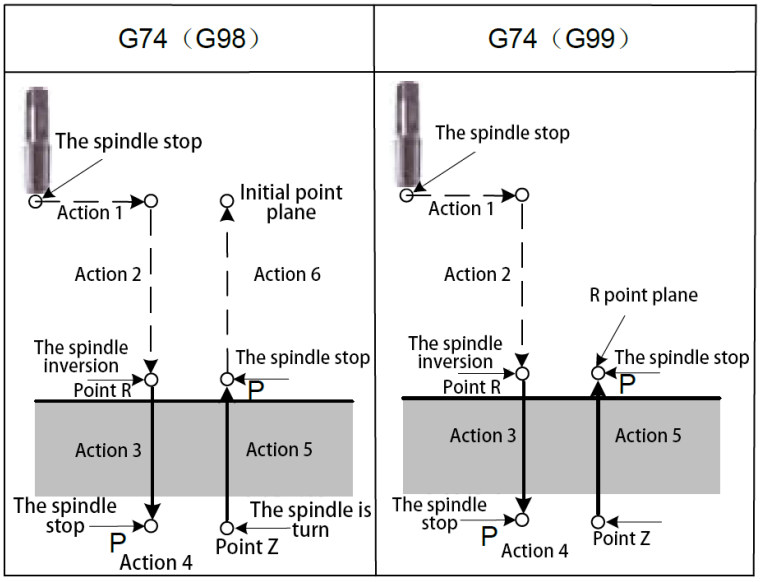


Fig. 3-4-19-1

After positioning along the X and Y axes, the Z axis quickly moves to point R, and the G74 spindle begins to invert, from point R to Z Point execution tapping, when tapping is complete, the spindle stops and performs a pause, then the spindle rotates the tool in the opposite direction to retreat to point R, the spindle stops, and then performs a quick move to the initial position. The feed velocity multiplier and spindle magnification are considered to be 100% while tapping is being executed.

Rigid mode:

You can specify the rigidity mode in any of the following ways:

(1) Specify M29 S***** before the tapping command section

(2) Specify M29 S***** in the program segment containing tapping instructions

When the G74 instruction and the M instruction are specified in the same program segment, the M code is executed at the same time as the first hole positioning action , and then the system processes the next drilling action.

When you specify the number of repetitions K, the M code is executed only for the first hole, and no M code is executed for subsequent holes .

P is a modal instruction, the minimum parameter value is set by the data parameter P336, and the maximum parameter value is set by the data parameter P337 . The p-value is less than the P336 parameter setpoint and runs at the minimum value, and is greater than the P337 parameter setpoint and runs at the maximum. If specified in the program segment where drilling is not performed, P cannot be stored as modal data.

Tool length compensation, if the tool length compensation G43, G44 or G49 is specified in the fixed cycle mode, the offset is added or unbiased while the R point is positioned.

Axis switching: The fixed cycle must be untied before switching the tapping axis. If the

tapping axis is changed in a rigid mode, an alarm is issued.

If S and axis movement commands are specified between M29 and G84, the system alarms. If M29 is specified in the tapping cycle, the system alarms.

Thread lead with expression: feed speed / spindle speed.

Z-axis feed speed = spindle speed * thread lead.

Example:

Spindle speed 1000rpm;

Thread lead 1.0mm;

Then, the Z axis feed speed = 1000*1=1000mm/min;

G00 X120 Y100 ； (Positioning).

M29 S1000 ； (specify the rigidity mode).

G74 Z-100 R-20 F1000 ； (Rigid tapping).

Limit:

F: An alarm is issued if the specified F value exceeds the upper limit of the cutting feed speed.

S: If the speed is higher than the maximum speed of the specified grade, the alarm is issued. The speed grade is set by the data parameter P209 (tapping cycle upper speed).

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameters NO:48#0 NO:14#0 set to 1) and G74, otherwise G74 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

Program restart: Restarting the program during rigid tapping is invalid.

Special case: When the position parameter 31.0=1 tapping pitch F is set, the programming tapping F is used to edit the tap pitch;

Example:

Spindle speed 1000rpm;

Thread lead 1.0mm;

Then, the Z axis feed speed = 1000*1=1000mm/min;

G00 X120 Y100 ； (Positioning).

M29 S1000 ； (specify the rigidity mode).

G74 Z-100 R-20 F1. 0 ； (Rigid tapping).

G80;

3.4.20 Right-hand rigid tapping G84

Instruction format:

M29 S_

G84 X_Y_Z_R_P_F_K_

Function: In the rigid mode, the control of the spindle motor is a servo motor, which can achieve high-speed and high-precision tapping. It can be guaranteed that the starting position of the tapping is consistent if the R point does not change. That is, the tapping command is repeated many times in one position, and the thread will not be snapped and rotten.

Illustrate:

X_Y_: hole positioning data;

Z_: incremental programming means specifying the distance from the R point to the bottom of the hole; Absolute programming represents the absolute coordinate value of the bottom of the hole;

R_: incremental programming represents the distance from the initial point plane to the R point; Absolute programming represents the absolute coordinate value of an R point;

P_: pause time at point R at the bottom of the hole or time at point R when rewinding;

F_: cutting feed speed;

K_: Number of repetitions.

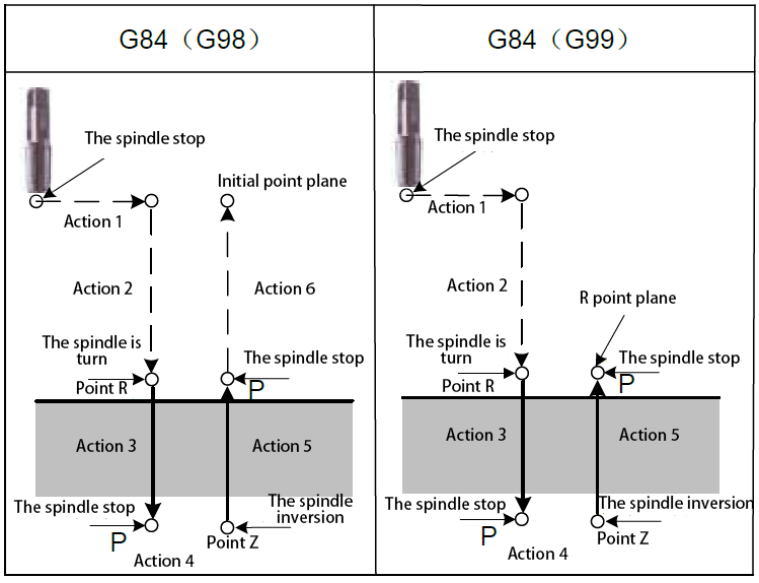


Fig. 3-4-20-1

After positioning along the X and Y axes, the Z axis performs a quick move to point R and the G84 spindle starts to rotate positively, from point R to Z Point execution tapping, when the tapping is complete, the spindle stops and performs a pause, then the spindle rotates the tool in the opposite direction, retreating to the R point and the spindle stops. Then perform a

quick move to the initial position.

The feed velocity magnification and spindle magnification are considered to be 100% while the tapping is being executed.

Rigid mode:

You can specify the rigidity mode in any of the following ways:

(1) Specify M29 S***** before the tapping command section

(2) Specify M29 S***** in the program segment containing tapping instructions

When the G84 instruction and the M instruction are specified in the same program segment, the M code is executed at the same time as the first hole positioning action , and then the system processes the next drilling action.

When you specify the number of repetitions K, the M code is executed only for the first hole, and no M code is executed for subsequent holes .

P is a modal instruction, the minimum parameter value is set by the data parameter P336, and the maximum parameter value is set by the data parameter P337 . The p-value is less than the P336 parameter setpoint and runs at the minimum value, and is greater than the P337 parameter setpoint and runs at the maximum. If specified in the program segment where drilling is not performed, P cannot be stored as modal data.

Tool length compensation, if the tool length compensation G43 G44 or G49 is specified in fixed cycle mode, the offset is added or unbiased while positioning to the R point.

Axis switching: The fixed cycle must be untied before switching the tapping axis. If the tapping axis is changed in a rigid mode, an alarm is issued.

If S and axis movement commands are specified between M29 and G84, the system alarms. If M29 is specified in the tapping cycle, the system alarms.

In the per-minute feed mode, the thread lead is expressed as "feed speed / spindle speed".

Z-axis feed speed = spindle speed * thread lead.

Example:

Spindle speed 1000r/min;

Thread lead 1.0mm;

Then, the Z axis feed speed = 1000*1=1000mm/min;

G00 X120 Y100 ； (Positioning).

M29 S1000 ； (specify the rigidity mode).

G84 Z-100 R-20 F1000 ； (Rigid tapping).

G80;

Limit:

F: An alarm is issued if the specified F value exceeds the upper limit of the cutting feed speed.

S: If the speed is higher than the maximum speed of the specified grade, the alarm is issued. The speed grade is set by the data parameter P209 (tapping cycle upper speed).

Cancel: You cannot specify group 01 G codes (G00 to G03) and G60 as modal G codes in the same program segment (bit parameter NO: 48#0 is set to 1) and G84, otherwise G84 will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

Program restart: Restarting the program during rigid tapping is invalid.

Special case: When the position parameter 31.0=1 tapping pitch F is set, the programming tapping F is used to edit the tap pitch;

Example:

Spindle speed 1000rpm;

Thread lead 1.0mm;

Then, the Z axis feed speed = 1000*1=1000mm/min;

G00 X120 Y100 ； (Positioning).

M29 S1000 ； (specify the rigidity mode).

G84 Z-100 R-20 F1. 0 ； (Rigid tapping).

G80;

3.4.21 Deep hole rigid tapping (chip evacuation) cycle

Instruction format:

M29 S_

G84（or G74） X_Y_Z_R_P_Q_F_K_

Function: In deep hole rigid tapping, perform several passes until the bottom of the hole.

Illustrate:

X_Y_: hole positioning data;

Z_: incremental programming means specifying the distance from the R point to the bottom of the hole; Absolute programming represents the absolute coordinate value of the bottom of the hole;

R_: incremental programming represents the distance from the initial point plane to the R point; Absolute programming represents the absolute coordinate value of an R point;

P_: pause time at point R at the bottom of the hole or time at point R when rewinding;

Q_: the depth of the cut feed for each cut;

F_: cutting feed speed;

K_: Number of repetitions.

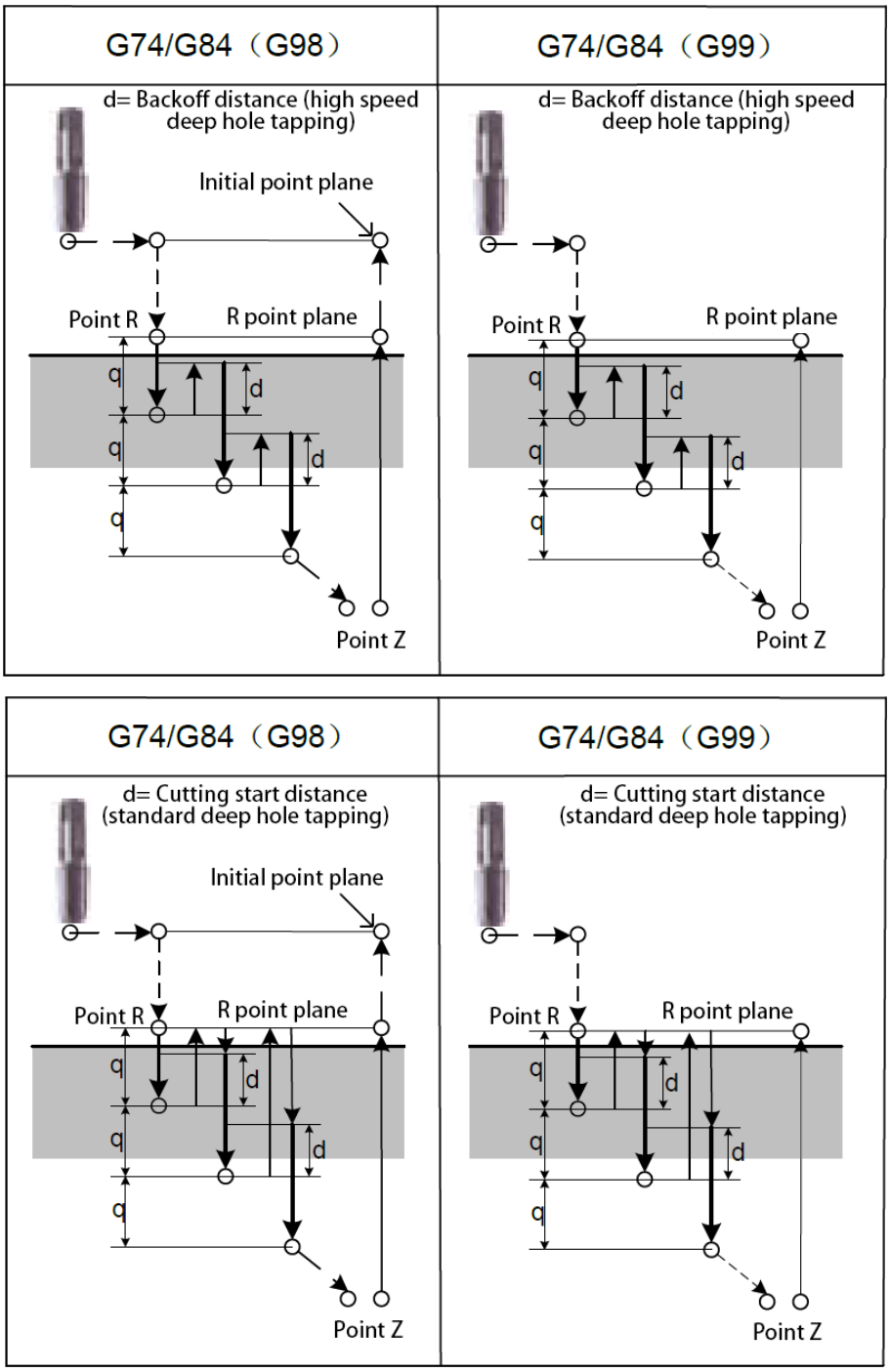


Fig. 3-4-21-1

There are two ways to have a deep hole rigid tapping cycle: a high-speed deep hole tapping cycle and a standard deep hole tapping cycle, set by the bit parameter NO: 58#1 .

When the bit parameter NO: 58#1=1, it is a high-speed deep hole tapping cycle: after positioning along the X and Y axes, perform a fast move to point R, and use the knife depth Q from point R (the depth of each cut feed) is performed, and then the tool return distance d

(set by the data parameter P339), set by the bit parameter NO: 58#6, when the rigid tapping retract is set, whether the magnification is valid and the bit parameter NO: 58#7 Specify the reverse speed magnification, and set whether the same time constant is used when rigid tapping into the knife and retract is set by the bit parameter NO: 58#5. When the Z point is reached, the spindle stops and then rotates backwards in the opposite direction.

When the bit parameter NO: 58#1=0, it is a standard deep hole tapping cycle: after positioning along the X and Y axes, perform a quick move to point R, and use the knife depth Q from point R (depth of each cut feed) Perform the cut, and then execute back to the R point, set by the bit parameter NO: 58#6, whether the magnification is valid when the rigid tapping retract is set, and the reverse rate is specified by the bit parameter NO: 58#7, from R The point to the point from the end point of the last cut d (set by the data parameter 339) re-perform the cut, by the position parameter NO: 58#5 to set whether the same time constant is used when rigid tapping the knife is entered and retracted. When the Z point is reached, the spindle stops and then rotates backwards in the opposite direction.

Limit:

F: An alarm if the specified F value exceeds the upper limit of the cutting feed speed.

S: If the speed is higher than the maximum speed of the specified grade, the alarm is issued. The speed grade is set by the data parameter P209.

Cancel: You cannot specify Group 01 G codes (G00 to G03) and G84 (or G74) in the same program segment, otherwise G84 (or G74) will be canceled.

Tool Offset: The tool radius offset is ignored during fixed cyclic positioning.

Program restart: Restarting the program during rigid tapping is invalid.

3.4.22 Fixed loop cancels G80

Instruction format: G80

Function: Cancel the pinning cycle.

Illustrate:

Cancel all fixed loops and perform normal operations. R points and Z points are also canceled. Other drilling, boring data is also canceled.

Example:

M3 S100； (The spindle starts to rotate).

G90 G99 G88 X300 Y-250 Z-150 R-120 F120； (Position, boring 1 hole, then return to point R).

Y-550； (Position, boring 2 holes, and return to point R).

Y-750； (Position, boring 3 holes, and return to point R).

X1000； (Position, boring 4 holes, then return to point R).

Y-550； (Position, boring 5 holes, then return to point R).

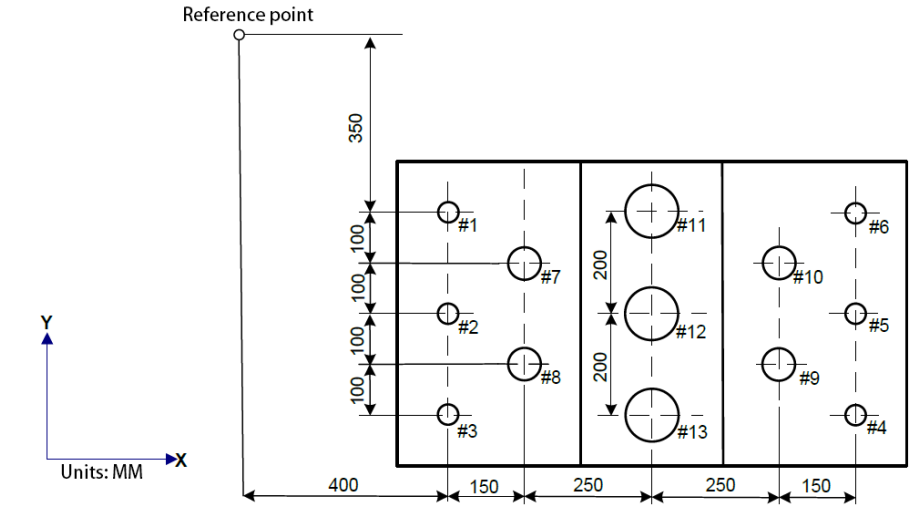
G98 Y-750； (Position, boring 6 holes, and then return to the initial position plane).

G80；

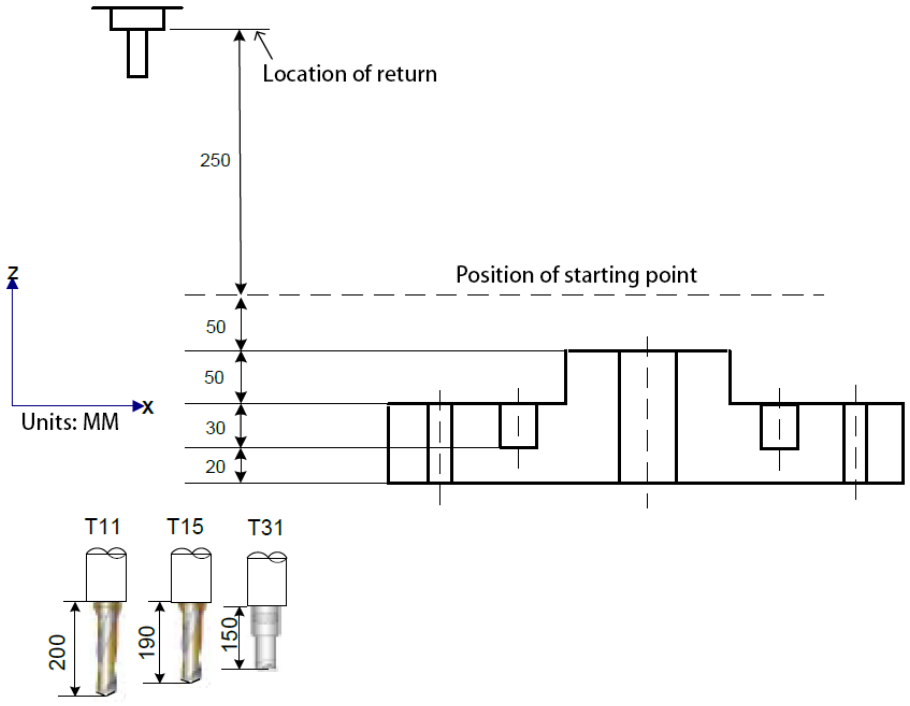
G28 G91 X0 Y0 Z0； (Return to reference point to unpin the loop).

M5； (Spindle stops rotating).

Example: The following uses tool length compensation, which is comprehensive to illustrate the use of fixed cycles.



1 ~ # 6... Drill Φ 10 hole
7 ~ # 10... Drill Φ 20 hole
11 ~ # 13... bore Φ 95 hole



The value of offset 11 is 200, the value of offset 15 is 190, and the value of offset number 31 is 150 as the offset is set separately.

The procedure is as follows:

N001 G92 X0 Y0 Z0 ； (The coordinate system is set at the reference point.)

N002 G90 G00 Z250 T11 M6； (tool change.)

N003 G43 Z0 H11 ； (Plane tool length compensation is performed at the initial point.)

N004 S300 M3 ； (Spindle start.)

N005 G99 G81 X400 Y-350 Z-153 R-97 F120 ； (Position and machine the #1 hole.)

N006 Y-550 ； (Position and machine the #2 hole to return to the R point plane.)

N007 G98 Y-750 ； (Machining the #3 hole after positioning returns to the initial point plane.)

N008 G99 X1200 ； (After positioning and machining the #4 hole, return to the R point plane.)

N009 Y-550 ； (Position and machine the #5 hole to return to the R point plane.)

N010 G98 Y-350 ； (Position and machine the #6 hole to return to the initial point plane.)

N011 G00 X0 Y0 M5 ； (Return to the reference point, the spindle stops.)

N012 G49 Z250 T15 M6 ； (Cancel the tool length compensation, change the tool.)

N013 G43 Z0 H15 ； (Initial point plane, tool length compensation.)

N014 S200 M3 ； (Spindle start.)

N015 G99 G82 X550 Y-450 Z-130 R-97 P30 F70 ； (After positioning the #7 hole, return to the R point plane.)

N016 G98 Y-650 ； (After positioning the #8 hole, return to the initial point plane.)

N017 G99 X1050 ； (After positioning the #9 hole, return to the R point plane.)

N018 G98 Y-450 ； (Position and machine the #10 hole to return to the initial point plane.)

N019 G00 X0 Y0 M5 ； (Return to the reference point, the spindle stops.)

N020 G49 Z250 T31 M6 ； (Cancel the tool length compensation, change the tool.)

N021 G43 Z0 H31 ； (Initial point planar tool length compensation.)

N022 S100 M3 ； (Spindle start.)

N023 G85 G99 X800 Y-350 Z-153 R47 F50 ； (Position and machine the #11 hole to return to the R point plane.)

N024 G91 Y-200 ；

Y-200 ； (After positioning and machining the #12, #13 holes, return to the R point plane.)

N025 G00 G90 X0 Y0 M5 ； (Return to the reference point, the spindle stops.)

N026 G49 Z0 ； (Eliminate tool length compensation.)

N027 M30 ； (Program stops.)

3.5 Tool compensation G code

3.5.1 Tool length compensation G43, G44, G49

Function:

G43 Specifies forward compensation for tool length.

G44 Specifies negative compensation for tool length.

G49 Eliminate tool length compensation.

Instruction format:

The system supports two tool length bias methods of A/B, with the position parameter N0 003#4 Set the tool length offset mode.

Way A:

G43 ⎫
⎬ Z_ H_
G44 ⎭

Method B:

G17 G43 Z_H；

G17 G44 Z_H；

G18 G43 Y_H；

G18 G44 Y_H；

G19 G43 X_H；

G19 G44 X_H；

Tool length offset mode canceled: G49 or H0.

Illustrate:

The purpose of the above directive is to move the specified axis command end point position by another offset. The difference between the hypothetical tool length value (usually set to the first knife) and the tool length value used during actual machining is preset in the bias memory, so there is no need to change the program, only need to change the tool length compensation value can be used to machine parts with different lengths.

G43, G44 specifies different offset directions, and the offset number is specified with an H code.

1. Offset direction

G43: Forward offset (the most common type of offset).

G44: Negative offset

Whether it is an absolute value instruction or an incremental value instruction, at G43, the coordinate value of the end point of the specified axis movement instruction in the program plus the offset specified with the H code (set in the offset memory); At G44, subtract the offset specified by the H code and take the coordinate value of its calculation as the end coordinate value.

G43, G44 is a modal G code that is valid until it encounters other G codes in the same group.

2. Specify the amount of bias

The length offset sign is specified by the H code, and the offset amount corresponding to the offset sign is added or subtracted from the Z-axis movement instruction value in the program to form a new Z-axis movement instruction . You can specify H00 ~ H256 with a bias sign as needed.

The range of offset amounts is set as follows:

Table 3-5-1-1

	Millimeter input
Compensation amount H	-999.9999 mm～+999.9999mm

The bias number 00, that is, the offset amount corresponding to H00 is 0. The offset amount corresponding to H00 cannot be set in the system.

Note: When the offset amount changes due to the change of the offset sign, the old offset amount is simply replaced directly with the new offset amount, not the new offset amount is added to the old compensation amount.

For example:

H01..... ..... The offset amount is 20

H02..... ..... Offset amount 30

G90 G43 Z100 H01 ； ..... Z walked to 120

G90 G43 Z100 H02 ； ..... Z walked to 130

3. The valid order of the bias signs

The current bias number takes effect as soon as the length offset modality is established, and when the offset sign changes, the new offset value immediately replaces the old offset

value.

For example:

O×××××；

H01；

G43 Z10； (1) The offset number H01 takes effect

G44 Z20 H02； (2) The offset number H02 takes effect

H03； (3) The offset number H03 takes effect

G49； (4) The bias is canceled and H00 takes effect

M30；

4. Cancel the tool length compensation

Cancel tool compensation with G49 or H00. Immediately after Directive G49 or H00, the system cancels the tool length compensation.

Note:

1. After the tool length bias B mode is performed along two or more axes, the bias of all axes is cancelled with G49, and the bias of the axis perpendicular to the specified plane is only canceled with H00.

2, the establishment and cancellation of the length compensation is recommended to add the Z axis movement command, otherwise the current point will be established and canceled the length compensation, therefore, when using G49, please make sure that the Z axis is at a safe height to prevent the knife from crashing or damaging the workpiece.

5. G53, G28 or G30 instructions in the tool length bias mode

When the G53, G28, or G30 command is specified in the tool length offset mode, the offset vector of the tool length offset axis is canceled when it moves to the specified position (where G53 is canceled when it moves to the command position, G28, G30 is canceled when moving to the reference point), but the modal code display does not switch to G49, and axes other than the tool length offset axis are not canceled. When G53 and G49 are in the same segment, all axes are removed from the length offset when they move to the command position; When G28 or G30 is in the same segment as G49, all axes are removed from the length offset when they move to the reference point. The canceled tool-length offset vector is restored at the next program segment being cached.

6. Specific examples of tool length compensation

(A) tool length compensation (machining #1, #2, #3 holes).

(B) H01 = offset -4

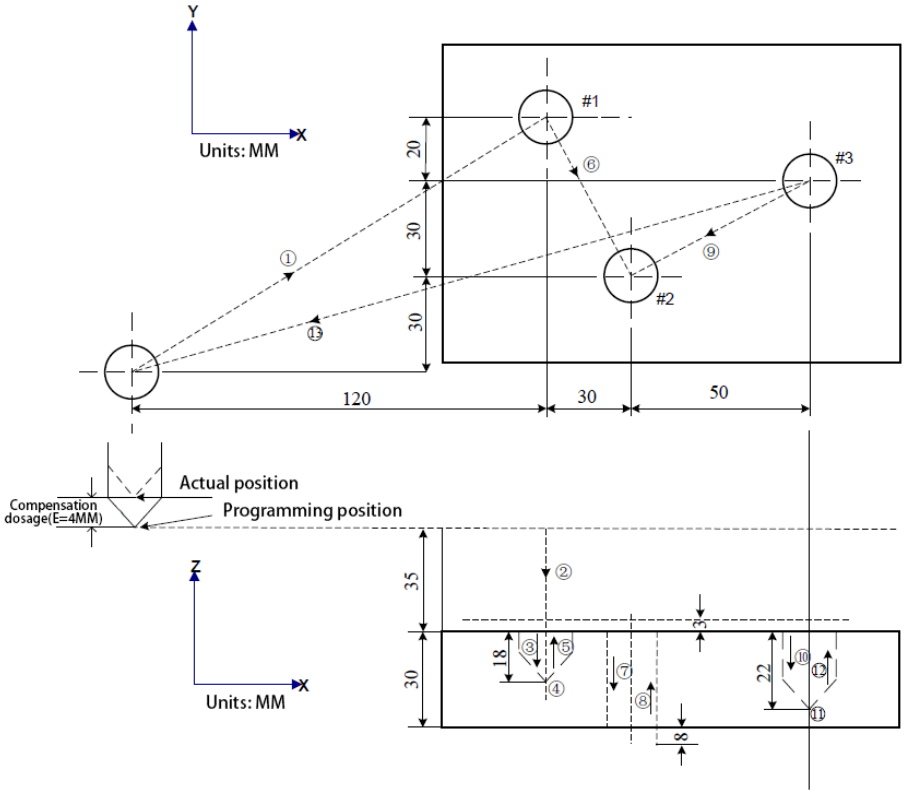


Fig. 3-5-1-1

N1 G91 G00 X120 Y80 ； (1)

N2 G43 Z-32 H01 ； (2)

N3 G01 Z-21 F200 ； (3)

N4 G04 P2000 ； (4)

N5 G00 Z21 ； (5)

N6 X30 Y-50 ； (6)

N7 G01 Z-41 F200 ； (7)

N8 G00 Z41 ； (8)

N9 X50 Y30 ； (9)

N10 G01 Z-25 F100 ； (10)

N11 G04 P2000 ； (11)

N12 G00 Z57 H00 ； (12)

N13 X-200 Y-60 ； (13)

N14 M30 ；

3.5.2 Tool radius compensation G40/G41/G42

Instruction format:

G41 D_ X_Y_

G42 D_ X_Y_

G40 X_Y_

Function:

G41 Specifies the left compensation for the direction of tool movement.

G42 Specifies the right side compensation for the direction of tool movement.

G40 eliminates tool radius compensation.

Illustrate:

1. Tool radius compensation function

As shown in the following figure, use a tool with a radius of R to cut workpiece A, the center path of the tool is B in the figure, and the distance of path B is R . The tool is offset from the workpiece

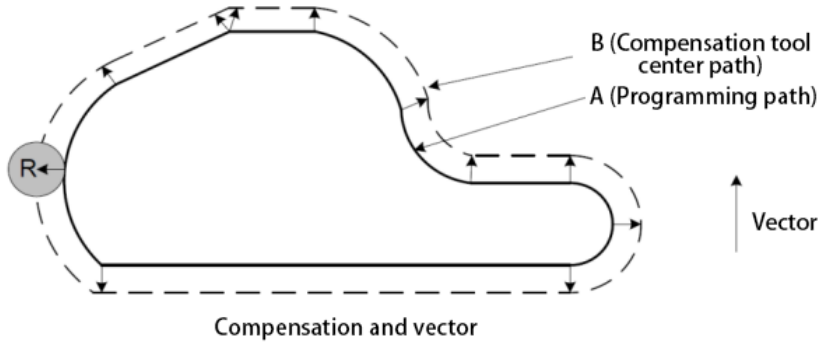


Fig. 3-5-2-1

The programmer uses the tool radius compensation mode to prepare the machining program, during the machining, the tool diameter is determined and entered into the CNC memory, and the tool path becomes the compensation path B.

2. Compensation amount (D value).

The D code specifies a radius bias sign, and the offset corresponding to the offset sign is added to or subtracted from the move instruction value in the program to form a new move instruction. You can specify D00 to D256 with a bias sign as needed.

With the LCD/MDI panel, the offset amount corresponding to the offset number can be set in the bias memory in advance.

The compensation amount is set within the following ranges:

Table 3-5-2-1

	Millimeter input
Compensation amount D	-999.9999mm～+999.9999mm

Note: The compensation amount of D00 is calculated by default to **0, which cannot be set or modified by the user.**

Changes to the compensation plane must be made after the compensation mode is canceled. No cancellation of the compensation mode and change the compensation plane system will be alarmed.

3. Plane selection and vector

Compensation calculations are performed in a plane selected by G17, G18, G19. This plane is called the compensation plane. For example, when an XY plane is selected, compensatory and vector calculations are performed in the program with (X,Y). Coordinate values for axes that are not in the compensation plane are not affected by compensation.

When performing simultaneous three-axis control, only the toolpaths projected in the compensation plane are compensated.

Changes to the compensation plane must be made after the compensation mode is canceled.

Table 3-5-2-2

Code G	Compensation plane
G17	X-Y plane
G18	Z-X plane
G19	Y-Z plane

4. G40**,** G41 **and G42**

Cancel and execute the vector with G40, G41, G42 command tool radius. They are combined with the G00, G01, G02, G03 instructions to define a pattern that determines the value and direction of the compensation vector.

Table 3-5-2-3

code	function
G40	Tool radius compensation is canceled
G41	Tool radius left compensation
G42	Right compensation of tool radius

5. G53, G28, G30 instructions in the tool radius compensation mode

When the G53, G28, or G30 command is specified in the tool radius compensation mode, the offset vector of the tool radius bias axis is canceled when it moves to the specified position (where G53 is canceled when it moves to the command position, G28, G30 is canceled when moving to the reference point), but the modal code display does not switch to G40, and axes other than the tool radius offset axis are not canceled. When G53 and G40 are in the same section, all axes cancel radius compensation when they move to the command position; When G28 or G30 is in the same segment as G40, all axes are removed from the length offset when they move to the reference point. The canceled tool radius compensation vector is restored on the next program segment that is cached.

Note: In compensation mode, you can choose whether the **compensation will be temporarily canceled when the G28 or G30** instruction is moved to the midpoint by the bit parameter **NO:19#4**

Eliminate tool radius compensation (G40).

In the G00, G01 state, use the following instructions, G40 X_ Y_;.

The old vector from the start point moves in a straight line toward the end point. In the G00 mode, each axial end point is moved rapidly. Use this command to take the system from the tool compensation state to the undo tool compensation state.

If only the G40; When there is no command to X_ Y_, the knife does not move.

Tool radius compensation left (G41).

1) G00, G01

G41 X_ Y_ D_ ； The instruction forms a new vector perpendicular to the direction of (X,Y) at the end of the program segment, and the cutter moves from the tip of the old vector at the starting point to the tip of the new vector.

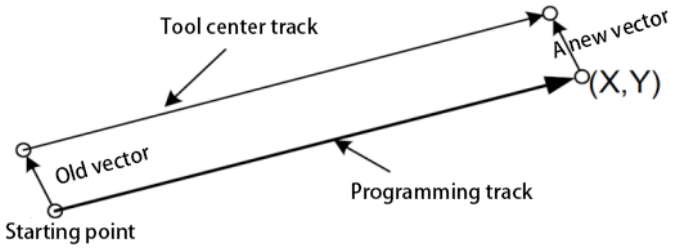


Fig. 3-5-2-2

When the old vector is zero, use this command to make the tool enter the tool radius compensation state from the unbiased state. In this case, the offset value is specified by the D code.

2) At G02, G03

G41.....;

.....

.....

G02/G03 X_ Y_ R_ ;

According to the above program, a new vector can be made, which is located on the line between the center of the arc and the end point, from the direction of the arc forward, pointing to the left (or right), and the tool center moves from the old vector tip of the arc to the new vector tip along the arc. But only if the old vector has been made correctly.

The offset vector is from the start or end point to the center of the arc or from the center.

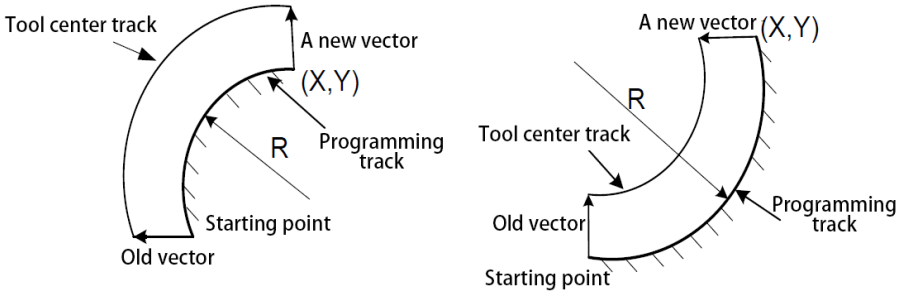


Fig. 3-5-2-3

Tool radius compensation right (G42).

The G42 is the opposite of the G41, with the tool offset on the right side of the workpiece in the direction of the tool's progress. That is to say, the vector direction made with G42 is exactly the opposite direction of the vector made by G41. Except that the vector is in opposite directions, the offset method is exactly the same as G41.

1) G00, G01

G42 X_ Y_ D_ ;

G42 X_ Y_ ;

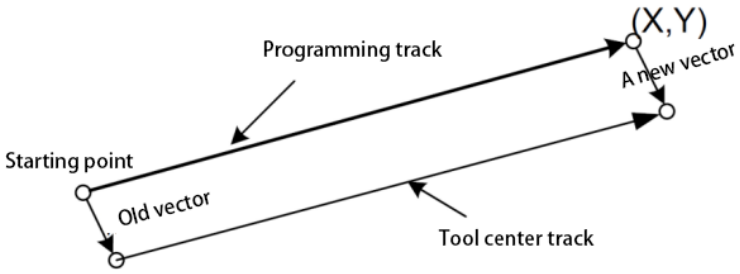


Fig. 3-5-2-4

2) At G02, G03

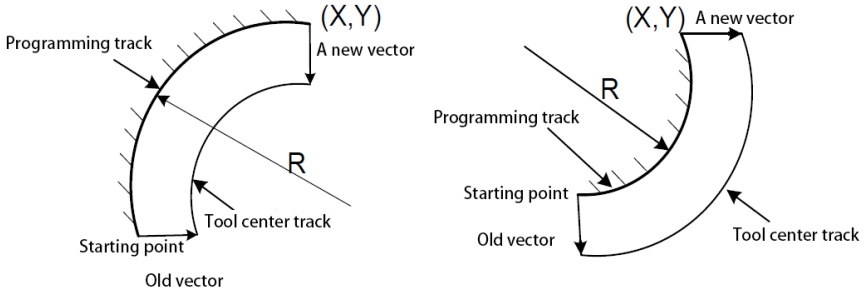


Fig. 3-5-2-5

6. General precautions about offset:

A. The assignment of the offset number

G41, G42, G40 are modal instructions, and the offset number is specified with a D code. This can be specified anywhere before the offset cancellation state changes to the tool radius compensation state. G41, G42 instructions must be followed by the move command, otherwise the alarm.

B. About entering the tool radius compensation state from the unoffset state

The movement command when entering the tool radius compensation state from the unoffset state must be positional (G00) or straight line interpolation (G01), not circular interpolation (G02, G03).

C. The tool radius compensates for the transition between left and right

When the offset direction changes from left to right, or from right to left, it is generally canceled by offset. However, positioning (G00) or straight line interpolation (G01) can be directly converted without being canceled by offset. The tool trajectory at this point is shown in the following figure:

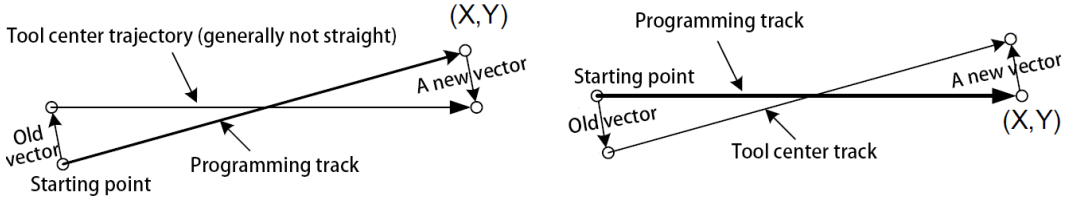


Fig. 3-5-2-6

```
{ G1 G41 D_X_ Y_ ;          { G42 D_X_ Y_ ;
  .....                       .....
  G1 G42 D_X_ Y_ ;            G41 D_X_ Y_ ;
```

D. The change in the offset

The change of the offset is generally done in the offset cancellation state, during the tool change, but for positioning (G00) and straight line interpolation can also be done in the offset state, as shown in the following figure.

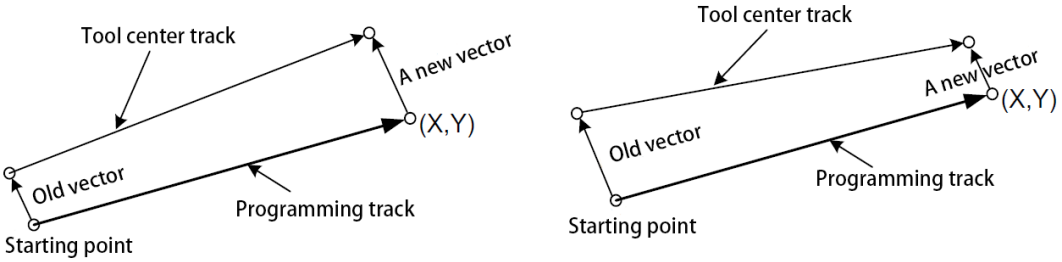


Fig. 3-5-2-7 (Change of Offset).

E. The positive and negative of the offset and the tool center trajectory

If the offset is set to a negative value, the workpiece is equivalent to the situation when G41 and G42 are all transformed on the program sheet , so that the cutting along the outside of the workpiece becomes cutting along the inside of the workpiece, and the original machining along the inside of the workpiece It becomes processed along the outside.

When programming in general as shown in the following figure, the offset is assumed to be positive:

When the tool trajectory is programmed as shown in (A) figure, if the offset is set to negative, then the tool trajectory is shown in Figure (B); Similarly, when the tool trajectory is programmed as shown in Figure (B), if the offset is set to negative, the tool trajectory is shown in Figure (A).

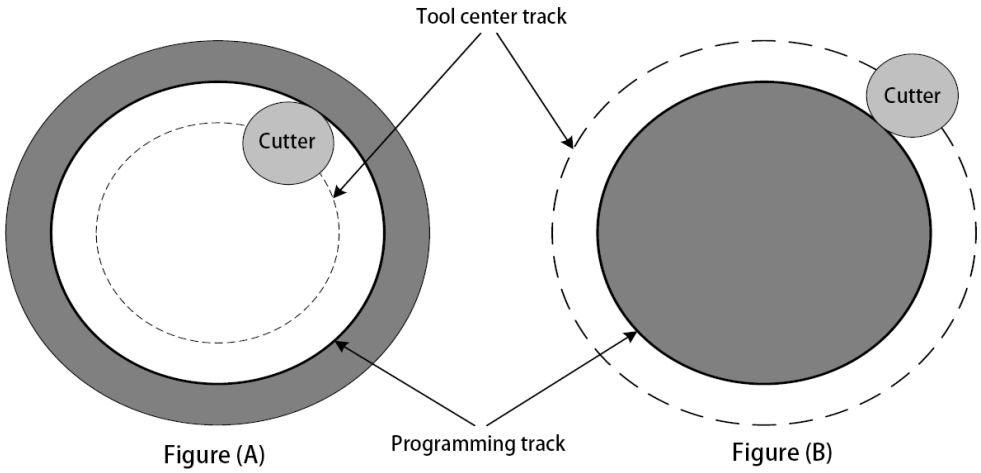


Fig. 3-5-2-8

Shapes with sharp corners are common (Shapes with Sharp Corner Arc Interpolation). However, when the offset is set to a negative value, the inner circle of the part cannot be machined. When cutting the inner sharp corner of a corner, an arc of the appropriate radius is inserted there, and the sleek transition can be made.

Left compensation or right compensation depends on whether the compensation direction is on the left or right side of the direction of movement of the tool relative to the workpiece (which is considered to be immobile). The G41 or G42 puts the system into compensation mode and G40 causes the system to cancel compensation mode.

An example of a compensatory procedure is as follows:

Program segment (1) is called start, and the G41 instruction in this section changes the compensation cancellation mode to the compensation mode. At the end of this section, the center of the tool compensates with the tool radius perpendicular to the next program path (from P1 to P2). The amount of tool compensation is specified with D07, that is, the compensation number is set to 7 and G41 indicates the left compensation of the toolpath.

After the compensation begins, when the workpiece shape is woven into such as P1 → P2... P9 → P10 → P11, toolpath compensation is performed automatically.

Example of a procedure for toolpath compensation

G92 X0 Y0 Z0 ；

(1) N1 G90 G17 G0 G41 D7 X250 Y550 ； (The amount of compensation must be pre-set with the compensation number).

(2) N2 G1 Y900 F150 ；

(3) N3 X450 ；

(4) N4 G3 X500 Y1150 R650 ；

(5) N5 G2 X900 R-250 ；

(6) N6 G3 X950 Y900 R650 ;

(7) N7 G1 X1150 ;

(8) N8 Y550 ;

(9) N9 X700 Y650 ;

(10) N10 X250 Y550 ;

(11) N11 G0 G40 X0 Y0 ;

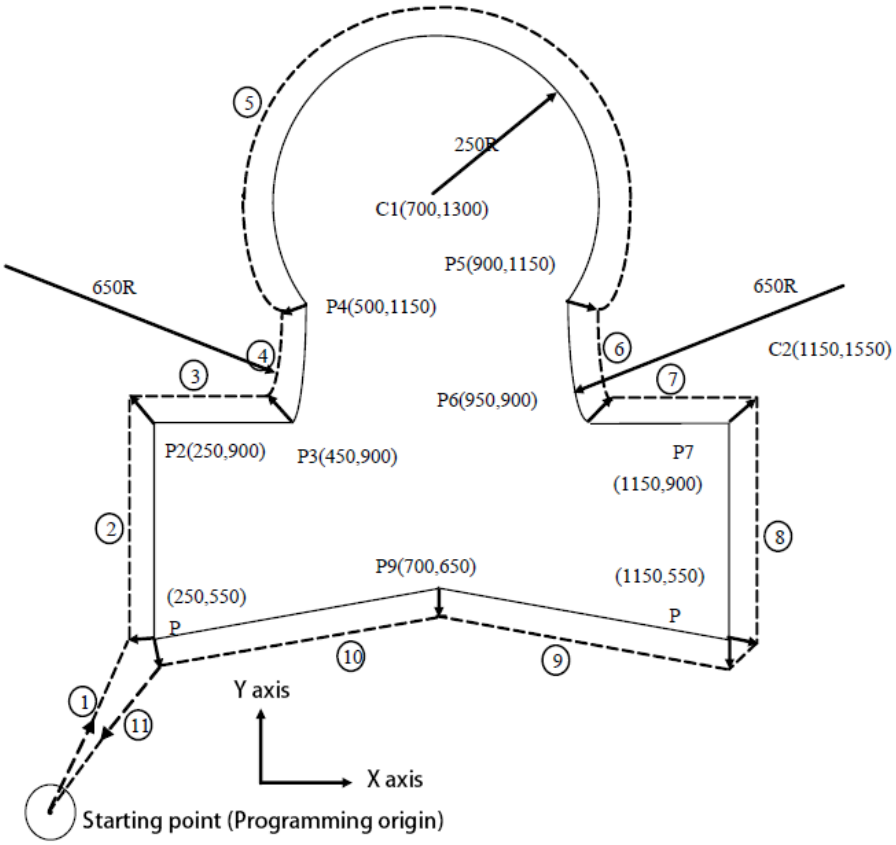


Fig. 3-5-2-9

3.5.3 A detailed description of the tool radius compensation

Conception:

Inner and outer side: When the angle of the tool trajectory established by the two program instructions exceeds 180°, the trajectory is called the inner side, and when the angle is between 0° and 180°, it is called the outside.

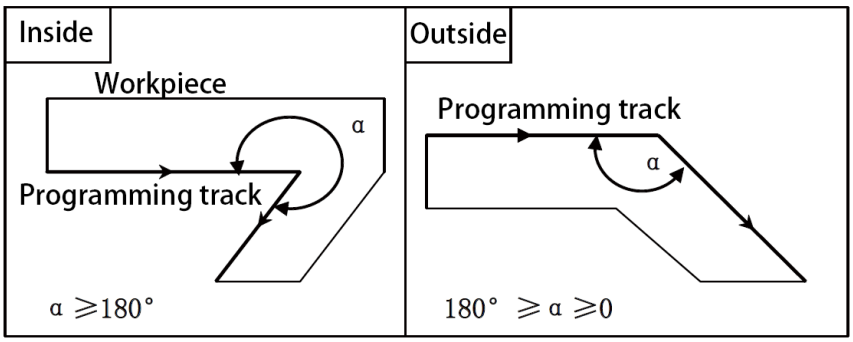


Fig. 3-5-3-1

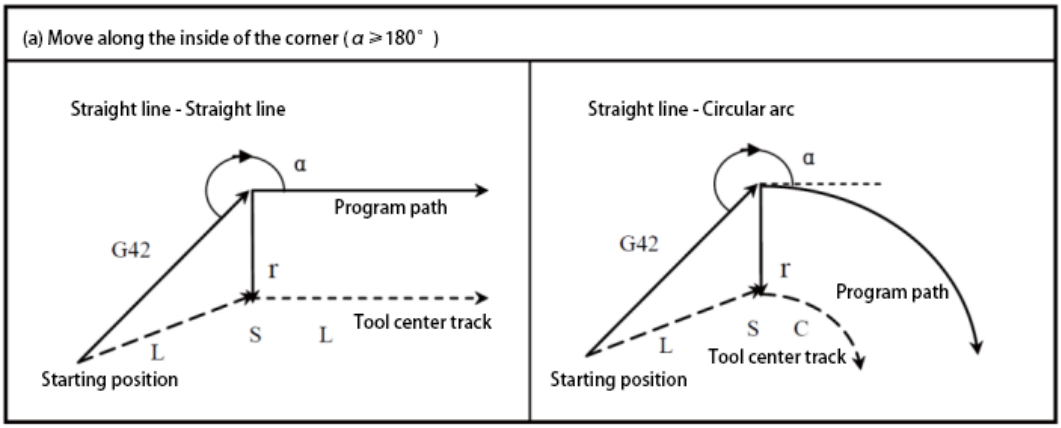
The meaning of the symbol

Use the following symbol in the following illustration

- S means that a single program segment is executed 1 time at this position
- SS means that a single program segment is executed 2 times at this location
- SSS means that a single program segment is executed 3 times at this location
- L indicates that the tool moves in a straight line
- C indicates that the tool moves along an arc
- r indicates the tool radius compensation value
- The intersection is where two programming tracks intersect each other after they are offset by r
- O indicates the tool center

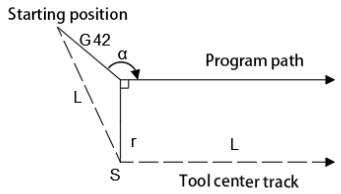
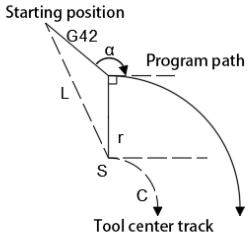
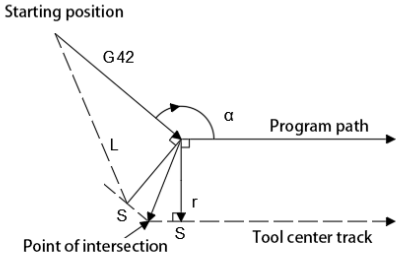
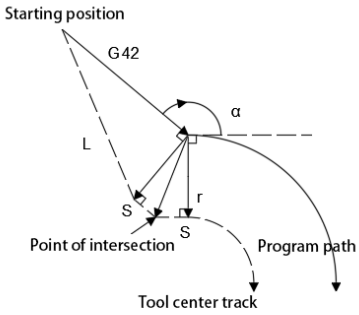
(1) **The tool moves when the knife is started**

When changing from offset cancellation mode to offset mode, the tool movement is shown in the following figure (Knife Starting).



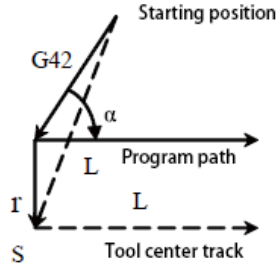
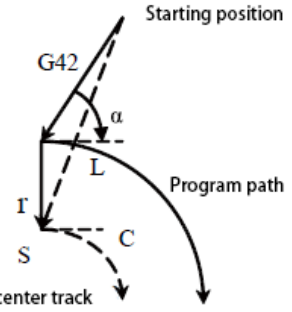
(b) Move along the outer side of the obtuse corner (180° > α ≥90°)

There are two types of tool path actions when compensation starts or cancellations, type A and type B, selected by the bit parameter NO: 19#3:

A	Straight line - Straight line  	Straight line - Circular arc 
B	Straight line - Straight line  Note: The intersection point is the location where the compensation paths of two consecutive program segments intersect.	Straight line - Straight line 

(c) Move along the outer side of the acute Angle (α < 90°)

There are two types of tool path actions when compensation starts or cancellations, type A and type B, selected by bit parameter NO: 19#3:

A	Straight line - Straight line  	Straight line - Circular arc 

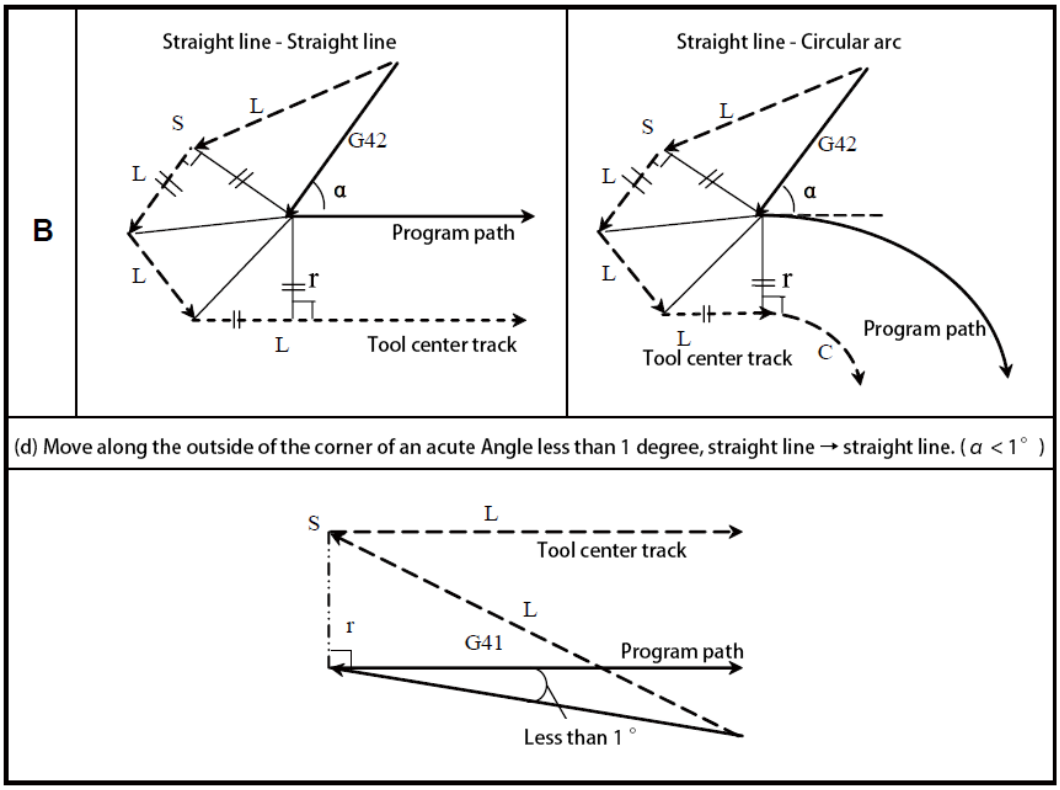


Fig. 3-5-3-2

⑵ The tool movement in offset mode

The compensation plane cannot be changed during the compensation mode execution, otherwise an alarm will be generated and the tool will stop. In the offset mode, the movement of the tool is shown in the following figure:

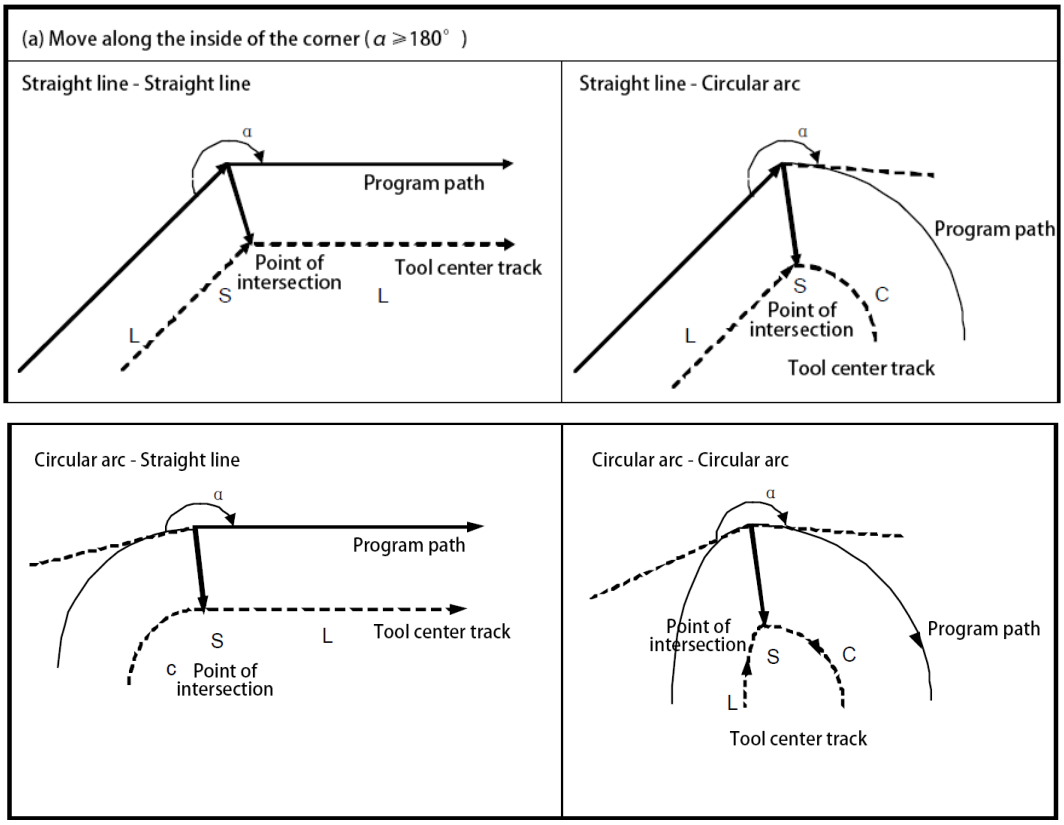


Fig. 3-5-3-3

(3) Special cases

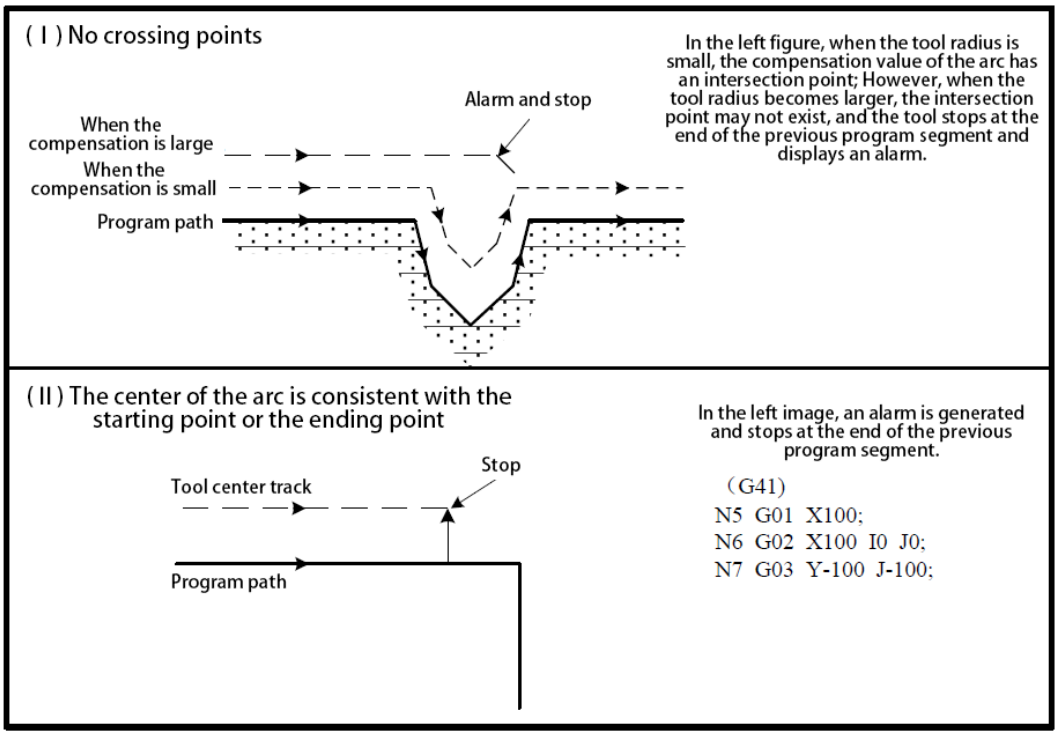


Fig. 3-5-3-4

(4) Tool movement in offset cancellation mode

In compensation mode, when a program segment that satisfies any of the following conditions is executed, the system enters compensatory cancellation mode, and the action of this program segment is called compensatory cancellation.

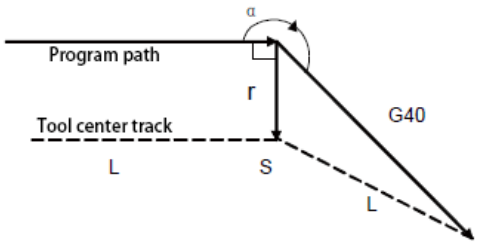
a) Directive G40

b) The tool radius compensation number is 0.

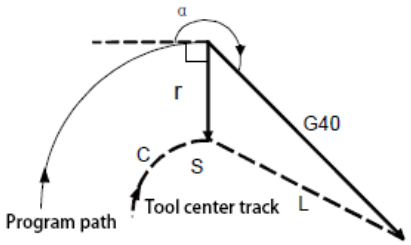
When performing compensatory cancellation, you cannot cancel with arc instructions (G03 and G02). If the command arc generates an alarm and the tool stops.

(a) Move along the inside of the corner ($\alpha \geqslant 180^\circ$)

Straight line - Straight line



Circular arc - Straight line

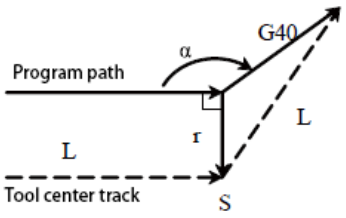


(b) Move along the inside of the corner ($90^\circ \leqslant \alpha < 180^\circ$)

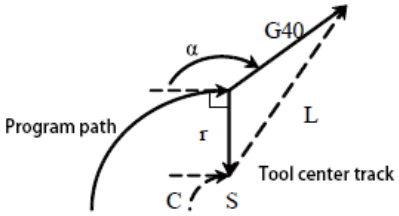
There are two types of tool path actions when compensation starts or cancellations, type A and type B, which are selected by the bit parameter NO:19#3

A

Straight line - Straight line

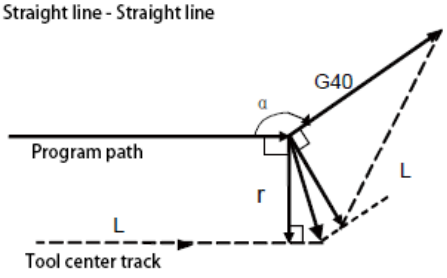


Circular arc - Straight line

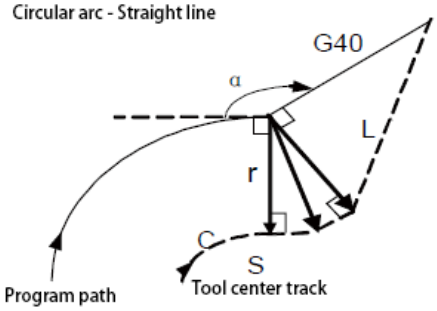


B

Straight line - Straight line



Circular arc - Straight line



(c) Move along the outer side of the corner with an acute Angle ($\alpha < 90^\circ$)

There are two types of tool path actions when compensation starts or cancellations, type A and type B, which are selected by the bit parameter NO: 19#3

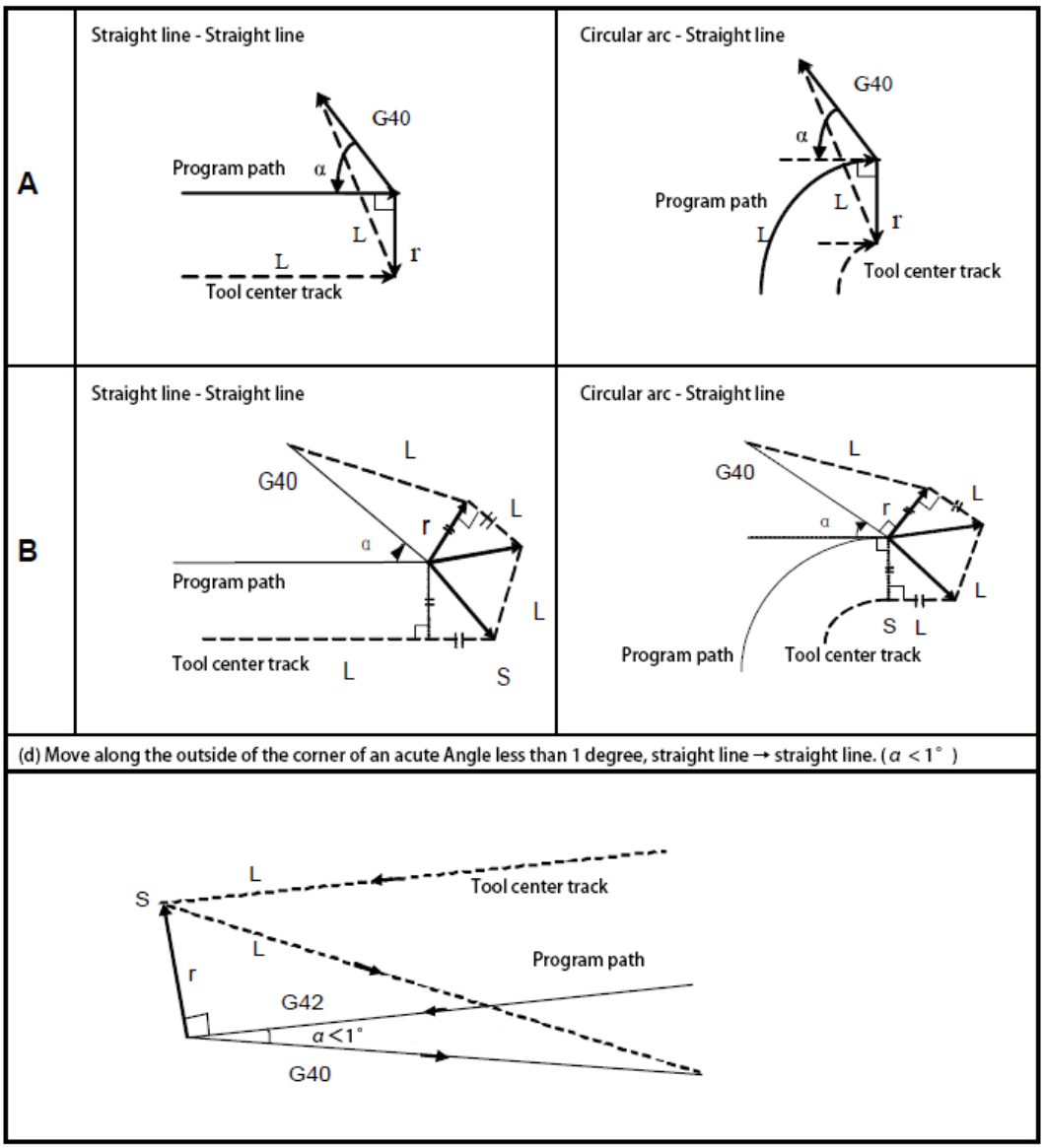


Fig. 3-5-3-5

(5) Change the direction of compensation in compensation mode

The tool radius compensation G code (G41 and G42) determines the direction of compensation, and the symbol of the compensation amount is as follows:

Table 3-5-3-1

Compensating amount symbol G-code	+	-
G41	Left side compensation	Right-hand compensation
G42	Right-hand compensation	Left side compensation

In special cases, the direction of compensation can be changed in the compensation mode. However, it is not possible to change the program section at the start of the start and the program segments that follow it. When compensating for a change in direction, there is no concept of inside and outside. The following compensation amounts are assumed to be positive.

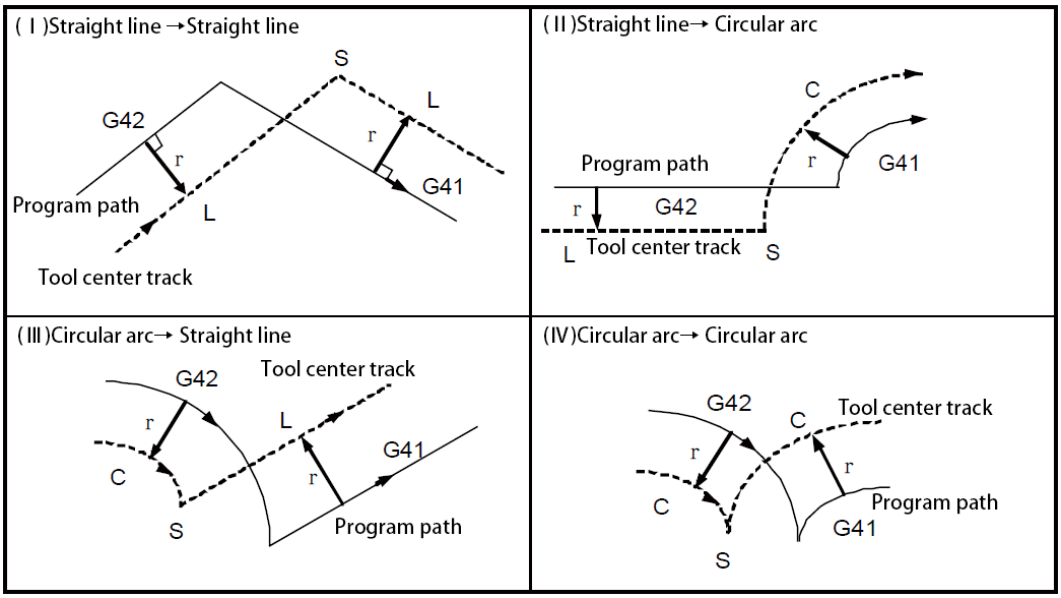


Fig. 3-5-3-6

(v) If compensation is performed normally, but there is no intersection

When using G41 and G42 to change the offset direction from program section A to program section B, if the intersection of the path is not required, the vector perpendicular to program section B is made at the beginning of program section B.

(1) Straight line----- Straight line

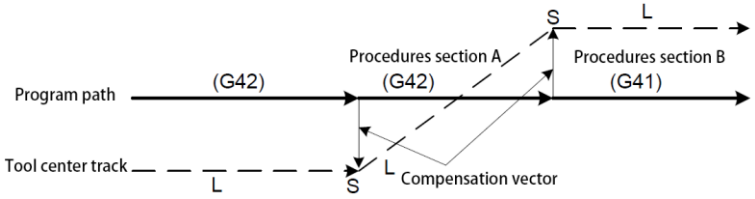


Fig. 3-5-3-7

(2) Straight line----- arc

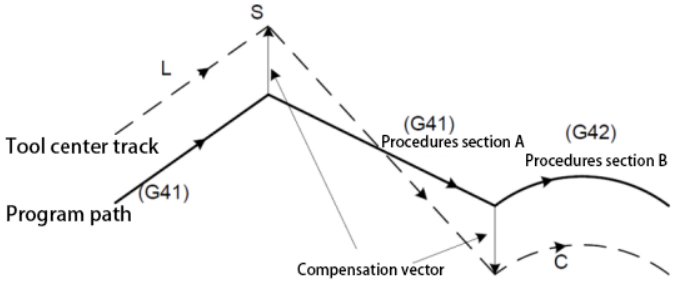


Fig. 3-5-3-8

(3) Arc----- Arc

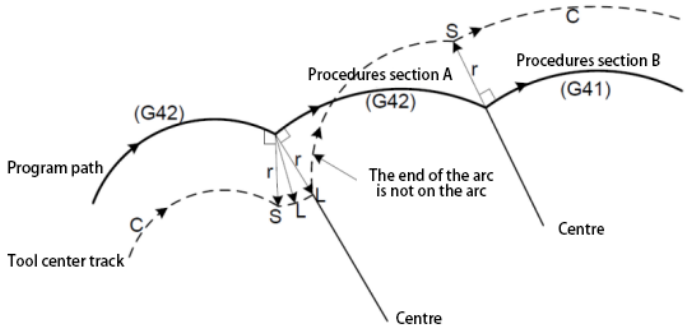


Fig. 3-5-3-9

(vi) When the tool radius compensation makes the tool center path longer than one week, the following conditions usually do not occur. However, when the G41 and G42 change, the following may occur:

Circular arc -- circular arc **(straight line -- circular arc**).

When changing the direction of the tool patch, the system should be alarmed, and the arc command cannot cancel the tool compensation when the tool number is D0!

Straight -- A straight line

The direction of the knife can be changed.

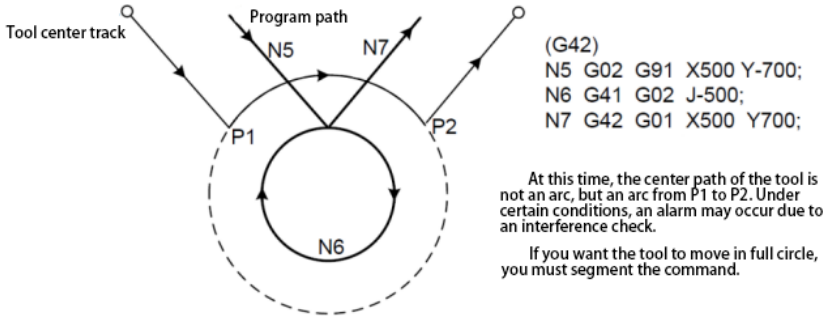


Fig. 3-5-3-10

6. **Temporary compensation cancellation**

In compensation mode, when the G28 or G30 code instructions are specified by the bit parameter NO:19#4, whether the compensation is temporarily canceled at the intermediate point.

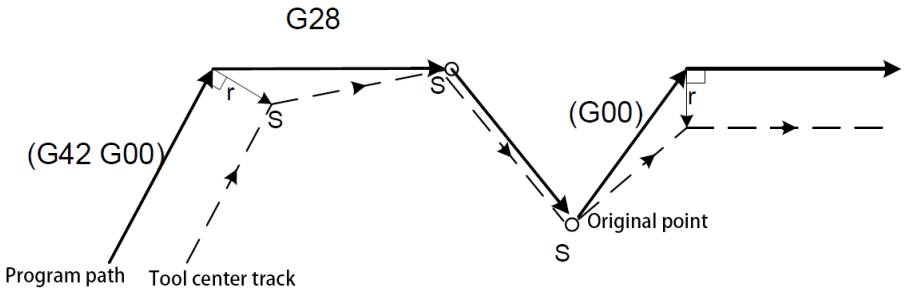


Fig. 3-5-3-1 1

For details of how to do this, please refer to the detailed description of compensation cancellation and compensation start.

a) **G28 automatically returns the reference point**

In compensation mode, if G28 is instructed, compensation will be canceled at the midpoint and the compensation mode will automatically resume after the reference point returns.

b) **G29 automatically returns from the reference origin**

In compensation mode, such as the instruction G29, compensation will be canceled at the intermediate point and compensation mode will be automatically restored in the next program segment.

Immediately after G28 command when:

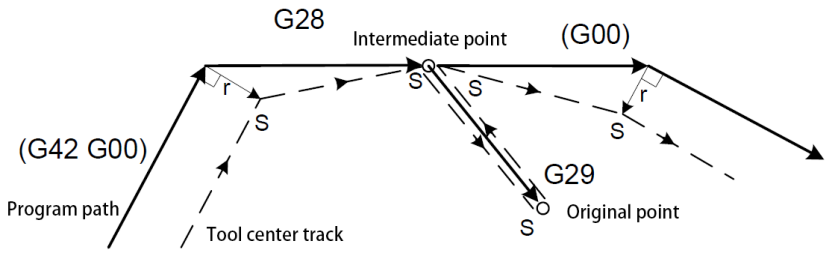


Fig. 3-5-3-12

When not immediately after G28 command:

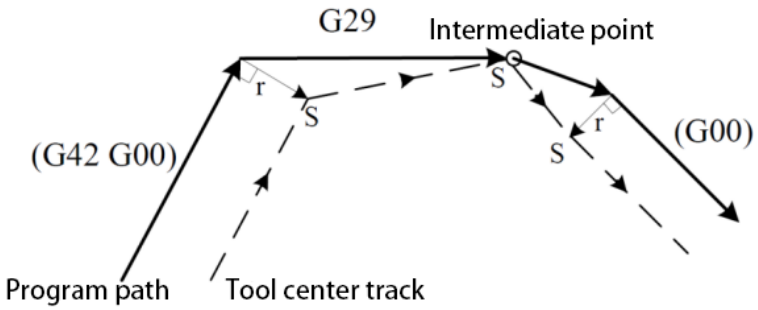


Fig. 3-5-3-13

7. **The tool radius compensation G code** in compensation mode

In the compensation mode, when the tool radius compensation G code (G41, G42) is specified, a vector at a right angle to the front program segment is formed relative to the

direction of movement, regardless of the inside and outside of the machine. However, if such a G code is specified in the arc instruction, the correct arc is not obtained.

When changing the direction of compensation with tool radius compensation G(G41, G42), refer to (5).

Straight line----- Straight line

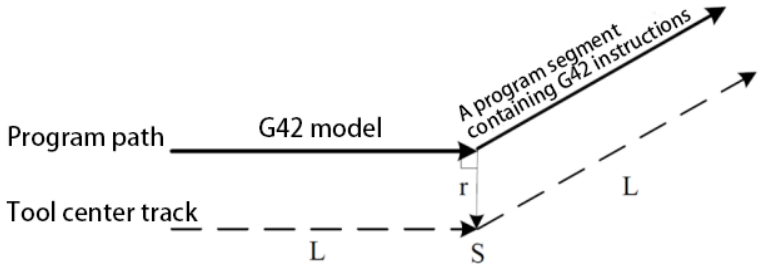


Fig. 3-5-3-14

Arc----- straight line

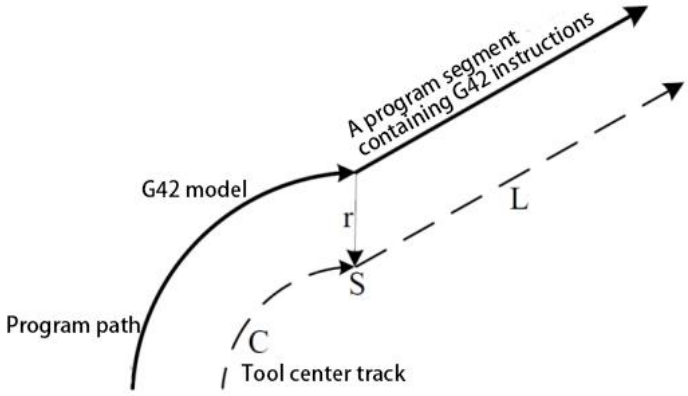


Fig. 3-5-3-15

8. **Temporarily cancel the instruction for the compensation vector**

In compensation mode, if G92 (Absolute Coordinate Programming) is specified, the compensation vector is temporarily canceled, after which the compensation vector is automatically restored. At this point, unlike the compensated cancellation mode, the tool moves directly from the intersection point to the instruction point where the compensation vector cancels it. When the compensation mode is restored, the tool moves to the intersection point again.

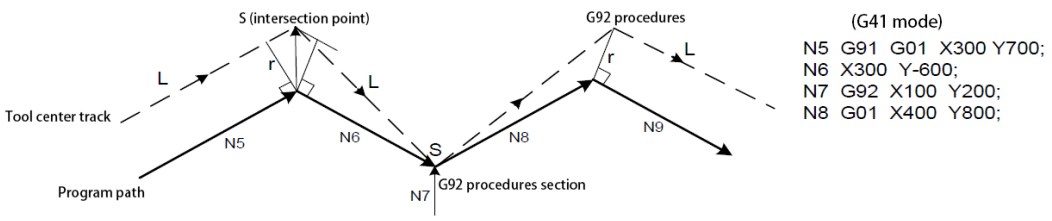


Fig. 3-5-3-16

9. The program segment where the tool does not move

There is no tool movement in the following program segments. In these program sections, the tool radius compensation mode does not move even if it is active.

(1) M05 ； (M code output).

(2) S21 ； (S code output).

(3) G04 X10； (Suspended).

(4) (G17) Z100 ； (No move instruction in the compensation plane).

(5) G90 ； (G code only).

(6) G01 G91 X0 ； (The amount of movement is zero).

a) The instruction at the beginning of the compensation

If the starting program section does not produce a movement, the system will generate a knife opening action in the next moving command.

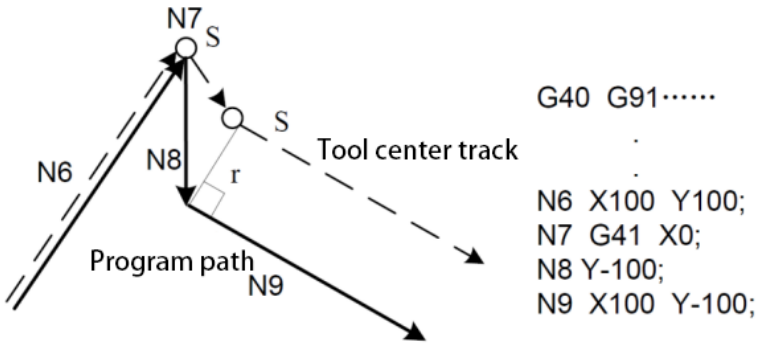


Fig. 3-5-3-17

b) When the compensation mode instruction is made

When only one program segment without tool movement is ordered in compensation mode, the vector and toolcenter path are the same as if the program segment is not commanded. (Refer to Item (3) Compensation Mode) This toolless moving program segment is executed at the single-program segment stop.

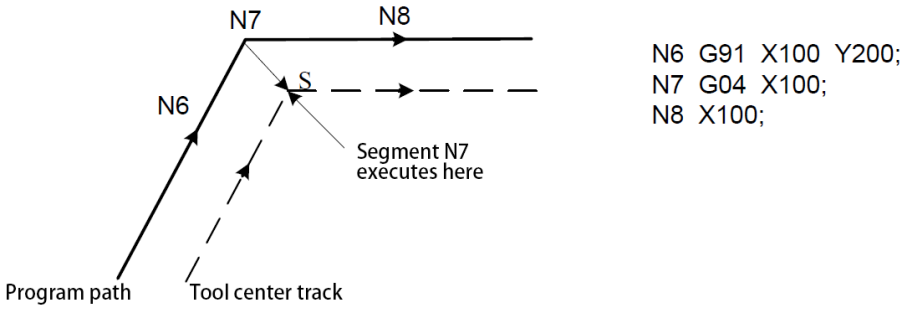
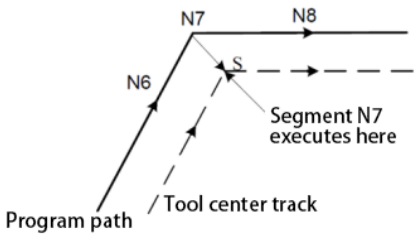


Fig. 3-5-3-18

However, when the amount of program segment movement is zero, even if only one

program segment is specified, the tool movement is the same as two or more program segments without tool movement instructions.



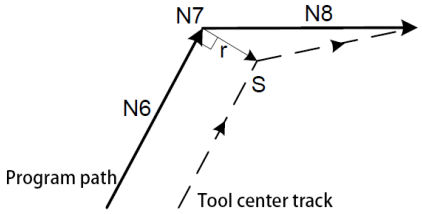

```
N6 G91 X100 Y200;
N7 X0;
N8 X100;
```

Fig. 3-5-3-19

Note: The above program segment is running under the conditions of G1G41, and the trajectory does not match the figure at **G0.**

c) **When ordered along with compensation cancellation**

When there is no tool movement in the program segment commanded together with the compensation cancellation, a vector of length as a compensatory quantity with a direction perpendicular to the direction of movement of the previous program segment is formed, and this vector is canceled on the next move instruction.




```
N6 G91 X100 Y100;
N7 G40;
N8 X100 Y0;
```

Fig. 3-5-3-20

10. **Move around corners**

If more than two vectors are generated at the end of the program segment, the tool moves from one vector straight line to another. This movement is called a corner movement.

If ΔVX ≤ the Δ V limit and Δ VY ≤ the ΔV limit, the later vector is ignored.

If these vectors are inconsistent, a movement along the corners is produced. This move belongs to the later program segment.

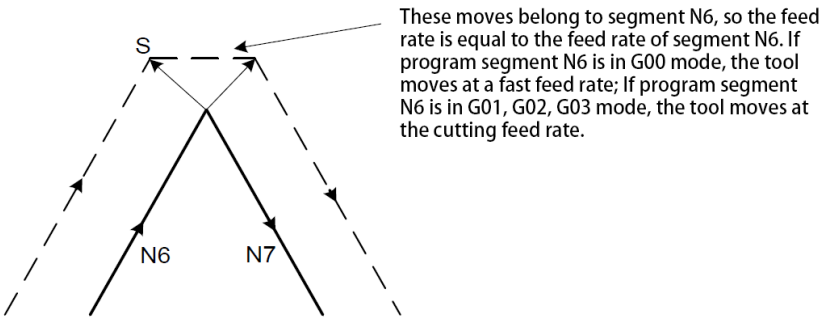
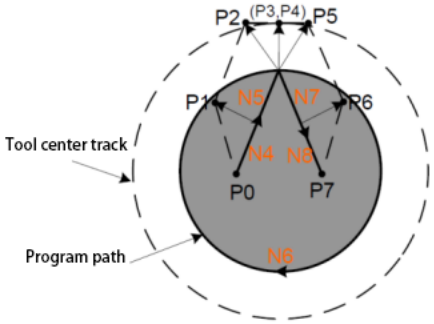


Fig. 3-5-3-21

However, if the path to the next program segment exceeds the semicircle, the above functions are not performed. Here's why:




```
N4 G41 G91 X150 Y200;
N5 X150 Y200;
N6 G02 J-600;
N7 G01 X150 Y-200;
N8 G40 X150 Y-200;
```

Fig. 3-5-3-22

If the vector is not ignored, the toolpath is as follows:

P0 → P1 → P2 → P3 (arc)→ P4 → P5 → P6 → P7

However, if the distance between P2 and P3 is ignored, P3 ignores. The toolpath is as follows:

Arc cutting for P0 → P1 → P2 → P4 → P6 → P7 section N6 is ignored.

11. Interferometric checking

Tool overcutting is called "interference". Interferocity can pre-check for tool over-cutting. If an interference is detected in the syntax check after loading the program, the system will alert you. Whether interference checking is performed when setting radius compensation by bit parameter NO:19#2 .

Basic conditions for interference:

(1) The toolpath direction is different from the program path direction. (The angle between paths is between 90 and 270 degrees.)

(2) In the case of arc machining, in addition to the above conditions, the angle between the start and end points of the toolcenter path and the angle between the start and end points of the program path are very different (more than 180 degrees) .

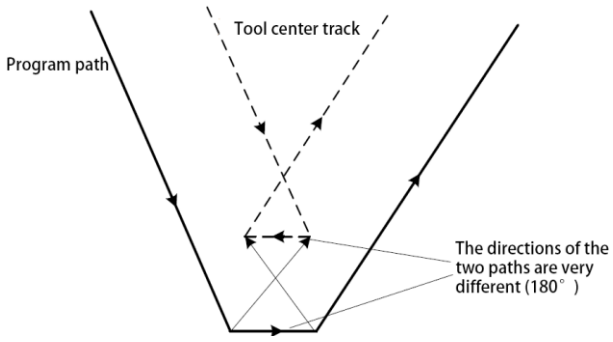


Fig. 3-5-3-2 3

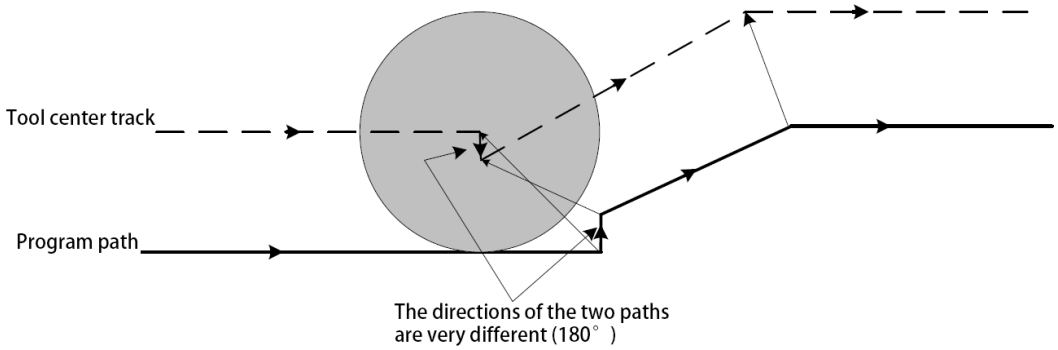


Fig. 3-5-3-24

12. **Manual action**

For manual operation in the tip radius compensation, please refer to the manual operation of the operation section. If tool length compensation is performed in tool radius compensation, the amount of tool radius compensation is considered to be changed.

13. **General considerations for compensation**

a) **The amount of instruction compensation**

The compensation amount is specified by the D code to specify the compensation amount number. Once specified, the D code remains valid until another D code is specified, or the compensation is canceled. In addition to specifying the amount of compensation for tool radius compensation, the D code is also used for the offset value of the tool bias.

b) **The amount of compensation for change**

Usually, when changing tools, the amount of compensation must change in cancellation mode. If the compensation amount is changed in the compensation mode, the new compensation amount is calculated at the end point of the program segment.

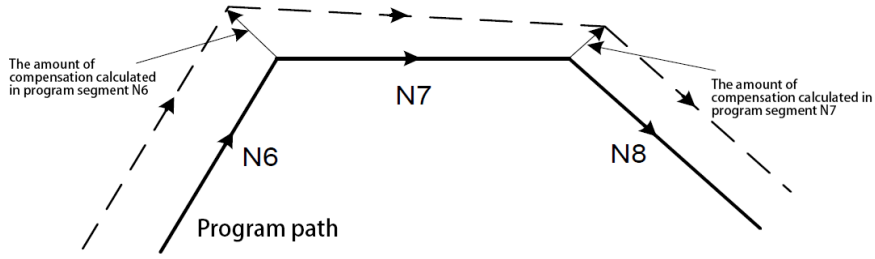


Fig. 3-5-3-25

c) **The plus or minus of the compensation amount and the toolpath center**

If the amount of compensation is negative (-) , the G41 and G42 in the program are exchanged with each other. If the tool center moves along the outside of the workpiece, it will move along the inside and vice versa.

The following example is shown. Generally, the amount of compensation when making a program is (+). When the toolpath is programmed as shown in Figure (a), if the compensation amount is negative (-), the tool center moves as shown in Figure (b) and vice versa. Thus the

same program can be cut into a male or female shape, and the clearance between them can be adjusted by a selective amount of compensation.

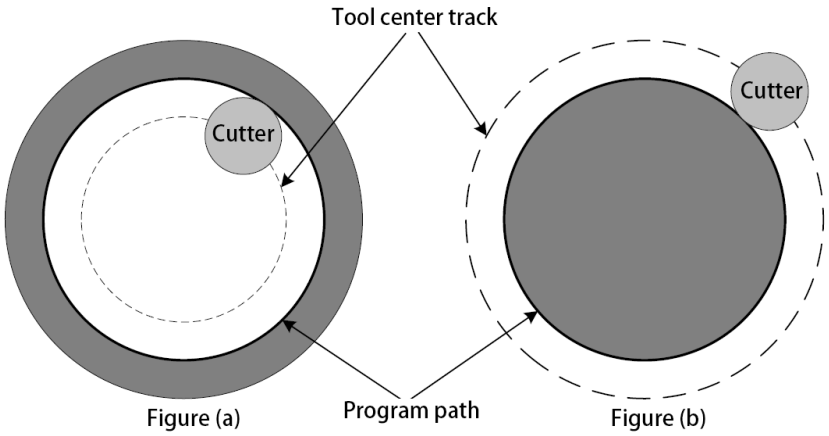


Fig. 3-5-3-26

d) **Overcutting is compensated with the tool radius**

(1) When machining with the inside of an arc smaller than the tool radius

When the corner radius is less than the tool radius, because the inside compensation of the tool will produce over-cutting, an alarm will be generated because the tool will produce over-cutting when stopped in a single-segment execution mode.

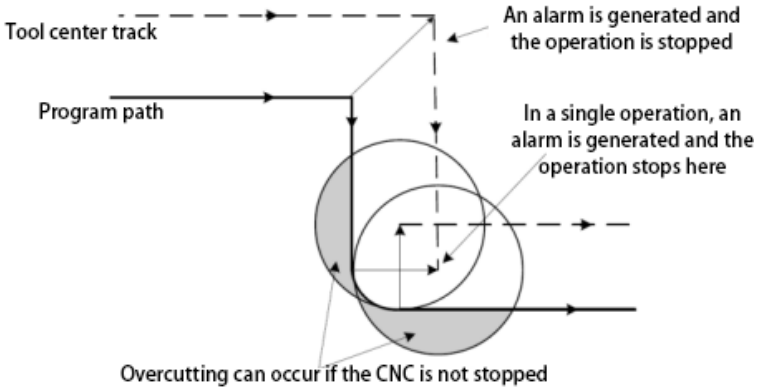


Fig. 3-5-3-27

(2) When machining with a truncated trench that is smaller than the tool radius

When machining with trufts smaller than the tool radius, excessive cutting occurs because the tool radius compensation forces the tool center path to move in the opposite direction of the program path.

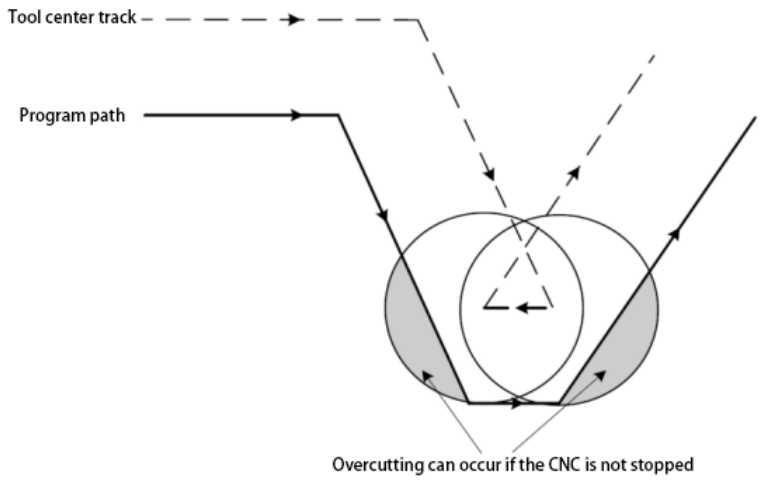


Fig. 3-5-3-28

(3) When the section difference is machined with a smaller radius than the tool radius

If there is a segment difference in the program that is smaller than the tool radius, when the segment difference is machined with the circular arc machining instruction, the normally compensated tool center path becomes the opposite of the program direction. The initial vector is ignored and the cutter moves straight to the second vector. Single-segment execution stops here. If machining is not in single-stage mode, the automatic operation continues. If the segment difference is straight, no alarm will be generated and the correct cutting will be made. However, the uncut part will remain.

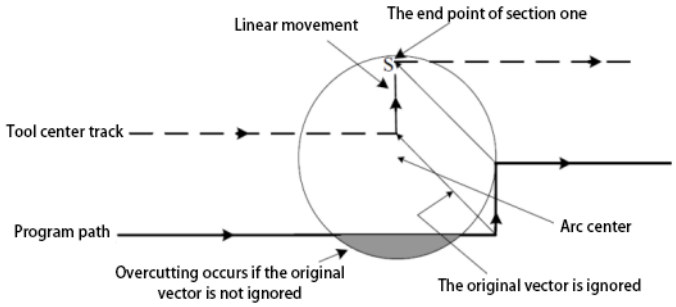


Fig. 3-5-3-29

Tool radius compensation starts and moves in the Z-axis

Generally, at the beginning of machining, after the tool radius compensation is effective, the tool moves along the Z axis a certain distance from the workpiece. In the above case, if you want to divide the movement along the Z axis into fast feed and cutting feed, please refer to the following two procedures:

If program segment N3 (Z axis move instruction).

They are divided as follows:

N1 G91 G00 X500 Y500 H01;

N3 Z-250;

N5 G01 Z-50 F1;

N6 Y100 F2;

N1 G91 G0 X500 Y500 H01;
N3 G01 Z-300 F1;
N6 Y100 F2;

When executing N3, N6 also enters the buffer, using the correct compensation between them, as shown in the right figure.

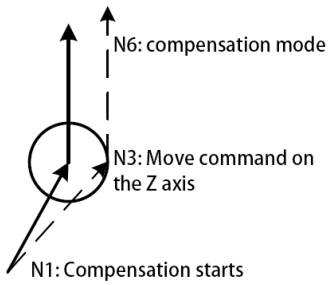


Fig. 3-5-3-30

3.5.4 Corner offset arc interpolation (G39).

Instruction format: G39 or

I_ J_

G39I_ K_

J_ K_

Function: During the tool radius compensation, G39 is instructed in the bias mode, and the corner bias arc compensation can be instructed, and the radius of the corner compensation is equal to the compensation value. Set by the bit parameter NO:19#7 in the radius compensation, whether the corner arc is valid.

Illustrate:

1. When specifying the G39 command, corner arc interpolation whose radius is equal to the compensation value can be performed.

2. The G41 or G42 in front of the instruction determines whether the arc is clockwise or counterclockwise, and G39 is a non-modal G code.

3. When programming with G39 instructions (without I, J, and K), an arc is formed at the corner, so the vector at the end of the arc is perpendicular to the beginning of the next program segment. As shown in the following figure:

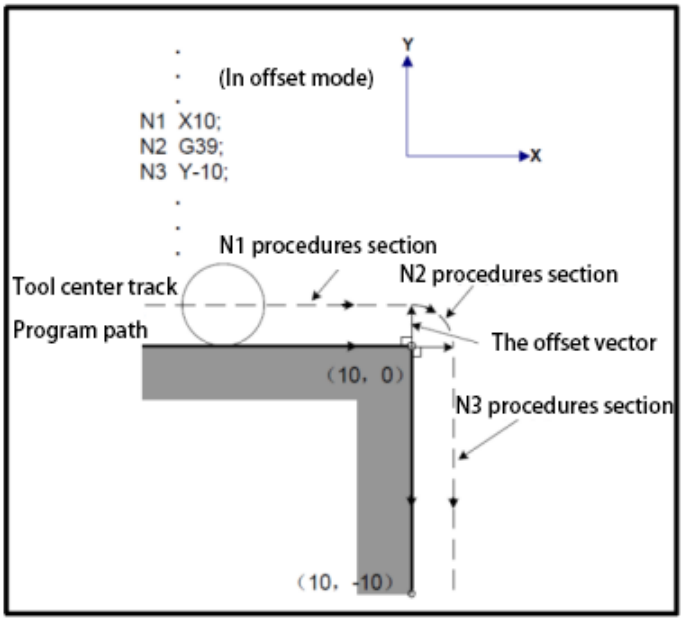


Fig. 3-5-4-1 G39 without I, J, and K

4. When G39 and I, J, and K are instructed, an arc is formed at the corner, so that the vector at the end of the arc is perpendicular to the one by I and J and the K value as determined by the vector. As shown in the following figure:

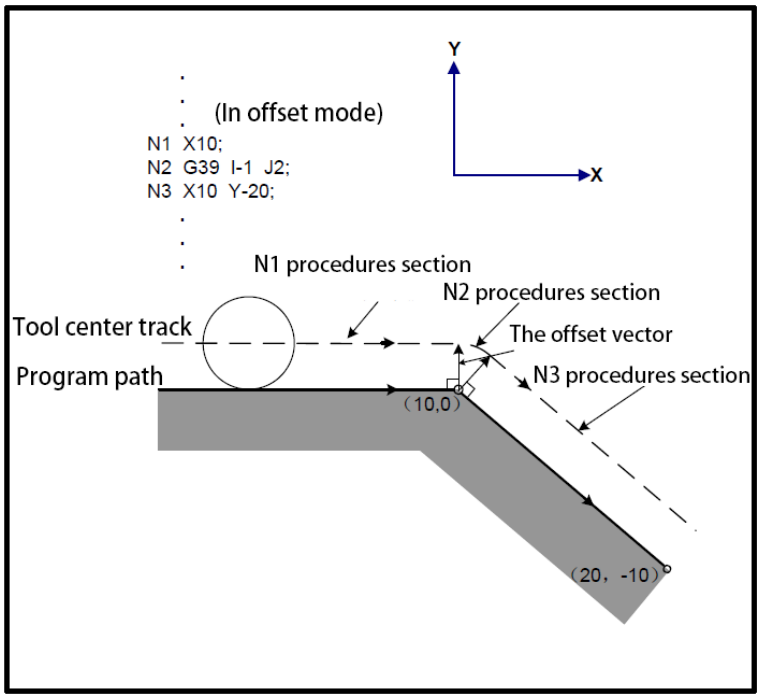


Fig. 3-5-4-2 G39 with I, J and K

3.5.5 Tool compensation value, compensation number with program input (G10).

Instruction format: G10 L10 P_ R_; The geometry compensation value for the H code

G10 L12 P_ R_ ； The geometry compensation value for the D code

G10 L11 P_ R_ ； The wear compensation value for the H code

G10 L13 P_ R_ ； The wear compensation value for code D

P: Tool compensation number.

R: The tool compensation value of the absolute value command (G90) method.

The tool complement value in the incremental value instruction (G91) mode, which is added to the specified tool compensation number value (and the tool compensation value).

Illustrate:

Valid input range for tool compensation values:

Geometric compensation: metric input ±999.999mm; The imperial input ± 99.9999inch.

Wear compensation: metric input ±999.999mm; The imperial input ± 99.9999inch.

Note: 1. When switching between the metric and imperial systems, the tool offset amount is set by the bit parameter **NO:3#0 whether the tool offset is automatically transformed.**

2. The maximum value of wear compensation is limited by the number parameter P331.

3.6 Feed G code

3.6.1 Feed mode G64/G61/G63

Instruction format:

Quasi-stop mode G61

Tapping method G63

Cutting method G64

Function:

Stop-and-go G61: Once specified, this feature is in effect until specified by G62, G63, or G64. The tool decelerates at the end of the program section to perform a spot check and then performs the next program section.

Tapping G63: Once specified, this feature is in effect until specified by G61, G62, or G64. The tool executes the next program segment at the end of the program segment without slowing down. When G63 is specified, neither the feed speed multiplier nor the feed pause are valid.

Cutting method G64: Once specified, this function is in effect until G61, G62 or G63 is specified. The tool executes the next program segment at the end of the program segment without slowing down.

Illustrate:

1. No parameter format.

2. G64 is the default feed mode of the system, and the end point of the program section does not slow down, and the next section is directly executed.

3. The purpose of the in-place inspection in the quasi-stop method is to check whether the servo motor has reached the specified position range.

4. In the accurate stop mode, the cutting method and the tapping method The trajectory of the tool movement is different.

For details, see Figure 3-6-1-1 below

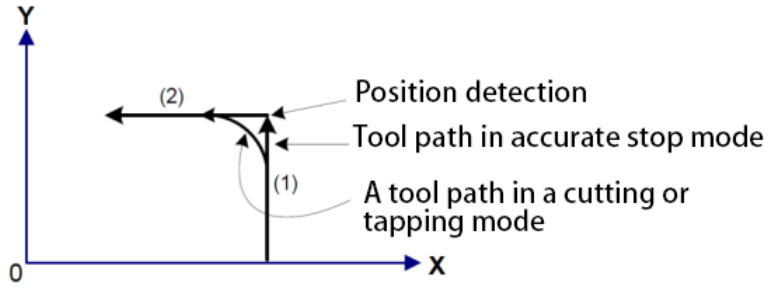


Fig. 3-6-1-1 Tool trajectory from program section 1 to program section 2

3.6.2 Automatic corner magnification (G62).

Instruction format: G62

Function: Automatic corner rate mode G62, once specified, until G61, G63 or G64 specified, this function is always valid. During tool radius compensation, when the tool moves along the inner corner, the cutting feed speed is multiplied to suppress the amount of cutting per unit time, so that good surface accuracy can be machined.

Illustrate:

1、 When performing tool radius compensation, when the tool moves in the inner corner and inner arc area, it automatically decelerates to reduce the load on the tool and machine a smooth surface.

2、 By bit parameter NO. 55#1 Set whether the automatic corner magnification function is effective. By bit parameter NO. 17#2 control automatic corner deceleration function (0: angle control, 1: speed difference control).

3、 When the G62 is specified and has the tool radius compensation function and the inner corner is machined, the feed rate is automatically adjusted at both ends of the corner. There are four types of inner corners shown in Figure 3-6-2-1. In the figure: 2°≤θ≤θp≤178°. θp is set with the data parameter P160.

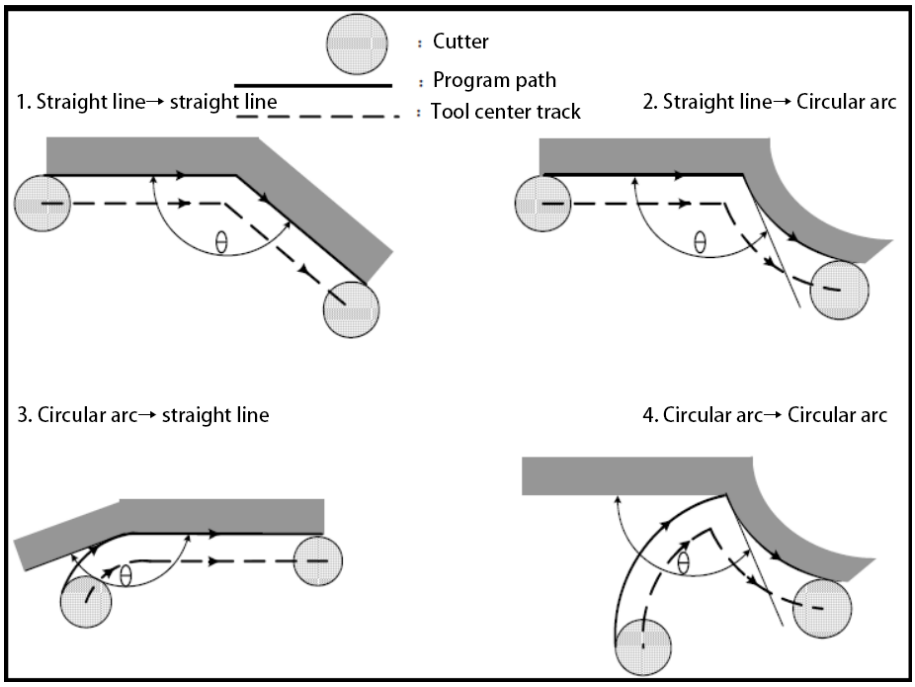


Fig. 3-6-2-1

4、 When the corner is determined to be an inner corner, the feed velocity multiplier is performed before and after the inner corner. The distance from which the feed magnification is performed is Ls and Le, and Ls and Le are the distances from the point on the center trajectory of the tool to the corner. As shown in Fig. 3-6-2-2, where Ls+Le ≤ 2mm.

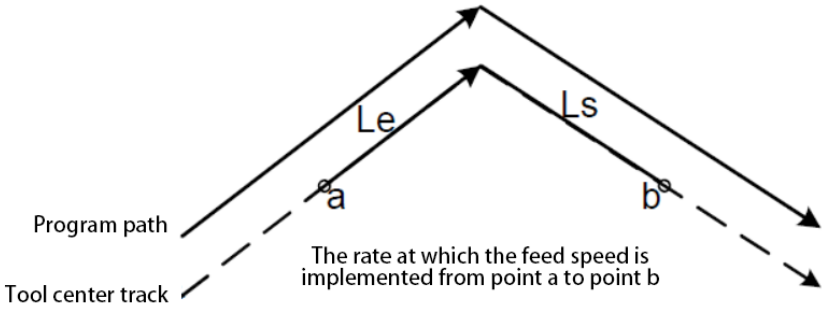


Fig. 3-6-2-2 Straight to straight line

5、 When the programming trajectory includes two arcs, if the start and end points are in the same quadrant or in adjacent quadrants, the feed velocity is multiplied, and the data parameter P161 is used to control the automatic corner deceleration of the lowest feed velocity, as shown in Fig. 3-6-2-3.

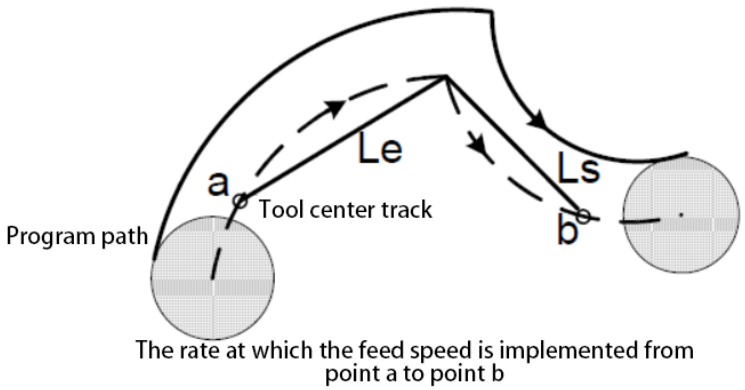


Fig. 3-6-2-3 Arc-to-arc

6、 Consider that a program has a straight line to an arc and an arc to a straight line, as shown in Fig. 3-6-2-4, the feed velocity from point a to point b and from point c to point d is multiplied.

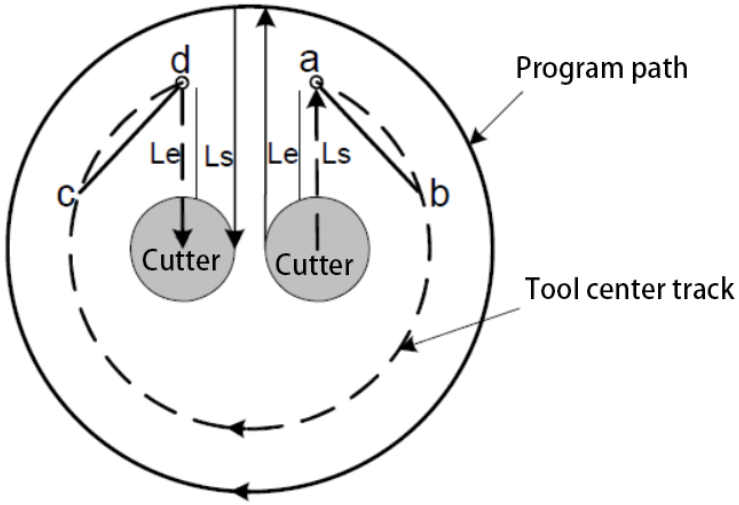


Fig. 3-6-2-4 Straight line to arc**, arc to straight line**

Limit:

1. During the acceleration/ deceleration period before interpolation, the inner corner is invalid.

2. If there is a knife starting program section before the corner or a program section including G41 or G42 after the corner, the inner corner multiplier is invalid.

3. If the bias is zero, the inner corner is not executed.

3.7 Macro function G code

3.7.1 User macro programs

A certain function implemented by a set of instructions is stored in memory in advance like a subroutine, and these functions are represented by one instruction. These functions can be

implemented by simply writing out the representative instructions in the program. This set of instructions is called the user macro program ontology, and the representative instructions are called "user macro instructions". The user macro program itself is sometimes referred to as a macro program, and the user macro is also known as a macro program call instruction.

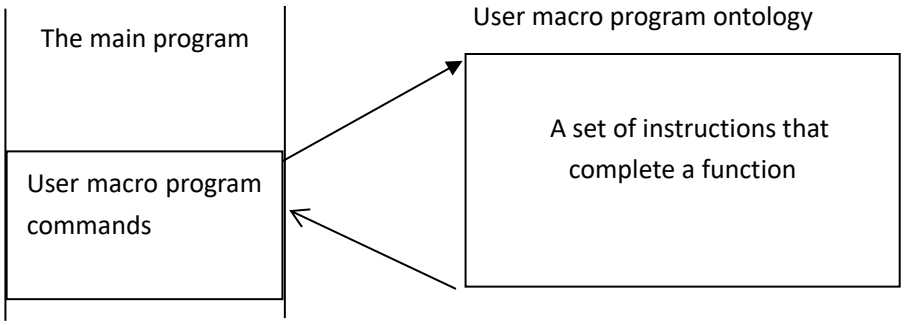


Fig. 3-7-1-1

Variables can be used in the user macro program ontology. Operations can be performed between variables, and macros can be used to assign values to variables.

3.7.2 Macro variables

In user macros, you can use general CNC instructions, as well as variable, arithmetic, and transfer instructions.

User macro programs start with the program number and end with M99 .

```
O0066;                    Program number
G65 H01 ……;               Operational
G90 G00 X#101 ……;         CNC instructions using variables
……
……
……
G65 H82 ……;               Jump instruction
……
……
M99;                      The user macro program ontology ends
```

Fig. 3-7-1-2 (Composition of the user macro program ontology).

1. How to use variables

Variables allow you to instruct the values of parameters in the user macro program body. Variable values can be assigned by the main program or set by LCD/MDI , or to calculated values when the user macro program ontology is executed.

Multiple variables can be used, which are distinguished by variable numbers.

(1) Representation of variables

The variable is represented by # subsequent variable numbers, in the following format:

#i(i = 1，2，3，4 .....)

(Example) #5 ，#109，#1005

(2) References to variables

Variables allow you to displace the numeric value after the parameter value.

(Example).

F#103 is the same as F15 instructions when #103=15.

G#130 is the same as G3 when #130=3.

Note: 1. Parameter words O and N (program number and sequence number) cannot refer to variables. Cannot be programmed with O#100, N#120.

2. If the maximum instruction value specified by the parameter value is exceeded, it cannot be used. #30 = 120, M#30 exceeds the maximum instruction value.

3. Display and setting of variable values: variable values can be displayed on the LCD screen, and the variable values can also be set by MDI.

2. The type of variable

Depending on the variable number, variables are divided into null, local, common, and system variables, all of which have different purposes and properties.

(1) Empty variable #0: (The variable is always empty, no value can be assigned to the variable).

(2) Local variables #1 ~ #50: Local variables can only be used to store data in macro programs, e.g. as a result of an operation. Automatic clearance when power is cut off, reset or program ends (M30, M02 execution). When a macro program is called, the argument assigns a value to the local variable.

(3) Common variables #100 ~ #199, #500 ~ #999:

Common variables are common in the main program and in the individual user macros called by the main program. That is, the variables used in a user macro program #i and the #i used by other macro programs are the same. Therefore, common variables for the result of operations in one macroprogram#i can be used in other macros.

The use of common variables is not specified in the system, and users can use them freely.

Table 3-7-2-1

The variable number	The variable type	function
#100 ～ #199	Common variables	Clear when power is turned off, reset all to "empty" when power is turned on
#500 ～ #999		Data is saved in a file and is not lost even if the power is lost

(4) System variables: System variables are used to read and write changes to various data at the CNC runtime. Each is as follows:

1) Interface input signal #1000 --- #1047 (read PLC input signal bitwise).
2) Interface output signal #1100 --- #1147 (the signal written bitwise to PMC).
3) Tool length compensation value #1500 --- #1755 (read/write).
4) Length wear compensation value #1800 --- #2055 (read/write).
5) Tool radius compensation value #2100 --- #2355 (read/write).
6) Radius wear compensation value #2400 --- #2655 (read/write).
7) Tool magazine data #2700 --- #2955 (read-only, not writable).
8) Alarm #3000
9) User Datasheet #3500 --- #3755 (read-only, not writable).
10) Modal information #4000 --- #4030 (read-only, not write-able).
11) Location information #5001 --- #5030 (read-only, not write-able).
12) Workpiece zero offset #5201 --- #5235 (read/write).
13) Additional workpiece coordinate system #7001 --- #7250 (read/write).

3. Detailed description of system variables

1) Modal information

Table 3-7-2-2

The	function	The group
#4000	G10,G11	00 groups
#4001	G00,G01,G02,G03	Group 01
#4002	G17,G18,G19	02 groups
#4003	G90,G91	03 groups
#4004	G94,G95	04 groups
#4005	G54,G55,G56,G57,G58,G59	05 groups
#4006	G20,G21	06 groups
#4007	G40,G41,G42	07 groups
#4008	G43,G44,G49	08 groups
#4009	G73,G74,G76,G80,G81,G82,G83,G84,G85,G86,G87,G88,G89	09 groups
#4010	G98,G99	10 groups
#4011	G15,G16	11 groups
#4012	G50,G51	12 groups
#4013	G68,G69	13 groups
#4014	G61,G62,G63,G64	14 groups
#4015	G96,G97	15 groups
#4016	To be extended	16 groups
#4017	To be extended	17 groups

#4018	To be extended	18 groups
#4019	To be extended	19 groups
#4020	To be extended	20 groups
#4021	To be extended	21 groups
#4022	D	
#4023	H	
#4024	F	
#4025	M	
#4026	S	
#4027	T	
#4028	N	
#4029	O	
#4030	P (Additional workpiece coordinate system now selected).	

Note: 1, P code is the currently selected additional workpiece coordinate system.

When G#4002 is executed, the value obtained in #4002 is 17, **18, or 19.**

3, modal information can not be written, can only be read.

2) Current location information

Table 3-7-2-3

The variable number	Location information	The relevant coordinate system	Read operation on move	Tool compensation value
#5001	X-axis program segment end point	The workpiece coordinate system	OK	The tip position (program instruction position) is not considered
#5002	Y-axis program segment end point			
#5003	Z-axis program segment end point			
#5004	4th axis program segment end point			
#5005	5th axis program segment end point			
#5006	The end point of the X-axis program	Machine coordinate system	No	Consider the tool datum point location (Machine coordinates)
#5007	The end location of the Y-axis			
#5008	Z-axis program segment end location			
#5009	4th Axis Program Segment End			
#5010	5th Axis Program Segment End			
#5011	X-axis program segment end point	The workpiece coordinate system		
#5012	The end location of the Y-axis			
#5013	Z-axis program segment end location			
#5014	4th Axis Program Segment End			
#5015	5th Axis Program Segment End			
#5016	The end point of the X-axis program		OK	
#5017	The end location of the Y-axis			
#5018	The Z-axis program segment end			

#5019	4th Axis Program End Point			
#5020	5th Axis Program Segment End			
#5021	X-axis tool length compensation			
#5022	Y-axis tool length compensation			
#5023	Z-axis tool length compensation			
#5024	4th axis tool length compensation			
#5025	5th axis tool length compensation			
#5026	X-axis servo position compensation			
#5027	Y-axis servo position compensation			
#5028	Z-axis servo position compensation			
#5029	4th axis servo position compensation			
#5030	5th axis servo position compensation			

Note: 1, ABSIO: in the workpiece coordinate system, the coordinate value of the end point of the previous program segment.

2. ABSMT: In the coordinate system of the machine tool**, the current machine tool coordinate system position.**

3. ABSOT: the current coordinate position in the workpiece coordinate system.

4. ABSKP: In the coordinate system of the workpiece, the position of the jump signal in the **G31 program section is valid.**

3) Workpiece zero offset and additional zero point offset:

Table 3-7-2-4

The	function
#5201	The zero offset value of the external workpiece on the 1st axis
...	...
#5205	Zero offset of the external workpiece in the 5th axis
#5206	1st axis G54 workpiece zero offset value
...	...
#5210	5th axis G54 workpiece zero offset
#5211	1st axis G55 workpiece zero offset value
...	...
#5215	5th axis G55 workpiece zero offset
#5216	1st axis G56 workpiece zero offset value
...	...
#5220	5th axis G56 workpiece zero offset
#5221	1st axis G57 workpiece zero offset value
...	...
#5225	5th axis G57 workpiece zero offset

#5226	1st axis G58 workpiece zero offset value
...	...
#5230	5th axis G58 workpiece zero point offset
#5231	1st axis G59 workpiece zero offset value
...	...
#5235	5th axis G59 workpiece zero offset
#7001	1st axis G54 P1 workpiece zero offset value
...	...
#7005	5th axis G54 P1 workpiece zero point offset
#7006	1st axis G54 P2 workpiece zero offset value
...	...
#7010	5th axis G54 P2 workpiece zero offset
#7246	1st axis G54 P50 workpiece zero offset value
...	...
#7250	5th axis G54 P50 workpiece zero offset

4. The number of parts machined and the number of parts that need to be machined:

The number of parts machined and the number of parts that need to be machined	
The variable number	Function (can be read and written)
#3901	Number of machined parts (number of completed) [maximum value of 99999, automatically cleared when the maximum value is exceeded].
#3902	The number of parts to be machined (target quantity) is required [99999 maximum].

When the value of #3901 is changed, the number of parts displayed in the position interface also changes, and the number of parts processed is also changed by the synthetic number parameter 365 record; When you change the value of parameter 365, the number of parts displayed in the position interface also changes, and the value of the system variable #3901 also changes.

When the value of #3902 is changed, the number of required parts displayed in the location interface also changes, and the number of requirements of the synthetic number parameter 366 changes; When you change the value of parameter 3 66, the number of required parts displayed in the location interface also changes, and the value of the system variable #3902 also changes.

5. Local variables

Address correspondence with local variables:

Table 3-7-2-5

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

Note: 1. Use English letters followed by numeric values for assignment, except for G, L, O, N, H, and P Each of the **20 English letters can be assigned a value to the independent variable, and each letter is assigned a value once, from A-B-C-D... To X - Y - Z, the assignment does not have to be done alphabetically, and addresses that are not assigned can be omitted.**

2. G65 must be specified before using any arguments.

6. Notes on the user macro program ontology

1) The method entered with the key

In the parameter words G, X, Y, Z, R, I, J, K 、F、H、M、S、T、P、Q Press the # key at the back of the button and the # will be entered.

2) In the MDI state, instructions can also be manipulated and instructions transferred.

3) H, P, Q, and R of the operation and transfer instructions are used as parameters of G65 commands before and after G65.

H02 G65 P#100 Q#101 R#102 ； Correct

N100 G65 H01 P#100 Q10 ； Correct

4) The range of variable values is: -9999.9999 ~ 9999.9999 .

5) The result of the variable value operation can be a decimal with a precision of 0.0001. Except for H11 (or operation), H12 (with operation), H13 (non-operation), H23 (residual operation) will ignore the fractional part of the variable during the calculation process, and other operations will not give up the decimal point for operation.

Example:

#100 ＝ 35， #101 ＝ 10，#102 ＝ 5

#110 ＝ #100÷#101 (＝ 3.5)

#111 ＝ #110×#102 (＝ 17.5)

#120 ＝ #100×#102 (＝ 175)

#121 ＝ #120÷#101 (＝ 17.5)

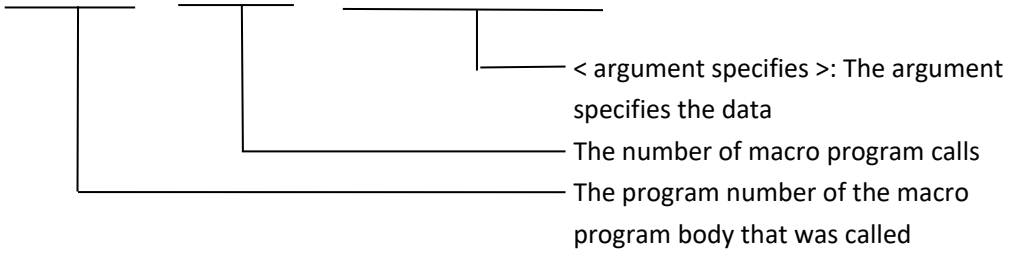
6) The execution time of the operation and transfer instructions varies depending on the conditions, and the general average can be considered as 10 milliseconds.

3.7.3 User macro program calls

When instruction G65 is given, the user macro program specified at address P is called, and the data is passed to the user macro body via arguments.

The directive format is as follows:

G65 P □□□□□ L □□□□ <arguments specified>;



After G65, specify the program number of the user macro program with address P, specify the number of macro program calls with L, and pass data to the macro program with arguments.

When duplicates are required, specify the number of repetitions from 1 to 9999 after address L, and the default number of times is 1 when L is omitted.

Specify using an independent variable whose value is assigned to the corresponding local variable.

Note: 1. When the **subroutine number specified with address P cannot be retrieved,** an alarm **(PS 078) is generated.**

2, **9000 ~ 9999** subroutine for the system reserved program, when the user calls the subprogram, the **system can execute the content of the subprogram, but the cursor will always stay in the M98 instruction segment, the program interface has been showing the main program content. (You can display the contents of the subroutine by modifying the bit parameter N0:29#5.)** ）

3. Macro programs cannot be called in DNC mode.

3.7.4 Operations and transfer instructions

1. General form:

G65 Hm P#I Q#J R#K ；

m:01 ~ 99 indicates the operation instruction or transfer instruction function.

#I: The name of the variable that stored the result of the operation.

#J: The name of the variable 1 on which the operation was performed. It can also be a constant. Constants are expressed directly, without #.

#K: The name of the variable on which the operation is performed2. It can also be a constant.

Meaning: #I = #J ○ #K (○ operation symbol, specified by H m).

(Example).

P#100 Q#101 R#102..... #100 = #101 ○ #102 ；

P#100 Q#101 R15 ..... #100 = #101 ○ 15 ；

P#100 Q-100 R#102..... #100 = -100 ○ #10

The H code specified with G65 has no effect on the selection of the offset amount.

Code format	function	definition
G65 H01 P#I Q#J	Assignment operations	#I =#J ； Assign the value of the variable #j to the variable #i
G65 H02 P#I Q#J R#K	Decimal addition	#I = #J + #K
G65 H03 P#I Q#J R#K	Decimal subtraction	#I = #J - #K
G65 H04 P#I Q#J R#K	Decimal multiplication	#I = #J × #K
G65 H05 P#I Q#J R#K	Decimal division	#I = #J ÷ #K
G65 H11 P#I Q#J R#K	Binary addition (or operation)	#I = #J OR #K
G65 H12 P#I Q#J R#K	Binary multiplication (and operations)	#I = #J AND #K
G65 H13 P#I Q#J R#K	Binary XOR	#I = #J XOR #K
G65 H21 P#I Q#J	Decimal opens square	#I = $\sqrt{\#J}$
G65 H22 P#I Q#J	Decimal takes the absolute value	#I = $\lvert\#J\rvert$
G65 H23 P#I Q#J R#K	Decimal takes the remainder	#I = ÷ remainder of (#J #K).
G65 H24 P#I Q#J	Decimal becomes binary	#I = BIN (#J)
G65 H25 P#I Q#J	Binary becomes decimal	#I = BCD (#J)
G65 H26 P#I Q#J R#K	Decimal multiplication and division	#I = #I × #J ÷ #K
G65 H27 P#I Q#J R#K	Compound square root	#I = $\sqrt{\#\mathrm{J}^2 + \#\mathrm{K}^2}$

G65 H31 P#I Q#J R#K	sine	#I = #J × SIN(#K)
G65 H32 P#I Q#J R#K	cosine	#I = #J × COS(#K)
G65 H33 P#I Q#J R#K	Cut the strings	#I = #J × TAN(#K)
G65 H34 P#I Q#J R#K	Cut anyway	#I = #J × ATAN(#K)
G65 H80 Pn	Unconditional transfer	Jump to program segment n
G65 H81 Pn Q#J R#K	Condition Transfer 1	If #J = #K , you are skipped to program segment n, otherwise executed sequentially
G65 H82 Pn Q#J R#K	Condition Transfer 2	If #J ≠ #K , it is skipped to program segment n, otherwise executed sequentially
G65 H83 Pn Q#J R#K	Condition Transfer 3	If #J > #K, jump to program segment n, otherwise execute sequentially
G65 H84 Pn Q#J R#K	Condition Transfer 4	If #J < #K, jump to program segment n, otherwise execute sequentially
G65 H85 Pn Q#J R#K	Condition Transfer 5	If #J ≥ #K , it is skipped to program segment n, otherwise executed sequentially
G65 H86 Pn Q#J R#K	Condition Transfer 6	If #J ≤ #K , it is skipped to program segment n, otherwise executed sequentially
G65 H99 Pn	Generate user alarms	Generate (3000+ n) user alarms

Fig. 3-7-4-1

2. Operation instructions:

1) Assignment of variables: #I=#J

G65 H01 P#I Q#J；

(Example) G65 H01 P#101 Q1005; （#101 = 1005）

G65 H01 P#101 Q#110； （#101 = #110）

G65 H01 P#101 Q-#102； （#101 = -#102）

2) Addition operation: #I=#J+#K

G65 H02 P#I Q#J R#K；

G65 H02 P#101 Q#102 R15; （#101 = #102+15）

3) Subtraction: #I=#J-K

G65 H03 P#I Q#J R#K；

G65 H03 P#101 Q#102 R#103; （#101 = #102-103）

4) Multiplication: #I=#J×#K

G65 H04 P#I Q#J R#K；

G65 H04 P#101 Q#102 R#103; （#101 = #102×#103）

5) Divide operation: #I=#J÷ #K

G65 H05 P#I Q#J R#K;

G65 H05 P#101 Q#102 R#103; （#101 = #102÷#103）

6) Logical addition (or): #I = #J OR.#K

G65 H11 P#I Q#J R#K;

G65 H11 P#101 Q#102 R#103; （#101 = #102.OR. #103）

7) Logical multiplication (and): #I=#J.AND.#K

G65 H12 P#I Q#J R#K;

G65 H12 P# 101 Q#102 R#103; （#101 = #102.AND.#103）

8) XOR: #I=#J.XOR.#K

G65 H13 P#I Q#J R#K;

G65 H13 P#101 Q#102 R#103; （#101 = #102.XOR.#103）

9) Square root: #I =$\sqrt{\#J}$

G65 H21 P#I Q#J;

10) Absolute value: #I=|#J|

G65 H22 P#I Q#J ;

(Example) G65 H22 P#101 Q#102; （#201 = |#102| ）

11) Take the remainder: #I=#J-TRUNC(#J/#K)× #K, TRUNC: Rounds off the fractional part

G65 H23 P#I Q#J R#K;

G65 H23 P#101 Q#102 R#103; （#101 = #102-TRUNC(#102/#103)×#103）

12) Compound multiplication and division: #I = (#I × #J) ÷ #K

G65 H26 P#I Q#J R#K;

G65 H26 P#101 Q#102 R#103; （#101 =（#101×#102）÷#103）

13) Composite square root: #I= $\sqrt{\#I^2 + \#K^2}$

G65 H27 P#I Q#J R#K;

G65 H27 P#101 Q#102 R#103; （#101 =$\sqrt{\#102^2 + \#103^2}$）

14) Sine: #I = #J • SIN (#K) (unit: degrees).

G65 H31 P#I Q#J R#K;

G65 H31 P#101 Q#102 R#103; （#101 = #102•SIN（#103））

15) Cosine: #I = #J • COS (#K) (unit: degrees).

G65 H32 P#I Q#J R#K;

G65 H32 P#101 Q#102 R#103; （#101 =#102•COS（#103））

16) Tangent: #I = #J • TAM (#K) (unit: degrees).

G65 H33 P#I Q#J R#K;

G65 H33 P#101 Q#102 R#103; （#101 = #102•TAM（#103））

17) Arctangent: #I=ATAN (#J/#K) (unit: degrees).

G65 H34 P#I Q#J R#K;

G65 H34 P#101 Q#102 R#103; （#101 =ATAN（#102/#103））

Note: 1. The unit of the angle variable is degrees.

2. In each operation, when the necessary Q and R are not specified, their values participate in the operation as zeros.

3. trunc: Rounding and discarding the fractional part.

3. Transfer command

1) Unconditional transfer

G65 H80 Pn; n: Sequence number

(Example) G65 H80 P120; (Go to N120 Program).

2) Conditional transfer 1#J.EQ.#K (=).

G65 H81 Pn Q#J R#K; n: Sequence number

(Example) G65 H81 P1000 Q#101 R#102 ;

When #101=#102, go to the N1000 program segment, and when #101 ≠ #102 , the program executes sequentially.

3) Conditional transfer 2#J.NE.#K (≠).

G65 H82 Pn Q#J R#K; n: Sequence number

(Example) G65 H82 P1000 Q#101 R#102 ;

When #101 ≠ #102, go to the N1000 program segment, and when #101 = #102 , the program executes sequentially.

4) Conditional transfer 3#J.GT.#K (>).

G65 H83 Pn Q#J R#K； n: Sequence number

(Example) G65 H83 P1000 Q#101 R#102 ；

When #101 > #102, go to the N1000 program segment, and when #101 ≤ #102 , the program executes sequentially.

5) condition transfer4#J.LT.#K (＜)

G65 H84 Pn Q#J R#K； n: Sequence number

(Example) G65 H84 P1000 Q#101 R#102 ；

当# 101 ＜ #102 , go toN1000 Program segment, when#101 ≥ #102 The program executes sequentially.

6) Conditional transfer 5#J GE.#K (≥)

G65 H85 Pn Q#J R#K； n: Sequence number

(Example) G65 H85 P1000 Q#101 R#102 ；

当# 101 ≥ #102 , go toN1000 Program segment, when#101 ＜ #102 The program executes sequentially.

7) Conditional transfer 6 #J LE.#K (≤)

G65 H86 Pn Q#J R#K； n: Sequence number

(Example) G65 H86 P1000 Q#101 R#102 ；

When #101 ≤ #102, go to the N1000 program segment, and when #101 > # 102 , the program executes sequentially.

Note: You can specify the sequence number with a variable. For example: G65 H81 P#100 Q#101 R#102; When the condition is met, the program moves to the program segment for the sequence number specified by **#100.**

4. Logic and, Logic Or, and Logic Non-Instruction

Example:

G65 H01 P#101 Q3；

G65 H01 P#102 Q5；

G65 H11 P#100 Q#101 Q#102；

5 is represented as binary 101, 3 is represented as 011, and the calculation result is #100=7;

G65 H12 P#100 Q#101 Q#102；

5 is represented as binary 101, 3 is represented as 011, and the calculation result is #100=1;

3.7.5 User macro program instances

1. Bolt hole circulation

On a circumference with a datum point (X0, Y0) and a radius of (R), with an initial angle of (A), N aliquots are machined.

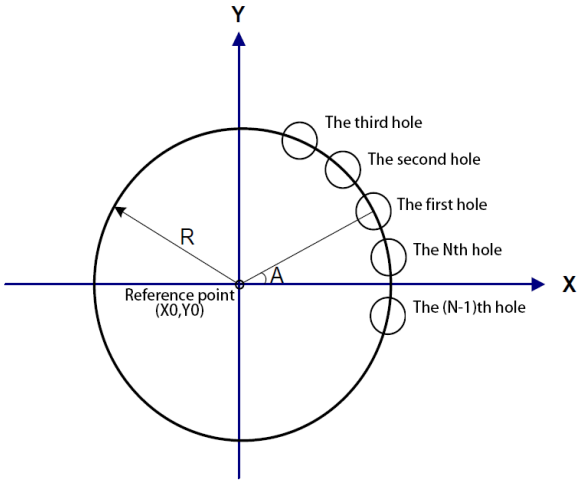


Fig. 3-7-5-1

X0,Y0 are the coordinate values of the bolt hole cycle datum point.

R: radius, A: start angle, N: number. The above parameters use the following variables.

#500: The coordinate value of datum point X (X0).

#501: The coordinate value (Y0) of the datum point Y

#502: Radius (R).

#503: Initial angle (A).

#504: N numbers

N > 0, reverse-clockwise, number N.

N ＜ 0 time, clockwise, numberN。

The following variables are used for operations in macro programs.

#100: Indicates the count (I) of the I hole

#101: The final value of the count (= | N|) (IE)

#102: The angle of the first hole (θI).

#103: The X coordinate value (Xi) of the first hole

#104: The Y coordinate value of the first hole (Yi).

The user macro program body can be written in the following form:

O1010;

#500 Datum point X coordinate value (center coordinate)

#501 Datum Point Y coordinate value (center coordinate)

#502 Radius of circumference R

#503 Start Corner A

#504 Number of holes N

#505 Punch depth

#506 Punch speed

N>0 moves counterclockwise

N<0 moves clockwise

#100 Number of holes

The final value of the #101 count

#102 I hole angle

#103 The X coordinate value of the first hole I

#104 Y coordinate value of the first hole I

```
M3S1200;
G90G54G40G17G80;
G0Z100;
G0X#500Y#501;// center position
N100G65H01P#100Q0;//I=0
G65H22P#101Q#504;//IE=N
N200G65H05P#105Q360R#101;// circumferential average distribution pore angle
G65H04P#106Q#105R#100;
G65H84P300Q#504R0;//Counterclockwise punching judgment
G65H02P#102Q#503R#106;// I hole angle
G65H80P400;
N300G65H03P#102Q#503R#106;// I hole angle
N400G65H32P#107Q#502R#102;
G65H02P#103Q#500R#107;//I hole X=R*COS#102
G65H31P#108Q#502R#102;
G65H02P#104Q#501R#108;//I-well X=R*COS#102
G99G81X#103Y#104Z#505R5F#506;//Punch G Code
G65H02P#100Q#100R1;//I=I+1
G65H84P200Q#100R#101;//
G80M05;
M30;
```

Appendix I. CNC Alarm

Alarm number	contents
0000	modifies the parameter that the power must be cut off once
0001	Failed to open file
0002	Input data is out of range
0003	The copied or renamed program number exists
0004	address not found
0005	There is no data behind the address
0006	Illegal use of minus signs
0007	Illegal use of decimal points
0008	The program file is too large to be fully loaded
0009	Enter an illegal address
0010	Incorrect G code
0011	No feed speed command
0012	Insufficient disk space
0013	The number of program files has reached the upper limit
0014	cannot command G95, spindle is not supported
0015	commands too many axes
0016	The current pitch error compensation point is out of range
0017	No permission to modify
0018	Modifications are not allowed
0019	Zoom function is not enabled
0020	exceeds radius tolerance
0021	has ordered illegal plane axes
0022	In the arc, R and IJK are all 0
0023	Arc interpolation is specified in both IJK and R
0024	The helix interpolation rotation angle is 0
0025	G12 cannot be in the same segment as other G instructions
0026	The file format is not supported by the system
0027	The length knife repair instruction cannot be the same as G92
0028	Illegal plane selection
0029	illegal bias value
0030	illegal compensation number
0031	G10 is ordered in the illegal P
0032	Illegal compensation value in G10

0033 knife patch C or chamfer in no intersection point

0034 Arc command cannot be created or canceled when the knife patch is created

0035 M99 did not cancel the C knife patch before the instruction

0036 cannot command G31

0037 The plane cannot be changed in knife patch C

0038 interference in the arc program segment

0039 Tool tip positioning error in Tool Patch C

0040 tool patch C execution changes the workpiece coordinate system

0041 There is interference in

0042 Non-moving instructions in Knife Patch C over ten

0043 Insufficient permissions

0044 does not allow instructions G27~G30 in a fixed cycle

0045 address Q not found or Q value is 0 (G73/G83)

0046 Illegal reference point return instruction

0047 Mechanical zeroing is required before executing this instruction

0048 The Z plane should be lower than the R plane

0049 The Z plane should be lower than the R plane

0050 When changing the fixed cycle mode, the position should be moved

0051 Wrong movement after chamfering or the chamfer value is too large

0052 Milling slot fixing cycle can not use the mirror function

0053 Too many address instructions

0054 DNC transmit error

0055 Chamfer or inverted R in the wrong move value

0056 M99 cannot be in the same paragraph as macro instructions

0057 Write to file failed, must power off and restart

0058 No end points found

0059 The program number was not found

0060 No sequence number found

0061 The X axis is not at the reference point

0062 The Z axis is not at the reference point

0063 The Y axis is not at the reference point

0064 The 4TH axis is not at the reference point

0065 The TH5 axis is not at the reference point

0066 The fixed loop must be unlocked before G10 is executed

0067 G10 does not support formatting

0068 The parameter switch is not turned on

0069 processing operation needs to close the U disk operation interface

0070 Memory capacity is insufficient memory

0071 No data end found

0072 Too many programs

0073 The program number is already in use

0074 illegal program number

0075 Protection

0076 Address P is not defined

0077 Subprogram nesting error

0078 Program number not found

0079 System usage time expires

0080 The data entered by is unreasonable

0082 The H code is instructed in G37

0083 G37 Illegal Axis Directive

0084 The button times out or is short-circuited

0085 Communication error

0087 X axis reference point returns unfinished

0088 Y axis reference point returns outstanding

0089 Z axis reference point returns to incomplete

0090 4TH axis reference point returns unfinished

0091 5TH axis reference point return is not complete

0092 is not on the axis of the reference point

0094 P type not allowed (coordinates)

0095 P Type Not Allowed (EXT OFS CHG)

0096 P Type Not Allowed (WRK OFS CHG)

0097 P type not allowed (automatic)

0098 G28 found in sequential return

0099 MDI is not allowed after retrieval

0100 The parameter is valid for writing

0101 power off memory data is confused, please make sure the location is correct

0110 position data exceeds the allowed range, please return to zero

0111 Compute Data Overflow

0112 is divided by zero

0113 Incorrect instruction

0114 Macro program format error

0115 illegal variable

0116 Write-protected variables

0118 Brace nesting error

0119 M00~M02,M06,M98,M99,M30 cannot be in the same section as other M instructions

0122 Quadruple macromodal-call

0123 Macros cannot be used in DNC

0124 Procedure ended illegally

0125 Macro program format error

0126 number of illegal loops

0127 NC and macros in the same program segment

0128 Sequence number of illegal macro

0129 Illegal argument address

0130 Illegal shaft operation

0131 Too much external alarm information

0132 No alarm number found

0133 Axis commands are not supported by the system

0134 The number of control axes in the system is greater than 3 axes can not use rigid tapping

0135 Illegal Angle Directive

0136 Illegal Axis Directive

0139 can not change the PLC control axis

0142 illegal proportional rate

0143 Zoom motion data overflow

0144 illegal plane selection

0148 illegal data settings

0149 G10L3 format error

0150 illegal tool group number

0151 No tool group number found

0152 Tool data cannot be stored

0153 did not cancel the C knife repair before changing the knife

0154 Tools in the unused life group

0155 Illegal T code in M06

0156 P/L instruction not found

0157 Too many tool sets

0158 Illegal tool life data

0159 Tool data setting is not completed

0160 In the polar coordinate mode, the arc can only be programmed with R

0161 The command cannot be executed in polar coordinate mode

0163 The command cannot be executed in rotation mode

0164 The instruction cannot be executed in zoom mode

0165 Please specify this directive in a separate program segment

0166 No axis is specified when returning to the reference point

0167 The coordinates of the intermediate point are too large

0168 The minimum pause time at the bottom of the hole should be less than the maximum pause time at the bottom of the hole

0170 Tool radius compensation is not removed when entering or exiting the subprogram

0172 In the program segment of the calling subprogram, P is not an integer or P is less than 0

0173 The number of subprogram calls should be less than 9999

0175 The fixed loop can only be performed in the G17 plane

0176 Rigid tapping does not specify the spindle speed before starting

0177 does not support spindle orientation function

0178 The spindle speed is not specified before the fixed cycle begins

0181 Illegal M Code

0182 Illegal S Code

0183 illegal T code

0184 The selected tool is out of range

0185 L is too small or L is undefined

0186 L is too big

0187 tool radius is too large

0188 U is too big

0189 The value of U is less than the tool radius

0190 V is too small or Q is undefined

0191 W is too small or Q is undefined

0192 Q is too small or Q is undefined

0193 I is undefined or J is 0

0194 J is undefined or J is 0

0195 D is undefined or D is 0

0198 illegal axis selection

0199 Macros are not defined

0200 Illegal S Way Directive

0201 No feed velocity was found in the rigid tapping

0202 Location LSI overflow

0203 Rigid tapping in the program is not correct

0204 illegal shaft operation

0205 Rigid mode DI signal off

0206 can not change the plane (rigid tapping)

0207 The tapping data is incorrect

0208 The command cannot be executed in the G10 mode.

0212 Illegal plane selection

0224 returns the reference point

0231 G10 Illegal format in L50 or L51

0232 There are too many spiral interpolation axes for the instruction

0233 device busy

0235 record ends

0236 Program reboot parameter error

0237 No decimal point

0238 Address duplicate error

0239 parameter 0

0240 G41/G42 is not allowed in the MDI mode

0241 handwheel pulse abnormality

0251 emergency stop alarm

0260 axis name is repeated, please modify the parameters NO.225~227

0451 X axis drive alarm

0452 Y-axis drive alarm

0453 Z axis drive alarm

0454 4TH axis drive alarm

0455 5TH axis drive alarm

0456 spindle drive alarm

0500 Soft limit overrun: -X

0501 Soft Limit Overhaul: +X

0502 Soft limit overpass: -Y

0503 Soft limit overrun: +Y

0504 Soft Limit Overflight: -Z

0505 Soft Limit Overpass: +Z

0506 Soft limit overpass: -4TH

0507 Soft Limit Overrun: +4TH

0508 Soft Limit Overpass: -Th5

0509 Soft Limit Overrun: +Th5

0510 Hard Limit Overrun: -X

0511 Hard Limit Overrange: +X

0512 Hard limit overpass: -Y

0513 Hard Limit Overpass: +Y

0514 Hard Limit Overflight: -Z

0515 Hard Limit Overhaul: +Z

0516 Hard limit overpass: -4TH

0517 Hard Limit Overrange: +4TH

0518 Hard Limit Overflight: -Th5

0519 Hard Limit Overrange: +Th5

0740 Rigid Tapping Alarm: Exceptional

0741 Rigid Tapping Alarm: Exceptional

0742 Rigid Tapping Alarm: LSI

0751 First spindle alarm detected (AL-XX)

0754 spindle abnormal torque alarm

1001 The address of the relay or coil is not set

1002 Function instructions for entering the code do not exist

1003 Function Directive COM/COME is not used correctly

1004 user ladder diagram exceeds the maximum number of allowed rows or steps

1005 Function Directive END1 or END2 is not used correctly

1006 There is an illegal output in the network

1007 Hardware failure or system interruption error causes the PLC to be unable to communicate

1008 The function instruction is not connected correctly

1009 The network horizontal line is not connected

1010 Power off while editing the ladder diagram causes loss of the network on the edit

1011 address data was not entered correctly

1012 The input symbol is undefined or the input address is out of range

1013 specifies illegal characters or data out of range

1014 The CTR address is duplicated

1015 The function instruction JMP/LBL is not handled correctly or exceeds the capacity

1016 The network structure is incomplete

1017 A network structure appears that is not currently supported

1019 TMR address duplicate

1020 The function directive is missing a parameter

1021 The PLC executes a timeout and the system automatically stops the PLC

1022 The function instruction name is missing

1023 The address or constant of the function instruction parameter is out of range

1024 There are unnecessary relays or coils

1025 The function instruction is not output correctly

1026 The number of network connection rows is out of the supported range

1027 The same output address is used in another place

1028 Ladder file format error

1029 The ladder file in use is missing

1030 There are incorrect vertical lines in the network

1031 User data area is full, instruction data table capacity

1032 The first stage of the ladder diagram is too large to be executed in time

1033 The SFT instruction exceeds the maximum number of permissible uses

1034 The function instruction DIFU/DIFD is not used correctly

1035 The conversion of the currently open ladder file was not successful

1036 PLC abnormal stop alarm

1037 The open ladder diagram is inconsistent with the data parameter setting ladder diagram

1039 The instruction or network is not within executable range

1040 The function instruction CALL/SP/SPE is not used correctly

1041 The horizontal conduction cable is connected in parallel with the node network

1042 The PLC system parameter file is not loaded

Appendix II, PLC alarm

A001.5 The tool magazine debug mode function is turned on and the program cannot be started, and K30.7 turns off debug mode

Alarm reason: After using the tool magazine function, open the tool magazine debugging function, and the system starts to run the program alarm;

Processing: press [reset key] to cancel the alarm, check whether the tool magazine is normal, if the tool magazine debugging function K30.7=0 is closed normally; If it is not normal, turn off the tool magazine debugging function after troubleshooting.

A001.6 The external loop open time is too long (with the normally open point) K20.0

Alarm reason: cyclic start button conduction time exceeds 4 seconds, system alarm;

Handling: Check whether the external loop start button of the system is connected to the normally open contact, and check whether the external switch is damaged.

A001.7 External Pause Open Time Is Too Long (Connect Normal Closed Point) K20.1

Alarm reason: external pause button conduction time exceeds 4 seconds, system alarm;

Handling: Check whether the external pause button of the system is connected to the normally closed contact, and check whether the external switch is damaged.

A002.0 The guard door is not closed and automatic operation is not allowed

Alarm reason: In the automatic mode, the protective door is not closed, the program is started, and the system generates an alarm;

Processing: Press [Reset] to cancel the alarm, if the protective door has been closed, detect whether the protective door sensor is damaged or whether the level selection is normal.

A002.1 Tool setter overstroke alarm

Alarm reason: When the overhaul signal of the tool instrument is selected, the override signal of the tool instrument has input, and the Z axis moves in the negative direction;

Processing: Press [Reset] to cancel the alarm and move the Z axis in the square direction.

A002.2 Set the total number of parts to be machined to complete the alarm

Alarm reason: the number of processed parts reaches the number of parameters 361 The total number of processed parts is set, and the number of processed parts is set to the prompt effective K16.3=1 when the alarm is set;

Processing: Press [Reset] to cancel the alarm, manually clear the number of processed parts.

A002.3 When the spindle is rotated, the tool shall not be released

Alarm reason: loose knife in spindle rotation, the system generates an alarm;

Processing: Press [Reset] to cancel the alarm.

A002.4 When the spindle rotates, the tool clamping in place signal is invalid alarm

Alarm reason: The tool clamping signal is not detected in the spindle rotation;

Handling: Press [Reset] to cancel the alarm, check the installation position of the tool clamping signal switch and whether it is damaged.

A002.5 The spindle shall not be started when the tool clamping in place signal is invalid

Alarm reason: the spindle is started when the tool clamping signal is not detected;

Handling: Press [Reset] to cancel the alarm, check the installation position of the tool clamping signal switch and whether it is damaged.

A002.6 The spindle tool is loosened and the spindle must not be started

Alarm reason: the tool is in a loose state to start the spindle;

Processing: Press [Reset] to cancel the alarm, and the spindle can only be started when the tool is clamped.

A004.1 The spindle analog voltage control state is not currently available and cannot perform the spindle jog function

Alarm reason: when the spindle control is a switch control, open the spindle jog function, the system alarm;

Processing: Press [Reset] to cancel the alarm, and the spindle jog function cannot be used when the spindle switch is controlled.

A004.2 M03, M04 code specified error

Alarm reason: the spindle is rotated forward or reversed, and the alarm is activated when the spindle is rotated in the opposite direction;

Processing: Press [Reset] to cancel the alarm, the spindle must be in the stopped state to run in the opposite direction.

A004.3 M63, M64 code specified error

Alarm reason: In the 2nd spindle forward or reverse state, the alarm is performed when the spindle rotates in the opposite direction;

Processing: Press [Reset] to cancel the alarm, the spindle must be in the stopped state to run in the opposite direction.

A004.4 Spindle shift time is too long

Alarm reason: When the spindle automatic shift is effective, the shift code M41-M44 is executed, and the code alarm is not completed within the time set by T045;

Processing: Press [Reset] to cancel the alarm and check whether the shift in place signal is installed correctly and whether the switch is damaged.

A008.0 Spindle speed position mode conversion abnormal

Alarm reason: The code alarm was not completed within the T024 setting time after executing the M29 code;

Processing: Press [Reset] to cancel the alarm, check whether the T024/T025 setting time is correct (T025<T024), and check whether the signal setting is correct (K21.0) after the position mode switching is completed.

A008.1 Spindle position speed mode conversion abnormal

Alarm reason: The alarm was not completed within the T028 setting time after executing the M28 code;

Processing: Press [Reset] to cancel the alarm and check whether the T027/T028 setting time is correct (T027<T028).

A008. 2 Insufficient air pressure alarm, level selection K20.2 detection delay T02

Alarm reason: insufficient air pressure alarm signal detected within the T02 setting time;

Processing: check whether the air pressure detection device is normal, and whether the test signal level selection K20.2 setting is correct;

After waiting for the air pressure to be sufficient, press [Reset] to cancel the alarm.

A010.1 The spindle speed is too high and the tool cannot be released

Alarm reason: the spindle stop state is issued, but the spindle has not slowed down to zero, the speed of the big D data D126 set when the loose alarm;

Processing: Press [Reset] to cancel the alarm and check the D126 setting value (D126=20); Or the spindle stops for a period of time and then loosens.

A010.4 Spindle tool clamping detection abnormality

Alarm reason: The clamping signal is not detected in the T020 set time after the tool is released;

Processing: Press [Reset] to cancel the alarm, check whether the T020 setting time is correct, check the installation position of the switch and whether the opening and closing are damaged.

A010.5 Spindle tool release detects abnormalities

Alarm reason: The release signal is not detected in the T09 set time after the tool is released;

Processing: Press [Reset] to cancel the alarm, check whether the T09 setting time is correct, check the installation position of the switch and whether the switch is damaged.

A010.7 Spindle loose knife tightening knife detection abnormality

Alarm reason: At the same time, the loose knife / clip knife is in place signal;

Handling: Press [Reset] to cancel the alarm and check the installation position of the switch and whether the switch is damaged.

A011.0 Indexing Table Locking Loose is not in place

Alarm reason: When the output clamping/loosening signal is output, the release/clamping in place signal is not detected within the T043/T044 time.

Handling: Press [Reset] to cancel the alarm and check the installation position of the switch and whether the switch is damaged.

A011.1 Indexing table lock loosening detects abnormalities

Alarm cause: Spot spot of loosening in place and clamping in place at the same time.

Handling: Press [Reset] to cancel the alarm and check the installation position of the switch and whether the switch is damaged.

A011.2 The indexing table is not released and cannot be moved to the 4th axis

Alarm reason: In the case of automatic indexing work effective K1.7=1, the system does not detect the release of the position signal when the 4th axis is moved;

Processing: Press [Reset] to cancel the alarm, and move the 4th axis in the indexing table

release state.

A011.3 Indexing table is not locked and cannot be moved the XYZ axis

Alarm reason: In the case of automatic indexing effective K1.7=1, K1.4=0, the system does not detect any axis of XYZ when the release of the arrival signal is not detected

Processing: Press [Reset] to cancel the alarm, move the XYZ axis in the indexing table clamping state or change K1.4 to 1.

A012.5 Spindle orientation timing detection anomaly

Alarm reason: spindle orientation in place signal has an input in case of directional alarm;

Processing: Press [Reset] to cancel the alarm and reset the spindle orientation in place signal input level (K21.1).

A012.6 Spindle tool release does not perform spindle positioning

Alarm reason: spindle directional output alarm in the case of tool release;

Processing: Press [Reset] to cancel the alarm, the tool is not oriented when the tool is released.

A012.7 (bucket type, disc type) The tool magazine is not at the origin and spindle positioning cannot be performed

(Turret) The Z-axis is in the tool change interval and spindle positioning cannot be performed

Alarm reason: bucket type tool magazine: the tool magazine is not in the back position, and the alarm is generated when the spindle is positioned;

Disc tool magazine: ATC tool arm is not in the origin position, and the alarm is generated when the spindle is positioned;

Turret type tool magazine: when the Z-axis is in the tool change section, the spindle generates an alarm when it performs positioning;

Processing: Press [Reset] to cancel the alarm, determine that the tool magazine is back to the original position, and then perform the spindle positioning operation.

A013.0 Command Knife Number T is 0

Alarm reason: turret type tool magazine: Execute the T0 code when changing the tool to generate an alarm

Processing: Press [Reset] to cancel the alarm, the turret magazine can not execute the T0 code

A013.1 Spindle orientation in place detection abnormality

Alarm reason: The spindle orientation signal is not detected after the system outputs the spindle orientation after the T023 and T018 setting time;

Processing: Press [Reset] to cancel the alarm, the spindle orientation in place signal X4.2 and +24V on/off effective, check whether the wiring is normal; Check whether the T023/T018 time setting is normal.

A013.2 The spindle tool number is inconsistent with the tool magazine and the tool magazine cannot be advanced

Alarm reason: After opening the tool magazine function, when the current tool number of the spindle is inconsistent with the knife number of the knife magazine cutterhead, the tool magazine is manually controlled in the manual / handwheel mode to advance alarm;

Processing: Press [Reset] to cancel the alarm, the tool magazine forward condition is that the current spindle tool number is equal to the tool magazine tool number, and the tool number position is required for manual rotation of the tool magazine transposition.

A013.3 The Z-axis is not at the **2/3rd reference point, and the magazine cannot be advanced**

Alarm reason: After opening the tool magazine function, when the Z axis is not in the 2nd or 3rd reference point, manually control the tool magazine forward alarm in the manual/handwheel mode;

Processing: Press [Reset] to cancel the alarm, the magazine forward condition is the Z-axis and then the 2nd or 3rd reference point position, and the G91G30 can be executed in the MDI mode Z0 returns to the second reference point.

A013.4 (Bucket Type) The magazine is not **rotating when it is not in the back position and the Z axis is not at** the **3rd** reference point

The (disc type) magazine tool holder is not in the return tool in place, and the tool magazine cannot be rotated

(Turret) The Z axis is not in the permissible rotation zone of the tool magazine, and the tool magazine is not rotatable

Alarm reason: bucket type tool magazine when the Z axis is not in the 3rd reference point or the tool magazine is not in the back position of the position, control the tool magazine rotation alarm;

Disc type tool magazine When the knife sleeve is not in the return tool position, control the tool magazine rotation alarm;

Turret type tool magazine when the Z axis is not in the allowable rotation range of the tool magazine, control the tool magazine rotation alarm;

Processing: Press [Reset] to cancel the alarm.

A013.5 Non-manual/handwheel mode does not allow manual control of the tool magazine

Alarm reason: After opening the tool magazine function, the tool magazine alarm is not manually controlled in the manual / handwheel mode;

Processing: Press [Reset] to cancel the alarm, and the manual control of the tool magazine conditions must be controlled in the manual/handwheel mode.

A013.6 (bucket type, disc type) The tool magazine is not zero **back to zero at the origin** position

The (turret) Z-axis is not in the magazine to allow rotation to be non-zero

Alarm reason: the bucket type knife magazine is not in the back position of the position, and the control tool magazine returns to zero alarm;

When the disc tool magazine knife holder is not in the return position, the control

tool magazine back to zero alarm;

Turret type tool magazine: when the Z axis is not in the allowable rotation range of the tool magazine, the control magazine back to zero alarm;

Processing: Press [Reset] to cancel the alarm, manually control the tool magazine back to zero condition when the back to zero mode tool magazine back to the origin position.

A013.7 Tool change first execute the T command tool number

Alarm reason: after opening the tool magazine function, the T code is not specified before executing the tool change instruction M6 alarm;

Processing: Press [Reset] to cancel the alarm, and specify the T code before executing the tool change instruction M6.

A014.0 The Z-axis does not return to the mechanical zero point and cannot be changed

Alarm reason: After opening the tool magazine function, the tool change is performed when the Z axis does not return to zero;

Processing: Press [Reset] to cancel the alarm, and specify the T code before executing the tool change instruction M6.

A014.1 (Bucket Type) The cutterhead is not in the counting position and cannot be entered

The cutterhead (disc type) is not in the counting position and cannot be reversed

The cutterhead (turret) is not in the counting position and the Z axis is not movable

Alarm reason: when the bucket type knife magazine cutterhead is not in the counting position, the control tool magazine forward alarm;

When the disc type cutterhead is not in the counting position, the control knife sleeve inverted knife alarm;

When the turret cutterhead is not in the counting position, the Z-axis movement alarm is controlled;

Processing: Press [Reset] to cancel the alarm, manually control the cutterhead of the tool magazine must be in the counting position.

A014.2 (bucket type, disc type) The tool magazine is not in **the origin position and the spindle is not rotatable**

The (turret type) Z-axis is in the tool change section, and the spindle cannot be rotated

Alarm reason: when the bucket type tool magazine is not in the back position, the starting spindle generates an alarm;

When the disc type tool magazine ATC tool arm is not at the origin, the starting spindle generates an alarm;

Turret tool magazine: when the Z axis is in the tool change area, the spindle is activated to generate an alarm;

Processing: Press [Reset] to cancel the alarm, manually control the tool magazine back to the origin position.

A014.3 The spindle tool is loosened and the tool change cannot be performed

Alarm reason: After opening the tool magazine function, the tool is released to perform the tool change alarm;

Processing: Press [Reset] to cancel the alarm, and change the tool after clamping.

A014.4 The current cutterhead has a knife and cannot be entered

Alarm reason: after opening the tool magazine function, the knife magazine advances when the current cutterhead has a knife;

Processing: Press [Reset] to cancel the alarm, the tool magazine returns to zero or reset the spindle tool number D105.

A014.5 The current cutterhead has a knife number and spindle and cannot be entered

Alarm reason: After opening the tool magazine function, the current cutterhead and spindle are detected when the knife is controlled when the tool is alarmed;

Processing: Press [Reset] to cancel the alarm and remove the knife on the spindle.

A014.6 The current spindle tool number is zero, the spindle has a knife, and the tool cannot be changed

Alarm reason: after opening the tool magazine function, the spindle detection knife has a knife when the current spindle tool number is 0;

Processing: Press [Reset] to cancel the alarm and remove the knife on the spindle.

A014.7 The spindle has a knife, the Z axis is not in the loading position, and the knife cannot be entered

Alarm reason: After opening the tool magazine function, the current spindle is detected with a knife, and the Z axis is not at the 2nd reference point, the control tool magazine tool entry alarm;

Processing: Press [Reset] to cancel the alarm and move the Z axis to the 2nd reference point.

A015.0 The tool magazine number counting signal detects abnormalities, and the tool magazine needs to return to zero

Alarm reason: after opening the tool magazine function, the alarm when the tool magazine count switch is not detected after the T102 setting time after the tool magazine rotation stops;

Processing: Press [Reset] to cancel the alarm and check whether the T102 setting time is correct (T102=2000); Check the magazine counter for damage; The magazine counter level is selected correctly (K30.5).

A015.1 The magazine origin is lost and the magazine needs to be returned to zero

Alarm reason: After opening the tool magazine function, there is a tool magazine back to zero switch in the case: the cutterhead count is 1 when the tool magazine back to zero switch is not detected or the cutlehead count is not 1 when the tool magazine back zero switch is detected;

Processing: Press [reset key] in the zero return mode to cancel the alarm, and the tool

magazine returns to zero.

A015.2 Tool magazine zero detection anomaly

Alarm reason: After opening the tool magazine function, the tool magazine returns to zero for more than 5 minutes without returning to zero to complete the alarm;

Processing: Press [Reset Key] to cancel the alarm, and the tool magazine returns to zero.

A015.3 The zero position of the tool magazine was set successfully

Alarm reason: After opening the tool magazine function, the tool magazine zero return to zero is successful or no zero switch in the case of holding down the [machine tool zero point key] 7 seconds after the alarm;

Processing: Press [Reset key] to cancel the alarm, the magazine count has been redesigned.

A015.4 Abnormal stop of tool change ensures that the tool magazine is normal

Alarm reason: after opening the tool magazine function, when there is an emergency stop, reset and alarm in the process of tool change, this alarm is issued;

Processing: Press [reset key] to cancel the alarm and check whether the tool magazine is normal

A015.5 No command tool number or duplicate command tool number in the tool table

Alarm reason: After opening the tool magazine function, the tool magazine search tool in the T command exceeds the tool magazine capacity or there are duplicate values in D0-D99 in the tool magazine data sheet;

Processing: press [reset key] to cancel the alarm, command the correct T command, or (when returning the knife) modify the current tool number D105 of the spindle; Check the magazine data sheets D0-D99 for errors.

A015.6 The tool magazine is in the forward state, and the spindle tool cannot be moved without releasing the Z-axis

Alarm reason: after opening the tool magazine function, when the tool magazine is in the forward state, when the tool is not released, the Z axis alarm is moved;

Processing: Press [Reset key] to cancel the alarm, the tool magazine must be released to move the Z axis when the tool magazine is in the forward state.

A015.7 (Bucket Type) The spindle is not positioned and the magazine cannot be advanced

The (turret type) spindle is not positioned, and the **Z-axis cannot enter the tool change area**

Alarm reason: (bucket type) after opening the tool magazine function, when the spindle has a positioning function, the unpositioned control tool magazine forward alarm;

(Turret) After opening the tool magazine function, when the spindle has a positioning function, the Z axis is not positioned to control the tool change area alarm;

Processing: Press [reset key] to cancel the alarm, so that the spindle positioning, and then control the tool magazine forward, the spindle does not have the positioning function to change K21.1 to 1.

A016.0 Tool magazine forward/inverted tool in place detects abnormalities

Alarm reason: After opening the tool magazine function, the forward / backward knife of the tool magazine is not detected after the T104 setting time;

Processing: Press [reset key] to cancel the alarm, check whether the T104 setting is correct (T104=3000), check whether the tool magazine is normal, and manually control the tool magazine rewind/retract.

A016.1 Tool magazine back/return tool in place detection abnormality

Alarm reason: After opening the tool magazine function, the tool magazine back / back tool is not detected after the T105 setting time is not detected back into place;

Processing: Press [reset key] to cancel the alarm, check whether the T105 setting is correct (T105=5000), check whether the tool magazine is normal, and manually control the tool magazine rewind/retract.

A016.2 Tool magazine forward and backward / **backward knife / return tool detection abnormality**

Alarm reason: After opening the tool magazine function, the tool magazine forward/reverse knife in place and the back / back knife in place signals are sensed at the same time;

Processing: Press [reset key] to cancel the alarm, check the installation position of the tool magazine forward and backward/reverse knife return knife in place switch, check the forward and backward/reverse knife return switch level selection (K30.3/K30.4).

A016.3 (disc type) The tool is not clamped and the arm cannot be returned to the point of origin

Alarm reason: when the tool is not clamped, control the ATC knife arm back to the original point;

Processing: Press [reset key] to cancel the alarm, the tool clamping can be performed to return the arm to the original point;

A016.4 The spindle tool cannot be reversed without clamping the tool magazine

Alarm reason: after opening the tool magazine function, the tool is not clamped to control the tool magazine back alarm;

Processing: Press [reset key] to cancel the alarm, and the tool can be clamped before the tool can be controlled to retreat.

A016.5 Abnormal magazine rotation forces the cessation of rotation

Alarm reason: After opening the tool magazine function, the tool magazine rotates after T110, T111, T103 time does not check the tool magazine counter signal change alarm;

Processing: press [reset key] to cancel the alarm, check whether there is rotation, if there is rotation, check whether T110, T111, T103 and the magazine counter signal is normal (T110=5000; T111=3000; T103=3000）; If there is no rotation, check for problems with the magazine circuit.

A016.6 (bucket type, disc type) The tool magazine is not in the origin position and cannot be started

The (turret type) Z axis is in the tool change interval and cannot be started

Alarm reason: the bucket type knife magazine is not in the back position, start the program alarm;

When the disc type tool magazine ATC tool arm is not at the origin, start the program alarm;

Turret tool magazine: when the Z axis is in the tool change area, the program alarm is started;

Processing: Press [Reset Key] to cancel the alarm, manually control the tool magazine back to the original point.

A016.7 The tool magazine cannot be zeroed out without mechanical zeroing

Alarm reason: After opening the tool magazine function, the tool magazine has a zero return switch and does not execute the tool magazine zero alarm in the mechanical zero return mode;

Processing: Press [reset key] to cancel the alarm, and execute the tool magazine to zero in the mechanical zero return mode.

A017.0 The tool magazine is not **at the origin and the Z axis is not movable**

Alarm reason: when the bucket barrel is not in the back position, move the Z axis;

Move the Z axis when the disc magazine arm is not in the origin position;

Processing: Press [Reset Key] to cancel the alarm, manually control the tool magazine back to the origin position.

A017.1 (Disc type) The tool arm is not at the origin and cannot be reversed/returned

Alarm reason: when the knife arm is not in the origin position, control the tool magazine inverted knife / return knife;

Processing: Press [reset key] to cancel the alarm, and return the knife arm to the origin position of the knife when manual.

A017.2 (Disc type) The tool magazine inverted knife is not in place, and the knife arm cannot be moved

Alarm reason: when the knife sleeve is not in the inverted knife position, control the ATC knife arm action;

Processing: Press [reset key] to cancel the alarm, and the knife sleeve is in place when manual.

A017.3 (Disc type) The spindle is not positioned and the knife arm cannot be moved

Alarm reason: when the spindle is not positioned, control the ATC knife arm action;

Processing: Press [Reset Key] to cancel the alarm, and the knife arm action can be performed after the spindle positioning is completed.

A017.4 (Disc type) The Z-axis is not at the 2nd reference point, and the knife arm cannot be moved

Alarm reason: When the Z axis is not at the 2nd reference point, control the ATC knife arm action;

Processing: Press [Reset Key] to cancel the alarm, and the Z axis must be at the 2nd reference point to perform the knife arm action.

A017.5 (disc type) The tool is not clamped and the knife arm cannot be snapped

Alarm reason: when the tool is not clamped, control the ATC knife arm clip knife;

Processing: Press [reset key] to cancel the alarm, the tool clamping can perform the knife arm clip;

A017.6 (Disc type) The tool is not loosened and **the arm cannot be pulled down and rotated**

Alarm reason: when the tool is not released, control the ATC arm pull-down rotation;

Processing: Press [reset key] to cancel the alarm, the tool is released to perform the knife arm action.

A017.7 (Disc) Tool arm running time is too long

Alarm reason: knife arm running time exceeds T115 set time alarm;

Processing: Press [reset key] to cancel the alarm, check whether the T115 data setting is reasonable, and check whether the knife arm brake signal is valid.

A018.0 (disc type) knife arm clipper is not in place

Alarm reason: When performing the knife action, the knife in place signal alarm is not detected after the T112 setting time

Processing: Press [reset key] to cancel the alarm, check whether the T112 data setting is reasonable, and check whether the knife clip knife in place signal is valid.

A018.1 (Disc type) Arm pull-down rotation does not rise in place

Alarm reason: When performing the pull-down rotation action, the button knife in place signal alarm is not detected after the T113 setting time

Processing: Press [reset key] to cancel the alarm, check whether the T113 data setting is reasonable, and check whether the knife in place signal is valid.

A018.2 (Disc type) The tool arm return point is not in place

Alarm reason: When performing the action back to the origin, the origin signal alarm is not detected after the T114 setting time

Processing: Press [reset key] to cancel the alarm, check whether the T114 data setting is reasonable, and check whether the origin in place signal is valid.

A018.3 (Disc) Tool arm operation abnormal

Alarm reason: when performing the knife arm action, the system issues an alarm when resetting, alarming, and emergency stopping;

Processing: Press the [reset key] to cancel the alarm.

A020.0 Guard door opened warning

Alarm reason: After opening the protective door detection function, a warning is issued when

the protective door signal is not detected;
Processing: Press [Reset Key] to cancel the alarm when closing the protective door, and the program cannot be started when the warning is triggered.

A020.1 Warning of insufficient lubricating oil

Alarm reason: Warning after delaying T03 set time in case of insufficient lubricating oil is detected;
Handling: Add lubricating oil, press [Reset key] to cancel the warning; Check that the lubricating oil detection level is selected correctly (K16.2).

A020.2 Warning of insufficient air pressure

Alarm reason: Warning is issued after delaying the T02 set time in the case of insufficient air pressure;
Treatment: check whether the intake element is normal, and cancel the warning after waiting for the air pressure to flush out; Check that the low air pressure detection level is selected correctly (K20.2) and warn that the program will not start when triggered.

A020.3 Zero warning for spindle tool number

Alarm reason: not warning when the current tool number of the spindle is 0 during the tool change process;
Processing: Check for a knife on the spindle, remove the tool to re-perform the tool change or modify the current tool size value D105 for the spindle.

A020.4 Tool magazine commissioning function opens warning

Alarm reason: tool magazine debugging function to open K30.7=1 trigger warning;
Processing: After confirming that the tool magazine is normal, turn off the debugging function to cancel the warning, and the warning cannot start when triggered.

A020.5 Warning in the tool magazine back to zero

Alarm reason: trigger warning when the tool magazine magazine returns to zero;
Processing: The warning is canceled after the end of the magazine back to zero, and the warning cannot be started when triggered.

A020.6 (Bucket Type) magazine warning that it is not in the back position
(Disc) knife arm is not at the origin warning
(Turret) Z-axis warning in the tool change zone

Alarm reason: (bucket type) is not in the process of changing the tool, the tool magazine is not in the back position to trigger the warning;
(Disc type) is not in the process of tool change, and the knife arm is not at the origin position to trigger the warning;
(Turret) is not during the tool change, the Z axis triggers a warning in the tool change zone
Processing: Controls the tool magazine to fall back to the back position of the tool magazine to cancel the warning, warning that the program cannot be started when triggered.

Appendix III Macro Program Supplemental

Instructions (2021 versions above).

1. Statement macro code (macro B).

1.1 Arithmetic and logical operations

Arithmetic and logical operations

function	Expression structure	remark
Define or assign a value	#i=#j	
addition subtraction multiplication division	#i=#j+#k #i=#j-#k #i=#j*#k #i=#j/#k	
or and XOR	#i=#j OR #K #i=#j AND #K #i=#j XOR #K	Logical operations are performed bit by bit as binary numbers
square root absolute value rounding Up rounded Rounded down Napierian logarithm Exponential function	#i=SQRT[#j] #i=ABS[#j] #i=ROUND[#j] #i=FUP[#j] #i=FIX[#j] #i=LN[#j] #i=EXP[#j]	
sine arcsine cosine inverse cosine tangent Cut anyway	#i=SIN[#j] #i=ASIN[#j] #i=COS[#j] #i=ACOS[#j] #i=TAN[#j] #i=ATAN[#j]/[#k]	The units of the angle are specified in degrees, e.g. 90°30' is expressed as 90.5 degrees
Switch from BCD to BIN Switch from BIN to BCD	#i=BIN[#j] #i=BCD[#j]	Used for signal conversion with PMC

Related instructions

1、Angular units

The angular units of the functions SIN, COS, ASIN, ACOS, TAN, and ATAN are degrees (°). If 90 ° 30 ′ should be expressed as 90.5 ° (degrees).

2、Arcsine #i=ASIN[#j]

i、The output range of the result is as follows,

When the bit parameter 25.7 NAT bit is set to 1, 90° ~ 270°;

When the bit parameter 25.7 NAT bit is set to 0, -90°~90°;

ii、When #j is out of the range of -1 to 1, a P/S alarm is issued.

iii、The constant replaces the variable #j.

3、Inverse cosine #i=ACOS[#j]

i、The output range is from 180° to 0°;

ii、A P/S alarm is issued when #j exceeds the range of -1 to 1;

iii、Constants can replace the variable #j.

4、Anyway cut #i=ATAN[#j]/[#k]

Specify the length of the two sides, separated by the slash '/'.

i、The value range is as follows,

When the position parameter 25.7 NAT bit is set to 1: 90°~270°;

[Example] #1=ATAN[-1]/[-1], #1=225° when specifying;

When position parameter 25 7 NAT bit set to 0 -90°~90°;

[Example] #1=ATAN[-1]/[-1] when #1=45.0° is specified;

ii、Constants can replace the variable #j.

5、Natural logarithm #i=LN[#j]

i. Constants can replace the variable #j.

6、Exponential function #i=EXP[#j]

i、Constants can replace the variable #j.

7、ROUND rounding function

When the arithmetic or logical operation code IF or WHILE contains the ROUND function, the ROUND function is rounded to the 1st decimal position.

For example:

When #1=ROUND[#2] is executed, at which point #2=1.2345 the value of variable 1 is 1.0.

8、Up rounded and bottomed rounded

When CNC processes numerical operations, if the absolute value of the integer generated after the operation is greater than the absolute value of the original number, it is called rounding up; If it is less than the absolute value of the original number, it is called rounding down. Handling of negative numbers should be handled with care.

For example:

Suppose #1=1.2 and #2=-1.2

When #3=FUP[#1] is executed, 2.0 is assigned to #3.

When #3=FIX[#1] is executed, 1.0 is assigned to #3.

When #3=FUP[#2] is executed, -2.0 is assigned to #3.

When #3=FIX[#2] is executed, -1.0 is assigned to #3.

1.2 Transfer and circulation

In the program, the flow direction of the control can be changed using GOTO statements and IF statements. There are three transfer and loop operations available.

1、GOTO statement (unconditional transfer).

2、The condition controls the IF statement.

3、WHILE loop statement.

1）Unconditional transfer (GOTO statement)

Transfer the program segment to the sequence number n. When an alarm is specified with a sequence number other than 1 to 99999, a sequence number can be specified using an expression.

Format: GOTOn; n: Sequence number (1~99999).

Example: GOTO1;

GOTO#101;

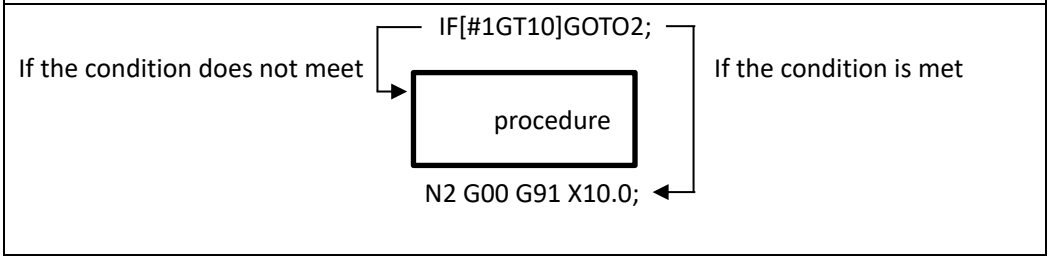
2） Conditional control (IF statement)

GOTO format: IF[conditional expression] GOTOn;

If the specified conditional expression is true, transfer the sequence number to the program segment of bit n; If the specified conditional expression is not true, the next program segment is executed sequentially.

Example:

If the value of variable #1 is greater than 10, transfer to the program segment with sequence number N2.



THEN format: IF[conditional expression]THEN< macro program statement >;

If the conditional expression is true, only one statement can be executed after THEN.

Example: IF[#1EQ#2]THEN#3=0;

If the value of #1 is equal to the value of #2, 0 is assigned to variable #3; if it is not equal, the assignment statement after THEN is executed sequentially.

Conditional expressions: Conditional expressions must include conditional operators, which can be flanked by variables, constants, or expressions, and conditional expressions enclosed in parentheses '['']'.

Conditional operators: The system can use the conditional operators listed in the following table.

Conditional operators	meaning
EQ or ==	Equals (=)
NE or <>	Not equal to (≠).
GT or >	Greater than (**>**).
GE or >=	Greater than or equal to (**≥**)
LT or <	Less than (**<**).
LE or <=	Less than or equal to (**≤**).

For example: IF[3<>2]GOTO2; The meaning is: if 3 is not equal to 2, then jump to the N2 program segment;

IF[#101>=7.22]THEN#101=SIN30； This means that if #101 is greater than or equal to 7.22, the assignment operation after THEN is performed. That is, assign a sine value of 30 degrees to variable #101.

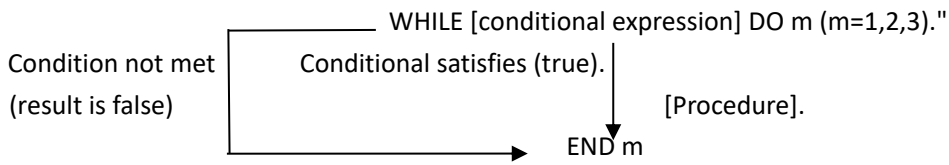
Typical program: The following program calculates the sum of integers 1 to 10.

```
O9500
#1=0； ..... and initialize to 0
#2=1； ..... The initial value of the addition is 1
N1 IF[#2 GT 10]GOTO2； ..... Transfer to N2 when the addition is greater than 10
#1=#1+#2； ..... Calculates the sum of two numbers
#2=#2+1； ..... The number is added by 1
GOTO1； ..... Unconditionally jump to program segment N1
N2 M30； ..... The program ends
```

3）**Loop (WHILE statement)**

Specify a conditional expression after WHILE, and when the specified condition is true, execute the program segment from DO to END; Otherwise, jump to the program segment after END.

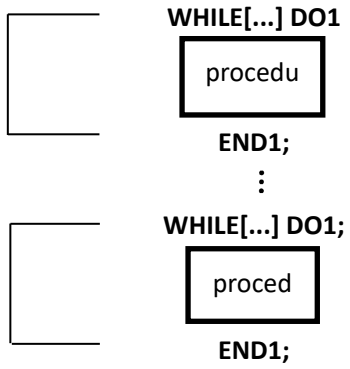
Example:



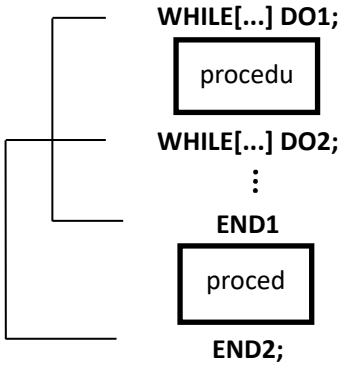
Description: When the specified condition is true, the program segment between DO and END is executed; Otherwise, the program segment after END is executed instead. The label after DO and the label after END should be consistent, and the label value can be 1, 2 or 3.

Nesting: DO, the designator (1 to 3) in the END loop can be used as many times as needed. However, an alarm is raised when there is a cross-repetitive cycle in the program.

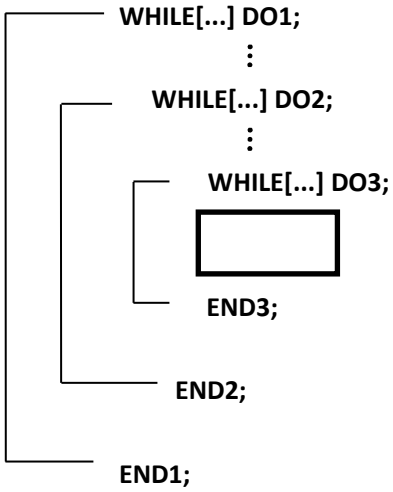
1. Labels (1 to 3) can be used as many times as required



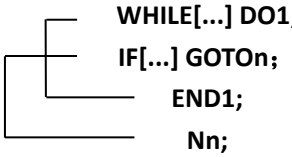
2. **The range of DO cannot be crossed**



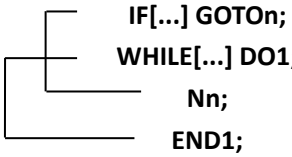
3. **The DO loop can be nested 3 layers**



4. **Control can go outside the loop**



5. **The transfer cannot enter the loop**



1.3 Macro alarm

Format: ERR_;
Range 0 to 200;
Alarm number 3000+n;
For example: E RR01 or E RR1;
The system alarm number is 3001.

1.4 Special instructions (substituting variable numbers with variables).

Format: "#"+ "9" + displacement variable number
Examples: #100=205, #205=500;
X# 9100 and X500 code functions the same;
X-#9100 and X-500 code functions the same;

It can be understood as #9_____ ， 500～999；
#9100 represents the macro variable number represented by the value within #100;
For example, #100=510, #510=101;

Write #9100, #9100=#510=101;

Appendix 4: Custom alarm content writing (2021 version above).

Bit parameter 32.3 [Read macro program alarm/comment (0:no 1:yes) valid].

Macro A alarm

G65H99P_;//[ERR_, the system displays alarm content] (within 25 words).

Range 0-200;

Macro B alarm

ERR_;//[ERR_, the system displays alarm content] (within 25 words).

Range 0-200;

For example:

ERR1;//[ERR1, quantity arrived];

The system displays macro program alarm 3001 number arrived;

Note: At the same time, the value of the system macro variable #3000 can be written for alarm, the range is also 0-200, and the alarm number is also consistent 3000-3200; The writing method is G65 H01 P#3000 Q_, or #3000= _;

Example: G65 H01 P#3000 Q0;

#3000=0;

G65H99P0;

ERR0; These four compilation methods have the same effect.

Modifying the system can read the program alarm comments, and then display the alarm content and report out in the system file WARN2;

Program macro variable comments can be read and saved in the system file MacroVar;

Specify the writable alarm range;

0650-0657； M89K0-K7 detection

3000-3200； Macro program alarm P0-P200

;

Write M89 K0-K7 alarm specification format:

M89 K0-K7 J_ ;//[ERR650-657,M89K0-K7 detection alarm];

M89 K0-K7 detection alarm; Alarm display content, while recorded in the file WARN, and then there is the same alarm number when there is no written alarm content, directly read the stored alarm internal display.

For example:

M89 K3 J5000;//[ERR653, M89 K3 ejection not in place].

or

[ERR653, M89 K3 top material not in place].

M89 K3 J5000;

As long as the program reads [ERR653, M89K3 ejection is not in place], the alarm content is saved in the system file 653 alarm;

When the next time the alarm display is not written, the same code is commanded, and the same alarm will display the content written before;

For example:

M89 K3 J5000;//

At this time, the alarm will still show that the M89 K3 top material is not in place;

Write a macro program alarm specification format

Macro A:

G65H99P_;//[ERR_, alarm display content];

Macro B:

E RR_;//[ERR_, alarm display content];

Alarm display content, recorded in file WARN2 at the same time, and then there is the same alarm number when there is no written alarm content, directly read the stored alarm internal display.

For example:

G65 H99 P0;//[ERR0, number of processes reached].

or

[ERR0, the number of processes reached].

G65 H99 P0;

As long as the program reads [ERR0, the number of processes reached], the alarm content is saved in the system file 3000 alarm;

When the next time the alarm display is not written, the same code is commanded, and the same alarm will display the content written before;

For example:

G65 H99 P0;

or

ERR0;

At this point the alarm will still show the macro alarm 3000 processing quantity arrived;

The program reads macro variable comments saved in the system file MacroVar:

Adjustable Notes:

#100-#199；

#500-#899； (#900-#999 reserved for internal use of the system);

Modify the specification format of comments;

//[#--, display content] (within 20 words).

//[#--, display content].

The display content will be saved in **MacroVar and can be viewed in public variables;**

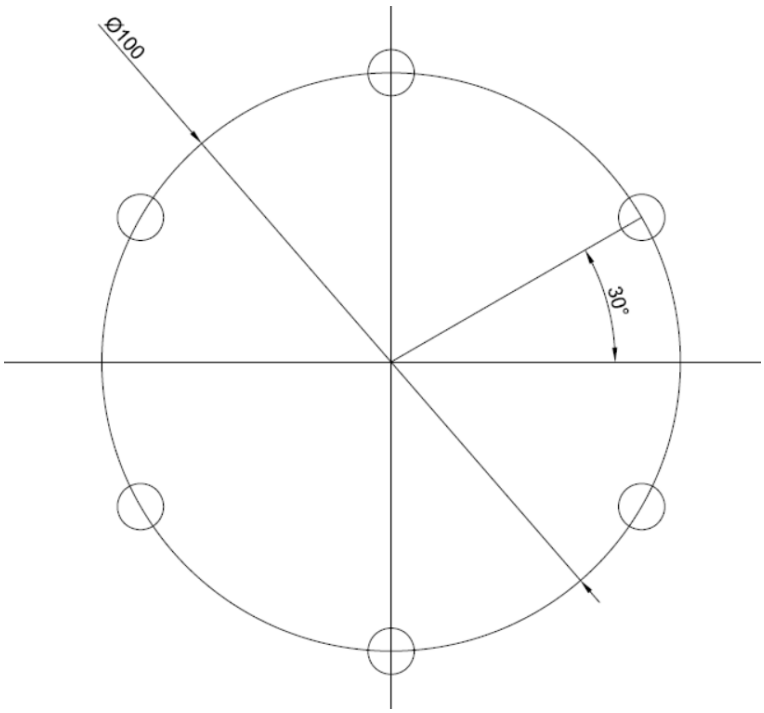
Macro variable comments are saved without changing the same macro variable comment.

Appendix V: Circumferential Drilling/Tapping Instructions (2021 and above versions have it).

Circumferential Hole Command: Drill/Spot Drill Cycle G181
(G 81 is called, and the drilling action is consistent with G81).
Instruction format: G181 X_ Y_ Z_ R_ I_ J_ K_ F_;
Illustrate:
X/Y: The center coordinate value
Z: Machining depth
R: R point plane (fast entry point).
I: Circumferential diameter
J: Starting angle
K: Aliquot
F: Feed speed

Circumferential Hole Command: Chip Evacuation Drilling Cycle G183
(Call G 83, drilling action consistent with G83).
Instruction format: G183 X_ Y_ Z_ R_ I_ J_ K_ Q_ F_;
Illustrate:
X/Y: The center coordinate value
Z: Machining depth
R: R point plane (fast entry point).
I: Circumferential diameter
J: Starting angle
K: Aliquot
Q: The amount of knife per feed
F: Feed speed

Circumferential Command: Tapping cycle G184/G174
(Call G 84, tapping action is consistent with G 84).
Instruction format: G184/G174 X_ Y_ Z_ R_ I_ J_ K_ Q_ P_ F_;
Illustrate:
X/Y: The center coordinate value
Z: Machining depth
R: R point plane (fast entry point).
I: Circumferential diameter
J: Starting angle
K: Aliquot
Q: The amount of knife per feed
P: The pause time at the bottom of the hole
F: Pitch/feed speed



Example: G181 X0 Y0 Z-10 R5 I100 J30 K6 F200;

Appendix VI: Axes 4/5 follow the special instructions for the Z axis

Parameter: bit parameter 42.1 G200-G205 follow the Z-axis function is valid (0: no, 1: yes).

Instruction format: G200 The 4th axis follows the Z axis

G201 Cancel the 4th axis following the Z-axis

G202 The 5th axis follows the Z axis

G203 Eliminate the 5th axis from following the Z-axis

G204 The 4th and 5th axes follow the Z axis

G205 Cancel the 4 and 5 axes following the Z axis

Note 1: Cancel the following during alarm, reset, or emergency stop, and require a separate line of instruction to follow and cancel the follow-up at the same time;

Note 2: The number of pulses emitted by the follow axis is determined by the Z axis, so when using the follow function, the follow axis gear ratio needs to be consistent with the Z-axis gear ratio;

Examples of use:

G54G90G17G80G40;

G0X0Y0Z20A20; At the same time at the absolute position of 2 0 begins to follow

G200;// Turn on the follow function
G81Z-20R5F120;//4th axis follows the Z axis to achieve 4th axis punching
X10Y10;
G80;
G201;// Cancel follow function

Appendix 7: Automatic screw clamping function

M72/M73 during drilling

Parameters: bit parameter 1 4.5 G73/G81/G82/G83/G85/G86/G88/G89 drilling screw automatic clamping (0: no 1: yes) effective

Automatic output clamping Y1.5(M72/M73);
The clamping control parameter T46 M72 outputs the clamping delay (default 200).
T47 M72/M73 code completion delay (default 200).

Working principle: When the command drilling command [G73/G81/G82/G83/G85/G86/G88/G89], the X Y axis moving positioning fast first output M 73 release the clamping (turn off Y1.5), wait for M 73 complete, then move X Y positioning, after the positioning is completed, output M 72 delay output clamping Y1.5, wait for M72 to finish, and then command the Z axis to move the hole, When multiple holes, repeat the above action.

Note: After the drilling is completed, the instruction M73 is required to lift the lead screw clamping;

Example:
G54G90G17G80G40;
G0X0Y0Z20; //
G73X10Y10Z-20R5F120Q5;//
G80; // Lift lead screw clamping

Note 1: At the end of the G73/G81/G82/G83/G85/G86/G88/G89 instructions, output M73 turns on the clamp holder.
Note 2: There can be no M code in the G80 segment of the end loop code, otherwise the alarm address is duplicated.
Note 3: Commanding the cutting code [such as G01/G02/G03 and other cutting instructions] in the loop code will cancel the loop code, so there can be no M code in the first cutting instruction section, otherwise the alarm address is repeated.
Note 4: If the cycle directive [G73/G81/G82/G83/G85/G86/G88/G89] is not used in the machining or the cutting instruction [G01/G02/G03] is cancelled, then run the other instructions [G76/G87/G74/G84/G22/G23]. etc. command] when the lead screw clamping function will not be released.

Appendix VIII: Rotation axis control mode (only available in version 2 021 or above).

1、 To set the rotation axis parameters:

Bit parameter 33.0 sets the X-axis type (0: linear axis 1: axis of rotation).

Bit parameter 33 1 Set the Y-axis type (0: linear axis 1: rotation axis).

Bit parameter 33.2 sets the Z-axis type (0: linear axis 1: axis of rotation).

Bit parameter 33.3 sets the 4TH axis type (0: linear axis 1: rotary axis).

Bit parameter 33.4 sets the 5TH axis type (0: linear axis 1: axis of rotation).

2、 Rotation axis type:

The system divides the rotation axis settings into 2 categories, 1) nearby rotation, 2) rotation by symbol;

Position parameter 34.2 When the X axis is the axis of rotation (0: Rotate 1: Rotate nearby in the direction of the symbol).

The bit parameter 34.3 When the Y axis is the axis of rotation (0: rotate 1 in the direction of the symbol: rotate nearby).

Bit parameter 35.3 When the Z axis is the axis of rotation (0: Rotate 1: Rotate Nearby in the Symbol Direction).

Bit parameter 35 1 When the 4TH axis is the axis of rotation (0: rotate 1: rotate in the direction of the symbol).

Bit parameter 37 1 5TH axis is the axis of rotation (0: rotate in the direction of the symbol 1: rotate nearby).

1) Nearest rotation control:

The nearest rotation control mode machine coordinates and absolute coordinates are cycled between 0-360 degrees, the relative coordinate coordinates can be set to cycle or partial loop, the shortest path control is performed during absolute programming, and the incremental movement is performed during incremental programming (the shortest distance is not performed).

N1 G90 G0 C0	The sequence number	The actual position amount	The absolute coordinate value of the end point	Machine coordinates
N2 G0 C300	N2	-60, negative movement	300	300
N3 C100	N3	-160, negative movement	100	100
N4 C360	N4	-100, negative movement	0	0
N5 C-30	N5	-30, negative movement	330	330

N6 C0	N6	30, forward movement	0	0
N7 C360	N7	0, do not move	0	0
N8 C720	N8	0, do not move	0	0
N9 C0	N9	0, do not move	0	0
N10 C360	N10	0, do not move	0	0
N11 C360	N11	0, do not move	0	0
N12 G91 C540	N12	540, forward movement	180	180
N13 G91 C360	N13	360, forward movement	180	180
N14 G91 C-200	N14	-200, negative movement	340	340

2) Select controls by symbol direction:

According to the symbol direction rotation control mode machine tool coordinates and absolute coordinates are cycled between 0-360 degrees, the relative coordinate coordinates can be set cyclic or partial cycle, according to the symbol rotation can be divided into two types of rotation axis A/B: (Note: A/B two types are invalid under the nearest rotation control).

Bit parameter 35.4 Sets the X axis to the type when rotating (0: axis of rotation A type 1: axis of rotation type B)

Bit parameter 35.5 sets the Y axis to the type when rotating (0: axis of rotation A type 1: axis of rotation type B)

Bit parameter 35.6 sets the Z axis to the type when rotating (0: axis of rotation A type 1: axis of rotation type B)

Position parameter 34.1 sets the 4TH axis to be the type when rotating (0: rotation axis A type 1: rotation axis B type)

Position parameter 36.1 sets the 5TH axis to be the type when rotating (0: rotation axis A type 1: rotation axis B type)

Axis of rotation type A: Execution is performed in the direction of the symbol when absolutely programmed, and the operation mode is similar to the linear axis, moving from C0 to C 300 in the positive direction, C300 to C200 when the negative direction moves, incremental programming to perform incremental movement (not to perform the shortest distance).

N1 G90 G0 C0	The sequence number	The actual position amount	The absolute coordinate value of the end point	Machine coordinates
N2 G0 C300	N2	300, forward movement	300	300
N3 C100	N3	-200, forward movement	100	100
N4 C360	N4	260, forward movement	0	0
N5 C-30	N5	-30, negative	330	330

		movement		
N6 C0	N6	-330, negative movement	0	0
N7 C360	N7	360, forward movement	0	0
N8 C720	N8	720, forward movement	0	0
N9 C0	N9	0, do not move	0	0
N10 C360	N10	360, forward movement	0	0
N11 C360	N11	360, forward movement	0	0
N12 G91 C540	N12	540, forward movement	180	180
N13 G91 C360	N13	360, forward movement	180	180
N14 G91 C-200	N14	-200, negative movement	340	340

Note: In type A/B, when the coordinates go to a multiple of 3 60 or 3 60, the next line is followed by 3 60 or 3 60 when the multiplier is valued, the coordinates are calculated starting from 0; For example the above program C 0-C360 (at this time walked a circle of 360 degrees) - C720 (at this time from 0 The segment started counting and went 2 laps).

Axis of rotation type B: Execution is performed in the direction **of** the symbol when absolutely programmed, and the mode of operation is always moving in the positive direction (moving in the negative direction when there is a negative symbol), moving in the positive direction from C0 to C 300, C300 Positive movement to C 200 [300-360(0)-200], incremental movement is performed during incremental programming (the shortest distance is not performed).

N1 G90 G0 C0	The sequence number	The actual position amount	The absolute coordinate value of the end point	Machine coordinates
N2 G0 C300	N2	300, forward movement	300	300
N3 C100	N3	160, forward movement	100	100
N4 C360	N4	260, forward moving	0	0
N5 C-30	N5	-30, negative movement	330	330
N6 C0	N6	60, forward movement	0	0
N7 C360	N7	360, forward movement	0	0
N8 C720	N8	720, forward movement	0	0

N9 C0	N9	0, negative movement	0	0
N10 C360	N10	360, forward movement	0	0
N11 C360	N11	360, forward movement	0	0
N12 G91 C540	N12	540, forward movement	180	180
N13 G91 C360	N13	360, forward movement	180	180
N14 G91 C-200	N14	-200, negative movement	340	340

Note: In type A/B, when the coordinates go to a multiple of 3 60 or 3 60, the next line is followed by 3 60 or 3 60 when the multiplier is valued, the coordinates are calculated starting from 0; For example the above program C 0-C360 (at this time walked a circle of 360 degrees) - C720 (at this time from 0 The segment started counting and went 2 laps).

1. Rotate the negative angle of type A/B by **symbol**

Bit parameter 37.6 Rotate the negative angle instruction type 0 by symbol (00:A-type 01:B-10:C-11:D type).

Bit parameter 37.7 Rotate the negative angle instruction type 1 by symbol (00:A-type 01:B-type 10:C-type 11:D type).

1. Negative angle editing does not exceed -360 cases
2. Negative angle edit more than -360 cases

1) Negative angle A type 37.7=0**, 37.6=0**

When it does not exceed -360, the shortest distance in the negative direction;
Above -360, take the negative (linear axis) distance.

N1 G0 C0	The sequence number	The actual amount of movement	Absolute coordinates	Machine seat coordinates
N2 C-260	N2	-260	100	100
N3 C-260	N3	0	100	100
N4 C-360	N4	-460 (-100-360)	0	0
N5 C-540	N5	-540 (-360-180)	180	180
N6 C-540	N6	-720 (-180-360-180)	180	180
N7 C-260	N7	-80	100	100
N8 C-720	N8	-820 (-100-360-360)	0	0

2) Negative angle B type 37.7=**0, 37.6=1**

When exceeding or not exceeding -360, take the negative (linear axis) distance.

N1 G0 C0	The sequence	The actual amount of movement	Absolute coordinates	Machine seat coordinates

	number			
N2 C-260	N2	-260	100	100
N3 C-260	N3	-360 (-100-260)	100	100
N4 C-360	N4	-460 (-100-360)	0	0
N5 C-540	N5	-540 (-360-180)	180	180
N6 C-540	N6	-720 (-180-360-180)	180	180
N7 C-260	N7	-440 (-180-260)	100	100
N8 C-720	N8	-820(-100-360-360)	0	0

3) Negative angle C type 37.7=**1, 37.6=0**

When it does not exceed -360, the shortest distance in the negative direction;

When it exceeds -360, the negative direction is relatively short distance, the continuous same instruction does not exceed -360 does not move, and the continuous same movement exceeds -360 degrees -360 displacement.

N1 G0 C0	The sequence number	The actual amount of movement	Absolute coordinates	Machine seat coordinates
N2 C-260	N2	-260	100	100
N3 C-260	N3	0	100	100
N4 C-360	N4	-100	0	0
N5 C-540	N5	-180	180	180
N6 C-540	N6	-360 (-180-180)	180	180
N7 C-260	N7	-80	100	100
N8 C-720	N8	-100	0	0

4) Negative angle D type 37.7=1**, 37.6=1**

When exceeding or not exceeding -360, the shortest distance in the negative direction;

N1 G0 C0	The sequence number	The actual amount of movement	Absolute coordinates	Machine seat coordinates
N2 C-260	N2	-260	100	100
N3 C-260	N3	0	100	100
N4 C-360	N4	-100	0	0
N5 C-540	N5	-180	180	180
N6 C-540	N6	0	180	180
N7 C-260	N7	-80	100	100
N8 C-720	N8	-100	0	0

Appendix IX: In-program invocation subroutine functions (2021.12 with subsequent versions).

Description: On the basis of the original M98P_ call subprogram, add the use of calling subprograms within the program (the main program and the subprogram are in the same program).

Parameter: bit parameter 29.6 M98 H_L_ subprogram function within the program (0: invalid 1: valid) [Invalid by default]；

Format: M98 H_ L_;

Description: H_ subroutine start line number;
L_ Number of calls;

When L_ is omitted, it is called once by default
The subprogram ends M99 and the subprogram returns to execute the next line of the main program;

Note 1: The subroutine must end with M99, otherwise the call will be wrong;
Note 2: When the subprogram does not end with M99, the subprogram is called only once, and the entire program ends;
Note 3: Subroutines are nested up to 10 levels;

Example:

```
O0001;
M3S1000;
....
....
M98 D10 L5;// Call the subroutine behind N10 5 times
N5 ....
   ....
M98 D20 L5;// Call the subroutine behind N20 5 times
N6 ....
N7 ....
......

M30;//End of program
N10;// subprogram D10
....
....
M99;// line N5 is returned at the end of the subroutine
N20;//subprogram D20
....
```

....
M99;// subroutine ends with line N6

Appendix X: New modifications to the speed control of the handwheel trial cutting simulation (2021.12 subsequent versions have).

The feed axis speed of the new handwheel simulation is modulated according to the frequency of the handwheel shaking, but the upper speed limit is still affected by the number parameter 149 and the number parameter 82;

Parameter: bit parameter 1 8.7 [handwheel test cut follows handwheel rotation speed (0: invalid 1: valid)];
Number Parameter 1No. 53 [Handwheel Trial Cutting Follows the Speed Change Base];

The larger the base number of the speed change base followed by the handwheel test cut, the faster the speed change (equivalent to the greater the acceleration and deceleration, the more obvious the jitter); The smaller the base of change, the slower the change (equivalent to the smaller the acceleration and deceleration, the smaller the jitter).

Appendix XI: M89 Custom Detection/Output (202 1.12 with subsequent versions).

- **Programming format description:**
  ① M89 K_ J_ R_ P_ I_ Q_
  K_: The heartbeat selects K0—K7
  J _ : Wait detection time, in milliseconds
  R_ : Repeat output interval time , in milliseconds
  P_: the number of repeated outputs in the range P0 is output 1 time, P1 is 2 repetitions, P2 is 3 repetitions, and so on
  I _ : Select Open output addresses I0—I7
  Q_: Select Turn off output addresses Q0—Q7
  Note: The output addresses that I/Q open and close correspond to one by one.

M89 output I_, wait for the detection K_, no detection in J_ time K_ turn off the output I_, after R_ time to open the output I_ alarm when the P_ is not detected within the number of cycles K_, and turn off the output at the same time;
If P_ completes the M89 code within K_ of the number of cycles, the Q_ output is turned off,

and if the Q_ is not filled in, it remains I_;

If I_=Q_, the M89 turns off the output when I_ is complete; If I_ ≠Q_, the output remains I_ and the Q_ output is turned off.

② M89 K_ J_

When M89 does not detect J_ within K_ waiting time, an alarm is generated;

M89 code completes while waiting for J_ K_ time to detect;

③ M89 I_

M89 output I_ hold

④ M89 Q _

M89 Off Output Q_

⑤ M89 I_ J_ Q_

M89 output I_ Elapsed time J_ Off output Q_;

If the Q_ is not filled in, the M89 output I_ the output is turned off J_ elapsed time;

If the I_ = Q_ time elapses J_ turn off the output;

If I_ ≠Q_, the output is held J_ after the I_ elapsed and the Q_ output is turned off.

⑥ M89 K_

M89 waits for K_ detected, the M89 code completes;

⑦ M89 K_ J_ I_

M89 output I_ Wait for detected K_ , J_ detected within K_ time, M89 code completed;

If the J_ is not detected within the time K_, the system generates an alarm;

When J_ or J=0 is not written, it waits for the detection K_ signal until the K_ signal is detected to complete the M89 code, and an alarm is generated;

⑧ M89 K_ J_ Q _

M89 closes output Q_ Wait for K_ detected, J_ detected within K_ time, M89 code completes;

If the J_ is not detected within the time K_, the system generates an alarm;

When J_ or J=0 is not written, it waits for the detection K_ signal until the K_ signal is detected to complete the M89 code, and an alarm is generated;

M89 L_ Signal Detection Minimum Hold Time M89 K Detection function is turned on, if there is no L insertion, the code is completed immediately as soon as there is a signal, and when there is L, it is necessary to continuously detect that the signal is greater than the L editing time to complete the code;

All formats with K can use L detection duration.

Note: When K0-K7 is not written after **M89, the M89 code is completed directly;**

M89K0-K7 detects the address relationship between the signal and the input port X

M89I0—I7/Q0—Q7 output/off signal corresponds to the address relationship with output port Y

Note: The output input address needs to be viewed in the "Debug Documentation" M 89 article corresponding to different system models

Appendix XII. M90/M91 program segment loop (202 1.12 subsequent versions have).

- **Programming format description:**

  M91 C_;// Cycle program segment starts, C_ number of cycles

  ;

  ;

  M90;//End of loop program segment

  Example:

  T01 M06;

  M3 S1000;

  G0 X32 Z5;

  M91 C5; The loop program section starts and loops 5 times (loop N10-N20 program segment).

  N10 G01 X30 Z0 F500;

  G01 Z-20;

  G01 X20 Z-40;

  G0 X32;

  N20 Z0;

  M90; The looping program segment ends

Appendix XIII: M92/M93 jump function (202 1.12 with subsequent releases).

M92 Conditional Jump Instruction:

- Programming format:

  M92 K_ N_ ;

  M92 L_ N_ ;

K_ Enter high; (The range K0--K7 corresponds to #1008--#1015=1).

L_ Input low; (Range L0--L7 corresponds to #1008--#1015=0)

N_ Jump program segment number;

When the conditions are met, jump to the N_ section;

Example:

M92 K0 N10;// When #1008=1, the program jumps to the N10 segment; G65 H81 P10 Q#1008 R1 instructions are the same

G0 X50 ;

N10 G0 X30 ;

G0 X0;

M92 L0 N10;// When #1008=0, the program jumps to N10 segment G65 H81 P10 Q#1008 R0 instructions are the same

M93 Unconditional jump instruction:

- Programming format:

  M93 N_ ;

  N_ Jump program segment number;

  Jump to N_ when running to M93 N_;

  Example:

  M93 N10;// Jump to N10 segment

  G0 X50;

  N10 G01 X30;

  When running to the M93 N10 segment, the program jumps to the N10 segment and does not execute G0 X50.

Note: To enter the address, please check the macro input section of the "Debug Documentation" corresponding to different system models

2022/ 04/27 5th Edition