HNC8 Series CNC Controller

Programming Manual

V2.4

Introduction

The manual may help you to quickly get familiar with the HNC-8 system, providing detailed information about commissioning, programming or application methods. Any updates or modification of the manual is not allowed without the written permission of Wuhan Huazhong Numerical Control Co., LTD (hereafter referred to as "HCNC"). Without HCNC's authorization or written permission, any units or individuals are not allowed to modify or correct the manual. HCNC will not be responsible for any losses thus incurred to customers.

In this manual we have tried as much as possible to describe all the various matters concerning of the system. However, we cannot describe all the matters which must not be done, or which cannot be done, because there are so many possibilities. Therefore, matters which are not especially described as possible in this manual should be regarded as "impossible" or "not allowed".

Copyright of the manual should be reserved by HCNC. Any units and individuals' publication or duplication should be deemed as illegal behavior and we will hold them accountable.

Please favor me your instruction for shortages and inadequacies of the manual.

⚠ **Note**

⚠ As to notes such as "Limitations" and "Usable functions", the specification provided by the machine tool manufacturer is superior to the manual. Please conduct dryrun before actual machining and confirm machining program, tool compensation volume and workpiece offset, and so on.

⚠ Please explain matters which are not described in the manual as "Infeasible".

⚠ The manual is prepared on the condition that all functions are configured. Please make a confirmation according to the specification provided by the machine tool manufacturer in use.

⚠ For relevant instructions for machine tools, please refer to the specification provided by the machine tool manufacturer.

⚠ Usable screens and functions differ with different NC systems (or versions). Please be sure to confirm specifications before use.

Content

1 Overview

1.1 Coordinate Axis

1.1.1 Machine Coordinate Axis

Function and Purpose

In order to simplify programming and ensure the versatility of the program, a unified standard has been formulated for the naming of coordinate axes and directions of CNC machine tools.

The fixed linear feed coordinate axes are represented by X, Y, Z which are referred to as the basic coordinate axes; the coordinate axes rotating around the X, Y, and Z axes are represented by A, B, and C which are referred to as the rotary coordinate axes.

1） Basic coordinate axes X, Y, Z

The direction of the machine tool axis depends on the type of machine tool and the layout of each component. The relationship between the X, Y, and Z coordinate axes is determined by the right-hand rule, as shown in Figure 1-1. In this figure, the thumb points in the positive direction of X axis, the index finger points in the positive direction of Y axis, and the middle finger points in the positive direction of Z axis.
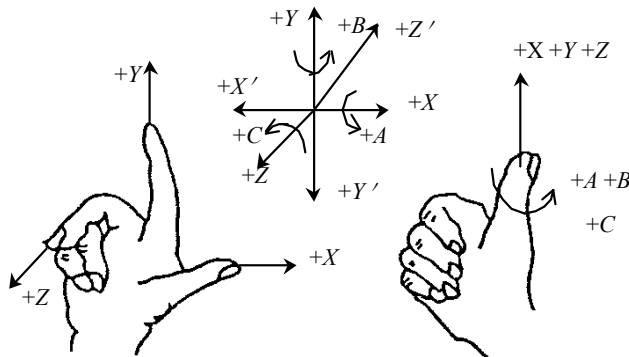
Figure 1-1 Machine coordinate axis

2） Rotary axis

The circular feed coordinate axes that rotate around the X, Y, and Z axes are represented by A, B, and C respectively. According to the right-hand screw rule as shown in Figure 1-1, the thumb points to +X, +Y, +Z directions, and the index finger and middle finger point to +A, +B, +C directions of circular feed motion.

1.1.2 CNC Control Axis

Function and Purpose

The standard HNC8 CNC controller has 3 axes; it can control up to 5 axes by adding auxiliary axes. When specifying each machining axis and direction, use the preset coordinate letter and the set direction.

Item	HNC-8 (Standard)
Basic controlled axes	3 axes
Extended controlled axes (total)	5 axes at maximum (including CS axis)
Basic simultaneously-controlled axes	3 axes
Extended simultaneously-controlled axes	5 axes at maximum

Axis name: 3 standard basic axes are X, Y, Z, and the the additional axes can be named A, B, C, U, V, W.

Note: The additional axis names U, V, W can be set by the parameters, and are not related to the UVW instruction words during lathe incremental programming. For details, see the instructions on lathe incremental programming.

1.2 Reference Point, Machine Origin, and Machine Coordinate System

Function and Purpose

Reference point: The machine reference point is a fixed mechanical point on the machine tool (For some machine tools it is determined by the travel switch and dog, and for some machine tools it is directly determined by the grating ruler zero, etc.)

Machine origin: The machine zero is a fixed point on the machine tool, and the CNC device uses it as a reference for position control. The machine zero is determined by the reference point and the system parameter "coordinate value of reference point in machine coordinate system".

Machine coordinate system: The machine coordinate system is the inherent coordinate system of the machine tool. It takes the machine zero as the origin, and each coordinate axis is parallel to the corresponding machine axis. The origin of the machine coordinate system is also called the machine origin or machine zero. In the machine coordinate system, the workpiece is always considered to be stationary, while the tool is considered to be moving.

Figure 1-2 shows the horizontal lathe with front tool post. Since it is a machine tool with a rotary spindle, first user determines the Z-axis direction: the spindle axis direction is the Z-axis direction, and the direction the tool moves away from the workpiece is the positive direction of Z-axis; then user determines the X-axis direction: in the machining plane of the machine tool, the direction perpendicular to the Z-axis is the X-axis direction, and the direction the tool moves away from the workpiece is the positive direction of X-axis; At last user determines the Y-axis direction: the Y-axis positive direction can be determined based on the right-hand Cartesian rule.
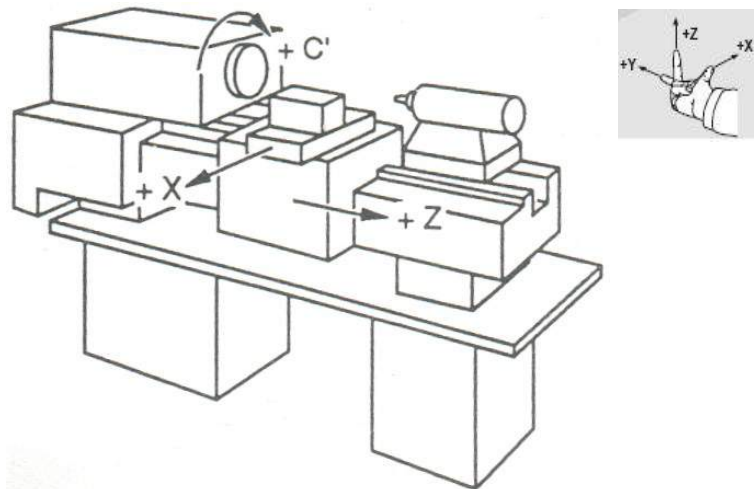
Figure 1-2 Horizontal lathe with front tool post

Figure 1-3 shows a single-column vertical milling machine (or machining center). Since it is a machine tool with a rotary spindle, first user determines the Z-axis direction: the spindle axis direction is the Z-axis direction, and the direction the tool moves away from the workpiece is the Z-axis positive direction; then user determines the X-axis direction: while the operator is facing the column and the worktable is moving, relative to the workpiece the direction of the tool moving

to the right is the X-axis positive direction; at last user determines the direction of the Y axis: according to the right-hand Cartesian rule, the positive direction of Y-axis is the direction tool moves to the column relative to the workpiece.

Taking the machine zero point as the origin, the coordinate system established with each axis of the machine tool is the machine coordinate system of the machine tool.
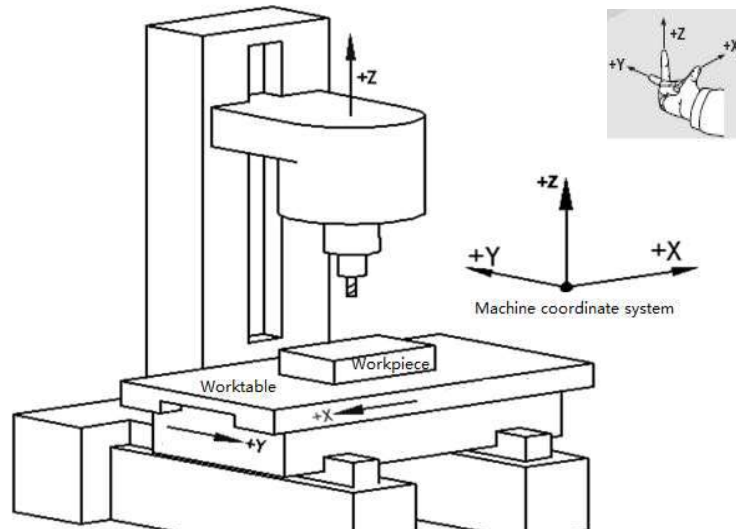
Figure 1-3 Vertical milling machine with single column

1.3 Workpiece Origin and Workpiece Coordinate System

Function and Purpose

The workpiece coordinate system is used by the programmer during programming. The programmer selects a known point on the workpiece as the origin (also referred to as the program origin), and establishes a coordinate system parallel to each axis of the machine tool, which is called the workpiece coordinate system. Once the workpiece coordinate system is established, it will remain valid until it is replaced by a new workpiece coordinate system.

The workpiece coordinate system is determined by the programmer, and generally a position that is convenient for processing and calculation is selected.

1.4 Coordinate System and Travel

Function and Purpose

The travel of the machine tool coordinate system is divided into effective travel and mechanical travel. The effective travel is the range of tool movement which is set by CNC parameters; the mechanical travel is determined by the travel switch, and its value is determined by the manufacturer. The relationship between the machine zero (M), machine reference point (m), workpiece coordinate origin (W), effective travel and mechanical travel of the machine coordinate

system is shown in the figure below.
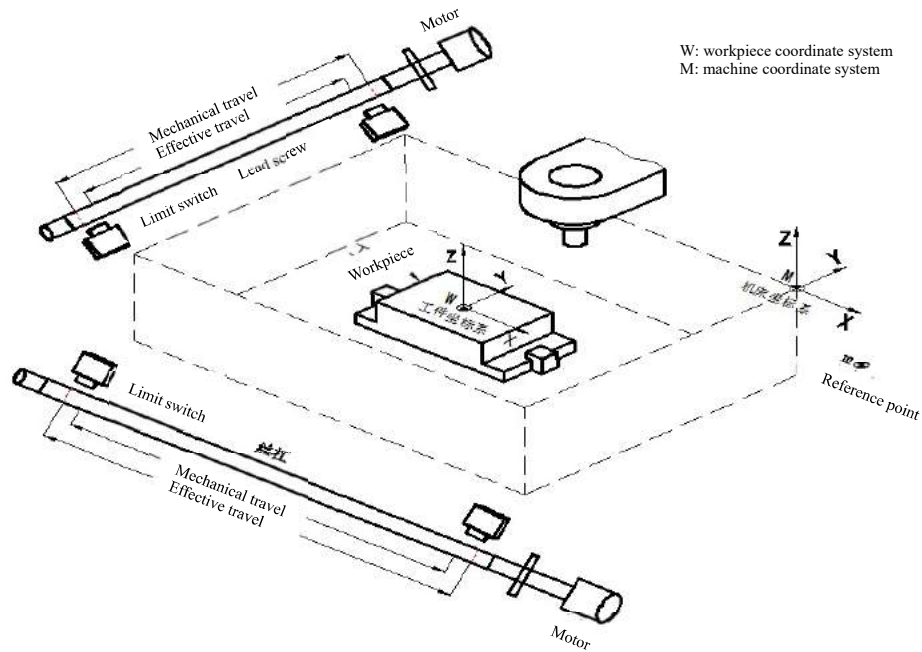
1.5   Positional Relationship between Coordinate Systems

Function and Purpose

The coordinate system set with the machine zero as the origin is referred to as the machine coordinate system. The machine tool manufacturer sets the machine zero for each machine tool. The workpiece coordinate system is the coordinate system used in workpiece processing. Generally, their position setting relationship is shown in the figure below.
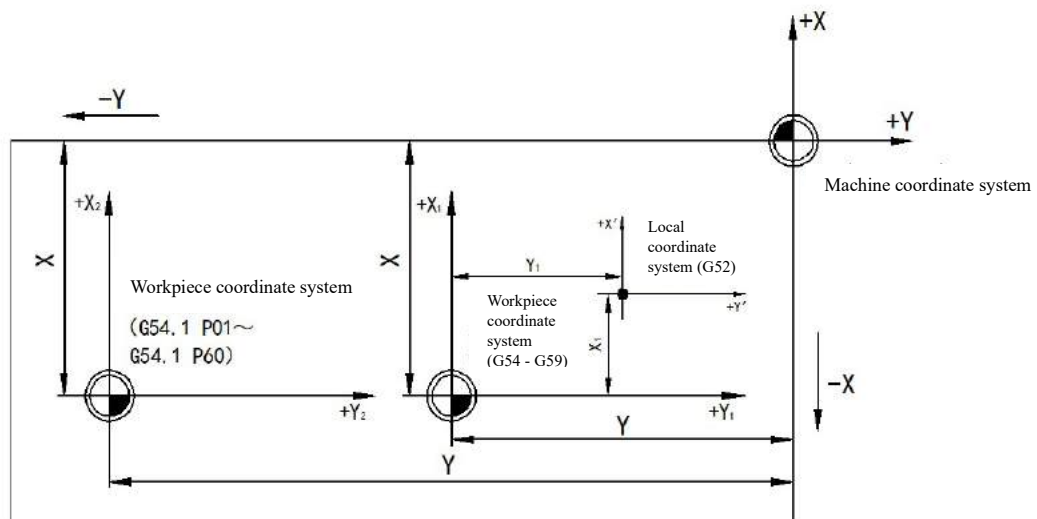
1.7   General Programming Methods and Steps

Function and Purpose

Programming means that the programmer determines the processing technology of the parts based on the drawing. The processing process, process parameters, processing path and auxiliary actions required in processing such as tool change, cooling, clamping, etc. are documented as the program list in accordance with the processing sequence and the command code and program format specified by the CNC machine tool used. Then all the contents in the program list are input into the CNC controller, so that the CNC machine tool can process based on the contents.

The general methods and steps for programming and processing are as follows:

Blueprint → Craft → Tool Path Calculation → Manual or CAD/CAM programming → Program input → Tool setting → Program Verification → Precutting → **Batch Machining**

Edit
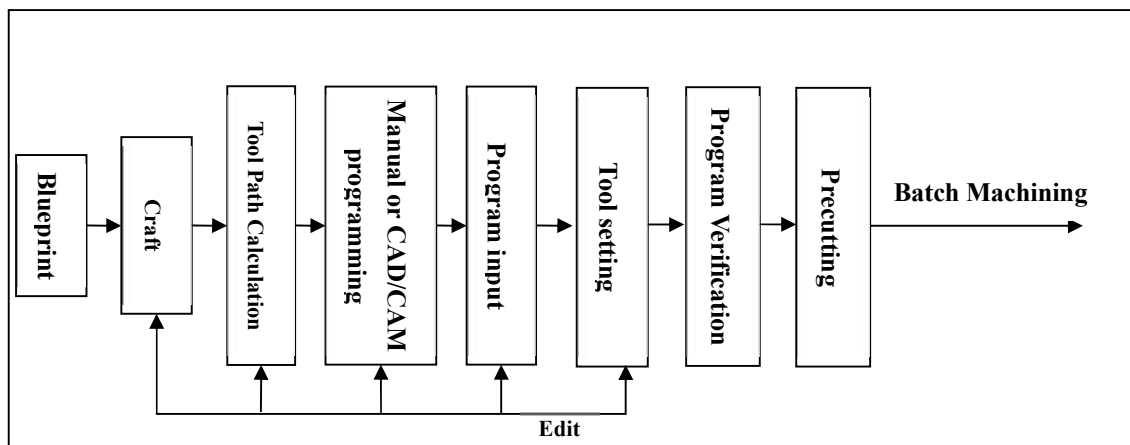
2 Program Format and Structure

2.1 Program Format

Function and Purpose

When providing information to a controller, the format specified is referred to as the program format. The format used by the controller is referred to as the word address format.

A program is a collection of "block" units, and a "block" is used to specify a machine action (sequence). These commands (blocks) are worked out in the order of actual tool movement. A program block is a collection of "word" units, and a "word" is used to specify a command of an operation. A word is a collection of characters (English letters, numbers, symbols), and the characters are arranged in a certain order.

2.1.1 Address and Command Word

Function and Purpose

The command information used by this controller consists of letters (A, B, C...Z), and numbers (0, 1, 2...9), and symbols (+, -, /...). These letters, numbers, and symbols are collectively referred to as characters. What is expressed in this form is referred to as a code. This controller uses ISO code which is written in notepad format.

Description

The initial letter of a word is referred to as the address, which defines the meaning of the following numerical information. A command word is composed of address characters (command characters) and numeric data with signs (such as the word with a defined dimension) or without signs (such as the preparatory function word G code). In this controller, the word is composed of a letter (word address) and several numbers following it. (User can also add symbols such as "-" at the beginning of the number).
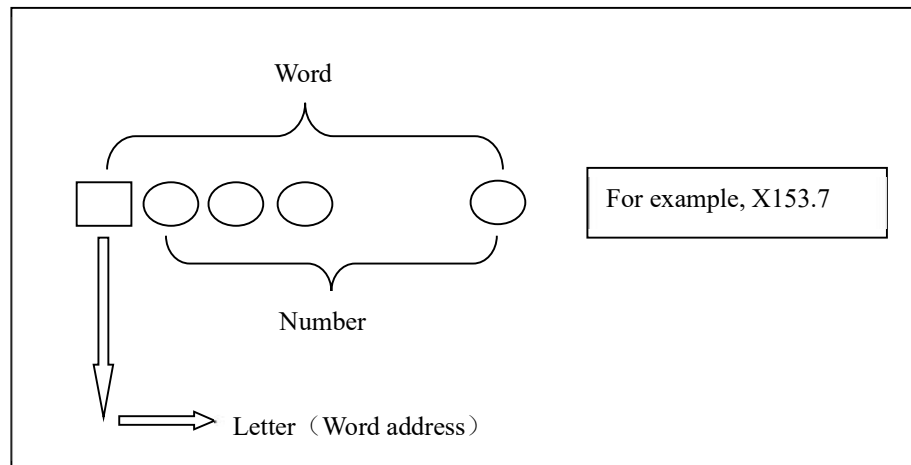
Structure of word

Different command characters and the subsequent numbers in the block determine the meaning of each command word. The main command characters contained in the CNC program block are shown in Table 2.1.

Table 2.1 List of command characters

Function	Address	Meaning
Parts program No.	Letter O	Program No.：O1~4294967295
Block No.	N	Block No.：N0~4294967295
Preperatary function	G	Command action mode (straight line, circular arc, etc.): G00 to G200
Dimension word	X，Y，Z	Traverse command of linear axis: ±21474
	A，B，C	Traverse command of rotary axis: ±21474
	U，V，W	Incremental programming command of lathe: ±21474
	R	Circular radius of canned cycle
	I，J，K	Coordinates of circle center relative to the starting point for canned cycle
Feedrate	F	Feedrate：0 to 50000
Spindle function	S	Spindle speed：0 to 100000
Tool function	T	Tool number：0 to 99
Auxiliary function	M	ON/OFF control at the machine side：0 to 99
	A	Worktable indexing
Compensation No.	H，D	Tool compensation number：00 to 99
Dwell	P，X	The dwell time: millisecond, second
Program No.	P	Sub-program number：1 to 4294967295
Repeat times	L	The number of repetitions of subprogram, the number of

		repetitions of canned cycle
Parameter	P，Q，R	Canned cycle parameter

2.1.2 Block and Block Number

Function and Purpose

The composition of the block: A program block is composed of more than one word. A group of single-step sequential commands or a line of several functional commands is referred to as a program block.

Block number: the number used to distinguish each block, which is composed of address N and the following 5 digits (1 to 99999). The block number is placed at the beginning of the block, and can be specified in any order. Any block number can be skipped except for the block number specified by the code command with special meaning. The block number can be specified for all blocks, and also can be specified only for the blocks required by the program. However, for convenience, in general the block number is specified in the order of processing steps.

Descriptions

A program block defines a command line to be executed by the CNC controller. The format of the block defines the syntax of the functional words in each block.

The structures of block number and block are as follows:

1 block (or command line)

N○○ G○○ X○○○ Y○○○ M○○ S○○ T○○

Block   Preparatory function   Dimension work   Auxiliary function   Spindle function   Tool function
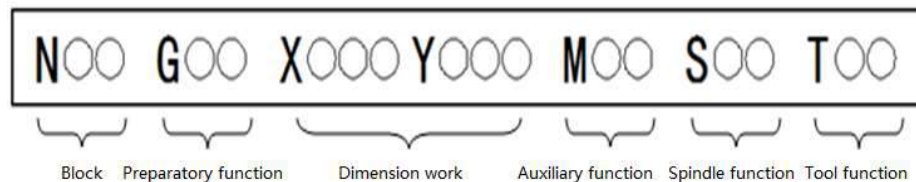
E.g. N100 G01 X24.7 Y59.31 M03 S1000 T0101; the part underlined is the block number

A block starts with the sequence number that identifies the block (the block number can also be absent), and ends with the block end code. In this manual, a newline indicates the end of the block. Like a notepad, a newline character indicates the end of the line. The preliminary function determines the content of the dimension character. In this manual, the dimension words are represented by IP.

2.1.3 General Structure of Program

Function and Purpose

A program of parts is executed in the order of the block line, not in the order of the block number, but it is recommended to write the block number in ascending order when writing the program.

Description

Program start: The first cell of the first line cannot be numbers or symbols other than "%".

Program end: the program end code is usually specified at the end of the program, and one of the following codes is to indicate the program end code.

Code	Meaning
M02	End of main program
M30	
M99	End of subprogram

Comment symbol: the content bracketed () or after the semicolon ";" is the comment text.

Single line command: When writing the processing G code program, some commands must be written in a single line. Such as M30, M02, M99, M6T, CTOS, STOC, G16, G15, G05.1, G04.

2.1.4 File Attributes of Program

Function and Purpose

User can set the access attribute for the file program to be writable or readable.

Description

Editing prohibited: The current loader can be set to "read-only" through interface operations. At this time, the file cannot be rewritten until it is set to "writeable" through interface operations.

In addition, user can also control the access properties of the program through the key switch of the project panel, but this key switch is effective on all programs in the program manager, that is, when the switch is turned off, all programs will become read-only until the switch is turned on.

2.1.5 Program and Program Name

Function and Purpose

Program: A parts program is composed of several blocks that follow certain structure, syntax and format rules, and each block is composed of several command words. The program name corresponds to each workpiece, or is a number to classify programs in the unit of subprogram. The program name is a number used to distinguish each program. Use the address "O" (capital letter O) and the following number or letter to specify.

The newly created program name on the system can be up to 7 numbers or letters; the system can read the program name created externally with greater than 7 characters.

Description

This system calls the program by calling the program name for processing or editing.
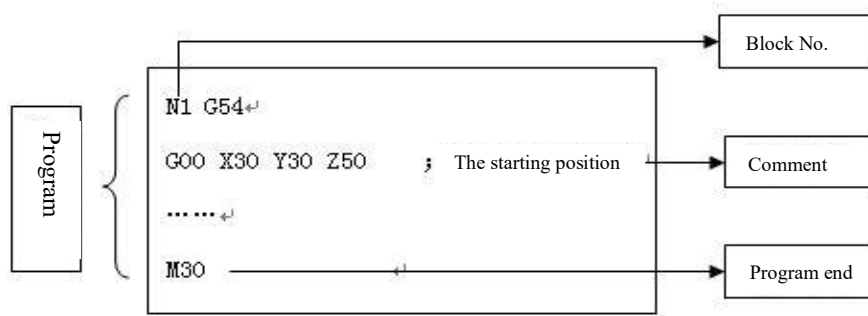
If the program end code is executed during the program execution, the CNC ends the program execution and goes into the reset state. When the subprogram end code is executed, the control returns to the main program that calls the subprogram.

Note

If the Optional Skip switch on the machine operation panel is turned on, for example,

/M02;

/M30;

/M99; Not considered to be the end of the program.

2.1.6 Optional Block Skip

Function and Purpose

This function means that the block after the "/" (slash) code in the processing program can be selectively ignored until the block ends.

Description

The specific function starting with the "/" symbol. It is used to choose whether the selective processing block is executed or not.

Detailed description: When the Optional Skip switch is ON, the line codes with the symbol "/" at the beginning of the block are skipped; when the Optional Skip switch is OFF, the line codes with the symbol "/" at the beginning of the block are executed, and the "/" symbol does not work.

For example, when certain blocks in the program do not need to be executed under certain circumstances, but need to be executed in another environment, this function can be used.

Note

The ON or OFF setting of the Optional Skip button on the CNC controller must be executed before the program runs.

The special "/" symbol for optional skip must be at the beginning of the block. (If it is inserted in the middle of the block, it is used as a division operation command of the user macro)

Example: N30 G1 X15/Y5; error, illegal symbol alarm

     /N30 G1 X15 Y5; correct

2.2   Main Program and Subprogram

Function and Purpose

When a same processing mode appears multiple times in a program, this mode can be edited into a program for repeated calling to simplify the program. Such a program is referred to as a subprogram, and the original program is referred to as the main program. The call of the subprogram is performed with M98 or G65 command, and the return from the subprogram is performed with M99 command. Refer to the chapter of user macro program for the specific subprogram calling of G65.

The subprogram can be called repeatedly up to 999 times. User can also call other subprograms in a subprogram, and up to 6 layers are supported.

Description

Structure of calling subprogram:
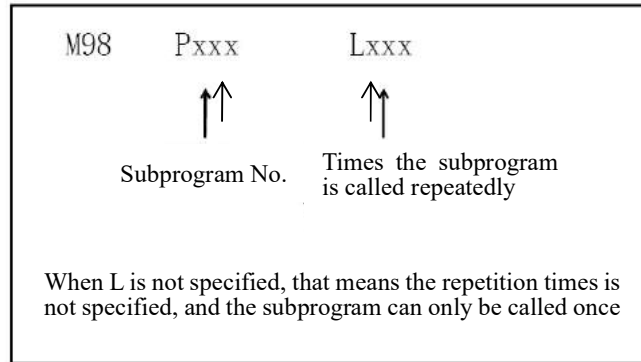
When a call to subprogram execution command occurs during the execution of the main program, the subprogram command is executed. When the subprogram is executed, return to the main program to continue the execution.
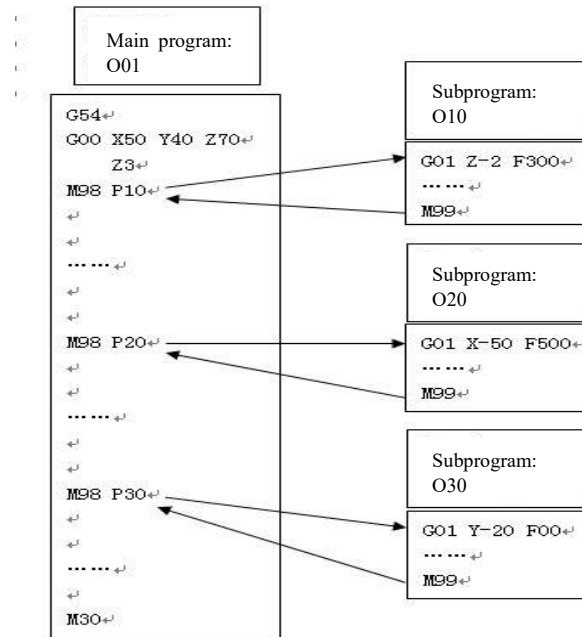
Special usage of M99: when it is used as the main program end command, return to the main program head to execute the program again.

There are two forms to call subprogram:

1) The subprogram is placed after the main program, that is, after the end of the program (M30 or M02). The program number called can start with % or the letter O;

2) When the subprogram is used as a separate program, it should be placed in the same disk (system disk or USB disk) as the main program.

⚠️ **Note**

1) In both forms, the subprogram placed after the main program has priority, that is, the first case is prior to the second case.

2) When calling a subprogram, the name of the subprogram cannot have a suffix, otherwise the alarm "File not loaded" is issued.

3) The subprogram cannot be executed in MDI, otherwise the alarm will be issued. "MDI cannot call external subprograms".

When the main program calls a subprogram, it is regarded as a first layer of subprogram, and the called subprogram can also call another subprogram. The depth can reach up to 6 layers, as shown below:
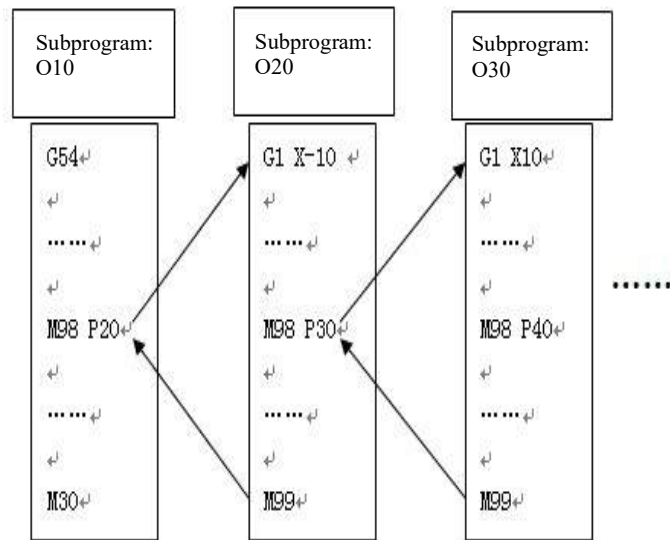
Subprogram structure nesting

2.3   Cautions before Machining

⚠ **Caution**

⚠ When creating a processing program, please select appropriate processing conditions and be careful not to exceed the performance, capacity, and limitations of the machine and NC. We do not take the above processing conditions into consideration for the cases mentioned in this manual.

⚠ Please perform dry run before actual machining to confirm the machining program, tool compensation amount, workpiece offset amount, etc.

3 Preparatory Function (G Code)

3.1 G Code Modal and Grouping

Function and Purpose

Modal of G code

G codes can be divided into two types according to their effective states: non-modal G codes and modal G codes.

Non-modal G code: It is valid only when the G code is specified, and it is invalid if it is not specified.

Modal G code: This type of G code is stored by the CNC system after it is executed once, and remains valid until it is replaced by other codes in the same group.

Grouping of G codes

G codes are divided into several groups according to their functions. Group 00 is for non-modal G codes, and the other groups are for modal G codes.

Description

1) Multiple G codes of different groups can be specified in the same block.

2) If multiple codes of the same group are specified in the same block, only the last specified code is valid.

3) If the G code of group 01 is specified in the canned cycle, just like the G80 command is specified, the canned cycle can be cancelled. But the G code of canned cycle does not affect the code of 01 group.

3.2 List of G codes (T)

After the controller is powered on, the one marked with " 【 】 " in the table is the initial modal in the same group, and the one marked with " 〖 〗 " is the equivalent macro name of the G code.

G code	Group No.	Function
G00	01	Rapid traverse positioning
【G01】		Linear interpolation
G02		Circular interpolation CW/ Helical interpolation CW
G03		Circular interpolation CCW/ Helical interpolation CCW
G04	00	Dwell
G07	00	Imaginary axis designation
G08		Read-ahead OFF
G09		Exact stop check

G10	07	Programmable data input
【G11】		Programmable data input cancel
G17	02	XY plane selection
【G18】		ZX plane selection
G19		YZ plane selection
G20	08	Inch input
【G21】		Metric input
G28	00	Reference point return
G29		Return from reference point
G30		Return from 2nd, 3rd, 4th, 5th reference points
G32	01	Thread cutting
【G36】	17	Diameter programming
G37		Radius programming
【G40】	09	Tool radius compensation cancel
G41		Left tool compensation
G42		Right tool compensation
G52	00	Local coordinate axis setting
G53		Direct machine coordinate system programming
G54.x	11	Extended workpiece coordinate system selection
【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
G60	00	Unidirectional positioning
【G61】	12	Exact stop mode
G64		Cutting mode
G65	00	Macro non-modal call
G71	06	Inner/Outer roughing multiple repetitive cycle
G72		Face end roughing multiple repetitive cycle
G73		Close turning multiple repetitive cycle
G76		Threading cutting multiple repetitive cycle
G80		Inner/Outer diameter cutting cycle
G81		Face end cutting cycle
G82		Thread cutting cycle
G74		Face end deep-hole drilling cycle
G75		Outer diameter pocket cutting cycle
G83		Radial drilling cycle
G87		Axial drilling cycle
G84		Axial rigid tapping cycle
G88		Radial rigid tapping cycle
【G90】	13	Absolute programming

G91		Incremental programming
G92	00	Workpiece coordinate system setting
G93		Inverse time feed
【G94】	14	Feed per minute
G95		Feed per revolution
【G97】	19	Constant linear speed control ON
G96		Constant linear speed control OFF
G101		Axis release
G102		Axis acquire
G103		Command channel loader
G103.1		Command channel loader running
G104		Channel synchronization
G108 『STOC』	00	Spindle is switched to C axis
G109 『CTOS』		C axis is switched to spindle
G110		Alarm
G115		Rotary axis angle resolution redefining

3.3 List of G codes (M)

G code	Group No.	Function
G00		Rapid traverse positioning
【G01】	01	Linear interpolation
G02		Circular interpolation CW/ Helical interpolation CW
G03		Circular interpolation CCW/ Helical interpolation CCW
G04	00	Dwell
G05.1	27	High speed high precision mode
G07		Imaginary axis designation
G07.1	00	Cylindrical interpolation
G08		Read-ahead OFF
G09		Exact stop check
G10	07	Programmable data input
【G11】		Programmable data input cancel
G12	18	Polar coordinate interpolation ON
【G13】		Polar coordinate interpolation OFF
【G15】	16	Programmable data input
G16		Programmable data input cancel
【G17】		XY plane selection
G18	02	ZX plane selection
G19		YZ plane selection
G20	08	Inch input
【G21】		Metric input

G24	03	Mirroring ON
【G25】		Mirroring OFF
G28	00	Reference point return
G29		Return from reference point
G30		Return from 2nd, 3rd, 4th, 5th reference points
【G40】	09	Tool radius compensation cancel
G41		Left tool compensation
G42		Right tool compensation
G43	10	Tool length Compensation plus
G44		Tool length Compensation minus
【G49】		Tool length Compensation Cancel
【G50】	04	Scaling ON
G51		Scaling OFF
G52	00	Local coordinate system setting
G53		Direct machine coordinate system programming
G54.x	11	Extended workpiece coordinate system programming
【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
G60	00	Unidirectional positioning
G61	12	Exact stop mode
【G64】		Cutting mode
G65	00	Macro non-modal call
G68	05	Rotation transformation enable
【G69】		Rotation transformation cancel
G73	06	Deep hole drilling cycle
G74		Reverse tapping cycle
G76		Fine boring cycle
【G80】		Canned cycle cancel
G81		Center drilling cycle
G82		Drilling cycle with pause
G83		Deep hole drilling cycle
G84		Tapping cycle
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle (manual)
G89		Boring cycle
G181		Circular groove cycle (type 1)
G182		Circular groove cycle (type 2)

G183		Circumferential groove milling cycle
G184		Rectangular pocket cycle
G185		Circular pocket cycle
G186		Face end milling cycle
G188		Rectangular boss cycle
G189		Circular boss cycle
【G90】	13	Absolute programming
G91		Incremental programming
G92	00	Workpiece coordinate system setting
G93		Inverse time feed
【G94】	14	Feed per minute
G95		Feed per revolution
【G98】	15	Start point return of canned cycle
G99		Reference point return of canned cycle
G101		Axis release
G102		Axis acquire
G103		Command channel loader
G103.1		Command channel loader running
G104		Channel synchronization
G106		Measurement data record and export
G108 〖STOC〗	00	Spindle is switched to C axis
G109 〖CTOS〗		C axis is switched to spindle
G110		Alarm
G115		Rotary axis angle resolution redefining
NURBS		NURBS spline interpolation
HSPLINE		HSPLINE spline interpolation

⚠ **Note**

After the controller is powered on, the one marked with " 【 】 " in the table is the initial modal in the same group, and the one marked with " 〖 〗 " is the equivalent macro name of the G code.

During the operation, if a code not included in the G code is specified, a program error (alarm code) will occur.

4   Auxiliary Function

4.1   M Command

Function and Purpose

The auxiliary function code is composed of the address word M and the following numbers. It is mainly used to control the parts program and auxiliary functions such as various auxiliary switch actions of the machine tool. Among them, M00, M01, M02, M30, M92, M93, M98, M99 are used to control the parts program. They are default auxiliary functions defined by CNC, are not determined by the machine tool manufacturer and unrelated to the PLC program. Other M codes are used for auxiliary switch actions of the machine tool, and their functions are not determined by the CNC, but specified by the PLC program, so the codes may vary with different machine tool manufacturers. The functions provided in this manual are conventional definitions. The specific conditions are subject to the machine description.

Description

Pre and post attributes for M code

When the M function is specified in the same block as the traverse command, the execution sequence of the M command may be pre, post, and synchronous.

Pre function: The M function is executed before the axis movement programmed in the block.

Post function: The M function is executed after the axis movement programmed in the block.

Synchronous function: The M function is executed simultaneously with the axis movement programmed in the block.

In the parameter configuration of this system, the M code table can be set to take effect in three ways: pre, post, and synchronous. Which state is applicable depends on the machine specifications.

M指令名称	M指令组号	M指令类型	M指令备注
M03	3	前置	主轴正转
M04	3	前置	主轴反转
M06	6	同步	未指定
M07	7	同步	未指定
M08	7	同步	冷却液开
M09	7	后置	冷却液关
M10	10	同步	未指定
M11	11	同步	未指定
M12	12	同步	未指定
M13	13	同步	未指定
M14	14	同步	未指定
M15	15	同步	未指定

(Note 1) The activation modes of M codes M00, M01, M02, M05, M30, M92, and M93 that are not displayed in the list are fixed and cannot be modified.

(Note 2) The M code that is set or defaulted to the post should not be specified together with the traverse command. It is better to be specified in a line separately, otherwise errors of some M codes may occur.

Multiple M command specification instructions

Usually only one M code is valid for a block. In this system, at most 4 M codes can be specified in a block. When users need to specify multiple M codes in a block, please note the following:

(1) Up to 4 M codes can be specified in a block, and multiple M codes in the same group are regarded as one.

(2) When multiple M codes of the same group are designated in a block, the last M code in the block is effective.

(3) When multiple different groups of M codes are designated in a block, the M codes in the block are all effective.

(4) M codes such as M00, M01, M02, M30, M99 cannot be specified together with other M codes.

(5) M codes such as M00, M01, M02, M30, and M99 require to be specified in a separate line, that is, the program line containing the above M codes can not only have one M code, and cannot have other execution commands including G and T commands.

(6) The corresponding relationship between M codes and functions depends on the specific settings of the machine tool manufacturer. See the machine tool manufacturer's manual for details.

Note

Although multiple M codes can be specified in a block, due to the different specifications of each machine tool and the different settings by each machine tool manufacturer, it is recommended not to specify multiple M codes in a block at the same time, otherwise unpredictable errors may occur.

4.1.1 Default Auxiliary Function of CNC

Function and Purpose

M00 Program dwell

When the CNC executes the M00 command, it will pause the execution of the current program to facilitate the operator to perform operations such as tool and workpiece size measurement, workpiece U-turn, manual speed change, etc.

During pause, the feed of the machine tool is held, and all the existing modal information remains unchanged. User should press the CycleStart button on the operation panel for the continued execution of subsequent program.

M00 is valid for the current program line, it is the post M function. Its post attribute is not allowed to be modified.

M01 Optional stop

If user presses the Optional Stop button on the operation panel, when the CNC executes the M01 command, it will suspend the execution of the current program and the machine tool will be in the feed hold state to facilitate the operator to perform the tool extension length and workpiece size measurement, workpiece U-turn, manual speed change and other operations. During pause, the feed of the machine tool stops, and all the existing modal information remains unchanged. User should press the CycleStart button on the operation panel for the continued execution of subsequent program.

If user does not activate the Optional Stop button on the operation panel, when the CNC executes the M01 command, the program will not pause and continue to execute.

M01 is valid in the current program line, it is the post M function. Its post attribute is not allowed to be modified.

M92 Program dwell (manual intervention)

When CNC executes M92, the program execution pauses and the machine is in feed hold state, but the difference from M00 is that at this time user can manually intervene in each axis, manually command its movement, and then switch to the Auto mode to continue the running of

the current program after pressing CycleStart button.

M92 can be used in the following occasions:

(1) In manual boring, when the boring tool is automatically processed to the bottom of the hole, the machine tool stops running, and changes the working mode to "Jog". Through manual operation, the tool can only retract after moving radially in the opposite direction of the tool nose for a certain distance to avoid the wall of the workpiece hole. The general milling machine can complete fine boring with this command without the need of spindle exact stop function.
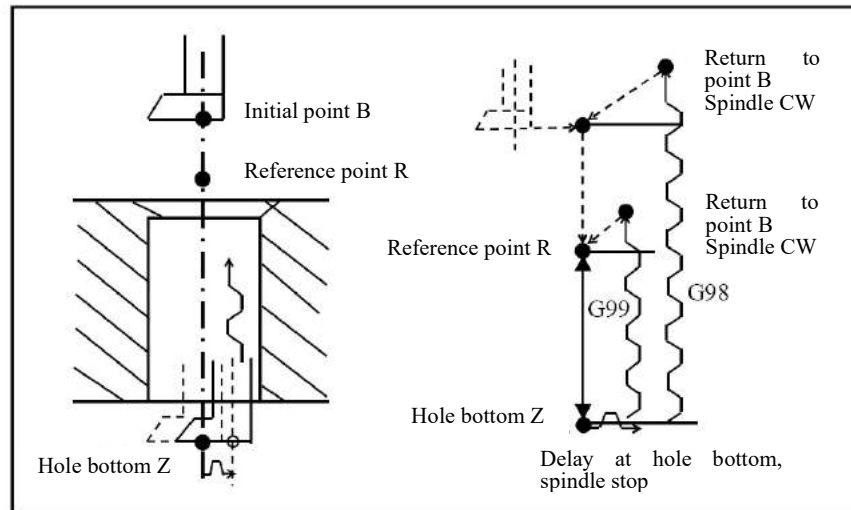
(2) During manual measurement, when the user manually measures the tool or workpiece through the dialogue interface, the CNC actually processes it as a manual measurement cycle, and M92 is inserted in the appropriate place in the cycle to guide the user to complete the entire measurement task. If the system executes M92 of the canned cycle, the program is paused. At this time, the system waits for the user to switch to the "Jog" mode to command the measuring axis to run to the measuring position, and then switches to the "auto" mode. User presses the CycleStart button to continue the measurement cycle.

M93 Program dwell (manual intervention is disabled)

The M93 command is equivalent to the M00 command. Unlike M92, the user cannot perform manual intervention when the program is paused with M93.

M02 End of program

M02 is in the last block of the main program. When the CNC executes the M02 command, the spindle, feed, and coolant of the machine tool stop, and the machining ends.

If user needs to re-execute the program which has been finished its execution with M02, user

has to call the program again, or press the "rerun" key in the submenu of automatic processing, and then presses the CycleStart button on the operation panel.

M02 is valid in the current program line. It is the post M function. Its pre and post attributes are not allowed to be modified, and it must be specified in a separate line.

M30 Program ends and returns

The functions of M30 and M02 are basically the same, but the M30 command also has the function of returning to the program head.

If user needs to re-execute the program which has been finished its execution with M30, just press the CycleStart button on the operation panel again.

(Note 1) This command must be placed in a separate line to take effect.

M98/M99 Subprogram call

If the program contains a fixed sequence or frequently repeated pattern, such a sequence or pattern can be stored as a subprogram in the memory to simplify the program.

(Note 1) The maximum number of subprogram calls (L) is 10,000 times.

(Note 2) A subprogram can be called from the main program.

(Note 3) A called program can also call another subprogram.

```
Structure of subprogram
%xxxx；   Subprogram No.
……；      Subprogram contents
M99；      Subprogram returns
Subprogram call (M98)
M98 P__ L__
     P__：   Subprogram No. called (in Arabic numerals)
     L__：   The number of times the subprogram is called repeatedly
```

Nested call of subprogram

When the main program call a subprogram with M98, it can be regarded as a first-level of subprogram call. The subprogram call can be nested up to 6 levels, as shown below:
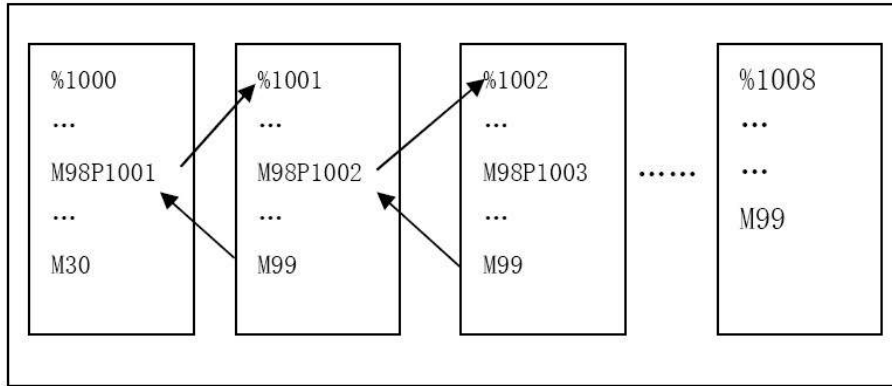
(Note 1) When the main program calls a subprogram with M98, G80 must be added before M99 in the subprogram to ensure the correct operation of the program.

(Note 2) M98/M99 need to be used in a separate line (avoid being in the same line with other commands).

Use M99 in the main program

If M99 is executed in the main program, the control returns to the beginning of the main program, and the main program is executed from its beginning, and loops forever.

4.1.2 Auxiliary Function Set by PLC

Function and Purpose

M03/04/05 Spindle control

M03 Starts the clockwise rotation of spindle at the speed programmed (viewed from the positive direction to the negative direction of Z axis).

M04 Starts the counterclockwise rotation of spindle at the speed programmed.

M05 Spindle stops

(Note 1) M03, M04, and M05 can mutually cancel each other.

(Note 2) The spindle can be switched from position mode to speed mode with M03/M04 instead of G109.

M06 Tool change

M06 is used to call a tool which needs to be installed on the spindle of the machining center. When this command is executed, the tool will be automatically installed on the spindle. For example: M06 T01; the tool 01 will be installed on the spindle.

(Note 1) M06 is effective in the current program line.

(Note 2) M06 needs to be used in a separate line (avoid being in the same line with other commands).

M07/08/09 Coolant control

M07 and M08 are to turn on the coolant pipeline (based on the definition of the machine tool manufacturer).

M09 is to turn off the coolant pipe.

M64 Workpiece counting

M64 will accumulate the number of completed workpieces in the system processing statistics

M19/M20 Spindle orientation

M19 is to enable spindle orientation.

M20 is to disable spindle orientation

4.2   Table of M Command Function and Regular Status

Code	Function	Pre/Post/Synchronous			Function remains valid until it is canceled	Function is valid in the block it exists
		Synchronous	Pre	Post		
M00	Program dwell			#		○
M01	Optional stop			#		○
M02	End of program			#		○
M03	Spindle rotation CW		○		○	

M04	Spindle rotation CCW		○		○	
M05	Spindle stop			#	○	
M06	Tool change		○			○
M07	Coolant 2 ON		○		○	
M08	Coolant 1 ON		○		○	
M09	Coolant OFF			○	○	
M10 to M18	Undefined					
M19	Spindle orientation		○			○
M20	Spindle orientation cancel			○		○
M21	Tool release (umbrella-type magazine)	○				○
M22	Tool clamping (umbrella-type magazine)	○				○
M23	Magazine forward (umbrella type magazine)	○				○
M24	Magazine backward (umbrella type magazine)	○				○
M25	Tool selection (umbrella-type magazine)	○				○
M26 to M29	Undefined					
M30	Program ends and returns to its head			#		○
M31 to M63	Undefined					
M64	Workpiece counting	○				○
M65 to M91	Undefined					

M92	Program dwell (manual intervention)			#		○
M93	Program dwell (manual intervention is not allowed)			#		○
M94 to M97	Undefined					
M98	Subprogram call			○		○
M99	Subprogram returns to main program			○		○
M100 to M999	Undefined					

"○"：The function is conventionally specified.

"#"： The function status is fixedly specified.

For the "undefined" codes, its function may be defined when the standard is revised in the future.

5   Spindle Function

5.1   Spindle Speed Setting

Function and Purpose

The spindle speed can be controlled by the number specified after the address S.

In addition to running in speed mode (rotating at a certain speed), the spindle can also operate in position mode (switching to the rotary axis for interpolation calculation).

Through the M command controlled by the PLC, the S command is processed and terminated.

Example

%1234

G92X0Y0Z20；Establish coordinate system

G00 X0 Y0 Z2；Positioned above the hole

M03S2000；CW spindle rotation

G41 G01 X20 D01 F300；Establish tool radius compensation

G03X40R10；Start the circular cutting

G03 I-40 Z-10 L5；Mill the hole in spiral

G03 I-40；Smooth the hole bottom

G03X20R10；Exit the circular cutting

G40G01X0；Cancel tool radius compensation

G0Z20；Lift the tool

M05；Spindle stops

M30；End of program

5.2 Constant Linear Speed Cutting Control（G96/G97）（T）

Function and Purpose

This function is only applicable to CNC lathe system. It adjusts the spindle speed with the movement of the tool nose position (constant linear speed cutting control), so that the speed of the cutting point is always a constant speed (fixed cutting speed), which improves the uniformity of the surface.

As the tool nose is moving to the workpiece origin, the spindle speed of the machine tool is getting higher and higher until the machine specification limit is reached, which is very dangerous. Therefore, please be sure to set to the maximum speed limit in the spindle speed limit setting command (G46).

Constant linear speed control when the constant speed cutting control command G96 is set to S314m/min.

Workpiece radius: 50mm (radius value) Workpiece radius: 25mm (radius value)
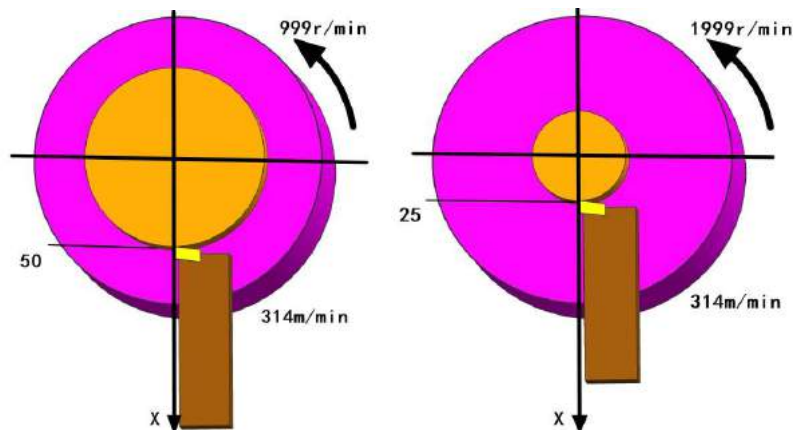
Because the linear speed is constant, the spindle speed is calculated as the tool nose position moves, and it changes automatically.

In the above example, because the linear speed (314 m/min) is constant, the speed changes from 999 (r/min) to 1999 (r/min) as the radius of the workpiece changes (50mm → 25mm).

The relationship between linear speed and spindle speed

n=1000*V /πD

n——machine spindle speed r/min

V——Cutting linear speed of workpiece circumference m/min

D——Diameter of workpiece mm

Command Format

G96 P_ S_ ;   To activate constant linear speed cutting control function of specified axis

G46　X_　P_ ;　　Spindle speed limit

G97　S_ ;　　　　To disable constant linear speed cutting control function of specified axis

Parameter	Meaning
P	The control axis of the constant linear speed cutting control specified with the G96 command, the axis specified by P is determined by the system axis parameters. 1～3 respectively represent the X, Y, and Z axes; The maximum speed limit of the spindle (r/min) when the G46 command specifies the constant linear speed cutting control;
S	Specify the constant linear speed (mm/min or inch/min) with G96 command; The designated spindle speed (r/min), after the constant linear speed cutting control is canceled with G97 command,;
X	Minimum spindle speed limit (r/min) for constant linear speed;

Description

(1) The system only supports the X-axis to control spindle speed for constant linear speed cutting control. The system has not yet opened the function of controlling the spindle speed with Y or Z axis for constant linear speed cutting control;

(2) G96/G97 is a pair of modal commands, and they can be mutually canceled;

(3) The G46 command function is only valid when the constant linear speed cutting control function is valid;

(4) To use the constant linear speed cutting control function, the spindle must be able to change speed automatically. (E.g., servo spindle, PWM spindle);

(5) When performing the constant linear speed cutting control function, if the spindle speed is greater than the maximum spindle speed, it will be clamped at the maximum spindle speed.

(6) The linear speed of spindle is specified with S command when the constant linear speed cutting control is enabled.

(7) The constant linear speed calculation is always executed at the time of the cutting feed command (such as G01).

(8) The constant linear speed cutting control cancel command (G97) is only available when the constant linear speed cutting control command is enabled.

Note

1) G96 must be followed by G46 to limit the maximum and minimum spindle speed. When the controlled axis of the constant linear speed cutting control approaches to the spindle center, the spindle speed increases, and the allowable speed of the workpiece and chuck may be exceeded, which may cause damage to the tool and machine tool, and even cause injury; while the controller axis of the constant linear speed cutting control moves away from the spindle center, the spindle

speed decreases, which will be less than the actual machining speed required, resulting in the inability to machine qualified parts. Therefore, G96 must be followed by G46 to limit the maximum and minimum spindle speeds, and when user specifies the constant linear speed cutting control during programming, pay attention to the distance between the controlled axis and the spindle center.

2) When G96 is commanded, do not omit the linear speed command "S_". When the command is omitted, the system will alarm.

5.3 Spindle Clamping Speed

Function and Purpose

The maximum and minimum spindle speeds are limited by G46 command. According to the specifications of the workpiece, the chuck installed on the spindle, and the tool, the spindle speed limit is set when the speed needs to be restricted, so that qualified products can be processed. It is often used in conjunction with constant linear speed control G96/G97.

Command format

G46　X_　P_; Spindle speed limit

Parameter	Meaning
P	Max. spindle speed limit（r/min） when constant linear speed cutting control is specified
X	Min. spindle speed limit （r/min） when constant linear speed cutting control is specified

Description

The speed is limited only in the constant linear speed cutting control mode.

Note

Spindle clamping speed G46 is often used in conjunction with the constant linear speed cutting control G96/G97. G96 must be followed by G46 to limit the maximum and minimum spindle speed. When the controlled axis of the constant linear speed cutting control approaches to the spindle center, the spindle speed increases, and the allowable speed of the workpiece and chuck may be exceeded, which may cause damage to the tool and machine tool, and even cause injury; while the controller axis of the constant linear speed cutting control moves away from the spindle center, the spindle speed decreases, which will be less than the actual machining speed required, resulting in the inability to machine qualified parts. Therefore, G96 must be followed by G46 to limit the maximum and minimum spindle speeds, and when user specifies the constant linear speed cutting control during programming, pay attention to the distance between the controlled axis and

the spindle center.

5.4 C/S-Axis Switching（CTOS/STOC）

Function and Purpose

In addition to running in speed mode (rotating at a certain speed), the spindle can also run in position mode (switching to the rotary axis for interpolation calculation). At this time, the spindle drive needs to support the speed mode/position mode switching function (ie C/ S-axis switching function).

In some applications such as the rigid tapping function, the C/S-axis switching function is required.

Command format

STOC/G108 IP；

CTOS/G109 IP；

Parameter	Meaning
IP	IP can be A/B/C, the number after them indicates the spindle number in the channel, the value ranges from 0 to 3; When the IP is absent after STOC, the No. 0 spindle is changed to the C axis by default; When the IP is absent after CTOS, the C axis is changed to to the No. 0 spindle by default.

Explanation: 1) G108 B0 means to switch No. 0 spindle to B axis; G109 B0 means to switch B axis back to No. 0 spindle. Generally, the G108 B0 function is used on the 5-axis machine tool; the 3-axis milling machine generally only uses G108 (no IP is written afterwards), and the No. 0 spindle becomes the C axis by default.

Example

%0007; Rigid tapping test program, R is the program zero
G92 G17 Z0.000
G109
M03 S1000.000; CW spindle rotation
M05
G90 G0 Z1
G108; Switch speed mode to position mode for spindle
G98 G84 Z-20.000 R1 P500 F1.000; Perform rigid tapping
G109; Switch position mode to speed mode for spindle
G01 Z0.000
M30

Note: The current version of the system software G84 canned cycle already includes the C/S-axis

switching function of G108/G109. When using G84, there is no need to write G108/G109. G84 can run normally whether G108/G109 is written or not in the program. There is an older version of the system software, whose G84 canned cycle does not include the C/S axis switching function of G108/G109, so when using G84 in the program, it needs to be used with G108/G109. They are listed here to explain the G108/G109 commands.

Note

1) The machine tool needs to be equipped with a spindle servo drive to support the speed mode/position mode switching function.

2) The spindle has never been rotated after power-on. Before using the C/S-axis switching function, the spindle needs to rotate several revolutions, otherwise the system will warn-"C/S switching requires manual homing".

3) In the same G code program, it is best not to use the STOC/CTOS macro commands frequently.

4) When the spindle is switched to C axis, the unit of C axis is deg/min.

5) It is not allowed to use any line function to skip between STOC and CTOS, and it is also not allowed to use any line to skip from elsewhere to between STOC and CTOS.

5.5 Spindle Orientation

Function and Purpose

Spindle orientation is to stop the spindle at a specific position.

In the processing of CNC machine tools, in order to realize automatic tool change and enable the manipulator to accurately load the tool into the spindle hole, the keyway of the tool must be axially aligned with the key position of the spindle; During boring, when retracting the tool, it is required that the tool can retract after moving a certain distance radially in the opposite direction of the tool nose to avoid scratching the workpiece. All of these require an accurate axial positioning function of spindle.

Command format

M19/M20 ；

Parameter	Meaning
M19	Enable spindle orientation
M20	Cancel spindle orientation

Note

1) Generally, the spindle orientation command does not need to be edited in the machining

program. It is executed by calling the canned cycle program in the system with commands such as boring or tool change (M06).

2) The spindle orientation position is set in the parameter PA39 of the spindle drive, or by modifying 105539 (spindle orientation position pulse) on the system side (the spindle is usually logical axis 5). The above takes HSV-180US spindle drive as an example. The unit of the parameters is pulse.

5.6 Spindle Synchronization Control（G146/G147）

Function and Purpose

With the improvement of spindle control technology, there are also new requirements for the mode and efficiency of CNC machining, such as polygon turning processing, load tapping where the spindle speed does not drop to zero, and dual-spindle workpiece exchange with continuous spindle rotation, all requiring spindle synchronization control function.

The spindle synchronization control of this system is also called the electronic gearbox function. This function controls the transmission ratio of the synchronous axis through programming, and performs high-precision motion coupling control of the spindle.

Through the coordination of programming commands and channel parameters, up to 3 groups of 6 spindles (master axis and slave axis) can be controlled.

Command Format

G146I_ J_ R_ P_　　 ；Enable synchronization

G147P_　　　　　　 ；Disable synchronization

Parameter	Meaning
I	Transmission ratio of master axis
J	Transmission ratio of slave axis
R	Phase angle deviation value of the master and slave axis
P	Synchronization group number (the system has designed a total of 3 groups of axis coupling control, the serial number is 1, 2, 3, and the default is 1)

Description

The system parameter corresponding to the command (channel parameter, * represents the channel number)

Parameter number	Parameter	Description
04*340	The first group of electronic gearbox master axis number	Set the logical axis number of master axis

04*341	The first group of electronic gearbox slave axis number	Set the logical axis number of salve axis
04*342	The first group of electronic gearbox master axis ratio	【I】Set the transmission ratio of master axis
04*343	The first group of electronic gearbox slave axis ratio	【J】Set the transmission ratio of slave axis
04*344	The first group of electronic gearbox synchronization type	Set the synchronization type of master and slave axes (0: actual position synchronization; 1: command position synchronization)
04*345	The first group of electronic gearbox phase ON	Set whether to synchronize the phase angle when the master and slave axes rotate (0: not synchronized 1: synchronized)
04*346	Phase angle of the first electronic gearbox	[R] Set the angle of the synchronization phase angle (0 to 360 degrees)
04*347 ～04*353	The second group of electronic gearbox parameter	The parameter description is similar with the first group
04*354 ～ 04*360	The third group of electronic gearbox parameter	The parameter description is similar with the first group

Example

Example 1

(1) The system is configured with dual channel. The spindles in the two channels need to synchronously complete the exchange of workpieces (spindle 5 and spindle 6)
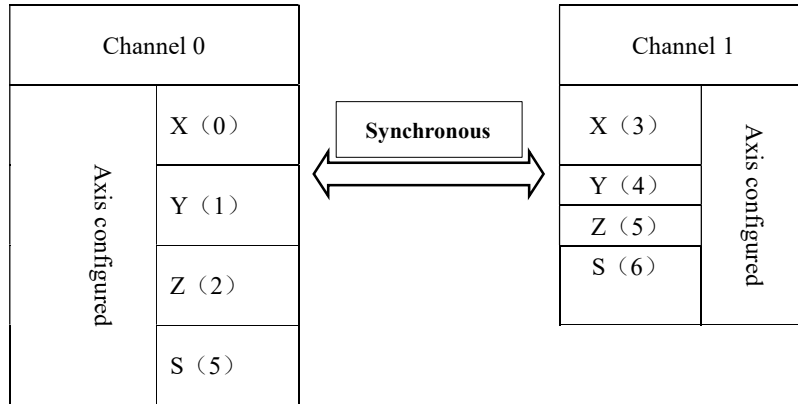
(2) Parameter setting

| Parameter | Channel 0 | | Channel 1 | | |
	Parameter No.	Value	Parameter No.	Value	Explanation

The first group of electronic gearbox master axis number	040340	0	041340	5	Set axis 5 as the master axis
The first group of electronic gearbox slave axis number	040341	0	041341	6	Set axis 6 as the slave axis
The first group of electronic gearbox master axis ratio	040342	0	041342	1	Set the transmission ratio
The first group of electronic gearbox slave axis ratio	040343	0	041343	-1	of master and slave axes
The first group of electronic gearbox synchronization type	040344	0	041344	0	Set to the actual position synchronization
The first group of electronic gearbox phase ON	040345	0	041345	1	Phase angle synchronization is enabled
The first electronic gearbox phase angle	040346	0	041346	0	

(3) Sample program

Channel 0	Channel 1
T0101 G99	G104P1
M3 S1000 ；Master axis starts	M4 S200; Slave axis rotates first (servo needs to be enabled first)
G4X2	
G104P1	G146; Synchronization starts (The synchronization parameters are based on the settings of above table)
G104P2 ；Wait for synchronization to complete	T2222
G0 Y0.0	G98
G99G0 X30.0 Z15.2	G28 Z0.0 X0.0 F5000
G4X0.5	G104P2
G104P3	G104P3
G104P4	M21; Chuck is automatically released
G4 X0.3	G98
G99G1X-2.0F0.06	G0 Z-182.0
M5 ；Master and slave axes stop	G1 Z-204 F5000
G104P5	G4 X1.0
G104P6	M22; Chuck is automatically clamped
M30	G4 X0.3
	G104P4
	G104P5
	G28 Z0.0
	M5
	G104P6
	G147; Synchronization ends
	M30

Example 2

(1) The system is configured with a single channel, one spindle and one power head.

The power head cooperates with the spindle to process 4, 6, and 8 polygons with the fly cutter (the fly cutter is equipped with 2 tools).

Channel 0	
Feed axis	X（0）
	Z（2）
Spindle	S（5）
Power head	S1（3）

(2) Parameter setting

Parameter	Channel 0		Description
	Parameter No.	Value	
Master axis No. of the first group of electronic gearbox	040340	5	Set axis 5 as master axis
Slave axis No. of the first group of electronic gearbox	040341	3	Set axis 3 as slave axis
Master axis ratio of the first group of electronic gearbox	040342	0	Not set
Slave axis ratio of the first group of electronic gearbox	040343	0	
The first group of electronic gearbox synchronization type	040344	0	Set to the actual position synchronization
The first group of electronic gearbox phase opens	040345	1	Phase angle synchronization is enabled
The first group of electronic gearbox phase angle	040346	0	Not set

(3) Sample program:

```
%1234
M103S1=0; Power head is enabled (slave axis)
M3S200; Master axis starts
G0Z30
G146 I1 J-2 R0; The synchronization is enabled with the transmission ratio of master
            and slave axes being 1:-2 (power head rotates reversely); the phase
            angle R is 0, at this time a quadrilateral is processed.
T1
G0Z2
```

X-23
G01X-12.44F2
Z0
Z-3F1
G0X-23
M3S200
G146I1J-3R0; The synchronization is enabled with the transmission ratio of master
and slave axes being 1:-2 (power head rotates reversely); the phase
angle R is 0, at this time an hexagon is processed.
G1X-17.6F1
Z-6
G0X-23
M3S200
G146I1J-4R0; The synchronization is enabled with the transmission ratio of master
and slave axes being 1:-2 (power head rotates reversely); the phase
angle R is 0, at this time an octagon is processed.
G1X-20.32F1
Z-9
G0X-23
Z50
X-50
G147; Synchronization ends
M30

Note

1) If I, J, R programming is specified with the G146 command, the corresponding functions set with the parameters will not take effect. At this time, it is based on the parameter settings in the program processing command. If I, J, and R are not specified in the G146 command, then the two spindle coupling control parameters are based on the channel parameter settings.

2) When there is no P parameter in the G146 command, the system uses the first group of electronic gearbox parameters by default.

3) If the master and slave axes are respectively set in two channels, when using the electronic gearbox function, the G146 command needs to be executed in the channel to which the slave axis belongs, and the parameters should also be set in the channel where the slave axis is located. Otherwise, when running the program in the master axis channel, the system will alarm: the program syntax error.

4) If G147 is absent in the program, the system will cancel the G146 mode during panel reset or emergency stop reset. In programming, please note that after enabling G146 synchronization, user needs to cancel synchronization with G147. Otherwise, when the program runs again, the system will alarm: the spindle is not ready, and send the command alarm.

6　Tool Function

6.1　T Command in Lathe System

Function and Purpose

In the process of machining parts on a lathe, different tools are often used. In order to simplify the program, it is assumed that the positions of the tool noses are the same during programming. However, due to the different shapes and installations of the tools, the actual positions of the noses cannot be consistent. The T command function of the lathe is used to realize the tool change control and the offset compensation processing for the inconsistency of the actual positions of the tools.

In addition, when the system realizes position offset compensation, it can also realize tool nose radius compensation under the same offset number.

Command Format

1. T command function format

T and the following 4 digits, see the figure below.

➤ The first 2 digits are the tool number (the currently selected tool number)

➤ The last 2 digits are the tool compensation number (the register number for storing the offset and the register number for storing the radius compensation)
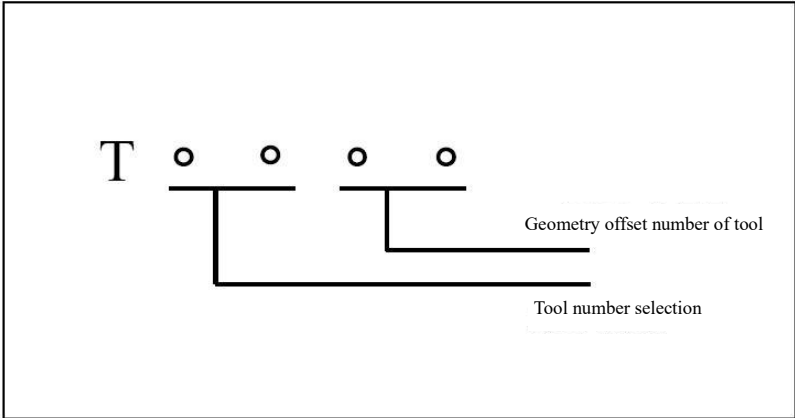
2. Relative deviation and absolute deviation

There are two ways to select the commonly used tool deviation: relative deviation and absolute deviation

➤ Relative deviation value

The position deviation of each tool nose relative to the datum tool nose. In this mode, the

program also needs the command to call the position relationship between the datum tool and the workpiece zero (such as G92 command).

➢ Absolute deviation value

When each tool is at the machine zero position (the machine returns to zero and the tool is at the machining tool position), the position deviation of the workpiece zero relative to each too nose. This method is adopted by this system.

📖 **Description**

1. Tool offset setting

The first two digits of the T command are the tool number for tool selection; the last two digits are the tool offset number for calling the tool offset. When the tool offset number is 00, it means that the offset is 0, that is, the offset function is cancelled.

The tool offset number can be the same as or different from the tool number, that is, a tool can correspond to multiple offset numbers (values).

The tool offset is set in the setting interface of the lathe, as shown in the figure below. The tool offset of the X axis and Z axis can be set through the tool offset setting and tool post translation setting. For specific operations, please refer to the lathe operation manual.

There is also a wear setting for the wear of the tool. When the tool is worn due to excessive use, the offset will become larger. When this value is set, the system will also calculate the wear value on the offset value to correct the tool offset amount (the specific amount of wear is determined after the workpiece is measured).

The parameter 000064 Tool Wear Accumulation Enable can make the tool wear value set every

time be accumulated to the tool offset of the system.

2. Three levels of coordinate system

➢ Machine coordinate system

➢ External zero offset, G54-G59 coordinate systems

➢ T command coordinate system

The low-level coordinate system is completed on the basis of the high-level coordinate system, that is, the calling of the T command offset and wear is completed on the machine coordinate system, the external zero offset, and the coordinate system of G54-G59.

| **Example** |

External zero offset X is 8, Z is 0

G54 coordinate system X is 4, Z is 0

No. 1 tool offset is X11, wear is 0

No. 2 tool offset is X14, wear is 3

No. 3 tool offset is X9, wear is -1

G54

T0101; Change No. 2 tool No. 2 and call No. 1 tool offset

G01 X5 When the workpiece coordinate system reaches 5, the actual machine X is 28 (5+8+4+11+0)

T0202; Change to No. 2 tool and call No. 2 tool offset

G01 X5 When the workpiece coordinate system reaches 5, the actual machine X is 34 (5+8+4+14+3)

T0303; Change to No. 3 tool and call No. 3 tool offset

G01 X5 When the workpiece coordinate system reaches 5, the actual machine X is 25 (5+8+4+9-1)

T0301; Change to No. 3 tool and call No. 1 tool offset

G01 X5 When the workpiece coordinate system reaches 5, the actual machine X is 28

(5+8+4+11+0)

Note

1) Pay attention to the position of the tool when the program executes tool change to prevent the tool touching other devices.

2) The tool offset value needs to be set for each tool installed.

3) Both the wear value and the offset value will be included in the program. If it is not needed, 0 is set.

4) Regardless of the relative offset or absolute offset mode, the tool offset value generally needs to be got by tool setting.

6.2　T Command in Milling System

Function and Purpose

The tool function is also called T function, which specifies the tool number. It is designated by the 4 digits (0～9999) after the address T.

By designating a value following the address T, a code signal is input to the machine tool to control the tool selection on the machine tool.

When the traverse command and the T command are specified in the same program line, there are three execution methods based on the M code type:

M06 is to set the synchronization type, and the traverse command and M code are executed simultaneously;

M06 is set to the Pre type, after the M code is executed, the traverse command is executed

M06 is set to Post type, after the traverse command is executed, the M code is executed

The choice of these three methods depends on the machine tool manufacturer's regulations
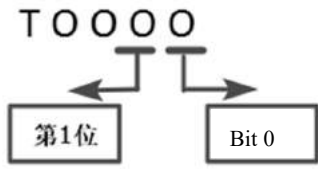
Description

When the T command is executed on the machining center, a code signal or pulse signal is input to the machine tool to control the magazine to rotate to the selected tool, and then wait until the tool change is automatically completed with M06.

● **Parameters related to magazine**

Machine user parameter **010089 T Command Control Mode** sets the tool change mode and tool processing mode selection of T command in binary.

Bit		Functions
Bit 0	0	T command only has tool selection function for the magazine which have the tool preselection function, e.g., manipulator magazine
	1	T commands have the functions of tool selection and tool change, e.g. magazine of drilling-tapping center
Bit 1	0	Disable tool machining mode
	1	Enable tool machining mode

NC parameter 000012 Tool Axis Selection Mode is used to determine which axis the G43/G44 tool length compensation should compensate.

0: Tool length compensation is always compensated to the Z axis.

1: The tool length compensation axis is switched according to the coordinate plane selection modal G command (G17/G18/G19), corresponding to the Z/Y/X axis respectively.

Channel parameter 040127 Starting Tool Number is used to set the starting tool number of magazine in the tool compensation table in the current channel, used in conjunction with the channel parameter Number of Tools

Channel parameter 040128 Number of Tools is used to set the number of tools in the current channel, which is consistent with the number (or add one) of tool positions of magazine in the current channel. If the starting tool number in channel 0 is set to 1, the number of tools is set to 5, the starting tool number in channel 1 is set to 6, and the number of tools is set to 10, then the data saved about the tools 1-5 in the tool compensation table (the tool offset is included for lathe system) is of the magazine 0 in the channel, and the data saved about the tools 6-15 is of the magazine 1 in the channel.

The channel parameter 040060 Number of Tool Data Saved by System is used to set how many tools (radius, length) the system saves. This parameter must be greater than or equal to the sum of the "number of tools" set in all channels.

- **Parameter related to big/small tool**

Machine user parameter 010099 Magazine Management Interface for Big/Small Tool

0: Disable the magazine management interface for big/small tool;

1: Enable the magazine management interface for big/small tool.

- **Parameter related to tool grouping and life management**

Channel parameter 040130 Tool Life Management Mode

0：Disable tool life function;

1：Enable tool life function, and grouping is not supported;

2： Enable tool life function, grouping is supported, and T command specifies tool group number;

3： Enable tool life function, grouping is supported, and T command specifies tool number (just for milling system);

Channel parameter 040133 Ignored Number of T Command Life Management.

Channel parameter 040135 Length Compensation of Tool Group in Milling: tool group length compensation number after tool grouping function is enabled.

Channel parameter 040136 Radius Compensation of Tool Group in Milling: tool group radius compensation number after tool grouping function is enabled.

- **Parameters related to multi-tool-edge function**

NC parameter 000372 Number of Tool Edges can enable the multi-tool-edge interface. When the number of tool edges is 0, the regular tool compensation interface is enabled; when the number of tool edges is 1-9, the multi-tool-edge interface is enabled.

Note

The tools in the magazine table are managed by the system and generally cannot be modified. For the umbrella type tool magazine, the M06 code and the T command are required to be written in the same block. Pay attention to the tool magazine table during tool change, the tool number of group 0 (such as 15) is the position number of the tool clamped on the spindle in magazine. If other tools need to be installed on the spindle, the tool must be returned to the position (No. 15) in magazine. At this time, there must be no tools in this position in the magazine, otherwise a collision will occur.

Therefore, when magazine loads the tool, it is recommended to install the tool on the spindle first, and then run the M code and T command (such as M06 T01) in the MDI mode to install the tool in the magazine through the spindle.

7   Feed Function

7.1   Overview

Function and Purpose

The feed function is that the CNC system issues a feedrate command to the servo drive, the servo drive controls the motor, and then controls the movement of the tool or worktable.

The feed function not only needs to specify the speed of the feed movement, but also needs to plan the speed for the start and stop of the feed movement. The CNC controller adopts automatic acceleration and deceleration control in the feed control. While the worktable is moving, the speed and direction of feed may change at the joint between the line segment and the line segment or between the line segment and the arc, which causes the oscillation of the machine tool, and the surface quality of the workpiece reduces. Therefore, automatic acceleration and deceleration control is adopted to avoid this phenomenon.

Description

The automatic acceleration and deceleration control of the CNC controller adopts the S-curve acceleration and deceleration planning method, which accelerates and decelerates during the start and end of the motion, so that the speed changes softly to adapt to the performance of the motor and reduce the impact on the machine tool. Real-time control chart. The chart below shows the real-time control of S-curve acceleration and deceleration
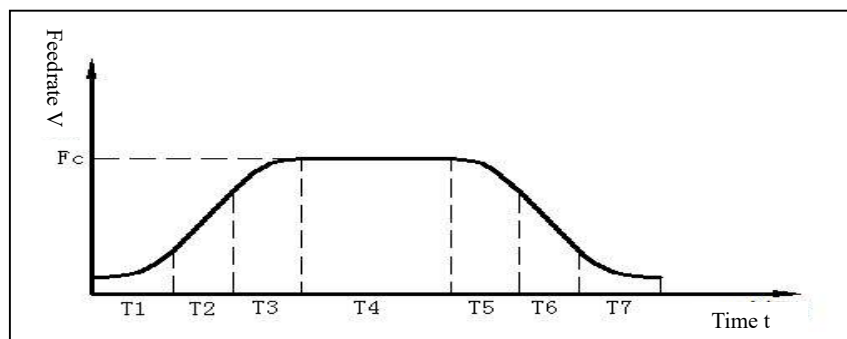
Through S-curve acceleration and deceleration, the traditional three-stage acceleration and deceleration is changed into a seven-stage acceleration and deceleration, forming an S curve.

It is divided into: acceleration section composed of T1, T2, and T3; T4 section of the feed at a constant speed V; deceleration section composed of T5, T6, and T7.

During the cutting, the change of the movement direction between the blocks will cause the rounded path of tool, as shown in the figure:
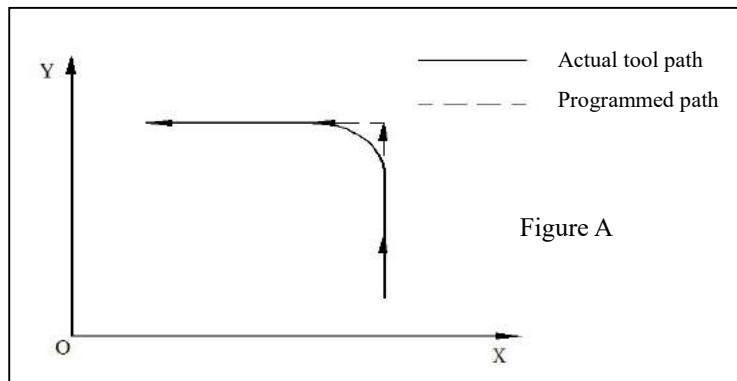
Figure A

In the circular interpolation, the radial error appears as shown in the figure:
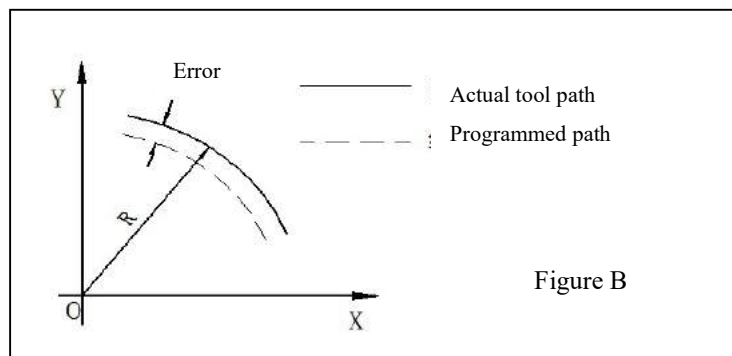
Figure B

The fillet trajectory in Figure A and the error shown in Figure B depend on the feedrate. Under normal circumstances, the greater the speed, the greater the fillet in Figure A, and the greater the error in Figure B.

Override: Use the knob switch on the machine operation panel to adjust the rapid traverse speed or cutting feedrate.

7.2   Feedrate Setting

Function and Purpose

According to the specifications of the machine tool, the maximum speed of the machine tool and the maximum speed of the cutting tool can be set. The maximum speed of the machine tool is the speed of the machine tool during non-cutting movement, which is mainly determined by the machine itself. The maximum speed of the cutting of the tool is the maximum speed that the machine tool can reach during cutting. Their speed can be specified in the following table:

Command mode	Description	Unit
Max. rapid traverse speed	The maximum rapid traverse speed must be the maximum value of all the speed setting parameters for the axis. The maximum rapid traverse speed is closely related to the ratio of the external pulse	mm/min

	equivalent numerator to denominator. This parameter must be set reasonably to avoid exceeding the motor speed range.	
Max. cutting feedrate	The maximum cutting feedrate is related to processing requirements; mechanical transmission is related to the load; the maximum processing speed must be less than the maximum rapid traverse speed. The rotary axis is affected by the converted rotation radius.	mm/min

The feedrate function has three forms: rapid traverse speed, cutting feedrate, and second feedrate.

7.2.1 Rapid Traverse Speed

Function and Purpose

It is the highest speed for the machine tool, and the rapid traverse speed of each axis can be set by parameters in the CNC system. The upper limit speed is set based on the condition of the machine tool. Please refer to the machine tool manual for details.

Description

The Maximum Time for Exact Stop Check (Parm 010166) and Positioning Tolerance (Parm 100060) during rapid traverse.

a) Parm 010166: Set the maximum time for detecting the axis positioning tolerance after rapid traverse positioning to a certain point. This parameter only takes effect when the axis parameter Parm 100060 Positioning Tolerance is not 0.

b) Parm 100060: set the allowable exact stop error for the coordinate axis rapid traverse positioning.

 0: The current axis has no positioning tolerance limit.

Greater than 0: When Parm 010166 Maximum Time for Exact Stop Check is reached, if the current axis machine coordinate still exceeds the set positioning tolerance, the CNC system will alarm.

The tool moves from the start point to the end point at the highest speed of each axis, and the rapid traverse speed is valid for G00, G28, and G29 commands.

(Note 1) The rapid traverse federate magnification is set based on the PLC editing and parameter setting, and generally has two types:

Type 1: Set the 4-stage magnification for 0%, 25%, 50%, and 100%.

Type 2: Set the magnification in 10% units, ranging from 0% to 100%.

7.2.2 Cutting Feedrate

Function and Purpose

When the CNC system processes the parts, the cutting feedrate is specified by the address F and the number. The tool moves at the cutting feedrate programmed. The cutting feedrate is valid for G01, G02, and G03 commands. There are three command modes for the feedrate unit of the milling system: G93, G94, G95.

Description

1）**Feed per minute**

Programming format

G94　；　G code of feed per minute (Group 14)

F_　；　Feedrate command　（mm/min or inch/min）

After G94 (feed per minute) is specified, the feed per minute of the tool is directly specified by the value after F. G94 is a modal code. Once G94 is specified, it will remain valid until G95 (feed per revolution) is specified. When the power is turned on, the default setting is feed per minute.

The switch on the machine operation panel can be used to to set the magnification for the feed per minute, and the magnification can be set from 0% to 150% (the override interval is determined by the MCP panel of the system). When G94 is specified, the feedrate F of the movement command specifies the movement amount of the tool per minute, with the unit mm/min (G21 mode) or inch/min (G20 mode).
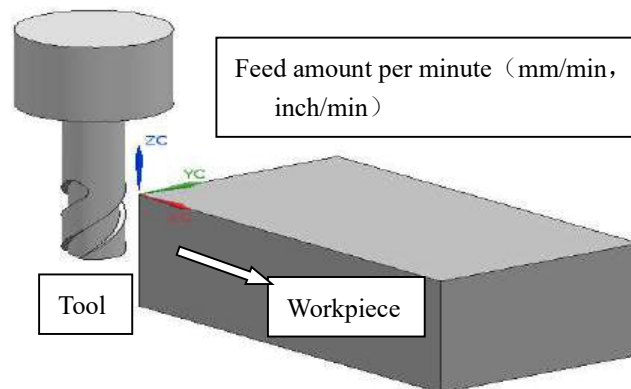
Example: feed per minute

G01　X-20　Y-10　F300　　　；Feedrate 300mm/min

2）Feed per revolution

Programming format

G95 　　；G code of feed per revolution (Group 14)

F_ 　　　；Feedrate command （mm/rev or inch/rev）

After G95 (feed per revolution) is specified, the value after F directly specifies the tool feed per revolution (G95) of the spindle. G95 is a modal code. Once G95 is specified, it will remain valid until G94 (feed per minute) is specified. With the switch on the machine operation panel, the federate override can be applied to the feed per revolution, and the magnification ranges from 0% to 150%.
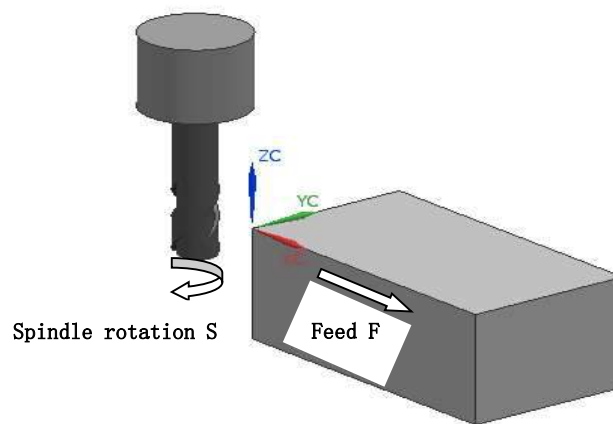
Example：G95

M03 S1000

G01 X-50 Y-20 F0.2 　　；　feedrate per revolution is 0.2mm/rev

Correspondence between feed per revolution and feed per minute: when spindle speed and feed per revolution F are given, for example spindle speed S1000 and feed F0.2.

Feed per minute（F）=1000（spindle speed）×0.2（feed per revolution）=200mm/min

3）Inverse time feed

Command format

G93 　　　；Inverse time feed command (Group 14)

F_ 　　　；Feedrate command（l/min）

By specifying G93, it becomes the inverse time specification mode. The inverse time feed function is realized by the reciprocal of the specified speed, that is, the time it takes to execute the current block. Use the F code to specify the inverse time FRN. The specified range of FRN is not restricted by the inch/metric input, and the range is from 0.001 to 9999.999.

$$FRN= \frac{1}{Time（min）} = \frac{Speed}{Distance}$$

Speed: mm/min (in metric) or inch/min (inch)

Distance: mm (in metric) or inch (in inch)

When linear interpolation (G01) and circular interpolation (G02, G03)

（1）If one program line is ended in 1 minute,

$$FRN = \frac{1}{Time（min）} = \frac{1}{1（min）} = 1$$ ；F1 is specified in program.

（2）If one program line is ended in 20 seconds,

$$FRN = \frac{1}{Time (sec)/60} = \frac{1}{20/60（min）} = 3$$ ；F3 is specified in program.

（3）When the traverse time for F0.5 is specified, the time needed for execution of one program line is,

$$Time（min） = \frac{1}{FRN} = \frac{1}{0.5} = 2$$ ；2 minutes are needed.

（4）When the traverse time for F5 is specified, the time needed for execution of one program line is,

$$Time（sec） = \frac{1}{FRN} = \frac{1}{5} = 0.2$$ ；12 seconds are needed.

Example

G01 X10

G93

G01 X20 F10   ；X axis moves 10mm after 0.1 minute (6 second)

The correspondence between the inverse time feed and the feed per minute: when the inverse time feed F is given, for example F10. In the formular FRN=0.1(min)=6(sec), then it takes 6 seconds to move the X axis 10mm. the feed rate per minute (F)=10/0.1=100mm/min.

Note

1) G93, G94, G95 are in the same group (Group 14). They are modal G codes, and mutually cancelable. G94 is the default modal.

2) When F is designated in G93 mode and the calculated speed exceeds the maximum cutting speed, the actual speed is clamped to the maximum cutting feed rate.

3) When circular interpolation is used in G93 mode, the speed is calculated from the arc radius,

not from the actual moving distance of the program line. Therefore, when the arc radius is longer than the arc distance, the actual time slows down; and when the arc radius is shorter than the arc distance, the actual time speeds up. The cutting feed in a canned cycle can also use inverse time feed.

4) The G93 command inverse time feed mode should be specified separately.

7.2.3 2nd Feedrate

Function and Purpose

The second feedrate E, which is different from the feedrate F, is generally used to limit the feedrate at the end of the block. For example, in NURBS curve interpolation, F command specifies the feedrate during interpolation, and E specifies the feedrate at the end of interpolation.

The feedrate F is a modal command, and the second feedrate E is a non-modal command, and it is used when the second feedrate is required. If E is not specified, 0 is the default.

The second feedrate is mainly used in more complicated interpolation control, and is currently only used in NURBS curve interpolation (G06.3).

(Note 1) The feedrate override has been configured in parameters 010308 to 010328, and ranges from 0% to 120%.

7.3 Feed Control Mode

Function and Purpose

During cutting and feed, the trajectory of the tool is different in different control modes. There are two cutting feed control modes for the CNC controller: exact stop and continuous cutting. For parts needing sharp edges and corners, exact stop control is used; while parts with fillet edge or small line segment programs, the continuous cutting is used.
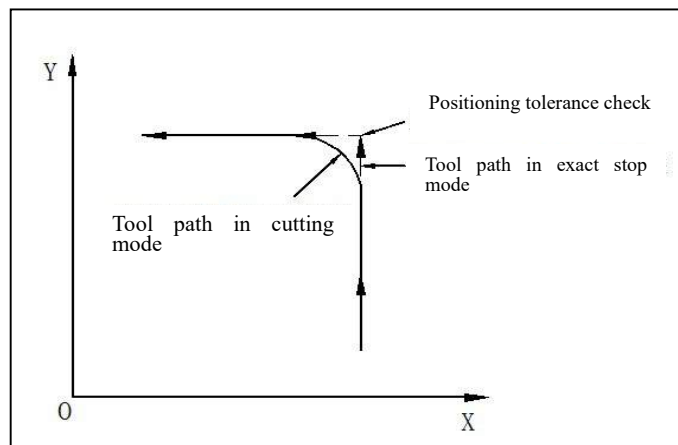
The correspondence between the two control modes is as follows:

Cutting mode	G code	Modal	Group No.	Description
Exact stop	G09	Non-modal	00	The tool decelerates at the end of the block and the positioning tolerance check is performed. Then the next block is executed.
	G61	Modal	12	The tool decelerates at the end of the block and the positioning tolerance check is performed. Then the next block is executed.
Continuous cutting	G64 (G05.1 Q0）	Modal		The tool executes the next block without deceleration at the end of the block.

Description

1） Exact stop control

Non-modal exact stop G09

G09 is a non-modal command, which is only valid in the block where the G09 is specified.

Modal exact stop G61

G61 is a modal command (group 12). Once it is specified, the function remains valid until G64 is specified.

2） Continuous cutting control G64

(1) G64 is a modal command (group 12). Once it is specified, this function is valid until G61 is specified.

(2) For continuous cutting, whether to perform exact stop check at the corner between line segments can be set by parameter G64 Parm 010169 (enable exact stop check at corner).

(3) The G64 is to set whether the exact stop at corner with G64 is performed. When this parameter is set to 1, the CNC system will enable the exact stop check at corner with modal G64.

(4) In modal G64, if the feed length of the two adjacent straight lines is less than or equal to 5mm and the vector angle is less than or equal to 36°, the CNC system will automatically adopt the arc transition, which is not controlled by this parameter.

Example

Example 1

Programming for the contour as shown in the figure: The programmed contour is required to match the actual contour.


```
%061
G54
G00 X0 Y0
G01 X30 Y20
G61 G01 Y60 F300
X90
.............
```
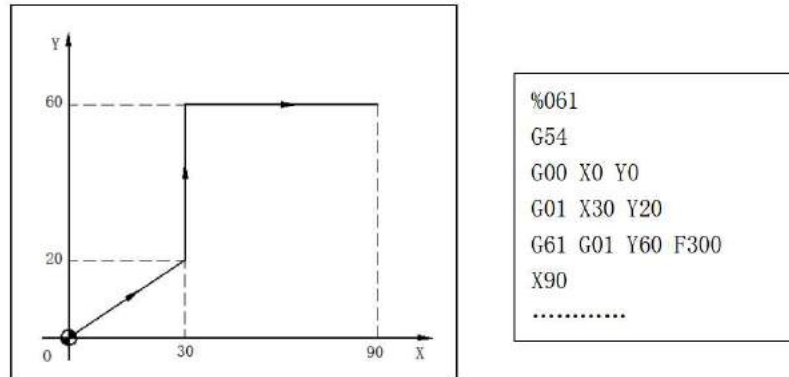
The programmed contour with G61 is the same as the actual contour.

Example 2

Programming for the contour as shown in the figure: Not stop is required between the blocks.


```
%064
G54
G00 X0 Y0
G01 X30 Y20
G64 G01 Y60 F300
X90
.............
```
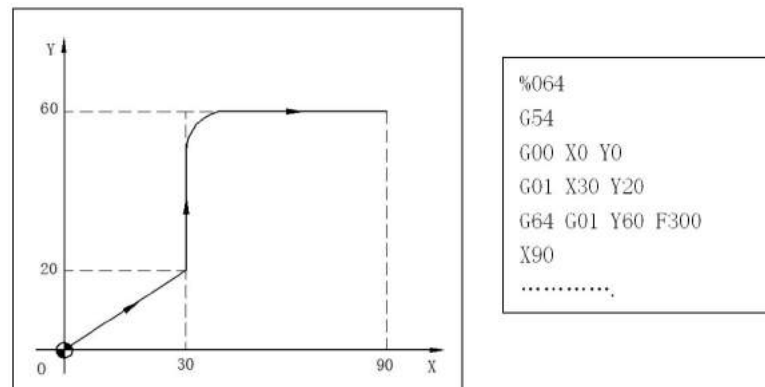
The programmed contour with G64 is different from the actual contour, and there is arc connecting between the line segments.

Note

1) The difference between G61 and G09 is that G61 is a modal command, while G09 is a non-modal command.

2) The programming axes in each block after G61 must stop exactly at the end of the block, and then the next block is continued to be executed. The programmed contour with G61 and G09 conforms to the actual contour.

3) G64 (G05.1 Q0) continuous cutting mode:

When the programmed axes after G64（G05.1 Q0）just start to decelerate (the programmed end point is not reached), the next block is executed. However, in the blocks with positioning commands (G00, G60) or with exact stop check (G09), and in blocks without motion commands, the positioning check is performed after the feedrate is decelerated to 0.

4) G61 and G64 (G05.1 Q0) are modal commands, which can be mutually cancelled.

5) The programmed contour with G64 (G05.1 Q0) is different from the actual contour. The difference depends on the F value and the angle between the two paths. The greater the F, the greater the difference.

7.4   Feedrate Control

Function and Purpose

When the CNC system is running the program in speed control, the tool needs to automatically decelerate when performing circular processing and corner processing to reduce the load on the tool, thereby reducing the impact on the machine tool, and avoiding tool marks on the workpiece.

Description

Circular speed control

(1) For circular cutting, the feedrate on the programmed path is controlled by the circular deceleration radius (Parm 040042) and the circular deceleration speed (Parm 040043).

a) Circular deceleration radius (Parm 040042): to set the maximum circular radius for deceleration. When the programmed circular radius is less than the set value, the feed cutting is executed at the set circular deceleration speed (040043). When the programmed circular radius is greater than the setting, the deceleration control is not performed. When 0 is set, the circular deceleration function is invalid.

b) Circular deceleration speed control (Parm 040043): to set the target speed for circular deceleration. When the programmed circular radius is smaller than the circular deceleration radius (040042), the feed cutting is executed based on the set value. When the programmed circular radius is greater than the circular deceleration radius (040043), the deceleration control is not performed. When 0 is set, the circular deceleration function is invalid.

(2) For the large circular radius, the speed control can be directly performed by circular

interpolation, or can be performed by Parm 040149 (Whether Arc is Discrete into Straight Line).

Parm 040149 Whether Arc is Discretized into Straight Line: when the arc can be discrete into straight line, the arc can be discretized into the connection of tiny line segments. Then for the case where the straight line meets the arc or the arc meets the arc, it can be equivalent to the straight line meets the straight line, and the speed at the junction of the two can be processed in the mode of deceleration at corner.

> 0: Turn off the function of arc discrete into straight line.

> 1: Turn on the function of arc discrete to straight line.

(3) When performing radius compensation, the automatic deceleration mode adopted for circular feedrate control is to use the parameter Parm 010044 Radius Compensation Circular Speed Strategy.

Radius compensation circular speed strategy Parm 010044: This parameter is used to adjust the circular speed after radius compensation.

> 0: Turn off the function

> 1: Speed after radius compensation = (circular radius after radius compensation / circular radius before radius compensation) * programmed speed

> 2: Speed after radius compensation = sqrt (circular radius after radius compensation/circular radius before radius compensation) * programmed speed

> 11 to 19: Speed after radius compensation = programmed speed* (0.1 to 0.9)

(Note 1) Modes 1 and 2 are the feedrate for the programmed path, which is determined by the programmed value F of the circular cutting feedrate and the circular radius, as shown in the figure. In the tool radius compensation mode, this function is valid.

1: $F \times \dfrac{Rc}{Rp}$

2: $F \times \sqrt{\dfrac{Rc}{Rp}}$

Rc: Tool center path radius

Rp: Programmed radius
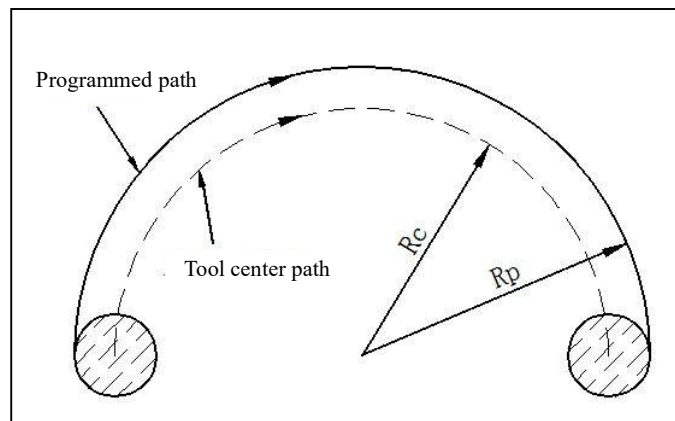
Corner speed control

When the CNC system is running the program, it uses the method shown in the figure to determine whether it is a corner or a straight line (or basically a straight line) between the program blocks.
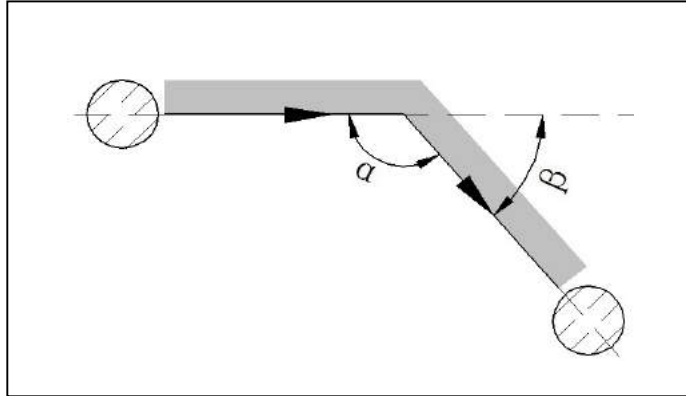
In the figure, α is the minimum internal angle (degree) of the corner to determine whether it is a corner of the connection between the line segment amnd the line segment, and the β angle is to determine whether it is a straight line or basically a straight line.

(1) Parameter related to the corner (α angle):

a) Parm 040141 Minimum internal angle for corner smoothing: During continuous small line segment interpolation, local speed reduction can be performed based on the actual programmed trajectory. When the sharp corner of the contour needs to be highlighted, the speed should be reduced to zero at the top of the sharp corner. This parameter is used to set the value of this angle. If the processed angle is less than this angle, the exact stop will be performed; if it is greater than this angle, other ways will be used to plan the speed reduction processing at this angle. If the maximum directed angle is set to 45°between the two small line segments which are allowed to be compressed and merged, this parameter should be set to 45.

b) Parm 040144 Deceleration scale factor at corner: For the polyline segment whose corner angle is greater than the minimum smoothing internal angle (040141), the feed is executed at the corner with circular transition, and the deceleration scale factor at corner can be used to control the deceleration speed at corner. The smaller the set value, the smaller the deceleration speed at corner, the smaller the corner roundnes, and the contour accuracy is theoretically higher; but the milling time at the corner becomes longer and the efficiency is reduced.

(2) Parameter related to collinear determination (β angle)

a) Maximum angle threshold in collinear determination: This parameter sets the maximum exterior angle between two adjacent line segments which are determined to be collinear. When the exterior angle is less than this value (radian value), the two line segments are collinear, otherwise they are not. 0.017 is the default value.

(3) For the length and deviation of the small line segment spline fitting, the upper limit length of the small line segment (mm) (Parm 040140), the lower limit length of the small line segment (mm) (Parm 040145), and the allowable contour error of the small line segment trajectory (Parm 040143)) are used for the spline fitting processing of the CNC system.

a) The upper limit length of the small line segment (mm) (Parm 040140): used in conjunction with the lower limit length of the small line segment to form the area range for the small line segment spline fitting.

b) Allowable contour error of small line segment trajectory (Parm 040143): During continuous small line segment interpolation, the small line segment can be compressed and merged based on the actual programmed trajectory. This parameter is to set the allowable contour error between the compressed and merged small line segment and the original programmed trajectory. When the contour error exceeds the set value, it will not be compressed.

c) The lower limit length of small line segment (mm) (Parm 040145): During spline interpolation, the spline smoothing (fitting) needs to be performed on the small line segment based on the actual programmed trajectory. This parameter is to set the shortest length of small line segment where the smoothing is allowed. If the length of the small line segment is less than the set value, the smoothing of this line segment is not performed.

8   Position Command Function

8.1   Mode I of Absolute Command and Incremental Command（G90/G91）

Function and Purpose

There are two modes to define the position of the target point: absolute position and incremental position.

For easier programming, the definition of the position should fit the workpiece drawing. When a fixed reference point is given in the drawing, it is more convenient to use absolute programming; and when the drawing size is given by the spacing between the contour nodes, it is more convenient to use incremental programming.

There are two modes of absolute command and incremental command in the CNC system: mode I and mode II. The mode 1 can be executed in lathe ystem (T) and milling system (M), and the mode II can only be executed in lathe system (T).

Command Format

Mode I of absolute command and incremental command.

G90 IP_   Absolute programming

Parameter	Meaning
IP	By absolute command G90, the traverse command is always executed with the workpiece coordinate system as the starting point

G91 IP_   Incremental programming

Parameter	Meaning
IP	By incremental command G91, the traverse command is always executed with the current point as the starting point

Description

Absolute command G90: define the address of the target point. The value after the address word is the coordinate value in the workpiece coordinate system;

Incremental command G91: define the address of the target point. The value after the address word is the directional distance value from the target point to the previous point;

When using G90, G91 commands, the coordinate movement values after the commands can be specified in absolute mode or incremental mode, but when defining the circular trajectory, the radius R and circle center I, J, K values must be incremental values. G90 and G91 are modal

commands and can be mutually cancelled. G90 is the default.

 Example

Example 1: The origin is the starting point, programming with G90 and G91; the tool is required to move from the origin to points 1, 2, and 3 in order.
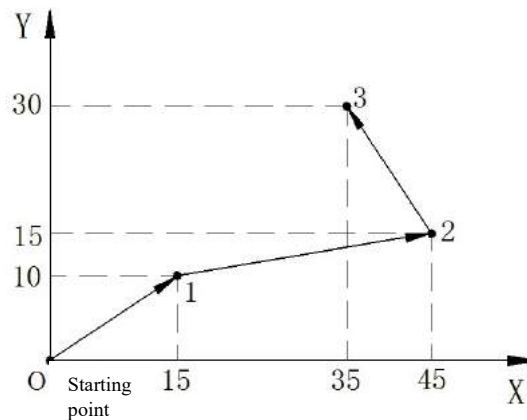
G90 programming
%
G54
G00 X0 Y0 M3 S1000
G90 G01 X15 Y10 F400
X45 Y15
X35 Y30
M30

G91 programming
%
G54
G00 X0 Y0 M3 S1000
G91 G01 X15 Y10 F400
X30 Y5
X-10 Y15
M30

 Note

1) When using the G90 command to run the program, the system is in the absolute mode. During running of the traverse command, the workpiece coordinate system is always used as the starting point to move to the workpiece coordinate system position specified by the program, regardless of the current position.

2) When using the G91 command to run the program, the system is in the incremental mode. During running of the traverse command, the current position is always the starting point and the value specified in the program is used as a relative value for moving.

3) G90 and G91 can be used in the same block, but pay attention to the difference caused by their order.

4) The mode 1 of absolute command and incremental command can be executed in both lathe system and milling system.

8.2 Mode II of Absolute Command and Incremental Command（X, Z/U, W）（T）

Function and Purpose

There are two modes of absolute command and incremental command in the CNC system: mode I and mode II. The mode 1 can be executed in lathe ystem (T) and milling system (M), and the mode II can only be executed in lathe system (T).

When using mode II for programming, the switching between absolute position and incremental position is simple. It is especially suitable for mixed programming, but it occupies the axis name definition of the auxiliary axis, so this mode is more suitable for lathe system.

Command Format

If user needs to use the mode II of absolute command and incremental command in lathe syste, it is necessary to configure the parameter 040033 [UVW incremental programming enable] in the channel parameter.

When the parameter 040033 is specified as 1, the incremental value programming can use U, V, W to represent the incremental values on X, Y, Z axis respectively. At this time, the absolute value is still specified by X, Y, and Z with G90. G90 is the default and does not need to be defined. The format of mode II is: U, V, and W are for incremental programming, and X, Y, and Z are for absolute programming. Of course, it can also be executed if the mode I is used for programming at this time.

When parameter 040033 is specified as 0, the mode I is enabled. At this time, U, V, W definition for incremental position cannot be executed.

The definition of channel parameter 040033 is shown in the figure below:

040030	通道的缺省进给速度(mm/min)	2000.000000	保存
040031	空运行进给速度(mm/min)	2000.000000	保存
040032	直径编程使能	0x0	复位
040033	UVW增量编程使能	1	保存
040034	倒角使能	1	复位
040035	角度编程使能	1	保存

Description

1. Linear incremental programming

G01 X10 Z5   Move to 10 on X of the workpiece coordinate, and 5 on Z of the workpiece coordinate;

G01 U10 W10   Move in the positive direction 10 on X based on the current coordinates, and

move in the positive direction 10 on Z based on the current coordinates.

2. Circular incremental programming

For circular programming, in addition to specifying the end position by absolute programming, incremental programming can also be used to specify the incremental value in each direction of the arc (UVW is also used), and it can also use XYZ and UVW for mixed programming.
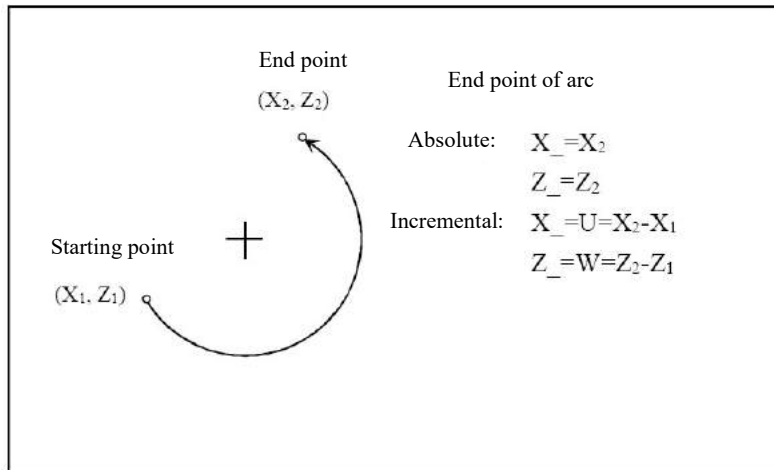
Example

Tool moves to Q from P (The diameter value is on X axis)

Absolute command：G90X400Z50
Incremental command：1.　G91X200Z-400
　　　　　　　　　　　2.　U200W-400
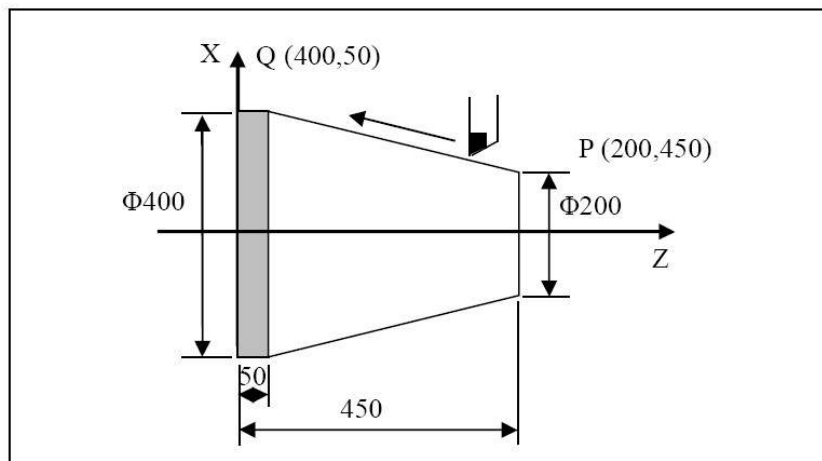
Note

1）UVW programming is only used for lathe system.

2）G91 is a modal command, and cannot be canceled by itself. The lines after G91 are all in G91 programming, and its modal can be canceled by G90.

8.3　Diameter Programming and Radius Programming（G36/G37）（T）

Function and Purpose

The workpiece on a CNC lathe is usually has a shape of rotating body, and its size on X axis can be specified both by diameter mode and radius mode.

Radius programming: Use the radius value to define the X axis position of the target point. There are two types: absolute radius value and incremental radius value. The absolute radius value is the radius coordinate value on X axis; the incremental radius value is the incremental radius value on X axis.

Diameter programming: Use the diameter value to define the X-axis position of the target point. There are also two types: absolute diameter value and incremental diameter value. The absolute diameter value is the diameter coordinate value on X axis; the incremental diameter value is the incremental diameter value on X axis.

Diameter programming is more in line with the drawing of parts, so the diameter programming is the default for lathe machine. The value in diameter programming is twice the value in radius programming.

The setting of diameter programming is often used on the X axis of the lathe, and other axes perpendicular to the rotary axis can also be set as the diameter programming axis when needed, such as the Y axis of some turning centers.

Command Format

G36;　　　Diameter programming

G37;　　　Radius programming

G36 and G37 can be used to switch between diameter and radius programming.

Description

When the channel parameter 040032 is set to 0X1, the X-axis diameter programming of the machine tool is enabled, and G36 or G37 can be used to set the diameter programming or radius programming.

| 040032 | 直径编程使能 | 0x1 | 复位 |

说明： 两个床加工工件的径向尺寸通常以直径方式标准，因此编程时可以直接使用标注的
直径方式编写程序。此时直径上一个编程单位的变化，对应径向进给轴半个单位的
移动量。该参数用来选择当前通道的编程方式
0x0 半径编程方式；0x1 X轴直径编程方式开；0x2 Y轴直径编程方式开
0x3 X、Y轴直径编程方式开。
注意：此参数开并且编程模式为G36时，直径编程才生效。

After the diameter programming is enabled, the default is G36 command (diameter programming). After switching to radius programming with G37, it is necessary to return to diameter programming with G36.

When using diameter programming, it is recommended to turn on the diameter display enable. NC parameter 000065 is the parameter of Diameter Display Enable. For example, when the parameter is set to 0X1, the X axis diameter display is enabled, and the X position value on the system display interface is in diameter. The tool offset and the offset and wear on the tool compensation interface are also displayed in diameter.

Example
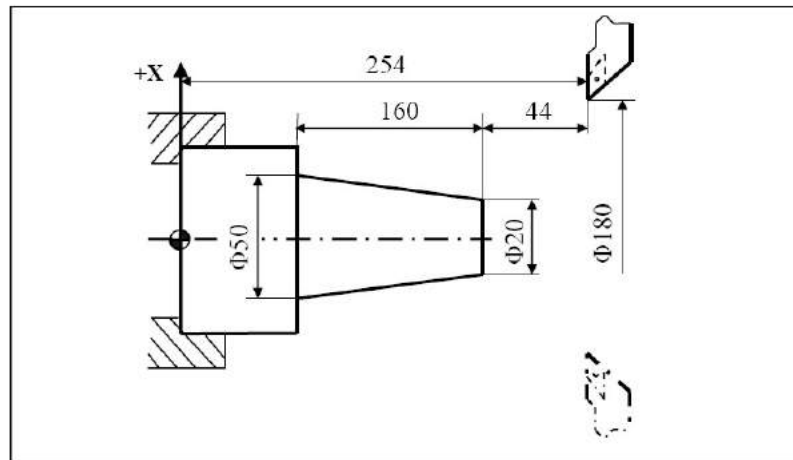
Diameter programming	Radius programming
%3341	%3342
N1 G92 X180 Z254	N1 G92 X90 Z254
N2 G36 G01 X20 W-44	N2 G37 G01 X10 W-44
N3 U30 Z50	N3 U15 Z50
N4 G00 X180 Z254	N4 G00 X90 Z254
N5 M30	N5 M30

Note

1) Z axis command input has nothing to do with diameter and radius programming;

2) When G02 and G03 are specified, the R, I and K are the radius values;

3) R, the feed amount on X axis used in a symple canned cycle, is specified in radius;

4) For the lathe and turning center systems, the default is G36 diameter programming;

5) The axial feedrate is specified by the change of radius.

8.3   Inch/Metric（G20/G21）

Function and Purpose

User can select the unit of size through G20, G21.

Command Format

G20 IP_   Inch

Parameter	Meaning
IP	After G20 is specified, the traverse distance after G20 is in inch.

G21 IP_   Metric

Parameter	Meaning
IP	After G21 is specified, the traverse distance after G21 is in metric.

Description

Only the command unit can be switched with G20/G21, but not the input unit.

G20/G21 only works on the linear axis, and is invalid to the rotary axis.

Note

1) G20 and G21 are modal commands and can be mutually cancelled. The default is G21 after the system is powered on.

2) The unit of data input in the G code is not related to the unit of data displayed on the HMI interface. G20 and G21 are only used to select the unit of data input in processing G codes, and cannot change the data unit displayed on the HMI interface.

3) Parameter 000025 [Inch/Metric] in NC parameters is used to set the data unit of the coordinate displayed on the interface.

9　Delay Function

9.1　Delay Function

Function and Purpose

This function can suspend the movement of the machine tool by G04 X_ or P_ command to delay the start of the next program. The delay time unit is X for seconds and P for milliseconds.

Command Format

G04 X__ / P__

Parameter	Meaning
X/P	Dwell time
X	Unit: second
P	Unit: millisecond

Example

Command	G04 X5	G04 X2.5	G04 P2000	G04 P1000.5
Delay seconds	5	2.5	2	1

Description

1) When the delay time is specified by X, the decimal point command is valid.

2) When the delay time is specified by P, the commands after the decimal point are ignored.

3) If there is a cutting command in the previous block, the delay time will be started to be calculated after the block is stopped.

4) When the machine tool is locked, the delay command is also valid.

5) During the automatic operation, user can specify G04 to suspend the tool feed, and the subsequent blocks will be executed automatically after the dwell time is reached.

6) The command value range of the dwell time: X minimum value is 0, maximum value 2000; P minimum value is 0, maximum value 2147400.

Note

1) When using this function, please issue the X or P command after the G04 command to make it clear that X or P is executed.

2) The number followed by X cannot exceed 2000, otherwise the system will alarm "Syntax-Illegal number".

10 Coordinate System

10.1 Overview

Function and Purpose

In processing of machine tool, when a position reached by the tool is preset, the tool can move to the specified position. This position should be given by the coordinate value in a certain coordinate system, and the coordinate value is specified by the programmed axis. In this way, the required workpiece can be processed based on the program.

When the 3 programmed axes are X, Y and Z axes, the coordinate values are specified as follows:

X_Y_Z_: This command is called size word.
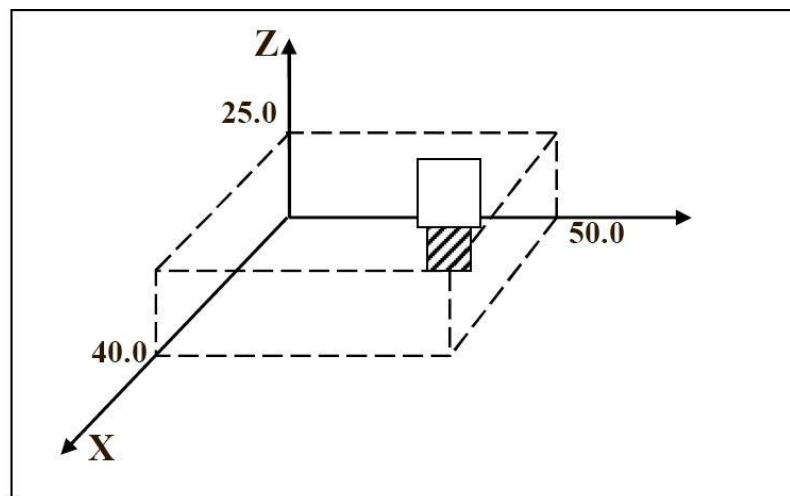
The tool position specified by X40.0Y50.0Z25.0

Description

The system supports the following three coordinate systems for user to choose: (1) machine coordinate system; (2) workpiece coordinate system; (3) local coordinate system.

The machine coordinate system is the fixed coordinate system of the machine tool, which represents the position inherently determined by the machine tool.

The workpiece coordinate system is the coordinate system used by the programmer during programming, and generally the reference point on the workpiece is regarded as the coordinate origin.

The local coordinate system is a coordinate system created on the workpiece coordinate system in order to simplify the creation of part of the processing program.

Note

The local coordinate system (G52) is valid in the coordinate system specified by workpiece coordinate systems 1 to 6.

10.2   Machine Coordinate System

Function and Purpose

The machine zero is a fixed mechanical point on the machine tool. The coordinate system established by the machine tool manufacturer with this point as the origin is referred to as the machine tool coordinate system.

After turning on the power, manual reference point return is executed to establish the machine coordinate system. Once the mechanical coordinate system is established, it remains unchanged until the power is cut off.

Command Format

G53 X__ / Y__ / Z__ /； Absolute size (target position in machine coordinate system)

Description

1) G53 is a non-modal command, which is only valid in the block where the machine coordinate system is specified. When user needs to execute direct machine tool coordinate system programming, G53 must be specified in the current line;

2) The target position specified by G53 cannot be in incremental programming, only absolute programming can be used. When the incremental command (G91) is specified, the G53 command is ignored.

3) When G53 command is specified, the tool radius compensation, tool length compensation, tool nose radius compensation and other functions are cleared.

4) Before specifying the G53 command, the machine coordinate system must be set. Therefore, the reference point must be returned manually or automatically by the G28 command after power-on. When an absolute encoder is used, this operation is unnecessary.

5) Machine coordinate system is set before calling G53. The system must establish the machine coordinate system through the reference point return.

6) The system reference point does not necessarily coincide with the origin of the machine coordinate system, and the relationship between them is shown in the figure below.

Machine origin

Reference point
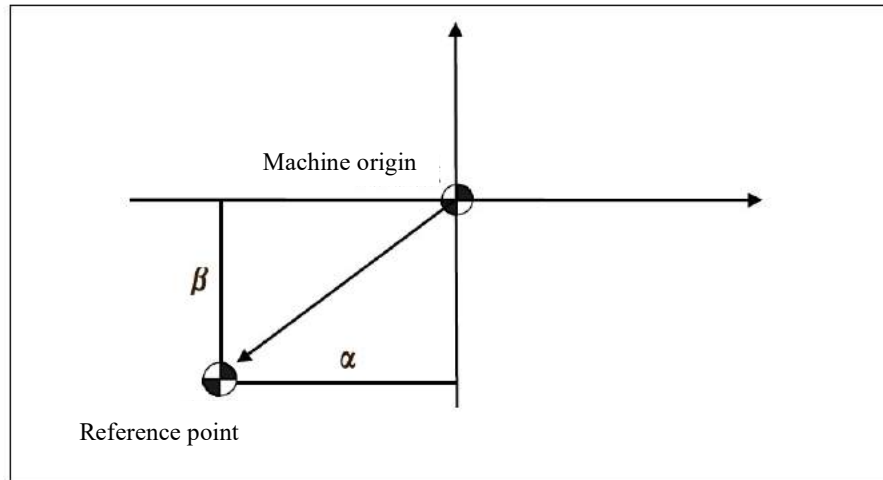
10.3 Workpiece Coordinate System

Function and Purpose

The workpiece coordinate system is used by the programmer during programming. The programmer selects a known point on the workpiece as the zero of the workpiece coordinate system. The introduction of the workpiece coordinate system is to simplify programming and reduce calculations.

There are three ways to establish a workpiece coordinate system:

(1) Use the workpiece coordinate system setting command (G92) to establish a workpiece coordinate system;

(2) Use the workpiece coordinate system selection commands (G54~G59) to establish a workpiece coordinate system;

(3) Use the extended workpiece coordinate system selection command (G54.01~G54.60) to establish a workpiece coordinate system;

For the lathe, in the absolute tool offset compensation mode, the workpiece coordinate zero can be set with the T command.

In addition, the zero of workpiece coordinate system established in the above ways can be modified by G10 command, and a new workpiece coordinate system can be formed.

10.3.1 Workpiece Coordinate System Setting (G92)

Function and Purpose

The meaning of the G92 command is to determine the position of the workpiece origin through the coordinate value of the tool location in the workpiece coordinate system. The position of the workpiece coordinate system changes with the change of the tool position when this command is

executed. The prerequisite for correct processing is that the operator must correctly set the tool position on the set coordinates through the tool setting.

Command Format

G92   IP(X…Y…Z…)_;

Parameter	Meaning
IP	The directed distance from the origin of the coordinate system to the starting point of the tool

Before executing the program containing the G92 command, user must perform the tool setting operation to ensure that the workpiece coordinate system origin established by the G92 command is consistent with the programmed origin.

Example

Example 1: use G92X30.0Z20.0; command to set the coordinate system (tool nose is the starting point of program)

Example 2: use G92X300.0Z1000.0; command to set the coordinate system (The starting point of program is the reference point on shank)

To move the tool nose to the command position in absolute programming, perform the tool length compensation for the difference between the tool nose and reference point

Note

1) When executing this block, only the workpiece coordinate system is established, and the tool does not move;

2) G92 is a non-modal command;

3) After the length compensation is added, if the coordinate system is set by the G92 command, the set coordinate system is the coordinate position before compensation. However, the G92

command must not be in the same line with the G command which causes the change of length compensation vector, for example, it cannot run in the following blocks;

- The block where G43/G44 is specified;

- The block where H code is specified in the G43/G44 mode;

- The block where G49 is specified in G43/G44 mode;

- In the G43/G44 mode, the block where is vector is restored after the compensation vector is temporarily cancelled by G28, G53, etc.

In addition, when the workpiece coordinate system is set by G92 command, the blocks before it stop, and the tool length compensation amount selected by MDI cannot be changed.

10.3.2 Workpiece Coordinate System Selection G54～G59（G54.X）

Function and Purpose

The six standard workpiece coordinate systems preset in the system can be selected through the six commands G54 to G59. The standard workpiece coordinate system zero is manually input through the HMI interface.

Description

In the control system, G54～G59 commands can be used to select the current workpiece coordinate system among 6 preset workpiece coordinate systems. When there are many workpiece sizes and there are relatively multiple different marking datums, the coordinate values of several datum points in the machine coordinate system can be input into the system in advance through MDI as the coordinate origin of G54～G59, and the system will remember these points automatically. Once one of the G54～G59 commands is executed, the origin of the workpiece coordinate system is the current programmed origin, and the absolute coordinates in the subsequent blocks are the values relative to the programmed origin.

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

Example

G54 G90 G00 X100 Y100 Z5；   Position to X100 Y100 Z5 position in G54 coordinate system

G55；   Set G55 as the current coordinate system

G00X30Y30；   Move to the point X30Y30 in G55

G52X45Y15；   Establish a local coordinate system G52 in the current G55

coordinate system

G00G90X35Y20；   Move to the point X35Y20 in G52

G53X35Y35；   Move to the point X35Y35 in G53 (machine coordinate

system)

> **⚠ Note**

1) The workpiece coordinate systems G54 to G59 are established after the power is turned on and the reference point is returned.

2) G54 coordinate system is automatically selected when the power is turned on.

3) Even if the workpiece coordinate system is switched by G54 to G59 and G54.X, the tool radius compensation of the designated axis will not be cancelled.

4) G54 to G59 and G54.X are modal commands.

10.3.3 Extended Workpiece Coordinate System Selection（G54.x）

Function and Purpose

In addition to the six workpiece coordinate systems specified by G54 to G59 for users to choose, the milling system also provides the extended workpiece coordinate systems.

The system provides 60 extended workpiece coordinate systems for users to choose.

Command Format

G54.X；To select No. X extended workpiece coordinate system

Parameter	Meaning
X	The index number of the extended workpiece coordinate system, ranges from 1 to 60, a total of 60.

Example

%1234

G54.18

G90 G00 X100 Y100 Z50; Position to X=100 Y=100 Z=50 in the 18th extension
coordinate system

M30

Note

Once the workpiece coordinate system is selected, it remains valid until it is selected by another workpiece coordinate system

Select the standard workpiece coordinate system 1 (G54) when the power is turned on

G54.1 P1 additional workpiece coordinate system 1

G54.1 P2 additional workpiece coordinate system 2

⋮

G54.1 P60 additional workpiece coordinate system 60

10.3.4 Workpiece Coordinate System Modification (G10)

Function and Purpose

Generally, the operator can modify the external workpiece origin offset or the workpiece origin offset through the HMI interface to change the workpiece coordinate system. And with this function, G10 command can be used to realize the above changes in the program. For details, see "Chapter 13 Programmable Data Input"

Command Format

G10 L2 Pp IP_;

Parameter	Meaning
P=0	External workpiece zero offset value
P=1 to 6	Workpiece zero offset of workpiece coordinate systems 1 to 6
IP	the workpiece origin offset of each axis in the absolute mode (G90); and the offset added to the workpiece zero set originally of each axis in the incremental mode (G91), and the result of the addition of the two is the new workpiece zero offset

Note

After the external workpiece zero offset is set, when the coordinate system is set with G92, the coordinate system is not affected by the external workpiece zero offset. For example, when "G92X100.0Y80.0" is commanded, the coordinate system where the current position of the tool is X=100.0, Y=80.0 is specified.

10.4 Local Coordinate System Setting (G52)

Function and Purpose

When programming on the workpiece coordinate system, for convenience, user can create a sub-workpiece-coordinate-system in the workpiece coordinate system. Such a sub-coordinate-system is called a local coordinate system.

Command Format

G52 IP_;   To set local coordinate system

G52 IP 0;   To cancel coordinate system

Parameter	Meaning
IP	To specify the origin of local coordinate system

Description

1) Use G52 IP_; command to set the local coordinate system in all workpiece coordinate systems. The IP value is the coordinate position in the workpiece coordinate system, and the position in the workpiece coordinate system is the origin of the local coordinate system.

2) Once the local coordinate system is set, the movement command of the specified axis will be the coordinates in the local coordinate system;

3) If user needs to cancel the local coordinate system or specify the coordinate value in the workpiece coordinate system, coincide the origin of the local coordinate system with the origin of the workpiece coordinate system.
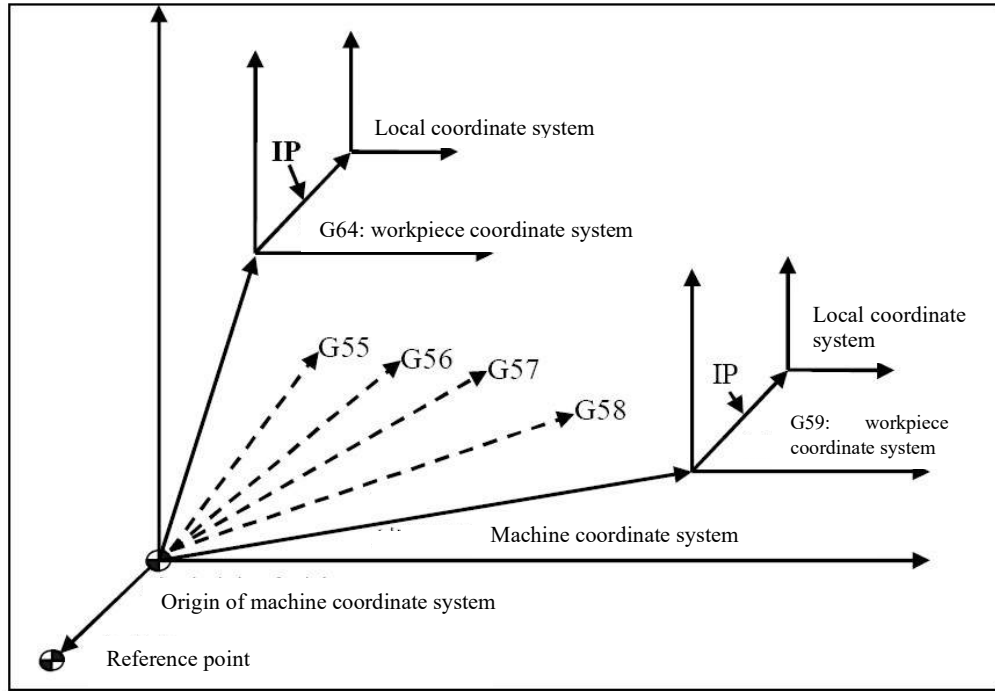
Local coordinate system

Example

%1234

G55；Select G55, assuming that the coordinates of G55 in the machine tool coordinate system are (10, 20)

G1 X10Y10F1000；Move to the machine coordinate system (20, 30)

G52 X30Y30；Establish a local coordinate system based on all workpiece coordinates, the origin of the local coordinate system is (30, 30)

G1 X0Y0；Move to the origin of the local coordinate system (the current position of the machine tool coordinate system is (40, 50))

G52 X0Y0；Cancel local coordinate system setting, and the system restores to G55 coordinate system

G1 X10Y10；Move to the machine coordinate system (20, 30)

M30

Note

1) When the manual reference point return of an axis is performed, the zero of the local coordinate system of the axis is consistent with the zero of the workpiece coordinate system. The result is the same as the result of the command G52 α0. (A: axis returning to reference point)

2) The local coordinate system setting does not change the workpiece coordinate system and the machine coordinate system.

3) When the workpiece coordinate system is set by G92 command, if the coordinate values of all axes are not commanded, the local coordinate system of the axis for which the coordinate value is not specified will not be cancelled, and will remain unchanged.

4) The offset in tool radius compensation is temporarily cleared with G52.

5) After the G52 block, the motion command is immediately specified in absolute mode

10.5　Coordinate Plane Selection（G17，G18，G19）

Function and Purpose

Coordinate plane selection G17/18/19 commands are used for the selection of the machining plane in operations such as circular interpolation, tool radius compensation (M), and rotation transformation (M).

Description

G code	Plane	X	Y	Z
G17	XY plane	X axis or its parallel axis	Y axis or its parallel axis	Z axis or its parallel axis
G18	ZX plane			
G19	YZ plane			

- X, Y, Z are determined by the axis address appearing in the block where G17, G18 or G19 is specified;

- When the axis address is omitted in the G17, G18 or G19 block, it is considered that the basic 3-axis address is omitted;

- When G17, G18, G19 are not specified in the block, the plane remains unchanged;

- The movement command has nothing to do with the plane selection;

- Parameter 000012 is used to set the tool length compensation axis;

Example

G17X_Y_　　　；XY plane

G18X_Z_　　　；XZ plane

G19Y_Z_　　　；YZ plane

X_Y_　　　；YZ plane is not changed

G17　　　；XY plane

G18　　　；XZ plane

G18Y_　　　；XZ plane, the movement on Y has no relationship with planes.

Note

G17, G18, and G19 are modal functions, which can be mutually cancelled. The default modal is

G17 after power-on. The movement command has nothing to do with plane selection. For example, when G17 G01 Z10 is commanded, the Z axis still moves.

10.6 Machine Origin and 2nd, 3rd, 4th, and 5th Reference Points

Function and Purpose

There is a fixed mechanical point on the machine tool which can be used as the reference point of the machine tool. This point is referred to as the machine origin, and its position is determined by the dog of reference point return or the grating zero point. The machine origin is the reference point of the basic mechanical coordinate system, and is a fixed mechanical point determined by the reference point return. The reference point return function can accurately move the tool to the set fixed position. The 1st, 2nd, 3rd, 4th, and 5th reference points are the positions preset with the coordinate values in the parameters based on the origin of the basic machine coordinate system.

The first reference point in this system is usually set to 0. The 2nd, 3rd, 4th, and 5th reference points are usually used for each buffer point and tool change point for the tool change action, so as to realize the correct tool change action. Therefore, when the parameter values of the second, third, fourth, and fifth reference points are set, please do not modify them at will, otherwise, the tool cannot be changed, the tool change action error or dangerous situation may occur.

The values of the 2nd, 3rd, 4th, and 5th reference points are set by parameters. The values are set based on the style of the magazine or the requirements of the machine tool manufacturer. Usually, the setting is completed before the machine leaves the factory.

Take axis 0 as an example, through the coordinate values set in the coordinate axis parameters (100017, 100021 to 100024), up to 5 reference points of the machine coordinate system can be specified.

Machine origin
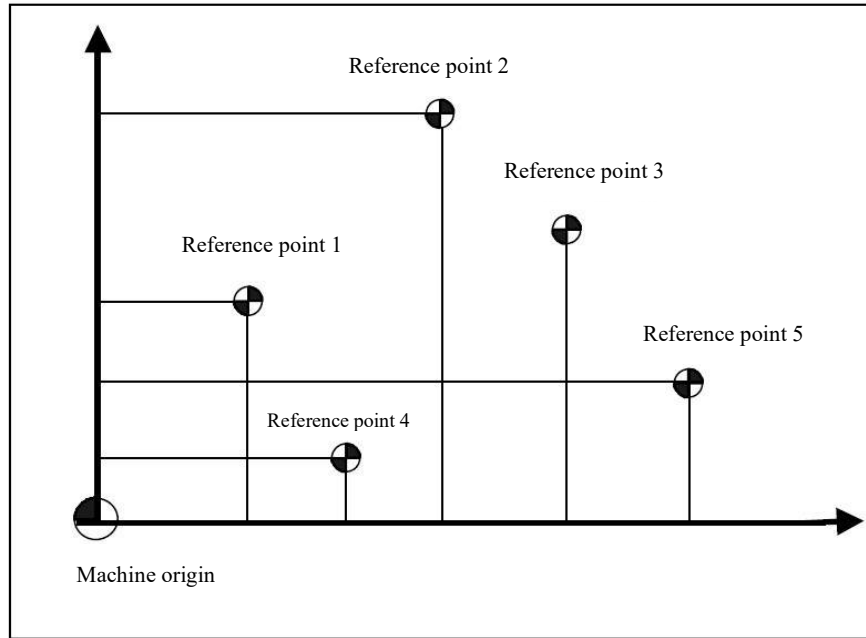
10.7   Reference Point Return (G28/G29)

Function and Purpose

Through Automatic reference point return, the position of the machine tool can be corrected to make the positions for machining or tool change consistent.

When returning to the reference point (G28) automatically, the tool quickly moves to the reference point after passing the intermediate point, and the specified intermediate point is stored by CNC.

When returning from the reference point (G29), it will automatically move to the specified point along the specified axis through the intermediate point.

The process of the reference point return and the return from reference point is shown in the figure below:

A Starting point of reference point return

C Target point of reference point return

B Intermediate point

Automatic reference point return （G28）A→B→R

Return from reference point （G29） R→B→C
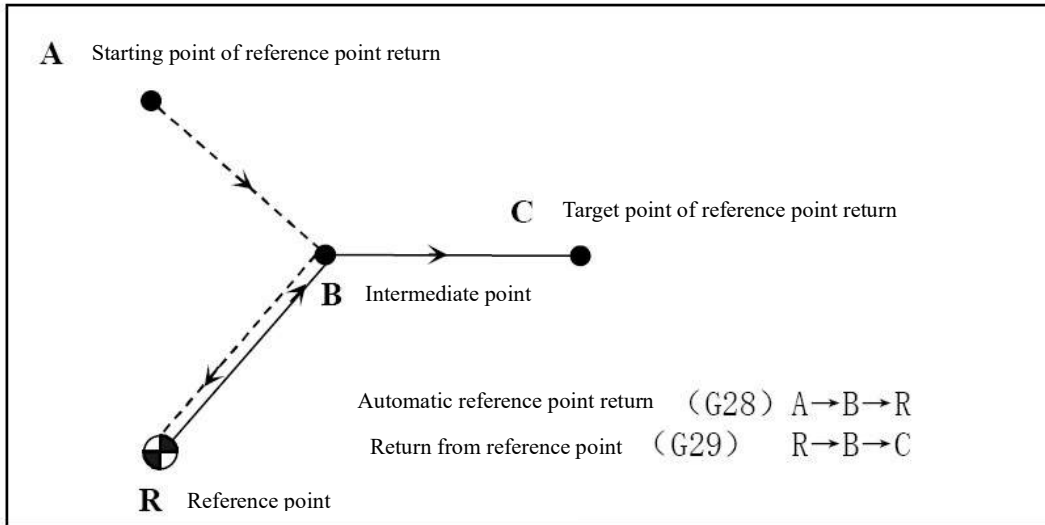
R Reference point

(Note 1) The return from the reference point (G29) is only valid for the M series.

Command Format

G28　IP_；　1st reference point return (automatic reference point return)

Parameter	Meaning
IP	In absolute mode (G90), to specify the absolute position of the intermediate point. In incremental mode (G91), to specify the distance between the intermediate point and the starting point. There is no need to calculate the specific amount of movement between the intermediate point and the reference point.

(Note 1) The coordinates of the IP command are the values in the workpiece coordinate system. When the automatic reference point return command is executed, only the axis with the designated intermediate point will move, and the axis without the designated intermediate point will not move.

G29 IP_；　Return from reference point

Parameter	Meaning
IP	In absolute mode (G90), to specify the position of the target point. In the incremental mode (G91), the intermediate point of G29 must be the intermediate point set by G28 last time, and G91 is executed for the coordinate value after G29 on the basis of the intermediate point of G28.

(Note 1) The coordinates of the IP command are the values in the workpiece coordinate system, and the intermediate point is the one previously specified with G28 and G30.

📖 **Description**

Parameter setting

G29 related parameters are as follows (only the channel 0 parameters are listed)

Parameter	Description
040112	G28 intermediate point takes effect one time

040112：This parameter is to set whether the G28 intermediate point takes effect one time or multiple times in the subsequent machining codes. If it takes effect multiple times, then the G28 intermediate point can be returned with G29 multiple times; if it takes effect one time, then it only works on the first G29 after the G28.

    0：G28 intermediate point takes effect one time.

    1：G28 intermediate point takes effect multiple times.

Example
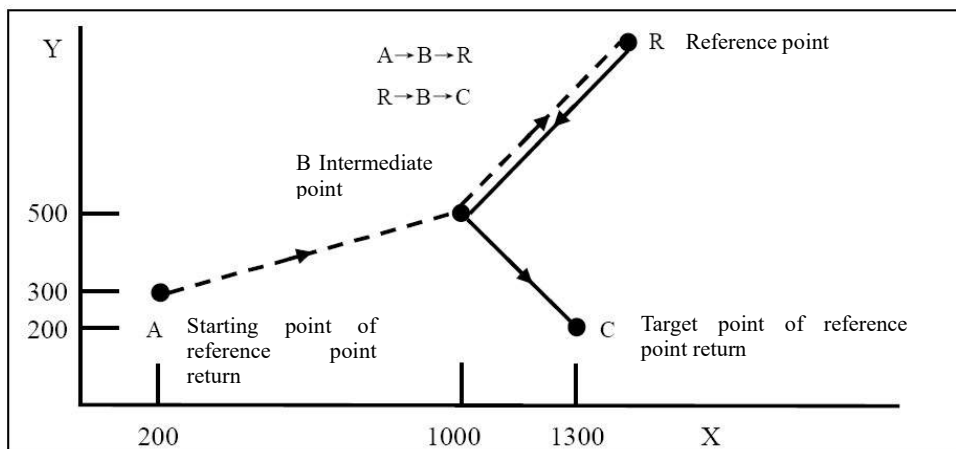
G54

G00 X200Y300

G28 G90 X1000.0 Y500.0； The program from A to B. After passing the intermediate point B, move to the reference point R

T6M06；Change tool at the reference point

G29 X1300.0 Y200.0；From the reference point R through the middle point B, move to the C specified by G29

M30；Program ends

10.8   2nd, 3rd, 4th,5th Reference Point Return (G30)

Function and Purpose

With G30 P2 (P3 P4 P5) command, the 2nd, 3rd, 4th or 5th reference points (origin) return can be performed.

Command Format

Return to the 2nd, 3rd, 4th or 5th reference points

G30 P2 IP_   ; Return to the 2nd reference point (P2 can be omitted)

G30 P3 IP_   ; Return to the 3rd reference point

G30 P4 IP_   ; Return to the 4th reference point

G30 P5 IP_   ; Return to the 5th reference point

Parameter	Meaning
IP	In absolute mode (G90), to specify the absolute position of intermediate point; In incremental mode, to specify the distance from intermediate point to the starting point. There is no need to calculate the specific amount of movement between the intermediate point and the reference point.

Description

Parameter setting:

Accurate reference point return enable

For the reference point return with G28, G30, the reference point return mode can be set to the accurate return by parameters. In this mode, the zero pulse position is required to be found when the reference point return is performed with G28, G30. The reference point return with G28 and G30 is of the normal mode by default, and the zero pulse doesn't need to be found, with the related parameter set to 0. When it is necessary to return to the reference point with high accuracy, please adopt the accurate reference point return and set the corresponding parameter to 1.

The relevant parameters of accurate reference point return are as follows (only the parameters in channel 0 are listed):

Parameter	Description
040110	G28 Z pulse search Enable
040111	G28/G30 rapid traverse positioning selection

040110: This parameter is to set whether to search for Z pulse when reference point return is

performed with G28. G28 Z Pulse Search Enable is only for incremental motors. For absolute motors, this parameter must be set to 0; for incremental motors, it can be set to 0 or 1.

> 0: search Z pulse.
>
> 1: not search Z pulse.

040111: This parameter is to set whether G28/G30 returns to the machine zero at the G00 rapid traverse speed after moving to the machine reference point at the speed of G01.

> 0: Return to the machine zero at the speed of G01.
>
> 1: Return to the machine zero at the speed of G00.

⚠️ **Note**

1) Tool length compensation and tool radius compensation cannot be canceled with G28/G29/G30.

2) When G28/G30 is executed, the tool completes the cancellation of tool radius compensation during the process from the starting point to the intermediate point; during the process from the intermediate point to the reference point, there is no tool radius compensation. When G29 is executed, there is no tool radius compensation during the process from the reference point to the intermediate point; the tool radius compensation is restored during the process from the intermediate point to the target point.

3) The intermediate point specified by G28/G30 in the tool length compensation will be cumulatively calculated with the length compensation, and the reference point to be returned finally will not be cumulatively calculated with the length compensation. Whether the tool length compensation will be restored after G28/G30 is executed is determined by NC parameter 000014.

The NC parameter 000014 is as follows:

Parameter	Description
000014	Whether to restore tool length compensation after G53/G28

This parameter is to set whether to automatically restore the tool length compensation function after G53 command is executed.

> 0: After executing G53 command, the tool length compensation function is not restored automatically.
> 1: After executing G53 command, the tool length compensation function is automatically restored.

Note: G30 is the same as G28.

4) Whether the target point specified by G29 in the tool length compensation and the intermediate point stored by G28 or G30 will be cumulatively calculated with the length compensation is related to whether the NC parameter 000014 and G28/G30 commands are before or after G43.

The parameter is set to 0. When the G28 command is before G43, the intermediate point and

target point of G29 operation will accumulate with the tool length compensation;

The parameter is set to 1. Whether the G28 command is before or after G43, the intermediate point and target point of G29 operation will be accumulated with tool length compensation;

5) When G28, G30, and G29 are executed, all axes move to the reference point or the intermediate point at the G00 rapid traverse speed or G01 speed, and the speed can be controlled by the rapid traverse override switch or the feed override switch.

6) During reference point return with G28/G30, first move to the intermediate point at the speed of G01, and then return to the reference point at the speed of G0. The speed of returning to the reference point can be set by channel parameter 040111 (take channel 0 as an example).

7) During the return from the reference point with G29, first move to the intermediate point at the speed of G0, and then to the target point at the speed of G01.

8) G29 should be executed after G28 and G30 are executed, otherwise the execution may be abnormal if there is no intermediate point stored.

11   Interpolation Function

11.1   Positioning (G00)

Function and Purpose

With this command, the machine tool can quickly and accurately move to the target position, and its moving trajectory can be set by parameters as linear or non-linear path.

Command Format

G00 X__Y__Z__α__ ；   (α is the auxiliary axis)

Parameter	Meaning
X, Y, Z, α	Represent the coordinate value. It is expressed as an absolute position or an incremental position with the G90/G91 state.

Description

(1) G00 is a modal command, which is the 01 group command in the G command code. G00 can be abbreviated as G0 in the programming.

(2) After G00 is specified, it will remain valid until it is replaced by the same group of G codes (i.e. G01, G02, G03, G32 in group 01), and then only the coordinate address needs to be commanded.

(3) With parameter 000013 [G00 Interpolation Enable], two tool paths can be specified:

   a) Non-linear interpolation rapid traverse positioning:

When the parameter is set to 0, the tool will move at the rapid traverse speed of each axis from the current position to the positioning target specified in the block.

   b) Linear interpolation rapid traverse positioning:

When the parameter is set to 1, the tool path is the same as the linear interpolation G01. The tool path is the shortest path between the start point and the end point. The positioning speed is automatically calculated within the range where the specified axis speed does not exceed its rapid traverse feedrate to ensure the shortest allocated time.
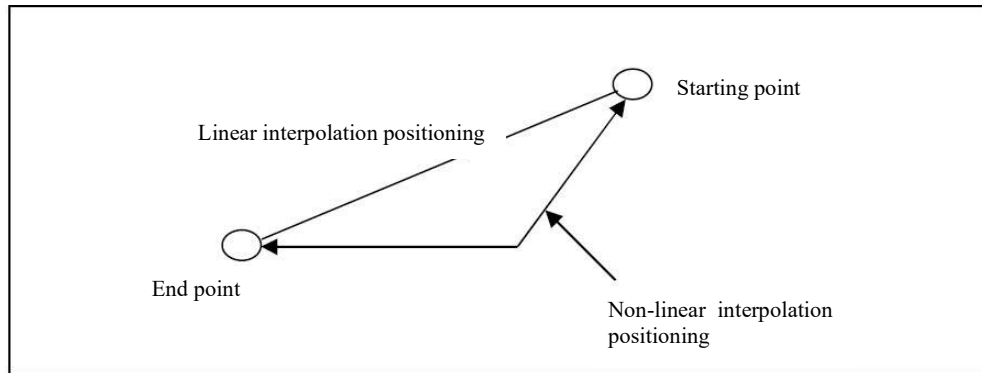
(4) The maximum rapid traverse speed in G00 command is set by the axis parameter [Maximum rapid traverse speed] (100034 for axis 0),(101034 for axis 1),(102034 for axis 2), etc., respectively corresponding to each axis setting, and cannot be specified by F .

(Note 1) The parameter value of the maximum rapid traverse speed cannot be greater than the maximum motor speed * screw pitch.

(5) When the program is running, the rapid traverse speed of G00 is controlled by the rapid traverse speed override on the system operation panel.

(6) G00 is generally used for rapid traverse positioning before processing or rapid traverse retraction after processing. In the positioning mode enabled by G00, the tool accelerates to a predetermined speed at the starting point of the block, and decelerates near the target position.The next block is executed after the tool is in the position.

(7) The allowable error of G00 rapid positioning is set by the system parameters (100060 for axis 0), (101060 for axis 1), (102060 for axis 2), etc. (the system parameter 010166 Maximum Time for Exact Stop Check sets the maximum time to check the positioning tolerance of the coordinate axis after reaching a certain point with the rapid traverse positioning (G00))

Example

Example 1: As shown in the figure below, programming with G00: the tool is required to move from point A to point B in rapid traverse positioning.
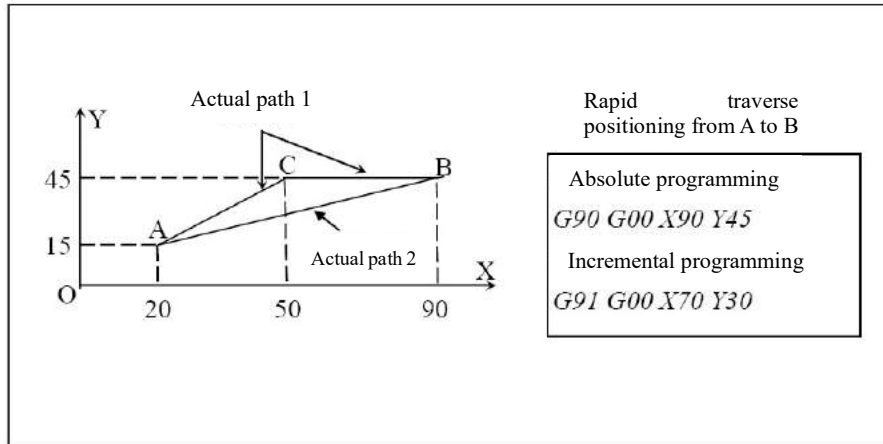
（1）In the non-linear interpolation mode, when the rapid traverse speeds on X axis and Y axis are the same, the rapid traverse positioning path from point A to point B is A—C—B;

（2）In the linear interpolation mode, when the rapid travers speeds on X axis and Y axis are the same, the rapid traverse positioning path from point A to point B is A-B;

Example 2: As shown in the figure below, use G00 to program, a→b→c→d (starting point is at origin X0, Y0).
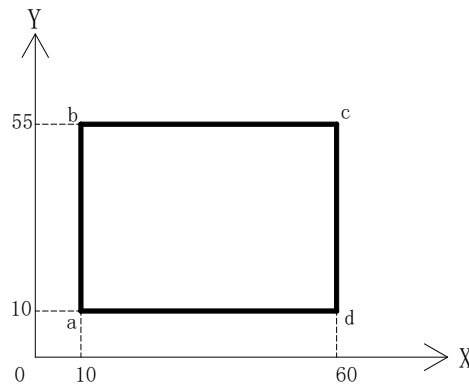
Machining path	Absolute programming	Incremental programming	Programmed position
Rapid traverse positioning to a point	G90G00X10Y10	G91G00X10Y10	(X10、Y10)
a→b	Y55	Y45	(X10、Y55)
b→c	X60	X50	(X60、Y55)
c→d	Y10	Y-45	(X60、Y10)

(1) The rapid traverse speed can be corrected by the rapid traverse override knob on the panel.

(2) There is no value after the G command, and the system alarms "Syntax-Illegal symbol".

11.2   Unidirectional Positioning (G60)

Function and Purpose

This function enables each positioning movement of the machine tool to be realized in one direction, by which the backlash of the machine tool can be eliminated and precise positioning can be achieved.

Command Format

G60 X__Y__Z__α__ ； (α is the auxiliary axis)

Parameter	Meaning
X,Y,Z,α	The coordinate value. With the mode of G90 or G91, it is expressed as an absolute position or an incremental position

Description

(1) G60 is a non-modal command. It is the 00 group command in the G command code, and is valid only for the current line.

(2) When the system runs the G60 command, it is executed at the G00 rapid traverse speed.

(3) When executing G60 command, the offset amount and direction are set through the following parameters. The positive and negative values determine the offset direction.

1st axis	Parm100030	G60 offset value vector of the first axis
2nd axis	Parm101030	G60 offset value vector of the second axis
3rd axis	Parm102030	G60 offset value vector of the third axis

(4) In order to eliminate the influence of backlash, the positioning in one direction of the axis can be commanded. As shown in the figure below, when the movement direction is consistent with the positioning direction, the positioning is performed in a conventional way; when the movement direction is inconsistent with the positioning direction, first move an additional offset along the movement direction, and then move an offset distance along the positioning direction, finally reach the end of the positioning.
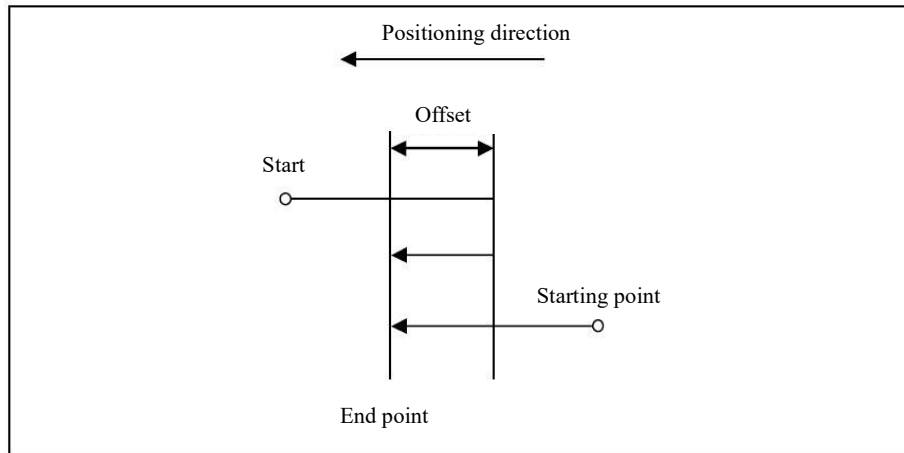
Example

Use unidirectional positioning from A→B (parameter 100030 value is -10)
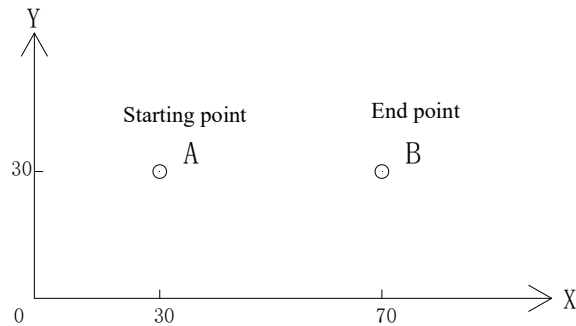
Programming commands	Commanded coordinates
N1 G54	
N2 G0X30Y30	N2　(X30、Y30)
N3 G60X70	N3　(X80、Y30)→(X70、Y30)
N4 M30	

Note

1) Even if the tool movement distance is zero, unidirectional positioning is performed;

2) The set overtravel in unidirectional positioning should be greater than the backlash of the corresponding axis, otherwise the backlash cannot be completely eliminated during unidirectional positioning

11.3 Linear Interpolation (G01)

Function and Purpose

This command uses the combination of coordinates and feedrate commands to move (interpolation) the tool linearly from the current point to the target point specified by the coordinate address at the speed specified in the address F. At this time, the feedrate specified by address F is always the linear speed in the direction of tool center movement.

Command Format

G01 X__Y__Z__α__F__ ；(α is the auxiliary axis)

Parameter	Meaning
X, Y, Z, α	The coordinate value. With the mode of G90 or G91, it is expressed as an absolute position or an incremental position
F	Feedrate（mm/min or °/min）

Description

(1) G01 is a modal command, which is the 01 group command in the G command code, and can be canceled by G00, G02, G03 and G32. G01 can be abbreviated as G1 in the programming.

(2) With the G01 command, the tool moves linearly from the current position to the end point specified in the block at the synthetic feedrate specified by F in the simultaneous-axis mode.

(3) The feedrate specified by F is valid until the new value is designated. Therefore, it does not need to be specified in every block.

(4) The maximum processing speed of G01 is set by the system axis parameter [maximum processing speed] (100035 for axis 0),(101035 for axis 1),(102035 for axis 2), etc., respectively corresponding to each axis. When the programmed F feedrate is greater than the maximum processing speed set by the axis parameter, the system executes at the maximum processing speed set by the axis parameter.

   (Note 1) The maximum processing speed set cannot be greater than the maximum motor speed * screw pitch

(5) When the program is running, the G01 processing speed is controlled by the speed override on the operation panel of the system

(6) When F feedrate is not specified after G01 command, and the programming is in G94 feed per minute, the system executes at the feedrate set by parameter 040030 "Default feedrate in channel(mm/min)"; When the programming is in G95 feed per revolution, the system executes at the feedrate set by parameter 040044 " Default feedrate in channel (mm/r)".

(7) The speed along each axis in each direction is as follows:

G91 G01 Xα Yβ ZγFf；

Speed along X axis　Fα=α×f/L；

Speed along Y axis　Fβ=β×f/L；

Speed along Z axis　Fγ=γ×f/L；

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

For the rotary axis, the feedrate is specified by the linear speed.

(8) When the linear axis α (such as X, unit mm) and rotary axis β (such as C, unit deg) perform the linear interpolation, the tangent speed in the Cartesian coordinate system of α and β is the specified F (mm/min) speed. The speed on β is obtained by converting the required time to deg/min using the above formula.

For example, G91 G01 X20.0 C40.0 F300.0；

Assume that the 40.0 deg of the C axis in metric is 40 mm.

The allocated time is:

$$\frac{\sqrt{20^2 + 40^2}}{300} \approx \textbf{0.14907min}$$

The speed on C axis is:

$$\frac{40\,\text{deg}}{0.14907\,\text{min}} \approx \textbf{268.3deg/min}$$

(9) The trajectory in linear interpolation and rotary interpolation is as follows:

Linear interpolation	Rotary interpolation
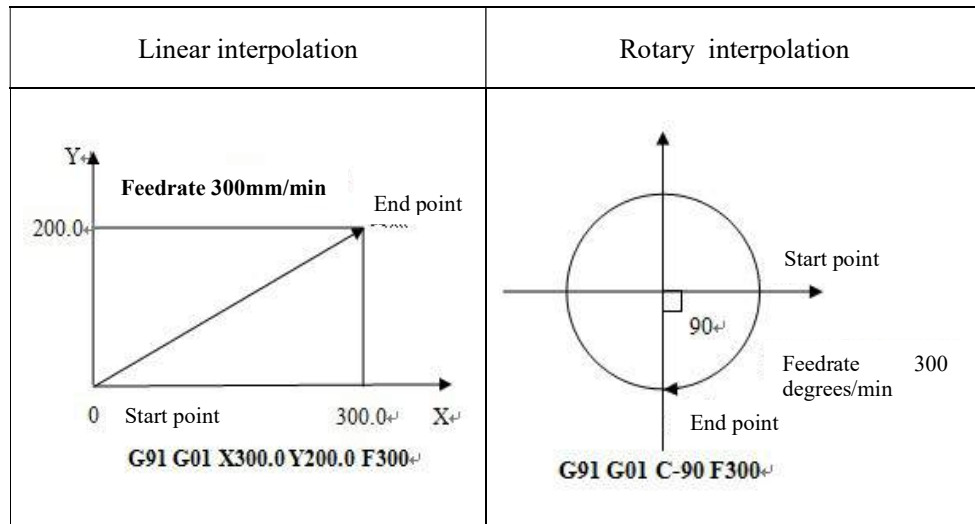
Example

Perform the cutting on P1→P2→P3→P4 at a feedrate of 600mm/min; P0→P1 is the rapid traverse positioning
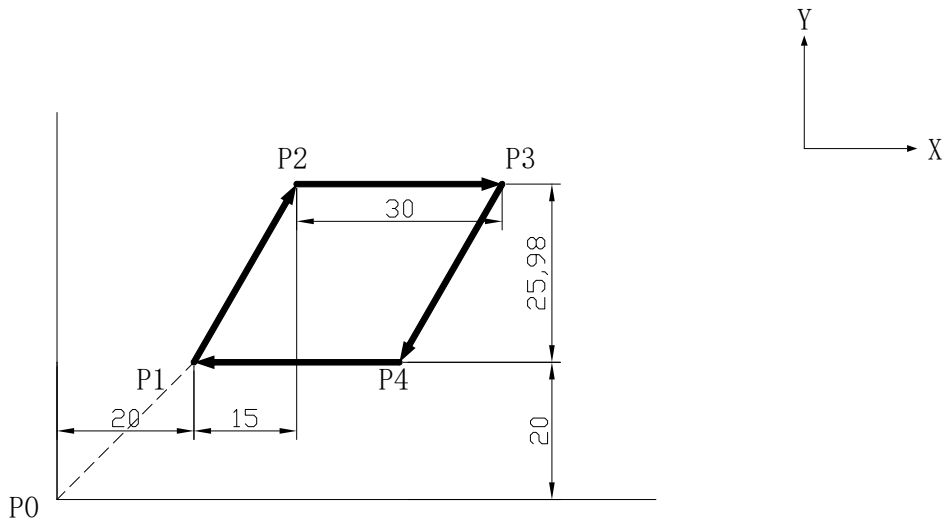
Processing path	Absolute programming	Incremental programming	Commanded position
Positioning	G54G0X0Y0	G54G0X0Y0	(X0、Y0)
P0→P1	X20Y20	G91X20Y20	(X20、Y20)
P1→P2	G1X35Y45.98F600	G1X15Y25.8F600	(X35、Y45.98)
P2→P3	X65	X30	(X65、Y45.98)
P3→P4	X50Y20	X-15Y-25.98	(X50、Y20)
P4→P1	X20	X-30	(X20、Y20)
Program ends	M30	M30	

11.4　Circular Interpolation（G02/G03）

✎　**Function and Purpose**

With this command, the tool moves to the end point in the specified plane (G17, G18, G19) along the specified circular direction.

Ⓕ　**Command Format**

$$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_Y\begin{Bmatrix}I\_J\_\\R\_\end{Bmatrix}F\_\quad \text{Circular interpolation in XY plane}$$

$$G18\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_Z\begin{Bmatrix}I\_K\_\\R\_\end{Bmatrix}F\_\quad \text{Circular interpolation in ZX plane}$$

$$G19\begin{Bmatrix}G02\\G03\end{Bmatrix}Y\_Z\begin{Bmatrix}J\_K\_\\R\_\end{Bmatrix}F\_\quad \text{Circular interpolation in YZ plane}$$

Parameter	Meaning
G02	Circular interpolation CW
G03	Circular interpolation CCW
G17	Circular interpolation in XY plane
G18	Circular interpolation in ZX plane
G19	Circular interpolation in YZ plane
X	Movement amount on X axis of circular interpolation or X coordinate of circular end
Y	Movement amount on Y axis of circular interpolation or Y coordinate of circular end
Z	Movement amount on Z axis of circular interpolation or Z coordinate of circular end
R	Radius (with sign, "+" minor arc, "-" major arc)
I	The distance from the circular starting point to the circular center on X (with sign)
J	The distance from the circular starting point to the circular center on Y (with sign)
K	The distance from the circular starting point to the circular center on Z (with sign)
F	Feedrate, modal

📖　**Description**

1）Circular command description

G02/G03 are modal commands, which are the 01 group commands in G codes. G02/G03 can be abbreviated as G2/G3 during programming.

2）Circular interpolation direction

The definition of the circular interpolation direction in each plane: In the Cartesian coordinate system, viewed from the positive to negative direction of the third axis, when the circular motion direction is consistent with the clockwise direction, it is the clockwise circular interpolation direction; when the circular motion direction is the same as the counterclockwise direction, it is the counterclockwise interpolation direction.

The third axis is Z axis in XY plane, Y axis in ZX plane, and X axis in YZ plane. The definitions of directions are as follows:
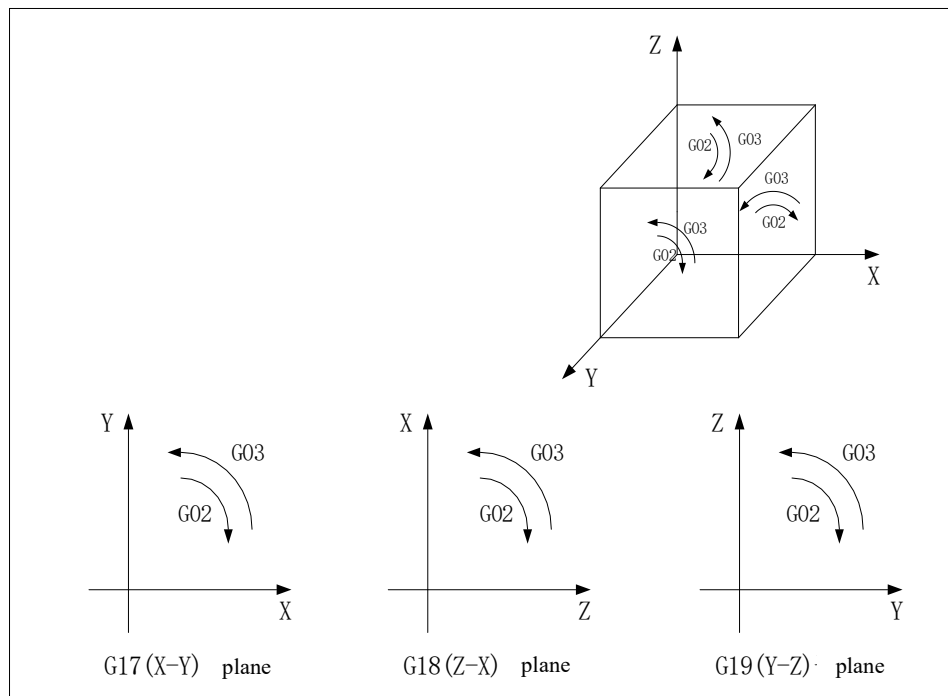
3）Circular end point

The circular end is specified with position command (X, Y, Z)

In absolute mode (G90), （X，Y，Z）specifies the absolute position of circular end; in incremental mode (G91), （X，Y，Z）specified the distance from the circular starting point to the circular end point.

End point

(X_2, Y_2)

End point of arc

G90 时： $X\_=X_2$

$Y\_=Y_2$

G91 时： $X\_=X_2-X_1$

$Y\_=Y_2-Y_1$

Starting point

(X_1, Y_1)
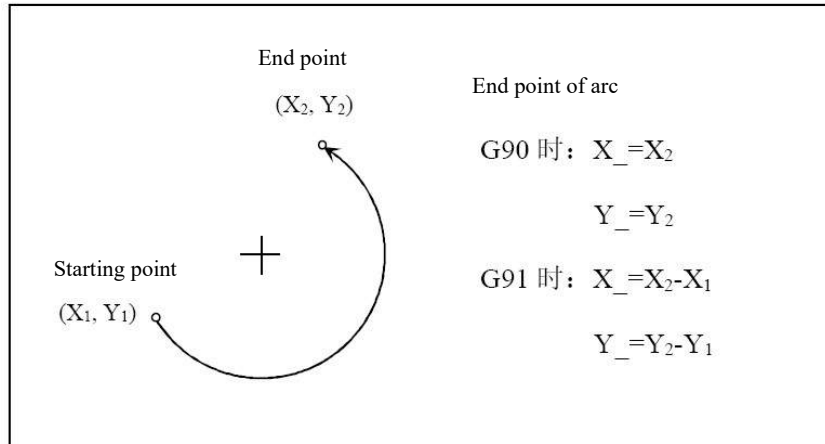
4）UVW incremental programming

In addition to XYZ, UVW can also be used to specify the circular end point.

For the turning system (T series), when the channel parameter [UVW incremental programming enable] (040033) is set to 1, UVW can be used instead of XYZ to represent the movement amount (increment) in G02/G03 on the XYZ-axis, and user can also use XYZ and UVW for programming simultaneously. (The premise is that the UVW axes are not designated as the traverse axes)

5）Circular center with IJK

Command (I, J, K) is used to specify the position of the circular center.

The value specified by (I, J, K) is the vector component viewed from the starting point to the circular center, and whether in G90 or G91 it is always an incremental value.

The sign of the values specified with (I, J, K) determines the direction.

The representation of the circular center is shown in the figure below

End point (X_2,Y_2) End point (Z_2,X_2) End point (Y_2,X_2)

Starting point (X_1, Y_1) J

Starting point (Z_1, X_1) I

Starting point (Y_1, Z_1) K

Circle center $(X0, Y0)$ I

G17 $I\_=X_0-X_1$

$J\_=Y_0-Y_1$

Circle center $(Z0, X0)$ K

G18 $K\_=Z_0-Z_1$

$I\_=X_0-X_1$

Circle center $(Y0, Z0)$ J

G19 $J\_=Y_0-Y_1$

G19 $K\_=Z_0-Z_1$
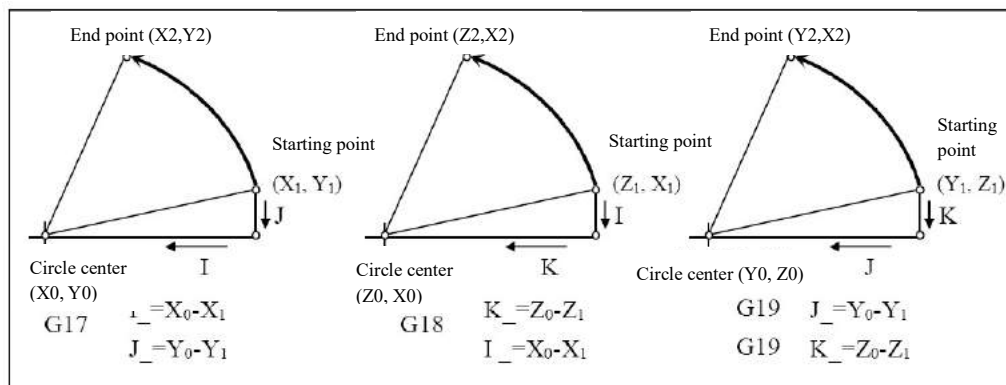
6）Full circle programming

If all the position commands (X, Y, Z) are omitted during programming, it means that the start point and the end point coincide. At this time, a full circle is designated with (I, J, K) programming.

If specified with R, it becomes an arc of 0 degrees, and the system alarms.

7）Semicircle programming

The circular center can be specified by the above-mentioned (I, J, K) as well as the radius. There are two cases for the radius being used to specify the center of the circle.

a) An arc with a center angle less than 180°;

b) An arc with a center angle greater than 180°;

Therefore, user should specify clearly what arc is for the programming, which is determined by the sign of the radius R. As shown below,

Arc of the center angle larger than 180°

②

End point

①

Starting point

Arc of the center angle smaller than 180°

Arc ①

G91 G02 X70 Y80 **R50** F500；

Arc ②

G91 G02 X70 Y80 **R-50** F500；
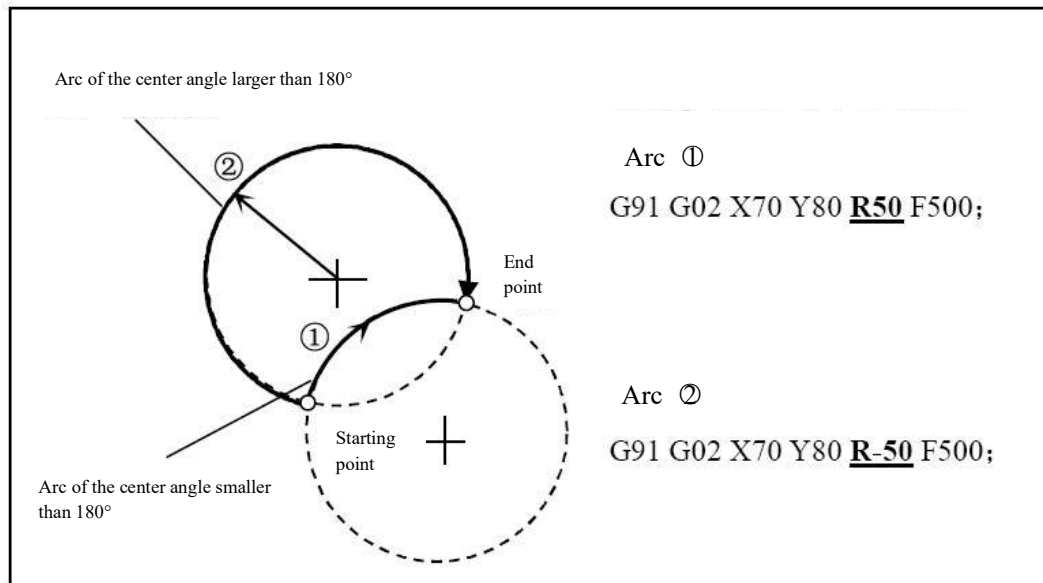
Parameter setting

Parameters of circular interpolation

If the difference between the radiuses of circular starting point and end point is greater than the setting value of [Circular Interpolation Contour Error] (000010), or the ratio of the difference between the radiuses of circular starting point and end point to actual radius exceeds the setting value of [Allowable deviation of circular radius] (000011), the system will alarm.

When parameter 010098 [Whether to switch to G01 when G02/G03 is by default] is set to 1, and the radius is not specified in the G02/G03 command, the block will be executed with G01.

Note

1）If I, J, K and R are specified at the same time in the non-full-cirlce circular interpolation, the arc specified with R is valid.

2）If the axis not in the plane is specified, the alarm will be issued.

3）For the semicircle or the acr with center angle close to 180 degrees, specifying the center position with R will result in the circular center   calculation error due to the rounding error in the programmed position. In this case, I, J, K should be used to specify the circular center.

Example
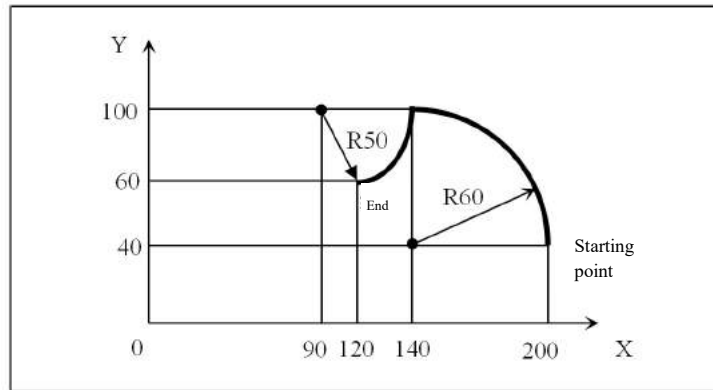
The programming for the tool path shown in the figure above is as follows:

Programming with R		
Absolute programming	Incremental programming	Commanded position
G54G0 X200.0 Y40.0 ;	G54G0 X200.0 Y40.0;	(X200、Y40)
G90 G03 X140.0 Y100.0 R60.0 F3000 ;	G91 G03 X-60.0 Y60.0 R60.0 F3000 ;	(X140、Y100)
G02 X120.0 Y60.0 R50.0;	G02 X-20.0 Y-40.0 R50.0;	(X120、Y60)
Programming with IJK		
Absolute programming	Incremental programming	Commanded position
G54 G0 X200.0 Y40.0;	G54 G0 X200.0 Y40.0;	(X200、Y40)
G90 G03 X140.0 Y100.0 I-60.0 F3000;	G91 G03 X-60.0 Y60.0 I-60.0 F3000;	(X140、Y100)
G02 X120.0 Y60.0 I-50.0;	G02X-20Y-40I-50.0	(X120、Y60)

11.5 3D Circular Interpolation（G02.4/G03.4）

 Function and Purpose

To execute the circular interpolation in three-dimensional space, in addition to the starting point (current position) and end point, it is also necessary to specify any point on the arc (intermediate point). Based on the 3 points (starting point, intermediate point, end point), the defined arc can be processed.

Currently the system has not yet opened this command.
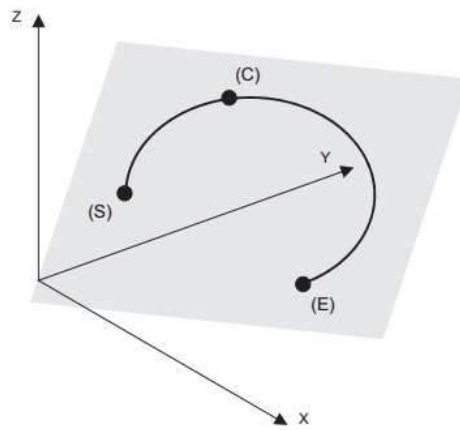
(S) Starting point (current position)
(E) End point
(C) Intermediate point

 Command Format

G02.4/G03.4 X__Y__Z__I__J__K__ F__ ；   （α is the auxiliary axis）

Parameter	Meaning
X, Y, Z, α	The coordinate value. With the mode of G90 or G91, it is expressed as an absolute position or an incremental position
I, J, K	The coordinates of the intermediate point in the space
F	Feedrate（mm/min）

 Example

Three arcs in space are machined as shown below
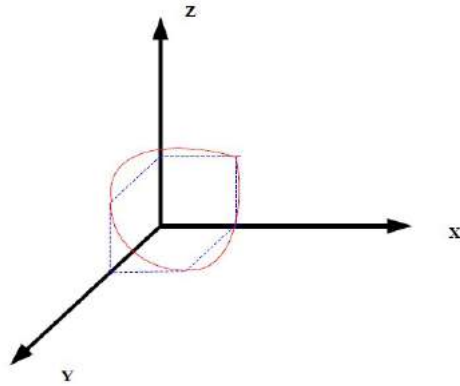
%877

G90 X80 Y0 Z80

F2000

G64

G03.4 X80 Y-80 Z0 I88 J0 K0

      X0 Y-80 Z80 I32 J-74 K32

      X80 Y0 Z80 I0 J0 K88

M30

⚠️ **Note**

(1) The intermediate point is specified by IJK, whether in G90 or G91 mode, it is always the directed distance from starting point to end point.

(2) G02.4 is same as G03.4, cannot specify the rotation direction.

(3) When any two of the circular starting point, intermediate point and end point coincide or the three points are in the same straight line, the system will alarm.

(4) Please disable tool radius compensation function during 3D circular compensation.

(5) 3D circular interpolation cannot be used to specify the full circle (the starting point is inconsistent with the end point); if need to specify the full circle, user could divide the full circle into several segments.

11.6 Thread Cutting（G32）（T）

Function and Purpose

The tool feeds while the spindle rotates, so that different types of threads can be machined, such as variable-pitch threads and multi-start threads. As shown below:

1. **Equal-pitch single-start thread**
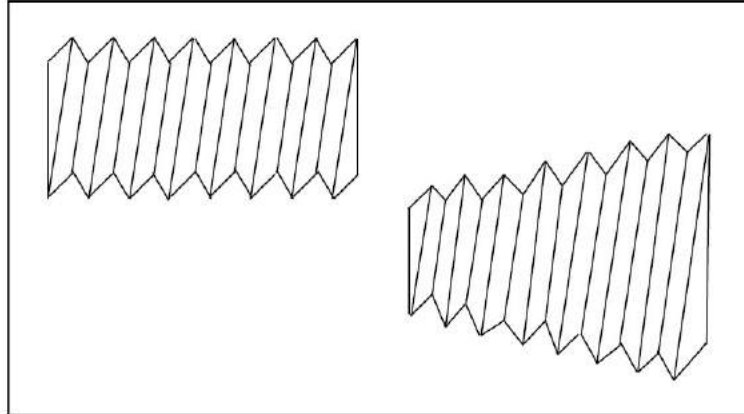
2. **Equal-pitch multi-start thread**

Specify the thread starting angle P to process multi-start threads. If P=180 degrees, the double-start thread can be processed.
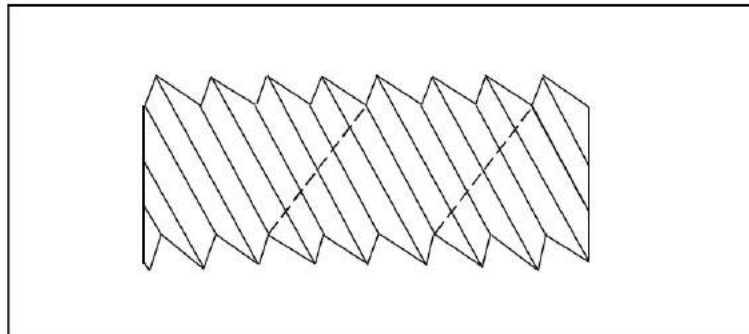
Command Format

G32 X(U)_ Z(W)_ F_ P_ R_ E_

Parameter	Meaning
XZ	Coordinates of thread end point (G90)
	Incremental value of thread end point relative to thread starting point (G91)
UW	Incremental value of thread end point relative to thread starting point
F	Thread lead; thread pitch for single-start thread (projection distance in the long axis direction)
P	Angle of thread starting point
R	Undercut on Z axis, in incremental, can be omitted
E	Undercut on X axis, in incremental radius, can be omitted

Description

The undercut amount is the size of the incomplete thread profile. The undercut length (axial dimension) of the ordinary thread is generally 1 to 2 times the pitch, and the radial length should be the thread height.

For the undercut machining, R specifies the distance on Z axis, and E specifies the distance on X-axis. The proportional relationship between the two determines the angle of undercut.

The signs of R and E values determines the undercut machining direction. The undercut machining direction should be consistent with the thread machining direction; otherwise the complete thread profile will be damaged.

Example

Programming for the thread shown as below. The thread lead is 1.5mm, and the feed amount (diameter value) each time is 0.8mm, 0.6 mm, 0.4mm, 0.16mm.
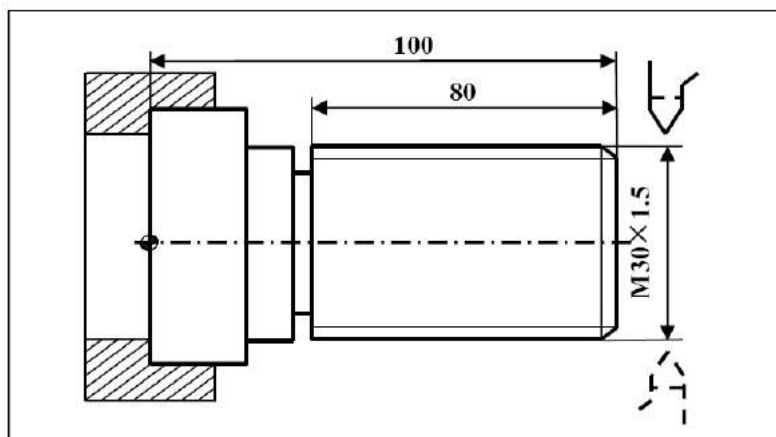
%3316

N1 T0101   (Set the coordinate system, select the No.1 tool)

N2 G00 X50 Z120   (Move to the starting point)

N3 M03 S300   (Spindle rotates at 300r/min)

N4 G00 X29.2 Z101.5   (Move to the thread starting point, acceleration stage 1.5mm,   feed depth 0.8mm)

N5 G32 Z19 F1.5   (Cutting to the thread end point, deceleration stage 1mm)

N6 G00 X40   (Rapid traverse retraction on X axis)

N7 Z101.5   (Rapid traverse retraction to thread starting point on Z axis)

N8 X28.6   (Rapid traverse to thread starting point on X axis with the feed depth 0.6mm)

N9 G32 Z19 F1.5   (Cutting to the thread end point)

N10 G00 X40   (Retraction on X axis)

N11 Z101.5   (Retraction to thread starting point on Z axis)

N12 X28.2   (Rapid traverse to thread starting point on X axis with feed depth 0.4mm)

N13 G32 Z19 F1.5   (Cutting to thread end point)

N14 G00 X40   (Retraction on X axis)

N15 Z101.5   (Retraction to thread starting point on Z axis)

N16 U-11.96   (Retraction to thread starting point on X axis with feed depth 0.16mm)

N17 G32 W-82.5 F1.5   (Cutting to thread end point)

N18 G00 X40   (Retraction on X axis)

N19 X50 Z120   (Return to tool setting point)

N20 M05   (Spindle stops)

N21 M30   (Main program ends and resets)

Note

(1) Do not modify the feedrate override and spindle override during thread cutting;

(2) It is very dangerous to stop the tool feed of thread cutting without stopping the spindle, it will suddenly increase the cutting depth. Therefore, the dwell function is invalid during thread cutting. If the feed hold button is pressed during thread cutting, the feed hold is invalid. Feed hold is only valid during non-thread processing;

(3) When thread cutting is executed in the single block state, the tool stops after the first non-thread cutting is executed.

(4) During thread cutting, the working mode is not allowed to change from auto mode to JOG mode, incremental mode or reference point return mode.

11.7　Helical Interpolation（G02/G03）

Function and Purpose

In addition to specifying circular interpolation, G02 and G03 can be used to perform helical interpolation by specifying the movement distance of the third axis, and perform thread milling and cavity-type parts hole processing.

Command Format

$$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_Y\_Z\_\begin{Bmatrix}I\_J\_\\R\_\end{Bmatrix}L\_F\_\quad \text{Circular interpolation in XY plane}$$

$$G18\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_Z\_Y\_\begin{Bmatrix}I\_K\_\\R\_\end{Bmatrix}L\_F\_\quad \text{Circular interpolation in ZX plane}$$

$$G19\begin{Bmatrix}G02\\G03\end{Bmatrix}Y\_Z\_X\_\begin{Bmatrix}J\_K\_\\R\_\end{Bmatrix}L\_F\_\quad \text{Circular interpolation in YZ plane}$$

Parameter	Meaning
G02	Circular interpolation CW
G03	Circular interpolation CCW
G17	Circular interpolation in XY plane
G18	Circular interpolation in ZX plane
G19	Circular interpolation in YZ plane
X	Movement amount on X axis of circular interpolation or X coordinate of circular end
Y	Movement amount on Y axis of circular interpolation or Y coordinate of circular end
Z	Z axis coordinate of end point in absolute programming; incremental amount of end point on Z axis relative to starting point (even with L)
R	Radius (with sign, "+" minor arc, "-" major arc)
I	The distance from the circular starting point to the circular center on X (with sign); for the conical interpolation in XZ plane, it is the height increase or decrease of one spiral rotation.
J	The distance from the circular starting point to the circular center on Y (with sign)

K	The distance from the circular starting point to the circular center on Z (with sign)
F	Feedrate, modal
L	Number of helical rotations (positive number without decimal point)

📖 **Description**

(1) The rotation direction of helical interpolation refers to the direction of the arc projected to the two-dimensional plane

(2) For full circle programming, if all the position commands (X, Y, Z) are omitted during programming, it means that the start point and end point coincide. At this time, a full circle is specified with (I, J, K). If R is used for specifying, it becomes a 0 degree arc, and the system alarms.

Example

Process the helix as shown below,
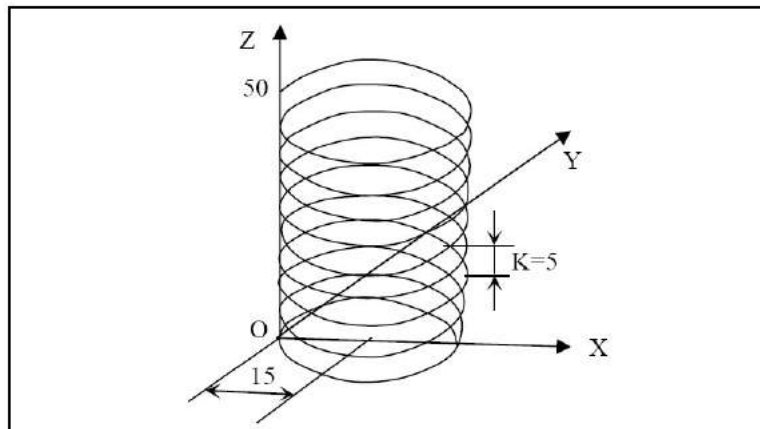
Programming with R		
Absolute programming	Incremental programming	Commanded position
G54 G0 X30 Y0 Z0;	G54 G0 X30 Y0 Z0;	(X30、Y0、Z0)
G90 G03 X0 Y0 Z50 R15 L10 F3500	G91 G03 X-30 Y0 Z50 R15 L10 F3500	(X0、Y0、Z50)
M30	M30	
Programming with IJK		
Absolute programming	Incremental programming	Commanded position

G54 G0 X30 Y0 Z0;	G54 G0 X30 Y0 Z0;	(X30、Y0、Z0)
G90 G03 X0 Y0 Z50 I-15 J0 K0 L10 F3500	G91 G03 X-30 Y0 Z50 I-15 J0 K0 L10 F3500	(X0、Y0、Z50)
M30	M30	

11.8   Imaginary Axis Specifying and Sine Interpolation (G07)

Function Purpose

If an axis is designated as an imaginary axis, the axis only participates in the interpolation computing, but does not move. If a certain axis is designated as the imaginary axis (axis without actual movement) in the helical interpolation, its running path is: the projected path of the helical interpolation on the plane perpendicular to the imaginary axis. This running path corresponds to sine line and cosine line (SIN interpolation or COS interpolation).

Normal helical interpolation
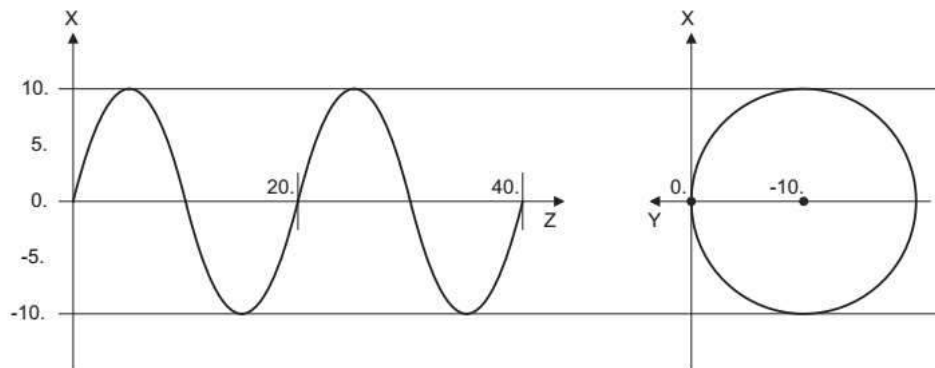
Helical interpolation in imaginary axis (Y-axis is the imaginary axis)
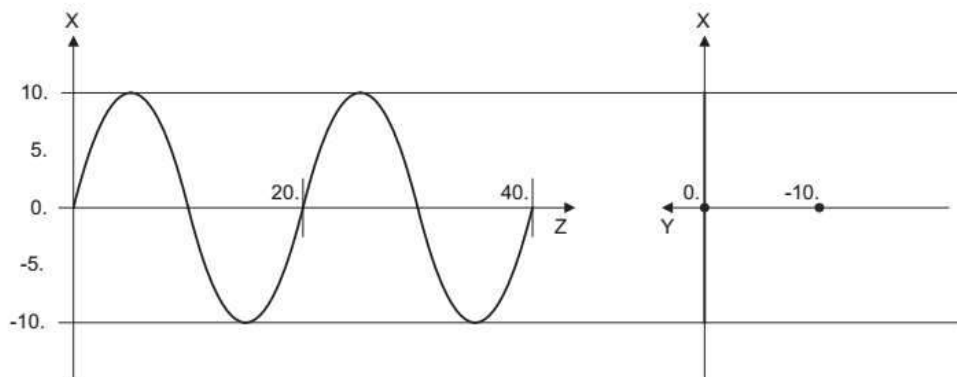
Command Format

G07 $\underline{\alpha}$0（$\underline{\alpha}$1）；

Parameter	Meaning
α0	Imaginary axis interpolation mode is enabled
α1	Imaginary interpolation mode is disabled
α	Axis name of imaginary axis

Description

(1) Between "G07 α0;" and "G07 α1;", the α axis is an imaginary axis.

(2) The axis designation for imaginary axis setting can be applied to all axes of the NC axis.

(3) Multiple imaginary axes can be set.

(4) If a command other than the imaginary axis interpolation mode Enable (0)/Disable (1) is enabled, it will be regarded as imaginary axis interpolation Disable (1). However, when an imaginary axis is specified without the value, it is regarded as imaginary axis interpolation Enable (1).

Example

N01 G07 Y0 ;	Y axis is used as an imaginary axis.
N02 G17 G02 X0 Y0 Z40 I0 J-10 L2 F50	SIN interpolation is performed in XZ plane
N03 G07 Y1 ;	Y axis is restored to the actual axis.
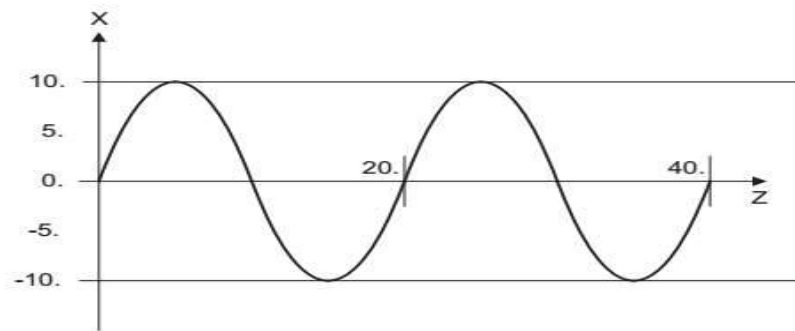
Note

(1) The interpolation functions that can be used in the imaginary axis interpolation are helical interpolation and spiral interpolation.

(2) The imaginary axis interpolation needs to be canceled at the time of high-speed high-precision （G05.1Q1/G05.1Q2/G05.1Q3)

(3) The imaginary axis interpolation is valid during auto operation, and invalid during JOG operation. The imaginary axis interpolation is also valid during handwheel interpolation.

(4) Although the traverse command of the imaginary axis is ignored, when the feedrate is allocated,

the allocation of the imaginary axis is the same as the actual axis.

(5) If the imaginary axis is set again during the imaginary axis interpolation, no error will occur, and the effect of the imaginary axis will continue to be maintained.

(6) When the imaginary axis is canceled, the usage of the non-imaginary axes is not affected.

(7) If reset, the imaginary axis is canceled.

11.9   Polar Coordinate Interpolation（G12/G13）

Function and Purpose

Polar coordinate interpolation is a contour control function, which converts the programmed position in the Cartesian coordinate system into linear axis movement (tool movement) and rotary axis movement (workpiece rotation). It can be programmed on the selected plane (polar coordinate interpolation plane) to complete the milling or grinding of the face contour of the rotary workpiece.

In the polar coordinate interpolation mode, linear interpolation and circular interpolation can be commanded, and absolute programming, incremental programming, diameter programming, radius programming, tool radius and length compensation can be used (polar coordinate interpolation is performed based on the compensated path).

This function is often used in turning centers with power head tools.
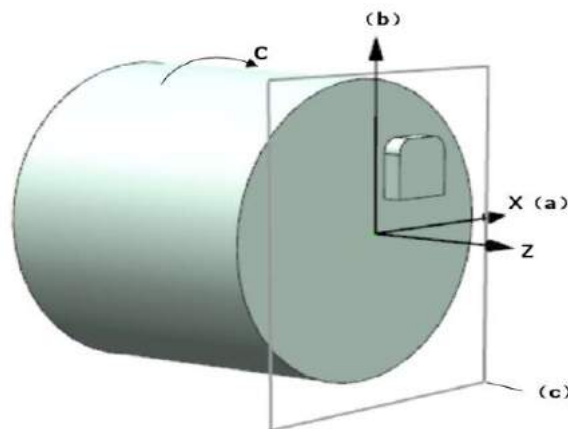
(a) Linear axis   (b) Rotary axis (imaginary axis)   (c) Polar coordinate
                          interpolation plane (G17 plane)

Command Format

G12 ；

G13 ；

Parameter	Meaning
G12	Polar coordinate interpolation is enabled.
G13	Polar coordinate interpolation is disabled.

Description

1）Polar coordinate interpolation plane

The polar coordinate interpolation plane uses the linear axis as the first right-angled axis of the plane, and the imaginary axis orthogonal to the linear axis as the second axis of the plane. The plane composed of 2 orthogonal axes is the polar coordinate interpolation plane, and the polar coordinate interpolation is executed on this plane. In the polar coordinate interpolation, the origin of the workpiece coordinate system is regarded as the origin of the coordinate system.

2）Polar coordinate interpolation parameter setting

The linear axis, rotary axis and imaginary axis in polar coordinate interpolation need to be set in the parameters in advance. The relevant parameters are as follows:

Parameter name	Parameter	Description	Default
Channel parameter CH0	Parm040095	Linear axis No. in polar coordinate interpolation	0 (X axis)
	Parm040096	Rotary axis No. in polar coordinate interpolation	5 (C axis)
	Parm040097	Imaginary axis No. in polar coordinate interpolation	1 (Y axis)

The linear axis and the imaginary axis of the polar coordinate interpolation coordinate system are determined by the above parameters. The linear axis is used as the horizontal axis of coordinate system in the polar coordinate interpolation and the imaginary axis is used as the vertical axis. In the execution plane formed by the linear axis and the imaginary axis, the corresponding circular interpolation address words are shown in the following table:

Parm040095 parameter setting	Linear axis name in polar coordinate interpolation	Plane	Circular definition address words
0	X	G17	I, J, R
1	Y	G18	J, K, R
2	Z	G19	I, K, R

3）Imaginary axis eccentricity compensation

When there is an error in the imaginary axis direction from the rotation axis center of the first axis in the plane, that is, the rotation axis center is not on the X axis, this function can be used for the compensation, and the system will perform polar coordinate interpolation after calculating the error. Enable this function and set the Parm040099 parameter in the channel parameters as the measurement error value.

Imaginary axis (C axis)

Rotary axis

(X, C)

Error in the imaginary axis direction

X axis

假想轴方向的误差
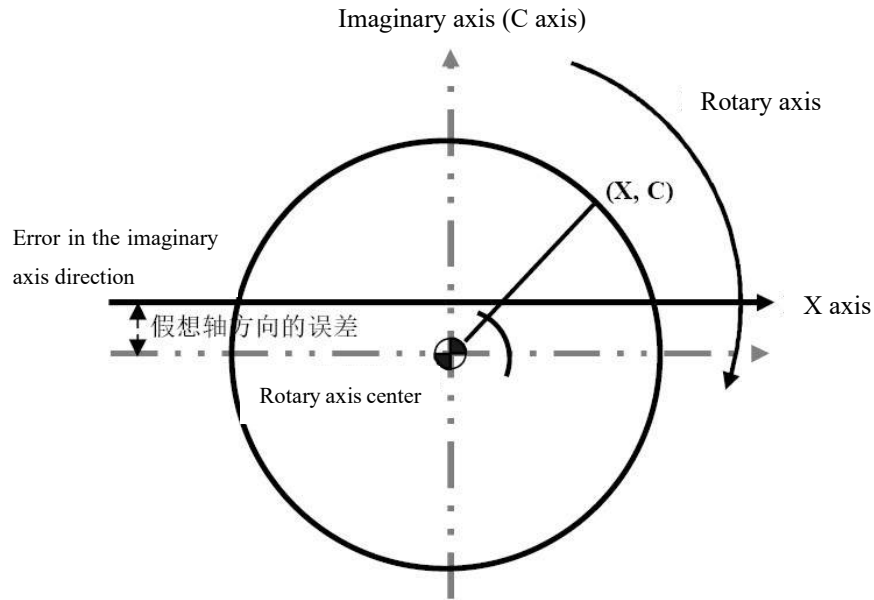
Rotary axis center

(X,C)   The point in the X-C plane (the rotary axis center is used as zero of X-C plane)

X   X axis coordinate in the X-C plane

C   Coordinate of imaginary axis in the X-C plane

P   Error in the imaginary axis direction

4）Coordinate system offset in polar coordinate interpolation

In polar coordinate interpolation, the following format can be used to translate the workpiece coordinate system.

G12   X_C_; (Polar coordinate interpolation used for X axis and C axis)

G12   Y_A; (Polar coordinate interpolation used for Y axis and A axis)

G12   Z_B; (Polar coordinate interpolation used for Z axis and B axis)

X-C (Y-A, Z-B) is used to specify the coordinate value of the center position of the rotary axis C (A, B) in the interpolation plane relative to the origin of the workpiece coordinate system (see the figure below)

G12.1 Xx Cc ;

C axis center

c

Workpiece coordinate system origin

x

X
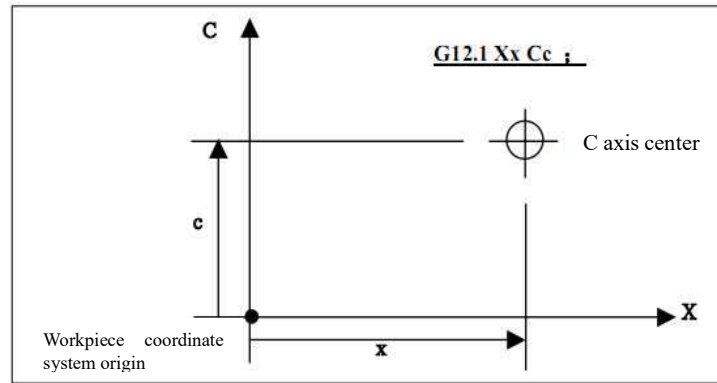
Example

Example 1: Face contour milling
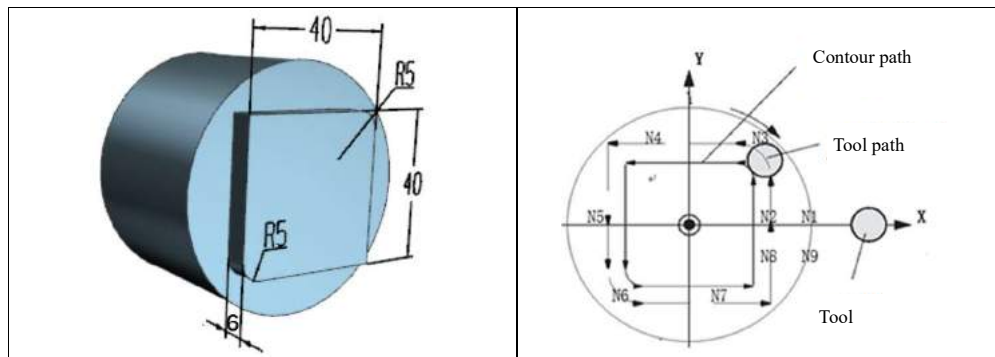
%1234

G54

T0101；The workpiece coordinate system is built at the center of rotation on the right end of the blank, with end milling cutter φ10

G108；Spindle is switched from speed mode to position mode

M103 S1=2000；Tool rotation CW

G37；Radius programming

G0X45Z50C0

G0Z-5

G12X0CO

G42D1G01X40F500（D1=5）

N1 G1X20

N2 C10

N3 G3X10 C20 R10

N4 G1X-20

N5 C-10

N6 G3X-10　C-20　　I10　J0

N7 G1X20

N8 C0

N9 G40X40

G13
G0Z10
G109
G0X120
M105
M30

Example 2: Face contour milling
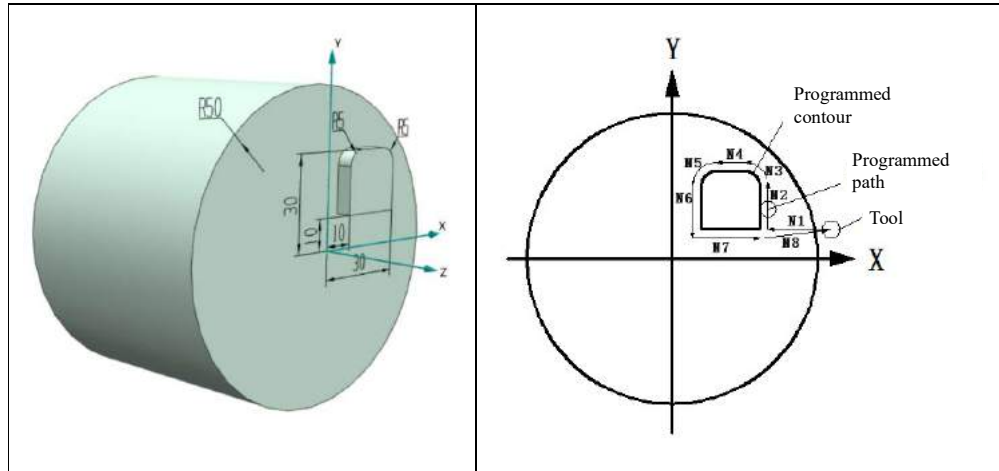
%1234

G54

T0101；The workpiece coordinate system is built at the center of rotation on the right end
      of the blank, with end milling cutter φ6

G108；Spindle is switched from speed mode to position mode

M103 S1=2000；Tool rotation CW

G37； Radius programming

G0X52Z50C0

G0 Z-5

G12X0CO

G1X50C10F500

N1 G42D1G01X30C10（D1=3）

N2 G1C25

N3 G03X25C30R5

N4 G1X15

N5 G03X10C25R5

N6C10

N7 X30

N8 G40X50

G13

G0Z10

G109；Spindle is switched from speed mode to position mode

G0X52

M105；Tool stops

M30

Note

Before entering the polar coordinate interpolation mode, cancel the tool radius compensation and tool length compensation, and specify the tool compensation in the polar coordinate interpolation mode.

11.10 Cylindrical interpolation（G07.1）

Function and Purpose

With this function, the shape of the side surface of the cylinder (the shape on the cylindrical coordinate system) is expanded into a plane, and the expanded shape is used as the plane coordinate for the program command, then the system converts it to the movement on linear axis and rotary axis of the cylindrical coordinate when performing machining to execute the contour control.
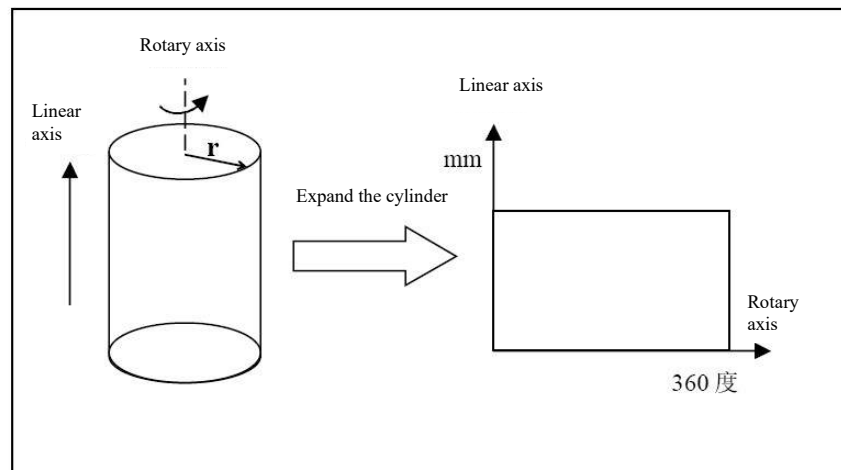
Command Format

G07.1 RC=<u>r</u>   ；   Start of cylindrical interpolation (r: radius of cylinder)

……

G07.1 RC=0   ；   End of cylindrical interpolation

Description

Circular interpolation

The circular interpolation (G02, G03) in the cylindrical interpolation plane is performed, and the coordinate system plane is determined based on the parallel axis in the cylindrical interpolation.

Parm040092 = 0	Set X axis as the horizontal axis in the cylindrical interpolation plane	G02, G03 are specified in X-C plane (G17) with IJR programming
Parm040092 = 1	Set Y axis as the horizontal axis in the cylindrical interpolation plane	G02, G03 are specified in Y-C plane (G19) with JKR programming
Parm040092= 2	Set Z axis as the horizontal axis in the cylindrical interpolation plane	G02, G03 are specified in Z-C plane (G18) with IKR programming

Compensation

The tool radius compensation and tool length compensation should be cancelled before entering the cylindrical interpolation mode, and the tool compensation should be specified in the cylindrical interpolation mode.

Related parameters

Parameter name	Parameter	Default	Description
Channel parameter (CH0)	Parm040090	5（C axis）	Rotary axis No. in cylindrical interpolation
	Parm040091	2（Z axis）	Linear axis No. in cylindrical interpolation
	Parm040092	1（Y axis）	Parallel axis No. in cylindrical interpolation

Example
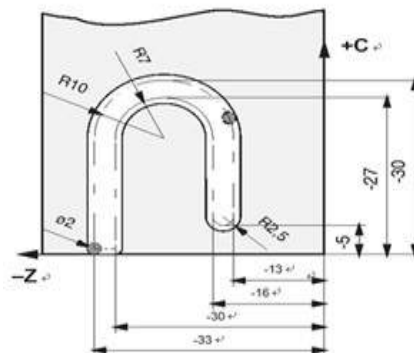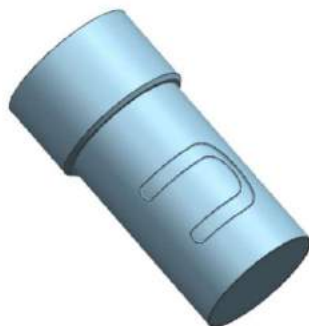
Example 1: Contour milling
G54
T0101；Radial face milling cutter ø2, workpiece ø30
G108
G19；Select a plane
M103 S1=1000；The second spindle rotates CW, the direction of tool rotation is related to the tool post
G0X50C0

G0 Z-31.5
G07.1RC=15
G95 G1 X29 F0.08
G42D1 G1 C1.5 Z-33
G1C20
G2C20Z-13R10
G1 C6.5
G2 C6.5Z-16R1.5
G1 C20
G3 C20Z-30R7
G1 C0 Z-33
G40   C1.5 Z-31.5
G0 X32
G0 Z10
G07.1 RC=0;
G109G18
G0X150Z150
M105
M30

Example 2: Contour milling

%1234	%1
G54	G91
T0101; Radial face milling cutter ø4, workpiece ø37	G42 D1G1Z-5C0F100
	G3 C-14.564Z-3.294 R5
G108	G1 C-16.006Z-14.047
G94	G2 C-29.128 R5
G19; Select a plane	G1 C-16.006 Z14.047
M103 S1=1000	G3 C-14.564 Z3.294 R5

G0X50C0
G0 Z50
G07.1 RC=18.5
G0Z5
M98P1L4
G90
G07.1 RC=0;
G109
G18
G0X100 Z100
M105
M30

G40G1 Z5
M99

11.11 Polar Coordinate Command（G15/G16）

Function and Purpose

When the angle and radius are marked on the drawings of the parts, the "polar diameter" and "polar angle" of polar coordinate are used to define the end positions of the program, which can greatly simplify the programming calculation.

In the specified plane of the polar coordinate system, the counterclockwise direction of the polar axis movement is the positive direction of the polar angle, and the clockwise direction of the polar axis movement is the negative direction of the polar angle.

Both "polar diameter" and "polar angle" can be specified under absolute programming/incremental programming (G90, G91).

Command Format

G17(or G18/G19)；

G16 ；

G90 X- Y- (or G91 X- Y-)；

G15；

To specify the plane of polar coordinate system	G17	XY plane: polar radius is specified with X axis, and polar angle is specified with Y axis.
	G18	ZX plane: polar radius is specified with Z axis, and polar angle is specified with X axis.
	G19	YZ plane: polar radius is specified with Y axis, and polar angle is specified with Z axis.
To specify the origin of polar coordinate system	G90	Workpiece coordinate system zero (or local coordinate system zero) is specified as the origin of polar coordinate system (polar origin)
	G91	The current position is specified as the origin of polar coordinate system

		(polar origin)
G16		Polar coordinate programming starts
G15		Polar coordinate programming ends
To specify polar diameter	X	In absolute programming: Polar diameter measured from the workpiece coordinate system zero (or local coordinate system zero)
		In incremental programming: Polar diameter measured from the current point
To specify polar angle	Y	In absolute programming: Polar angle measured with workpiece zero as the polar origin
		In incremental programming: Polar angle measured with the current point as the polar origin

📖 **Description**

Polar coordinate system origin setting

There are two ways to set the polar coordinate system origin:

(1) At the time of programming with polar coordinate, if the polar origin is defined in absolute mode (G90), the polar coordinate system origin will be the workpiece origin or the local coordinate system origin.

(2) At the time of programming with polar coordinate, if the polar origin is defined in incremental mode (G91), the polar coordinate system origin will be the current point.

Polar coordinate command position

(1) When workpiece coordinate system origin is the center of polar coordinate,

The workpiece coordinate system origin is the polar coordinate center when the radius is specified in absolute mode.

Only when the local coordinate system (G52) is used, the origin of the local coordinate system is the center of the polar coordinate.
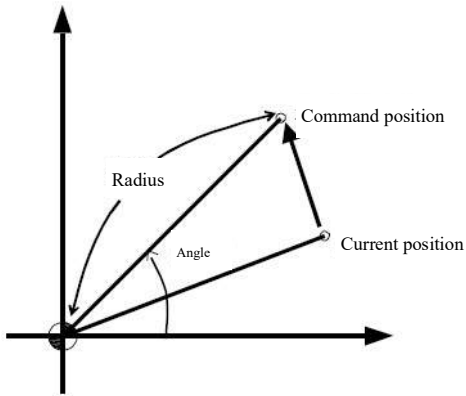
When the angle is in absolute mode
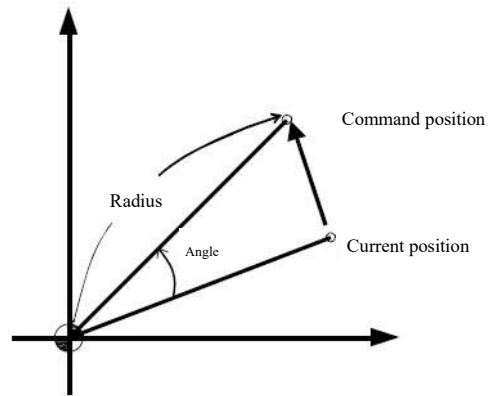
When the angle is in incremental mode

(2) When the current position is the center of polar coordinate,

The current position is the polar coordinate center when the radius is specified in incremental mode.
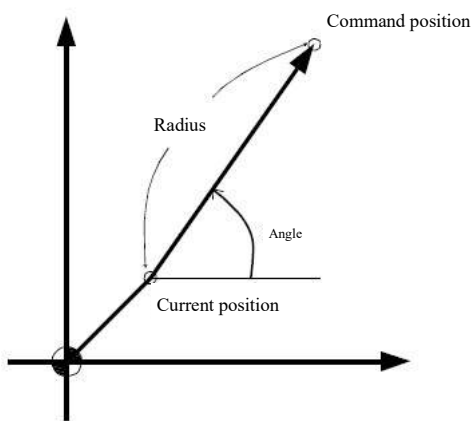
When the angle is in absolute mode
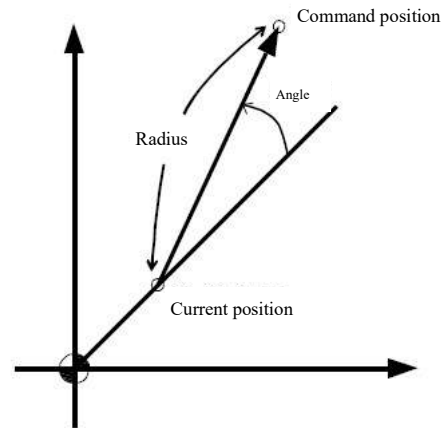
When the angle is in incremental mode

(3) When radius command is ignored,

When the radius is ignored, the origin of the workpiece coordinate system is the polar coordinate center, and the distance from the polar coordinate center to the current position is the radius. Only when the local coordinate system (G52) is used, the local coordinate origin is the polar coordinate center.
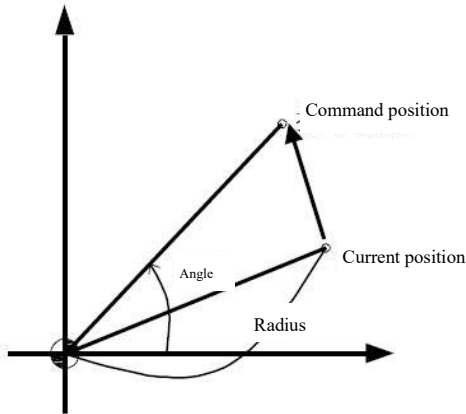
When the angle is in absolute mode
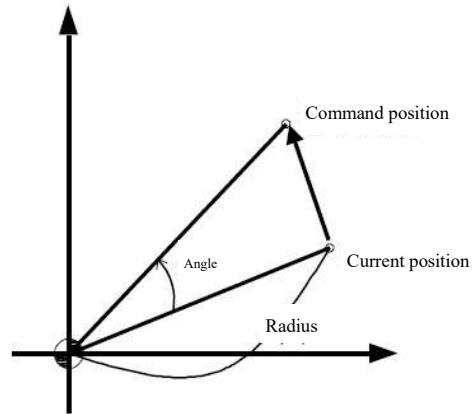
When the angle is in incremental mode

(4) When angle command is ignored,

When the angle command is ignored, the angle of the current position in the workpiece coordinate system is the angle command.

When the radius is commanded in the absolute mode, the origin of the workpiece coordinate system is the polar coordinate center. Only when the local coordinate system (G52) is used, the origin of the local coordinate system is the polar coordinate center.

In addition, when the radius is specified in the incremental mode, the current position is the polar coordinate center.
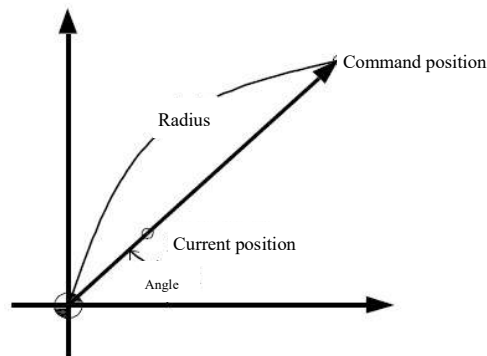
When the angle is in absolute mode
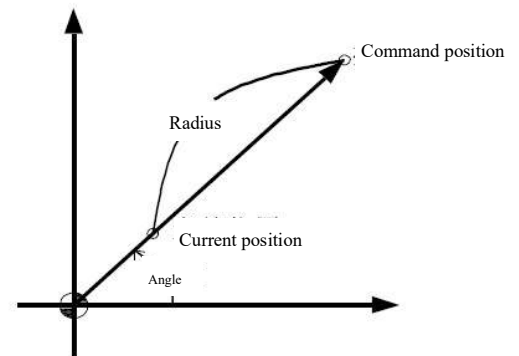
When the angle is in absolute mode

Example

Example 1: Mill a hexagon with the polar coordinate programming, the workpiece origin is set at the center of the hexagon (as shown below), and the processing path is a→b→c→d→e→f.
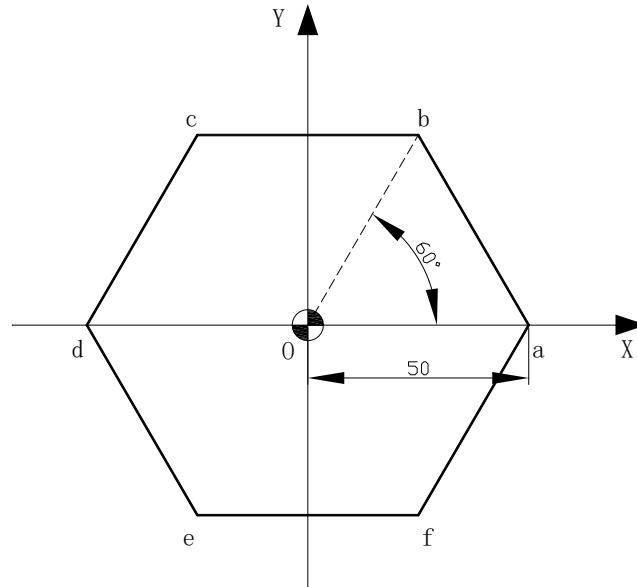
Processing path	Absolute programming	Incremental programming	Commanded position
Position to a point	G54G0X50Y0	G54G0X50Y0	(X50、Y0)
Polar coordinate command starts	G17G90G16	G17G90G16	(X50、Y0)
a→b	G1X50Y60F120	G1X50Y60F600	(X25、Y43.3)
b→c	Y120	G91Y60	(X-25、Y43.3)
c→d	Y180	Y60	(X-50、Y0)
d→e	Y240	Y60	(X-25、Y-43.3)
e→f	Y300	Y60	(X25、Y-43.3)
f→a	Y360	Y60	(X50、Y0)
Polar coordinate command is disabled	G15	G15	(X50、Y0)
Program ends	M30	M30	

Example 2: As shown in the figure below, use polar coordinate programming to drill 3 holes with a diameter of 10mm on the workpiece.

(The workpiece coordinate system zero is O, and the rotation polar origin A of polar coordinate is outside the worktable the machine tool)

12
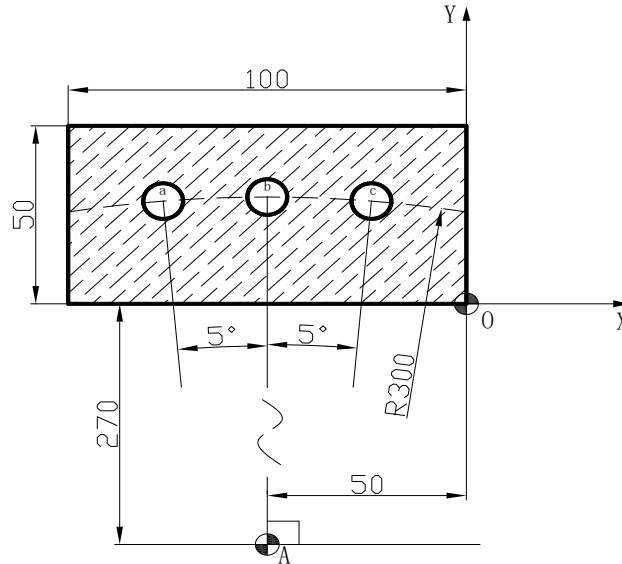
Processing path	Absolute programming	Incremental programming	Commanded position (X, Y, Z)
Position above the zero	G54G0X50Y0Z30	G54G0X50Y0Z30	(50、0、30)
Establish G52 local coordinate system	G52X-50Y-270	G52X-50Y-270	(50、270、Z30)
Polar coordinate command starts	G17G90G16	G17G90G16	(50、270、Z30)
Drill hole at c point in the plane	G82X300Y85Z-10R5F100	G82X300Y85Z-10R5F100	(26.147，298.858，-10)
Drill hole at b point in the plane	Y90	G91Y5	(0、300、-10)
Drill hole at a point in the plane	Y95	Y5	(-26.147，298.858，-10)
Polar coordinate command is disabled	G15	G15	(-26.147，298.858，30)
Local coordinate system is cancelled	G52X0Y0	G52X0Y0	(-76.147，28.858、30)
Program ends	M30	M30	

Note

(1) The axis commands accompanied by the following commands will not be regarded as polar coordinate commands;

- Dwell G04

- Programmable data input G10

- Local coordinate system G52

- Workpiece coordinate system change G92

- Machine coordinate system selection G53

- Coordinate rotation G68

● Scaling G51

(2) In polar coordinate mode, the angle and radius R cannot be specified;

(3) For polar coordinate programming, when radius is specified in absolute mode, the workpiece origin is set as the polar coordinate center; when radius is specified in incremental mode, the current position is set as the polar coordinate center. However, when only the angle is specified in the command, regardless of whether it is in absolute or incremental mode, the workpiece origin is set as the polar coordinate center.

11.12   NURBS Spline Interpolation

Function and Purpose

This function only needs to specify the NURBS (non-uniform rational B-spline) curve (degree/weight/node/control point) used in curve/curved surface processing to realize NURBS curve processing without replacing tiny line segments.

In NUBRS interpolation, interpolation is performed at the commanded speed, but at positions with large curvature, the speed is limited to a speed that does not exceed the allowable acceleration of the machine tool.

Command Format

NURBS P_ K_ X_ Y_ Z_ W_ F_ E_ ;

Parameter	Meaning
P	Order of NURBS curve, only cubic spline is supported. P is 4
K	Node
X, Y, Z	Coordinate value of control point
W	Weighting
F	Feedrate
E	2nd feedrate

Description

Cancel interpolation

NURBS belongs to the 01 group of modal, and the NURBS interpolation modal can be released by specifying G01 or G00.

Order of curve

P is to specify the order of NURBS curve;

When P=4, it indicates the cubic NURBS curve;

P is a modal address word, and P will remain valid until it is changed or other modal commands of group 01 are specified.

Node

In NURBS interpolation, user must specify the first control point as the starting point and the final control point as the end point.

In addition, when specifying the node of the first block, please use the following format:

Single spline

NURBS P4 K{0,0,0,0,1} X1 Y0 Z0

Double-spline

NURBSB P4 K{0,0,0,0,0.5}Q{10,0,0,38.28,0,28.28} W1F60

Weighting

The weight is the weight of the designated control point in the same block. When it is omitted, the default value is 1.0

Compensation

Tool radius compensation cannot be used in NURBS curve interpolation.

Description

Single spline NURBS is generally used for three-axis small line segment interpolation.

Double-spline NURBS is generally used for five-axis small line segment interpolation.

Example

R=50mm Please use the single spline NURBS interpolation for the full circle shown as below, R=50mm.
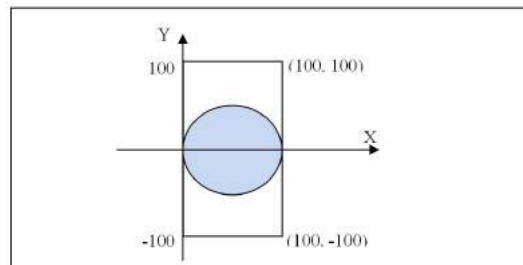
%0001

G54

G90G17F500G64

G01X0Y0Z0

NURBS P4 K{0.0,0.0,0.0,0.0,0.5} X0.0Y0.0Z0.0 W1.0

K0.5   X0.0000 Y100.0   W0.3333

K0.5   X100.0 Y100.0   W0.3333

K1.0   X100.0 Y0.0   W1.0

K1.0   X100   Y-100.0   W0.3333

K1.0   X0.0   Y-100   W0.3333

K1.0   X0.0   Y0.0   W1.0

M30

11.13   HSPLINE Spline Interpolation

Function and Purpose

HSPLINE is the abbreviation of Hermite SPLINE. Hermite interpolation function can also improve the processing effect of small line segments and make the processed surface smooth. Unlike the NURBS curve, the Hermite curve passes through the control points, while the Nurbs curve does not pass through the control points. The system performs spline interpolation by specifying the control points and vectors of the Hermite curve

Command Format

HSPLINE P_X_Y_Z_I_J_K_W_F_ ;

Parameter	Meaning
P	Degree of HSPLINE spline curve
X,Y,Z	Coordinates of control point
I,J,K	Vector of control point
W	Weighting
F	Order of Hermite curve

Description

Cancel interpolation

HSPLINE belongs to the 01 group modal, and the HSPLINE interpolation modal can be cancelled by specifying G01 or G00.

Degree of curve

The degree of HSPLINE spline curve is specified with P, and this value is 3 currently.

Compensation

The tool radius compensation cannot be used in the HSPLINE curve interpolation mode.

Example

Example 1: Use cubic Hermite spline to interpolate the following spatial curve
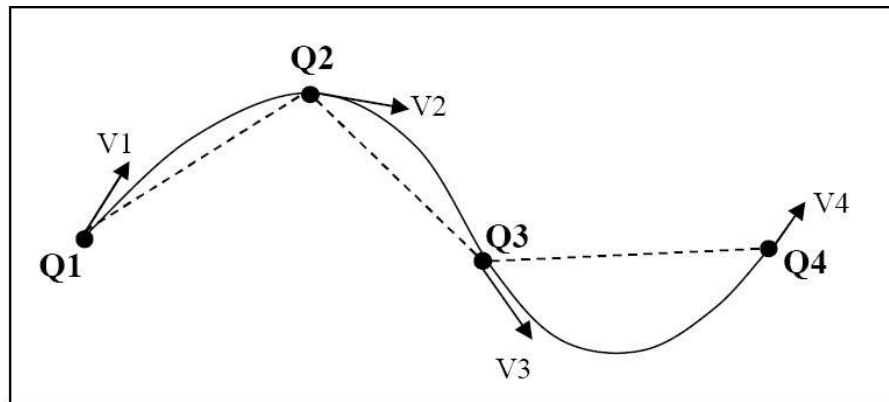

```
%0001
G54G0X0Y0Z0
G90G17 F1000G64
X0.005Y-0.987Z0.04
HSPLINE P3 X0.005 Y-0.987 Z0.040 I1.000 J-0.026 K-0.002；Q1
X0.748   Y-0.727   Z0.027   I0.756   J0.655   K-0.016        ；Q2
X1.049   Y-1.097   Z0.023   I0.967   J0.256   K-0.011        ；Q3
X1.249   Y-0.727   Z0.053   I0.497   J0.866   K0.050      ；Q4
M30
```

Example 2: Use Hermite spline interpolation to program R50 full circle (starting point X0Y0) in G17 plane

```
%1234
G54G01 X0 Y0 Z0 F2000
HSPLINE P3 X0 Y0 Z0    I0 J1 K0   （0°）
X6.698729 Y25 Z0 J0.866025 I0.5 K0；(30°)
X14.644661 Y35.355339 Z0 I0.707107 J0.707107 K0(45°)
X25 Y43.301270 Z0 J0.5 I0.866025 K0 ；(60°)
X50 Y50 Z0 I1 J0 K0；(90°)
X75 Y43.301270 Z0 J-0.5 I0.866025 K0 ；(120°)
X85.355339 Y35.355339 Z0 I0.707107 J-0.707107 K0 ；(135°)
X93.301270 Y25   Z0 J-0.866025 I0.5 K0；  (150°)
```

X100 Y0 Z0 I0 J-1 K0 ；(180°)
X93.301270 Y-25 Z0 J-0.866025 I-0.5 K0；(210°)
X85.355339 Y-35.355339 Z0 I-0.707107 J-0.707107 K0 (225°)
X75 Y-43.301270 Z0 J-0.5 I-0.866025 K0；(240°)
X50 Y-50　Z0 I-1 J0 K0（270°）
X25 Y-43.301270 Z0 J0.5 I-0.866025 K0；(300°)
X14.644661 Y-35.355339 Z0 I-0.707107 J0.707107 K0（315°）
X6.698729 Y-25　Z0　K0 J0.866025 I-0.5 K0；(345°)
X0 Y0 Z0 I0 J1 K0　（360°）
M30

12 Tool Compensation Function

12.1 Turning Tool offset Compensation（T）

Function and Purpose

When editing the program, it is generally assumed that the positions of the tool noses are the same, but due to the difference in tool shape and installation, the actual positions of the tool noses cannot be consistent. The position deviation between the programmed assumption and actual positions is handled with the tool offset compensation for lathes.

Command Format

The first two digits of the T command (turning system) are the tool number, and the last two digits are the tool offset number, that is, the number of the register where the tool offset value is stored. The call of lathe tool offset compensation is implemented by the last two digits of the command T.

The format of T command is : T and the following 4 digits. See the figure below.

➤ The first two digits is the tool number (the currently selected tool number)

➤ The last 2 digits is the tool offset number (the number of the register where the offset is stored and the number of the register where the radius compensation is stored)
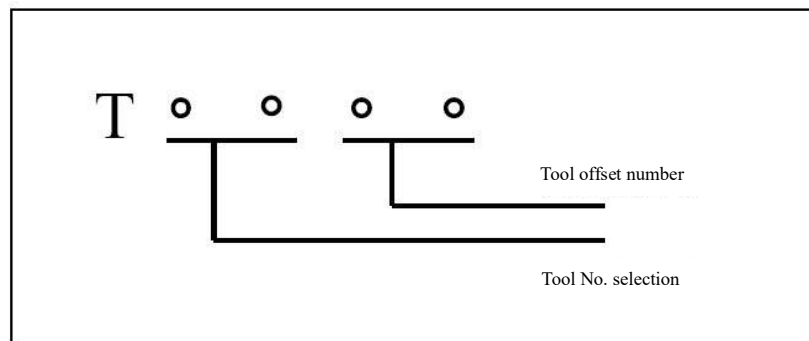
Description

1. Tool offset compensation setting

One tool needs to use one tool number, and one tool nose position needs to use one tool offset number. When the tool has only one tool nose position, the tool number and tool offset number are generally the same number for easy memory, such as T0101. When a tool has multiple tool nose positions to be used, a tool number needs to correspond to multiple offset numbers. To

facilitate memory, the unit digit value of the tool number generally keep the same with that of the tool offset number, such as T0202, T0212, T0222.

2. Cancel tool offset compensation

When the offset number is set to 00, it means that the tool offset function is cancelled and the offset is 0.

3. Relative offset compensation and absolute offset compensation

There are two modes of the tool offset value: incremental offset and absolute offset

➢ Incremental offset

The position deviation of each tool nose relative to the standard tool nose. In this mode, the program also needs a command to call the positional relationship between the standard tool and the workpiece zero (such as the G92 command).

➢ Absolute offset

When each tool is at the machine zero (the machine returns to zero and the tool is at the machining position), the position deviation of the workpiece zero relative to the nose of each tool is shown as below:
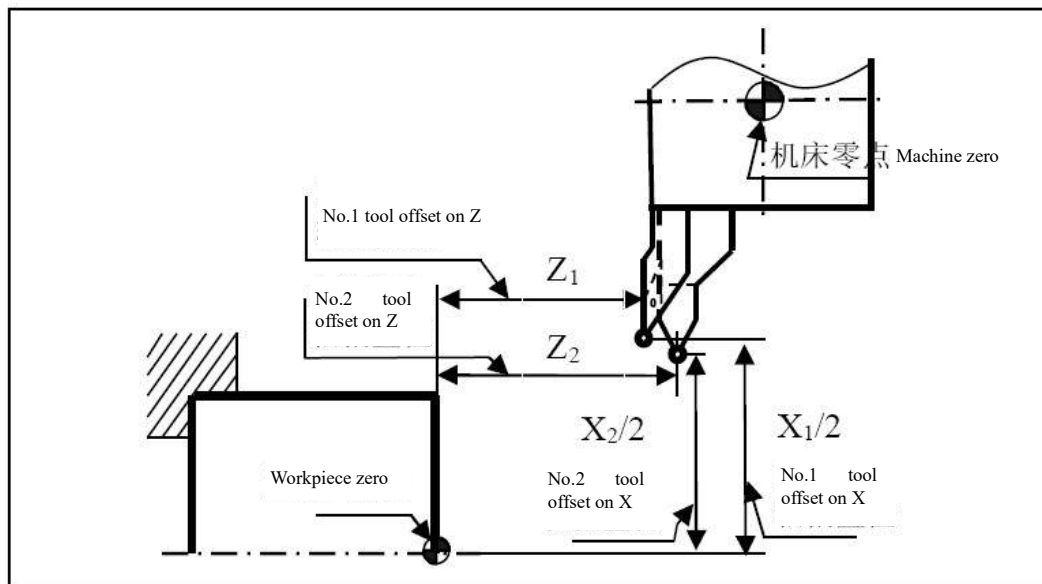
The HNC8 system adopts the absolute offset compensation mode. In the above figure, the absolute offset value of No. 1 tool is: X axis X1 (diameter value), Z axis Z1; the absolute offset value of No. 2 tool is: X axis X2 (diameter value), Z axis Z2.

During programming, the tool number and offset number of each tool are generally set to be the same for user to remember. As shown in the figure above, T0101 and T0202 are used to execute tool change and offset compensation. At this time, the offset values X1 Z1 of No. 1 tool must be stored in the No. 1 offset compensation register, and the offset values X2 Z2 of No. 2 tool must

be stored in the No. 2 offset compensation register.

When the system executes the tool offset compensation, the machining coordinate system of No.1 and No.2 tools are set as (X1, Z1) and (X2, Z2) respectively. Therefore, though the geometric dimensions of tools 1 and 2 are different, and the distances of both tools relative to workpiece zero are different when tool post is at the machine zero, the coordinate system established for each is coincident with the workpiece coordinate system.

4. Tool wear compensation

The error caused by tool wear is handled with wear compensation. The system stores the wear compensation amount and the offset compensation amount in the same register address number. Therefore, when the T command is used for offset compensation in the program, the position offset and wear stored in the register will be called for compensation.

5. Tool path for tool offset and wear compensation

The figure below shows the tool position point trajectory for the situation: first establishing the tool offset and the wear compensation, and then canceling the tool offset and the wear compensation.
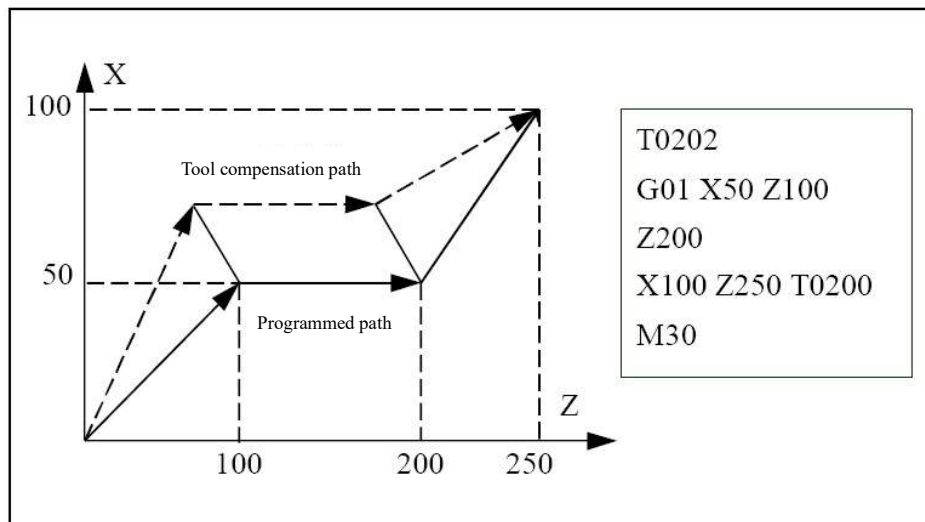
Example

N1 G00X100Z140
N2T0313 (No. 3 tool and No. 13 tool offset is selected)
N3X200Z150

As shown in the figure below, if the tool path has compensation values in the X and Z directions relative to the programmed path (the vector composed of the compensation components in the X and Z directions is called the compensation vector), then the end position in the block is added or subtracted by the compensation amount (compensation vector) specified by the T code is the end

position of the tool path section.
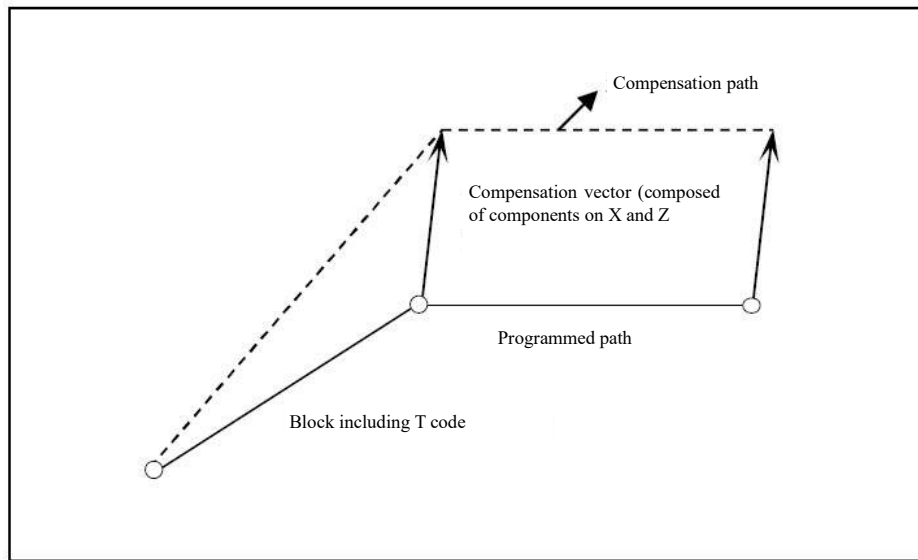
⚠️ **Note**

1) The tool offset compensation value can be automatically generated through the tool setting operation, or it can be manually input into the offset compensation register after tool setting and calculation. Please refer to the operation manual for specific operations.

2) Both the wear value and the offset value will be included in the program. If they are not needed, be sure to set to 0.

12.2   Turning Tool Nose Radius Compensation（G40/G41/G42）（T）

✎   **Function and Purpose**

The CNC program is generally aimed at a certain point on the tool which is the tool position to compile the path of the tool. The tool position of the turning tool is an imaginary tool nose point in an ideal state. This point is small enough and strong enough, and the workpiece contour is finally machined by this point. However, due to process or other factors, the actual lathe tool nose is often not an ideal point but a circular arc, so the actual cutting point of the tool (the final contact point between the tool and the workpiece) will be changed on the circular arc of tool nose. This position deviation between the actual cutting point and the tool position will inevitably cause overcutting or undercutting of the workpiece, as shown in the figure below.
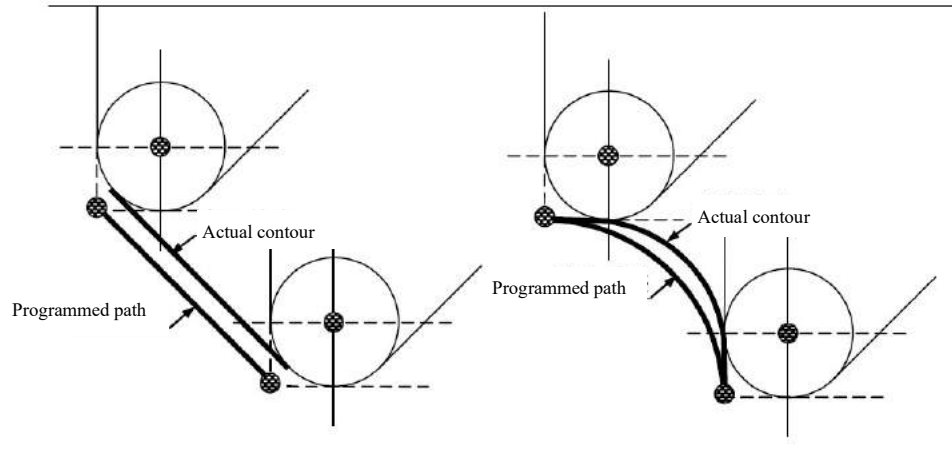
The machining error caused by the tool nose being not an ideal point but a circular arc can be eliminated by the tool nose radius compensation function.

Command Format

G code	Workpiece position	Tool path
G40	Tool nose radius compensation is cancelled	Move along tool path
G41	Tool nose radius compensation left	The left of the programmed path is compensated.
G42	Tool nose radius compensation right	The right of the programmed path is compensated.

Note: Radius compensation does not support the programs in interrupt format, such as G31.

Description

1. Imaginary tool nose point

➢ The imaginary tool nose point during programming;

➢ The point is assumed to be small enough and strong enough during programming, without interference and cutting point position change;

➢ The contour of the workpiece is formed by the final processing trajectory of this point;

➢ This point is the description point of the programmed path;

2. Tool position

➢ Standard imaginary tool nose point;

➢ This point is the description point of programmed path;

➢ This point is the tool setting point;

➢ 9 positions can be used as the tool position for each tool nose. There are a total of 10 numbers of tool position;

➢ No. 0 and No. 9 tool positions are the circular center of the tool, and the other No. 1 to 8 tool positions respectively correspond to a point position;

➢ The main principle of tool position selection is ease of operation;

➢ According to the numbering rules of tool position, there is a certain difference in the visual expression of tool positions between machine tools with front tool post and with rear tool post

3. Tool position of machine tool with front tool post
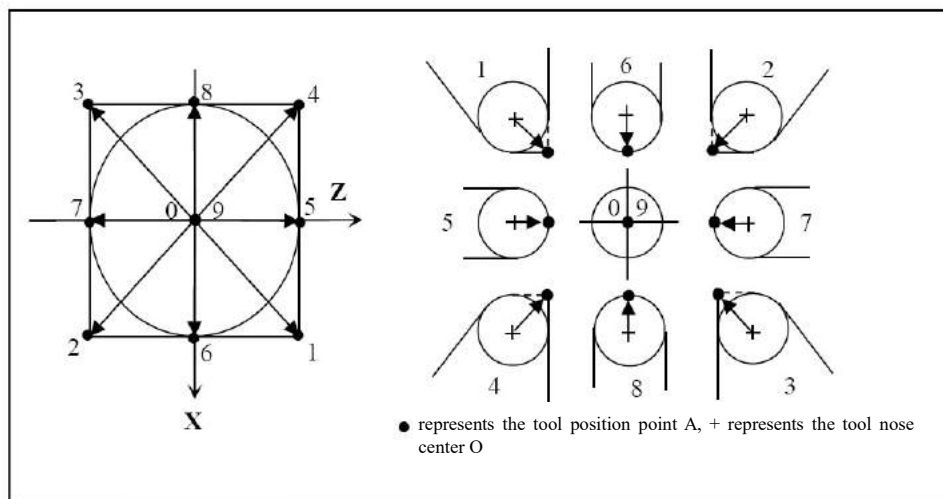
● represents the tool position point A, + represents the tool nose center O

4. Tool position of machine tool with rear tool post

● represents the tool position point A, + represents the tool nose center O
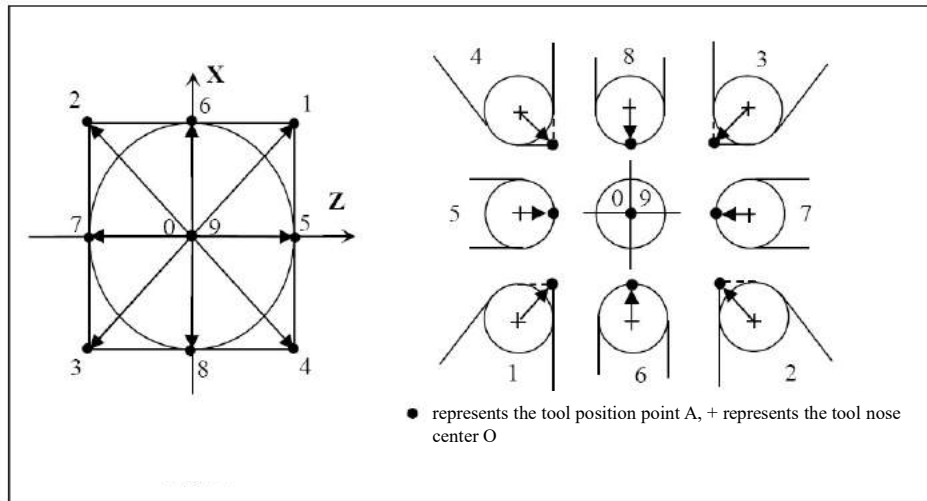
5. Tool nose radius compensation left/right（G41\G42）

➢ How to determine the right or the left,

✧ G41（compensation left）：On the positive side of the third axis, user views along the direction of tool movement, when the tool is on the left side of the tool path, tool nose radius compensation left is determined.

✧ G42（compensation right）：On the positive side of the third axis, user views along the direction of tool movement, when the tool is on the right side of the tool path, tool nose radius compensation right is determined.

➢ Tool nose radius compensation left/right for machine tool with front tool post and rear tool post

➢ Because the direction of the third axis for machine tools with the front and rear tool post, the directions of the tool nose radius compensation are different. The third axis of the machine tool with rear tool post points to the same direction as the customary direction; the third axis direction of the machine tool with front tool post is opposite to the customary direction. See the figure below for details

✧ Figure of tool nose radius compensation for tool machine with front tool post
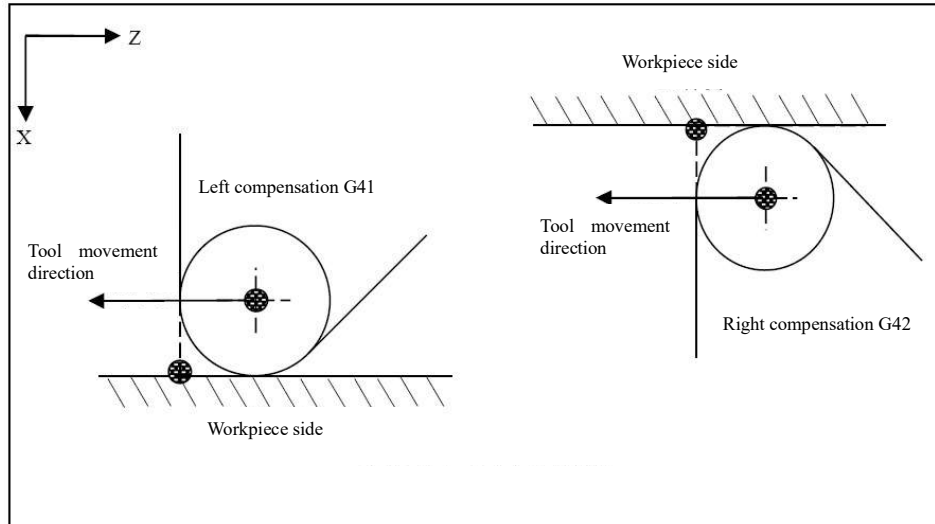
✧ Figure of tool nose radius compensation for tool machine with rear tool post
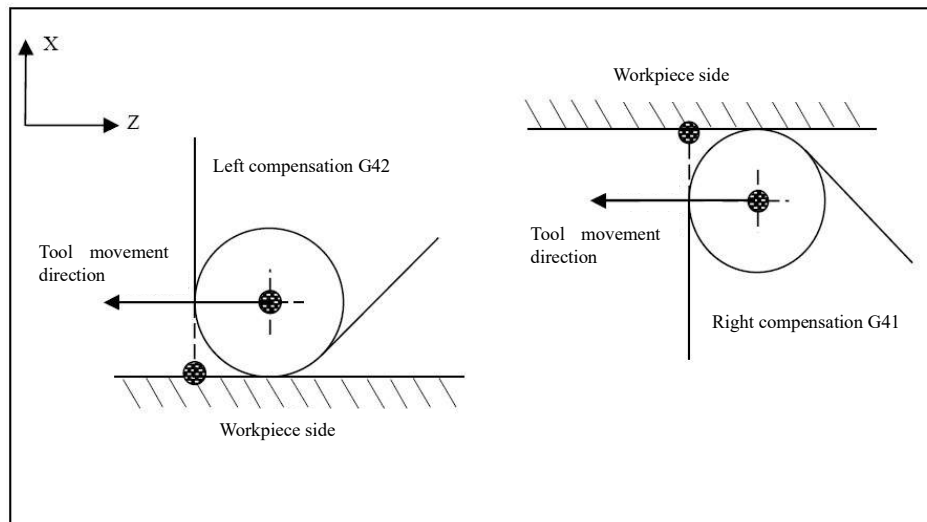
6. Add or cancel tool nose radius compensation

➢ The tool radius compensation can only be added or cancelled in the G00 or G01 block. When it is added or cancelled in the G02 or G03 block, the system will alarm;

➢ When adding or canceling tool radius compensation, it must be in the traverse command block and there is enough moving distance, otherwise overcutting will occur;

➢ When adding tool radius compensation, user needs to specify the corresponding tool nose radius compensation value. The register for storing the radius compensation value has the same number as the offset compensation register, so the tool offset number must be specified with the T command;

➢ After the execution of the tool radius compensation (G41/G42) block is completed, the center of the tool stops directly above the normal direction of the starting point of the lower path.

➢ Before executing the block of canceling tool radius compensation (G40), the center of the tool stops directly above the normal direction of the end point of the upper path.
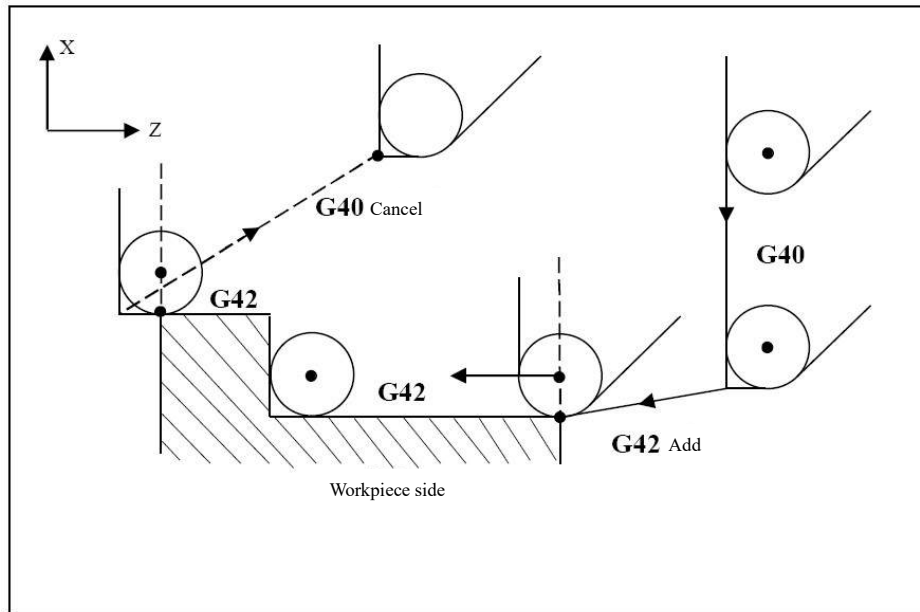
Example

Programming for the parts shown in below figure with tool radius compensation.
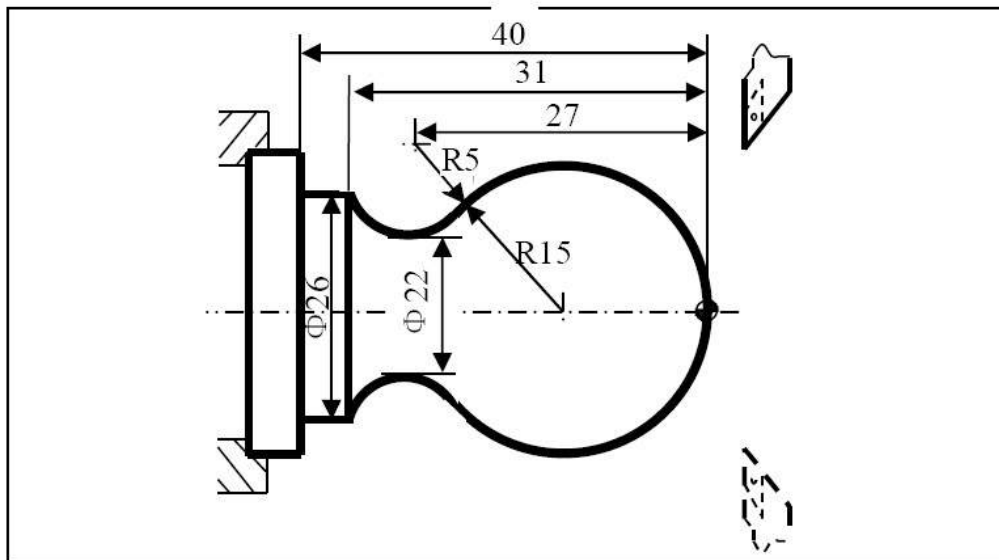
%3323

N1 T0101   (Change to the No. 1 tool and determine its coordinate system)

N2 M03 S400   (Spindle rotates CW at 400r/min)

N3 G00 X40 Z5　(Move to the starting position of program)

N4 G00 X0　(Tool moves to the workpiece center)

N5 G01 G42 Z0 F60　(Tool radius compensation is added, and move close to workpiece)

N6 G03 U24 W-24 R15　(Process R15 arc)

N7 G02 X26 Z-31 R5　(Process R5 arc)

N8 G01 Z-40　(Process Φ26 outer circle)

N9 G00 X30　(Exit the processed surface)

N10 G40 X40 Z5　(Cancel radius compensation, return to the starting position of program)

N11 M30　(Spindle stops, main program ends and resets)

Note

(1) G40, G41, and G42 are all modal codes, which can be mutually cancelled.

(2) Only G00 or G01 instead or G02 or G03 can be used for the creation and cancellation of tool nose radius compensation.

(3) When using G41 or G42 to enable radius compensation, the corresponding radius compensation register with radius compensation value must be designated. The radius compensation register number is the same as the tool offset compensation register number.

(4) There must be a traverse command and a traverse distance in the block where the tool radius compensation Enable (G41/G42) or the tool radius compensation Disable (G40) is added.

12.3　Milling Tool Length Compensation（M）

Function and Purpose

It is to set the difference between the tool length during programming and the actual tool length in the tool length compensation register. Calling this function can move the tool to the programmed end position without modifying the program, thus making the program universal.
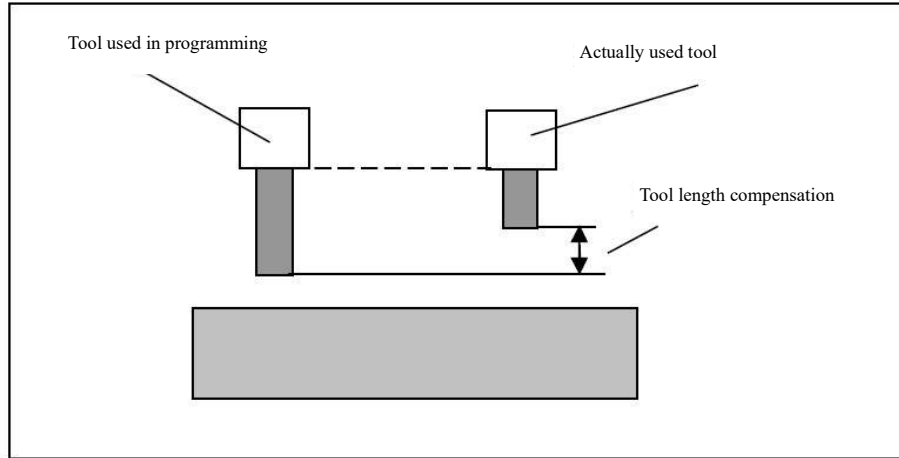
Command Format

G43/G44 H_ Z_

…...

G49

Tool length compensation is specified with G43 and G44.

Parameter	Meaning
G43	Tool length compensation plus (The tool length compensation value is added to the theoretical position in the tool axis direction)
G44	Tool length compensation minus (The tool length compensation value is subtracted from the theoretical position in the tool axis direction)
H	The number of tool length compensation amount in the tool compensation table
G49	Cancel tool length compensation

Description

Tool length compensation type

(1) According to the tool length compensation axis, the following two tool length compensation types can be used,

Type	Description	Format
A	Tool length compensation along basic Z axis	G43/G44 Z_H_
B	Tool length compensation along the vertical direction of the selected plane G17：XY plane；G18：ZX plane； G19：YZ plane；	G17 G43/G44 Z_H_ G18 G43/G44 Y_H_ G19 G43/G44 X_H_

(2) The tool length compensation types A and B are set by the parameters, and the related parameter are as below,

Parameter	Description
000012	Tool axis selection mode

000012：This parameter is used to determine which axis the G43/G44 tool length compensation function should be performed on.

0: The tool length compensation is always performed on Z axis;

1: Tool length compensation axis is switched based on the coordinate plane selection (G17/G18/G19) corresponding to Z/Y/X-axis respectively.

Movement amount of tool length compensation

(1) Movement amount of tool length compensation: when G43 or G44 tool length compensation is executed, the movement amount is calculated according to the below formula:

G43 Z_ H_；Z_ + H_（tool length compensation）Compensaiton in + direction

G44 Z_ H_；Z_- H_（tool length compensation）Compensation in - direction

As shown in the above calculations, regardless of whether the absolute command or the incremental value is used, the actual end point is of the coordinates which is compensated with the specified compensation amount based on the end point coordinates of programmed traverse command

(2) When there is length wear in the tool compensation table: for the execution of tool length compensation command G43 or G44, the movement amount is calculated according to the following formula

G43 Z_ H_；Z_ + H_（tool length compensation）+ H_（tool length wear）

G44 Z_ H_；Z_- H_（tool length compensation）- H_（tool length wear）

(3) When there is coordinate value in the workpiece coordinate system: for the execution of tool length compensation command G43 or G44, the movement amount is calculated according to the following formula

G43 Z_ H_；Z（workpiece coordinate system）+ Z_ + H_（tool length compensation）

G44 Z_ H_；Z（workpiece coordinate system）+ Z_- H_（tool length compensation）

(4) When there is coordinate value in the workpiece coordinate system and length wear in the tool compensation table: for the execution of tool length compensation command G43 or G44, the movement amount is calculated according to the following formula

G43 Z_ H_；Z（workpiece coordinate system）+ Z_ + H_（tool length compensation）+ H_（tool length wear）

G44 Z_ H_；Z（workpiece coordinate system）+ Z_- H_（tool length compensation）- H_（tool

length wear）

(5) When there is coordinate value in the workpiece coordinate system, and there is a coordinate value of external zero offset: for the execution of tool length compensation command G43 or G44, the movement amount is calculated according to the following formula

G43 Z_ H_；Z（workpiece coordinate system）+ Z（external zero offset）+ Z_ + H_（tool length compensation）

G44 Z_ H_；Z（workpiece coordinate system）+ Z（external zero offset）+ Z_ - H_（tool length compensation）

(6) When there is coordinate values in workpiece coordinate system and external zero offset, and there is length wear in the tool compensation table: for the execution of tool length compensation command G43 or G44, the movement amount is calculated according to the following formula

G43 Z_ H_；Z（workpiece coordinate system）+ Z（external zero offset）+Z_ + H_（tool length compensation）+ H_（tool length wear）

G44 Z_ H_；Z（workpiece coordinate system）+ Z（external zero offset）+Z_ - H_（tool length compensation）- H_（tool length wear）

Tool length compensation number

(1) The working range of the compensation number depends on the parameter setting. The relevant parameters are as follows:

Parameter	Description
000060	Number of tools stored in system

NC000060: This parameter is to set the number of tool of which the data (tool offset, wear, radius, tool nose direction, etc.) is saved in the tool compensation table. The value must be greater than or equal to the sum of the tools in each channel.

Max. value：1000   Default：100   Min. value：0

(2) When the commanded compensation number exceeds the range, the system will give an alarm of "illegal tool compensation number".

(3) After the compensation number specified in G43 or G44 block, it cannot become the modal called later to be effective.

For example, H1 modal in N1 line cannot take effect in N4 line in the following program.

N1 G43 Z0 H1；   Perform tool length compensation via H1

N2 G0 X0 Y0；

N3 G49 Z0；   Tool length compensation is cancelled

N4 G43 Z0；   Tool length compensation will not be performed through H1 again, and the compensation number must be specified again.

(4) In the G43/G44 modal, when G43/G44 is commanded again, the compensation will be performed according to the new compensation number, that is, the new offset value will not be added to the old offset value when the offset number is changed.

For example, the N2 line of the following program is compensated according to the new compensation number H2.

N1 G43 Z0 H1； Tool length compensation is performed via H1

N2 G43 Z0 H2； Tool length compensation is performed with the new compensation number H2

Cancel the tool length compensation

(1) After the system restarts and M02, M30, G49 are executed, tool length compensation is cancelled.

(2) H0 is specified, tool length compensation is cancelled.

(3) When Reset or Emergency Stop is pressed during tool length compensation modal, tool length compensation is cancelled.

(4) When tool length compensation type B is used, the following items must be noted for canceling tool length compensation with G49.

(a) When the length compensation is cancelled with G49, only the length compensation in the last selected plane before G49 command is cancelled;

For example, in the below program, G49 in N6 line only cancels the length compensation in G19 plane.

N1 G28 X0 Y0 Z0；

N2 G92 X0 Y0 Z0；

N3 G17 G43 Z0 H1； Tool length compensation in XY plane is performed via H1

N4 G18 G43 Y0 H2； Tool length compensation in ZX plane is performed via H2

N5 G19 G43 X0 H3； Tool length compensation in YZ plane is performed via H3

N6 G49； The length compensation only in G19 plane is cancelled

N7 G0 X0 Y0 Z0；

(b) When using G49 to cancel the length compensation in all planes, the plane selection command (G17/G18/G19) must be specified after the G49 command, and it must be specified in a separate line;

For example, the correct programming based on the above is as following

N1 ......   or   N1 ......

   ....

N6 G49 G17；　　　　　　　　N6 G49 G17；

N7 G49 G18；　　　　　　　　N7 G18；

N8 G49 G19；　　　　　　　　N8 G19；

N9 ......　　　　　　　　　　　N9 ......

Wrong way: G49 G17 G18 G19, only the length compensation in the last plane selected before the G49 command.

Actions when other commands are executed in the tool length compensation modal

(1) Whether the tool length compensation will be restored after G53 or G28 is executed in the tool length compensation modal depends on the parameter setting. The relevant parameters are as follows:

Parameter	Description
000014	Whether tool length compensation is restored after G53/G28

000014：This parameter is to set whether to automatically restore the tool length compensation function after executing G53/G28 commands.

　　0: After executing G53/G28 command, the tool length compensation function will not be restored automatically.

　　1: After executing G53/G28 command, the tool length compensation function will be restored automatically.

(2) Whether the tool length compensation will be restored after the G30 command is executed in the tool length compensation modal depends on the setting of parameter 000014

(3) When G29 is executed in tool length compensation modal, the tool length compensation will not be cancelled, and cannot be controlled by 000014 parameter.

For example, the machine coordinate is Z55.0 after N4 line is executed when H1=50 in the following program

　　N1 G28 Z10；
　　N2 G90 G92 Z0；
　　N3 G43 Z0 H1；　Tool length compensation is performed via H1.
　　N4 G29 Z5；　Machine coordinate is Z55.0. Machine coordinate is Z-45.0 when G44 is used.
　　N5 G49；　Tool length compensation is cancelled.

(4) When G54 to G59 commands are executed in the tool length compensation modal to change the workpiece coordinate system, the tool length compensation will not be cancelled, and the end point coordinates of the compensation axis will be recalculated based on the given coordinate system.

Example

Create the machining program for the parts shown in the figure below with tool length compensation. Requirements are as follows:

(1) Set the workpiece coordinate system in the G54 coordinate system based on the workpiece zero as shown in the figure

(2) Process according to the path 1 to 13 indicated by the arrow

(3) Write out the coordinate value in the workpiece coordinate system and the compensation value in the tool compensation table before programming

(4) Write the comment and machine coordinate of each line of program

(5) The program must be safe and reliable
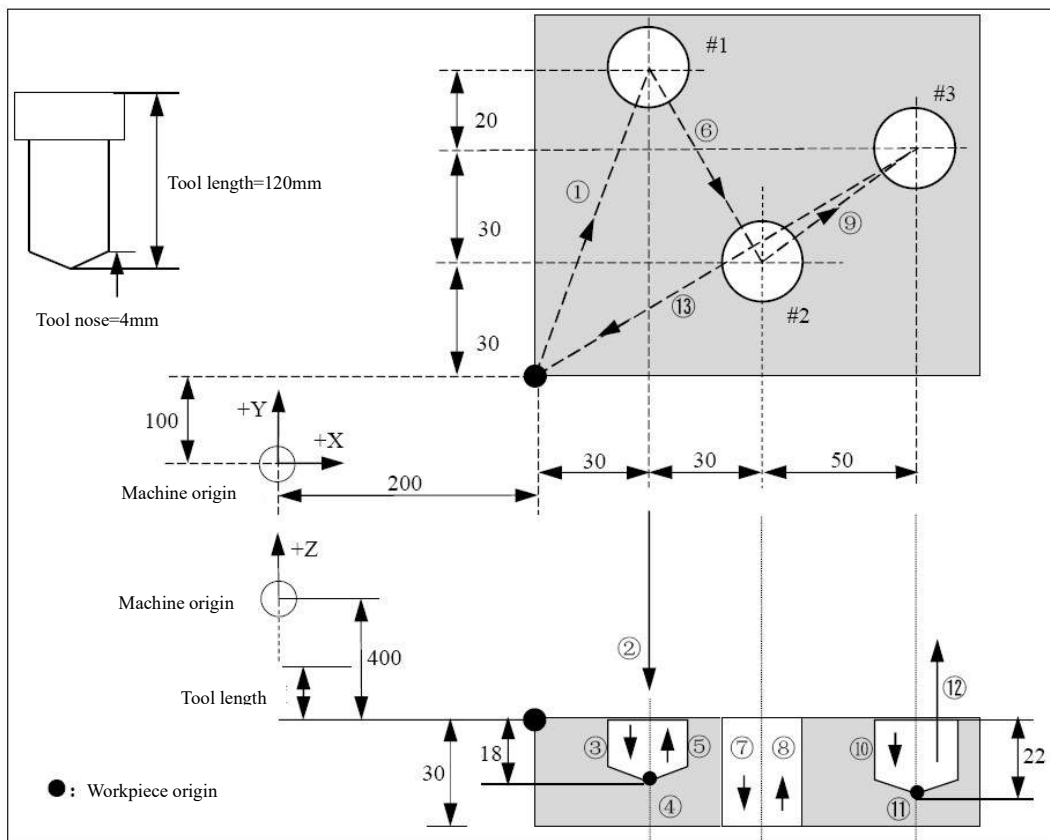
Workpiece coordinate system：G54（X200，Y100，Z-280） 　　Compensation value：H01=-120

%1234
G54 G90 M03 S600 ； Select workpiece coordinate system G54, use absolute command G90 to program, spindle rotates CW
G00 Z50 ； Rapid traverse to Z50, （Z-230）

X0 Y0 ；X0 Y0, Rapid traverse to X0 Y0, （X200，Y100）

X30 Y80 ；①, Rapid traverse to X30 Y80（X230，Y180）

G43 Z5 H01 ；②, Establish tool length compensation, and rapid traverse to Z25， （Z-395）

G01 Z-18 F300 ；③, Z-18Process #1 hole to Z-18， （Z-418）

G04 P2000 ；④, Pause for two seconds at the bottom of the hole

G00 Z5 ；⑤, Rapid traverse to Z5， （Z-395）

X60 Y30 ；⑥, Rapid traverse to X60 Y30， （X260，Y130）

G01 Z-40 ；⑦, Process #2 hole to Z-40， （Z-440）

G00 Z5 ；⑧, Rapid traverse to Z5， （Z-395）

X110 Y60 ；⑨, Y60Rapid traverse to X110 Y60， （X310，Y160）

G01 Z-22 ；⑩, Process #3 hole to Z-22， （Z-422）

G04 P2000 ；⑪, Pause for two seconds at the bottom of the hole

G49 G0 Z50 ；⑫, Cancel the tool length compensation, and rapid traverse to Z50， （Z-230）

X0Y0 ；⑬, Rapid traverse to X0 Y0， （X200，Y100）

M05 ； Spindle stops

M30 ； Program ends

⚠ Note

(1) The direction of tool length compensation is always perpendicular to the plane selected by G17/G18/G19.

(2) The compensation number H_ must be given after G43/G44 command, otherwise the system will issue an alarm of "No compensation number specified".

(3) The compensation axis (X/Y/Z) must be specified after G43/G44 command, or the traverse command of compensation axis must be specified before the next G43/G44 command, otherwise the length compensation will be invalid.

(4) G43/G44/G49 are all modal codes and can be mutually cancelled, but they can be mutually cancelled only when they are not specified in the same line. When G43 and G44 are specified in the same line, the system will alarm; when G49 is specified in the same line as G43/G44, G49 will not take effect.

(5) G43/G44/G49 For G43/G44/G49, after startup, emergency stop and reset, G49 state is the default.

(6) When using G43/G44 to specify H0, the tool length compensation will be cancelled, but the modal display area is still G43/G44, and will not be changed to G49, so when H_ is given separately at this time, the tool length compensation will take effect again.

(7) When G49 is not used, use H0 to cancel the tool length compensation, and then add the tool length compensation with H_; When H0 is used to cancel the tool length compensation, there is no tool length compensation until H_ is used again.

For example,

N1 G43 Z0 H1	;	Tool length compensation is performed with H1
N2 G43 Z0 H0	;	Tool length compensation is cancelled with H0
N3 G0 Z5	;	There is no tool length compensation
N4 H2	;	Tool length compensation is performed with H2
N5 G0 Z0	;	There is tool length compensation
N6 G49	;	Tool length compensation is cancelled

(8) After the tool length compensation is set in MDI, it will always be valid for the subsequent MDI program. The tool length compensation will only be cancelled after HO is specified with reset, emergency stop and MDI, as well as G49 command being executed on MDI interface.

(9) After the tool length compensation is set in MDI, it will not be effective for the program in the program file. After switching from the MDI interface to the program interface, the modal becomes the G49 modal, so it will not be effective.

(10) When the G43/G44 tool length compensation program is stopped with reset or emergency stop, the tool length compensation will be cancelled. When the program is stopped with the dwell or switching mode, the tool length compensation will not be cancelled.

(11) After the tool length compensation is set with G43/G44, the tool length compensation will be performed based on   the corresponding H number.

(12) In the auto mode, when using the breakpoint save and restore functions after the tool length compensation command, the system can correctly establish the tool length compensation after the program resumes execution. If there are multiple G43/G44 tool length compensation commands in the program, the system can also correctly establish tool length compensation through the corresponding G43/G44 commands after the breakpoint is restored.

(13) After the tool length compensation command is used to specify the line operation with any line function, the system can correctly establish the tool length compensation; If there are multiple G43/G44 tool length compensation commands in the program, the system can also correctly establish tool length compensation through the corresponding G43/G44 commands.

For example, when the following program runs from line N3 with any line function, the system can correctly establish tool length compensation

```
N1 G43 Z0 H1
N2 G0 Z5
N3 G0 Z10；    Use any line function to run from this line
N4 G0 Z15
N5 G0 Z20
N6 G49；
```

12.4　Milling Tool Radius Compensation（G40/G41/G42）（M）

　Function and Purpose

When the CNC machine tool mills the contour of the workpiece, the tool center is usually used as the path for easy programming. The tool radius is not considered when programming, and the tool path is programmed based on the contour size of the workpiece. But there will be an offset (tool radius) between the actual tool path and the workpiece contour. Therefore, the offset of the tool center path is needed during programming to make the tool path consistent with the workpiece contour, which is called tool radius compensation function.

In addition, the tool radius is changed due to tool change or tool wear. With this function, user can directly modify the corresponding radius compensation value or wear value in the tool compensation table without modifying programming.

After the tool radius compensation is performed, the tool path is shown as the dotted line in the figure below:
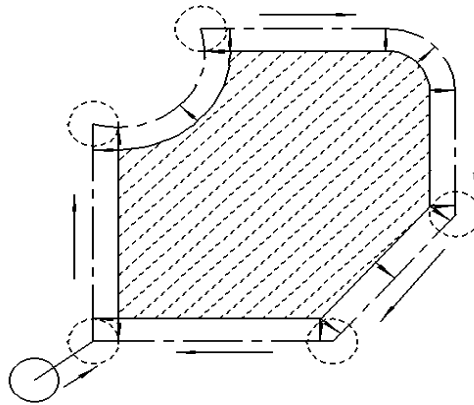
　Command Format

$$\text{G17 G41/G42} \begin{cases} \text{G00 X\_ Y\_ D\_} \\ \text{G01 X\_ Y\_ D\_} \end{cases}$$

$$\text{G18 G41/G42} \begin{cases} \text{G00 X\_ Z\_ D\_} \\ \text{G01 X\_ Z\_ D\_} \end{cases}$$

$$\text{G19 G41/G42} \begin{cases} \text{G00 Y\_ Z\_ D\_} \\ \text{G01 Y\_ Z\_ D\_} \end{cases}$$

Parameter	Meaning
G17/G18/G19	To specify compensation plane, XY, XZ, ZY-plane respectively.

G41/G42	Tool radius compensation is valid. G41: Tool radius compensation Left; G42: Tool radius compensation Right
D	To specify the compensation number

📖 **Description**
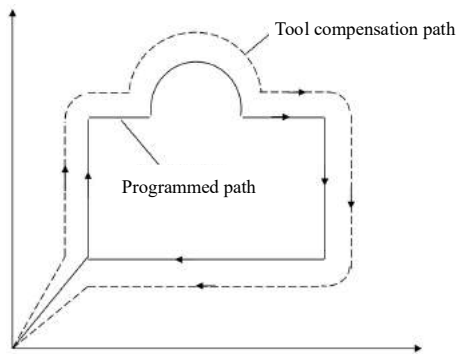
1) Tool radius compensation direction

Description:
G41 Tool radius compensation Left
Viewed from the positive direction to the negative direction of the third axis that is not in the machining plane, the tool is on the left side of the workpiece relative to the direction of tool movement

a) Left compensation

G41：Offset is performed to the left side of the tool path (as shown in the figure a below).

G42：Offset is performed to the right side of the tool path (as shown in the figure b below).
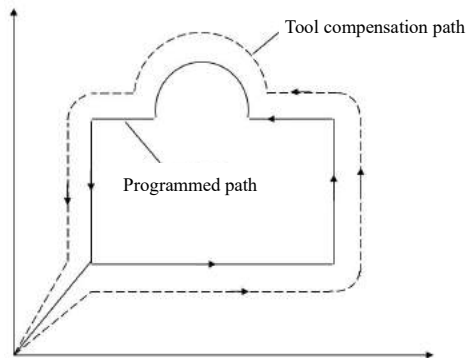
Description:
G42 Tool radius compensation Right
Viewed from the positive direction to the negative direction of the third axis that is not in the machining plane, the tool is on the right side of the workpiece relative to the direction of tool movement

b) Right compensation

2) Tool radius compensation number

The effective range of tool compensation numbers in this system defaults to 99 groups, and the effective range of compensation numbers can be set by NC parameter 000060 (number of tool data saved by the system).

3) Tool radius compensation plane selection

Offset plane	Plane selection	IP
XY	G17	X_Y_
ZX	G18	X_Z_

15

YZ	G19	Y_Z_

The radius compensation calculation is performed on the plane determined by the G17/G18/G19 commands. The plane on which the compensation calculation is performed is called the compensation plane. The axis coordinate values that are not in the compensation plane will not be compensated. The projection of the tool path in each plane is compensated at the time of simultaneous 3-axis control.

The compensation plane must be switched when radius compensation is cancelled. If the plane is switched during compensation, the alarm "Coordinate plane cannot be switched during tool radius compensation" will appear and the machine tool will stop.

4) Tool radius compensation amount setting

Use the D code to call the compensation amount set in the tool compensation table by specifying the number of the tool radius compensation amount.

The D code remains valid until another D code is designated.

The change of the tool radius compensation amount is usually carried out in the G40, reset or tool change mode.

In general, the compensation amount is positive (+). If the compensation amount is negative (-), G41 and G42 are interchanged. If the compensation amount is positive, the tool center moves around the outer contour of the workpiece, then if the compensation is negative, it will move around the inner side, as shown in the figure below.

When the compensation amount is positive, the tool path is shown in Figure (1). If the compensation amount is changed to a negative value (-), the tool path is shown in Figure (2). Therefore, for the same program, both male and female shapes can be processed by the positive and negative values of the compensation, and the gap between them can also be adjusted with the compensation.
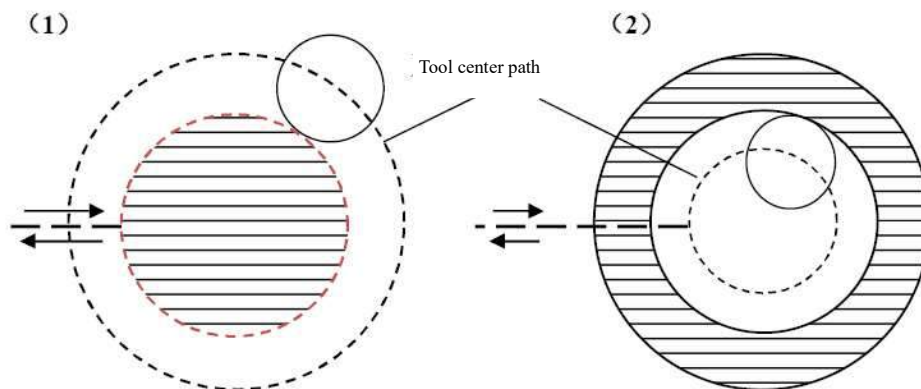
Example
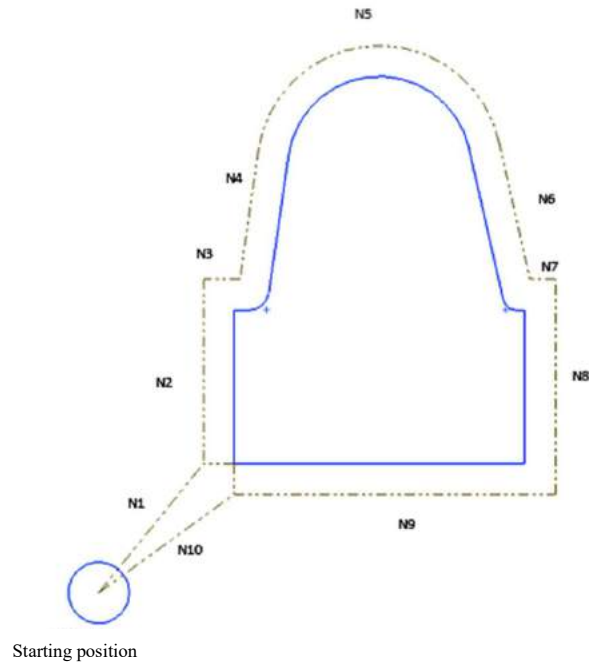
Starting position

G92 X0 Y0 Z0；   The absolute value is specified, and the tool is at the starting point

N1 G90 G17 G00 G41 D01 X25 Y25；   Tool radius compensation (tool starts) starts

N2 G01 Y50 F150；   Process from N2

N3 X30；   Process from N3

N4 G03 X40 Y60 R10.0；   Process from N3 to N4

N5 G01 X50 Y90；   Process from N4

N6 G02 X90 Y90 R20；   Process from N5

N7 G01 X100 Y60；   Process from N6

N8 G03 X110 Y50 R10；   Process from N6 to N7

N9 G1 X120；   Process from N7

N10 X120 Y25；   Process from N8

N11 X25 Y25；   Process from N9

N12 G00 G40 X0 Y0；   Cancel tool radius compensation, and tool returns to the starting
                           position (X0,Y0,Z0)

Note

1) The length compensation H command is ignored in the tool radius compensation, and only the D command is valid.

2) The plane selection for tool radius compensation is specified by G command (G17/G18/G19), and only the axes in the specified plane are compensated, the axes not in the plane are not compensated.

12.5 Detailed Explanation of Tool Radius Compensation

12.5.1 Tool Radius Compensation Action

Function and Purpose

The process of tool radius compensation is divided into three steps: tool compensation establishment, tool compensation in progress and tool compensation cancelation. As shown below:
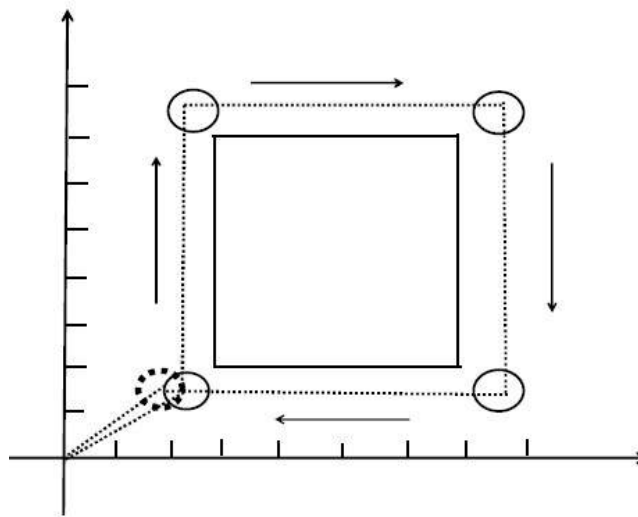
Description

Tool radius compensation establishment and start

When the compensation is canceled, and the radius compensation command is used to establish tool radius compensation, if the following compensation establishment conditions are not met, there may occur compensation overcutting, undercutting or alarms.

(1) The establishment of tool radius compensation mode is valid only in G00/G01 traverse command mode;

(2) The tool radius must be established on the axis where movement distance is not 0 in the

compensation plane. The compensation cannot be established when only the third axis movement is performed; however, in special cases, the system also supports the establishment of tool compensation with a movement amount of 0, and in order to prevent errors, it is recommended to give a movement greater than the tool radius value when creating tool compensation.

(3) The establishment is usually performed with G01 to ensure the safety of the tool and workpiece

(4) When the sign of the compensation value changes, the compensation direction of G41/G42 will also change.When the compensation value of G41 is positive, the compensation direction is on the left, and when the compensation value is negative, the compensation direction is on the right. G42 is also like this.

(5) When using tangential cutting or normal cutting to establish tool compensation, if the establishment conditions are not met, auxiliary line segments can be added to establish tool compensation when cutting into the workpiece.

(6) The number of tool radius compensation is: 0＜D＜maximum compensation number.

(7) The establishment of tool radius compensation in circular interpolation (G02/G03) is invalid, the system will alarm, and the machine stops moving

(8) In the process of radius compensation, G00 (rapid traverse positioning), G01 (linear interpolation) or G02/G03 (circular interpolation) is used to realize compensation. If two or more blocks where the tool does not move (such as auxiliary functions, pause, etc.) are executed during radius compensation, there will occur overcutting or undercutting.

In the process of tool radius compensation

(1) When starting the radius compensation, whether it is in auto mode or single-block mode, three traverse commands must be read in. If there is no three traverse commands, the execution will be performed after reading up to 5 consecutive blocks.

(2) When the radius compensation is valid, the compensation plane must not be switched, otherwise it will alarm and stop the movement.

(3) In the state of radius compensation, the linear movement of the milling cutter and the radius for the inner circular cutting must be greater than or equal to the radius of the milling cutter, otherwise it may cause interference and overcutting.

When tool radius compensation is cancelled

The cancellation of tool radius compensation needs to be executed in the G00 or G01 block. When any of the following conditions is met, the system enters the radius compensation cancellation state.

(1) When the machine is turned on, the system will initialize and restore the default tool compensation cancellation state.

(2) After machine is reset.

(3) The tool compensation cancellation command G40 is executed.

(4) M02 and M30 commands with the reset function are executed.

(5) When the block where the tool radius compensation offset number is 0 is executed.

Example

%0001
G54G90G17G0Z50
X0Y0
M3S10000
G41G01X30Y20D01 ； Tool radius compensation is established, and register value of
                      No.1 tool compensation is called
Y70
X70
Y20
X30
G40G01X0Y0 ； Tool radius compensation is cancelled.
M30

Note

It is invalid when the radius compensation is cancelled by the circular command (G02/G03). If the circular command is designated for the cancellation, an alarm will be issued and the machine tool will stop moving.

12.5.2 Tool Radius Compensation Action Diagram

Function and Purpose

Inside and outside: When the included angle of the tool path established by two programs exceeds 180°, the path is referred to as "inside". When the included angle is between 0° and 180°, it is referred to as "outside".

Inside	Outside
Workpiece, Programmed path, α, **180° ≤ α**	Programmed path, Workpiece, α, **0° ≤ α ＜180°**
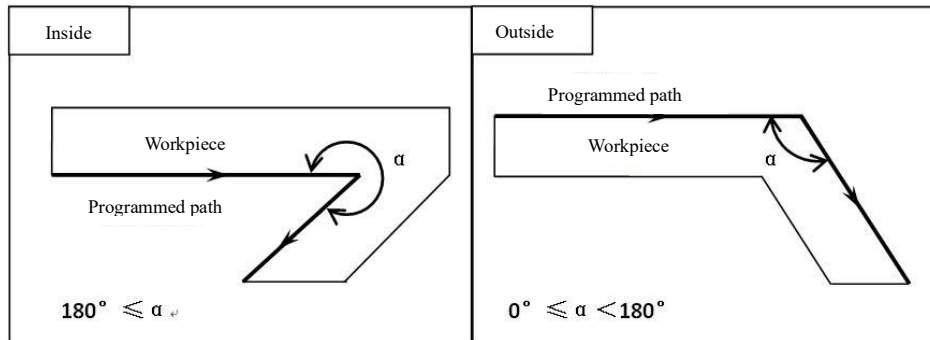
📖 **Description**

Action of tool radius compensation establishment and start

Meaning of sign

—S indicates that this position is the starting point for cutting.

—L indicates that the tool moves in a straight line.

—C indicates that the tool moves along an arc.

—r indicates tool radius compensation value.

(1) The tool moves around the inside of the corner (α≥180°)
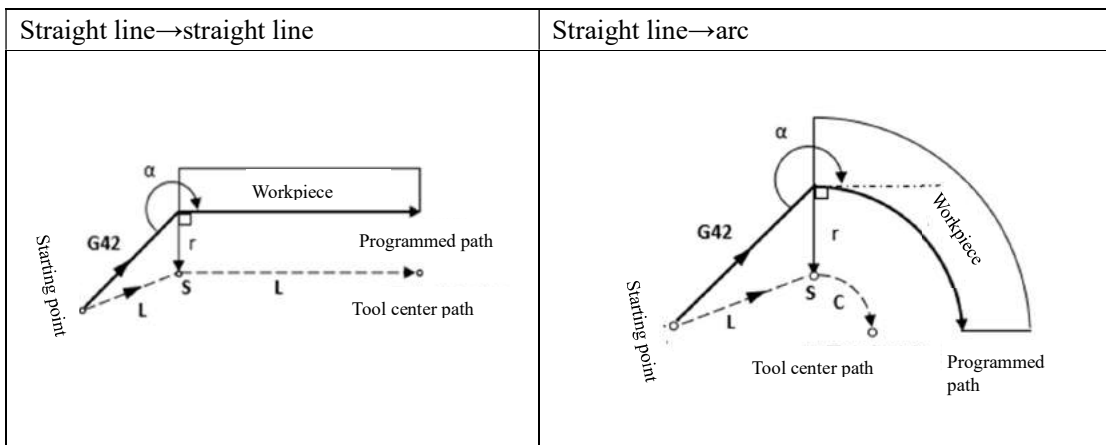
Straight line→straight line	Straight line→arc

(2) The tool moves around the outside of the corner (90°≤α＜180°)

Straight line→straight line	Straight line→arc

(3) The tool moves around the inside of the corner （α＜90°）
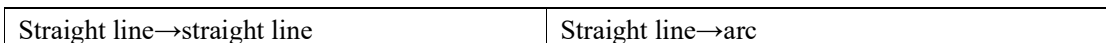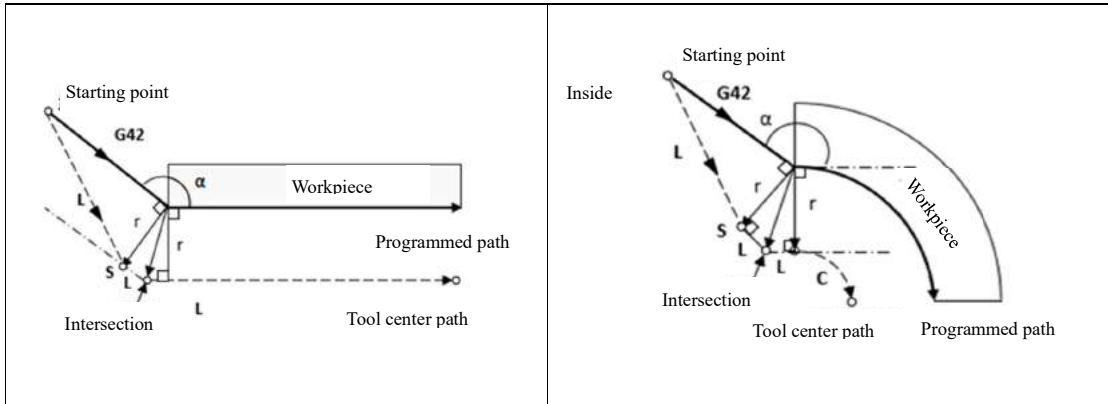
Straight line→straight line	Straight line→arc
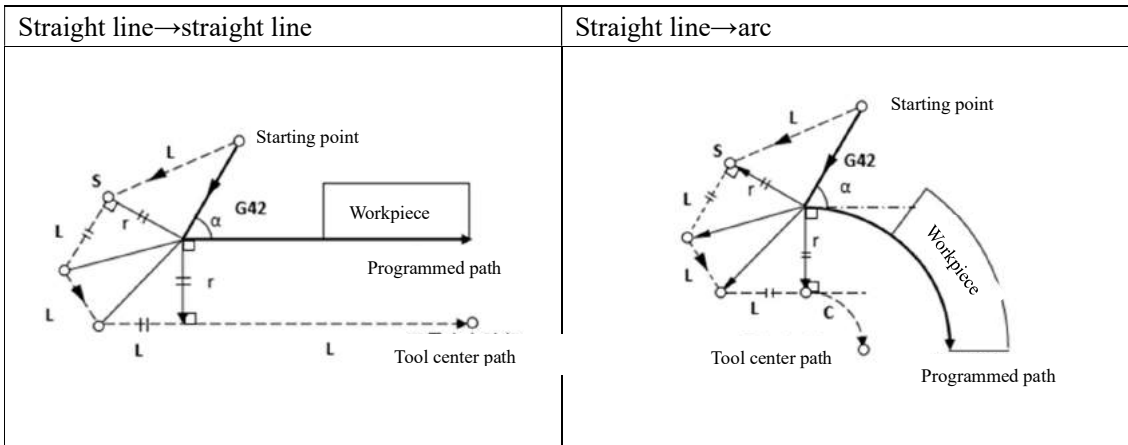
(4) The tool moves around the outside of the acute corner less than 1° (α<1°)

Straight line→straight line
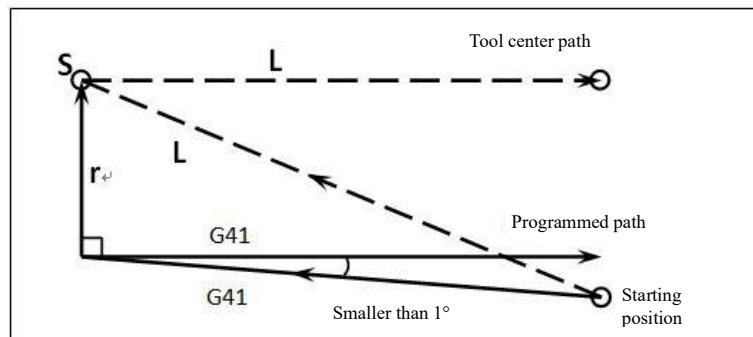
(5) There is no tool movement command in the starting block

If there is no tool movement command in the starting block, no offset is established.

G91 G40 .... N6 X100 Y100 N7 G41 X0 N8 Y-100 N9 Y-100 X100	
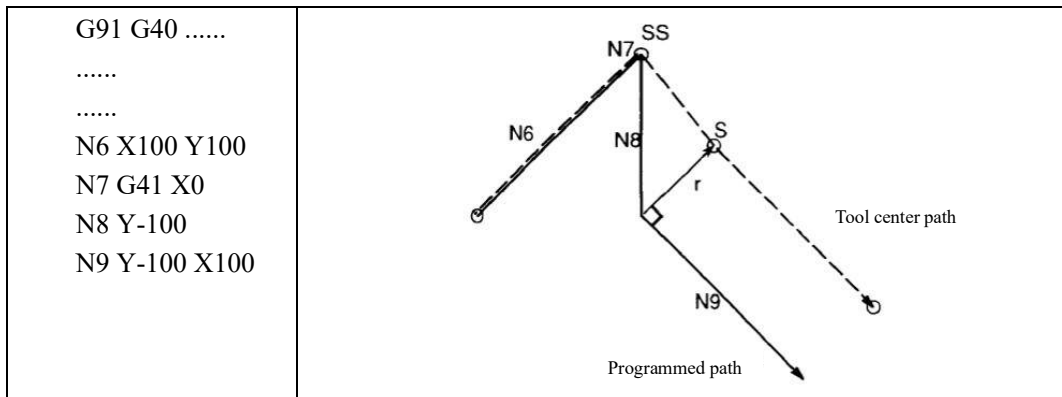
Action of tool radius compensation in progress

For the programmed path (G00, G01, G02, G03), the system calculates the tool path from a straight line/arc, and performs tool radius compensation.

During the compensation, if the same compensation command (G41/G42) is specified, the second compensation command will be ignored.

During compensation, if five blocks without movement are consecutively specified, overcutting or undercutting will occur.

In tool radius compensation, when M00 is designated, read-ahead is prohibited.

（1）The tool moves around the inside of the corner (α≥180°)
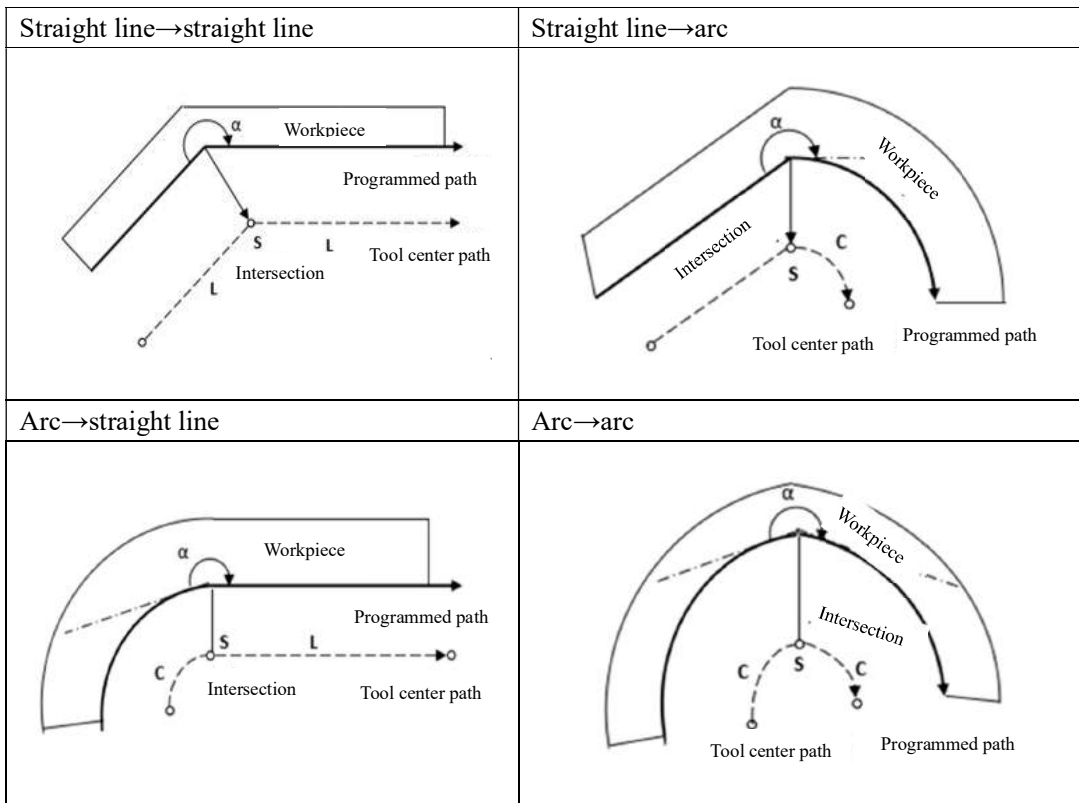
Straight line→straight line	Straight line→arc
Arc→straight line	Arc→arc

（2）The tool moves around the outside of the corner (90°≤α＜180°)

Straight line→straight line	Straight line→arc
α Workpiece Programmed path S Intersection L Tool center path L	Programmed path α Tool center path L r Workpiece S L C Intersection
Arc→straight line	Arc→arc
α Workpiece r Programmed path C S L Intersection L Tool center path	Programmed path α r r Workpiece Tool center path α C S L L L C Intersection

（3）The tool moves around the inside of the corner（α＜90°）

Straight line→straight line	Straight line→arc
L L r α Workpiece S r Programmed path L L r L L Tool center path	L L r α S r Workpiece L L r L C Tool center path Programmed path
Arc→straight line	Arc→arc
C S r α Workpiece L L r L r Programmed path L Tool center path	Programmed path C S L r α L r Workpiece L L C Tool center path

Action at the time of tool radius compensation cancellation

To cancel the tool radius compensation, there must be a movement command other than the circular command. If the radius compensation is canceled in the circular command, the system will issue an alarm.

When any of the following conditions is met, the tool radius compensation is cancelled.

a) G40 is executed.

b) Compensation number D00 is executed.

(1) In the block where there is movement cancellation, when the tool moves around the inside of the corner ($\alpha \geq 180°$)

Straight line→straight line	Straight line→arc
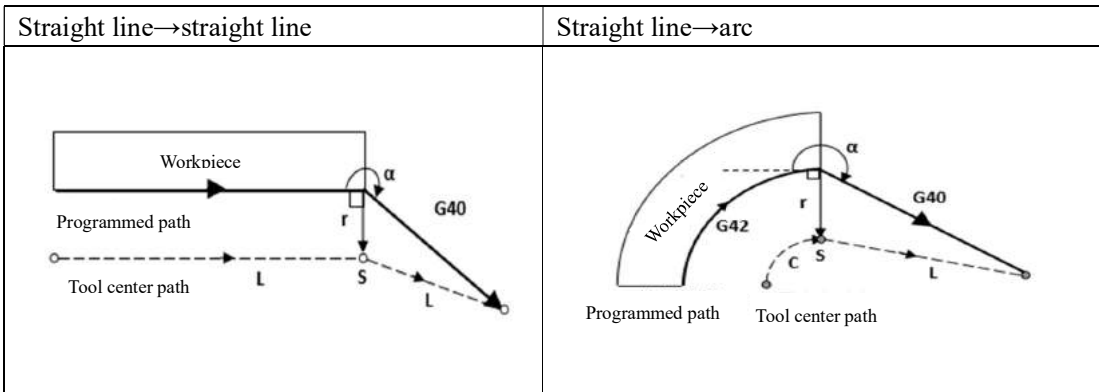
(2) In the block where there is movement cancellation, when the tool moves around the outside of the obtuse angle ($90° \leq \alpha < 180°$)

Straight line→straight line	Straight line→arc
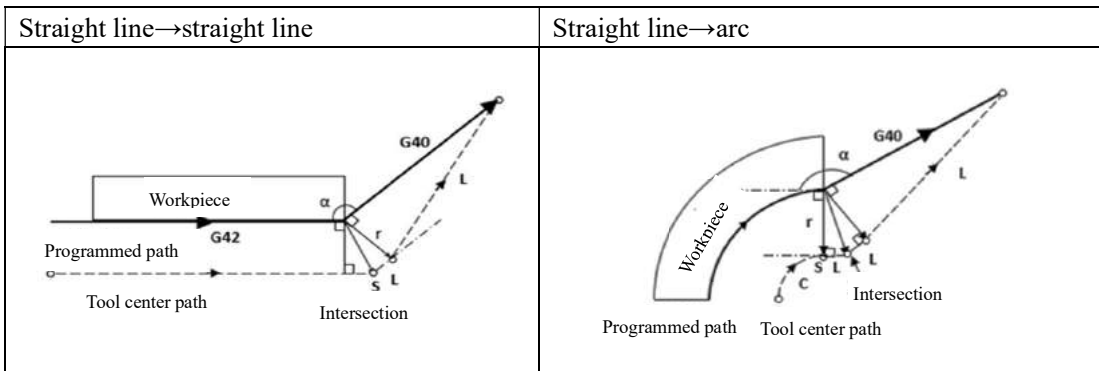
(3) In the block where there is movement cancellation, when the tool moves around the outside of the acute angle ($\alpha < 90°$)

Straight line→straight line	Straight line→arc
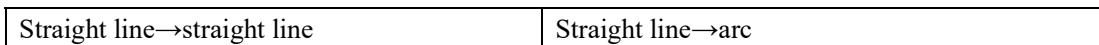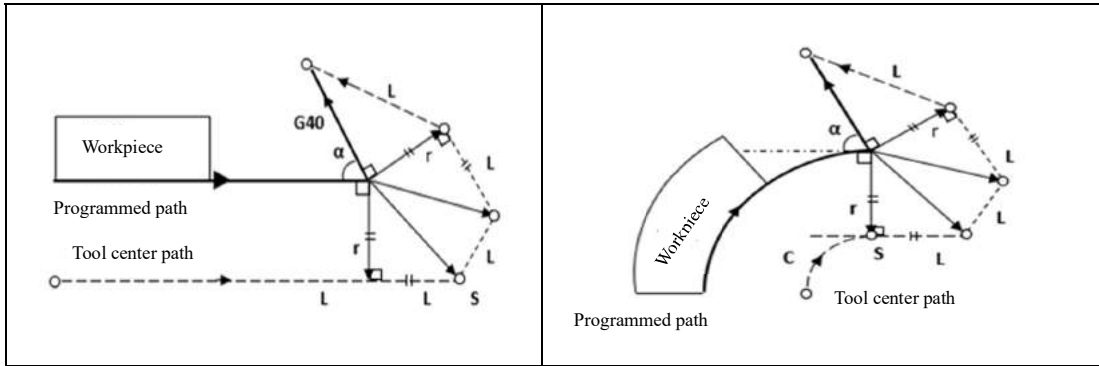
12.5.3 Compensation Direction Change During Tool Radius Compensation

Function and Purpose

The tool radius compensation direction is determined by the tool radius compensation command (G41/G42) and the sign of the compensation value.

Tool radius compensation command	Compensation value +	Compensation value -
G41	Compensation left	Compensation right
G42	Compensation right	Compensation left

The system does not support the use of G41 and G42 to change the compensation direction, and an alarm "The tool compensation direction is not allowed to be changed during radius compensation" will appear during use. It is not recommended to change the compensation direction by changing the compensation sign during the compensation process, because different programming compensation methods and use environments may result in different effects.

Example

Note: The following changes the compensation direction by changing the sign of the compensation amount during the compensation process. Examples of programming will show the following paths. (It is not recommended to use tool direction change).

The compensation direction is changed (straight line to straight line)

●There is a path intersection when changing the compensation direction

%1234
G90G54G0X-50Y50
G41G01X-50Y20D1 (compensation value is 5)
X-30Y0
X30
X50Y20D2 (compensation value is -5)
Y50

	M30

● There is no path intersection when changing the compensation direction

	%1234
	G90 G54 G0 X-60 Y-30
	G41 G01 X-50 Y-20 D1 (compensation value is 5)
	X-30 Y0
	X30 Y0
	X50 Y-20 D2 (compensation value is -5)
	Y-50
	M30

The compensation direction is changed in the way of (straight line to arc)

● There is a path intersection when changing the compensation direction

	%1234
	G90 G54 G0 X-50 Y30
	G41 G01 X-50 Y0 D1 (compensation value is 5)
	X-30 Y0
	G2 X30 Y0 I30 J0 D2 (compensation value is -5)
	G1 X50 Y0 D1 (compensation value is 5)
	G40
	M30

● There is no path intersection when changing the compensation direction

	%1234
	G90 G54 G0 X-50 Y30
	G41 G01 X-50 Y0 D1 D1 (compensation value is 5)
	X-30 Y0
	G3 X30 Y0 I30 J0 D2 (compensation value is -5)
	G1 X50 Y0 D1 (compensation value is 5)
	M30

The compensation direction is changed (arc to arc)

Note: For the direction change programming for arc to arc, the system does not support the following two programming methods.

● There is a path intersection when changing the compensation direction

G41 feed direction D01　　　D02　　　D02	%1234 G90 G54 G0 X-50 Y0 G41 G01 X0 Y0 D1 (compensation value is 5) G2X53.64Y18.47I30J0 X140.66Y0I41.55　　　　J-18.47D2 (compensation value is -5) G1X150 M30 This programming mode is not supported for the direction change)

● There is no path intersection when changing the compensation direction

G41 feed direction D01　　　　D01　　　　D02	%1234 G90 G54 G0 X-50 Y0 G41 G01 X0 Y0 D1 (compensation value is 5) G3X18.02Y-4.04I9.03J1.7 X37.26Y-2.18I9.26 J3.7 G2X53.12Y10.61I13.99J-1.11D2 (compensation value is -5) G1X50 M30 (This programming mode is not supported for the direction change)

Linear reciprocating motion changes the direction of tool compensation vector

D1 D2	%1234 G90G54G0X-30Y0 G41G1X0Y0D1 (compensation value is 5) X80 G01X0D2 (compensation value is -5) G40 G01 X-30 M30

12.5.4 When Tool Radius Compensation is not Executed

Function and Purpose

16

In radius compensation mode, when the commands (G28, G53) are executed, the compensation vector will be temporarily invalid. After this type of command is executed, it will automatically return to the compensation mode, and there is no need to cancel or add the compensation. The point where the compensation vector is cancelled is the command point of the program. When the compensation mode is restored, go directly to the command point and complete the restoration of the compensation vector.

📖 **Description**

(1) Reference point return command G28 in the radius compensation mode

At the intermediate point (there is no intermediate point when G91 is used, it is directly the reference point), the compensation vector becomes 0

Example：%1234

G90G54G0X-30Y0

N1G41G1X0Y0D1

N2X40Y15

N3G28X50Y20

N4X80 Y30

N5X120 Y20

M30
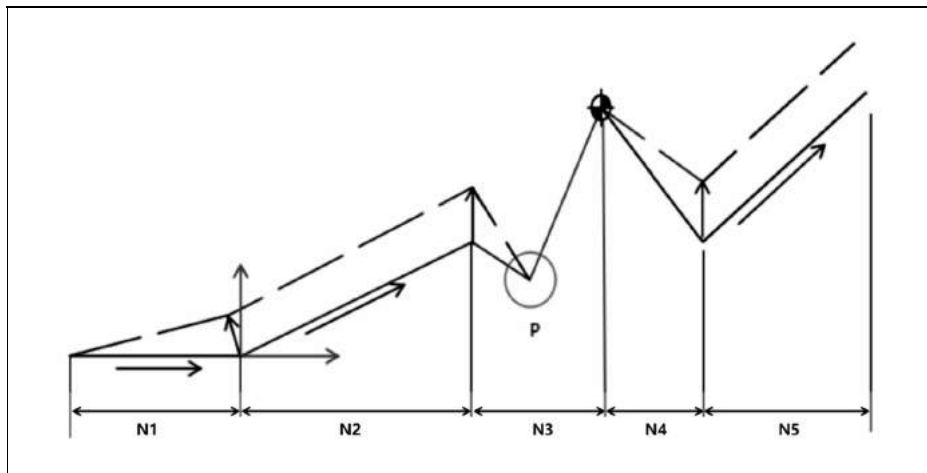
········ Tool center path after compensation

——— Tool path when compensation is not executed

Point P is the position of the intermediate point at the time of reference point return. If there is no intermediate point, it directly returns to the reference point.

(2) Direct machine coordinate system programming command G53 in radius compensation mode

In the G53 command (basic machine coordinate system selection), the compensation vector will be temporarily cancelled.

12.5.5 There is a Block without movement in Tool Radius Compensation

Function and Purpose

In the process of tool compensation creation and execution, the following 5 blocks will be read ahead to establish the tool compensation path, so when there are continuous blocks without movement in the program, the system will deal with it as follows.

The following blocks are regarded as non-moving blocks,

Non-moving block type	Example
M auxiliary code	M03/M04/M05/M06/M08
S speed command	S300
T tool number command	T6/Grouping T101
G04 dwell command	G04 X5/G04 P5000
Programmable data input block	G10 L10 P_
Coordinate system setting	G92 X0 Y0
The third axis movement command in the compensation plane	G17 Z10
G code	G90/G94/G95/G08/G09/G64/G65
The movement amount is 0	G91 X0

M00, M01, M02, and M30 are treated as M codes of read-ahead prohibition.

Description

When starting to establish compensation,

> At the beginning of the compensation establishment, there are more than 5 con secutive blocks without movement and the read-ahead prohibition M command, it does not affect the establishment of compensation.


```
%1234
G92X0Y0
G0 X30 Y60
G41 D1
G4X2
F1000
S500
M3
N1G1X20Y-50
X50Y-20
M30
```
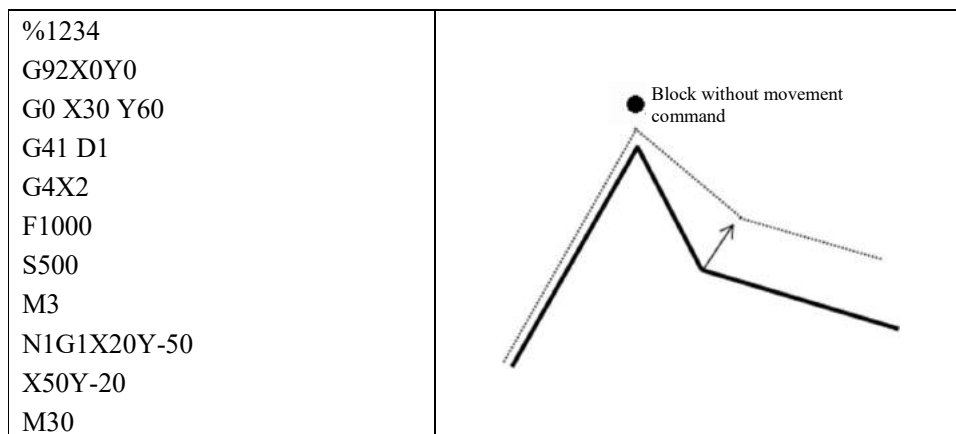
(Note 1) When G41 is used to establish tool compensation, there is no movement amount, and there is no movement in 5 consecutive blocks. The tool compensation path is executed when the block N1 is executed. At this time, the compensation path is not affected when the entire program is creating tool compensation.

In the compensation mode

When there are no over 4 continuous blocks without movement, and there is no M command of read-ahead prohibition in the compensation mode, the vector that the length equal to the compensation value will be generated in the vertical direction of the movement direction of the previous and next blocks. The non-movement block is executed at the vector stop point after compensation.


```
%0001
G90G54G0X0Y-30
G42G01X0Y0D1
N1X30Y50
N2G04P1000  ;  Program block
without movement
N3X80
……
```
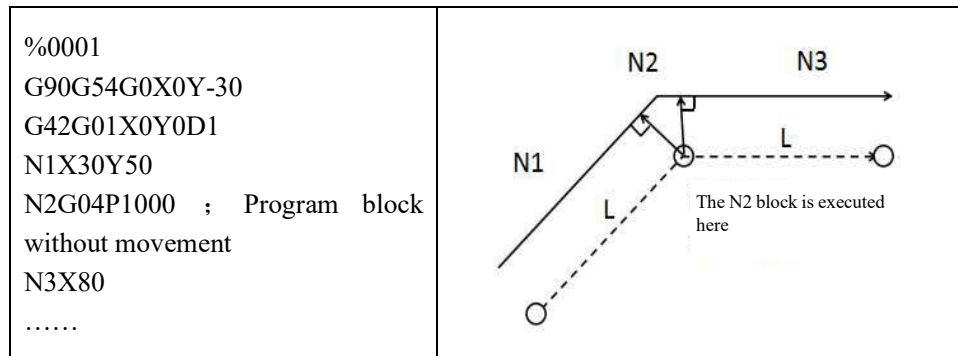
Several blocks without tool movement should not be commanded continuously.

If there are more than 4 consecutive non-movement blocks, and the M commands of read-ahead prohibition, a vector with a length equal to the compensation value will also be generated in the vertical direction of the movement direction of the previous and next blocks, and the non-movement block will stop at the vector point after the compensation.


```
%0001
G90G54G0X0Y-30
G42G01X0Y0D1
N1X30Y50
G04P1000
G91X0
M3S300
F2000
M00    ;  Non-movement block
N3X80
……
```
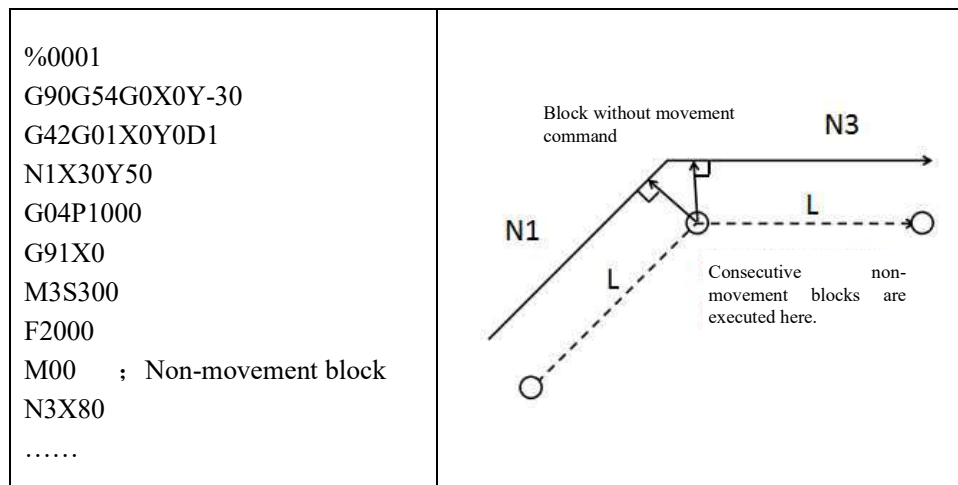
Note: The block without movement in the figure means: execute the program between N1 and N3.

The non-movement command is executed while the compensation is cancelled

When the non-movement block and G40 command are executed at the same time, only the compensation vector is cancelled.

%0001 G90G54G0X0Y-30 G42G01X0Y0D1 N1X30Y50 N2G40　；Non-movement block N3X80Y30	
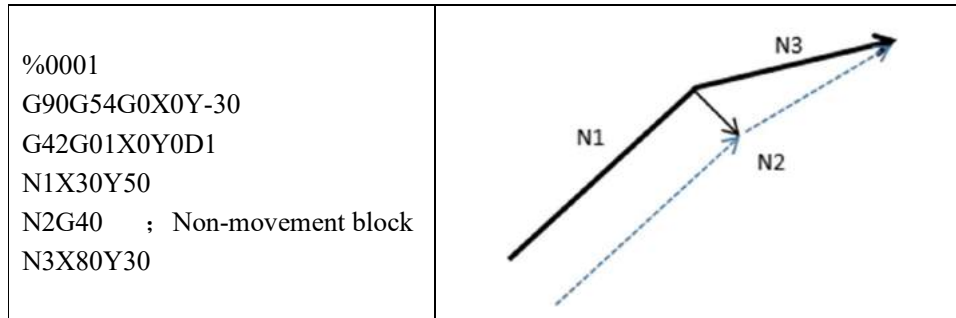
12.5.6 Action Insertion in Tool Radius Compensation

MDI insertion command

1) Non-movement command is inserted in MDI (tool path will not change)

%0001 G90G54G0X0Y-30 G42G01X0Y0D1 N1X30Y50 N2　　；Insert the execution of M3S1000 in MDI N3X55Y20 ……	
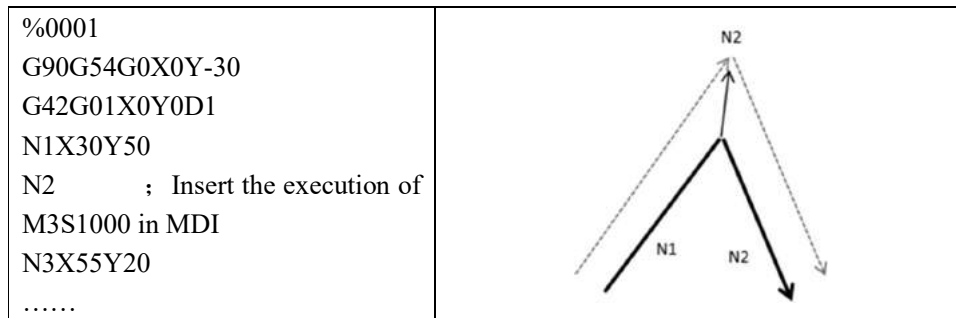
The MDI mode can be switched to only when the feed is held, so when running to the N1 block, the single-block running mode is switched to, and after entering the feed hold, the MDI mode is switched to for the insertion of a non-movement command.

2) Insert a movement command in MDI

The system runs the program of radius compensation in auto mode. If a movement command is input in MDI and the position of the machine tool is changed when the program is paused, the system will alarm that it is not at the breakpoint position when it returns to the program to run. It can continue to run after resuming the breakpoint position, and the compensation is added normally.

Manual insertion action

The system runs the program in radius compensation in auto mode. If each axis of the machine tool is moved in JOG mode, and the position of the machine tool is changed when the program is paused, the system will alarm that it is not at the breakpoint position when it returns to the program to run. It can continue to run after resuming the breakpoint position, and the compensation is added normally.

12.5.7 Change of Compensation Value in Tool Radius Compensation

✎	**Function and Purpose**

In principle, do not change the compensation number randomly in the radius compensation mode. Different compensation numbers may correspond to different compensation values, and the programmed trajectory will also change. If the compensation number is changed, the following actions are performed.

The program code format for changing compensation number is as follows,

G41G01……D(r1)

G0/G01/G02/G03　X_ Y_

G0/G01/G02/G03　X_ Y_

G0/G01/G02/G03　X_ Y_ D(r2) ； 　Compensation number is changed from r1 to r2

G0/G01/G02/G03　X_ Y_

……

📖	**Description**

1) Path of straight line→straight line

The programmed path is an acute angle,

G90G54G0X-30Y0 G41G1X0Y0D01 （Compensation value is 5） N1X20Y15 N2X40Y15 N3X60Y10D02 （ Compensation value is 10） N4X80 ……	
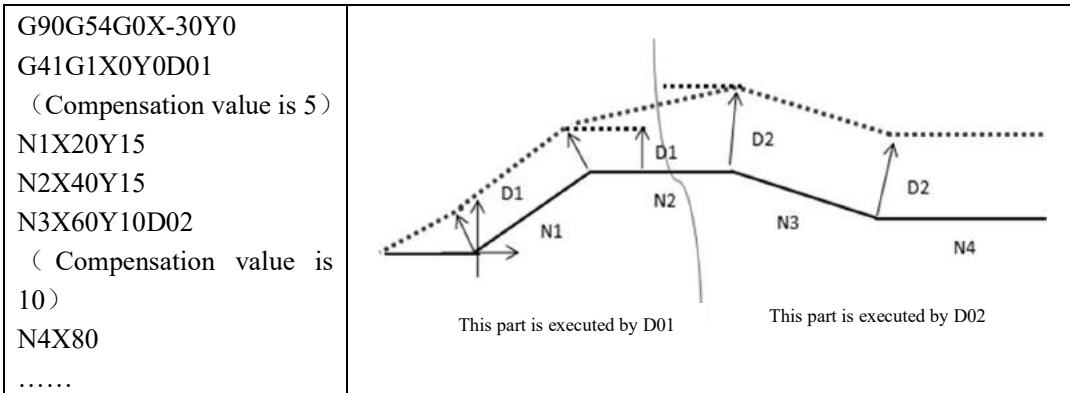
 Tool center path after compensation

Tool path before compensation

The programmed path is an obtuse,

| G90G54G0X-30Y0
G41G1X0Y0D01
（Compensation value is 5）
N1X50Y0
N2X40Y-20
N3X-20D02（Compensation value is 10）
…… | |
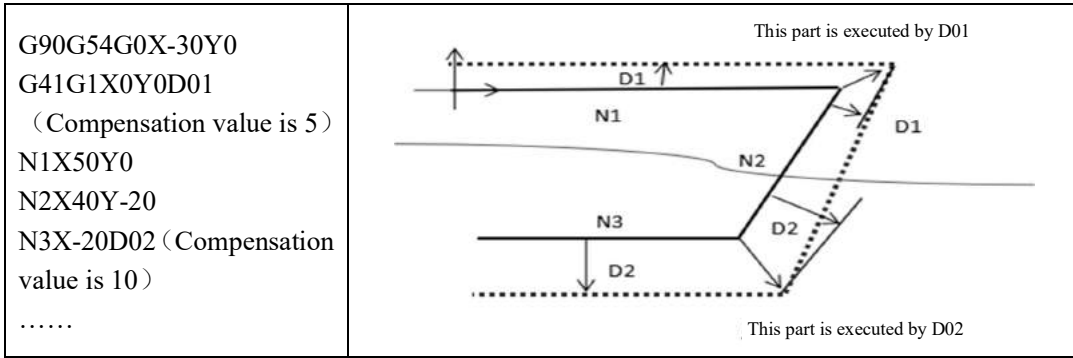
2) Path of straight line→arc

Change of compensation amount inside sharp corners

| G90G54G0X-30Y0
G41G1X0Y0D01
（Compensation value is 5）
N1X20Y0
N2G2X60Y0I20J0D02
N3X90Y0 （Compensation value is 10）
…… | |
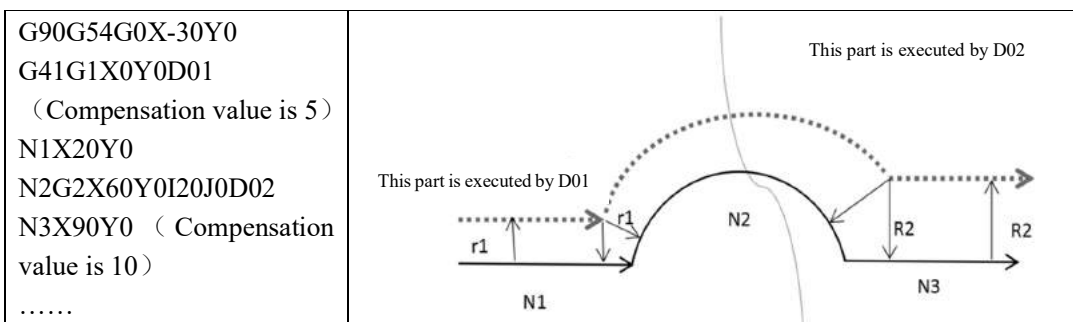
Change of compensation amount outside the sharp corner

| G90G54G0X-30Y0
G41G1X0Y0D01
（Compensation value is 5）
N1X20Y0
N2G3X40Y-20I20J0D2
N3X90 （Compensation value is 10）
…… | |
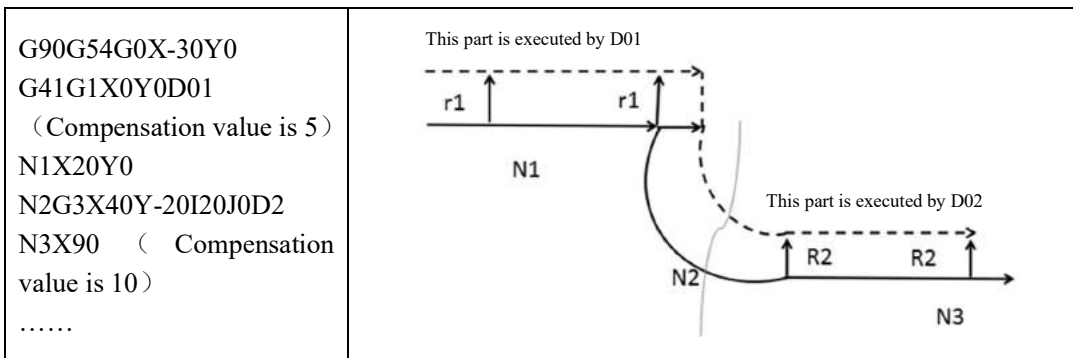
12.5.8 Interference Check

Function and Purpose

Usually, the overcutting of tool may occur after program read-ahead and tool radius compensation, which is referred to as interference. The interference check function can check the tool overcut in advance (even if there is no overcut, the interference check is performed). However, this function cannot find out all interference.

Description

There are two modes for interference check via parameter setting.

Function	Parameter	Action
Interference check alarm function	Interference alarm control Enable	Before the overcut interference occurs during the compensation, the system will alarm and stop running
Interference check avoidance function	Auto correct interference Enable	The path is changed to avoid overcutting

Note: When parameter 010046 is set to 0, the interference alarm function is turned on.

When parameter 010046 is set to 1, the auto interference correction function is turned on.

When parameter 010047 (number of radius compensation interference check blocks) is set to 4, 3 blocks is read ahead for check interference.

As the compensation interference condition

In the pre-reading 4 blocks, when there are movement commands in 3 blocks, and the compensation calculation vector on each movement command contact crosses the compensation path, it is regarded as the interference.
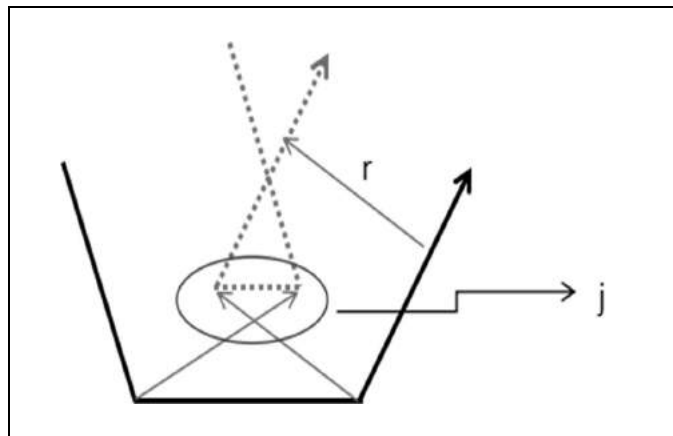
r: compensation amount   j: the position the cross vector occurs

Tool center path after compensation

Tool path before compensation

Two modes are set to run the program with interference, the specific situations are as follows:

Example 1: use a tool with a larger diameter to process a parts containing a sharp corner and a line segment
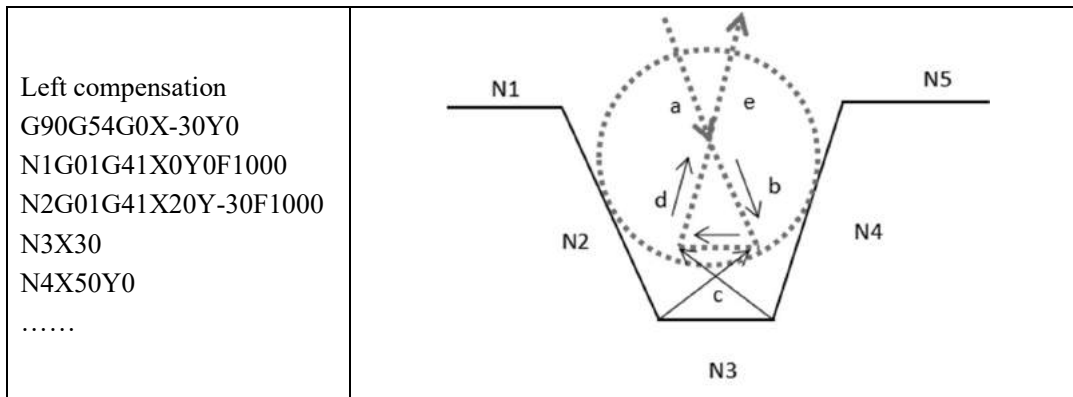
Left compensation G90G54G0X-30Y0 N1G01G41X0Y0F1000 N2G01G41X20Y-30F1000 N3X30 N4X50Y0 ……	

1) Interference alarm

When parameter 010046 is set to 0, the interference alarm function is turned on. The parameter 010047 is set to 4 by default; When executing the above program, if an alarm is issued before N1 block, system stops running.

2) Interference avoidance

The intersection calculation of N2 and N4 blocks is executed, the interference avoidance vector is created, the compensation path is automatically corrected, and the path is corrected to the a-e path.

Example 2: Use a tool with a larger diameter to process the parts containing an arc of a smaller radius
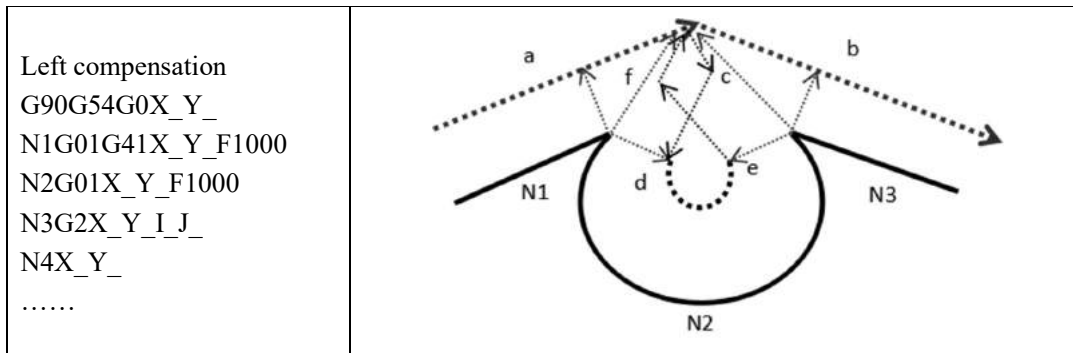
Left compensation G90G54G0X_Y_ N1G01G41X_Y_F1000 N2G01X_Y_F1000 N3G2X_Y_I_J_ N4X_Y_ ……	

1) Interference alarm

When parameter 010046 is set to 0, the interference alarm function is turned on. The parameter 010047 is set to 4 by default; When executing the above program, if an alarm is issued before N1 block, system stops running.

2) Interference avoidance

Through the interference check processing, due to the path crossing occurs at points c and f, the intersection calculation of N1 and N3 blocks is executed, the interference avoidance vector is created, the compensation path is automatically corrected, and the path is corrected to the a-b path.

13 Programmable Data Input（G10/G11）

13.1 Programmable Data Input Command（G10/G11）

Function and Purpose

Through G10/G11 command, user can dynamically modify system data in the program. G10 is a modal command. When G10 is specified, the programming data input mode can be entered, and the modified system data will take effect in time. When G11 command is called, this mode is cancelled.

The function list is as follows:

Function	G code
Input G54~G59 workpiece coordinate system origin	G10 L2 Pp IP_
Input G54.X extended workpiece coordinate system origin	G10 L20 Pp IP_
System parameter output	G10 L53 PpRr
Cancel user-defined input	G11
Milling tool geometry compensation value (length compensation) H input	G10 L10 PpRr
Milling tool geometry compensation value (radius compensation) D input	G10 L12 PpRr
Turning tool offset data input	G10 L14 Pp X_ Z_ R_ Q_ Y_ J_ K_
Single cutting time input	G10 L78 Pp

13.2 Workpiece Coordinate System Origin Input

Function and Purpose

According to the G10 command, the workpiece zero point (G54 to G59) offset can be set/changed from the beginning of the program. In the absolute (G90) mode, the specified compensation amount becomes the new compensation amount; in the incremental (G91) mode, the currently set compensation amount is added to the specified compensation amount to become the new compensation amount.

Command Format

G10 L2 Pp_ IP_

Parameter	Meaning
Pp	To specif the workpiece origin offset of the workpiece coordinate systems 1-6.

	➢ 1 corresponds to G54 workpiece coordinate system
	➢ 2 corresponds to G55 workpiece coordinate system
	➢ 3 corresponds to G56 workpiece coordinate system
	➢ 4 Corresponds to G57 workpiece coordinate system
	➢ 5 corresponds to G58 workpiece coordinate system
	➢ 6 corresponds to G59 workpiece coordinate system
IP	The workpiece origin offset of axis in absolute mode. It will be added to the original offset of the workpiece origin of axis in incremental mode.

Example

%1002

G54 ； G54 initial value

G01X0Y0Z0

G90G10L2P1X100Y100Z100 ； G54 workpiece coordinate system zero is changed to (100,100,100) in absolute mode.

G11

G01X20Y20Z20 ； The command value of the machine coordinate system is (120, 120, 120)

G91G10L2P1X50Y50Z50 ； G54 workpiece coordinate system zero is changed to (150,150,150) in incremental mode.

G11

G90G01X20Y20Z20 ； The command value of the machine coordinate system is (170, 170, 170)

M30

13.3 Extended Workpiece Coordinate System Origin Data Input

Function and Purpose

According to the G10 command, the offset of the extended workpiece coordinate system (G54.1 to G54.60) can be set/changed from the beginning of the program. In the absolute (G90) mode, the specified compensation amount becomes the new compensation amount; in the incremental (G91) mode, the currently set compensation amount is added to the specified compensation amount to become the new compensation amount.

Command Format

G10 L20 Pp_ IP_

Parameter	Meaning

Pp	The specified code p of the workpiece coordinate system of the workpiece origin offset value: 1 to 60, corresponding to the X value in the G54.X coordinate system;
IP	The workpiece origin offset of axis in absolute mode; It will be added to the original offset of the workpiece origin of axis in incremental mode.

Example

%1003
G54 ； G54 initial value
G01X0Y0Z0
G90G10L20P1X50Y50Z50 ； G54.1 workpiece coordinate system zero is changed to
(50,50,50) in absolute mode.
G11
G01X20Y20Z20 ； The command value of the machine coordinate system is (70, 70, 70)
G91G10L20P1X50Y50Z50 ； G54.1 workpiece coordinate system zero is changed to
(100,100,100) in incremental mode.
G11
G90G01X20Y20Z20 ； The command value of the machine coordinate system is (120, 120,
120)
M30

13.4 System Parameter Data Output

Function and Purpose

System parameters is output to the current channel variable specified by Rr, #0～#49

Command Format

G10 L53 Pp__Rr_

Parameter	Meaning
Pp	Parameter ID index number
Rr	Variable address（#0～#49）

Example

%1004
G54
G01X0Y0Z0
G10L53P010340R1 ； Read the value of machine user parameter 010340 and assign it to variable #1
G10L53P010341R2 ； Read the value of machine user parameter 010341 and assign it to variable #2

G11
M30

13.5 Milling Tool Length Compensation Data Input

Function and Purpose

With the G10 command, the milling tool length compensation can be set or changed in the program. In the absolute (G90) mode, the specified compensation amount becomes the new compensation amount; in the incremental (G91) mode, the currently set compensation amount is added to the specified compensation amount to become the new compensation amount.

Command Format

G10 L10 Pp_ Rr_

Parameter	Meaning
Pp	Tool offset number
Rr	Tool compensation data

Example

%0005
G54
G01X100Y100Z100
G90G10L10P1R2 ； No.1 tool length compensation is changed to 2 in absolute mode
G11
G43Z50H1 ； H1=2
G49Z0
G91G10L10P1R8 ； No.1 tool length compensation is changed to （8+2=10）in incremental mode
G11
G90G43Z50H1 ； H1=10
G49Z0

M30

 Note

If the tool length compensation is changed with G10, the tool length compensation needs to be called again with G43 or G44 to take effect.

13.6　Milling Tool Radius Compensation Data Input

 Function and Purpose

With the G10 command, the milling tool radius compensation can be set or changed in the program. In the absolute (G90) mode, the specified compensation amount becomes the new compensation amount; in the incremental (G91) mode, the currently set compensation amount is added to the specified compensation amount to become the new compensation amount.

 Command Format

G10 L12 Pp Rr

Parameter	Meaning
Pp	Tool offset number
Rr	Tool compensation data

 Example

%0006
G54
G01X100Y100Z100
G10L12P2R1 ；No.2 tool radius compensation is changed to 1 in absolute mode
G11
G41X20Y20D2 ；D2=1
X30
G40X0Y0
G91G10L12P2R1 ； No.2 tool radius compensation is changed to (1+1=2) in incremental
　　　　　　　　　mode
G11
G90G41X20Y20D2 ；D2=2
X30
G40X0Y0
M30

⚠ **Note**

If the tool radius compensation is changed with G10, the tool radius compensation needs to be called again with G41 or G42 to take effect.

13.7 Lathe Tool Offset Data Input

✎ **Function and Purpose**

With the G10 command, the lathe tool compensation, wear, tool nose offset can be set or changed. In the absolute (G90) mode, the specified compensation amount becomes the new compensation amount; in the incremental (G91) mode, the currently set compensation amount is added to the specified compensation amount to become the new compensation amount.

⚙ Command Format

G10 L14 Pp X_ Z_ R_ Q_ Y_ J_ K_

Parameter	Meaning
Pp	Tool offset number
X	Tool compensation data X
Z	Tool compensation data Z
R	Tool nose radius compensation value R
Q	Imaginary tool nose direction
Y	Tool compensation data Y
J	Tool radial wear J
K	Tool axial wear K

Example

Modify the tool offset data of No. 1 in the tool offset table (for example, X axis offset is -100, Z axis offset is -50, tool nose radius is 0.2, tool nose position number is 3) and call this tool offset:

```
%1007
G10L14P1X-100Z-50R0.2Q3
G11
T0101
……
M30
```

13.8   Get and Modify Single Cutting Time

Function and Purpose

The single cutting time on the main processing interface can be modified with G10L78.

Description

P parameter specifies the modified single processing time, unit: second

After modifying the "single cutting time", the "this cutting time" and "cumulative cutting time" are updated simultaneously.

Example

    %0008
    G54
    G01X100Y100Z100
    G10L78P60；   The single cutting time is changed to 60S
    G10L78P1000；   The single cutting time is changed to 1000S
    G11
    M30

Note

#1471: The current "single cutting" time can be read, but it is best to add G08 command to pause the read-ahead, otherwise the read time may not be accurate;

14 Standard Canned Cycle of Lathe（T）

14.1 Simple Cycle of Lathe

There are five simple cycles for lathe system,

G command	Function
G80	Innter (outer) diameter cutting cycle
G81	End face cutting cycle
G82	Thread cutting cycle
G74	End face deep-hole drilling cycle
G75	Outer diameter grooving cycle

For the cycle in this chapter, a G code block is used to complete the processing operation of multiple block commands, so that the program can be simplified

Note

1) The cycles described in this chapter can only be used for lathe system.

2) The command format cannot be used for FANUC system.

14.1.1 Inner (Outer) Diameter Cutting Cycle

Function and Purpose

With this function, the operation control of 4 linear trajectories can be realized through one block. The trajectory starts from the starting point A, and returns to the starting point A by A→B→C→D→A, and finally completes a simple cycle processing, in which the the first and the forth trajectories are the rapid traverse movements, and the second and the third trajectories are movements at processing speed. The trajectories are shown in the figure below.

This function is suitable for simple inner or outer diameter cutting cycle.

1. Inner/Outer diameter cutting cycle of cylindrical surface
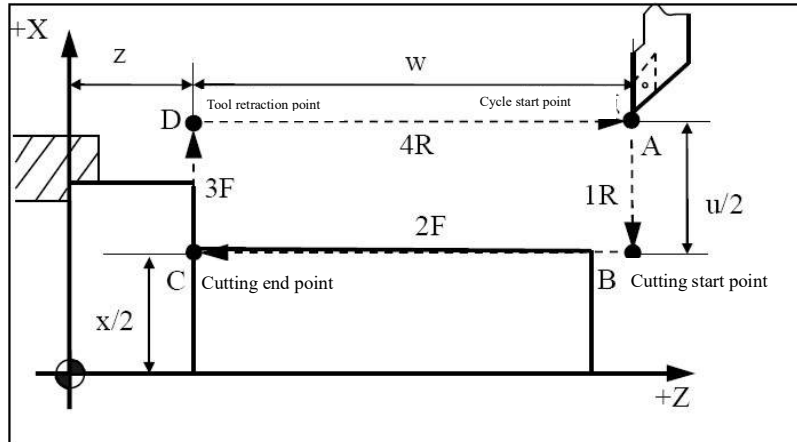
2. Inner/Outer diameter cutting cycle of conical surface
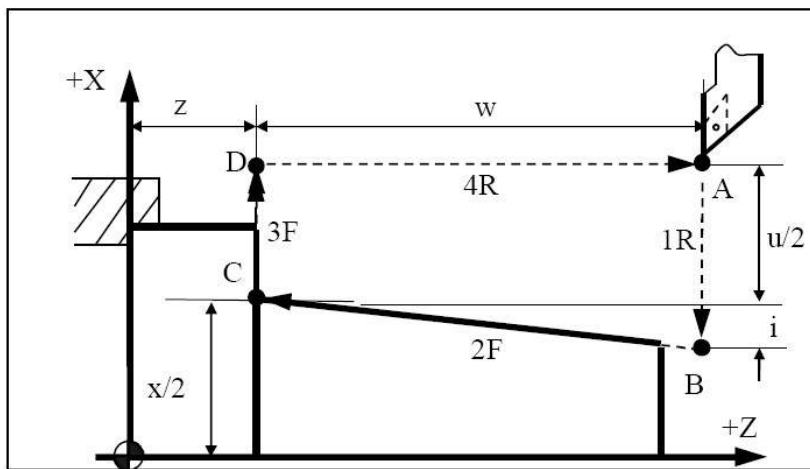
![F] **Command Format**

1. Inner/Outer diameter cutting cycle of cylindrical surface

G80 X_/U_ Z_/W_ F_

Parameter	Meaning
X/U Z/W	The coordinates of end point C in workpiece coordinate system in absolute mode; the directional distance from cutting end pint C to the cycle start point A in incremental mode, which is represented in U and W in the diagram, and the sign is determined by the directions of path 1 and path 2.
F	Feedrate (mm/min)

2. Inner/Outer diameter cutting cycle of conical surface

G80 X_/U_ Z_/W_ I_F_

Parameter	Meaning
X/U Z/W	The coordinates of end point C in workpiece coordinate system in absolute mode; the directional distance from cutting end pint C to the cycle start point A in incremental mode, which is represented in U and W in the diagram, and the sign is determined by the directions of path 1 and path 2.
I	The radius difference between the cutting start point B and the cutting end point C (the sign of I determines the reverse taper or the forward taper).
F	Feedrate (mm/min)

Example

Example 1: Machining the workpiece as shown in the figure below with G80 command to rough and finish simple conical parts.
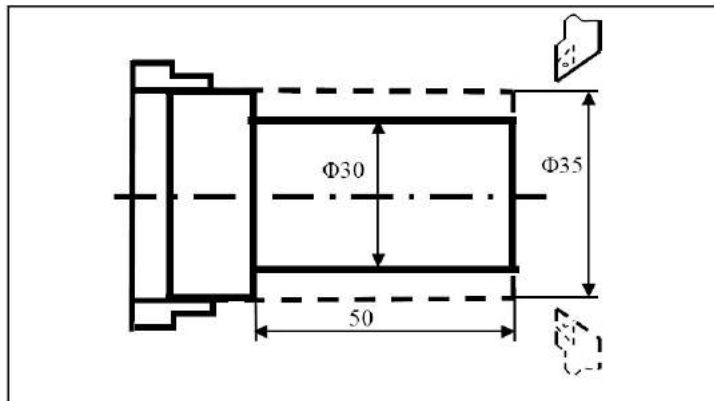
%3320

N1 T0101

N2 M03 S460

N3 G00 X90Z20

N4 X40 Z3

N5 G80 X31 Z-50 F100

N6 G80 X30 Z-50 F80

N7 G00X90 Z20

N8 M30

Example 2: Machining the workpiece as shown in the figure below with G80 command to process simple cone parts by roughing and finishing
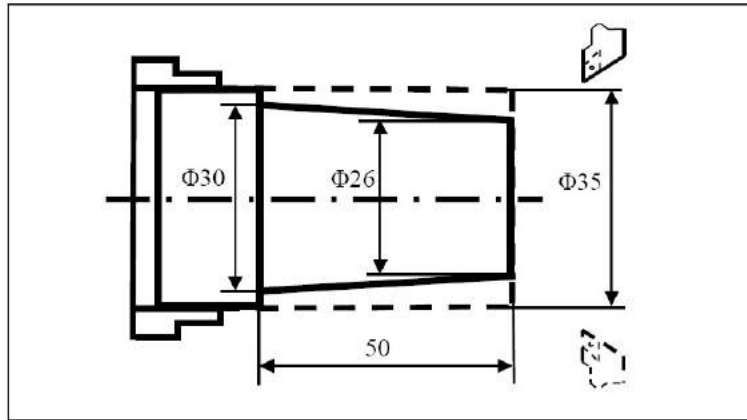
%3321

N1 T0101

N2 G00 X100Z40 M03 S460

N3 G00 X40 Z5

N4 G80 X31 Z-50 I-2.2 F100

N5 G00 X100 Z40

N6 T0202

N7 G00 X40 Z5

N8 G80 X30 Z-50 I-2.2 F80

N9 G00 X100 Z40

N10 M05

N11 M30

14.1.2 End Face Cutting Cycle (G81)

Function and Purpose

With this function, the operation control of 4 linear trajectories can be realized through one block. The trajectory starts from the starting point A, and returns to the starting point A by A→B→C→D→A, and finally completes a simple cycle processing, in which the the first and the forth trajectories are the rapid traverse movements, and the second and the third trajectories are movements at processing speed. The trajectories are shown in the figure below.

This cycle is suitable for flat face cutting and taper face cutting.

1. Flat face cutting

Cutting path A→B→C→D→A
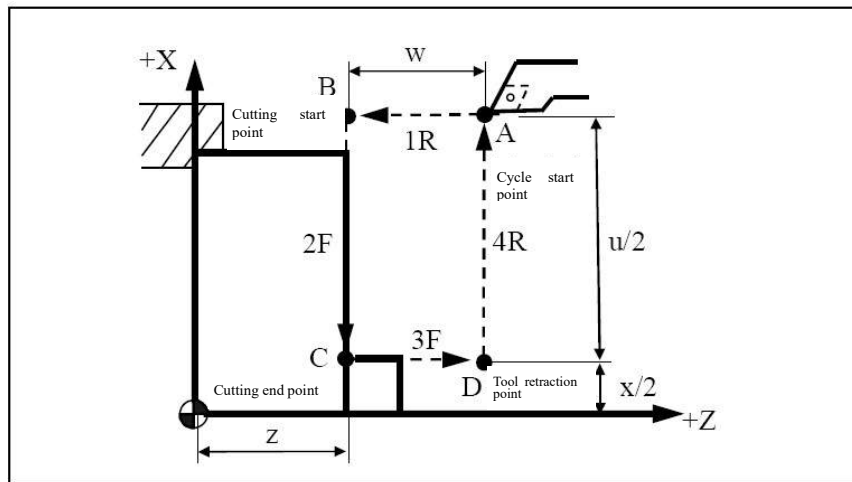
2. Taper face cutting

Cutting path A→B→C→D→A
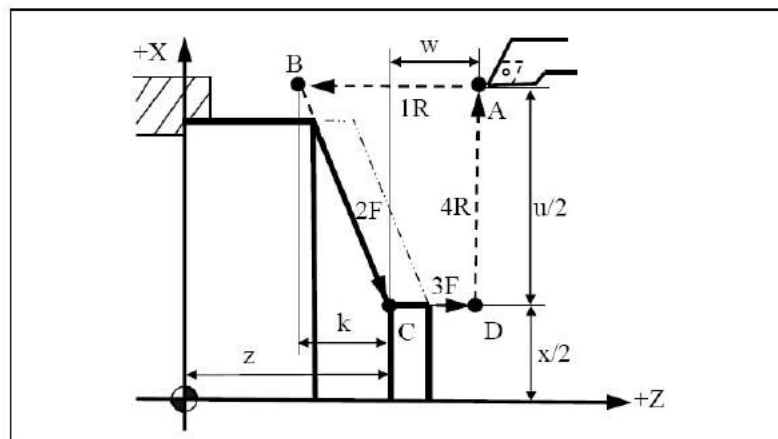
(F) **Command Format**

1. Flat face cutting

G81 X_/U_ Z_/W_ F_

Parameter	Meaning
X/U Z/W	The coordinates of end point C in workpiece coordinate system in absolute mode; the directional distance from cutting end pint C to the cycle start point A in incremental mode, which is represented in U and W, and the sign is determined by the directions of path 1 and path 2.
F	Feedrate (mm/min)

2. Taper face cutting

G81 X_/U_ Z_/W_ K_ F_

Parameter	Meaning
X/U Z/W	The coordinates of end point C in workpiece coordinate system in absolute mode; the directional distance from cutting end pint C to the cycle start point A in incremental mode, which is represented in U and W, and the sign is determined by the directions of path 1 and path 2.
K	The directional distance along Z from cutting start point B to cutting end point C.
F	Feedrate (mm/min)

Example

Process the workpiece as shown in the figure below with G81 command, and the dotted line represents the blank.
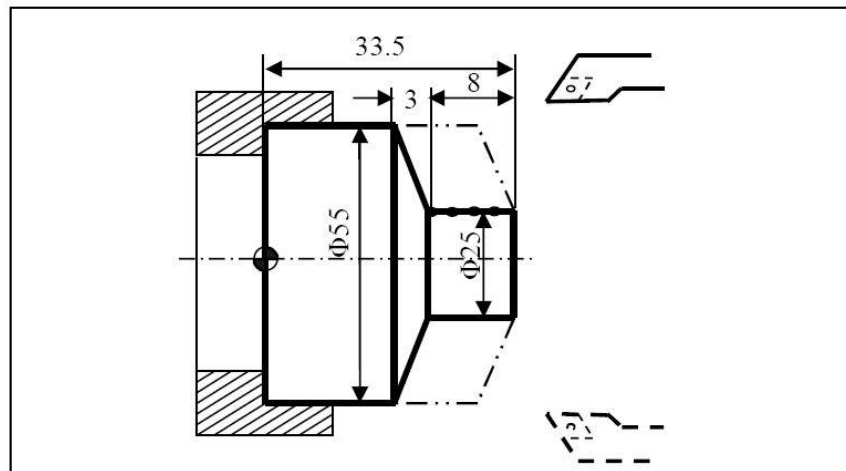
%3323

N1 T0101   ; Set coordinate system and select No.1 tool

N2 G00 X60 Z45   ; Move to the cycle start position

N3 M03 S460   ; Spindle rotation CW

N4 G81 X25 Z31.5 K-3.5 F100   ; The first cycle, the cutting depth is 2mm

N5 X25 Z29.5 K-3.5   ; The cutting depth is 2mm each time

N6 X25 Z27.5 K-3.5   ;   The starting point of each cutting is 3mm away from the outer surface of the workpiece, so the K value is -3.5

N7 X25 Z25.5 K-3.5   ; The forth cycle, the cutting depth is 2mm

N8 M05   ; Spindle stops

N9 M30   ; Main program end, and reset

Note

(1) If no F value is specified, it will cut at the default speed.

(2) The end face cutting requires special tools for end face cutting

(3) This canned cycle automatically recognizes diameter programming and incremental programming, no need to switch modes specifically

(4) Before running this program, the spindle must be rotated.

(5) This program can be executed with tool compensation.

14.1.3 Thread Cutting Cycle（G82）

Function and Purpose

1. Straight thread cutting cycle

The cutting path A→B→C→D→A
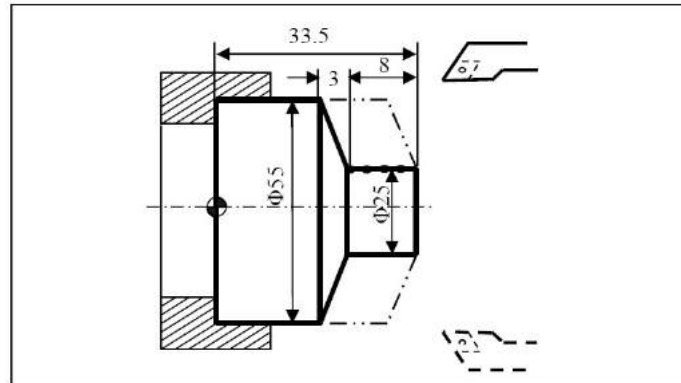
2. Taper thread cutting cycle

The cutting path A→B→C→D→A
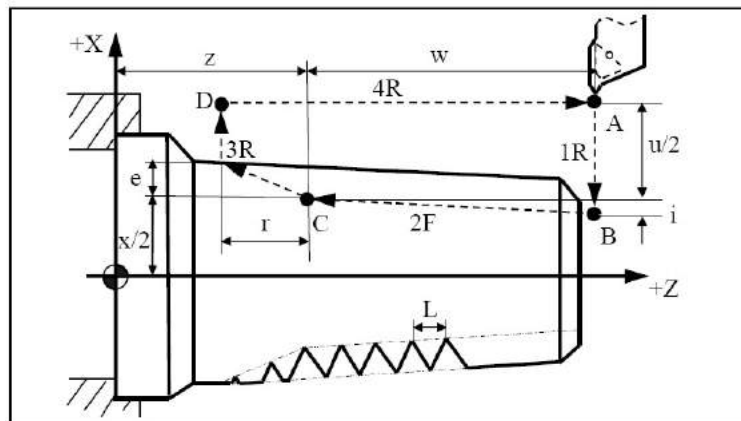
Command Format

G82 X_/U_ Z_/W_ I_ R_ E_ C_ P_ F_

Parameter	Meaning
X/U Z/W	The coordinates of end point C in workpiece coordinate system in absolute mode; the directional distance from cutting end pint C to the cycle start point A in incremental mode, which is represented in U and W in the diagram.
I	It is the radius difference between the thread start point B and the thread end point C. Its sign is the sign of the difference (no matter it is absolute programming or incremental programming). This parameter can be omitted for straight thread.

R E	The undercut amount for thread cutting. R and E are both vectors, R is the Z-direction retraction amount; E is the X-direction retraction amount. R and E can be omitted, indicating that the retraction function is not required
C	Number of thread starts. When it is 0 or 1, it indicates the single-start thread.
P	For single-start thread cutting, it is the spindle rotation angle of the spindle reference pulse from the cutting start point (default value is 0); for multi-start thread cutting, it is the corresponding spindle rotation angle between the cutting start points of adjacent thread starts.
F	Thread lead; (mm/r)

Example

As shown in the figure below, the workpiece is programmed with G82, and the blank shape has been processed.
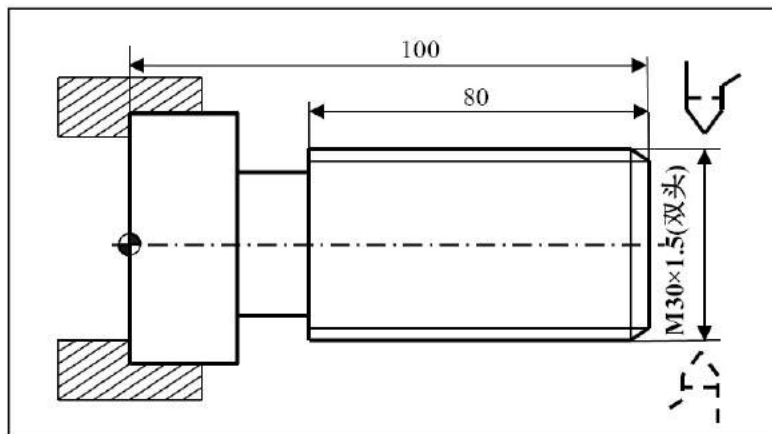
%3324

N1 G54 G00 X35 Z104   ;   Select coordinate system G54, move to cycle start point

N2 M03 S300   ;   Spindle rotates at 300r/min

 N3 G82 X29.2 Z18.5 C2 P180 F3   ;   The first cycle for thread cutting, the cutting depth is 0.8mm

N4 X28.6 Z18.5 C2 P180 F3   ;   The second cycle for thread cutting, the cutting depth is 0.6mm

N5 X28.2 Z18.5 C2 P180 F3   ;   The third cycle for thread cutting, the cutting depth is 0.4mm

N6 X28.04 Z18.5 C2 P180 F3   ; The forth cycle for thread cutting, the cutting depth is 0.16mm

N7 M30   ; Spindle stops, main program end, and reset

![Note icon] **Note**

(1) If the retraction function is required, please note that the signs of the R and E values must be coordinated with the thread cutting direction. Retraction in the opposite direction of thread processing may damage the thread. At the same time, only R can be specified without specifying E, but if E is specified, R must be specified. When R and E are zero, the undercut amount is forced to F×0.68;

(2) The thread cutting cycle is the same as the G32 thread cutting. In the feed hold state, the cycle will stop after completing all actions;

(3) The override cannot be changed at the time of thread cutting;

(4) F value can be inherited, if there is no value to inherit, it will alarm;

(5) The 90-degree undercut is #54019 in the user-defined macro variable.

14.1.4 End Face Deep-hole Drilling Cycle（G74）

![Function and Purpose icon] **Function and Purpose**

This cycle can perform deep hole drilling on the end face. As shown in the figure below.

The tool path A→B→C→D→A
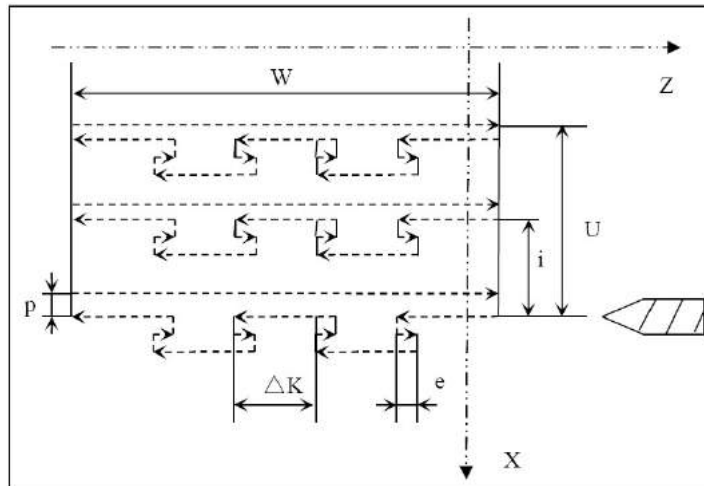
![Command Format icon] **Command Format**

G74 X_/U_Z_/W_ Q($\triangle$K)_R(e)_ I(i)_P(p)_

Parameter	Meaning
X/U	The coordinates of end point of hole bottom on X in workpiece coordinate system in absolute programming; in incremental programming, the directional distance from end point of hole bottom to the cycle start point, which is

	represented by U in the graph. This value can be left blank.
Z/W	The coordinates of end point of hole bottom on Z in workpiece coordinate system in absolute programming; in incremental programming, the directional distance from end point of hole bottom to the cycle start point, which is represented by W in the graph.
R	Undercut amount on Z, can only be positive, and can be left blank.
Q	Feed depth, can only be positive.
I	The width of each cut when drilling a wide hole, can only be positive, and can be left blank.
P	Retract amount on X. When I is specified, P can only be positive; when I is not specified, P can be both positive and negative, and can be left blank.

Description

Blank length: 406.1. radius 120.0. scaling 1.748
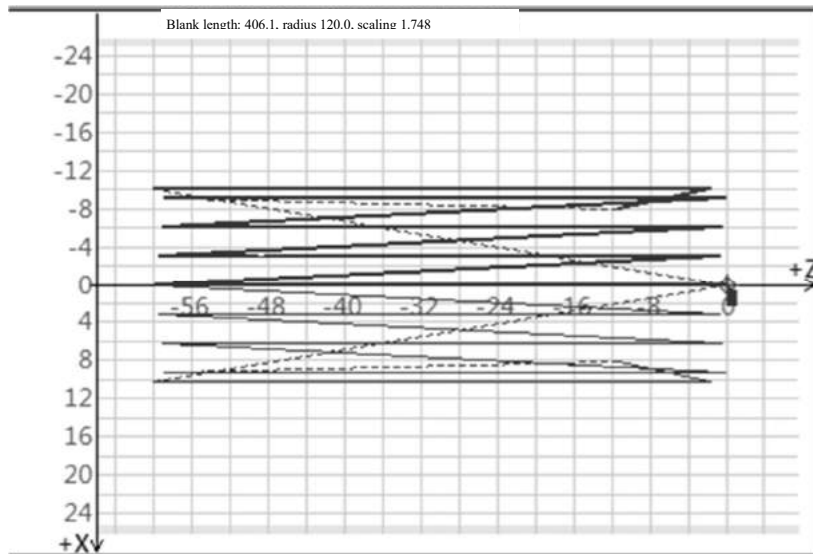
As shown in the figure above, it is the drilling trajectory of G74. Each time after drilling into the bottom of the hole, it moves Q distance in the X direction and then drills again until the remaining width in the X direction is less than Q, then completes the X axis movement, and drills finally so as to achieve the drilling of a fixed width hole.

Example
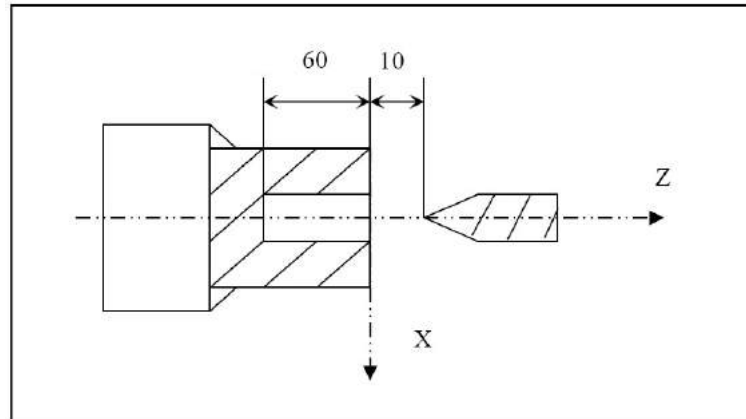
%1234

T0101

M03S500

G01 X0 Z10F200

G74 X-10Z-60R1Q5I3P1

M30

⚠ Note

(1) Modal inheritance is not supported. If the following lines of G74 are XZ coordinates, the program will not continue drilling, but o run the coordinates with G01.

(2) If need to drill continuously, user needs to write multiple lines of G74 continuously, and commands such as RQIP will not be inherited.

(3) Clockwise drilling and counterclockwise drilling are supported.

(4) Pay attention to selecting appropriate speed values (including dry run). Even for the virtual pattern in the dry run, the system will display the actual tool path when the speed is too fast. When the speed is adjusted to a high level in dry run, the drilling pattern will be distorted.

(5) Pay attention to the width of the tool at the time of wide groove cutting.

14.1.5 Outer Diameter Grooving Cycle (G75)

✎ Function and Purpose

This cycle is used for grooving the outer diameter of the workpiece.
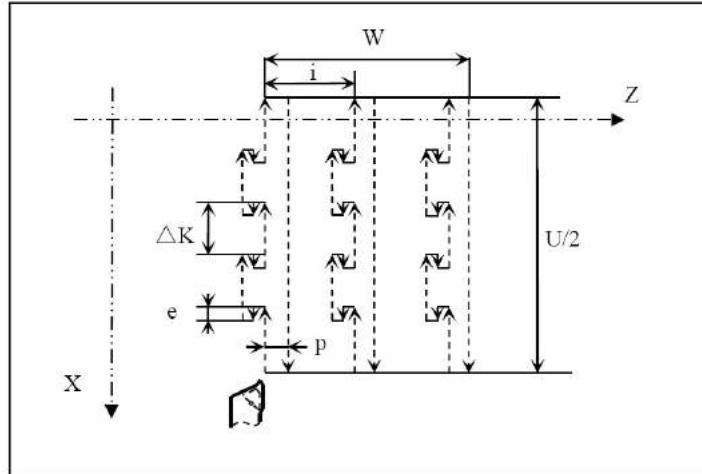
Command Format

G75X_/U_Z_/W_ Q(△K)_R(e)_ I(i)_P(p)_

Parameter	Meaning
X/U	The coordinates of end point of hole bottom on X in workpiece coordinate system in absolute programming; in incremental programming, the directional distance from end point of hole bottom to the cycle start point, which is represented by U in the graph.
Z/W	The coordinates of end point of hole bottom on Z in workpiece coordinate system in absolute programming; in incremental programming, the directional distance from end point of hole bottom to the cycle start point, which is represented by W in the graph.
R	Retract amount on X, can only be positive, and can be left blank.
Q	Feed depth, can only be positive.
I	Groove width, can only be positive, and can be left blank.
P	Undercut amount on Z. When I is specified, P can only be positive; when I is not specified, P can be both positive and negative, and can be left blank.

Example

Example 1: G75 outer diameter grooving cycle programming example.
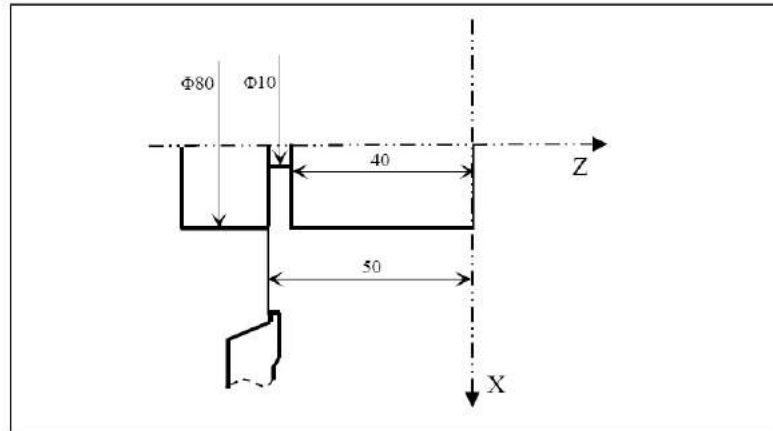
%1234

T0101

M03S500

G01 X50 Z-50F200

G75 X10Z-40R1Q5I3P2

M30

⚠️ **Note**

---

(1) If K is not defined, user macro #54005 needs to be set as the tool retraction amount.

(2) If Q is not defined, the Q in the previous line will be inherited. If Q is not defined in the first line, then Q will be 0.

(3) If R is not defined, then R is 0 by default.

(4) Modal inheritance is not supported. If the following lines of G75 are XZ coordinates, the program will not continue drilling, but to run the coordinates with G01.

(5) If need to drill continuously, user needs to write multiple lines of G75 continuously, and commands such as RQIP will not be inherited.

14.2 Drilling Canned Cycle for Lathe System

There are four drilling cycles for lathes system:

G command	Function
G83	Axial drilling cycle
G87	Radial drilling cycle
G84	Axial rigid tapping cycle
G88	Radial rigid tapping cycle

With this cycle, a block containing G code is used to complete the machining operations with multiple block commands, so that the program can be simplified.

Note

(1) Cycles described in this chapter only can be used for lathe system.

(2) Commands G83, G87, G84, and G88 do not have positioning function. It is necessary to use G01 and G00 outside the canned cycle for positioning.

14.2.1 Axial Drilling Cycle（G83）/Radial Drilling Cycle（G87）

Function and Purpose

Turning centers with powered tool often require axial or radial hole processing. The G83 command of this system can be used to drill axial holes; G87 can be used to drill radial holes. For the two processing methods, there only is the difference in the feed axis, and the drilling trajectory is exactly the same. The axial drilling is fed along Z axis and the radial drilling is fed along X axis.

In addition, axial drilling is also suitable for conventional lathes. At this time, the workpiece or drill can be installed on the lathe spindle to form the main cutting movement, and the drill or workpiece can be installed on the tool holder to perform the Z-axis feed movement, thereby realizing the axial drilling.

Command Format

1. Axial drilling cycle

G83Z(W)_R_Q_K_P_F_H_

2. Radial drilling cycle

G87X(U)_R_Q_K_P_F_H_

Parameter	Meaning

X/Z	Coordinates of hole bottom
R	Distance from initial level to R level
Q	Cutting depth
P	Dwell time at the hole bottom
F	Feedrate
K	Retraction distance
H1	High speed deep hole drilling. Retraction of the specified distance K
H2	Deep hole drilling. Retraction to R point
H3	Spot drilling. Drill directly to the hole bottom

Description

According to the different technological requirements of drilling process, this function sets three kinds of drilling cycle trajectories, namely conventional hole drilling, high speed deep hole drilling, and deep hole drilling. The three modes are set by H1, H2, H3 respectively.

H1 is set as high-speed deep hole drilling, chip breaking needs to be taken into consideration, and the chip removal effect is not very good;

H2 is set as deep hole drilling, chip breaking and chip removal need to be taken into consideration;

H3 is set as conventional hole drilling, which is mainly suitable for shallow hole processing, without considering chip removal and chip breaking.

- **High-speed deep hole drilling --- H1 mode**

| G83 or G87 (G98) | G83 or G87 (G99) |
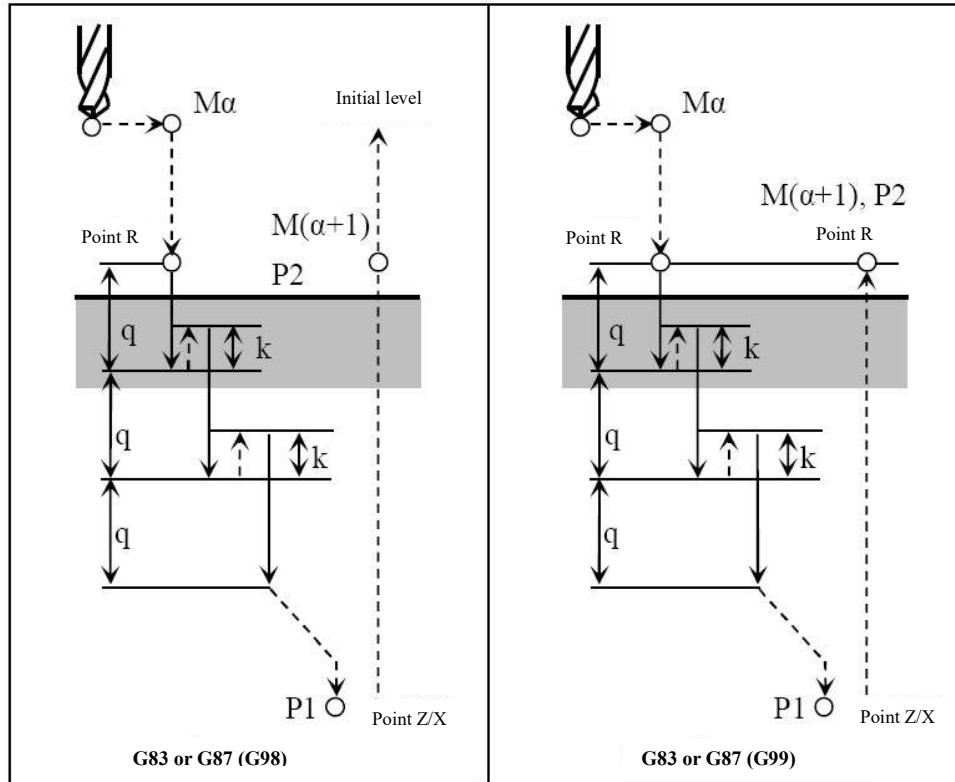
➢ Deep hole processing can be realized through repeated advance and retreat;

➢ The retraction amount K is small after drilling each time. Chip breaking can be achieve, and chip removal effect is not very good.

● **Deep hole drilling – H2 mode**

| G83 or G87 (G98) | G83 or G87 (G99) |
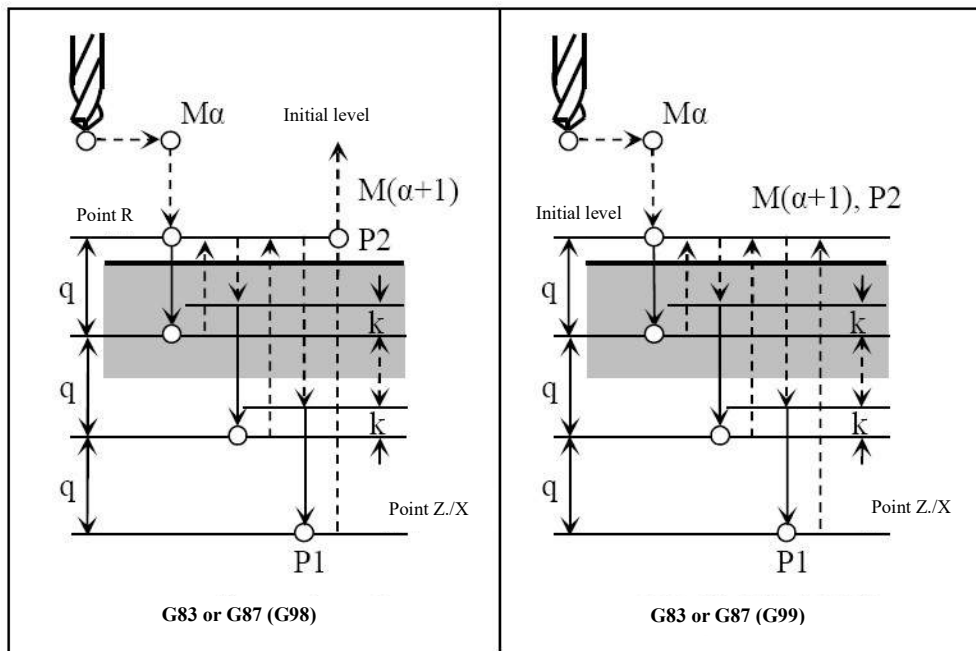
➢ Deep hole processing can be realized through repeated advance and retreat;

➢ After each drilling, the tool exits the workpiece to the safety level R point, which has a good chip removal effect, and the chip breaking can be achieved.

● **Spot drilling – H3 mode**

G83 or G87 (G98)   G83 or G87 (G99)
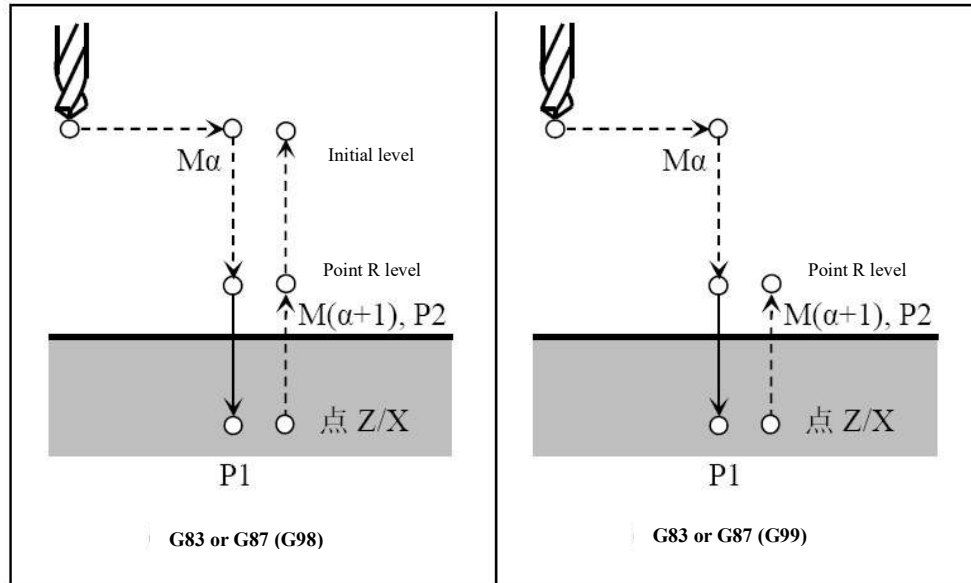
➢ Drill directly to the bottom of the hole each time, and then quickly exit;

➢ Designed for shallow hole drilling, no chip breaking and chip removal.

Example

To process the workpiece as shown in the diagram, the code format is as follows:

> %1111
>
> G54X0Z50
>
> G98G83Z-10R10Q5K2P1000F200H1
>
> G99G83Z-10R10Q5K2P1000F200H1
>
> G0X0Z50
>
> G98G83Z-10R10Q5K2P1000F200H2
>
> G99G83Z-10R10Q5K2P1000F200H2
>
> G0X0Z50
>
> G98G83Z-10R10Q5K2P1000F200H3
>
> G99G83Z-10R10Q5K2P1000F200H3
>
> M30
>
>
> %1111
>
> G54Z0X50
>
> G98G87X-10R10Q5K2P1000F200H1
>
> G99G87X-10R10Q5K2P1000F200H1
>
> G0Z0X50
>
> G98G87X-10R10Q5K2P1000F200H2
>
> G99G87X-10R10Q5K2P1000F200H2
>
> G0Z0X50
>
> G98G87X-10R10Q5K2P1000F200H3
>
> G99G87X-10R10Q5K2P1000F200H3
>
> M30

Note

(1) When H=1, the tool retracts the specified retraction distance K. It is stipulated in the canned cycle that when drilling with H1 and H2 modes, the amount of feed Q and the amount of retract k must be specified.

(2) Even if P, Q, K are not specified, if P, Q, K, are specified with previous G83, the variable will still be inherited. If a G83 line is normally specified, for the following G83 only the coordinates need to be specified. However, G01 , G00 will terminate this G83 modal, and for G83 after G01, G00, G02, G03 commands all parameter must be specified.

(3) The feed amount and the retraction amount must be larger than 0.

(4) If PWM spindle is used, C command cannot be used.

14.2.2 Axial Rigid Tapping Cycle (G84)/ Radial Rigid Tapping Cycle (G88)

Function and Purpose

Turning centers with powered tool often require axial or radial thread processing. The G84 command of this system can be used for tapping of axial thread; G87 can be used for tapping of radial thread. For the two processing methods, there only is the difference in the feed axis, and the drilling trajectory is exactly the same. The axial tapping is fed along Z axis and the radial tapping is fed along X axis.

In addition, axial tapping is also suitable for conventional lathes. At this time, the workpiece or tap can be installed on the lathe spindle to form the main cutting movement, and the tap or workpiece can be installed on the tool post to perform the Z-axis feed movement to realize the axial tapping.

Command Format

1. Axial rigid tapping cycle

G84 Z(W)_R_P_Q_E_J_K_F_H_

Parameter	Meaning
Z	Coordinates of hole bottom
R	Distance from initial level to R level
P	Dwell time at hole bottom
F	Feedrate
Q	Feed amount
K	Retraction amount
E1	Forward tapping
E2	Reverse tapping
J1	Tapping of the first spindle C axis
J2	Tapping of the second spindle A axis
H1	Retract the specified retraction amount K
H2	Retract to R point
H3	Directly move to hole bottom

2. Radial rigid tapping cycle

G88X(U)_R_E_Q_K_H_P_F_ (Only tapping of the second spindle A axis)

Parameter	Meaning
E1	Forward tapping
E2	Reverse tapping
Q	Feed amount
K	Retraction amount
H1	Retract the specified retraction amount K
H2	Retract to R point
H3	Directly move to hole bottom

Description

When executing this cycle, along the axis of the tool, the spindle rotates clockwise and taps to the bottom of the hole, and then the spindle counterclockwise rotates and retracts. According to the G98 or G99 command, the tap returns to the initial plane (G98) or the R level (G99). The specific trajectory is shown in the figure below.

| G84 or G88 (G98) | G84 or G88 (G99) |
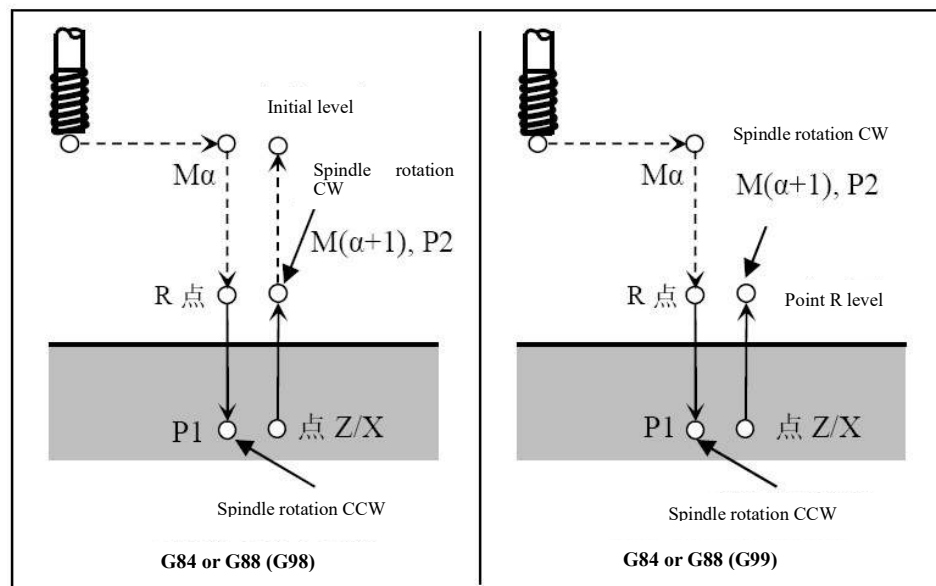
Example

%1111

M3 S1=1000; No. 1 spindle rotates

G0X50Z50

M5

G84Z-10R20P1000F1000H1

M33 S2=1000; No. 2 spindle rotates

G4P1000

M55

G84Z-10R20P1000F1H2

G88X-10R20P1000F1

M30

⚠ Note

(1) If the processing mode H is not specified in the first line, the H3 mode is defaulted, and the tool dills directly to the hole bottom.

(2) If the tapping spindle is not defined, C axis tapping is the default.

(3) During G84 and G88 rigid tapping, the feedhold and feedrate override modification are not allowed.

(4) For G84 rigid tapping, when P parameter 37 is set to 0, servo spindle tapping is selected; when P parameter 37 is set to 1, PMW spindle tapping is selected.

(5) Spindle rotates first during G84 tapping.

(6) Movement on Y axis is not supported for G88 radial drilling

(7) PMW spindle is not supported for G88.

14.3　FANUC Mode of Lathe System

✎ Function and Purpose

HCNC lathes system is compatible with the FANUC standard program format, which realizes the universality of the program and facilitates the user's program management.

When the parameter "010164 FANUC command support" is set to 0X2, the programming format of the lathe system is switched to the FANUC program format. At this time, the HCNC programming format can no longer be used.

Due to the essential difference between the two systems, the HCNC system is mainly compatible with FANUC's standardized programming format. For the non-standard formats not specified in the FANUC manual, the HCNC system cannot achieve complete compatibility.

This section focuses on the attentions for using FANUC mode in HCNC system.

📖 **Description**

1. **Note for FANUC mode canned cycle:**

   ➢ When G83 and G84 are used in combination, the R value in G84 cannot be omitted. The modal of the R value across the canned cycle are not inherited;

   ➢ P and Q in G74 and G75 can support decimals (unit is um), but FANUC only supports integers, and will alarm when it is decimal;

   ➢ P and Q in G76 can support decimals (unit is um), while FANUC only supports integers, and will alarm when it is decimal;

   ➢ G74 and G75 do not support each execution of a feed P as a single block, but the completion of entire execution as a single block

   ➢ When the G92 is executed in single block, the feed will be held at the starting point of the cycle after one cutting is finished. It does not support step-by-step operation during threading. In addition, the feed hold button is invalid during the thread cutting process, and it does not support FANUC's return to the cycle start point after pressing the feed hold;

   ➢ G76 in HCNC format is not allowed in FANUC mode;

   ➢ For all canned cycles, if only the canned cycle G code is specified without any commands after it, the canned cycle will be skipped directly.

2. **Note for multiple repetitive cycle in FANUC mode:**

   ➢ G71 retraction mode: retreat by one feed amount from the highest point of all points in the canned cycle;

   ➢ G71 and G72 are only allowed to establish and cancel tool compensation in ns and nf blocks. While tool compensation is established before entering G71 and G72 in FANUC;

   ➢ In the rough turning cycle G71, the movement cycle action to the turning starting point is G00, while G00 and G01 modes can be selected in FANUC.

   ➢ The retraction from the bottom of G71 with groove: after the last cutting at the bottom is completed, the tool moves along the contour, and then performs the next groove cutting.

   ➢ For the G72 with groove, even if the finishing amount on X is specified in the program, the finishing amount on X is not reserved.

   ➢ For the G71 with groove, even if the finishing amount on Z is specified in the program, the finishing amount on Z is not reserved.

3. **Note for parameter setting in FANUC mode**

   ➢ NC parameter 000023 F feedrate display mode can only be set to 0 and 1. 0 is set as the actual speed, and 1 is set as the specified speed. When 1 is set, the F value always displays as the specified value during G84 tapping.

   ➢ For the machine user parameter 010164, setting of 2 indicates FANUC mode, and setting of 1 indicates HCNC mode.

   ➢ The channel parameter 040179 maximum angle threshold of collinearity judgment is set to the default value 0.017.

   ➢ The default value of the machine user parameter 010160 F speed display is set to 0.

   ➢ The machine user parameter 010161 is the special parameter in the FANUC mode, and it is used as the non-monotonic alarm tolerance in Z direction of the multiple repetitive cycle.

   ➢ The tool offset must be specified with the T command before G50.

15 Canned Cycle for Milling System (M)

15.1 Standard Canned Cycle for Milling System

Function and Purpose

There is a relatively fixed operating sequence for the processing of hole parts. With this function, the canned cycle subprogram can be called through a single block to complete a fixed processing action. Therefore, programming can be simplified, and memory can be effectively used. Table 15.1(a) is the canned cycle command list of the CNC system.

Table 15.1 (a) Drilling canned cycle list

G command	Drilling (-Z direction)	Action at hole bottom	Retraction (+Z direction)	Use
G73	Intermittent cutting feed	Dwell	Rapid traverse retract	High-speed deep hole drilling cycle
G74	Cutting feed	Dwell—spindle rotation CW	Cutting retract	Reverse tapping
G76	Cutting feed	Spindle orientation	Rapid traverse retract	Fine boring
G80	——	——	——	Cancel cycle
G81	Cutting feed	——	Rapid traverse retract	Spot drilling, boring
G82	Cutting feed	Dwell	Rapid traverse retract	Drilling, boring
G83	Cutting feed	Dwell	Rapid traverse retract	Deep hole drilling cycle
G84	Cutting feed	Dwell—spindle rotation CCW	Cutting retract	Tapping cycle
G85	Cutting feed	——	Cutting retract	Boring
G86	Cutting feed	Dwell—spindle stop	Rapid traverse retract	Boring
G87	Cutting feed	Spindle rotation CW	Rapid traverse retract	Reverse boring
G88	Cutting feed	Dwell—spindle stop	JOG	Boring
G89	Cutting feed	Dwell	Cutting retract	Boring

Description

Generally there are the following six sequential actions for the hole machining cycle:

Sequential action 1: X and Y axis positioning (different planes may have different positioning axes)

Sequential action 2:rapid traverse to machining plane R point

Sequential action 3: execute drilling action

Sequential action 4: action at hole bottom (hole buttom level Z point)

Sequential action 5: retract to R level

Sequential action 6: retract to the initial level B point in rapid traverse

Detailed breakdown is shown in Figure 1.1 (a)
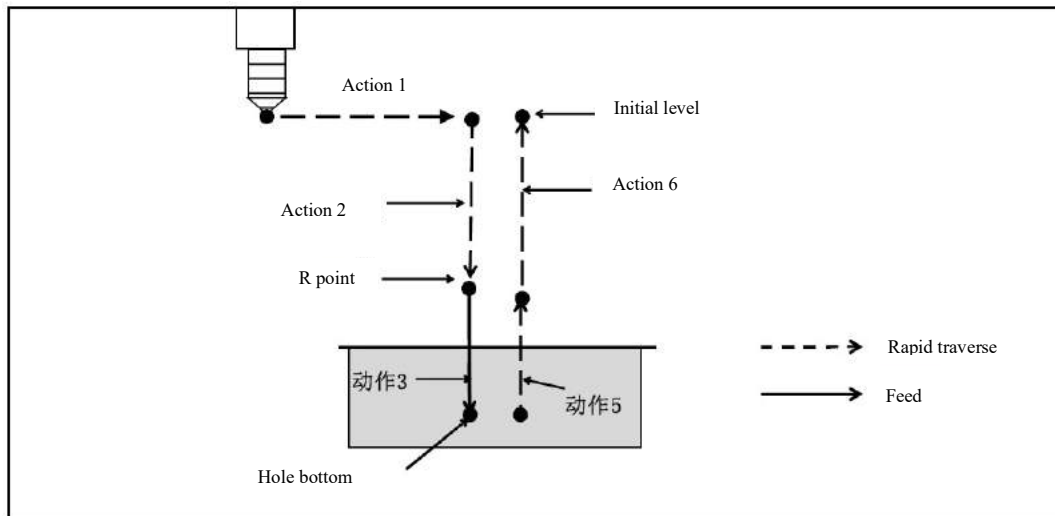
Figure 1.1（a）

- **Positioning plane**

The positioning plane is determined by the plane selection mode G17, G18, and G19.

The positioning axis is an axis other than the drilling axis.

Positioning plane	Positioning axis
G17	X, Y
G18	X, Z
G19	Y, Z

- **Drilling axis**

Hole machining canned cycles include drilling cycles, tapping cycles, boring cycles, etc., but in order to simplify the description, this section will collectively refer to various canned cycles as drilling cycles, and drilling axis, tapping axis, and boring axis are collectively referred to as drilling axis.

The drilling axis is a reference axis (X, Y, Z) that does not form the positioning plane or an axis parallel to it.

The reference axis or parallel axis used as the drilling axis is determined by the positioning plane

of drilling (G17, G18, G19).

If the axis address of drilling axis is not specified, the reference axis is assumed to be the drilling axis.

Positioning axis	Drilling axis
G17	Z-axis
G18	Y-axis
G19	X-axis

● **Drilling data**

G73, G74, G76 and G81 to G89 are all modal G code commands, which are valid until they are cancelled. In these drilling cycle commands, the specified parameters are also modal data,and are kept until they are modified or cleared.

● **Length compensation G43/G44/G49**

G73, G74, G76, and G81 to G89 drilling canned cycles can use tool length compensation. The compensation point is used to establish compensation in the initial plane. G43 is for positive compensation, and G44 is for negative compensation. The use of compensation needs to be established with the tool compensation table. Different tool compensation registers are called to achieve different compensation data.

Note: Through NC parameter 000012 Tool Axis Selection Mode, the Z axis can always be set as the drilling axis. When the parameter is set to 0, the tool length compensation is always compensated on the Z axis. When the parameter is set to 1, the compensation axis is switched based on the machining coordinate selection plane. See drilling axis positioning plane.

●**Movement amount of drilling axis G90/G91**

The movement amount on the drilling axis can be expressed by the coordinate value of the positioning point or the relative movement amount, which is specified by the G90 or G91 command in the program, as shown in Figure 1.1(b):
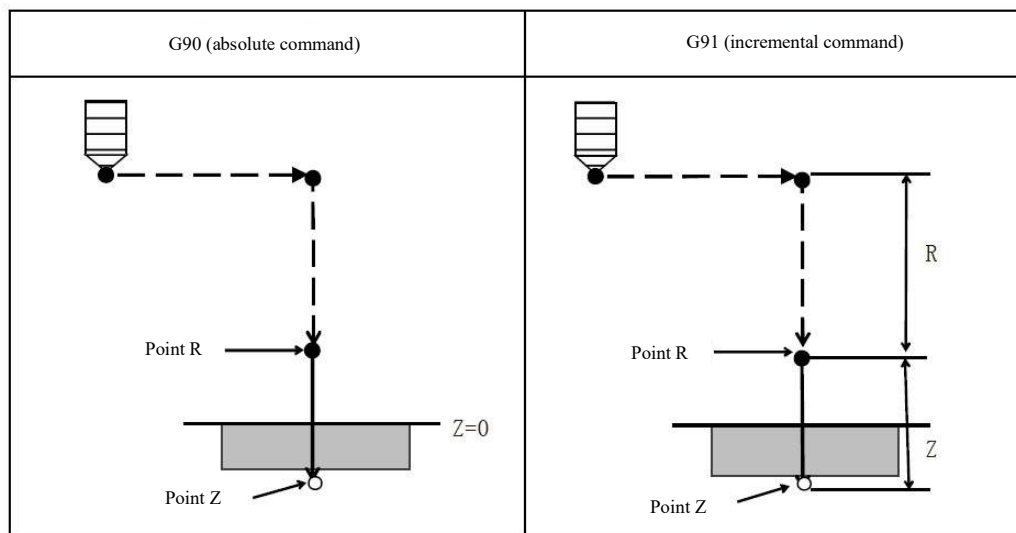
Figure 1.1(b)

● **Return to plane G98/G99**

Whether to return the tool from the hole bottom to the machining level (point R) or to the initial level (point B) is specified by G98 and G99. Figure 1.1(c) shows the action when G98 or G99 is specified. Generally, G99 is used for the first drilling and G98 is used for the final drilling. Even if drilling is performed in G99 mode, the initial level will not be changed.
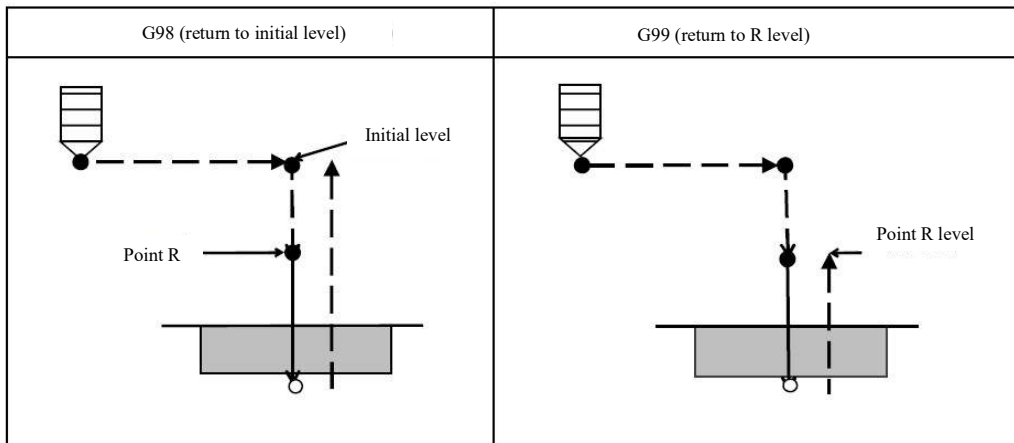
图 1.1(c)

● **Repeat**

When user wants to repeat equidistant drilling, use L_ to specify the number of repetitions with G91 relative coordinate command.

L is only valid in the block where it is specified.

Specify the position of the first hole in incremental mode (G91).

If it is specified in absolute mode (G90), the drilling is repeated at the same hole position.

Maximum value of number of repetitions L is 9999

Specify 0 or an integer value from 1 to 9999 for L.

● **Single block**

When the hole machining cycle is performed in a single block, the control device stops at the end points 1, 2, and 6 in Figure 1.1(a). Therefore, 3 times are required to drill a hole. At the end of actions 1 and 2, the indicator light of feed dwell lights u. At the end of action 6, if the number of repetitions is left, it will stop by feeding dwell. If there is no repetition number, it will stop in the single block stop state. In addition, the R point of G87 will not stop. G88 also stops after dwell at point Z.

● **Cancel**

G80 or the 01 group of G codes is used to cancel the canned cycle.

The 01 group of G codes include:

G00 : positioning (rapid traverse)

G01 : linear interpolation

G02 : circular interpolation or helical interpolation (CW)

G03 : circular interpolation or helical interpolation (CCW)

G60 : unidirectional positioning

● **Explanation of symbols in the figure**

The following describes each canned cycle.

The images used in these descriptions are represented by the following symbols.
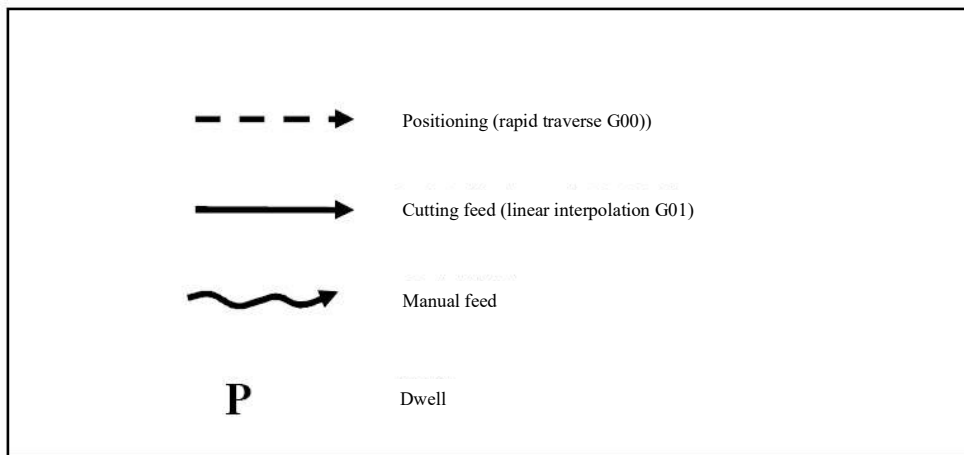
Note

1） When executing a canned cycle block that does not contain X, Y, Z traverse axis commands, this line will not produce tool movement, but the cycle parameter modal value of the current line will be saved;

2） Specifying the 01 group of G codes or specifying G80 will cancel the current canned cycle G code modal, and also clear the cycle parameter modal value;

3） If the canned cycle needs to be repeated by specifying L, when L is specified as 0, an alarm message will appear;

4） When using G53 command in a canned cycle block, its positioning data X, Y are still the original workpiece coordinate system data, not the coordinate system data specified by G53;

5） When switching the drilling axis for drilling, please cancel the drilling canned cycle;

6） To use this canned cycle function, user needs to set the machine user parameter 010083 drilling canned cycle type to 0 to call the HCNC canned cycle.

15.2　High-speed Deep-hole Drilling Cycle（G73）

Function and Purpose

This cycle is for high-speed deep hole drilling. When this command is executed, the tool intermittently cuts and feeds to the bottom of the hole. Intermittent feed makes it easy to break chips, remove chips, add coolant adding during deep hole machining, and the amount of tool retraction is not large.

Command Format

（G98/G99) G73 X_ Y_ Z_ R_ Q_ P_ K_ F_ L_;

Parameter	Meaning
X Y	Coordinate position of hole center in XY plane in absolute programming (G90); The incremental value of hole center relative to the starting point in XY plane in incremental programming (G91);
Z	Coordinate value of Z point of hole bottom in absolute programming; incremental value of Z point of hole bottom relative to the reference point R in incremental programming (G91);
R	Coordinates of reference point R in absolute programming (G90); the incremental value of reference point R relative to the intial point B in incremental programming (G91);
Q	Hole drilling depth downward (incremental value, negative);
P	Dwell time at the hole bottom, in the unit of ms
K	Retraction amount upward (incremental value, positive)
F	Feedrate of drilling;
L	Cycle times (when repeat drilling is needed)
G73（G98）	**G73（G99）**
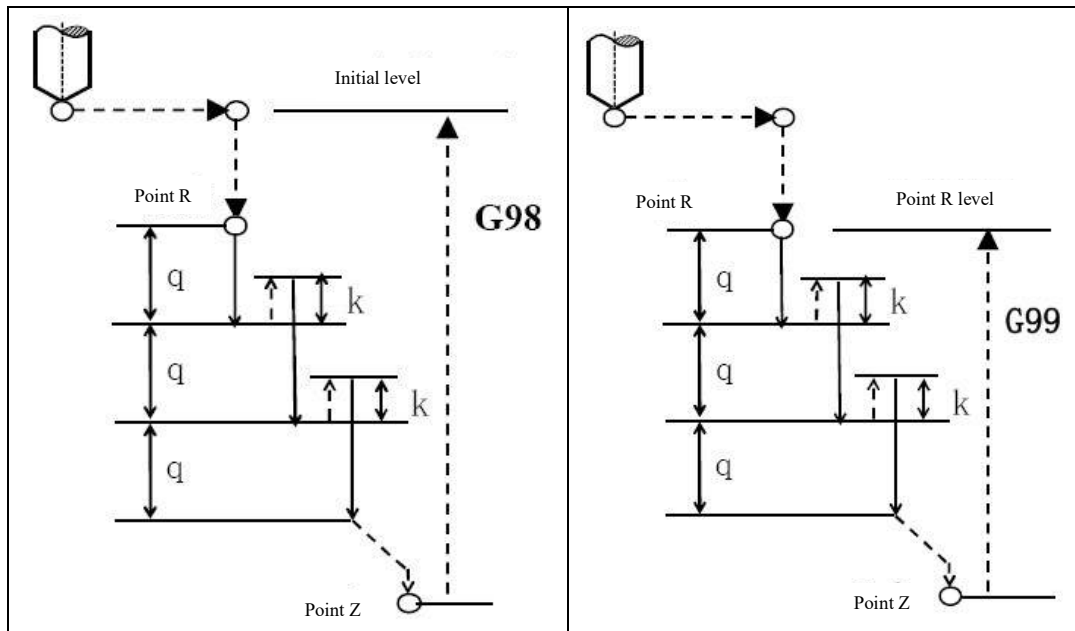
📖 **Description**

● Drilling action

（1）Tool position point moves to B point above the hole center in rapid travers;

（2）Rapid traverse to workpiece surface R point;

（3）Drill downward at F speed with depth of q;

（4）Tool retracts upward with distance of K;

（5）Repeat steps 3 and 4 several times;

（6）Drill to Z point at hole bottom;

（7）Pause at the hole bottom P seconds (spindle keeps rotating);

（8）Return to R point (G99) or B point upward.

● The high-speed deep hole drilling cycle executes intermittent feed along Z axis. When this cycle is used, the chips can be easily discharged from the hole, and a small retraction value can be set, which allows effective drilling.

● Use the auxiliary function to rotate the spindle before specifying G73 (M code).

● When G73 code and M code are specified in the same block, the M code will be executed at the same time as the first positioning action, and then the system will process the next drilling action.

● When the number of repetitions L is specified, the M code is only executed in the first hole, and the M code is not executed in the second and subsequent holes

- When the tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is added while positioning to the R point

- When Z, K, Q movement amounts are zero, this command is not executed.

- |Q|>|K|；

Example

Process the holes as shown in the figure below:
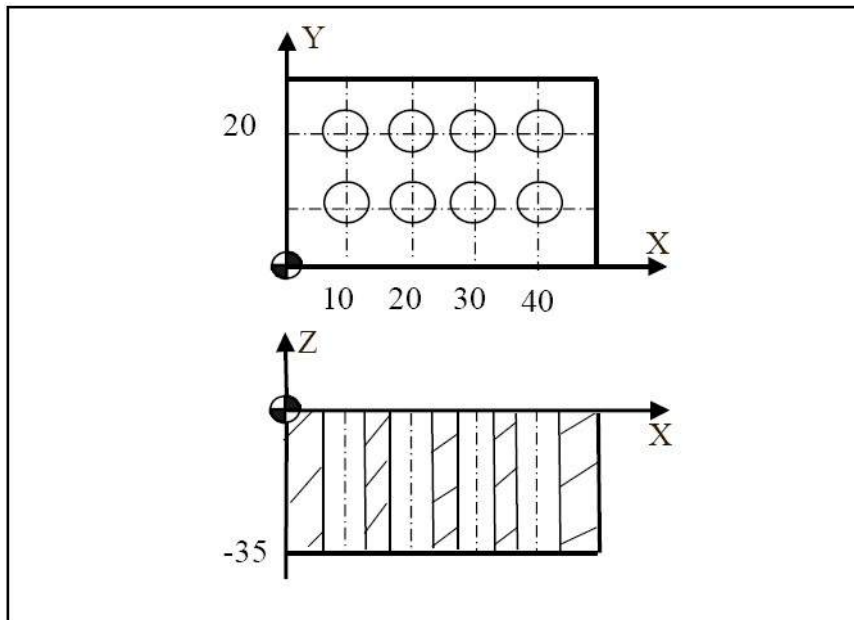
Example 1 (absolute programming, return to R level):

N10   G54 G0 X0 Y0 Z80；Establish coordinate system, move to the safe starting point
N20   M03 S500； Spindle rotates CW
N30   G0 Z20； Position to the intial level
N40   G99 G73 X10 Y10 Z-35 R5 Q-3 K1 F200； After positioning, perform drilling 1,
                                                                         and return to R level
N50   X20； After positioning, complete all drilling, and return to R level
N60   X30
N70   X40
N80   Y20
N90   X30
N100 X20
N110 X10
N120 G80； Cancel G73 canned cycle
N130 G28 G91 X0 Y0 Z0； Return to reference point
N140   M30； Program ends

Example 2 (absolute and incremental mixed programming, return to the initial level):

N10   G54 G90 G0 X0 Y0 Z80； Establish coordinate system, move to the safe starting point

N20   M03 S500； Spindle rotates CW

N30   G0 Z20； Position to the intial level

N40   G98 G73 X10 Y10 Z-35 R5 Q-3 K1 F200； After positioning, perform drilling 1, and return to R level

N50   G91 X10 L3； Increment X10 in turn to complete all drilling, and then return to the initial level

N60   Y10

N70   X-10 L3

N80   G80； Cancel G73 canned cycle

N90   G28 G91 X0 Y0 Z0； Return to reference point

N100   M30； Program ends

Note

(1) Axis switching: Before switching the drilling axis, the canned cycle must be cancelled.

(2) Modality: G73 command data is stored as modal data, and the same data can be omitted.

(3) Drilling: Do not drill in the block that does not include X, Y, Z, R or any other additional axis. If the moving position of Z is zero, the command is not executed.

(4) Q/R: Specify Q/R in the drilling block. If they are specified in the drilling block, they cannot be stored as modal data.

(5) Cancel: Do not specify the G codes of group 01 (G00 to G03, etc.) in the G73 block. Otherwise, G73 will be cancelled.

(6) Tool position offset: In the drilling canned cycle, the tool position offset is ignored.

15.3   Reverse Tapping Cycle（G74）

Function and Purpose

This cycle is for reverse tapping. There are two types of reverse tapping. One is the reverse tapping cycle. The tapping trajectory goes directly from point R to the point Z at the bottom of the hole; the other is the reverse pecking tapping cycle. Repeated intermittent feed is performed during the tapping process.

In the reverse tapping cycle, when the bottom of the hole is reached, the spindle rotates CW and exits. During the tapping or retraction, the tool feeds a thread lead along the tapping axis for each revolution of the spindle, the feed relationship does not change even during acceleration and deceleration.

Command Format

15.3.1 Reverse Tapping Cycle

(G98/G99)G74 X_ Y_ Z_ R_ P_ F_ L_J_；

Parameter	Meaning
X Y	Absolute position of hole in absolute programming (G90); The distance from the current position to the hole position in incremental programming (G91).
Z	Absolute position of hole bottom in absolute programming (G90); The distance from hole bottom to R point in incremental programming (G91).
R	Absolute position of R point in absolute programming (G90); The distance from R point to initial level in incremental programming (G91).
P	The dwell time at the hole bottom, in the unit of ms.
F	Thread lead.
L	Repeat times (can be omitted when L=1).
J	J1 A-axis tapping; J2 B-axis tapping; J3 C-axis tapping.
G74（G98）	**G74（G99）**

G74（G98）   G74（G99）
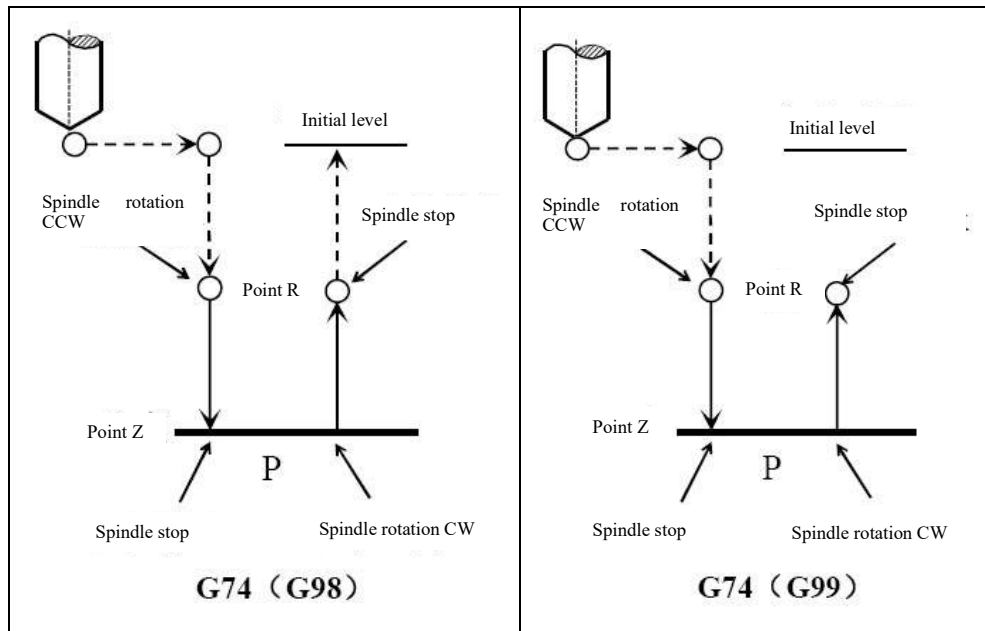
● Reverse tapping action:

(1) Rotate the spindle through the auxiliary function, and the tool position point moves to point B above the hole center in rapid traverse;

(2) The spindle rotates counterclockwise and rapid traverse to the surface of the workpiece to point R

(3) The system calculates the feedrate based on the pitch speed, and taps at the feedrate downward. The depth is Z;

(4) Tap to the hole bottom, and spindle stops;

(5) Dwell at the hole bottom after the dwell time (P) is specified;

(6) Spindle rotates CW, return to R point upward;

(7) Spindle stops;

(9) Return to R point (G99) or B point (G98);

15.3.2 Reverse Peck Tapping Cycle

(G98/G99)G74 X_ Y_ Z_ Q_ K_R_ P_ F_ L_E_J_ ；（Peck tapping）

Parameter	Meaning
X Y	The absolute position of hole in absolute programming (G90); The distance from the current position to the hole position in incremental programming (G91).
Z	The absolute position of hole bottom in absolute programming (G90); The distance from hole bottom to R point in incremental programming (G91).

Q	The feed amount in peck tapping;
K	The retraction amount in peck tapping;
R	The absolute position of R point in absolute programming (G90); The distance from R point to the intial level in incremental programming (G91).
P	The dwell time at the hole bottom, in the unit of ms;
F	The thread lead;
L	Repeat times (it can be omitted when L=1);
E	E1： Peck tapping, feed Q, retract K; E2： Peck tapping, feed Q, return to R level.
J	J1 A-axis tapping; J2 B-axis tapping; J3 C-axis tapping.

● **Reverse tapping cycle action (peck tapping):**

(1) Rotate the spindle through the auxiliary function, and the tool position point moves to point B above the hole center in rapid traverse;

(2) The spindle rotates counterclockwise and rapid traverse to the surface of the workpiece to point R;

(3) The system calculates the feedrate based on the pitch speed, and taps at the feedrate downward. The depth is Q;

(4) Spindle stops;

(5) When dwell time (P) is specified, dwell is performed;

(6) Spindle rotates clockwise, taps upwards and returns to R point;

(7) Spindle stops;

(8) Spindle rotates counterclockwise. Tap downward at the feedrate, the depth is (times*Q);

(9) Repeat steps 4, 5, 6, 7, 8, and arrive at Z point of hole bottom;

(10) Dwell P ms at hole bottom;

(11) Spindle rotates clockwise. Retract upward to R point (G99) or B point (G98) in rapid traverse.

Description

- Perform tapping with spindle rotation CCW. When it reaches the bottom of the hole, the spindle rotates clockwise for return. This cycle processes a reverse thread.

- During reverse tapping, the feedrate override is omitted. The feed is paused, and the machine doesn't stop unit the retraction is completed.

- Before G74 is specified, use the auxiliary function (M code) to counterclockwise rotate spindle.

- When G74 command and M code are specified in the same block, the M code will be executed at the same time as the first positioning action, and then the system will process the next drilling action.

- When the number of repetitions L is specified, the M code is only executed for the first hole, and is not executed for the second and subsequent holes.

- When the tool length offset G43, G44 or G49 is specified in the canned cycle, the offset is added while positioning to the R point.

Example

Process the holes as shown in the figure below:
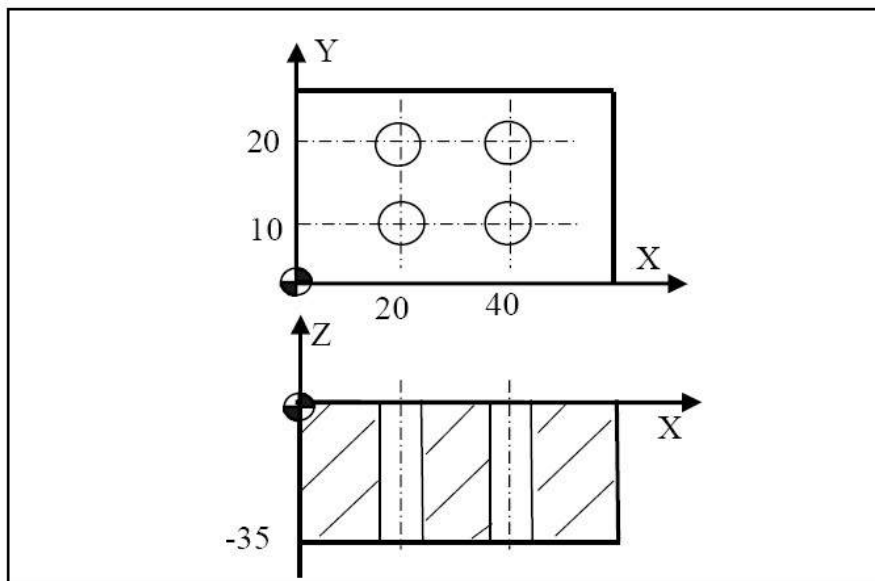
Example 1 (Absolute programming, tapping, return to R level)

%3343

N10   G54 G0 X0 Y0 Z80；   Establish coordinate system, move to safe starting point;

N20   M04 S500；   Spindle rotation CCW starts;

N30   G0 Z20；   Position to the initial level;

N40   G99 G74 X20 Y10 Z-35 R5 P500 F1；   Complete the tapping of hole 1 after
                                        positioning, then return to R level

N50   X40；   Complete tapping of all holes, then return to R level

N60   Y20

N70   X20

N80   G80；   Cancel G74 tapping cycle

N90   G28 G91 X0 Y0 Z0；   Return to reference point

N100 M30；   Program ends

Example 2 (absolute, incremental mixed programming, peck tapping, return to initial level)

%3343

N10   G54 G90 G0 X0 Y0 Z80；   Establish coordinate system, move to safe starting
                                    point;

N20   M04 S500；   Spindle starts;

N30   G0 Z20；   Position to the initial level;

N40   G98 G74 X20 Y10 Z-35 R5 Q-3 K2 E1 F1；   Perform peck tapping of hole 1 after
                                        positioning, then return to initial level;

N50   G91 X20；   Perform peck tapping of the remaining 3 holes, then return to initial
                    level;

N60   Y10

N70   X-20

N80   G80；   Cancel G74 canned cycle;

N90   G28 G91 X0 Y0 Z0；   Return to reference point;

N100   M30；   Program ends.

⚠️ **Note**

(1) F feedrate in tapping: F (feedrate) specified in the program is invalid during rigid tapping, and the feedrate along the tapping axis is calculated by the following formula: feedrate = spindle speed × thread lead

(2) Tapping method: C-axis tapping, the servo spindle is regarded as the C-axis, and the interpolation method is used to tap, which can achieve high-speed and high-precision tapping.

(3) Axis switching: please cancel canned cycle before switching drilling axis.

(4) Modality: G74 command data is stored as modal data, and the same data can be omitted.

(5) Reverse tapping: do not drill in the block that does not contain X, Y, Z, R or any other additional axis. If the moving position of Z is zero, the command will not be executed; point Z must be lower than the level of point R, otherwise alarm will be issued.

(6) Position mode G108 or STOC: Before using the tapping command G74, please pay attention to use STOC command to switch the control mode of the spindle servo motor from speed mode to position mode. After tapping is completed, user can use CTOS to switch the control mode of the spindle servo motor from position mode to speed mode, and the servo spindle is used as a normal spindle.

(7) Cancel: Do not specify the G codes of group 01 (G00 to G03, etc.) in the G74 block. Otherwise, G74 will be cancelled.

(8) Tool position offset: In the drilling canned cycle mode, the tool position offset is ignored.

(9) R: Specify R in the drilling block. If R is specified in the non-drilling block, it cannot be stored as modal data.

(10) P: Please specify P in the drilling block. If it is specified in the non-drilling block, it cannot be stored as modal data.

(11) Q: Please specify Q in the drilling block. If it is specified in the non-drilling block, it cannot be stored as modal data.

(12) Reset: When the reset is performed during rigid tapping, the rigid tapping mode cannot be released, and the machine tool cannot stop the tapping action. Emergency stop is availarble.

(13) Feed hold, single block: In G74 (G84) mode, the feed hold will be executed after each tapping action is completed, and the subsequent tapping will continue to be performed after pressing the cycle start button. In single block mode, the feed hold will be executed after each tapping action is completed, and the next tapping action will be completed after pressing the cycle start button.

(14) Machine lock: The machine lock is also valid for G74 (G84). Even if G74 (G84) is executed when the machine is locked, the drilling axis will not move. Therefore, the spindle will not move.

15.4   Fine Boring（G76）

Function and Purpose

This cycle is for precision hole boring.

In the fine boring cycle, when the bottom of the hole is reached, the spindle stops cutting, and the tool nose leaves the machining surface of the workpiece and returns.

Command Format

(G98/G99) G76 X_ Y_ Z_ R_ I_ J_ P_ F_ L_；

Parameter	Meaning
X Y	Absolute position of hole in absolute programming (G90); Distance from the current position to position of hole in incremental programming (G91). UV programming is not supported.

Z	Absolute position of hole bottom on Z in absolute programming (G90); Distance from the hole bottom to R point in incremental programming (G91).
R	Absolute position of R point on Z in absolute programming (G90); Distance from R point to initial level in incremental programming (G91).
I	Offset on X, can only be positive;
J	Offset on Y, can only be positive;
P	Dwell time at hole bottom (unit: ms);
F	Cutting feedrate;
L	Repeat times (it can be omitted when L=1)

G76（G98）	G76（G99）

📖 **Description**

● Fine boring cycle action

(1) Move to the tool position to point B above the hole center in rapid traverse

(2) Move close to the surface of the workpiece to point R in rapid traverse

(3) Bore downward at F speed, reach point Z at the bottom of the hole

(4) Dwell Z ms at hole bottom (spindle keeps rotating)

(5) Spindle orientation, spindle stops rotating

(6) The boring tool quickly moves I or J in the opposite direction of the tool nose

(7) Retract upward to R level (G99) or B level (G98) in rapid traverse

(8) Quickly move I or J in the positive direction of the tool nose, and the tool position point will return to point R or B above the hole center

(10) Spindle resumes CW rotation

● The drilling axis must be Z axis;

● Z point must be lower than R level; otherwise an alarm will be issued;

● G76 command data is stored as modal data, the same data can be omitted;

● Before using the command G76, please use the corresponding M code to rotate the spindle.

Example

Process the hole as shown below,
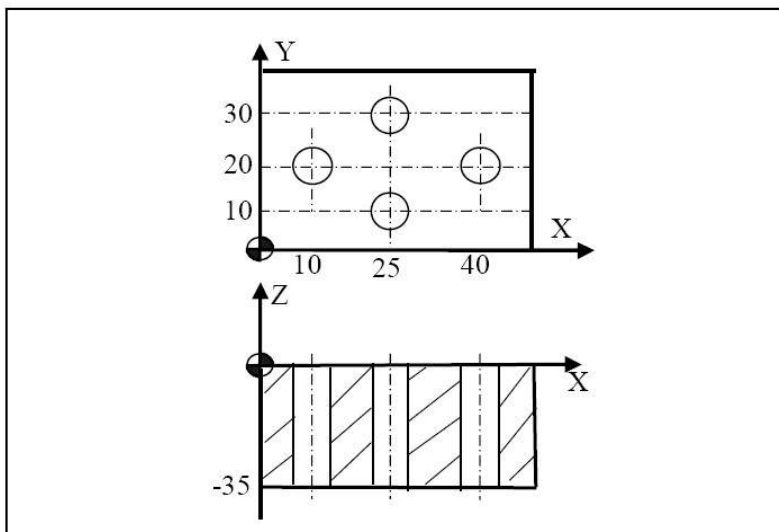
Example 1 (absolute programming, return to R level):

    %3341
    N10 G54 G90 G0 X0 Y0 Z80； Establish coordinate system, move to safe starting point
    N20M03 S600； Spindle rotation CW starts
    N30G00 Z20； Position to initial level
    N40 G99 G76X10Y20 Z-35 R5 I1 P2000 F100； Complete boring of hole 1 after positioning, return to the level of point R
    N50X25 Y30； Complete boring of the remaining holes after positioning, return to the level of point R
    N60 X40 Y20
    N70 X25 Y10
    N80 G80； Cancel G76 boring cycle
    N90 G91G28 X0 Y0 Z0； Return to reference point
    N100 M30； Program end

Example 2 (absolute and incremental mixed programming, return to the initial level):

    %3343

N10　G54 G90 G0 X0 Y0 Z80；　Establish coordinate system, move to safe starting point

N20　M03 S500；　Spindle rotation CCW starts

N30　G0 Z20；　Position to initial level

N40　G98 G76 X10 Y20 Z-35 R5 I5 P500 F100；　Complete boring of hole 1 after positioning, return to the level of point R

N50　G91 X15 Y10；　Complete boring of the remaining holes in incremental programming, return to the initial level

N60　X15　Y-10

N70　X-15 Y-10

N80　G80；　Cancel G76 boring cycle

N90　G28 G91 X0 Y0 Z0；　Return to reference point

N100　M30；　Program end

![Note icon] **Note**

(1) Axis switching: Before switching the boring axis, please cancel the canned cycle.

(2) Modality: G76 command data is stored as modal data, the same data can be omitted;

(3) Boring processing: do not drill in the block that does not include X, Y, Z, R or any other additional axis. If the moving position of Z is zero, the command is not executed;

(4) I J R: if specify I J R in the boring block, I or J can only be a positive value. If I or J is specified as a negative value, the sign is ignored. If they are specified in the non-boring block, they cannot be stored as modal data;

(5) Cancel: Do not specify the G codes of group 01 (G00 to G03, etc.) in the G76 block. Otherwise, G76 will be cancelled;

(6) Tool position offset: in the boring canned cycle mode, the tool position offset is ignored.

15.5 Drilling Cycle (Center Drills)（G81）

Function and Purpose

This cycle is for the normal drilling.

The feed cutting is performed to the hole bottom, and the tool returns from the hole bottom in rapid traverse.

Command Format

(G98/G99) G81 X_ Y_ Z_ R_ F_ L_ ;

Parameter	Meaning
X Y	The hole position data. The absolute position of the hole in absolute programming, and the distance from the current position to the hole position in incremental programming;
Z	The position of hole bottom. The absolute position of hole bottom on Z in absolute programming (G90); the distance from hole bottom to R point in incremental programming (G91);
R	Position of R point. The absolute position of R point on Z in absolute programming (G90); the distance from R point to initial level in incremental programming (G91);
F	Cutting feedrate
L	Number of repetitions (can be omitted when L=1, generally used for multi-hole processing, so X and Y should be incremental values)

G81（G98）	G81（G99）

Description

● **Action**

Tool position point moves to the B point above hole center in rapid traverse;

Move close to the workpiece surface to R point in rapid traverse;

Drilling downward at F speed to the hole bottom Z point;

Spindle keeps rotating; retract upward to R point (G99) or B point (G98) in rapid traverse.

- **Spindle rotation**

Before G81 is specified, the auxiliary function (M code) is used to rotate spindle.

- **Auxiliary function**

When G81 and M code are specified in the same block, the M code is executed during the initial positioning.

When the number of repetitions L is specified, the above actions will only be executed for the first time. The M code will not be executed after the second time.

- **Tool length compensation**

When tool length compensation (G43, G44, G49) is specified in the drilling canned cycle, the compensation is used when positioning to the R point.

Example

Process the hole shown as below,
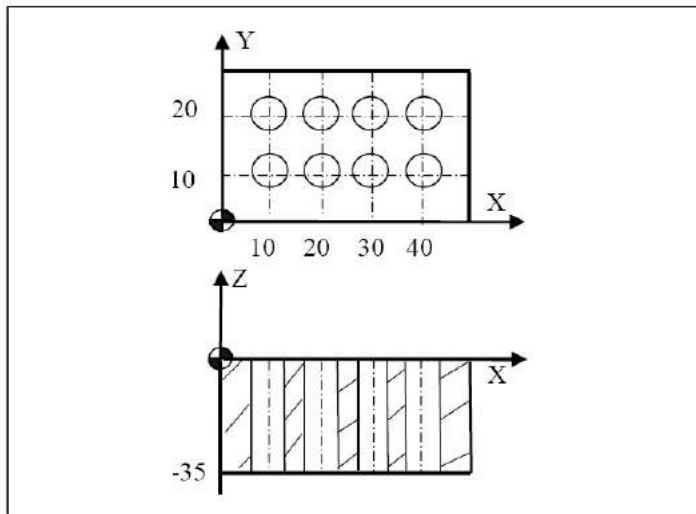
Example 1

%3343
G54G0 X0 Y0 Z30； Return to the safe height of workpiece zero
M03 S600； Spindle rotation CW
G0 Z10； Position to the initial plane
G90G99 G81 X10 Y10 R5 Z-35 F200； Position to the hole (10 10), perform drilling, return to R level
G91X10L3； Drill the last three holes on horizontal axis, return to R level
Y10； Position to the hole (40 20), perform drilling, return to R level
G91X10L3； Drill the last three holes on horizontal axis

X-10L3； Drill the first three holes on horizontal axis, and return to R level

G80； Cancel drilling canned cycle

G28 G91 Z0； Return to machine zero of Z axis

G28 G91 X0 Y0； Return to machine zero of X axis and Y axis

M30； Program end

⚠ Note

(1) Axis switching: please cancel canned cycle before switching drilling axis.

(2) Modality: G81 command data is stored as modal data, the same data can be omitted;

(3) Drilling: Drilling cannot be in the block that does not include X, Y, Z, R or any other additional axis. If the moving position of Z is zero, the command is not executed;

(4) Cancel: Do not specify the G codes of group 01 (G00 to G03, etc.) in the G81 block. Otherwise, G81 will be cancelled.

(5) Tool position offset: In the drilling canned cycle mode, the tool position offset is ignored.

15.6   Drilling Cycle with Pause（G82）

    Function and Purpose

This cycle is used for normal drilling, counternboring, and blind hole processing.

The cutting feed proceeds to the bottom of the hole. Tool pauses at the bottom of the hole, and then retracts from the bottom of the hole in rapid traverse.

This cycle can improve the accuracy of the hole depth.

This command is the same as G81 except that it needs to pause at the bottom of the hole.

    Command Format

(G98/G99) G82 X_ Y_ Z_ R_ P_ F_ L_ ;

Parameter	Meaning
X Y	The absolute position of the hole in absolute programming, and the distance from the current position to the hole position in incremental programming;
Z	The absolute position of hole bottom in absolute programming (G90); the distance from hole bottom to R point in incremental programming (G91);
R	The absolute position of R point in absolute programming (G90); the distance from R point to initial level in incremental programming (G91);
P	Dwell time at the hole bottom (unit: ms)
F	Cutting feedrate
L	Number of repetitions (can be omitted when L=1, generally used for multi-hole processing)
G82（G98）	**G82（G99）**

📖 **Description**

- **Action**

(1) Tool position point moves to B point above hole center in rapid traverse;

(2) Move close to the workpiece surface to R point;

(3) Drill downward at F speed to Z point of hole bottom;

(4) Spindle keeps original rotating. Pause P ms

(5) Retract upward to R point (G99) or B point (G98)

- **Spindle rotation**

Before G82 is specified, the auxiliary function (M code) can be used to rotate spindle.

- **Auxiliary function**

When G81 and M code are specified in the same block, the M code is executed during the initial positioning.

When the number of repetitions L is specified, the above actions will only be executed for the first time. The M code will not be executed after the second time.

- **Tool length compensation**

When tool length compensation (G43, G44, G49) is specified in the drilling canned cycle, the compensation is used when positioning to the R point.

Example

Example 1

%3343

G54G0 X0 Y0 Z30；　Return to the safe height of workpiece zero

M03 S600；　Spindle rotation CW

G0 Z10；　Position to the initial level

G99 G82 X10 Y10 R5 Z-35 P1000 F200；　Position to the hole (10 10), perform drilling, return to R level

X10Y20；　Position to the hole (10 20), perform drilling, return to R level

Y20Y15；　Position to the hole (20 15), perform drilling, return to R level

X30Y10；　Position to the hole (30 10), perform drilling, return to R level

X30Y20；　Position to the hole (30 20), perform drilling, return to R level

G80；　Cancel drilling canned cycle

G28 G91 Z0；　Return to machine zero of Z axis

G28 G91 X0 Y0；Return to machine zero of X axis and Y axis

M30

> **Note**

(1) Axis switching: please cancel canned cycle before switching drilling axis;

(2) Modality: G81 command data is stored as modal data, the same data can be omitted;

(3) Drilling: Drilling cannot be in the block that does not include X, Y, Z, R or any other additional axis. If the moving position of Z is zero, the command is not executed;

(4) Cancel: Do not specify the G codes of group 01 (G00 to G03, etc.) in the G82 block. Otherwise, G82 will be cancelled.

(5) Tool position offset: In the drilling canned cycle mode, the tool position offset is ignored.

(6) P: Please specify P in the drilling block. If it is specified in the non-drilling block, it cannot be stored as modal data.

15.7　Deep Hole Drilling Cycle (G83)

> **Function and Purpose**

This cycle is for deep hole processing.

This canned cycle is used for intermittent feed on Z axis. After drilling of each hole downward, the tool quickly retracts to the reference point R. The retraction amount is larger, which is better for chip removal and coolant adding.

> **Command Format**

(G98/G99) G83 X_ Y_ Z_ R_ Q_ K_ F_ L_ P_ ；

Parameter	Meaning
X Y	The absolute position of the hole in absolute programming, and the distance from the current position to the hole position in incremental programming;
Z	The absolute position of hole bottom in absolute programming (G90); the distance from hole bottom to R point in incremental programming (G91);
R	The absolute position of R point in absolute programming (G90); the distance from R point to initial level in incremental programming (G91);
Q	Drilling depth downward (incremental value, negative)
K	Distance from the upper surface of the machined hole (incremental value, positive) Note: K cannot be larger than Q
F	Cutting feedrate
L	Number of repetitions (can be omitted when L=1, generally used for multi-hole processing)
P	Dwell time at the hole bottom (unit: ms)

G83（G98）	G83（G99）

Description

● **Action**

(1) Tool position point moves to B point above hole center in rapid traverse;

(2) Move close to the workpiece surface to R point;

(3) Drill downward at F speed with depth of q;

(4) Retract upward to R point;

(5) Move down to the top of the processed hole, at a distance of k;

(6) Drill downward at F speed with depth of （q+k）;

(7) Repeat steps 4, 5, 6. Reach Z point at hole bottom;

(8) Pause P ms at hole buttom (Spindle keeps original rotating).

(9) Retract upward to R point (G99) or B point (G98).

- **Spindle rotation**

Before G83 is specified, the auxiliary function (M code) can be used to rotate spindle.

- **Auxiliary function**

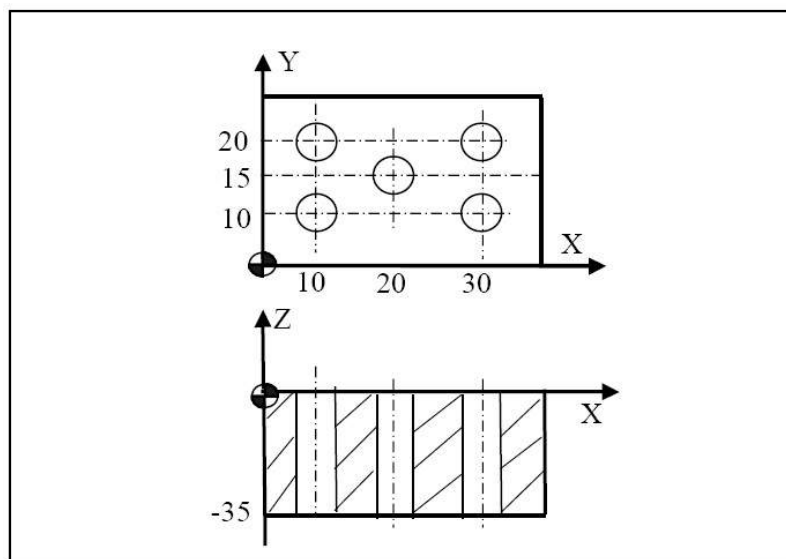When G83 and M code are specified in the same block, the M code is executed during the initial positioning. When the number of repetitions L is specified, the above actions will only be executed for the first time. The M code will not be executed after the second time.

- **Tool length compensation**

When tool length compensation (G43, G44, G49) is specified in the drilling canned cycle, the compensation is used when positioning to the R point.

Example

Process the hole as shown below.
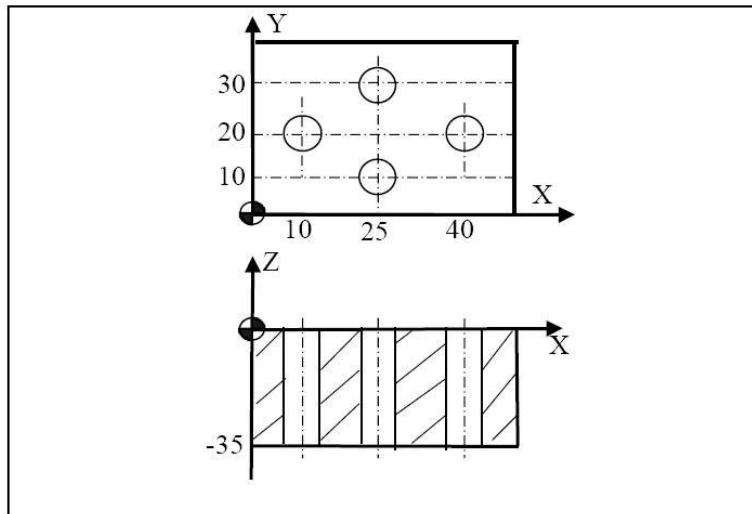
Example 1

%3343
G54G0 X0 Y0 Z30； Return to the safe height of workpiece zero
M03 S800； Spindle rotation CW
G0 Z10； Position to the initial level
G99 G83 X10 Y20 R5 Z-35 Q-1 K0.2 F200； Position to the hole (10 20), perform

drilling, return to R level

    X25Y30；   Position to the hole (25 30), perform drilling, return to R level

    X40Y20；   Position to the hole (40 20), perform drilling, return to R level

    X25Y10；   Position to the hole (25 10), perform drilling, return to R level

    G80； Cancel drilling canned cycle

    G28 G91 Z0；   Return to machine zero of Z axis

    G28 G91 X0 Y0；   Return to machine zero of X axis and Y axis

    M30；   Program end;

Note

(1) Axis switching: please cancel canned cycle before switching drilling axis.

(2) Modality: G81 command data is stored as modal data, the same data can be omitted;

(3) Drilling: Drilling cannot be in the block that does not include X, Y, Z, R or any other additional axis. If the moving position of Z is zero, the command is not executed;

(4) Cancel: Do not specify the G codes of group 01 (G00 to G03, etc.) in the G81 block. Otherwise, G81 will be cancelled.

(5) Tool position offset: In the drilling canned cycle mode, the tool position offset is ignored.

(6) P: Please specify P in the block of drilling. If it is specified in a non-drilling block, it cannot be stored as modal data.

(7) Q: Please specify Q in the block of drilling. If it is specified in a non-drilling block, it cannot be stored as modal data.

15.8 Tapping Cycle (G84)

Function and Purpose

With this function, the tapping of spindle rotation CW can be realized. There are two types of reverse tapping. One is the tapping cycle, that is, the tapping path goes directly from point R to point Z at the bottom of the hole; the other is the peck tapping cycle, that is, repeated intermittent feed is performed during the tapping process.

In the tapping cycle, when the spindle rotates clockwise in tapping, and reaches the bottom of the hole, the spindle rotates counterclockwise and exits. During the tapping or exit process, the spindle feeds a thread lead along the tapping axis for each revolution of the spindle, and the feed relationship does not change even during acceleration and deceleration.

When the PWM spindle is used, the tapping spindle moves synchronously with the spindle, which is referred to as following tapping; when the servo spindle is used, both the spindle motor and servo motor work in the position control mode, and the tapping is performed via the interpolation

between tapping axis and spindle, which is referred to as the synchronous tapping. The programming commands are the same for the following tapping and synchronous tapping.

Note that the pitch of the tap used for tapping should be consistent with that in the programming

15.8.1 Tapping Cycle（G84）

Command Format

(G98/G99)G84 X_ Y_ Z_ R_ P_ F_ L_ J_ ;

Parameter	Meaning
X Y	The absolute position of the hole in absolute programming (G90), and the distance from the current position to the hole position in incremental programming (G91);
Z	The absolute position of hole bottom in absolute programming (G90); the distance from hole bottom to R point in incremental programming (G91);
R	The absolute position of R point in absolute programming (G90); the distance from R point to initial level in incremental programming (G91);
P	Dwell time at the hole bottom (unit: ms)
F	Thread lead
L	Number of repetitions (can be omitted when L=1)
J	J1 A-axis tapping; J2 B-axis tapping; J3 C-axis tapping
G84（G98）	**G84（G99）**

- **Action**

(1) Rotate the spindle through the auxiliary function, and the tool position point will move to point B above the hole center in rapid traverse;

(2) Spindle rotates CW, and approaches to the workpiece surface in rapid traverse, and reaches to R point;

(3) System calculates feedrate base on the pitch speed; tap downward at the feedrate with the depth of Z;

(4) Tap to the hole bottom, spindle stops;

(5) After the dwell time P is specified, pausing at hole bottom is performed;

(6) Spindle rotates CCW, tap upward in rapid traverse, retract to R point;

(7) Spindle stops;

(8) Retract upward to R point (G99) or B point (G98) in rapid traverse.

15.8.2 Peck Tapping Cycle (G84)

(G98/G99)G84 X_ Y_ Z_ Q_ K_R_ P_ F_ L_ E_J_；（peck tapping）

Parameter	Meaning
X Y	The absolute position of the hole in absolute programming (G90), and the distance from the current position to the hole position in incremental programming (G91)
Z	The absolute position of hole bottom in absolute programming (G90); the distance from hole bottom to R point in incremental programming (G91)
Q	Feed amount
K	Retraction amount
R	The absolute position of R point in absolute programming (G90); the distance from R point to initial level in incremental programming (G91);
P	Dwell time at the hole bottom (unit: ms)
F	Thread lead
L	Number of repetitions (can be omitted when L=1)
E	E1: peck tapping, feed Q, retract K E2, peck tapping, feed Q, retract to R level
J	J1 A-axis tapping; J2 B-axis tapping; J3 C-axis tapping

- **Peck tapping cycle action**

(1) Rotate the spindle through the auxiliary function, and the tool position point will move to

point B above the hole center in rapid traverse;

(1) Spindle rotates CW; approaches to the workpiece surface in rapid traverse, and reaches to R point;

(1) System calculates feedrate based on the pitch speed, tap downward at the feedrate with the depth of Z;

(2) Spindle stops;

(3) After the dwell time P is specified, pausing at hole bottom is performed;

(4) Spindle rotates CCW; tap upward in rapid traverse, retract to R point;

(5) Spindle stops;

(6) Spindle rotates CW, tap downward at the feedrate with the depth of (times*Q)

(7) Repeat steps 4, 5, 6, 7, 8. Reach Z point at hole bottom;

(8) Dwell P ms at hole bottom (spindle keeps original rotation)

(9) Retract upward to R point (G99) or B point (G98) in rapid traverse.

Description

- **Spindle rotation**

Before G84 is specified, the auxiliary function (M code) can be used to rotate spindle. The spindle revolutions may affect the tapping speed.

- **Auxiliary function**

When G84 and M code are specified in the same block, the M code is executed during the initial positioning. When the number of repetitions L is specified, the above actions will only be executed for the first time. The M code will not be executed after the second time.

- **Tool length compensation**

When tool length compensation (G43, G44, G49) is specified in the drilling canned cycle, the compensation is used when positioning to the R point.

- **Feedrate in tapping**

During rigid tapping, the F (feedrate) specified in the program is invalid, and the feedrate along the tapping axis is calculated by the following formula:

Feedrate = spindle speed × thread lead

- **Tapping mode**

C-axis tapping: The servo spindle is used as the C-axis. High-speed and high-precision tapping can be achieved using interpolation for tapping;

Example

Process the hole as shown below.
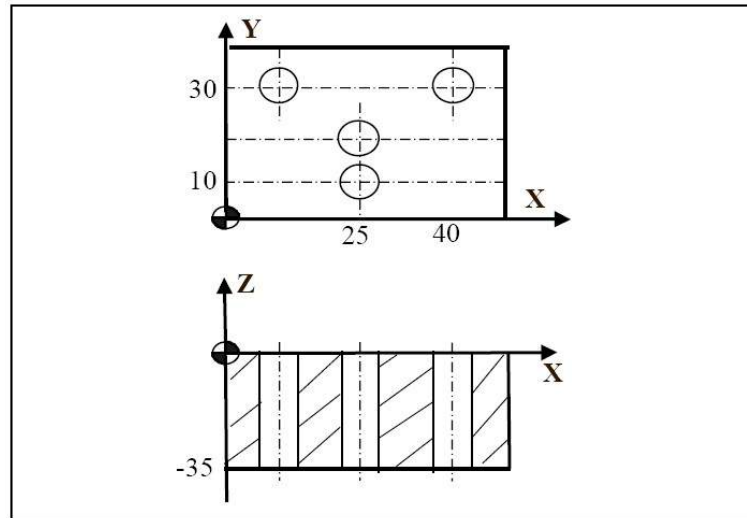
Example 1

%3343

G54G0 X0 Y0 Z30；　Return to the safe height of workpiece zero

M03 S3000；　Spindle rotation CW

G0 Z10；　Position to the initial level

G90G99 G84 X25 Y10 R5 Z-35P1000 F1；　Position to the hole (25 10), perform drilling, return to R level

X25Y20；　Position to the hole (25 20), perform drilling, return to R level

X10Y30；　Position to the hole (10 30), perform drilling, return to R level

X40Y30；　Position to the hole (40 30), perform drilling, return to R level

G80；　Cancel canned cycle;

G28 G91 Z0；　Return to machine zero on Z axis

G28 G91 X0 Y0；　Return to machine zero on X axis and Y axis

M30；Program end

⚠ **Note**

Axis switching: please cancel canned cycle before switching drilling axis.

Modality: G84 command data is stored as modal data, the same data can be omitted.

Drilling: Drilling cannot be in the block that does not include X, Y, Z, R or any other additional axis. If the moving position of Z is zero, the command is not executed. Z point must be lower than R level, otherwise an alarm will be issued.

Position mode G108 or STOC: Before the tapping command G84 is used, please switch the spindle servo motor from the speed mode to the position mode by the STOC command. After the tapping, user could switch the spindle servo motor from position mode to speed mode by CTOS

command, and use the servo spindle as the ordinary spindle.

Scaling and Rotation: the rotation or scaling are not supported in the process of rigid tapping.

Cancel: Do not specify the G codes of group 01 (G00 to G03, etc.) in the G84 block. Otherwise, G84 will be cancelled.

Tool position offset: In the drilling canned cycle mode, the tool position offset is ignored.

P: Please specify P in the drilling block. If it is specified in the non-drilling block, it cannot be stored as modal data.

Q: Please specify Q in the drilling block. If it is specified in the non-drilling block, it cannot be stored as modal data.

Reset: when the reset is executed during rigid tapping, the rigid tapping mode cannot be released, and the machine tool cannot stop tapping.

Emergency stop can be effective.

Feed hold, single block: In G84 (G74) mode, the feed hold will be executed after each tapping action is completed, and the subsequent tapping will continue after pressing the cycle start button.

The feed hold is executed after each tapping action is completed in single block mode, and the next tapping action will be completed after pressing the cycle start button.

Machine lock: machine lock is also effective to G84 (G74).

Even if G84 (G74) is executed in the machine lock state, the drilling axis will not move. Therefore, the spindle will not move.

15.9   Boring Cycle (G85)

Function and Purpose

This cycle is for hole boring.

In the boring cycle, when the bottom of the hole is reached, the cutting feed is executed until tool returns to the R point. If in G98 mode, the tool will quickly return to the initial level after reaching R point.

Command Format

(G98/G99) G85 X_ Y_ Z_ R_ P_ F_ L_;

Parameter	Meaning
X Y	The absolute position of the hole in absolute programming, and the distance from the current position to the hole position in incremental programming;

Z	The absolute position of hole bottom in absolute programming (G90); the distance from hole bottom to R point in incremental programming (G91);
R	The absolute position of R point on Z in absolute programming (G90); the distance from R point to initial level in incremental programming (G91);
P	Dwell time at the hole bottom (unit: ms)
F	Cutting feedrate
L	Number of repetitions (can be omitted when L=1)

G85（G98）	G85（G99）

Description

- Boring cycle actions

（1）Tool position point moves to B point above hole center in rapid traverse;

（2）Move close to the workpiece surface, and reach to R point;

（3）Bore downward at F speed;

（4）Reach Z point at hole bottom;

（5）Retract upward to R point at F speed (spindle keeps rotating);

（6）Retract upward to B point in rapid traverse if in G98.

- The drilling axis must be Z axis;

- Z point must be lower than B level; otherwise, the alarm will be issued;

- If the movement amounts of Z, Q, K are zero, then the command will not be executed;

- G85 command data is stored as the modal data, and the same data can be omitted;

- Please use corresponding M code to rotate spindle before G85 is used;

- When G85 and M code are specified in the same block, the M code is executed at the

same time as the first positioning action, and then the system processes the next boring action;

● When the number of repetitions L is specified, the M code is only executed for the first hole, and the M code is not executed for the second or later holes;

● When the tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset will be added while positioning to the R point;

Example

Machine the hole as shown below,
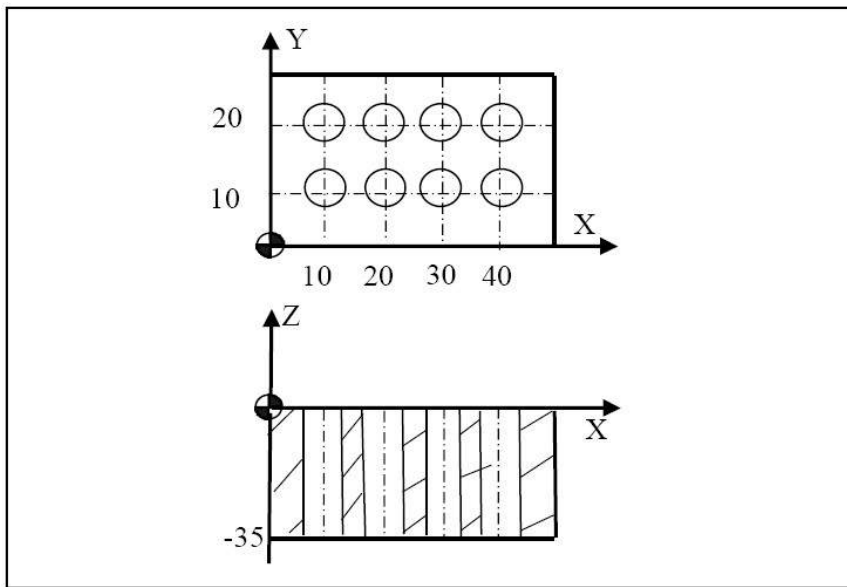
Example 1 (absolute programming, return to R level)

%3341
N10 G54 G90 G0 X0 Y0 Z80； Establish coordinate system, move to the safe initial point
N20M03 S600； Spindle rotates CW
N30G00 Z20； Position to initial level
N40 G99 G85 X10Y10Z-35 R5 P500 F100； After positioning, complete boring 1, then return to R level
N50X20； After positioning, complete all boring, then return to R level
N60 X30
N70 X40
N80 Y20
N90 X30
N100 X20
N110 X10

N120 G80； Cancel G85 boring cycle

N130 G91G28 X0 Y0 Z0； Return to reference point

N140 M30；Program end

Example 2 (absolute and incremental mixed programming, return to the initial level)

%3343

N10 G54 G90 G0 X0 Y0 Z80； Establish coordinate system, move to the safe initial point

N20 M03 S500； Spindle rotates CW

N30 G0 Z20； Position to initial level

N40 G98 G85 X10 Y10 Z-35 R5 P500 F100； After positioning, complete boring 1, then return to R level

N50 G91 X10 L3； Increment X10 in turn to complete boring 3 holes, and then return to the initial level

Y10

X-10 L3

N60 G80； Cancel G85 boring cycle

N70 G28 G91 X0 Y0 Z0； Return to reference point

N80 M30；Program end

⚠ **Note**

(1) Axis switching: Before switching the boring axis, the canned cycle must be cancelled.

(2) Modality: G85 command data is stored as modal data, and the same data can be omitted.

(3) Boring: Drilling is not performed in the block that does not include X, Y, Z, R or any other additional axis. If the moving position of Z is zero, this command is not executed.

(4) Cancel: Do not specify the G codes of group 01 (G00 to G03, etc.) in the G85 block. Otherwise, G85 will be cancelled.

(5) Tool position offset: In the boring canned cycle mode, the tool position offset is ignored.

15.10 Boring Cycle (G86)

✎ **Function and Purpose**

The action performed by G86 is the same as that of G81, but the spindle stops at the bottom of the hole and then quickly retracts. It is mainly used for low precision boring.

⚙ **Command Format**

(G98/G99) G86 X_ Y_ Z_ R_ F_ L_；

Parameter	Meaning
X Y	The absolute position of the hole in absolute programming(G90), and the distance from the current position to the hole position in incremental programming(G91);
Z	The absolute position of hole bottom in absolute programming (G90); the distance from hole bottom to R point in incremental programming (G91);
R	The absolute position of R point in absolute programming (G90); the distance from R point to initial level in incremental programming (G91);
F	Cutting feedrate
L	Number of repetitions (can be omitted when L=1)

G86（G98）	G86（G99）

Description

● **Action**

After positioning along X-axis and Y-axis, the tool position quickly moves to point B above the hole center;

Move close to the workpiece surface, and reach to R point;

Drill downward at F speed;

Reach to Z point of hole bottom;

Spindle stops rotating;

Retract upward to R point (G99) or B point (G98)

Spindle resumes CW rotation.

● **Spindle rotation**

Rotate spindle with auxiliary function (M code) before using G86.

When continuously performing the drilling with a short distance from the hole position and the initial level to the R point level, the spindle may not be able to rotate normally before entering the hole cutting action. In this case, do not specify the number of repetitions L, and user needs to insert GO4-based pause before each drilling action to let time out.

● **Tool length compensation**

When the tool length compensation (G43, G44, G49) is specified in drilling canned cycle, this compensation should be used at the time of positioning to R point.

● **Auxiliary function**

When G86 command and M code are specified in the same block, the M code is executed during the initial positioning. When the number of repetitions L is specified, the above actions will be executed only the first time, and M codes will not be executed after the second time.
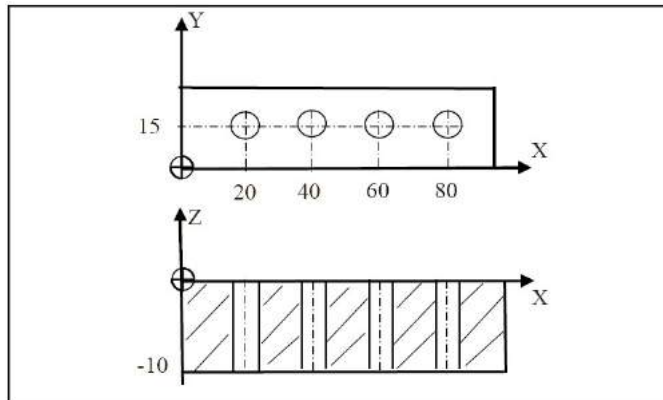
Example

Example 1：

%3353； Reaming with a reamer
G54G00X0Y0Z50； Establish coordinate system, reach to the safe starting point
M3 S2000； Spindle starts
G99 G86G90X20Y15R20 Z-10 F120； After positioning, drill hole 1, and return to R level
G91 X20 L3； Ream holes 2, 3, 4 in turn, and return to the R level after each hole is milled
G80 G91 G28 X0 Y0 Z0； Cancel G86 boring cycle, return to the reference point
M30； Program end

Note

(1) Cancel canned cycle before switching drilling axis.

(2) The drilling axis must be Z axis in G17 plane.

(3) If the moving position is zero on Z, this command will not be executed.

(4) G86 data is saved as the modal data, and the same data can be omitted.

(5) Z point must be lower than R level, otherwise the alarm will be issued.

(6) Drilling is not performed in the blocks that not includes X, Y, Z, R or any other additional axis.

(7) The tool position offset is omitted in the drilling canned cycle mode.

(8) Do not specify the G code (G00-G03) of group 01 in the G86 block; otherwise, G86 will be cancelled.

15.11   Reverse Boring Cycle (G87)

Function and Purpose

This command is generally used for boring the hole with small upper surface and large lower surface, and Z point at hole bottom is generally above the reference point R, which is different from other commands.

Command Format

(G98/G99) G87X_ Y_ Z_ R_ I_ J_ P_ F_ L_ ;

Parameter	Meaning
X Y	The absolute position of the hole in absolute programming(G90), and the distance from the current position to the hole position in incremental programming(G91);
Z	The absolute position of hole bottom on Z in absolute programming (G90); the distance from hole bottom to R point in incremental programming (G91);
R	The absolute position of R point in absolute programming (G90); the distance from R point to initial level in incremental programming (G91);
I	Offset amount on X axis;
J	Offset amount on Y axis;
P	Dwell time at hole bottom (unit: ms)
F	Cutting feedrate;
L	Number of repetitions (can be omitted when L=1);

G87（G98）	G87（G99）
Initial point offset (I, J) Z point Positioning R point R Reverse offset (I, J)	Unavailable

Description

- **Actions**

Tool position point moves to B point above hole center in rapid traverse, after positioning along X axis and Y axis

Spindle orientates and stops rotation

The boring tool moves I or J in the opposite direction of tool nose;

Move to R point in rapid traverse

The boring tool moves I or J in the positive direction of tool nose, tool position point returns to hole center X, Y

Spindle rotates CW

Boring upward at F speed, reach to Z point at hole bottom;

Dwell P ms at hole bottom (spindle keeps rotating)

Spindle orientates and stops rotating

Tool moves I or J in the opposite direction of the tool nose;

Retract to B point (G98) upward in rapid traverse

Move I or J in the positive direction of tool nose, tool position point returns to B point above hole center

Spindle restores to CW rotation.

- **Spindle rotation**

Rotate spindle with auxiliary function (M code) before using G87.

- **Tool length compensation**

When the tool length compensation (G43, G44, G49) is specified in drilling canned cycle, this compensation should be used at the time of positioning to R point.

- **Auxiliary function**

When G87 command and M code are specified in the same block, the M code is executed during the initial positioning. When the number of repetitions L is specified, the above actions will be executed only at the first time, and M codes will not be executed after the second time.

Example
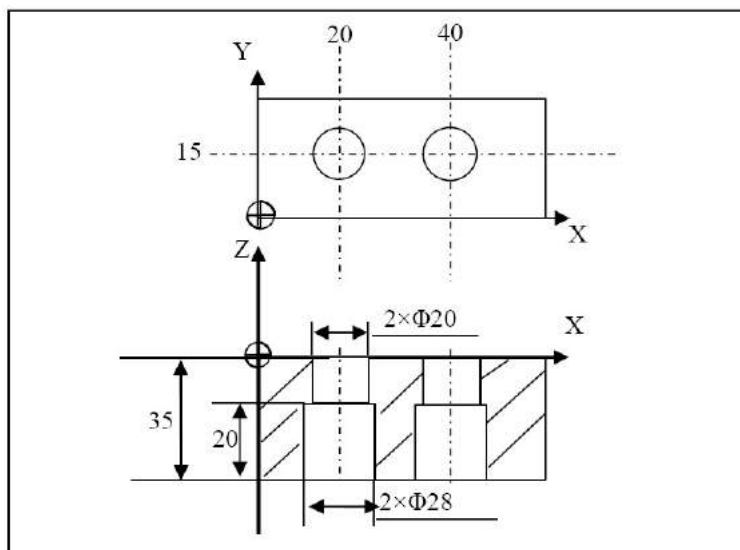
%3355

G54G0X0Y0Z50；Establish coordinate system, reach to safe starting point

M03 S600； Spindle starts

G54G0Y15

G98G87G90 X20I5R-40 P2000 Z-15 F120； Bore hole 1 after positioning

G91 X20； Bore hole 2 after positioning

G80 G91 G28 X0 Y0 Z0； Cancel G87 reverse cycle, return to reference point

M30； Program end

Note

(1) Cancel canned cycle before switching drilling axis.

(2) The drilling axis must be Z axis in G17 plane.

(3) If the moving position is zero on Z, this command will not be executed.

(4) Z point must be higher than R level, otherwise the alarm will be issued.

(5) G87 data is saved as the modal data, and the same data can be omitted.

(6) G87 can only use G98.

(7) Rotate spindle with the corresponding M code before using G87.

(8) Drilling is not performed in the blocks that not includes X, Y, Z, R or any other additional axis.

(9) The tool position offset is omitted in the drilling canned cycle mode.

(10) Do not specify the G code (G00-G03) of group 01 in the G87 block; otherwise, G87 will be cancelled.

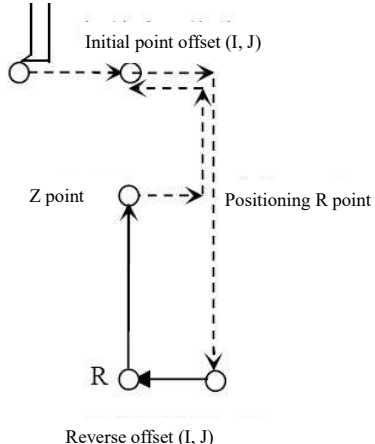## 15.12   Boring Cycle (Manual) (G88)

Function and Purpose

With this command, the position of the initial point B or reference point R is memorized before boring. When the boring tool automatically processes to the bottom of the hole, the machine stops running, and the working mode is manually changed to JOG. After moving radially in the opposite direction of the tool nose for a certain distance, the tool moves to above the height of point B or point R and avoids the workpiece. Then the working mode is restored to auto, the program is cycle restarted, and the tool position point returns to point B or point R. General milling machines can complete fine boring with this command without the spindle exact stop function.

Command Format

G98（G99）G88 X_ Y_ Z_ R_ P_ F_ L_ ；

Parameter	Meaning
X Y	The absolute position of the hole in absolute programming(G90), and the distance from the current position to the hole position in incremental programming(G91);
Z	The absolute position of hole bottom on Z in absolute programming (G90); the distance from hole bottom to R point in incremental programming (G91);
R	The absolute position of R point on Z in absolute programming (G90); the distance from R point to initial level in incremental programming (G91);
P	Dwell time at hole bottom (unit: ms)
F	Feedrate of boring;
L	Number of repetitions (generally for multi-hole processing, X and Y are incremental values)
G88（G98）	**G88（G99）**
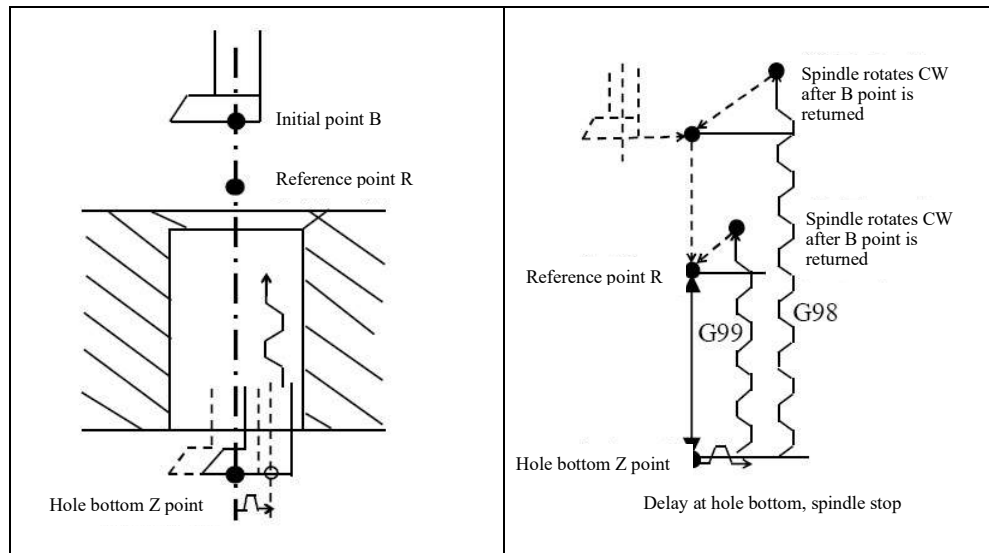
Description

● **Action**

(1) The tool position point moves to B point above hole center after positioning along X axis and Y axis;

(2) Move close to workpiece surface, reach to R point;

(3) Boring downward at F speed, reach to Z point at hole bottom;

(4) Dwell Pms at hole bottom (spindle keeps rotating);

(5) Spindle stops;

(6) In JOG mode, tool moves radially in the opposite direction of the tool nose for a certain distance, reaches to above the height of B point or R point, avoiding workpiece;

(7) Press cyclestart button in auot mode, the tool moves to R point (G99) or B point (G98) in rapid traverse;

(8) Spindle restores to rotation CW automatically;

● **Spindle rotation**

Rotate spindle with auxiliary function (M code) before using G88.

● **Tool length compensation**

When the tool length compensation (G43, G44, G49) is specified in drilling canned cycle, this compensation should be used at the time of positioning to R point.

● **Auxiliary function**

When G87 command and M code are specified in the same block, the M code is executed during the initial positioning. When the number of repetitions L is specified, the above actions will be

executed only for the first time, and M codes will not be executed after the second time.

Example
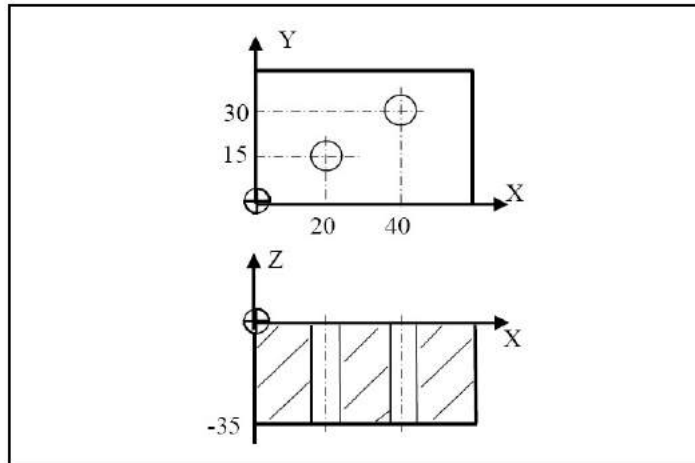
%3357
G54G01X0Y0Z50；Boring with a single-edge boring tool
M03 S600；Spindle starts
G98G88G90 X20Y15R5 P2000Z-40F100；After positioning, drill hole 1, return to R level,
　　　　　　　　　　　　　　　　　pause 2 seconds at hole bottom
G99 G91 X20 Y15；After positioning, drill hole 2, return to initial level
G80 G28 G91 X0 Y0 Z0；Cancel G88, return to reference point
M30；Program end

Note

(1) Cancel canned cycle before switching drilling axis.

(2) The drilling axis must be Z axis in G17 plane.

(3) If the moving amount is zero on Z, this command will not be executed.

(4) Z point must be lower than R level, otherwise the alarm will be issued.

(5) G88 data is saved as the modal data, and the same data can be omitted.

(6) If G99 is used, manually move the tool to the position higher than R point.

(7) If G98 is used, manually move the tool to the position higher than R point.

(8) Use corresponding M code to rotate spindle before using G88.

(9) Drilling is not performed in the blocks that not includes X, Y, Z, R or any other additional axis.

(10) The tool position offset is omitted in the drilling canned cycle mode.

(11) Do not specify the G code (G00-G03) of group 01 in the G88 block; otherwise, G88 will be cancelled.

15.13   Boring Cycle (G89)

Function and Purpose

The cycle is for boring.

Command Format

(G98/G99) G89 X_ Y_ Z_ R_ P_ F_ L；

Parameter	Meaning
X Y	The absolute position of the hole in absolute programming(G90), and the distance from the current position to the hole position in incremental programming(G91);
Z	The absolute position of hole bottom in absolute programming (G90); the distance from hole bottom to R point in incremental programming (G91);
R	The absolute position of R point in absolute programming (G90); the distance from R point to initial level in incremental programming (G91);
P	Dwell time at hole bottom (unit: ms)
F	Cutting feedrate
L	Number of repetitions (usually used for multi-hole processing, so X or Y should be an incremental value)

G89（G98）	G89（G99）

Description

● **Actions**

This cycle is similar as G86, and the dwell is performed at hole bottom.

● **Spindle rotation**

Rotate spindle with auxiliary function (M code) before using G89.

● **Auxiliary function**

When G89 command and M code are specified in the same block, the M code is executed during the initial positioning. When the number of repetitions L is specified, the above actions will be executed only for the first time, and M codes will not be executed after the second time.

Example
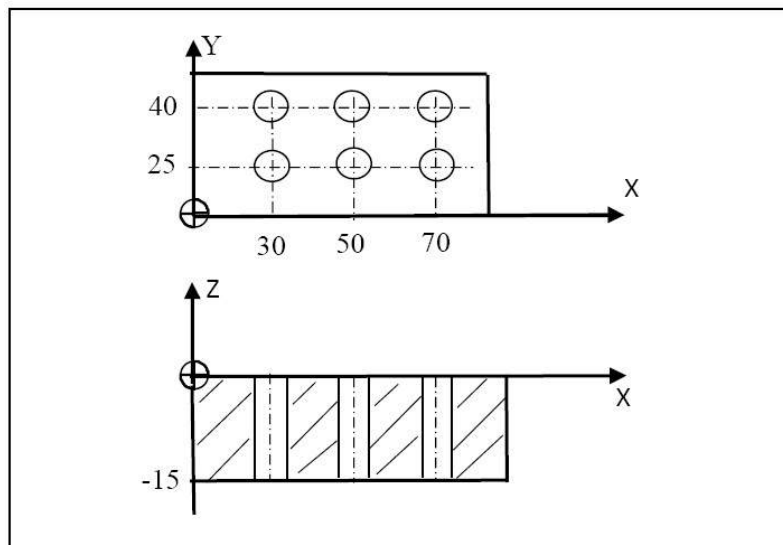
G54G01X0Y0Z50； Safe start point
M3 S1000； Spindle starts rotating
G90 G99 G89 X30 Y25 Z-15 R10 P1000 F120； After positioning, bore hole 1, pause 1 second at hole bottom, return to R point
X50Y25； Bore hole 2, pause 1 second at hole bottom, return to R point
X70Y25； Bore hole 3, pause 1 second at hole bottom, return to R point
X70Y40； Bore hole 4, pause 1 second at hole bottom, return to R point
X50Y40； Bore hole 5, pause 1 second at hole bottom, return to R point
G98 X30Y40； Bore hole 6, pause 1 second at hole bottom, return to initial level
G80 G28 G91 X0 Y0 Z0； Cancel boring, return to reference point
M30； Program end

Note

(1) Cancel canned cycle before switching drilling axis.

(2) The drilling axis must be Z axis in G17 plane.

(3) Z point must be lower than R level, otherwise the alarm will be issued.

(4) G89 data is saved as the modal data, and the same data can be omitted.

(5) G89 is similar as G86, and the dwell is performed at hole bottom.

(6) If the moving amount on Z is zero, G89 is not executed.

(7) Use corresponding M code to rotate spindle before using G89.

(8) Drilling is not performed in the blocks that not includes X, Y, Z, R or any other additional axis.

(9) The tool position offset is omitted in the drilling canned cycle mode.

(10) Do not specify the G code (G00-G03) of group 01 in the G89 block; otherwise, G86 will be cancelled.

15.14　Drilling Canned Cycle Cancel (G80)

Function and Purpose

This command is used to cancel drilling canned cycle

Command Format

G80；

Example

```
N0     M3 S1000； Spindle starts rotation
N10    G0 Z20； Position to initial level
N20    G90 G99 G81 X30. Y25. Z-2. R5.F120.；Position to hole 1, return to R point
N30    Y50；Position to hole 2, return to R point
N40    Y70；Position to hole 3, return to R point
N50    X100；Position to hole 3, return to R point
N60    Y80；Position to hole 4, return to R point
N70    G98 Y100；Position to hole 6, return to initial level
N70    G80 G28 G91 X0 Y0 Z0；Cancel canned cycle, return to reference point
N80    M5；Spindle stops rotating
N90    M30；Program end
```

Note

(1) After canceling all drilling canned cycles, resume the normal operation;

(2) Cancel R level and Z level;

(3) Other drilling parameter data are also cancelled; after canceling all drilling canned cycles, normal operation will be resumed;

16 Extended Canned Cycle（M）

16.1 Extended Canned Cycle (M)

Function and Purpose

The CNC controller can use extended fixed cycles to process some hole parts with simple contour, simplifying programming. The extended canned cycles include: engraving canned cycles, drilling extended cycles, circular pocket cycles, circumferential groove milling cycles, rectangular pocket cycles, end face milling cycles, rectangular boss cycles, circular boss cycles, etc.

16.2 Engraving Canned Cycle (G1025)

Function and Purpose

In actual processing, after the workpiece is processed, relevant information should be engraved on the surface of the workpiece, including the workpiece model, machine number, processing time and date, etc.

The engraving of the specified character information can be realized at the specified position with this engraving cycle, and the character size and direction can be adjusted.

Command Format

G1025C_ F_ E_ H_ V_ B_ Q_U_W_A_R_O_S_ P="_";

Parameter	Meaning
C	The tool lifting height relative to the engraving level. If this parameter is not defined, it is 10 by default. It is generally a positive value.
F	The composite feedrate when engraving. If this parameter is not defined, it defaults to 1000
X_ Y_	The absolute position of center point of the first character in XY direction in absolute programming (G90) The distance from center point of the first character to the starting point in XY direction in incremental programming (G91)
B	The absolute position of B point on the engraving level in absolute programming (G90) The distance from B point on the engraving level to the initial level in Z direction in incremental programming (G91)
Q	The feed depth relative to the engraving level. It is generally a positive value.

U	The engraved character size (the font height). If this parameter is not defined, the default is 5(mm). It is generally a positive value.
W	The distance between two characters. If this parameter is not defined, the default is 0.8 (mm).It ranges from 0.4 to 2.0. If this range is exceeded, an alarm will be issued.
A	Character rotation angle. 1: 0°; 2: 90°; 3: 180°; 4: 270°. It cannot be used simultaneously with R. If both A and R are not defined, the rotation angle defaults to 0°.
R	The rotation angle of the character. Range: -360°≤setting value≤360°. It cannot be used at the same time with A. If A and R are not defined, the rotation angle defaults to 0°.
O	Date and time type. 1: Month day-hour minute (Example 1214-1328); 2: Month day (Example 1215) 3: Year month day (Example 151214); 4: Year month day-hour minute (Example 151214-1328). If this parameter is not defined, the default is 1, that is, month day-hour minute.
P=""	To specify the engraving characters in the quotation marks, up to 60 characters. The currently specified characters are 0 to 9, A to Z, -, ., # (# indicates time and date). For example, P="AB-123" means the engravings of AB-123; P="N-#-18" means the engravings of N-1214-1328-18.
S	Spindle speed. If it is not defined, the modal speed value is valid.

Description

● **Actions**

After the positioning on X and Y axes, the tool moves to the center point coordinates (coordinates HV) of the first character at the G00 speed; meanwhile, the tool positions to the initial level in rapid traverse;

Move down (C height) on Z axis in rapid traverse,and position to the engraving level B;

Feed (L depth) with G01 for engraving corresponding characters;

After engraving, return to the initial level on Z axis in rapid traverse;

Move one character spacing, and perform the engraving of the second character according to the above actions until all the specified characters have been engraved;

● **Actions**

B point: engraving level
C line segment: lifting height C
L line segment: engraving depth L
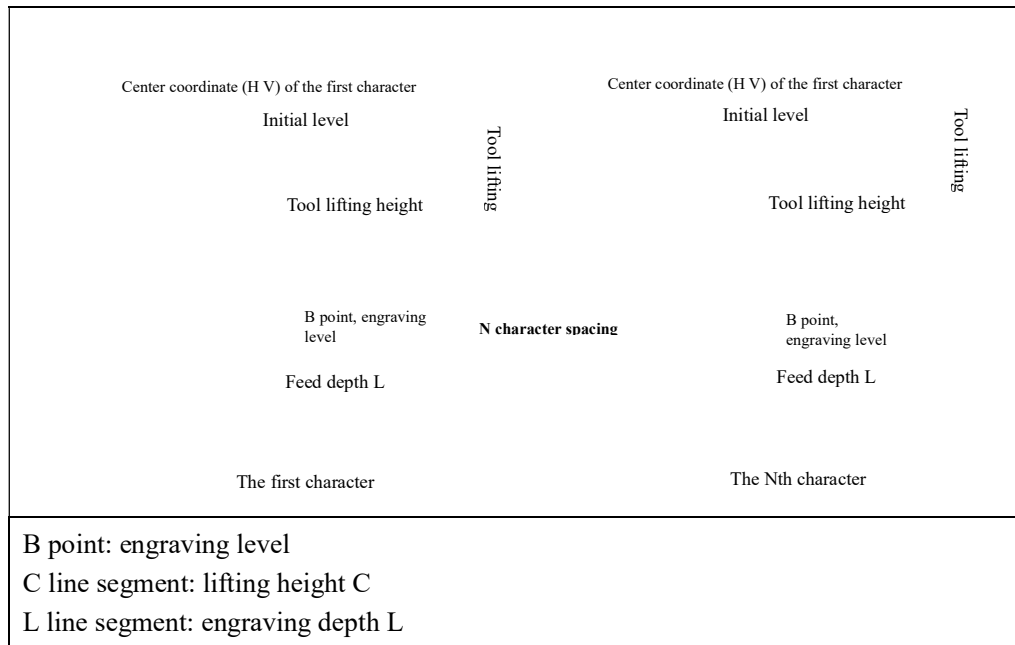
- **Tool length compensation**

When the tool length compensation (G43, G44, G49) is specified in engraving canned cycle, this compensation should be used at the time of positioning to B point.

- **Spindle rotation**

Use the auxiliary function (M code) to rotate spindle before G1025 is specified, and specify the spindle speed cyclically. If it is not defined, the modal speed value is valid.

- **Command use**

Engraving cycle program import

For the existing system needing to be upgraded to V1.25.00 and above, export USERDEF.CYC user-defined canned cycle through data management, copy the provided G1025 cycle program to USERDEF.CYC, and then reload USERDEF.CYC to the system. Then it can be used after power off and restart.

Alarms for parameters of engraving cycle

All parameters in the engraving cycle are non-modal, and each parameter needs to be defined when it is called multiple times.

The parameter value that must be defined

The center position of the first character X and Y must be defined, the engraving level B must be defined, and the engraving depth Q must be defined:

If the X parameter value is not defined, the alarm will prompt "Engraving cycle-X axis positioning coordinate X is not set"

If the Y parameter value is not defined, the alarm will prompt "Engraving cycle-Y axis positioning coordinate Y is not set".

If the value of B parameter is not defined, the alarm will prompt "Engraving cycle- engraving level B is not defined".

If the Q parameter value is not defined, the alarm will prompt "Engraving cycle-engraving depth Q is not defined".

Default W parameter value

If the character spacing W is defined, the default spacing between characters is 0.8mm, and the value of W ranges from 0.4 to 2.0. If the input number is not within the range, the alarm will prompt "Engraving cycle-character spacing exceeds the set range" .

A parameter value

Rotation angle A needs to be set to 1, 2, 3, 4. If it is set to other values, an alarm will prompt "Engraving cycle-Rotation angle setting error".

Note: Rotation angle parameters A and M cannot be set at the same time. If A and M are set at the same time, an alarm will be shown "Engraving cycle-A and M cannot be used at the same time to set the rotation angle"

M parameter value

The value of any rotation angle setting parameter M must be less than 360 and greater than -360. If it is set to a value larger than 360 or less than -360, an alarm will be displayed "Engraving cycle-rotation angle setting error".

Note: Rotation angle parameters A and M cannot be set at the same time. If A and M are set at the same time, an alarm will be shown "Engraving cycle-A and M cannot be used at the same time to set the rotation angle"

Q parameter value

The date and time type setting parameter O needs to be set to 1, 2, 3, and 4, and if it is set to other values, an alarm will be displayed "Engraving cycle-date and time type O setting error.

P parameter value

The engraving characters in P="_" can only be the provided characters, including 0 to 9, A to Z, -,., # (# indicates the time and date). If other characters are specified, such as *, an alarm will be displayed "Engraving cycle-the specified characters are not recognized".

Example

Example 1

%1234

G54G01X20Y20

G0Z30

G1025 A1 C10 F200 E800 H25.0 V50.0 B2.0 Q3.0 U2   O1 S1000 P="AD-#-28"

G90G0Z30

G1025 A1 C10 F200 E800 H25.0 V45.0 B2.0 Q3.0 U5   S1000 P="AD-#-28"

G90G0Z30

G1025 A1 C10 F200 E800 H25.0 V35.0 B2.0 Q3.0 U10   S1000 P="AD-#-28"

M30

The interpolation trajectory is shown in the figure below,
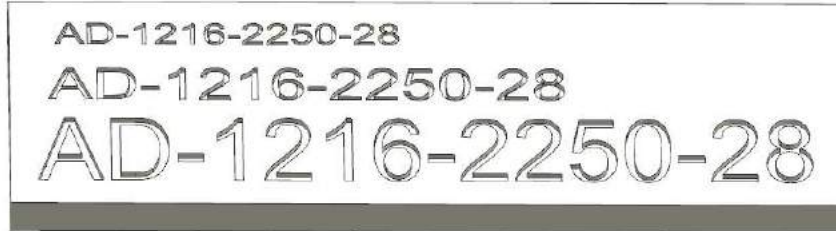
Example 2

%1234

G54G01X20Y20

G0Z30

G1025 A1 C10 F800 E800 H25.0 V45.0 B2.0 Q3.0 U5 O1 S1000 P="AD-#-28"

G90G0Z30

G1025 A2 C10 F800 E800 H20.0 V55.0 B2.0 Q3.0 U5 O1S1000 P="AD-#-28"

G90G0Z30

G1025 A3 C10 F800 E800 H15.0 V45.0 B2.0 Q3.0 U10 O1S1000 P="AD-#-28"

G90G0Z30

G1025 A4 C10 F800 E800 H20.0 V35.0 B2.0 Q3.0 U3 O1 S1000 P="AD-#-28"

M30

The interpolation trajectory is shown in the figure below,
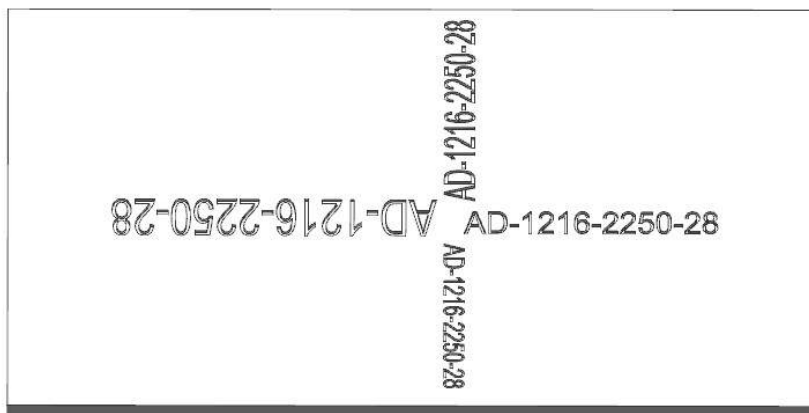
⚠ Note

The engraving cycle can realize the engraving of characters 0 to 9, A to Z, -, . , and   time and date (# indicates time and date). The character font format is Arial, with the default font height 5mm and the character width 3.09mm (character height*0.618). The default character spacing is 0.4mm, and the total number of characters for a single engraving does not exceed 60.

16.3   Drilling Type

16.3.1 Circumferential Hole Drilling Cycle (G70)

✎ Function and Purpose

On the circle centered on the coordinate (X,Y) with radius I, start from the point formed by the X axis and the angle J, divide the circle into N equal parts, and perform the drilling of N holes. Based on the values of Q and K, the G81 or G83 standard canned cycle is executed for the drilling of each hole. The position movement between holes is carried out in G00 mode.

G70 is a modal command, and the following command word is non-modal.

⚙ vCommand Format

(G98/ G99) G70 X_ Y_ Z_ R_ I_ J_ N_ [Q_K_P_] F_ L_；

Parameter	Meaning
X、Y	Coordinates of circle center
Z	Coordinates of hole bottom
R	Coordinates of R point in absolute programming; distance from intial point B to R point in incremental programming.
I	Radius of circle
J	Angle of initial drilling point, the positive value indicates the counterclockwise direction
N	Number of holes, the positive value indicates the drilling in counterclockwise direction, and the negative value indicates the drilling in clockwise direction;
Q	Feed depth, directed distance
K	Each time the tool is retracted and fed again, the distance from the last machined surface when the rapid traverse feed is converted to the cutting feed
P	Dwell time at hole bottom, unit: ms
F	Cutting feedrate

L	Number of cycles (can be omitted when L=1)

Description

- ● **Action**

(1) After positioning along X axis and Y axis with drilling angle, the tool quickly moves to point B above the hole center;

(2) Move close to the workpiece surface in rapid traverse, reach to R point;

(3) Drilling downward at F speed;

(4) Reach to hole bottom Z point;

(5) Dwell P ms at hole bottom (spindle keeps rotating);

(6) Retract upward to R point (G99) or B point (G98) in rapid traverse;

(7) Spindle keeps rotating;

(8) Program end.

- ● **Spindle rotation**

Rotate spindle with auxiliary function (M code) before using G70.

When continuously performing drilling of a short distance from the hole position and the initial level to the R point level, the spindle may not be able to rotate normally before entering the hole cutting. In this case, do not specify the number of repetitions L, and user needs to insert the G04-based pause before each drilling action to take some time out.

- ●**Tool length compensation**

When the tool length compensation (G43, G44, G49) is specified in drilling canned cycle, this compensation should be used at the time of positioning to R point.

- ● **Auxiliary function**

When G70 command and M code are specified in the same block, the M code is executed during the initial positioning. When the number of repetitions L is specified, the above actions will be executed only for the first time, and M codes will not be executed after the second time.

Example

Example 1: To drill four holes in the four axis directions in the XY plane. This cycle is executed twice and the G81 drilling is executed at the bottom of the hole.

G98 G70 X10 Y10 Z0 R20 I10 J0 N4 F200 L2

Example 2: To drill four holes at 45 degrees in the XY plane. This cycle is executed once and the G81 drilling is executed at the bottom of the hole.

G99 G70 X10 Y10 Z10 R50 I10 J45 N-4 F200

Example 3: To drill four holes at -45 degrees in the XY plane. This cycle is executed once and the G81 drilling is executed at the bottom of the hole.

G99 G70 X10 Y10 Z10 R50 I10 J-45 N-4 F200

Example 4: To drill four holes at -45 degrees in the XY plane. This cycle is executed once, the G81 drilling is executed at the bottom of the hole, and the Q value is invalid.

G99 G70 X10 Y10 Z10 R50 I10 J-45 N-4 Q-10 F200

Example 5: To drill four holes at -45 degrees in the XY plane. This cycle is executed once, and the G81 drilling is executed at the bottom of the hole.

G99 G70 X10Y10Z10R50 I10J-45N-4 Q0 F200

G99 G70 X10Y10Z10R50 I10J-45N-4 K0 F200

G99 G70 X10Y10Z10R50 I10J-45N-4 Q0K0 F200

Example 6: To drill four holes at -45 degrees in the XY plane. This cycle is executed once, and the G83 drilling is executed.

G99 G70 X10Y10Z10R50 I10J-45N-4 Q-10 K5 F200

Example: Use a Φ10 drill to process the hole shown in the figure:
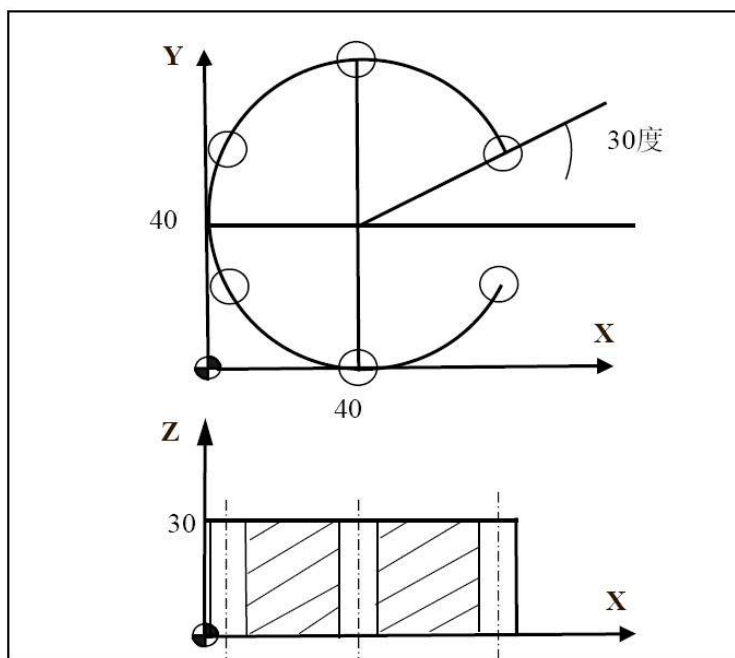
N10 G55 G00 X0 Y0 Z80
N20 G98G70G90X40Y40R35Z0I40J30N6P2000-10K5F100
N30 G90 G00 X0 Y0 Z80
N40 M30

⚠️ **Note**

(1) Cancel canned cycle before switching drilling axis.

(2) The drilling axis must be Z axis in G17 plane.

(3) If the moving position is zero on Z, this command will not be executed.

(4) Z point must be higher than R level, otherwise the alarm will be issued.

(5) I cannot be omitted; otherwise the alarm will be issued.

(6) G70 data is saved as the modal data, and the same data can be omitted.

(7) Use the corresponding M code to rotate spindle before using G70.

(8) Drilling is not performed in the blocks that not include X, Y, Z, R or any other additional axis.

(9) When Q is larger than zero or K is smaller than zero, the system reports errors; when Q or K is zero or is not defined, the G81 center drilling cycle is execute for each hole, and P is invalid; when values of Q and K are correct, the G83 deep hole drilling cycle is executed fro each hole, and P is valid.

(10) The tool position offset is omitted in the drilling canned cycle mode.

(11) Do not specify the G code (G00-G03) of group 01 in the G70 block; otherwise, G70 will be cancelled.

(12) When the angle is defaulted, the angel value =360/n.

16.3.2 Circular Hole Drilling Cycle (G71)

Function and Purpose

On the arc centered by the coordinates (X, Y) with radius I, start from the point formed by the X axis and the angle J, and drill N holes at an interval of O angle. The G81 or G83 standard canned cycle is executed based on the Q and K values for each hole. The movement of the positions between holes is carried out in G00 mode. G71 is a modal command, and the following command word is non-modal.

Command Format

(G98/G99) G71 X_ Y_ Z_ R_ I_ J_ N_ O_ [Q_ K_ P_] F_ L_ ;

Parameter	Meaning
Z	Coordinates of hole bottom
R	Coordinates of R point in absolute programming; distance from initial point B to R point in incremental programming
X，Y	Coordinates of acr center
I	Radius
J	Angle of initial drilling point, the positive value indicates the counterclockwise direction
O	Angle between holes. The positive value indicates the drilling in counterclockwise direction, and the negative value indicates the drilling in clockwise direction
N	Number of holes including the starting point

Q	Feed depth, directed distance
K	Each time the tool is retracted and fed again, the distance from the last machined surface when the rapid traverse feed is converted to the cutting feed
P	Dwell time at hole bottom, unit: ms
F	Cutting feedrate
L	Number of cycles (can be omitted when L=1)

📖 **Description**

● **Actions**

(1) After positioning along X axis and Y axis with the drilling angle, the tool position moves to point B above the hole center;

(2) Move close to the workpiece surface in rapid traverse, reach to R point;

(3) Drilling downward at F speed;

(4) Reach to the hole bottom Z point;

(5) Dwell P ms at hole bottom (spindle keeps rotating);

(6) Retract upward to R point (G99) or B point (G98) in rapid traverse;

(7) Spindle keeps rotating;

(8) Program end.

● **Spindle rotation**

Rotate spindle with auxiliary function (M code) before using G71.

When continuously performing drilling of a short distance from the hole position and the initial level to the R point level, the spindle may not be able to rotate normally before entering the hole cutting. In this case, do not specify the number of repetitions L, and user needs to insert G04-based pause before each drilling action to take some time out.

Do not need to consider the above situation for some machine tools.Please refer to the manual provided by the machine tool manufacturer.

● **Tool length compensation**

When the tool length compensation (G43, G44, G49) is specified in drilling canned cycle, this compensation should be used at the time of positioning to R point.

● **Auxiliary function**

When G71 command and M code are specified in the same block, the M code is executed during the initial positioning. When the number of repetitions L is specified, the above actions will be

executed only for the first time, and M codes will not be executed after the second time.

| **Example**

Use Φ10 drill bit to process the hole as shown,
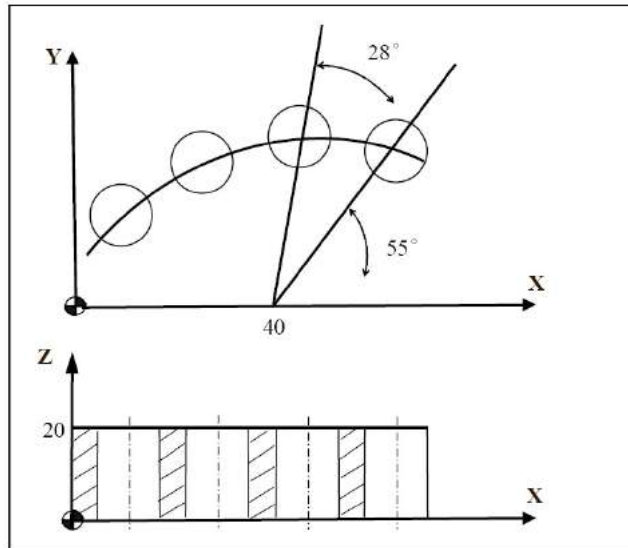
%3359
N10 G55 G00 X0 Y0 Z80
N20 G98G71G90X40Y0G90R25Z0I40J55O28N4P2000-10K5F100
N30 G90 G00 X0 Y0 Z80
N40 M30

| **Note**

(1) When Q is larger than zero or K is smaller than zero, the system reports errors; when Q or K is zero or is not defined, the G81 center drilling cycle is executed for each hole, and P is invalid; when values of Q and K are correct, the G83 deep hole drilling cycle is executed fro each hole, and P is valid.

(2) The total arc angle N×O cannot be greater than or equal to 360 degrees, otherwise it will not be executed;

(3) Cancel canned cycle before switching drilling axis.

(4) The drilling axis must be Z axis in G17 plane.

(5) If the moving amount is zero on Z, this command will not be executed.

(6) Z point must be higher than R level, otherwise the alarm will be issued.

(7) The values of I and O cannot be omitted, otherwise the alarm will be issued.

(8) G71 data is saved as the modal data, and the same data can be omitted.

(9) Use corresponding M code to rotate spindle before using G71.

(10) Drilling is not performed in the blocks that not include X, Y, Z, R or any other additional axis.

(11) The tool position offset is omitted in the drilling canned cycle mode.

(12) Do not specify the G code (G00-G03) of group 01 in the G71 block; otherwise, G71 will be cancelled.

16.3.3 Straight Line Hole Cycle (G78)

Function and Purpose

The cycle starts with the position (X, Y), and drill N holes in the direction formed by the X axis and J angle with the interval I. The G81 or G83 canned cycle is performed for each hole based on the value of Q and K. The movement between the holes is carried out in G00 mode. G78 is a modal command, and the following command word is non-modal.

Command Format

(G98/G99)G78 X_Y_Z_R_I_J_N_[Q_K_P]_F_L_;

Parameter	Meaning
X Y	Coordinates of the first hole
Z	Coordinates of hole bottom. Coordinates of R point in absolute programming; distance from intial point B to R point in incremental programming
I	Distance between holes
J	The starting angle formed by the straight line and the positive direction of X-axis, the positive value indicates the counterclockwise direction;
N	Number of holes including the starting point
Q	Feed depth, directed distance
K	Each time the tool is retracted and fed again, the distance from the last machined surface when the rapid traverse feed is converted to the cutting feed
P	Dwell time at hole bottom, unit: ms

Description

● **Action**

（1）After positioning along X axis and Y axis with drilling angle, the tool position quickly moves to point B above the hole center;

（2）Move close to the workpiece surface in rapid traverse, reach to R point;

（3）Drill downward at F speed;

（4）Reach to hole bottom Z point;

（5）Dwell P ms at hole bottom (spindle keeps rotating);

（6）Retract upward to R point (G99) or B point (G98) in rapid traverse;

（7）Spindle keeps rotating;

（8）Program end with M30.

- **Spindle rotation**

Rotate spindle with auxiliary function (M code) before using G78.

When continuously performing drilling of a short distance from the hole position and the initial level to the R point level, the spindle may not be able to rotate normally before entering the hole cutting action. In this case, do not specify the number of repetitions L, and user needs to insert GO4-based pause before each drilling action to take some time out.

Do not need to consider the above situation for some machine tools. Please refer to the manual provided by the machine tool manufacturer.

- **Tool length compensation**

When the tool length compensation (G43, G44, G49) is specified in drilling canned cycle, this compensation should be used at the time of positioning to R point.

- **Auxiliary function**

When G78 command and M code are specified in the same block, the M code is executed during the initial positioning. When the number of repetitions L is specified, the above actions will be executed only for the first time, and M codes will not be executed after the second time.

Example

Example: Use a Φ10 drill to process the hole shown in the figure.
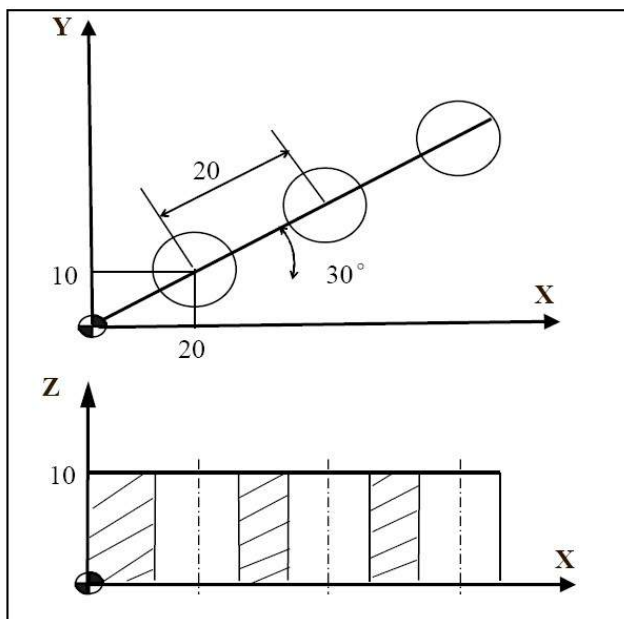
%3360
N10 G55 G00 X0 Y0 Z80
N20 G98G78G90X20Y10G90R15Z0I20J30N3P2000-10K5F100
N30 G90 G00 X0 Y0 Z80
N40 M30

⚠ **Note**

(1) Cancel canned cycle before switching drilling axis.

(2) The drilling axis must be Z axis in G17 plane.

(3) If the moving amount is zero on Z, this command will not be executed.

(4) The Z point must be higher than R level, otherwise the alarm will be issued.

(5) The I value cannot be omitted, otherwise the alarm will be issued.

(6) G78 data is saved as the modal data, and the same data can be omitted.

(7) Use corresponding M code to rotate spindle before using G78.

(8) Drilling is not performed in the blocks that not include X, Y, Z, R or any other additional axis.

(9) An error is reported when Q is greater than zero or K is smaller than zero; an error is reported when the feed distance Q is smaller than the retract distance K; when Q or K is zero or is not defined, the G81 center drilling cycle is executed for each hole, at this time P is invalid; when the values of Q and K are both correct, G83 deep hole cycle is executed for each hole, and P is valid at this time.

(10) The tool position offset is omitted in the drilling canned cycle mode.

(11) Do not specify the G code (G00-G03) of group 01 in the G78 block; otherwise, G78 will be cancelled.

16.3.4 Chess Type Hole Cycle (G79)

Function and Purpose

The cycle starts with the position (X, Y), drills N holes in the direction parallel to X axis with the interval I, and uses the interval J for drilling in the X axis direction, a total of O cycles. G81 or G83 canned cycle is performed based on the values of Q and K for each hole. The movement between holes is performed in G00 mode. G79 is modal, and the following command word is non-modal.

Command Format

(G98/G99)G79 X_Y_Z_R_I_J_N_[Q_K_P]_F_L_

Para meter	Meaning
X，Y	Coordinates of the first hole
Z	Coordinates of hole bottom
R	Coordinates of R point in absolute programming; distance from initial point B to R point in incremental programming
I	Distance between holes on X. The positive value indicates the drilling in the positive direction of X axis, and the negative value indicates the drilling in the negative direction of X axis.
N	Number of holes including the start point in the direction of X
J	Distance between holes on Y. The positive value indicates the drilling in the positive direction of Y axis, and the negative value indicates the drilling in the negative direction of Y axis.
O	Number of holes including the start point in the direction of Y
Q	Feed depth, directed distance
K	Each time the tool is retracted and fed again, the distance from the last machined surface when the rapid traverse feed is converted to the cutting feed
P	Dwell time at hole bottom, unit: ms

Description

● **Action**

(1) After positioning along X axis and Y axis with the drilling angle, the tool position quickly

moves to point B above the hole center;

(2) Move close to the workpiece surface in rapid traverse, reach to R point;

(3) Drill downward at F speed;

(4) Reach to hole bottom Z point;

(5) Dwell P ms at hole bottom (spindle keeps rotating);

(6) Retract upward to R point (G99) or B point (G98) in rapid traverse;

(7) Spindle keeps rotating;

(8) Program end with M30.

●**Spindle rotation**

Rotate spindle with auxiliary function (M code) before using G79.

When continuously performing drilling of a short distance from the hole position and the initial level to the R point level, the spindle may not be able to rotate normally before entering the hole cutting action. In this case, do not specify the number of repetitions N, and user needs to insert GO4-based pause before each drilling action to take some time out.

It is unnecessary to consider the above situation for some machine tools.Please refer to the manual provided by the machine tool manufacturer.

● **Tool length compensation**

When the tool length compensation (G43, G44, G49) is specified in drilling canned cycle, this compensation should be used at the time of positioning to R point.

● **Auxiliary function**

When G79 command and M code are specified in the same block, the M code is executed during the initial positioning. When the number of repetitions N is specified, the above actions will be executed only for the first time, and M codes will not be executed after the second time.

Example

Example: Use a Φ10 drill to process the hole shown in the figure.
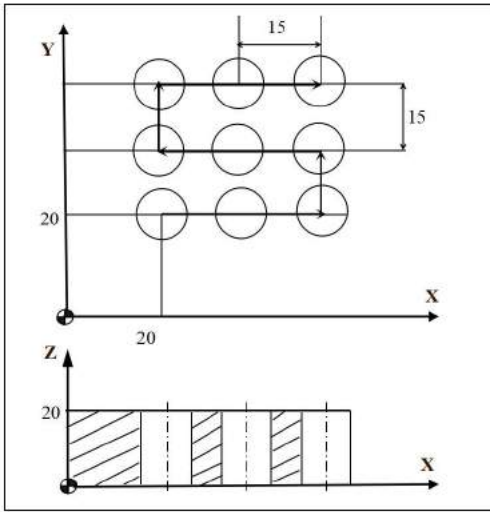
%3361
N10 G55 G00 X0 Y0 Z80
N20 G98G79G90X20Y20G90R25Z0I15N3J15O3P2000-10K5F100
N30 G90 G00 X0 Y0 Z80
N40 M30

⚠ **Note**

(1) Cancel canned cycle before switching drilling axis.

(2) The drilling axis must be Z axis in G17 plane.

(3) If the moving amount is zero on Z, this command will not be executed.

(4) Z point must be higher than R level, otherwise the alarm will be issued.

(5) The values of I, O, J cannot be ignored, otherwise the alarm will be issued.

(6) G79 data is saved as the modal data, and the same data can be omitted.

(7) Use corresponding M code to rotate spindle before using G79.

(8) Drilling is not performed in the blocks that not include X, Y, Z, R or any other additional axis.

(9) An error is reported when Q is greater than zero or K is smaller than zero; an error is reported when the feed distance Q is less than the retract distance K; when Q or K is zero or is not defined, the G81 center drilling cycle is executed for each hole, at this time P is invalid; when the values of Q and K are both correct, G83 deep hole cycle is executed for each hole, and P is valid at this time.

(10) The tool position offset is omitted in the drilling canned cycle mode.

(11) Do not specify the G code (G00-G03) of group 01 in the G79 block; otherwise, G79 will be cancelled.

16.4   Milling Cycle

16.4.1 Circular Groove Milling Cycle (Type 1) (G181)

    Function and Purpose

This cycle can be used to machine grooves arranged in an arc, and the groove width is determined by the tool diameter.
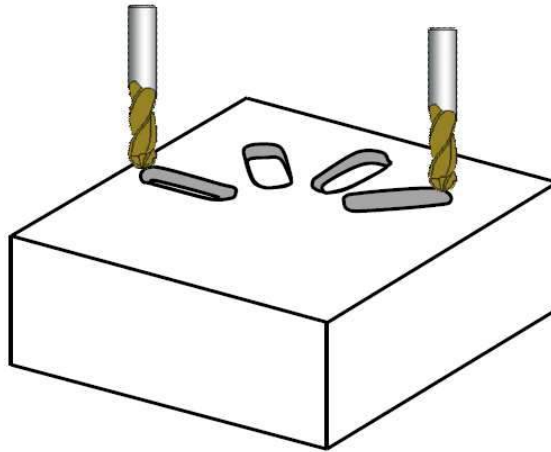
    Command Format

(G98/G99) G181 R_ Z_ N_ K_ X_ Y_ I_ A_ B_ F_ Q_ V_；

Parameter	Meaning
R	Coordinates of R point in absolute programming; distance from intial point B to R point in incremental programming
Z	Coordinates of groove bottom in absolute programming; incremental value from groove bottom to reference point R in incremental programming
N	Number of grooves（can be omitted when L=1）
K	Length of groove
X	Center position of the arc formed by grooves. Coordinate of the first axis in the current plane in absolute programming; incremental value relative to the starting point in incremental programming
Y	Center position of the arc formed by grooves. Coordinate of the second axis in the current plane in absolute programming; incremental value relative to the starting point in incremental programming
I	Radius of the arc formed by grooves
A	Starting angle (-180 to 180 degrees. It is a positive value for counterclockwise, and negative value for clockwise, and it can be omitted when A=0)

B	Incremental angle (can be omitted when B=360/N; The negative value indicates the milling of grooves in counterclockwise direction, and the positive value indicates the milling of grooves in clockwise direction)
F	Milling speed
Q	Max. feed depth (can be omitted when Q=groove depth, one-time cutting to the end)
V	Tool radius

📖 **Description**

Parameter diagram
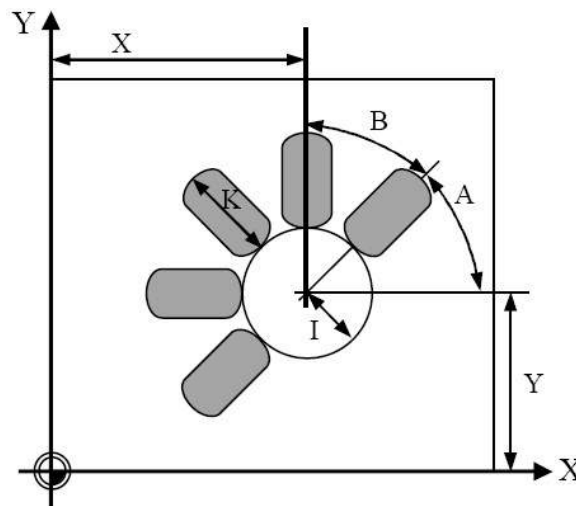
Milling steps

(1) Select the initial position to enter the cycle. This point can be any position, but it must be ensured that every groove can be executed from this point without contour collision;

(2) From the initial position to the reference position R above the near end of the first groove, the near end refers to the end close to the center of the arc, and the far end is the opposite. The groove specified by the starting angle parameter A is the first machining groove;

(3) Feed down to the specified depth at the milling speed, and then mill the groove back and forth to the groove bottom. Each depth feed is performed at the end of the groove;

(4) The application axis (usually the Z axis) retracts to the reference position R. The tool moves to the end of the next groove along the selected shortest path, and mills back and forth to the bottom;

(5) After the last groove is machined, retract the tool to the initial position B or reference position R of the cycle according to the current G98 or G99 modal, and the cycle ends.

N=3
A=180
B=-90

Milling in counterclockwise direction
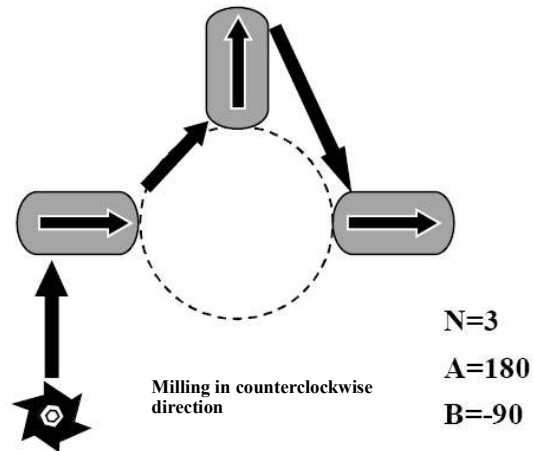
Example

Process 4 rectangular grooves as shown in the figure below, the groove length is 30mm, the groove depth is 23mm, and the cutting depth is 6mm.
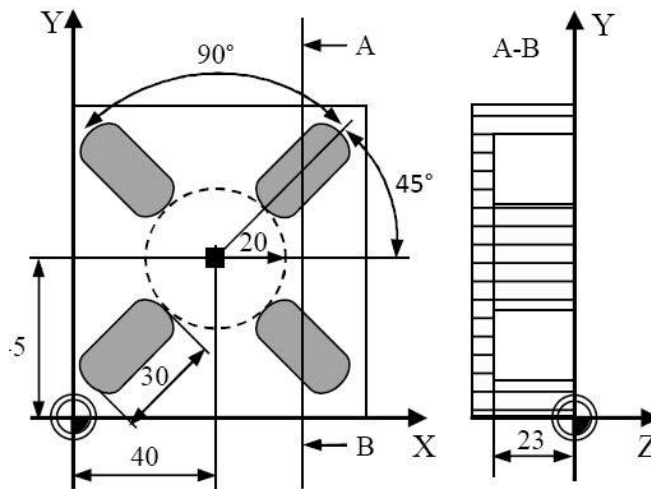
%0526
N10   G54 X0 Y0 Z5
N20   G17 G90
N30   T10
N40   M06
N50   M03 S600
N60   G181 R0 Z-23 N4 K30 X40 Y45 I20 A45 B90 F100 Q6 V5
N70   M30

Note

(1) The number of grooves N is input as a non-negative integer, the negative sign is ignored in the cycle, and the non-integer will be rounded.

(2) The maximum feed depth is specified by Q. If the groove depth cannot be divisible by Q, the feed depth of the last cut will be less than Q.

(3) The milling direction between the grooves is related to the sign of B. If B is positive, it will be processed counterclockwise from the first groove to the last groove. If B is negative, the groove will be processed clockwise. If B is not specified, B=360/N is cycled, and the milling of grooves is performed counterclockwise.

(4) The input of K, I, and Q are non-negative values. If the input is negative, the negative sign will be ignored in the cycle;

(5) Rotate the spindle before entering cycle.

16.4.2 Circular Groove Milling Cycle (Type 2) (G182)

Function and Purpose

This cycle can process the grooves arranged in annular. The longitudinal axes of these grooves are arranged radially. This cycle is different from G181. The groove width can be specified by parameters instead of the tool diameter.

At the same time, this cycle can specify roughing or finishing.
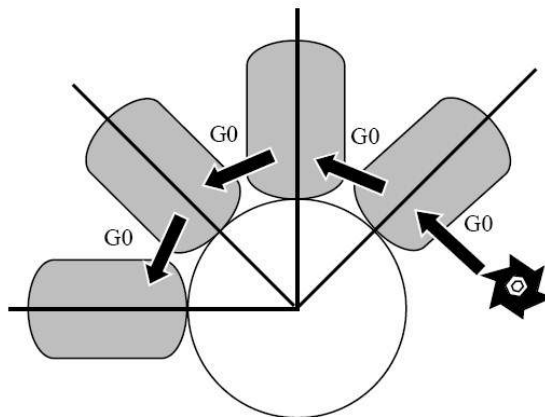
Command Format

(G98/G99)G182 R_Z_N_K_W_X_Y_I_A_B_F_Q_ E_O_H_U_P_C_D_V_ ;

Parameter	Meaning
R	Reference point R coordinates in absolute programming; incremental value of

	reference point R relative to the initial level.
Z	Coordinate value of groove bottom in absolute programming; incremental value of groove bottom relative to the reference point R in incremental programming.
N	Number of grooves (can be omitted when N=1)
K	Length of groove
W	Width of groove (can be omitted when W=tool diameter)
X	Center position of the arc formed by grooves. Coordinate of the first axis in the current plane in absolute programming; incremental value relative to the starting point in incremental programming
Y	Center position of the arc formed by grooves. Coordinate of the second axis in the current plane in absolute programming; incremental value relative to the starting point in incremental programming
I	Radius of the arc formed by grooves
A	Starting angle (-180 to 180 degrees. It is a positive value for counterclockwise, and negative value for clockwise, and it can be omitted when A=0)
B	Incremental angle (can be omitted when B=360/N; The negative value indicates the milling of grooves in counterclockwise direction, and the positive value indicates the milling of grooves in clockwise direction)
F	Milling speed in roughing
Q	Max. feed depth in roughing(can be omitted when Q=groove depth- finishing allowance of groove bottom)
E	Finishing allowance of groove edge (can be omitted when E=0)
O	Finishing allowance of groove bottom (can be omitted when O=0)
H	Max. feed depth in finishing (can be omitted when H=Q)
U	Feedrate in finishing (can be omitted when U=F)
P	Spindle speed in finishing (can be omitted when P=spindle speed or default speed before entering cycle)
C	Milling direction for the groove (can be omitted when C=3) 0: milling in the same direction; 1: milling in the reverse direction; 2: milling in the G02 direction; 3: milling in the G03 direction
D	Machining type (can be omitted when D=1) 1: roughing; 2: finishing
V	Tool radius

📖 **Description**

Graphical representation
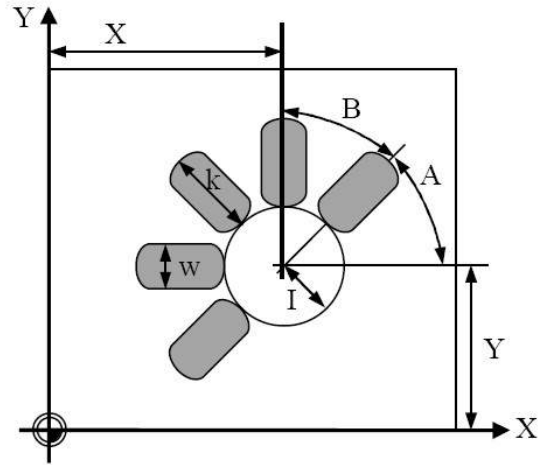
Milling direction

The spindle must be specified to rotate before entering the cycle. The cycle calculates a reasonable milling direction based on the spindle rotation direction M3/4 before the cycle is entered and the milling direction set by the user. The selection of the milling direction is shown in the following table:

Milling direction (cycle parameter C)	Specify M03 or M04 before entering cycle	
	M03 spindle rotation CW	**M04 spindle rotation CCW**
0：milling in the same direction	G03	G02
1：milling in the reverse direction	G02	G03
2：G02 direction	G02	G02
3：G03 direction	G03	G03
Omitted	G03	G03

Example

To machine 4 grooves. Groove length 30mm, groove width 15mm, groove depth 23mm, finishing allowance 0.5mm, milling direction G02, feed depth in roughing 6mm, tool radius 5mm.
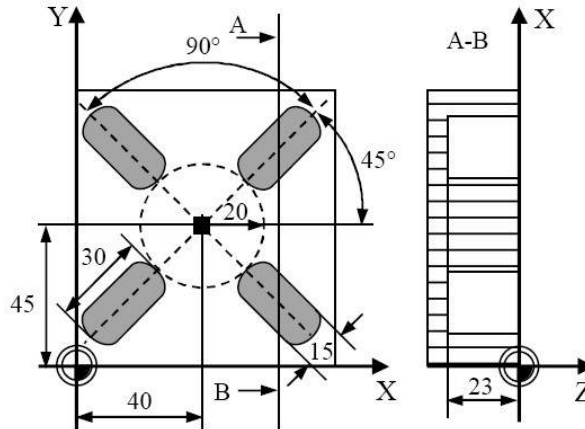
%0527

N10 G54 G17 G90
N20 T10
N30 M06
N40 M03 S600
N50 G182R5Z-23N4K30X40Y45W15I20A45B90F100Q6E0.5O0.5C2V5
N60 M30

Note

(1) The tool radius cannot exceed the specified groove width W, otherwise an alarm will be generated;

(2) The number of grooves N is input as a non-negative integer, the cycle will ignore the negative sign, and round off the non-integer;

(3) Q and H both refer to the maximum feed depth. Note that when the groove depth cannot be divisible by feed depth Q or H, the last cut will be less than Q or H;

(4) The milling direction C specifies the milling direction of a single groove, and the angle increment B specifies the milling direction between the grooves. If B is positive, the processing will be performed counterclockwise from the first groove to the last groove; and if B is negative, it will be clockwise. If B is not specified, the cycle will process B=360/N, and the grooves will be milled counterclockwise;

(5) K, W, I, E, O, Q, H are input as non-negative numbers. If the input is negative, the negative sign will be ignored in the cycle;

(6) The spindle must be rotated before entering the cycle. In addition, the finishing allowance E of the specified groove edge cannot exceed the groove width W/2, and the finishing allowance O of the groove bottom cannot exceed the groove depth, otherwise an alarm will be generated. For the alarm information, please refer to chapter 16.4.9.

16.4.3 Circumferential Groove Milling Cycle (G183)

Function and Purpose

This cycle can process the grooves distributed on the circle, and can specify rough machining, finishing or comprehensive machining.
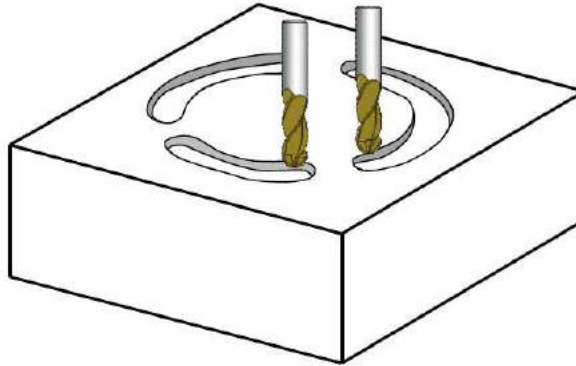
Command Format

(G98/G99)G183R_Z_N_K_W_X_Y_I_A_B_F_Q_E_O_H_U_P_C_D_V_；

Parameter	Meaning
R	Coordinates of R point in absolute programming; distance from reference point R to initial level in incremental programming
Z	Coordinate value of groove bottom in absolute programming; incremental value of groove bottom relative to reference point R in incremental programming.
N	Number of grooves （can be omitted when L=1）
K	Angle of groove length (0 to 360 degrees, unit: degree)
W	Groove width (can be omitted when W=tool diameter)
X	Center position of the arc formed by grooves. Coordinate of the first axis in the current plane in absolute programming; incremental value relative to the starting point in incremental programming.
Y	Center position of the arc formed by grooves. Coordinate of the second axis in the current plane in absolute programming; incremental value relative to the starting point in incremental programming.
I	Radius of circle arranged by grooves
A	Starting angle (-180 to 180 degress, positive for counterclockwise, and negative for clockwise. It

	can be omitted when A=0)
B	Incremental angle (can be omitted when B=360/N; The negative value indicates the milling of grooves in counterclockwise direction, and the positive value indicates the milling of grooves in clockwise direction)
F	Milling speed in roughing
Q	Max. feed depth in roughing (can be omitted when Q=groove depth-finishing allowance of groove bottom)
E	Finishing allowance of groove edge (can be omitted when E=0)
O	Finishing allowance of groove bottom (can be omitted when O=0)
U	Feedrate in finishing (can be omitted when U=F)
P	Spindle speed in finishing (can be omitted when P=spindle speed or default speed before entering cycle)
C	Milling direction for grooving (can be omitted when C=3) 0：milling in the same direction; 1: milling in the reverse direction; 2: milling in G02 direction; 3: milling in G03 direction
D	Machining type (can be omitted when D=1) 1: roughing; 2: finishing
V	Tool radius

Description

Graphical representation
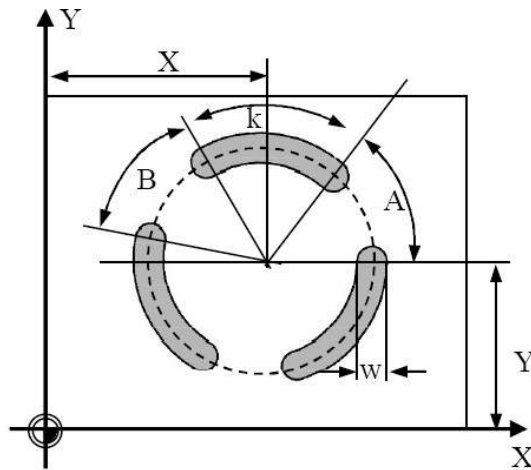
Milling direction

The spindle must be specified to rotate before entering the cycle. A reasonable milling direction is calculated based on the spindle rotation direction M3/4 and the milling direction set by the user before entering the cycle. The selection of the milling direction is shown in the following table:

Milling direction (cycle	Specify M03/M04 before entering cycle

parameter C)	M03 spindle rotation CW	M04 spindle rotation CCW
milling in the same direction	G03	G02
milling in the reverse direction	G02	G03
2：G02 direction	G02	G02
3：G03 direction	G03(1)	G03
省略 Omitted	G03	G03

Steps

(1) When cycle runs, use G00 to reach the reference level R;

(2) Milling the current groove back and forth from inside to outside, the milling steps are similar to G182;

(3) After finishing machining a groove, the tool retracts to the reference level and moves to the next groove;

(4) After processing all grooves, retract the tool with G98/G99 to complete the cycle.
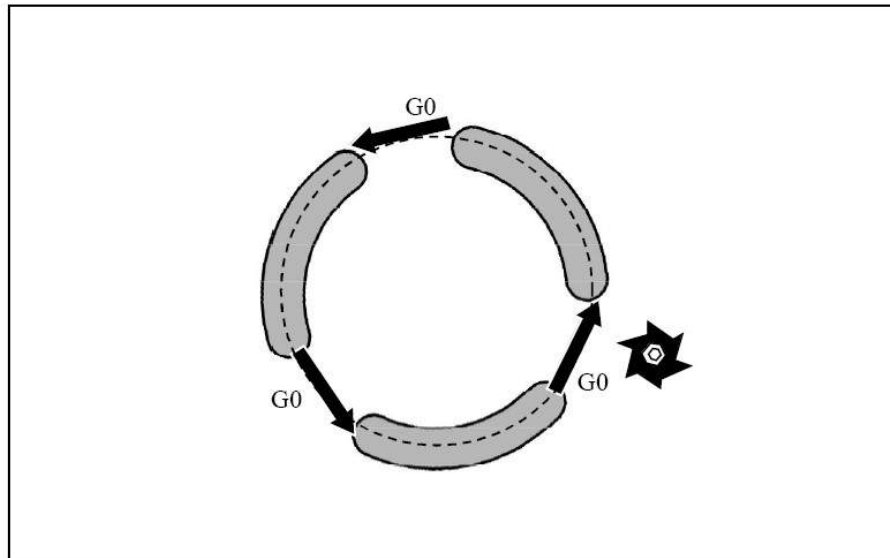
Example

Machining three grooves on a circle with a center of (X60, Y60) and a radius of 42mm in the XY plane. The groove size: width is 15mm, groove length angle is 70 degrees, groove depth is 23mm, starting angle is 0 degrees, incremental angle is 120 degrees, the finishing allowance on the groove contour is 0.5mm, and the depth of each feed is 6mm. The same speed and feed rate are used for finishing and roughing. The finishing is completed in one cut, and the tool radius is 5mm.
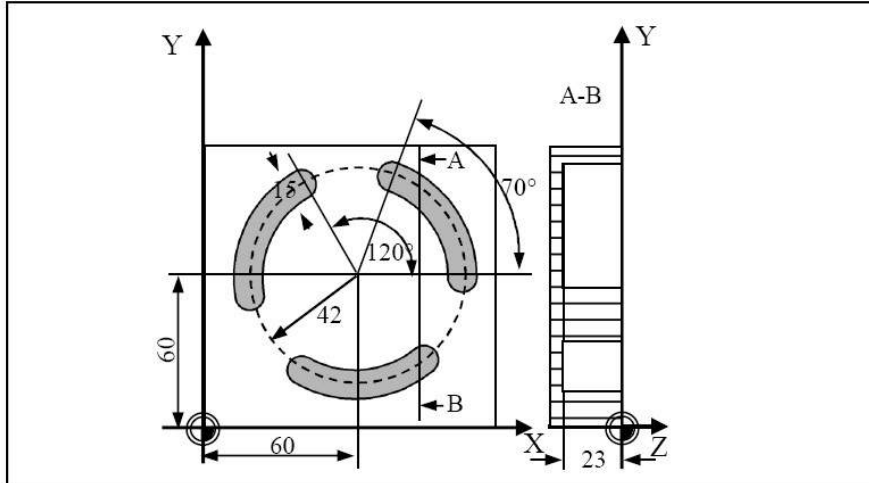
%0528
N10 G54 G17 G90
N20 T10
N30 M06
N40 M03 S600
N50 G00 X60 Y60 Z5
N60G183R2Z-23N3K70W15X60Y60I42A70B120F100Q6E0.5O0.5V5
N70 M30

16.4.4 Rectangular Pocket Milling Cycle (G184)

 Function and Purpose

This cycle is used for rough machining and finishing of rectangular pockets with circular corners.
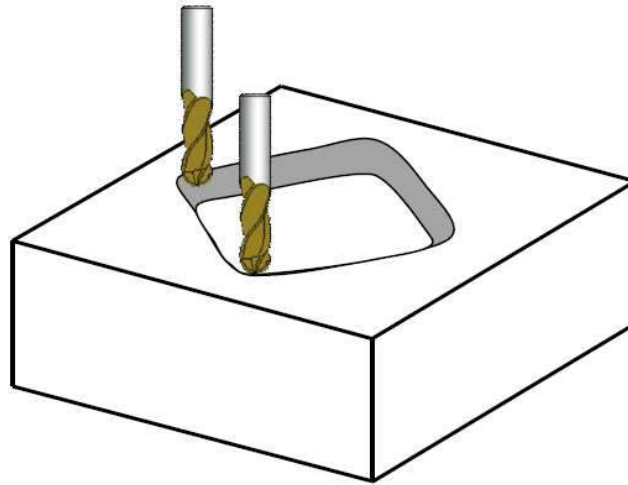
 Command Format

(G98/G99)G184R_Z_K_W_X_Y_I_A_F_ Q_E_O_H_U_P_C_D_V_；

Parameter	Meaning
R	Coordinates of R point in absolute programming; distance from initial level to R point in incremental programming
Z	Coordinate of pocket bottom in absolute programming; incremental value of pocket bottom relative to reference point R in incremental programming.
K	Pocket length
W	Pocket width
X	Center of pocket. Coordinate of the first axis in the current plane in absolute programming; incremental value relative to the starting point in incremental programming.
Y	Center of pocket. Coordinate of the second axis in the current plane in absolute programming; incremental value relative to the starting point in incremental programming.
I	Radius of circular corners of rectangular pocket (can be omitted or specified as 0 when I=W/2)
A	The angle between the long side of the rectangular groove and the positive direction of the first axis in the plane (can be omitted when A=0)
F	Milling speed in roughing

Q	Max. feed depth in roughing (can be omitted when Q=groove depth-finishing allowance of groove bottom)
E	Finishing allowance of groove edge (can be omitted when E=0)
O	Finishing allowance of groove bottom (can be omitted when O=0)
H	Max. feed depth in finishing (can be omitted when H=Q)
U	Feedrate in finishing (can be omitted when U=F)
P	Spindle speed in finishing (can be omitted when P=spindle speed or default speed before entering cycle)
C	Milling direction (can be omitted when C=3) 0：milling in the same direction; 1: milling in the reverse direction; 2: milling in G02 direction; 3: milling in G03 direction
D	Machining type (can be omitted when D=1) 1: roughing; 2: finishing
V	Tool radius

Description

Graphical representation


Milling direction

The spindle must be specified to rotate before entering the cycle. A reasonable milling direction is calculated based on the spindle rotation direction M3/4 and the milling direction set by the user before entering the cycle. The selection of the milling direction is shown in the following table:

Milling direction (cycle parameter C)	M03/M04 Specify M03/M04 before entering cycle	
	M03 spindle rotation CW	M04 spindle rotation CCW
0：milling in the same direction	G03	G02
1：milling in the reverse direction	G02	G03
2：G02 direction	G02	G02
3：G03 direction	G03	G03

Omitted	G03	G03

Steps

(1) Position to the starting position of the cycle. This point can be any position, but it must be ensured that the workpiece can be positioned from this point without collision;

Roughing (D=1)

(2) The midpoint of the edge of the wide side (finishing allowance of the pocket edge is left)is positioned with G00, and the depth is fed with a feed amount Q. The pocket surface is milled from the outside to the inside in the milling direction defined by C, and the tool returns to the same cutting point, feed from the groove surface again to the finishing allowance at the bottom of the groove.

Finishing（D=2）

(3) The midpoint of the edge of the wide side (finishing allowance of the pocket edge is left)is positioned with G00, and the depth is fed with an amount H. The pocket surface is milled from the outside to the inside in the milling direction defined by C, and the tool returns to the same cutting point, feed from the groove surface again to the finishing allowance at the bottom of the groove.

(4) After machining, move the tool to the initial level or reference level with G98/G99, and the cycle ends.

Example

Machine a pocket in the G17 plane with a length of 60mm, a width of 40mm, a corner radius of 8mm, and a depth of 17.5mm. The pocket forms a 0 degree angle with X axis. The finishing allowance of edge is 0.75mm, and the finishing allowance of bottom is 0.2mm. The pocket center is X60Y40, the feed depth is 4mm, and the tool radius is 5mm. Only rough machining is performed, as shown below.
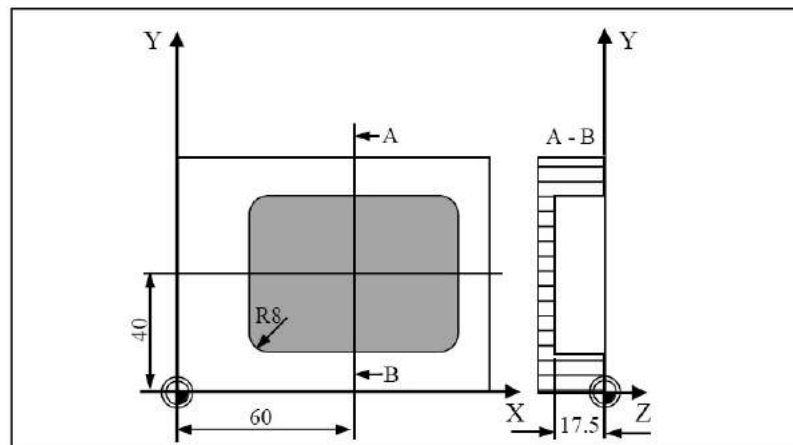
%0526

N10 G54 G90 G17
N20 T20
N30 M06
N40 M04 S600
N50 G00 X60 Y40 Z5
N60 G98G184R5Z-17.5K60W40X60Y40I8F120Q4E0.75O0.2D1V5
N70 M30

⚠️ **Note**

(1) This cycle requires the use of end mills.

(2) Q and H respectively define the maximum feed depth during roughing and finishing. Note that when the machining allowance cannot be divisible by Q or H, the last cut will be smaller than Q and H;

(3) N, K, W, I, E, O, Q, H are input as non-negative numbers. If the input is negative, the negative sign will be ignored in the cycle;

(4) Please refer to Section 16.4.9 for alarms generated by this cycle;

(5) If the input pocket width is greater than the pocket length, the cycle will automatically exchange them to the expected position;

(6) The C specified the milling direction such as G182.

(7) The spindle rotation must be specified before entering cycle.

16.4.5 Circular Pocket Milling Cycle (G185)

Function and Purpose

This cycle is used for processing circular pocket. Finishing and roughing are supported.
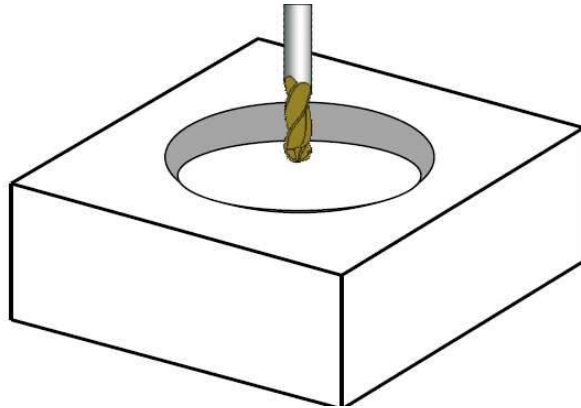
Command Format

(G98/G99)G185R_Z_X_Y_I_F_Q_E_O_H_U_P_C_D_V_;

Parameter	Meaning
R	Coordinates of R point in absolute programming; distance from reference point B to R point in incremental programming
Z	Coordinate value of pocket bottom in absolute programming; incremental value of pocket bottom relative to reference point R in incremental programming
X	Center of pocket. Coordinate of the first axis in the current plane in absolute programming; incremental value relative to the starting point in incremental programming.
Y	Center of pocket. Coordinate of the second axis in the current plane in absolute programming; incremental value relative to the starting point in incremental programming.
I	Radius of pocket
F	Milling speed in roughing
Q	Max. feed depth in roughing (can be omitted when Q=pocket depth-finishing allowance of pocket bottom)
E	Finishing allowance of pocket edge (can be omitted when E=0)
O	Finishing allowance of pocket bottom (can be omitted when O=0)
H	Max. feed depth in finishing (can be omitted when H=Q)
U	Feedrate in finishing (can be omitted when U=F)
P	Spindle speed in finishing (can be omitted when P=spindle speed or default speed before

	entering cycle)
C	Milling direction (can be omitted when C=3) 0：milling in the same direction; 1: milling in the reverse direction; 2: milling in G02 direction; 3: milling in G03 direction
D	Machining type (can be omitted when D=1) 1: roughing; 2: finishing
V	Tool radius

📖 **Description**

Graphical representation

Milling direction

The spindle must be specified to rotate before entering the cycle. A reasonable milling direction is calculated based on the spindle rotation direction M3/4 and the milling direction set by the user before entering the cycle. The selection of the milling direction is shown in the following table:

Milling direction (cycle parameter C)	Specify M03/M04 before entering cycle	
	M03 spindle rotation CW	M04 spindle rotation CCW
0： milling in the same direction	G03	G02
1： milling in the reverse direction	G02	G03
2： G02 direction	G02	G02
3： G03 direction	G03	G03
Omitted	G03	G03

Steps

(1) Selecting a position as the starting position of the cycle, this position can be random, and it must be possible to reach to workpiece from this position without collision.

(2) Roughing（D=1）

Move to the reference level above the edge of pocket with G00 (leaving finishing allowance of the edge), feed the amount Q, mill the pocket surface from outside to inside based on the milling direction specified by C, return to the same point and mill the surface again to the finishing allowance at the bottom of the pocket.

(3) Finishing（D=2）

Move to the reference level above the edge of pocket with G00 (leaving finishing allowance of the edge), feed the amount H, mill the fillishing allowance from outside to inside based on the milling direction specified by C, and return to the same point and mill the surface again to the finishing allowance at the bottom of the pocket. Then mill the fillishing allowance of bottom from outside to inside based on the milling direction specified by C

(4) After the pocket is machined, move the tool to the initial level or reference level with G98/G99, and the cycle is completed.

Example

Mill a circular pocket. The center is X50 Y50, the radius is 100mm, the pocket depth is 50mm, the finishing allowance of the pocket bottom and the pocket edge are 2mm and 1.5mm respectively, the roughing depth is 4mm, and the tool radius is 5mm.

```
%1022
G54 X0 Y0 Z40
G17 G90
T10
M06
M03 S650
G99 G185 R0 Z-50 X50 Y50 I100 F300 Q4 E1.5 O2 V5D1;  Roughing
X50 Y50 I100 P800 H1.5 D2;    Finishing
M30
```

Notes

(1) Please refer to section 16.4.9 for the alarms generated in this cycle;

(2) Q and H are the maximum feed depths for roughing and finishing respectively. Note that when the feed allowance cannot be divisible by Q or H, the last cut will be less than Q or H;

(3) I, E, O, Q, and H are specified as non-negative numbers. If a negative number is specified, the negative sign will be ignored in the cycle;

(4) C specifies the milling direction such as G182;

(5) The spindle rotation must be specified before entering cycle.

16.4.6 End Face Milling Cycle（G186）

![icon] **Function and Purpose**

This cycle can mill any rectangular end face in finishing and roughing.

![icon] **Command Format**

(G98/G99)G186R_Z_N_W_X_Y_ I_ A_F_Q_J_O_H_K_U_P_C_D_V_

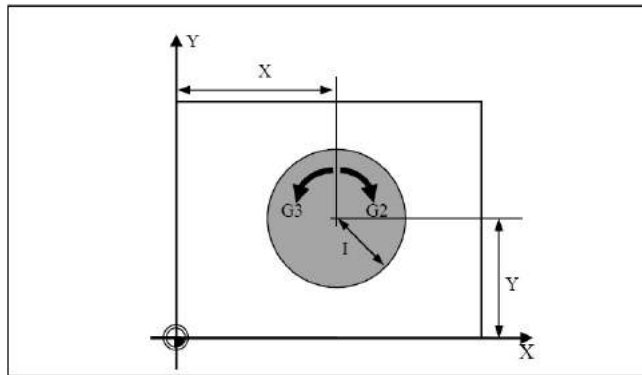| Parameter	Meaning
R	Coordinates of reference point R in absolute programming; distance from initial level B to R point in incremental programming
Z	Coordinate value of bottom in absolute programming; incremental value of bottom relative to reference point R in incremental programming
N	Length of the first axis of workpiece
W	Length of the second axis of workpiece
X、Y	Starting position. Coordinate of the first axis in the current plane in absolute programming; incremental value relative to the current point in incremental programming.
I	Safety margin in milling direction (can be omitted, I=tool radius)
A	The angle between the long side of the end face and the positive direction of the first axis in the plane (can be omitted, A=0)
F	Milling speed in roughing
Q	Max. feed depth in roughing (can be omitted, Q=milling depth–finishing allowance of bottom)
J	Milling width in roughing (can be omitted, J=tool diameter×80%)
O	Finishing allowance of workpiece bottom (can be omitted, O=0）
H	Max. feed depth in finishing (can be omitted, H=Q）
K	Cutting width in finishing (can be omitted, K=tool diameter×80%）
U	Milling speed in finishing (can be omitted, U=F)
P	Spindle speed in finishing (can be omitted, P=spindle speed or default speed before entering cycle)
C	Milling direction (can be omitted, C=0) 0: Bidirectional machining of 1st axis in the selected plane 1: Bidirectional machining of 2nd axis in the selected plane 2: Unidirectional machining of 1st axis in the selected plane 3: Unidirectional machining of 2nd axis in the selected plane
D	Machining type (can be omitted, D=1) 1: roughing, 2: finishing
V	Tool radius

Description

C=0，D=1, bidirectional machining of X axis
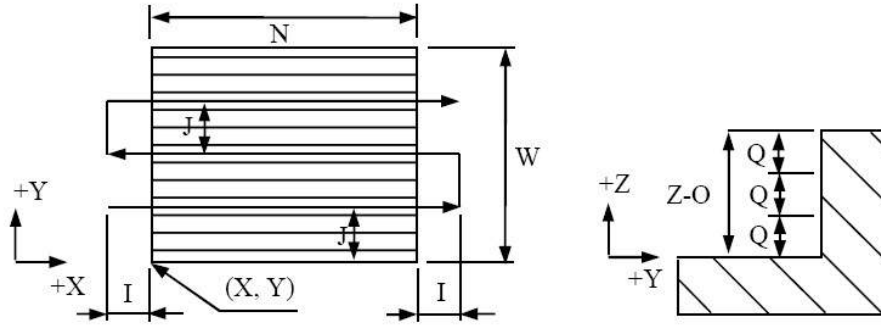
C=1，D=1，unidirectional machining of X axis
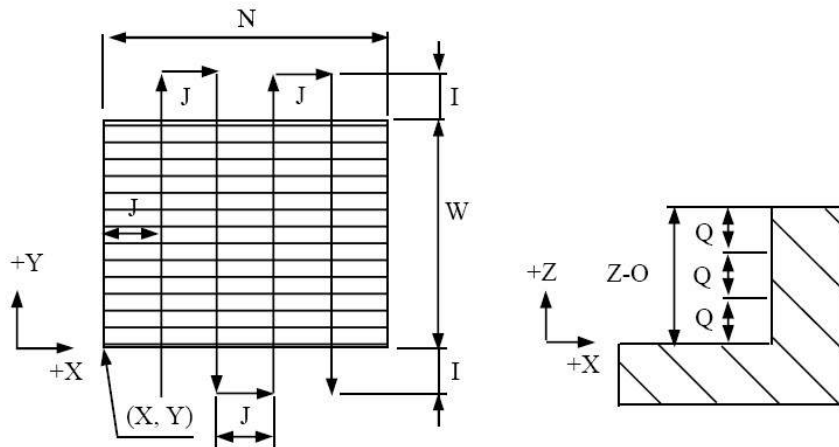
C=2，D=1，unidirectional machining of X axis
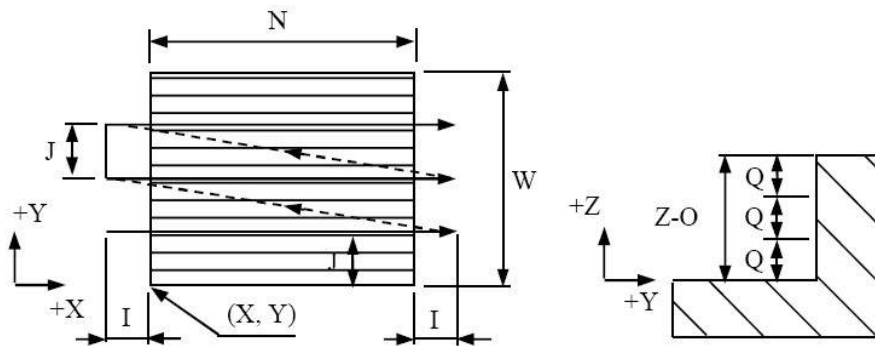
C=3，D=1，unidirectional machining of Y axis
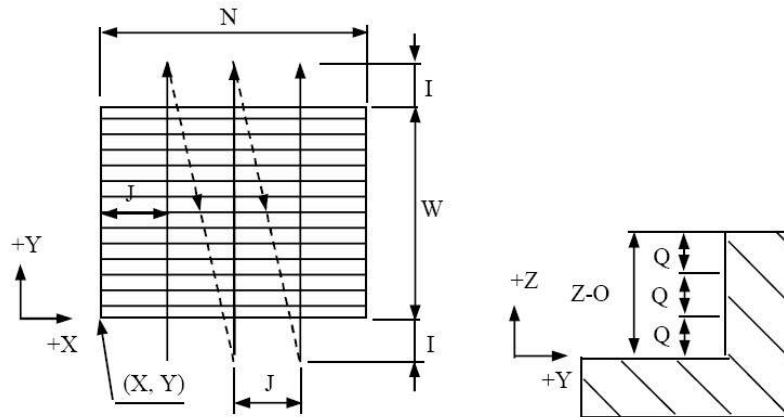
(Note 1) The above table only lists end face milling cycle in G17 plane (rough machining), for G18/G19 plane and so on, finishing and hybrid machining (D=2) are also similar.

Example

For milling a rectangular end face, the end face dimensions and related process parameters are as follows:

Initial level 10mm; reference level 2mm, only rough machining is allowed, each milling width 10mm, each feed depth 6mm, total milling depth 11mm, milling starting point (100, 100), end face size 60mm×40mm, the safety margin in milling direction is 5mm. Bidirectional milling of X axis is allowed, the processing feedrate of surface is 500mm/min, and the milling tool radius is 5mm.

```
N10     G54 X0 Y0 Z20
N20     G17 G90
N30     T10
N40     M06
N50     M03 S650
N60     G00 X0 Y0 Z20
N70     G99G186 Z-11 R0 N60 W40 X100 Y100 I5 F500 Q6 J10 V5
N80 M30
```

Note

1) N, W, I, O, Q, J, H, K are designated as non-negative numbers. If a negative number is input, the negative sign will be ignored in the cycle;

2) For the feed width (J, K) and feed depth (Q, H), when the feed amount cannot be divisible by it, the last cut will be smaller than the feed width or depth;

3) The spindle must be commanded to rotate before entering the cycle;

4) Please refer to section 16.4.9 for the alarms generated in this cycle.

16.4.7 Rectangular Boss Milling Cycle (G188)

Function and Purpose

Rectangular boss milling cycle is for processing rectangular bosses of any size in the plane. The rectangle bosses can have corner arcs. User can choose the type of machining, finishing, rough machining or hybrid machining.
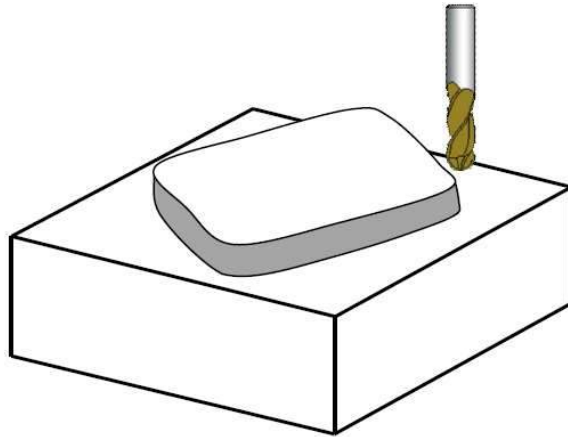
Command Format

(G98/G99)G188R_Z_N_W_X_Y_J_K_I_A_F_Q_E_O_H_U_P_C_D_V_；

Parameter	Meaning
R	Coordinates of R point in absolute programming; distance from reference point R to initial level in incremental programming
Z	Coordinates of boss bottom in absolute programming; distance from boss bottom to reference point R in incremental programming
N	Length of boss
W	Width of boss
X	Center of boss. Coordinate of the first axis in the current plane in absolute programming; incremental value relative to the starting point in incremental programming.
Y	Center of boss. Coordinate of the second axis in the current plane in absolute programming; incremental value relative to the starting point in incremental programming.
J	Length of boss blank
K	Width of boss blank
I	Radius of corner of boss (can be omitted, I=W/2）
A	The angle between the long side of the rectangular boss and the positive direction of the first axis in the plane (can be omitted, A=0)
F	Milling speed in roughing

Q	Max. feed depth in roughing (can be omitted, Q=pocket depth-finishing allowance of boss bottom)
E	Finishing allowance of boss edge (can be omitted when E=0)
O	Finishing allowance of boss bottom (can be omitted when O=0)
H	Max. feed depth in finishing (can be omitted when H=Q)
U	Feedrate in finishing (can be omitted when U=F)
P	Spindle speed in finishing (can be omitted when P=spindle speed or default speed before entering cycle)
C	Milling direction (can be omitted when C=3) 0：milling in the same direction; 1: milling in the reverse direction; 2: milling in G02 direction; 3: milling in G03 direction
D	Machining type (can be omitted, D=1) 1: roughing; 2: finishing

Description

Graphical representation
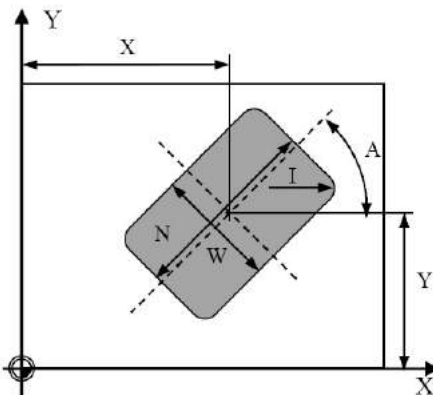
Enter contour, exit contour
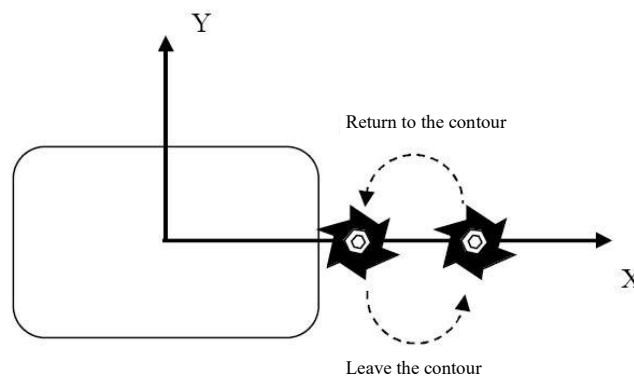
In order to ensure the smoothness when the tool enters the workpiece, the cycle automatically adds a semicircle path when entering and exiting the workpiece during execution. The radius is determined by the cycle parameters, and the direction is opposite to the milling direction. If the

G2 direction is specified for milling, then the semicircle inserted here is in the G3 direction.

Size of boss blank
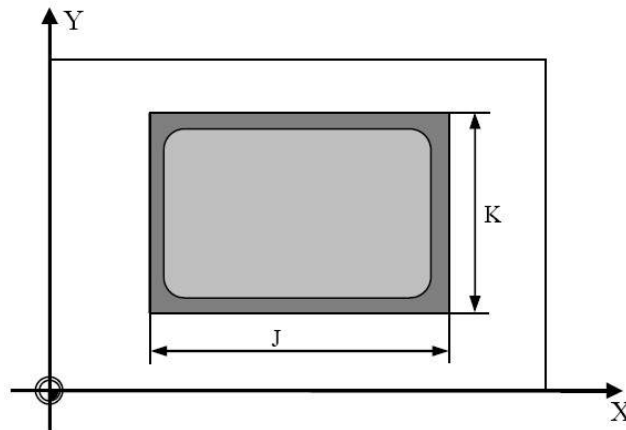
For the processing of pre-poured workpieces, the blank size of the rectangular boss can also be considered, which is symmetrical to the boss size, and the center point is also (X, Y).

Milling direction

The spindle must be specified to rotate before entering the cycle. A reasonable milling direction is calculated based on the spindle rotation direction M3/4 and the milling direction set by the user before entering the cycle. The selection of the milling direction is shown in the following table:

Milling direction (cycle parameter C)	Specify M03/M04 before entering cycle	
	M03 spindle rotation CW	M04 spindle rotation CCW
0: milling in the same direction	G03	G02
1: milling in the reverse direction	G02	G03
2: G02 direction	G02	G02
3: G03 direction	G03	G03
Omitted	G03	G03

Steps

(1) Select a position as the starting position of the cycle. This point needs to be positioned on the right side of the boss in the positive direction of the first axis in the plane. Note that the semicircle automatically added needs to be taken into account.

(2) Roughing（D=1）

Move to the reference level above the wide side of the boss with G00, feed an amount, mill the blank surface to the finishing allowance of the edge based on the milling direction, exit the workpiece contour, rapid traverse to the starting point with G00, and feed to machine the surface of boss to the finishing allowance of the bottom.

(3) Finishing（D=2）

Move to the reference level above the wide side of the boss with G00, feed an amount, enter the workpiece contour to mill the finishing allowance of the edge based on the milling direction, exit the workpiece contour, rapid traverse to the starting point with G00, and then feed to machine the mill the bottom to the finishing allowance; and mill the finsihing allowance of the bottom.

(4) After the boss is machined, move the tool to the initial level or reference level with G98/G99, and the cycle is completed.

Example

Machine the rectangular boss as shown in the figure below, the boss size is 60mm×40mm, the blank size is 80mm×50mm, and the tool radius is 3mm.
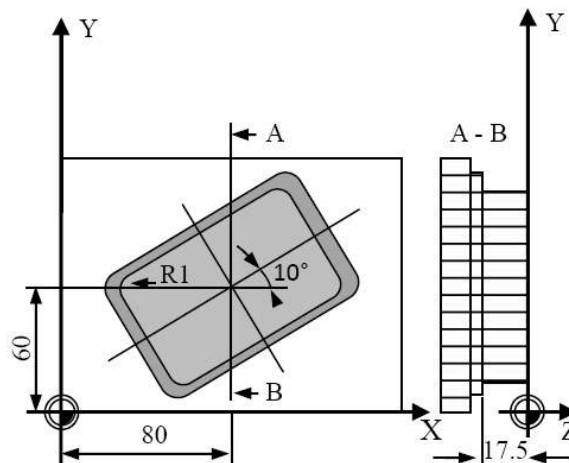

```
%1019
G17 G54 G90
T10
M06
M03 S650
G98 G188 R2Z-17.5N60W40X80Y60J80K50I15A10F200Q11E2O1V3
```

M30

 Note

(1) Please refer to section 16.4.9 for the alarms generated in this cycle;

(2) W, J, K, I, E, O, Q, H are designated as non-negative numbers. If a negative number is specified, the negative sign will be ignored in the cycle;

(3) The maximum feed depth of finishing and roughing is specified with Q and H respectively. When the feed amount cannot be divisible by it, the last cut will be smaller than Q or H;

(4) The spindle must be commanded to rotate before entering the cycle;

(6) If the specified wide side is greater than the long side, the wide side and the long side will be automatically exchanged by the cycle, and the corresponding rotation will be performed to meet the desired boss position.

16.4.8 Round Boss Milling Cycle (G189)

 Function and Purpose

This cycle is to machine the round boss of any size in the plane.
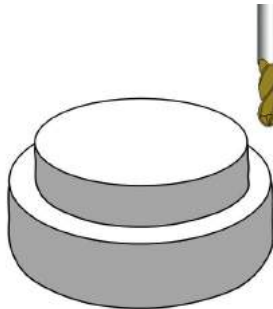
 Command Format

(G98/G99)G189R_Z_X_Y_ I_J_F_ Q_E_O_H_U_P_C_D_V_；

Parameter	Meaning
R	Coordinates of R point in absolute programming; distance from reference point R to initial level in incremental programming
Z	Coordinates of boss bottom in absolute programming; incremental value of boss bottom relative to reference point R in incremental programming
X	Center of boss. Coordinate of the first axis in the current plane in absolute programming; incremental value relative to the starting point in incremental programming.

Y	Center of boss. Coordinate of the second axis in the current plane in absolute programming; incremental value relative to the starting point in incremental programming.
I	Radius of round boss
J	Radius of blank
F	Milling speed in roughing
Q	Max. feed depth in roughing (can be omitted)
E	Finishing allowance of boss edge (can be omitted, E=0)
O	Finishing allowance of boss bottom (can be omitted, O=0)
H	Max. feed depth in finishing (can be omitted, H=Q)
U	Feedrate in finishing (can be omitted, U=F)
P	Spindle speed in finishing (can be omitted, P=spindle speed or default speed before entering cycle)
C	Milling direction (can be omitted, C=3) 0：milling in the same direction; 1: milling in the reverse direction; 2: milling in G02 direction; 3: milling in G03 direction
D	Machining type (can be omitted, D=1) 1: roughing; 2: finishing
V	Tool radius

📖 **Description**

Enter contour, exit contour
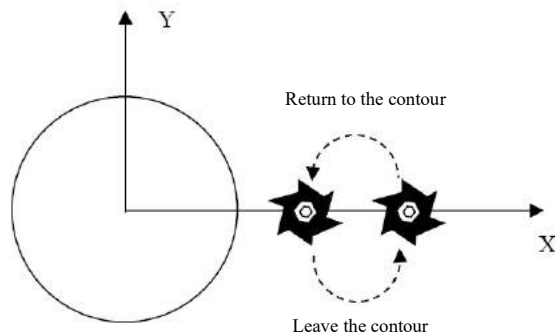
Like the G188 cycle, this cycle also adds a semi-circle every time the tool horizontally enters or exits the blank workpiece to ensure the smooth transition of the moving from boss edge to the blank, and the radius of the semi-circle is automatically calculated by the cycle.

Blank size

Like the G188 cycle, this cycle can also set the size of the blank, and the center point is also (X, Y).

Milling direction

The spindle must be specified to rotate before entering the cycle. A reasonable milling direction is calculated based on the spindle rotation direction M3/4 and the milling direction set by the user before entering the cycle. The selection of the milling direction is shown in the following table:

Milling direction	Specify M03/M04 before entering cycle	
(cycle parameter C)	M03 spindle rotation CW	M04 spindle rotation CCW
0： milling in the same direction	G03	G02
1： milling in the reverse direction	G02	G03
2： G02 direction	G02	G02
3： G03 direction	G03	G03
Omitted	G03	G03

Steps

(1) Select a position as the starting position of the cycle. This point needs to be positioned on the right side of the boss in the positive direction of the first axis in the plane. Note that the semicircle automatically added by the cycle needs to be taken into account.

(2) Roughing（D=1）

Move to the reference level above the wide side of the boss with G00, feed an amount, mill the blank surface to the finishing allowance of the edge based on the milling direction, exit the workpiece contour, rapid traverse to the starting point with G00, and feed to machine the surface of boss to the finishing allowance of the bottom.

(3) Finishing（D=2）

Move to the reference level above the wide side of the boss with G00, feed an amount, enter the workpiece contour to mill the finishing allowance of the edge based on the milling direction, exit the workpiece contour, rapid traverse to the starting point with G00, and then feed to machine the mill the bottom to the finishing allowance; and mill the finsihing allowance of the bottom.

(4) After the boss is machined, move the tool to the initial level or reference level with G98/G99, and the cycle is completed.

Example

Machine the round boss as shown in the figure below, the blank radius is 55mm, the feed depth for each cut is 10mm, and the tool radius is 5mm.
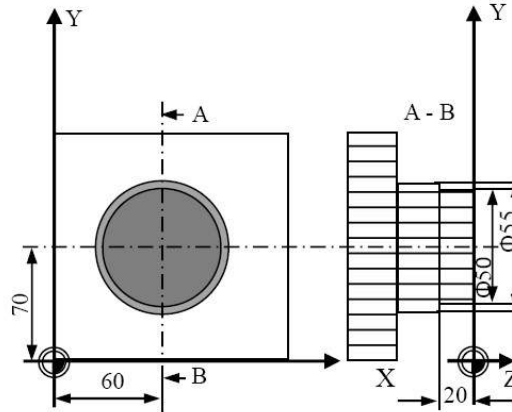
%1020
G17 G54 G90
T10
M06
M03 S650
G98G189 R2Z-20X60Y70I25J27.5F200Q10E1O1V5
M30

> ⚠️ **Note**

(1) Please refer to section 16.4.9 for the alarms generated in this cycle.

(2) J, K, I, E, O, Q, H are designated as non-negative numbers. If a negative number is specified, the negative sign will be ignored in the cycle.

(3) The maximum feed depth for roughing and finishing is specified with Q and H respectively, when the feed amount cannot be divisible by it, the last cut will be smaller than Q or H;

(4) The spindle must be commanded to rotate before entering the cycle;

16.4.9 Milling Cycle Alarm Diagnosis

	Function and Purpose

When the milling canned cycle is executed, if the system recognizes an error, an alarm will be generated and the execution of the current cycle will be interrupted. It will continue after modified by user.

This section lists the possible alarms that the system may generate during the execution of the milling cycle, and analyzes the cause of the alarms. User can modify the program based on that before continuing to execute the cycle.

	Description

Alarm No.	Message	Source	Cause and comments
800	No tool radius defined	G181 G182 G183 G184 G185 G186 G188 G189	No tool radius V is specified before entering cycle.
801	No reference level defined	G181 G182 G183 G184 G185 G186 G188 G189	If R is not specified in this block, and the R modal value cannot be detected by the cycle, this alarm will be generated. This parameter is required to specify the groove depth in increments or when returning to the R level at the end of the cycle, so this parameter should be defined.
802	No groove bottom position defined	G181 G182 G183 G184 G185 G186 G188 G189	The position of groove bottom needs to be defined, otherwise the groove depth cannot be determined in the cycle.
803	Number of grooves defined as zero	G181 G182 G183	If the number of grooves is specified as 0, an alarm will be generated. The number of grooves should be specified

			as an integer larger than 0.
804	Groove lenth defined too small	G182	The groove length must be greater than the groove width, otherwise this alarm will be generated.
805	Tool radius too large	G181 G182 G183 G184 G185 G186 G188 G189	If the diameter of the milling cutter exceeds the groove length defined by the cycle, this alarm will be generated. User can use a milling cutter with a smaller radius to complete the milling.
806	No center position defined	G181 G182 G183	If the arc center position is not explained in the block, and the cycle fails to detect the modal position, this alarm will be generated.
807	No radius defined	G181 G182 G183	If there is no arc radius modal value, the radius value must be specified in this row, otherwise this alarm will be generated.
808	Interference between grooves	G181 G182 G183	Considering the angle between the milling cutter radius and the grooves, there may be interference between the machined grooves, which will affect the contour shape of the groove. The interference check will be performed before the cycle, and user will be prompted in time.
809	Number of grooves conflicts with angle increment defined	G181 G182 G183	。Due to the improper definition of the number of grooves and the angle between grooves, this alarm will be generated if the number of grooves ×angle between grooves> 360 degrees.
810	Max. feed depth defined too large	G181 G182 G183 G184 G185 G186 G188 G189	The alarm is generated when the maximum feed depth Q is greater than the groove depth. If this alarm occurs, reduce Q.
811	Please command spindle	G181	Before the cycle is executed, the

	to rotate before entering cycle	G182 G183 G184 G185 G186 G188 G189	current spindle status is detected. This alarm will be generated if the spindle is not rotating.
812	Center position not defined for circular pocket or boss	G184 G185 G188 G189	If the position of circular pocket center or boss center is not defined, this alarm will be generated.
813	Radius position not defined for circular pocket or boss	G185 G189	If the radius of round pocket or boss is not defined, this alarm will be generated.
814	Finishing allowance of edge too large	G182 G183 G184 G185 G188 G189	The reserved edge finishing allowance is too large to complete the machining. Please reduce the allowance.
815	Finishing allowance of bottom too large	G182 G183 G184 G185 G186 G188 G189	The reserved finishing allowance of edge is too large to complete the machining. Please reduce the allowance.
816	Max.feed depth in finishing defined too large	G182 G183 G184 G185 G186 G188 G189	When the maximum feed depth H is larger than the groove depth, this alarm is generated, then reduce H.
819	Size of workpiece to be machined not defined	G186	For G186, the end face size of the workpiece to be processed must be specified, such as the length and the width, otherwise this alarm will be issued.
820	No milling starting position defined	G186	It is necessary to specify the starting point of milling for face milling cycle G186, which is generally the lower left corner of the workpiece in

			machining plane. If it is not specified, this alarm will be generated.
821	Safety margin defined too small	G186	A safety margin must be specified for the face milling cycle for a good milling effect, and this value is not less than the milling tool radius.
822	Milling width defined too large in roughing	G186	The milling width is not larger than tool diameter in roughing for face milling cycle G186.
823	Milling width defined too large in finishing	G186	The milling width is not larger than tool diameter in finishing for face milling cycle G186.
824	No blank size defined for boss	G188 G189	For round boss G189 or rectangular boss G188, the blank size needs to be defined, otherwise this alarm will be generated.
826	No length or width defined for pocket or boss	G181 G182 G183 G184 G188	If the groove length or width is not specified in this block, and the modal value of the groove length is not detected by the cycle, this alarm will be generated.
829	Corner radius defined too large for rectangular pocket and boss	G184 G188	The circular corner can be defined for the rectangular groove or rectangular boss cycle, but it cannot be greater than the half of the wide side, otherwise this alarm will be generated.
830	Blank size defined smaller than machining size	G188 G189	For round boss G189 or rectangular boss G188, the blank size must be larger than the contour size, otherwise this alarm will be generated.
873	Tool radius cannot be zero	G181 G182 G183 G184 G185 G186 G188 G189	V represents the compensation number in the tool compensation table. The value filled in the compensation number is the tool radius. This value cannot be zero, otherwise this alarm will be generated.
874	No finishing allowance defined	G182 G183 G184 G185 G186	During finishing machining, the finishing allowance of groove wall and the finishing allowance of groove bottom cannot be unspecified or specified as 0 at the same time,

		G188	otherwise this alarm will be generated.
		G189	
817	Milling direction definition error	G182	C sets a value other than 0, 1, 2, and 3. If a milling direction not supported by the system is defined, this alarm will be issued.
		G183	
		G184	
		G185	
		G186	
		G188	
		G189	
818	Machining type definition error	G182	D sets a value other than 1 and 2. If a milling direction not supported by the system is defined, this alarm will be issued.
		G183	
		G184	
		G185	
		G186	
		G188	
		G189	

17 Multiple Repetitive Cycle in Lathe（T）

This kind of canned cycle can simplify programming, and describe the tool path of roughing using the shape data of finishing. The system provides four multiple repetitive cycles for users:

G71： Inner (outer) diameter roughing multiple repetitive cycle

G72： End face roughing multiple repetitive cycle

G73： Closed contour multiple repetitive cycle

G76： Thread cutting multiple repetitive cycle

Using multiple repetitive cycle commands, users only need to specify the finishing path and the feed amount of roughing, and the system will automatically calculate the roughing path and the number of cuttings.

Note

1. The cycles described in this chapter are only for lathe system.

2. For the multiple repetitive cycle of G71, G72, G73, please note the following:

- The block designated by address P should have G00 or G01 command of group 01, otherwise an alarm will be generated;

- The multiple repetitive cycle cannot run in MDI.

- In the multiple repetitive cycles G71, G72, G73, the block where the sequence number designated by P, Q should not include M98 subprogram call or M99 subprogram return command;

- In the multiple repetitive cycles G71, G72, G73, the tool compensation can be performed only in the blocks of which sequence numbers designated by P and Q.

17.1 Inner (Outer) Diameter Roughing Multiple Repetitive Cycle (G71)

Function and Purpose

This cycle includes the inner (outer) diameter roughing multiple repetitive cycle with pocket and without pocket.

This command executes rough machining as shown in the figure below, and the tool returns to the starting point of the cycle. The finishing path A→A'→B'→B is executed sequentially based on the following commands.

1. Path of inner/outer diameter roughing multiple repetitive cycle without pocket
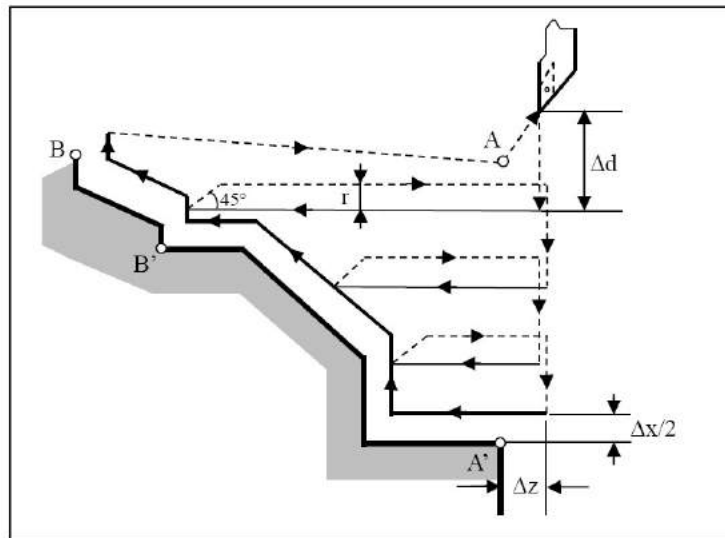
2. Path of inner/outer diameter roughing multiple repetitive cycle with pocket

Command Format

1. Inner/outer diameter roughing multiple repetitive cycle without pocket

G71 U(Δd) R(r) P(ns) Q(nf) X(Δx) Z(Δz) F(f) S(s) T(t)

Parameter	Meaning
U	Cutting depth, without sign. The direction is determined by vector AA′
R	Retraction amount
P	Sequence number of the first program block (AA' in the below figure) for finishing path
Q	Sequence number of the last program block (BB' in the below figure) for finishing path
X	Finishing allowance in X direction
Z	Finishing allowance in Z direction
FS T	F, S, T in G71 are valid in roughing, and F, S, T between block ns and nf are valid in finishing

2. Inner/outer diameter roughing multiple repetitive cycle with pocket

G71 U(Δd) R(r) P(ns) Q(nf) E(e) F(f) S(s) T(t)

Parameter	Meaning
U	Cutting depth, without sign. The direction is determined by vector AA′
R	Retraction amount
P	Sequence number of the first program block (AA' in the below figure) for finishing path
Q	Sequence number of the last program block (BB' in the below figure) for finishing path
E	Finishing allowancewhich is the equal height distance in the X direction; positive for outer diameter cutting, and negative for inner diameter cutting
F S T	F, S, T in G71 are valid in roughing, and F, S, T between block ns and nf are valid in finishing

Description

In the G71 cutting cycle, the cutting feed direction is parallel to the Z axis, so it is suitable for the inner and outer diameter cutting along the axial direction.

This cycle is a rough machining cycle. It is necessary to reserve an appropriate allowance for subsequent finishing. The finishing allowance in the X\Z axis direction is X(x) and Z(z), x is the allowance on X, z is the allowance on Z.

Since the direction of the allowance is determined by the programmer based on the processing technology, the system determines the direction of the feed according to the direction of the finishing allowance set by the program. The direction of the finishing allowance (positive and negative settings) must be correct, otherwise it will cause an error in the feed direction of the tool.

The relationship between the sign of the finishing allowance and the feed direction is shown in the figure below. (+) means that the contour allowance is kept in the positive direction of the axis, and (-) means that the contour allowance is kept in the negative direction of the axis.

Machine tool with rear tool post   Machine tool with front tool post
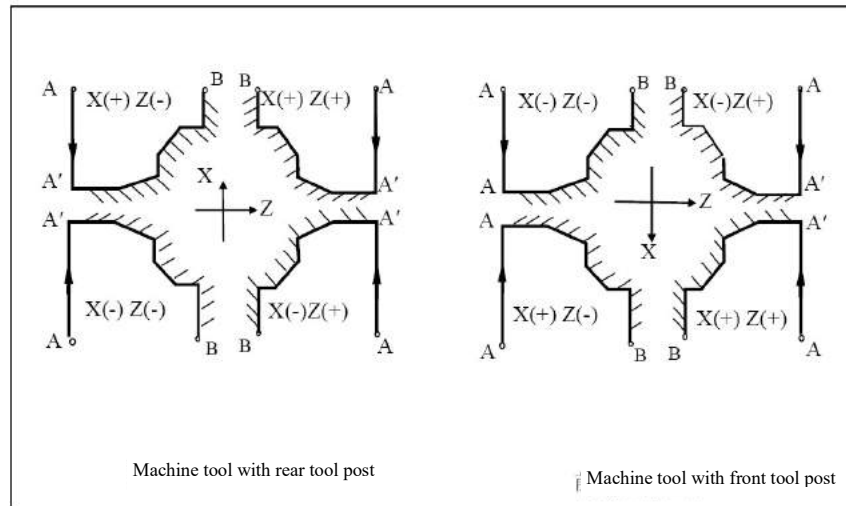
Example

Example 1: Use the outer diameter roughing multiple repetitive cycle to compile the machining program of the part shown in the figure below: the starting point of the cycle is required to be A (46, 3), and the cutting depth is 1.5mm (radius), the retraction amount is 1mm, the finishing allowance in the X direction is 0.4mm, and the finishing allowance in the Z direction is 0.1mm. The dotted line is the workpiece blank.
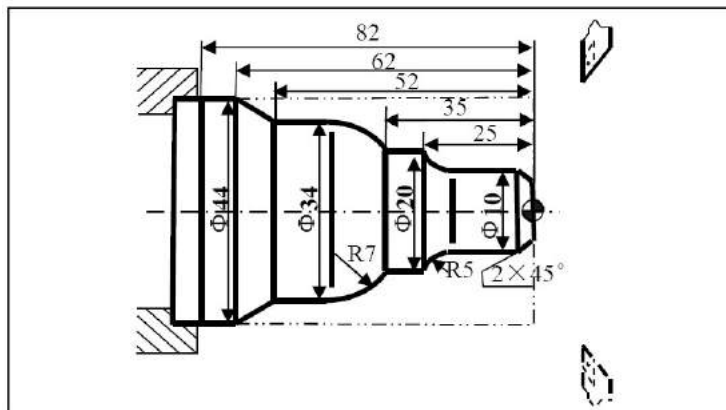
%3325

T0101; Set coordinate system, select No.1 tool

N1 G00 X80 Z80;　Move to starting point of program

N2 M03 S400;　Spindle rotates CW at 400r/min

N3 G01 X46 Z3 F100;　Move to starting point of cycle

N4 G71U1.5R1P5Q14X0.4 Z0.1;　Rough cut：1.5mm Finish cut：X0.4mm Z0.1mm

N5 G00 X0;　Finishing contour starts, move to extension line of chamfer

N6 G01 X10 Z-2 ;　Finishing 2×45°chamfer

N7 Z-20;　Finishing Φ10 outer circle

N8 G02 U10 W-5 R5;　Finishing R5 arc

N9 G01 W-10;　Finishing Φ20 outer circle

N10 G03 U14 W-7 R7;　Finishing R7 arc

N11 G01 Z-52;　Finishing Φ34 outer circle

N12 U10 W-10;　Finishing outer cone

N13 W-20;　Finishing Φ44 outer circle

N14 U1;　Finishing contour end

N15 X50;　Exit the machined surface

N16G00 X80 Z80;　Return to tool setting point

N17 M05;　Spindle stop

N18 M30;　Main program end and reset

Example 2: Use the inner diameter roughing multiple repetitive cycle to compile the processing program of the part shown in the figure below: the starting point of the cycle is required to be A (6, 5), and the cutting depth is 1.5mm (radius). The retraction amount is 1mm, the finishing allowance in the X direction is 0.4mm, and the finishing allowance in the Z direction is 0.1mm. The dotted line is the workpiece blank.
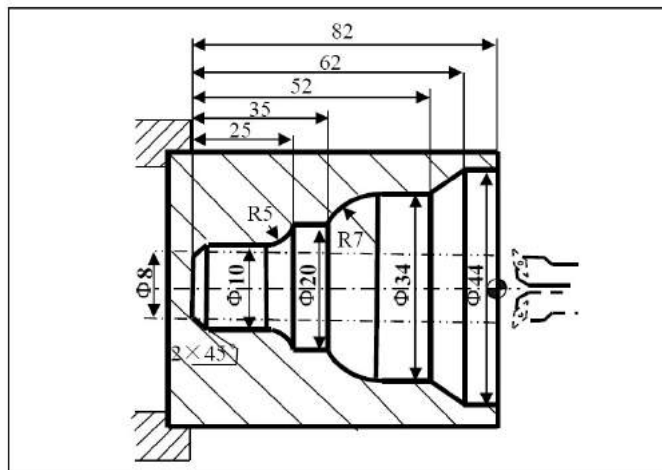
%3326

N1 T0101;　Select No. 1 tool, and determine the coordinate system

N2 G00 X80 Z80;　Move to starting point of program or the tool change position

N3 M03 S400;　Spindle rotates CW at 400r/min

N4 X6 Z5;　Move to starting point of cycle

G71U1R1P8Q16X-0.4Z0.1 F100;   Machining in inner diameter roughing cycle

N5 G00 X80 Z80;   After rough cutting, move to tool change position

N6 T0202;   Change to No.2 tool, and determine the coordinate system

N7 G00 G41X6 Z5;   Tool nose radius compensation is added to No. 2 tool

N8 G00 X44;   Finishing contour starts, move to Φ44 outer circle

N9 G01 Z-20 F80;   Finishing Φ44 outer circle

N10 U-10 W-10;   Finishing outer cone

N11 W-10;   Finishing Φ34 outer circle

N12 G03 U-14 W-7 R7;   Finishing R7 arc

N13 G01 W-10;   Finishing Φ20 outer circle

N14 G02 U-10 W-5 R5;   Finishing R5 arc

N15 G01 Z-80;   Finishing Φ10 outer circle

N16 U-4 W-2;   Finishing chamfer 2×45°, finishing contour end

N17 G40 X4;   Exit machined surface, cancel tool radius compensation

N18 G00 Z80;   Exit inner hole of workpiece

N19 X80;   Return to starting point of program or tool change position

N20 M30;   Spindle stop, main program end and reset

Example 3: Use the outer diameter roughing multiple repetitive cycle with pocket to compile the processing program of the part shown in the figure below, where the dashed line is the workpiece blank.

%3327

N1 T0101;   Change to No.1 tool, and determine the coordinate system

N2 G00 X80 Z100；   Move to starting point of program or tool change position
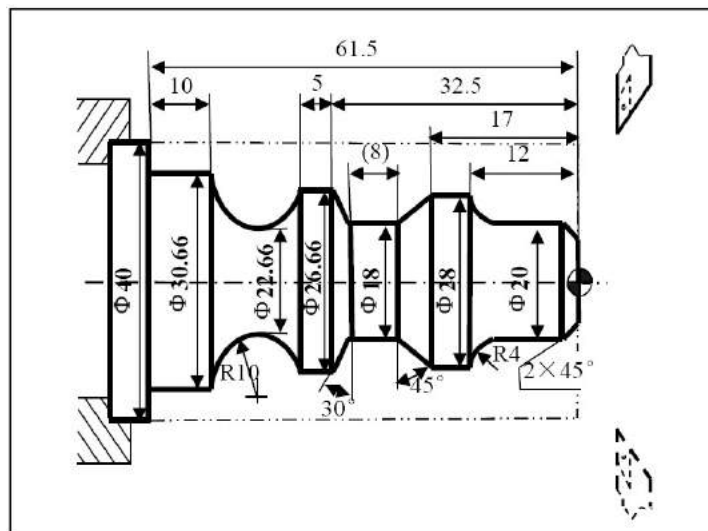
M03 S400;   Spindle rotates CW at 400r/min

N3 G00 X42 Z3;   Move to the starting point of cycle

N4G71U1R1P8Q19E0.3F100 ；　Machining in roughing cycle with pocket

N5 G00 X80 Z100；　Move to tool change position after roughing

N6 T0202;　Change to No. 2 tool, and determind the coordinate system

N7 G00 G42 X42 Z3;　Tool radius compensation is added to No. 2 tool

N8 G00 X10;　Finishing contour starts, move to extension line of chamfer

N9 G01 X20 Z-2 F80;　Finishing chamfer 2×45°

N10 Z-8 ;　Finishing Φ20 outer circle

N11 G02 X28 Z-12 R4;　Finishing R4 arc

N12 G01 Z-17;　Finishing Φ28 outer circle

N13 U-10 W-5;　Finishing tangent cone

N14 W-8 ;　Finishing Φ18 outer circle

N15 U8.66 W-2.5;　Finishing tangent cone

N16 Z-37.5 ;　Finishing Φ26.66 outer circle

N17 G02 X30.66 W-14 R10　;　Finishing R10 tangent arc

N18 G01 W-10;　Finishing Φ30.66 outer circle

N19 X40；　Exit machined surface, finishing contour end

N20 G00 G40 X80 Z100;　Cancel tool radius compensation, return to tool change position

N21 M30;　Spindle stop, main program end and reset

⚠ Note

(1) The G71 comand must have P, Q addresses ns, nf which must correspond to the start and end sequence numbers of the finishing path, otherwise the cycle processing cannot be performed;

(2) The block of ns must be G00/G01 command, that is, the movement from A to A'must be a linear or point positioning movement;

(3) In the blocks from sequence number ns to sequence number nf, subprogram should not be included;

(4) The block of G71ns must be monotonically increasing or decreasing when there is not groove.

(5) The rough turning cycle is realized by the G71 command with addresses P and Q. The F S and T functions specified in the motion command are invalid. But the F S and T functions specified in the G71 block or the previous block are valid.

(6) When the constant speed limit is used for cutting speed control, the G96 or G97 specified in the motion command between "ns" and "nf" is invalid. It is valid to specify G96 or G97 in the G71 block or the previous block.

(7) If user needs to process contours with non-monotonic increasing or decreasing in Z direction, the processing path between "H1", "ns" and "nf" in the G71 command line. If both X axis and Z axis are monotonous increasing or decreasing, the machining allowance at this time is represented

by "XZ"; if there is a groove on the processed contour path in the X direction, the machining allowance at this time is represented by "E".

(8) In principle when machining the outer circle, the cycle starting point must be established higher than the highest point of the contour; when machining the inner hole, it must be lower than the lowest point of the contour. (The starting point of the cycle can be controlled to be 0 to 1mm lower than or higher than the highest point of the contour through the machine user parameter 010161)

(9) 10 pockets are supported at maximum for the machining of contour with pocket.

(10) G71 cannot be in the same line as M code

(11) In the any line with scanning mode, the block between ns and nf cannot be specified.

17.2　End Face Roughing Multiple Repetitive Cycle (G72)

Function and Purpose

This cycle is similar as G71.

The G72 turning is shown in the figure below, where the finishing path is A→A'→B'→B
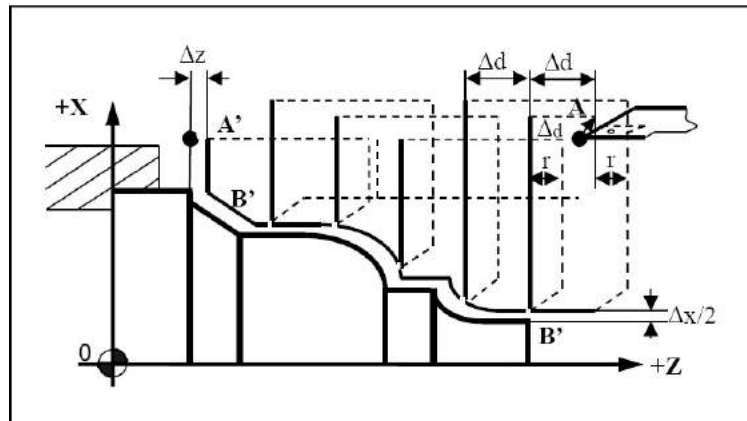
Command Format

G72W($\triangle$d) R(r) P(ns) Q(nf) X($\triangle$x) Z($\triangle$z) F(f) S(s) T(t);

Parameter	Meaning
W	Cutting depth, without sign. The direction is determined by vector AA′
R	Retraction amount
P	Sequence number of the first program block (AA' in the below figure) for finishing path

Q	Sequence number of the last program block (BB' in the below figure) for finishing path
X	Finishing allowance in X direction
Z	Finishing allowance in Z direction
F ST	F, S, T in G72 are valid in roughing, and F, S, T between blocks ns and nf are valid in finishing

Description

In the G72 cutting cycle, the cutting feed direction is parallel to the X axis, so it is suitable for face cutting along the X axis.

This cycle is a rough machining cycle. It is necessary to reserve an appropriate allowance for subsequent finishing. The finishing allowance in the X\Z axis direction is X(Δx) and Z(Δz), Δx is the allowance on X, Δz is the allowance on Z.

Since the direction of the allowance is determined by the programmer based on the processing technology, the system determines the direction of the feed according to the direction of the finishing allowance set by the program. Therefore, the direction of the finishing allowance (positive and negative settings) must be correct, otherwise it will cause an error in the feed direction of the tool.

The relationship between the sign of the finishing allowance and the feed direction is shown in the figure below. (+) means that the contour allowance is kept in the positive direction of the axis, and (-) means that the contour allowance is kept in the negative direction of the axis.

Machine tool with rear tool post

Machine tool with front tool post
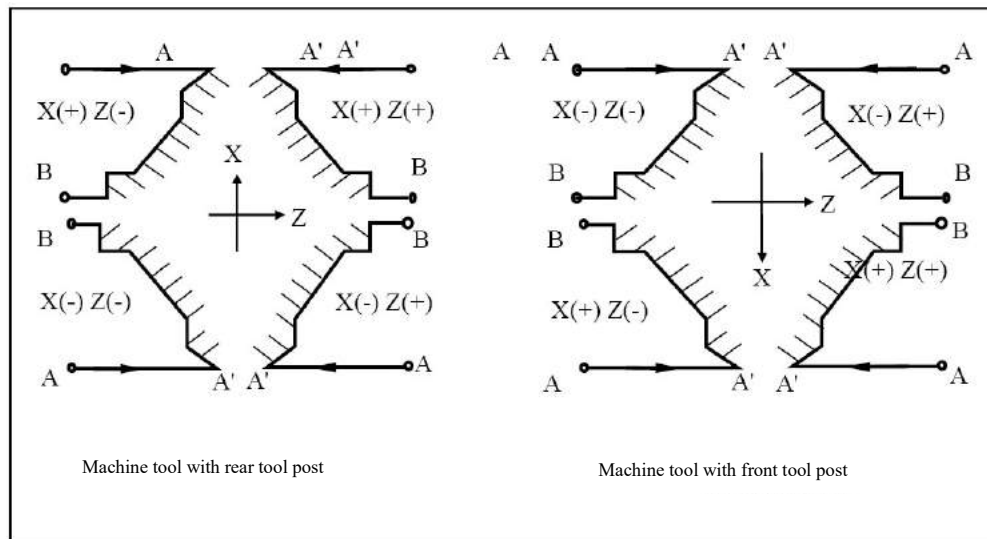
Example

Example 1: Compile the machining program of the part as shown in the figure below: The starting point of the cycle is required to be A (80, 1), and the cutting depth is 1.2mm. The retraction amount is 1mm, the finishing allowance in the X direction is 0.2mm, and the finishing allowance in the Z direction is 0.5mm. The dotted line is the workpiece blank.
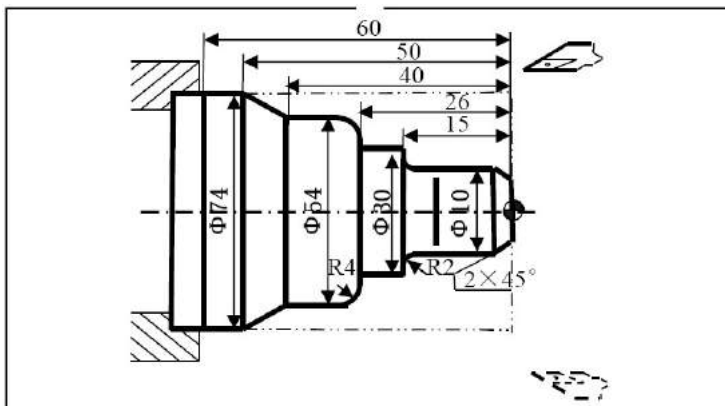
%3328

N1 T0101;   Change to No. 1 tool, and determine the coordinate system

N2 G00 X80 Z80;   Move to starting position of program

N3 M03 S400;   Spindle rotates CW at 400r/min

N4 X80 Z1;   Move to starting point of cycle

N5 G72W1.2R1P8Q17X0.2Z0.5F100 ;   Machining in face roughing cycle

N6 G00 X100 Z80;   Move to tool change position after roughing

N7 G42 X80 Z1;   Tool nose radius compensation is added

N8 G00 Z-53;   The finishing contour starts, and move to extension line of chamfer

N9 G01 X54 Z-40 F80; Cone finishing

N10 Z-30;   Φ54 outer circle finishing

N11 G02 U-8 W4 R4;   R4 arc finishing

N12 G01 X30;   Z26 face finishing

N13 Z-15;   Φ30 outer circle finishing

N14 U-16;   Z15 face finishing

N15 G03 U-4 W2 R2;   R2 arc finishing

N16 G01 Z-2;   Φ10 outer circle finishing

N17 U-6 W3;   2×45°chamfer finishing, contour finishing end

N18 G00 X50;   Exit machined surface

N19 G40 X100 Z80;   Cancel radius compensation, return to starting position of program

N20 M30;   Spindle stop, main program end and reset

Example 2: Compile the machining program of the part as shown in the figure below: The starting point of the cycle is required to be A (6, 3) and the cutting depth is 1.2mm. The retraction amount is 1mm, the finishing allowance in the X direction is 0.2mm, and the finishing allowance in the Z direction is 0.5mm. The dotted line is the workpiece blank
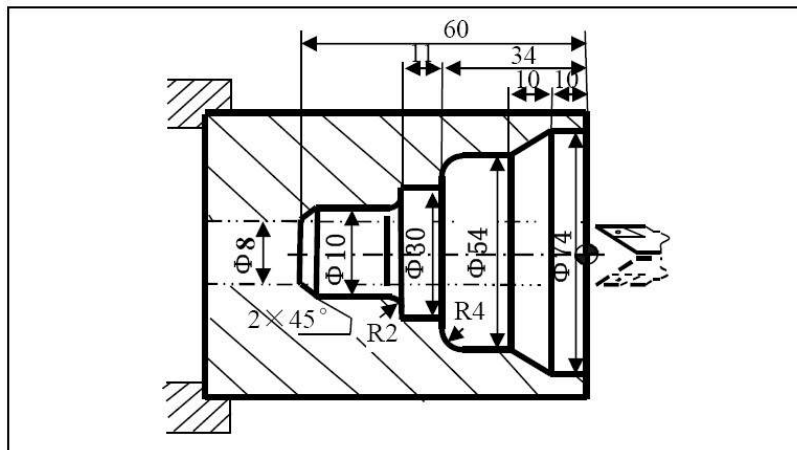
%3329

N1 T0101;   Set coordinate system

N2 G00 X100 Z80;   Move to starting point

N3 M03 S400;   Spindle rotates CW at 400r/min

N4 G00 X6 Z3;   Move to starting point of cycle

N5 G72W1.2R1P6Q16X-0.2Z0.5F100;   Machining in face roughing cycle

N6 G00 Z-61;   Contour finishing starts, move to extension line of chamfer

N7 G01 U6 W3 F80;   2×45°chamfer finishing

N8 W10;   Φ10 outer circle finishing

N9 G03 U4 W2 R2;   R2 arc finishing

N10 G01 X30;   Z45 face finishing

N11 Z-34;   Φ30 outer circle finishing

N12 X46；   Z34 face finishing

N13 G02 U8 W4 R4;   R4 arc finishing

N14 G01 Z-20;   Φ54 outer circle finishing

N15 U20 W10;   Cone finishing

N16 Z3;   Φ74 outer circle finishing, contour finishing end

N17 G00 X100 Z80;   Return to tool setting position

N18 M30;   Spindle stop, main program end and reset

⚠️ **Note**

(1) The rough turning cycle is realized by the G72 command with addresses P and Q. The F, S and T functions specified in the motion command are invalid. But the F, S and T functions specified in the G72 block or the previous block are valid.

(2) When the constant speed limit is used for cutting speed control, the G96 or G97 specified in the movement command between "ns" and "nf" is invalid. It is valid to specify G96 or G97 in the G72 block or the previous block.

(3) The block between sequence numbers "ns" and "nf" cannot call subprograms.

(4) The block trajectory between "ns" and "nf" must be gradually increased or decreased in X direction. Non-monotonously increasing or decreasing contour shape processing is not currently supported.

(5) In principle, the cycle starting point on Z-axis must be higher than the highest Z point of the contour; (the cycles tarting point on Z can be controlled to be 0 to 1mm lower than the highest point of the contour through the Machine User Parameters 010161).

(6) G72 command cannot be the same line as the M code.

(7) In the any line with scanning mode, the blocks between ns and nf cannot be specified

17.3　Closed Turning Multiple Repetitive Cycle (G73)

Function and Purpose

This cycle function is suitable for the machining of blanks with rough outlines, such as forging and casting parts. When processing this kind of blank parts, this function can effectively achieve a relatively balanced margin.

Command Format

G73 U(ΔI) W(ΔK) R(r) P(ns) Q(nf) X(△x) Z(△z) F(f) S(s) T(t)；

Parameter	Meaning
W	Cutting depth, without sign. The direction is determined by vector AA′
R	Retraction amount
P	Sequence number of the first program block (AA' in the below figure) for finishing path
Q	Sequence number of the last program block (BB' in the below figure) for finishing path
X	Finishing allowance in X direction
Z	Finishing allowance in Z direction
F ST	F, S, T in G72 are valid in roughing, and F, S, T between block ns and nf are valid in finishing

Description

When this function is cutting the workpiece, the tool path is shown in the figure below. The tool gradually feeds to make the path gradually close to the final shape of the part, and finally cut into the shape of the workpiece. The finishing path is A→A'→ B'→B.
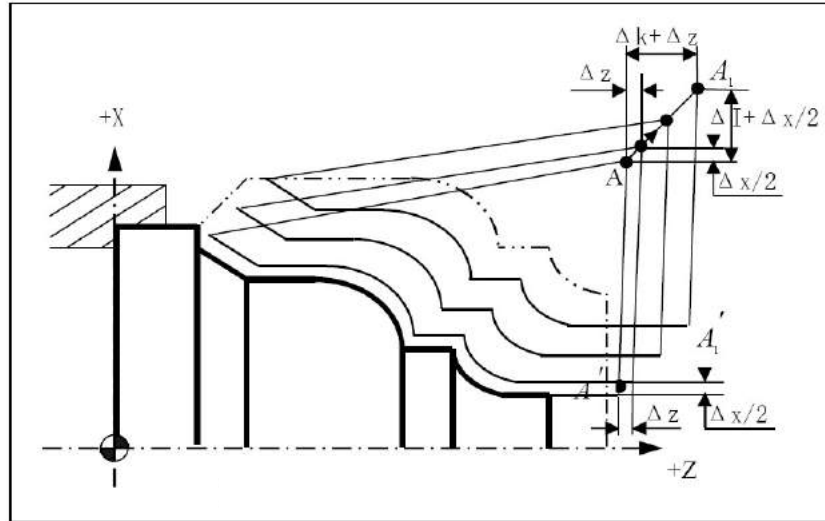
ΔI： Roughing allowance in X axis direction

Δk： Roughing allowance in Z axis direction

r： Number of rough cuts

ns： Sequence number of the first program block (AA' in the below figure) for finishing path

nf： Sequence number of the last program block (BB' in the below figure) for finishing path

Δx： Finishing allowance in X axis direction

Δz： Finishing allowance in Z axis direction

f，s，t： F, S, T in G73 are valid in roughing, and F, S, T between block ns and nf are valid in finishing

Note

ΔI and ΔK represent the total cutting amount during roughing, the number of roughing is r, the cutting amount in X and Z direction is ΔI/r and ΔK/r;

The cycle machining is realized based on the P and Q values in G73 block. Note the signs of $\triangle$x and $\triangle$z, $\triangle$I and $\triangle$K.

Example

Example 1: Set the starting point of cutting at A (60, 5); the rough machining allowances in the X and Z directions are 3mm and 0.9mm respectively; the number of rough machining is 3; the finishing allowances in the X and Z directions are 0.6mm and 0.1mm respectively. The dotted line part is the workpiece blank.
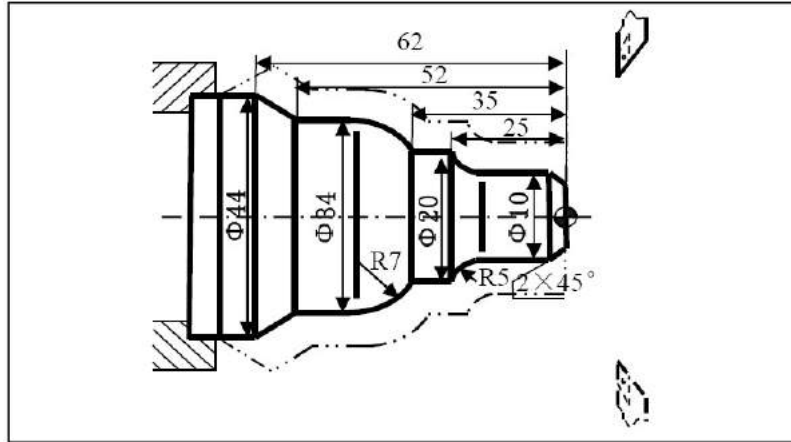
%0330

N1 T0101;   Set coordinate system, select No.1 tool

N2 G00 X80 Z80;   Move to starting point of program

N3 M03 S400;   Spindle rotates CW at 400r/min

N4 G00 X60 Z5;   Move to starting point of cycle

N5 G73U3W0.9R3P6Q13X0.6Z0.1F120;   Closed roughing cycle

N6 G00 X0 Z3;   Contour finishing starts, move to extension line of chamfer

N7 G01 U10 Z-2 F80;   2×45°chamfer finishing

N8 Z-20;   Φ10 outer circle finishing

N9 G02 U10 W-5 R5;   R5 arc finishing

N10 G01 Z-35;   Φ20 outer circle finishing

N11 G03 U14 W-7 R7;   R7 arc finishing

N12 G01 Z-52;   Φ34 outer circle finishing

N13 U10 W-10;   Cone finishing

N14 U10;   Exit machined surface, contour finishing end

N15 G00 X80 Z80;   Return to starting point of program

N16 M30;   Spindle stop, main program end and reset

⚠ Note

(1) The rough turning cycle is realized by the G73 command with addresses P and Q. The F, S and T functions specified in the motion command are invalid. But the F, S and T functions specified in the G72 block or the previous block are valid.

(2) When the constant speed limit is used for cutting speed control, the G96 or G97 specified in the movement command between "ns" and "nf" is invalid. It is valid to specify G96 or G97 in the G73 block or the previous block. The blocks between the sequence numbers na and nf cannot call subprogram.

(3) G72 cannot be the same line with M code.

(4) In the any line with scanning mode, the blocks between ns and nf cannot be specified.

(5) The number of divisions R can only be given as an integer value. If the specified value is a decimal, it will be rounded up.

17.4   Thread Cutting Multiple Repetitive Cycle (G76)

Function and Purpose

Threading is a forming processing, and the cutting amount per cut cannot be too large, so processing a threaded part often requires multiple processing to complete.

This command can use a cycle command to realize multiple reciprocating motions and finally complete the processing of threaded parts. It includes multiple roughing and at least one finishing.

Command Format

G76 C(c) R(r) E(e) A(a) X(x) Z(z) I(i) K(k) U(d) V(Δdmin) Q(Δd) P(p) F

Parameter	Meaning
C	Number of finishing (c=1 to 99), modal value
R	Undercut length on Z (The positive or negative R value determines the direction of undercut on Z axis), modal value
E	Undercut length on X (The positive or negative E value determines the direction of undercut on X axis), modal value
A	Tool nose angle (A value is a two-digit integer), modal value; the value is larger than 10° and smaller than 80°
X Z	Coordinate of the effective thread end point C in absolute programming; directional distance from the end point C of thread to starting point A of cycle in incremental programming
I	Radius difference between the starting point and the end point of thread cutting (The positive or negative I value determines the direction of taper); I=0 for straight thread, and I is negative for normal tapered thread
K	Thread cutting height (K is the radius, and is generally the thread tooth height)
U	Finishing allowance (d is the radius)

V	Min. cutting depth (Δdmin is the radius); when the nth cutting depth $\Delta d \sqrt{n} - \Delta d \sqrt{n-1}$ is smaller than Δdmin, the cutting depth is set to Δdmin
Q	The first cutting depth (Δ**d is the radius**)
P	Spindle rotation angle from the spindle reference pulse to the cutting start point
F	Thread lead (same as G32), F is in metric

Description

- For machining path of G76 thread multiple repetitive cycle, see the figure below,
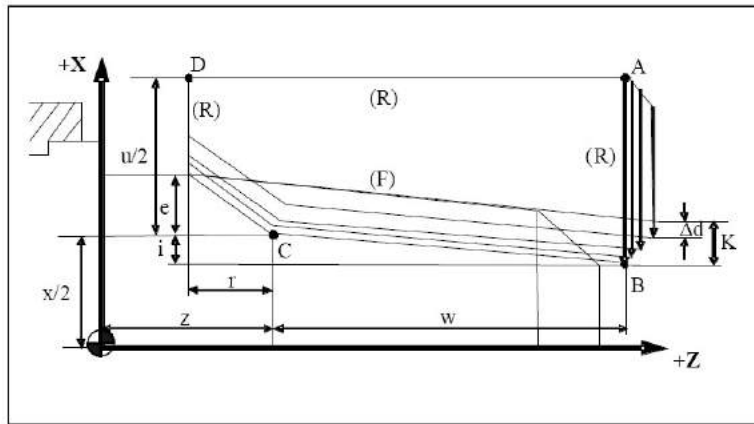
There are 4 paths for each reciprocation,

        1st path A→B, R rapid traverse feed

        2nd path B→C, F speed thread feed

        3rd path C→D, R rapid traverse feed (including E, R undercut path)

        4th path D→A, R rapid traverse feed

- Cutting depth setting for thread multiple repetitive cycle, see the figure below,
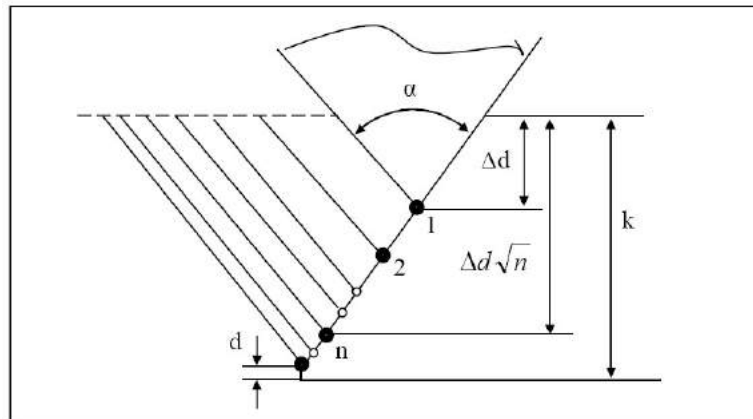
Example

Example 1: Use the thread cutting multiple repetitive cycle G76 for programming. The thread to be processed is ZM60×2, the workpiece size is shown in the figure below, and the size in brackets is obtained according to the standard. (Tan1.79=0.03125).
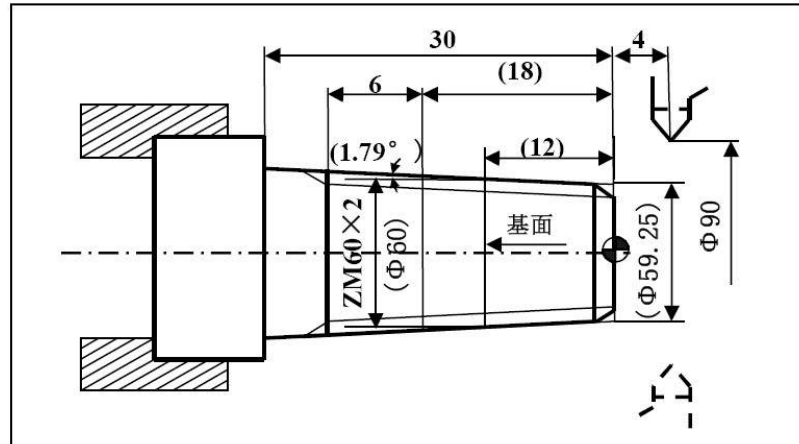
%3331

N1 T0101;   Select No. 1 tool, and determine the coordinate system

N2 G00 X100 Z100;   Move to the starting point of program or tool change position

N3 M03 S400;   Spindle rotates CW at 400r/min

N4 G00 X90 Z4;   Move to the starting point of simple cycle

N5 G80 X61.125 Z-30 I-1.063 F80;   Machine the outer surface of tapered thread

N6 G00 X100 Z100 M05;   Move to the starting point of program or tool change position

N7 T0202;   Change to No. 2 tool, and determine the coordinate system

N8 M03 S300;   Spindle rotates at 300r/min

N9 G00 X90 Z4;   Move to the starting point of thread

N10 G76C2R-3E1.3A60X58.15Z-24I-0.875K1.299U0.1V0.1Q0.45F2

N11 G00 X100 Z100;   Return to the end point of program or tool change position

N12 M05;   Spindle stop

N13 M30;   Main program end and reset

Note

(1) The cycle processing is realized based on the X(x) and Z(z) commands in the G76 block. In

incremental programming, note the sign of u and w (determined by the direction of the tool path AC and CD);

(2) G76 is used to perform the single-side cutting, which reduces the force on the tool tip. The cutting depth for the first cut is Δ d, the total cutting depth for the nth cut is $\Delta d \sqrt{n}$, and the amount of cutting depth in each cycle is $\Delta d \ (\sqrt{n} - \sqrt{n-1})$;

(3) In the single-side cutting diagram, the cutting speed from point B to C is specified by the thread cutting speed, and other paths are the rapid traverse feed.

(4) During thread cutting, the feed hold and reset bottons are invalid;

(5) During thread cutting, feed override is invalid;

(6) To avoid tool interference, the starting point of the thread cutting cycle should be set above the apex of the thread pitch;

(7) When the tapered thread cutting is executed, the thread pitch is calculated in the way of axis, and the actual processing speed is calculated in the way of generatrix.

18 Programming Simplifying Function (M)

18.1 Mirroring Function（G24，G25）

Function and Purpose

When editing a symmetrical shape processing program, user only needs to program any one side of the shape for the processing of the other side of the shape, which can save the time required for programming. At this time, the most effective function is the mirroring function.

For example, as shown in the figure below, when there is a program for processing the shape A on the left, the shape B that is symmetrical to the A can be completed on the right by mirroring the program.
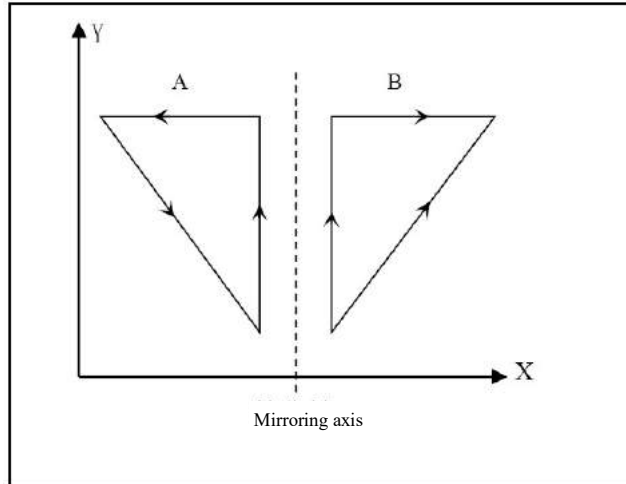
Mirroring axis

Command Format

G24 IP_; Establish mirroring

…… Tool path programming commands

G25 IP0; Cancel mirroring

Parameter	Meaning
IP	Mirroring axis position

Description

(1) In G24, the mirroring axis and mirroring enter coordinates are specified in absolute or incremental programming.

(2) In G24, user can specify axisymmetric mirroring and point-symmetric mirroring.

    a)   Axisymmetric mirroring

    b)   (G17/G18/G19)   G24 α_/β_;

    c)   …… ;

    d)   G25;

Parameter	Meaning
G17/G18/G19：	Plane selection for mirroring, this plane should include programmed tool path
G24 α_/β_：	Symmetry axis of mirroring. Only one of α_ and β_ can be specified. α represents the first axis of the selected plane, and β represents the second axis of the selected plane. If an axis on a non-selected plane is specified, the

	system alarms.
……:	Programmed tool path commands
G25α0/β0:	Cancel mirroring. When running the program, only write G25, or α and β after G25 are specified as any values, the mirroring function can be cancelled.

Point symmetry mirroring

(G17/G18/G19) G24 α_ β_;

…… ;

G25;

Parameter	Meaning
G17/G18/G19:	Plane selection for mirroring, this plane should include the programmed tool path
G24 α_β_:	Symmetry axis of mirroring. When α_ or β_ is omitted, the current tool position is defaulted. If an axis on a non-selected plane is specified, the system alarms.
……:	Programmed tool path commands
G25 α0 β0:	Cancel mirroring. When running the program, only write G25, or α and β after G25 are specified as any values, the mirroring function can be cancelled.

The diagram of axisymmetric and point-symmetric mirroring is as follows:
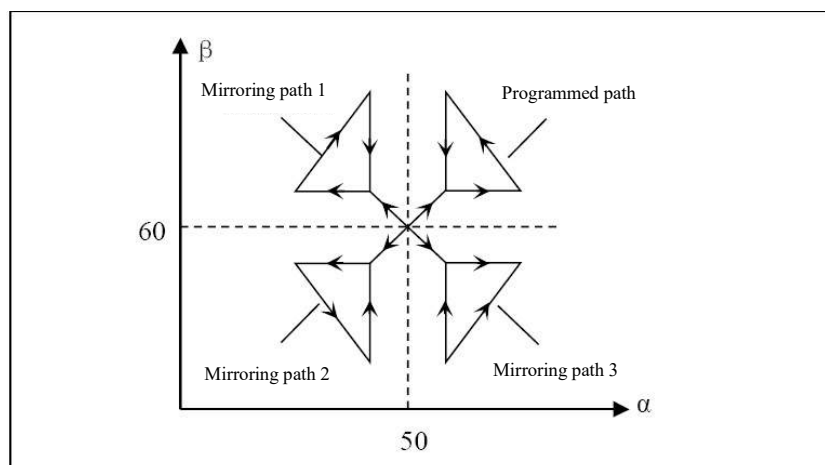
A mirror image path 1 and the programmed path are axisymmetrical, the axis of symmetry is α=50;

B mirror image path 2 and the programmed path are point-symmetrical, and the

symmetrical point is (50, 60);

C mirror image path 3 and the programmed path are axisymmetric, and the symmetry axis is β=60;

(3) By specifying G24 α_, the β-axis symmetry mirror can be established.

(4) When the β-axis mirror image is established, specify G25 α0 to cancel the β-axis mirror image. For example, when G24 X0 Y0 is specified to establish the point-symmetric mirroring, user can cancel the Y-axis symmetrical mirroring and only specify the X-axis symmetrical mirroring by specifying G25 X0.

(5) When only mirroring is specified on the first axis of the specified plane, the rotation direction and compensation direction in arc, tool radius compensation, and coordinate rotation are all reversed.

(6) The mirror image center of the local coordinate system will move due to the preset coordinate system and the change of the workpiece coordinate.

(7) G24 block and G25 block must be specified seperately.

(8) G24 is a modal command, can be canceled by G25 after mirroring function is finished.

(9) When no axis follows G25, all mirroring is canceled.

(10) When the reference point return command (G28, G30) is executed in mirror image, the mirror image is valid in the motion before the intermediate point is reached, so the mirror image is not executed in the motion from the intermediate point to the reference point.
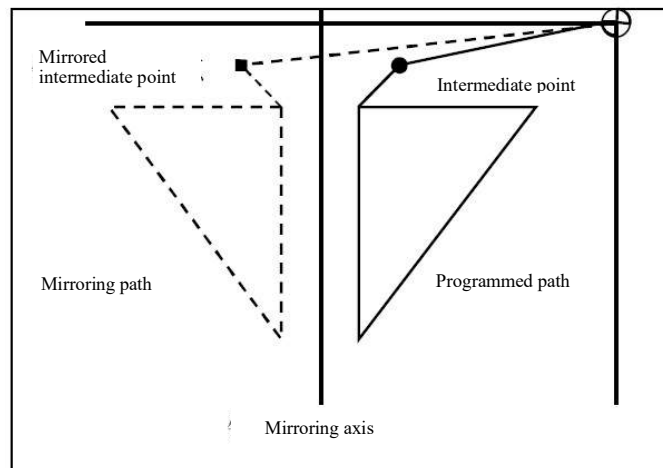
(11) When the return from origin command (G29) is specified in the mirroring, the mirroring is performed to the intermediate point.

(12) The mirror image is not executed to G53.

(13) The mirror image is processed after the tool radius compensation (G41, G42) is executed, so the cutting path as shown in the figure below is performed.

Programmed path

— ∙ — ∙ — Path when only the tool compensation is executed

———— Path when only the mirroring is executed

— ∙∙ — ∙∙ — Path when both mirroring and tool compensation are executed
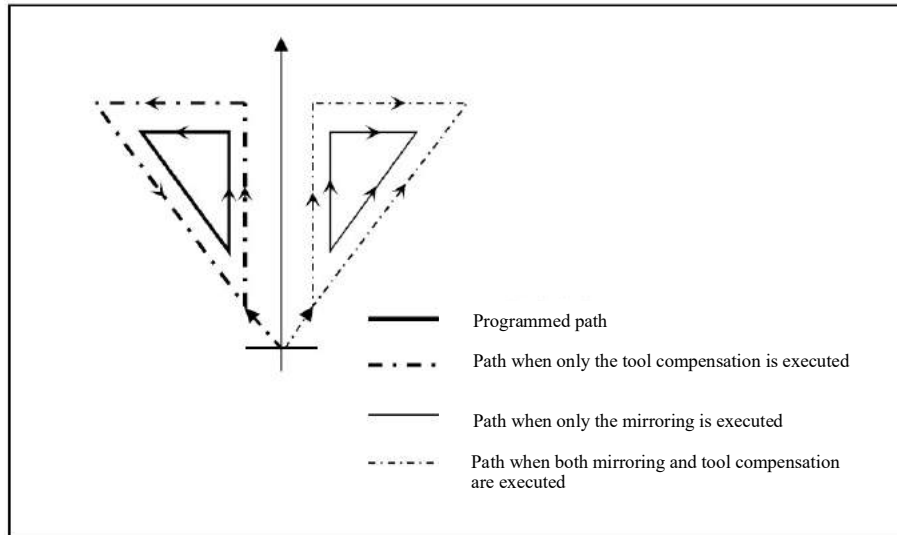
Example

Use the mirror image function to program the contour as shown in the figure: set the starting point of the tool to be 100mm from the upper surface of the workpiece, and the depth of cut is 5mm.
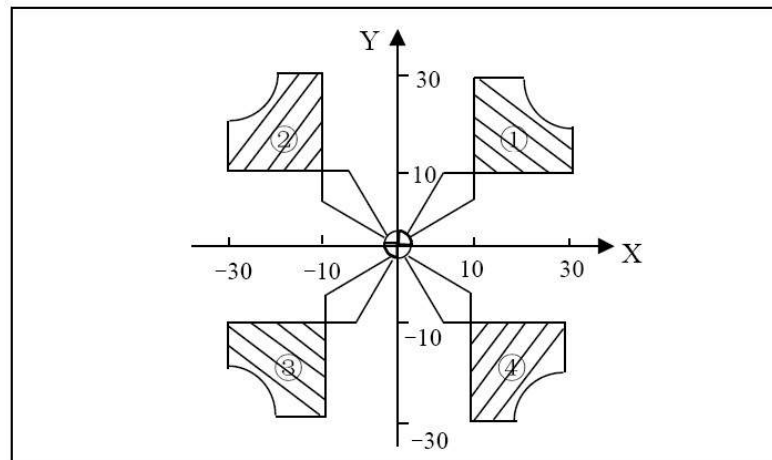
%3331   ； Main program
G92 X0 Y0 Z100
G91 G17 M03 S600
M98 P100   ； Machining ①
G24 X0   ； Y-axis mirror image, the mirroring position is X=0
M98 P100   ； Machining ②
G24 Y0   ； X, Y axis mirror image, the mirroring position is (0, 0)
M98 P100   ； Machining ③
G25 X0   ； X-axis mirroring continues to be effective, cancel Y-axis mirroring
M98 P100   ； Machining ④
G25 X0 Y0   ； Cancel mirroring

M30

%100   ；Subprogram (processing program of figure ①)

N100 G41 G00 X10 Y4 D01

N120 G43 Z10 H01

N130 G01 G90 Z-3 F300

N140 G91 Y26

N150 X10

N160 G03 X10 Y-10 I10 J0

N170 G01 Y-10

N180 X-25

N185 G00 Z10

N190 G90 G49 G00 Z100

N200 G40 X0 Y0

N210 M99

18.2　Scaling Function（G50，G51）

![icon] **Function and Purpose**

When executing the scaling function, the programmed path is enlarged or reduced based on the given scaling factor.

![icon] **Command Format**

G51 IP_ P_ ；　Scaling starts

……

G50　　　　；　Cancel scaling

Parameter	Meaning
IP	Specify the coordinates of the scaling center point. If it is not specified, specify the current point as the scaling center point. This command always specifies the absolute position of the scaling center in the workpiece coordinate system.
P	Specify the scaling factor of each axis. All axes are scaled by this factor.

![icon] **Description**

(1) Designation of scaling axis, scaling center and magnification

After issuing the G51 command, the scaling mode is entered. The G51 command only specifies the scaling axis, scaling center and magnification and doesn't specify the movement.

Although executing the G51 command will enter the scaling mode, in fact the effective axis for scaling is only the axis for which the scaling center is set.

(a) Scaling center

The center of the scaling is specified based on the absolute/incremental mode (G90/G91). In the G51 block, whether it is in incremental or absolute mode, the center of the scaling refers to the absolute position in the workpiece coordinate system.

Even if the current position is the center, it must be specified.

The effective axis for scaling is limited to the axis for which the scaling center is specified.

(b) Scaling magnification

The scaling magnification is specified with the address P.

The magnification for scaling only can be specified after G51.

Scaling magnification ranges from 0.000001 to 99999999999

When the scaling magnification is not specified, 1 is specified by default.

(c) A program error will occur in the following situations.

The scaling magnification exceeds the upper limit of the magnification command range.

(2) Scaling is cancelled. After specifying G50, the scaling is cancelled.

Example

Example 1: Use the scaling function to create the program with 1/2 scaling and the program without scaling in the figure below.
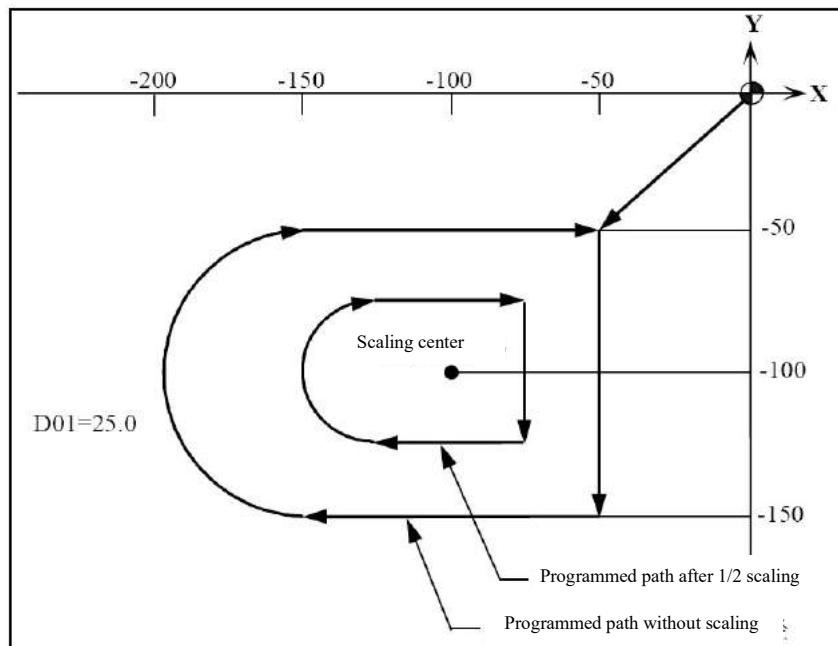

```
%1234
G92 X0 Y0 Z0；
G90 G51 X-100. Y-100. P0.5；
G00 G43 Z-200. H02；
G41 X-50. Y-50. D01；
G01 Z-250. F1000；
Y-150. F200；
X-150.；
G02 Y-50. J50.；
G01 X-50.；
```

G00 G49 Z0；

G40 G50 X0 Y0；

G90 G51 X-100. Y-100. P1；

G00 G43 Z-200. H02；

G41 X-50. Y-50. D01；

G01 Z-250. F1000；

Y-150. F200；

X-150.；

G02 Y-50. J50.；

G01 X-50.；

G00 G49 Z0；

G40 G50 X0 Y0；

M30;

Example 2: Use the scaling function to create the program of the contour as shown in the figure below: The vertices of the triangle ABC are A (10, 30), B (90, 30), C (50, 110), and the triangle A'B'C 'is the zoomed one, where the scaling center is D (50, 50), the scaling factor is 0.5 times, and the starting point of the tool is 50mm from the upper surface of the workpiece.
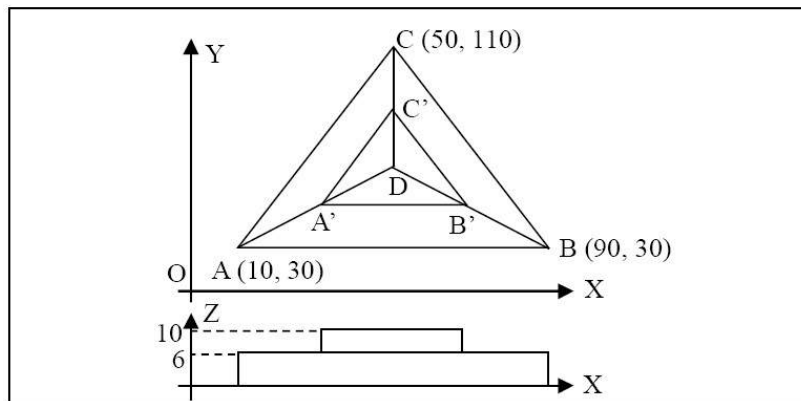
```
%3332                   ;  Main program
G92 X0 Y0 Z60
G17 M03 S600 F300
G43 G00 Z14 H01
X110 Y0
#51=0
M98 P100                ;  Machine triangle ABC
#51=6
G51 X50 Y50 P0.5        ;  Scaling center (50,50), scaling factor 0.5
M98 P100                ;  Machine triangle A'B'C'
G50                     ;  Cancel scaling
G49 Z60
G00 X0 Y0
M30
%100                    ;  Subprogram (the processing programi of triangle ABC)
N100 G41 G00 Y30 D01
N120 Z[#51]
N150 G01 X10
N160 X50 Y110
N170 G91 X40 Y-80
N180 G90 Z[#51]
N200 G40 G00 X110 Y0
N210 M99
```

⚠ Note

(1) When there is tool compensation, perform scaling first, and then tool radius compensation and tool length compensation. The scaling does not change the tool radius compensation value and tool length compensation value.

(2) Scaling is only valid for traverse commands in auto mode. It is invalid for the traverse in JOG. When the movement is in JOG mode, the breakpoint must be returned first.

(3) Scaling is valid to the axes of which X, Y, Z are specified, and invalid to the axes of which X, Y, Z are not specified.

(4) The G51 block must be specified separately.

(5) If M02 or M03 command is specified in scaling mode, or NC reset is executed, the scaling is canceled.

(6) When the coordinate system is offset (G92, G52 commands) or switched in the scaling mode, then the scaling center will be offset based on the coordinate system offset amount.

(7) When G28, G30, or G29 command is secified in scaling mode, the scaling is not canceled.

(8) When the G51 command is issued in the scaling mode, the axis of the newly-specified center becomes the effective scaling axis. The magnification is based on the newly specified G51.

(9) When G60 (unidirectional positioning) is issued in scaling mode, neither the final positioning point nor the crawling amount is scaled.

(10) When the graphics rotation is specified in the scaling mode, the graphics rotation center and the radius will be scaled.

(11) If the scaling command is issued in the graphics rotation subprogram, the radius of the graphics rotation is not scaled, but only the shape determined by the subprogram is scaled.

18.3　Rotation Transformation（G68，G69）

Function and Purpose

The rotation transformation is used to rotate the programmed processing path around the rotation center by a specified angle. If the shape of the workpiece is composed of many identical graphics, the graphics unit can be compiled into a subprogram which is then called by the rotation transformation command of the main program.

Command Format

G17/G18/G19；　　Select a rotation plane

G68 IP_ P_；　　Establish rotation transformation

……

G69；　　　　　　Cancel rotation transformation

Parameter	Meaning
IP	To specify coordinates of the rotation center. If it is not specified, it is the current point of the tool. The absolute position in the workpiece coordinate system whether it is in absolute or incremental mode
P	Rotation angle (unit: degree), counterclockwise rotation indicates a positive value, clockwise rotation indicates a negative value

Description

(1) The coordinate of the rotation center are always specified as an absolute value. Even if it is specified as an incremental address, it will not be treated as an incremental value. Regardless of the rotation angle specified in G90 or G91, P is always the absolute value of the angle of the first axis in positive direction in the specified reference plane.

(2) If the coordinate of the rotation center is omitted, the position where the G68 command is located will become the rotation center.

(3) The value of rotation angle P ranges from -360 to 360. The value is positive for counterclockwise direction, and negative for clockwise direction. When the angle command exceeds 360 degrees, the system will issue that the parameter is illegal.

(4) The rotation angle P is a modal value and will not change until a new angle is specified next time. The rotation angle can be omitted. If the rotation angle is omitted when G68 is specified for the first time, P will be regarded as 0. If the rotation angle is omitted when G68 is not specified for the first time, P will inherit the previous rotation angle.

(5) Use G69 to cancel it after the rotation transformation is completed. If M02 or M30 is specified in the coordinate rotation mode or the reset button is pressed, the coordinate rotation will be cancelled.

(6) In coordinate rotation mode, G68 is not displayed on the modal information screen, and G69 is not displayed after the mode is cancelled.

(7) The coordinate rotation function is valid in both auto and single-block working modes.

Example

Use the rotation function to program the contour as shown in the figure: the starting point of the tool is 50mm from the upper surface of the workpiece, and the depth of cut is 5mm.
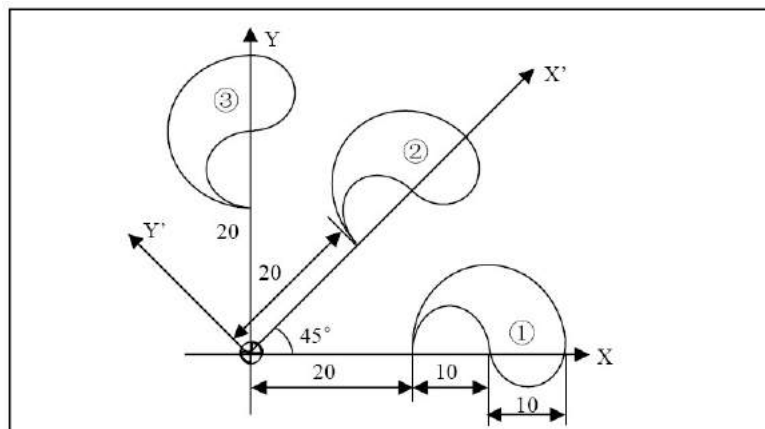
```
%3333          ；Main program
N10 G92 X0 Y0 Z50
N15 G90 G17 M03 S600
N20 G43 Z-5 H02
N25 M98 P200          ；Machine ①
N30 G68 X0 Y0 P45        ；Rotate 45°
N40 M98 P200          ；Machine ②
N60 G68 X0 Y0 P90        ；Rotate 90°
N70 M98 P200          ；Machine ③
N20 G49 Z50
N80 G69 M05 M30       ；Cancel rotation
%200                  ；Subprogram (Program of graphic ①)
G41 G01 X20 Y-5 D02 F300
N105 Y0
N110 G02 X40 I10
N120 X30 I-5
N130 G03 X20 I-5
N140 G00 Y-6
N145 G40 X0 Y0
N150 M99
```

![warning icon] **Note**

(1) The traverse command following G68 and G69 must be an absolute value command.

(2) In coordinate rotation mode, G codes related to the reference point (G28, G29, G30, etc.) and commands used to change the coordinate system (G52, G54 to G59, G54X, G92, etc.) cannot be specified. Otherwise, the coordinate rotation command needs to be cancelled first.

(3) G68 and G69 are specified in the tool radius compensation mode, and the rotation plane must be consistent with the tool radius compensation plane.

(4) If the coordinate rotation command is designated without the coordinate rotation specifications, it will be executed according to the detailed instructions (2) and (3).

(5) The G68 block must be specified separately.

(6) Tool radius compensation, tool length compensation, tool offset and other compensation operations are operated after the coordinate rotation is performed. When rotation and scaling are needed, user should enable the rotation function first and then the scaling function, otherwise system will prompt "nesting order change error".

19 User Macro and Subprogram Calling

19.1 User Macro Program

Function and Purpose

A user macro program is a program with specific control functions created by variables, logic operations, and control commands.

User macro program is a programming method similar to high-level language, which allows users to use variables, logic operations and conditional transfer and other control commands, makeing it more convenient to create the same program than traditional methods. At the same time, some same processing operations can be programmed into general programs with macro programs for users to call in cycles, as shown in the figure below for bolt hole processing.
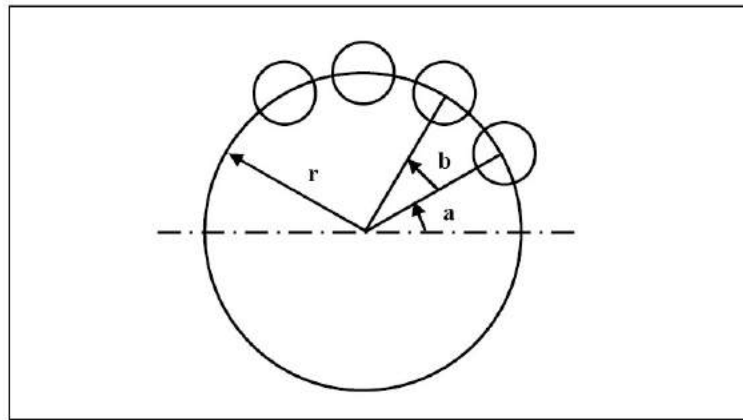
After storing the bolt hole machining program in the above figure created by the macro program in the CNC, user can call this program to machine the bolt hole at any time, and just fill in the bolt hole attributes such as the number of holes and deviation angle when calling. This is as if a bolt hole function has been added to the CNC.

19.1.1 Variable

Function and Purpose

A certain address in the program is not directly specified with a numerical value, but with a variable. When the program is running, the variable is assigned based on a predetermined situation to improve the generality of the program.

In the macro program, user can use variables in the parameters of the axis movement distance, such as G01 X[#54]F1000, at this time #54 is the variable, and user can perform some operations such as assignment before calling.

Command Format

#○○○=□□□□

#○○○=[Expression]

Variable representation	Description	Example
#A	A is composed of numbers from 0 to 9	#21
#B（expression）	Value A	#23
	Expression Operation command Expression	#1+#2
	Function [expression]	COS[#3]

Note

It is not allowed to use variable names directly in user macro programs. Variables are designated by the variable symbol (#) and the following variable number.

Description

Variable types

According to the purpose of variables, variables can be divided into local variables, global variables, and system variables. In addition, the access attributes of different variables are also different. Some variables are read-only, and some are readable and writable.

Constant

Some constants with constant values are defined in the system for users to use. The attributes of these constants are read-only.

PI：Pi π

TRUE：True, used for conditional judgment, indicating that the condition is ture

FALSE： False, used for conditional judgment, indicating that the condition is not ture

Note

When the constant PI is used, due to the calculation error, it needs to be dealt with at the time of ending conditions, otherwise an abnormal situation will occur.

Local Variable

Local variables refer to the variables used in the macro program. The local variable #i (i is a value, such as #10) used in macro program A is called in the current state, which is different with the #i used in macro program A called in other states. Therefore, when multiple levels of macros are called, such as calling macro B from macro A, if the local variable used in macro A is incorrectly used in macro B, the value will be destroyed.

The system provides #0 to #49 as current local variables, and their access attributes are readable and writable.

The system provides 6 levels of nesting. The corresponding local variables of each level are as follows. The access attributes of these local variables are readable:

- ➢ #200～#249　Level 0 local variable
- ➢ #250～#299　Level 1 local variable
- ➢ #300～#349　Level 2 local variable
- ➢ #350～#399　Level 3 local variable
- ➢ #400～#449　Level 4 local variable
- ➢ #450～#499　Level 5 local variable

Global variable

Different from local variables, global variables are commonly used between subprograms and macro programs, and their values remain unchanged. That is, #i used in one macro is the same as #i used in other macros. In addition, the public variable #i calculated by a certain macro can be used in other macros.

The system provides #50 to #199 as global variables, and their access attributes are readable and writable.

System variable

System variables are variables whose purposes are fixed in the system. There are 3 types of attributes for the system variables: read-only, write-only, and read/write. The attributes varies with system variables.

Undefined variable

The value of variables not defined in the system defaults to 0

Example：%1234
G54
G90G01 X10Y10F1000
X[#1]Y40　　; The coordinate value of the workpiece coordinate system is (0, 40)
M30

User-defined variable

User-defined variable：500 to 999 50000 to 54999		
#500 - #999	R/W	Global variable
#50000 - #54999	R/W	Global variable

Note

When machine user parameter 010091 "#500 - #999 user macro variable enable" is 1, #500 - #999 are valid user-defined variables. User-defined variables are saved after power off. "USERMACCFG.XML" needs to be configured (configured based on usage requirements, too many variables configured will occupy too much system storage space), and the configured file is correctly imported into the system in the "Data Management-User Macro Variable Name" interface.

USERMACCFG.XML（the file comes from the internal CNC system）：

```
<?xml version="1.0" encoding="GB2312"?><USERMACCFG version="1.0">
<item no="500" name="用户宏变量 1" type="FLOAT" />
<item no="501" name="用户宏变量 2" type="FLOAT" />
<item no="502" name="用户宏变量 3" type="FLOAT" />
<item no="503" name="用户宏变量 4" type="FLOAT" />
<item no="504" name="用户宏变量 5" type="FLOAT" />
<item no="505" name="用户宏变量 6" type="FLOAT" />
   ...
<item no="999" name="用户宏变量 500" type="FLOAT" />

<!--相关用户宏-->
<item no = "50000" name = "用户宏变量" type= "INT"></item>
<item no = "50001" name = "用户宏变量" type= "INT"></item>
<item no = "50002" name = "用户宏变量" type= "INT"></item>
<item no = "50003" name = "用户宏变量" type= "INT"></item>
<item no = "50004" name = "用户宏变量" type= "INT"></item>
<item no = "50005" name = "用户宏变量" type= "INT"></item>
...
<item no = "50100" name = "用户宏变量" type= "INT"></item>
</USERMACCFG>
```

Note

The numeric format in type needs to be configured correctly (FLOAT represents the data type of floating point, and INT represents the data type of integer), otherwise the macro program cannot run normally.

Channel-related variables

Variable number	Attribute	Description
Channel parameter		
Channel 00：（00000 to 03999)		
#0～#49	R/W	Current local variable
#50～#199		Reserved
#200～#249	R	Level 0 local variable
#250～299	R	Level 1 local variable
#300～#349	R	Level 2 local variable
#350～#399	R	Level 3 local variable

#400～#449	R	Level 4 local variable
#450～#499	R	Level 5 local variable
#1000～#1008	R	Machine position of current channel axis (9-axis)
#1009	R	Diameter programming of lathe
#1010～#1018	R	Programmed machine position of current channel axis (9-axis)
#1019		Reserved
#1020～#1028	R	Programmed workpiece position of current channel axis (9-axis)
#1029		Reserved
#1030～#1038	R	Workpiece origin of current channel axis (9-axis)
#1039	R	Coordinate system
#1040～#1048	R/W	G54 origin of current channel axis (9-axis)
#1049	R	G54 axis mask
#1050～#1058	R/W	G55 origin of current channel axis (9-axis)
#1059	R	G55 axis mask
#1060～#1068	R/W	G56 origin of current channel axis (9-axis)
#1069	R	G56 axis mask
#1070～#1078	R/W	G57 origin of current channel axis (9-axis)
#1079	R	G57 axis mask
#1080～#1088	R/W	G58 origin of current channel axis (9-axis)
#1089	R	G58 axis mask
#1090～#1098	R/W	G59 origin of current channel axis (9-axis)
#1099	R	G59 axis mask
#1100～#1108	R	G92 origin of current channel axis (9-axis)
#1109	R	G92 axis mask
#1110～#1118	R	Break-off position of current channel axis (9-axis)
#1119	R	Break point axis mark
#1120～#1149	R/W	Modal variable of canned cycle
#1150～#1189	R	Groups 0 to 39 modal of G code
#1190	R	User-defined input
#1191	R	User-defined output
#1192～#1199		Reserved
#1200～#1209	R	AD input
#1210～#1219	R	DA output
#1220	R	M3/4/5
#1221	R	G94 F value
#1222	R	Tapping F value
#1223~#1226	R	Tapping spindle speed
#1227	R	Valid radius compensation D number

#1228	R	Valid length compensation H number
#1229	R	cmd_feed
#1300~#1308	R	Relative zero of current channel axis (9-axis)
#1309		Reserved
#1310~1318	R	Programmed machine position of current channel axis (9-axis)
#1319		Reserved
#1320~#1328	R	G28 intermediate point
#1329	R	G28 axis mask
#1330~#1338	R	G52 origin
#1339		Reserved
#1340~#1349	R	G31 measurement machine command
#1350~#1359		Reserved
#1360~#1369	R	G31 measurement machine actual
#1370~#1399		Reserved
#1400~#1408	R/W	G54 offset
#1409		Reserved
#1410~#1418	R/W	G55 offset
#1419		Reserved
#1420~#1428	R/W	G56 offset
#1429		Reserved
#1430~#1438	R/W	G57 offset
#1439		Reserved
#1440~#1448	R/W	G58 offset
#1449		Reserved
#1450~#1458	R/W	G59 offset
#1459~#3999		Reserved

Note

The variables corresponding to the origin and offset of the workpiece coordinate system G54 - G59 in the current channel are both readable and writable, and are saved after power off.

Variables related to the extended coordinate system

Variable number	Attribute	Description
Channel variable		
Channel 00：(40100 to 40639)		
#40100~#40108	R/W	G54.1P1
#40109~#40117	R/W	G54.1P2
#40118~#40126	R/W	G54.1P3
#40127~#40135	R/W	G54.1P4
#40136~#40144	R/W	G54.1P5
#40145~#40153	R/W	G54.1P6

#40154～#40162	R/W	G54.1P7
#40163～#40151	R/W	G54.1P8
#40172～#40180	R/W	G54.1P9
#40181～#40189	R/W	G54.1P10
#40190～#40198	R/W	G54.1P11
#40199～#40207	R/W	G54.1P12
#40208～#40216	R/W	G54.1P13
#40217～#40225	R/W	G54.1P14
#40226～#40234	R/W	G54.1P15
#40235～#40243	R/W	G54.1P16
#40244～#40252	R/W	G54.1P17
#40253～#40251	R/W	G54.1P18
#40262～#40270	R/W	G54.1P19
#40271～#40279	R/W	G54.1P20
#40280～#40288	R/W	G54.1P21
#40289～#40297	R/W	G54.1P22
#40298～#40306	R/W	G54.1P23
#40307～#40315	R/W	G54.1P24
#40316～#40324	R/W	G54.1P25
#40325～#40333	R/W	G54.1P26
#40334～#40342	R/W	G54.1P27
#40343～#40351	R/W	G54.1P28
#40352～#40360	R/W	G54.1P29
#40361～#40369	R/W	G54.1P30
#40370～#40378	R/W	G54.1P31
#40379～#40387	R/W	G54.1P32
#40388～#40396	R/W	G54.1P33
#40397～#40405	R/W	G54.1P34
#40406～#40414	R/W	G54.1P35
#40415～#40423	R/W	G54.1P36
#40424～#40432	R/W	G54.1P37
#40433～#40441	R/W	G54.1P38
#40442～#40450	R/W	G54.1P39
#40451～#40459	R/W	G54.1P40
#40460～#40468	R/W	G54.1P41
#40469～#40477	R/W	G54.1P42
#40478～#40486	R/W	G54.1P43
#40487～#40495	R/W	G54.1P44
#40496～#40504	R/W	G54.1P45
#40505～#40513	R/W	G54.1P46
#40514～#40522	R/W	G54.1P47
#40523～#40531	R/W	G54.1P48
#40532～#40540	R/W	G54.1P49

#40541～#40549	R/W	G54.1P50
#40550～#40558	R/W	G54.1P51
#40559～#40567	R/W	G54.1P52
#40568～#40576	R/W	G54.1P53
#40577～#40585	R/W	G54.1P54
#40586～#40594	R/W	G54.1P55
#40595～#40603	R/W	G54.1P56
#40604～#40612	R/W	G54.1P57
#40613～#40621	R/W	G54.1P58
#40622～#40630	R/W	G54.1P59
#40631～#40639	R/W	G54.1P60

Note

The variables corresponding to the origin and offset of the workpiece coordinate system G54~G59 in the current channel are both readable and writable, and are saved after power off.

Variables related to tool

Tool data：#70000 to #89999 Each tool occupies 200 numbers, a total of 100 tools, a total of 20,000 numbers Relative coding range of No. 0 tool：000 to 199 Relative coding range of No. 1 tool：200 to 399 Relative coding range of No. 99 tool：18000 to 19999		
#70005	R	Lathe tool nose direction
#70006	R/W	Tool length of milling cutter or X offset value of lathe tool
#70007	R	Y offset of lathe tool
#70008	R	Z offset of lathe tool
#70009		Reserved
#70010		Reserved
#70011	R/W	Tool radius of milling cutter or tool nose radius of lathe tool
#70012~#70028		Reserved
#70029	R/W	Length wear of milling cutter or Z offset wear of lathe tool
#70030		Y offset wear of lathe tool
#70034	R/W	Radius wear of milling cutter or X offset wear of lathe tool
#70035~#70100		Reserved
#70101	R	Tool life monitoring type
#70104	R	Max. life of cutting time
#70105	R	Life of pre-warning cutting time
#70106	R	Life of current cutting time

#70107	R	Life of max. cutting times
#70108	R	Life of pre-warning cutting times
#70109	R	Life of actual cutting times

Note

The variables corresponding to the tool radius compensation value, length compensation value, and wear value can be read and written, and can be saved after power off.

（F） **Class A command (T)**

#50000	X coordinate of tool measuring instrument calibration	#50003	Z coordinate of tool measuring instrument calibration
#[50006+N]	Avarage measured value of No. N tool on Z	#[51006+N]	Average measured value of No. N tool on X
#54005	Tool retraction amount	#54006	Tool nose angle A
#54007	Finishing times	#54008	(FUNAC) Q value
#54009	（F）R value	#54010	Undercut angle
#54011	Thread chamfer amount	#54012	G83 Axial drilling cycle H1/H2 Retraction mode
#54013	G83 Retraction amount	#54014	G84 Tapping axis selection
#54015	G84 H1/H2 Retraction mode	#54016	G84 Retraction amount
#54017	G88 Tapping axis selection	#54990	Measuring instrument center position X
#54992	Measuring instrument center position Z		

Note

The (F) Class A command (T) function in the CNC system occupies the above-mentioned user-defined macro variables. Please do not use the above-mentioned user-defined macro variables.

Workpiece measurement F

Variable number	Description	Variable number	Description
#600	Distance from actual center to trigger point in X positive direction	#630	Center position value in X plane or X direction
#601	Distance from actual center to trigger point in X negative direction	#631	Center position value in Y plane of Y direction
#602	Distance from actual center to trigger point in Y negative direction	#632	Z plane position value
#603	Distance from actual center to trigger point in Y positive direction	#633	Position offset value in X direction
#604	Probe length value	#634	Position offset value in Y direction
#605	Eccentricity of probe in X direction	#635	Position offset value in Z direction

#606	Eccentricity of probe in Y direction	#636	Dimension value: width/diameter
#607	Trigger radius of probe in X direction	#637	Dimension offset vlaue
#608	Trigger radius of probe in Y direction	#638	Angle value (unit: degree)
#609	Second measurement speed of probe		

Note

The workpiece measurement function in the CNC system occupies the above-mentioned user-defined macro variables. If the debugger uses other user-defined macro variables when debugging the workpiece measurement function, instead of the user-defined macro variables listed in the above table, the debugger will replace or add them to the table, and inform and provide it to the user. Please do not use the above user-defined macro variables.

Tool measurement

Variable number	Description	Variable number	Description
#642	Laser beam width of tool setter	#649	2nd measurement speed in tool setting
#643	Tool setter laser on X	#650	Center position of tool setter on X
#644	Tool setter on Y	#651	Center position of tool setter on Y
#645	Height difference 1 (laser height-base height)	#652	Absolute safety height of tool setter
#646	Height difference 2 (base height-workpiece height)	#653	Max. allowable value for tool break detection
#647	Rapid traverse speed in tool setting	#654	Tool number of datum tool
#648	1st measurement speed in tool setting	#660	Datum tool length—relative tool setter surface
		#661~#676	1~16 号刀，刀长相对对刀仪表面 No.1 to 16 tools,

Note

The workpiece measurement function in the CNC system occupies the above-mentioned user-defined macro variables. If the debugger uses other user-defined macro variables when debugging the workpiece measurement function, instead of the user-defined macro variables listed in the above table, the debugger will replace or add them to the table, and inform and provide it to the user. Please do not use the above user-defined macro variables.

19.1.2 Operation Command

Function and Purpose

Arithmetic operators, functions and other operations can be flexibly used in macro programs, which is very convenient to realize complex programming requirements. As shown in the table

below.

Operation type	Operation command	Meaning
Arithmetic operation	#i = #i + #j	Addition, #i plus #j
	#i = #i - #j	Subtraction, #i minus #j
	#i = #i * #j	Multiplication, #1 times #j
	#i = #i / #j	Division, #i divided by #j
Conditional operation	#i EQ #j	（＝）Equal to
	#i NE #j	（≠）Not equal to
	#i GT #j	（＞）Larger than
	#i GE #j	（≥）Larger than or equal to
	#i LT #j	（＜）Smaller than
	#i LE #j	（≤）Smaller than or equal to
Logic operation	#i = #i & #j	Logic and
	#i = #i | #j	Logic or
	#i = ~#i	Logic not
Function	#i= SIN[#i]	Sine (unit: radians)
	#i=ASIN[#i]	Arc sine
	#i=COS[#i]	Cosine (unit: radians)
	#i=ACOS[#i]	Arc cosine
	#i=TAN[#i]	Tangent (unit: radians)
	#i=ATAN[#i]	Arc tangent
	#i=ABS[#i]	Absolute value
	#i=INT[#i]	Round (round down)
	#i=SIGN[#i]	Sign
	#i=SQRT[#i]	Square root
	#i=#A POW[#i]	#A to the #i power
	#i=LOG[#i]	Log
	#i=PTM[#i]	Pulse revolutions mm
	#i=PTD[#i]	Pulse revolution degree
	#i=RECIP[#i]	Reciprocal
	#i=EXP[#i]	Exponent, an exponent based on e (2.718)
	#i=ROUND[#i]	Rounding
	#i=FIX[#i]	Round down
	#i=FUP[#i]	Round up

Note

When using trigonometric functions, pay attention to check "000349 trigonometric function selection, 0: radian, 1: angle", and adjust the calculation during programming based on the parameters.

Example

Example 1

A sum of 1 to 100

O1234

#1=0；................................. Initial value of solution

#2=1；................................ Initial value of addend

N1 WHILE[#2 LE 100] ；The addend cannot exceed 100, otherwise jump to N2 after ENDW

#1 =#1 + #2；........................... Calculate solution

#2 =#2 +1；............................. Next addend

ENDW；.................................. Transfer to N1

N2 M30；................................ Program end

19.1.3 Macro Statement

Function and Purpose

The following blocks are macro statements:

1) Blocks containing variables and operation commands;

2) Blocks containing conditional judgment statements or cycle statements;

3) Blocks containing macro program calling commands.

Description

Expression

Any calculation formulas including signs such as "+", "-", "*", "/", "[", "]", SIN, etc. are referred to as expressions. Shown as follows

1. -#4
2. SIN[#4+#5]*COS[[#4+#5]/#6]

Note

The priority in [] is higher than +-*/, for example [[#4+#5]/#6], first calculate [#4+#5], then calculate /#6. For expressions, in order to ensure the correctness of calculations, it is recommended to use [] for expressions, such as [-#5]. The usage such as -[#5] is not recommended.

Assignment statement

Transferring the value of a constant or an expression to a macro variable is referred to as an assignment, and this statement is referred to as an assignment statement, as follows:

#5 = 145 / SQRT[3] * COS[40*PI/180]
#6 = 123

Conditional judgement statement

Two kinds of conditional judgement statements are supported:

IF [Conditional expression];   Type 1
……
ENDIF

IF [Conditional expression];   Type 2
……
ELSE
……
ENDIF

For the conditional expressions in the IF statement, user can use simple conditional expressions or compound conditional expressions, as shown in the following example:

When #4 is equal to #5, assign 1 to #6
IF [#4 EQ #5]
#6 = 1
ENDIF

When #4 is equal to #5, and #6 is equal to #7, assign 2 to #6

IF [#4 EQ #5] AND [#6 EQ #7]
#6 = 2
ENDIF

When #4 is equal to #5, or #6 is equal to #7, assign 1 to #6, otherwise assign 2 to #6

IF [#4 EQ #5] OR [#6 EQ #7]
#6 =1
ELSE
#6 = 2
ENDIF

Cycle statement

The conditional expression is specified after WHILE. When the specified conditional expression is satisfied, execute the program from WHILE to ENDW, and execute the cycle until the conditional expression is not satisfied. When the specified conditional expression is not met, exit the WHILE cycle and execute the program blocks after ENDW.

The calling format is as follows,

WHILE [Conditional expression]

……

ENDW

Infinite cycle

When the conditional expression in WHILE is always true, an infinite cycle can be realized. For example,

WHILE [TRUE];or WHILE [1]
……
ENDW

Jump statement

GOTO _

Use GOTO to jump to the specified label

GOTO is followed by a number. For example, with GOTO 5, the system jumps to the N5 block (N5 must be written at the beginning of the block).

Nesting

For IF statements or WHILE statements, the system allows nested statements, but with certain restrictions, as follows:

IF statement supports up to 6 levels of nested calls, and the system will issue an error if 6 levels are exceeded;

The WHILE statement supports up to 6 levels of nested calls, and the system will issue an error if 6 levels are exceeded;

The system supports mixed use of IF statement and WHILE statement, but the matching relationship between IF-ENDIF and WHILE-ENDW must be met.

Example

Edit the ellipse processing program (ellipse expression: $X=A×COSα$; $Y=B×SINα$).
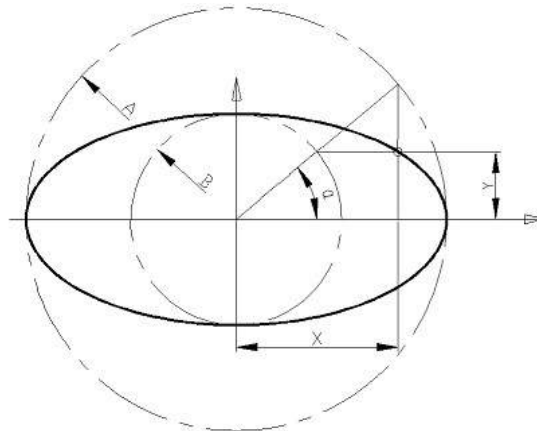

```
%0001
#0=5              ;   Define the tool radius R value
#1=20             ;   Define A value
#2=10             ;   Define B value
#3=0              ;   Define the initial value of the step angle α, unit: degree
G92 X0 Y0 Z10
M3S1000
G00 X[2*#0+#1] Y[2*#0+#2]
G01 Z0
G41 X[#1]D01F200
WHILE #3 GE [-360]
G01 X[#1*COS[#3*PI/180]] Y[#2*SIN[#3*PI/180]]
#3=#3-5
```

ENDW
G01X[#1]Y0
G01 G91 Y[-2*#0]
G90 G00 Z10
G40 X0 Y0
M30

19.2   Macro Program Calling

Function and Purpose

The system supports the following three ways to call the macro program:

(1) Non-modal call: G65

(2) Call macro program with G code

(3) Call macro program with M command

19.2.1 Argument Specification Rules

Function and Purpose

Argument specification rules

In the user macro program, when the argument needs to be transmitted as a local variable, the actual argument value must be specified after the address.

When the user calls the macro program, the system will copy the content of the arguments (A to Z) in the current block to the corresponding local variables #0 - #25 of the current layer of the corresponding user macro program, and also copy the absolute position of the nine axes (XYZABCUVW) of the workpiece coordinate system in the current channel to the local variables #30 - #38 in the current channel.

Macro variable	Argument name	Macro variable	Argument name	Macro variable	Argument name
#0	A	#1	B	#2	C
#3	D	#4	E	#5	F
#6	G	#7	H	#8	I
#9	J	#10	K	#11	L
#12	M	#13	N	#14	O
#15	P	#16	Q	#17	R
#18	S	#19	空	#20	U
#21	V	#22	W	#23	X
#24	Y	#25	Z	#26	Reserved
#27	Reserved	#28	Reserved	#29	Reserved
#30	X axis position	#31	Y axis position	#32	Z axis position
#33	A axis position	#34	B axis position	#35	C axis position
#36	U axis position	#37	V axis position	#38	W axis position

Macro variable is defined and determined

Format: AR[#Variable number]

Return value: 0: Indicates that the variable is not defined;

        90: Indicates that the variable is defined as absolute mode G90;

        91: Indicates that the variable is defined as incremental mode G91

Note: The system macro AR[] is used to determine whether the macro variable is defined and defined as incremental or absolute mode;

Example

```
%1234
G92X0Y0Z0
M98P9990X20Y30Z40
M30
%9990
IF [AR[#23] EQ 0] OR [AR[#24] EQ 0] OR [AR[#25] EQ 0]；    If the X, Y or Z value is not
                                                    defined, it returns.
M99
ENDIF
IF AR[#23] EQ 90   ； If X value is in absolute mode G90
#23=#23-#30          ； X value is converted to be in incremental mode, #30 is the absolute
coordinate of X
ENDIF
……
M99
```

19.2.2 Non-modal Call (G65)

Function and Purpose

When G65 is specified, the user macro program specified by the parameter P is called, and at the same time, the arguments and the variables needed by the user macro program are transferred to the user macro program.

Command Format

G65 P_ L_ [Argument address word]；

Parameter	Meaning
P	Program number required to be called
L	Number of repeated calls

Argument address word	Data that the user needs to transfer to the macro program

Note

（1）G65 is a non-modal command. User needs to specify G65 in this line every time the macro program is called;

Example

Example

%0032
G54G0X0Y0Z100
M3S1000
G65P100L5X50Y50Z-30R5F200
G00X50Z10
M30
%100
G01X[#23]Y[#25]F[#5]
G81Z[#25]R[#17]F[#5]
G0Z50
M30

19.2.3 Calling Macro Program with G Code

Function and Purpose

In addition to calling macro program with non-modal code (G65), user can also call the macro program in the form of G code. Currently, the macro program in the form of G code canned cycle can only be called for canned cycle. For the specific code, please refer to the drilling and milling chapter and this chapter 17.2.3.

Function

Call the user-defined subprogram in user canned cycle with G command.

Command Format

G_：

Parameter	Meaning
G	The number of the called subroutine in USERDEF.CYC (Arabic numerals)

Note:

1) The system provides G1000 to G1999 to call the user-defined subprogram in the canned cycle

with G command.

2）G1000 to G1999 respectively correspond to the called subprogram number (in Arabic numerals) %1000 to %1999 in USERDEF.CYC, for example, G1010 corresponds to the called subprogram number %1010 in USERDEF.CYC.

Example

Add user canned cycle %1010 in USERDEF.CYC

```
%1010;
G01 X30 Y0 F3000
#0 = 0.0
#1 = 30.0
#2 = 2.1
#3 = 3.4
WHILE [#0 LE 360]
G1 X[ #1 * COS[#0*PI/180]]Y[ #1 * SIN[#0*PI/180]]F3000
#0 = #0 + 0.1
ENDW
G01X30Y0F3000
G80
M99

%1244 Main program
G92X0Y0Z50
G01X30Y-20F3000
M3S3000
Z0F1000
G1010 (Call user-defined canned cycle)
G01Y20F3000
G00Z50
M30
```

19.2.4 Calling Macro Program with M Command

Function and Purpose

Calling macro program with M command is realized by calling a custom subprogram with an M code.

There are two forms as described in the "Command Format" below. For the macro program calling with M98, when M98 is executed, first call the subprogram number of the internal subprogram. If there is no such subprogram number, search for the number of the external subprogram. If there is no such subprogram in both, an error will be reported; for calling macro program with M code,

can the user-defined subprogram in the canned cycle can be called.

The corresponding M code parameter settings of user-defined subprogram in canned cycle are shown in the following figure. The user-defined parameters 010360 to 010373 respectively correspond to the %1007 to %1020 subprogram in USERDEF.CYC.

Command Format

M98 P_;

Parameter	Meaning
P	The program number to be called in the program

Note: when M98 is specified, the user macro program following P parameter is called.

M_;

Parameter	Meaning
M	The input value of user-defined parameter

Example

The parameter 010360 M Code Corresponding to User Canned Cycle G1007 is set as 13, that is, the %1007 program in USERDEF.CYC can be called by the M13 command in the processing program.

```
%1007; Add user-defined subprogram 1007 in the USERDEF.CYC file
G59
G64G01 X30 Y0 F3000
#0 = 0.0
#1 = 30.0
WHILE [#0 LE 360]
G1 X[ #1 * COS[#0*PI/180]]Y[ #1 * SIN[#0*PI/180]]F3000
#0 = #0 + 0.1
ENDW
G01X30Y0F3000
G80
M99


%1234; Main program
G54
G1X0Y0Z50
G01X30Y-20F3000
M3S3000
Z0F1000
M13; Call the matching 1007 subprogram with M13
```

G01Y20F3000

G00Z50

M30

Parameter type	Parameter	Parameter name	Value	Activation
Machine user parameter	010360	M code corresponding to user canned cycle G1007	13	Save
	010361	M code corresponding to user canned cycle G1008	0	Save
	010362	M code corresponding to user canned cycle G1009	0	Save
	010363	M code corresponding to user canned cycle G1010	0	Save
	010364	M code corresponding to user canned cycle G1011	0	Save
	010365	M code corresponding to user canned cycle G1012	0	Save
	010366	M code corresponding to user canned cycle G1013	0	Save
	010367	M code corresponding to user canned cycle G1014	0	Save
	010368	M code corresponding to user canned cycle G1015	0	Save
	010369	M code corresponding to user canned cycle G1016	0	Save
	010370	M code corresponding to user canned cycle G1017	0	Save
	010371	M code corresponding to user canned cycle G1018	0	Save
	010372	M code corresponding to user canned cycle G1019	0	Save
	010373	M code corresponding to user canned cycle G1020	0	Save

⚠ **Note**

(1) The above canned cycle, rotation/mirroring/scaling and G91 cannot be used at the same time.

(2) When using the M command to call a subprogram, it is necessary to add G80 before M99 at the end of the program.

19.2.5 Classification of Subprogram

Function and Purpose

Internal subprogram

The called program which is in the same file with the main program is referred to as internal subprogram.

External subprogram

The called program is stored separately in another file, which is referred to as external subprogram.

The file name of the external subprogram must start with the letter O.

Example

Example for internal subprogram

The G code file name is OTEST, and %111 is an internal subprogram, which is in the same file as the main program %1001 and is called by M98 in the main program.

```
%1001; Main program
G92 X0 Y0 Z50
G91 G01 Z10 F400
M98 P111; Call subprogram 111
G4X1
M30
%111; Subprogram
G01X10Y10Z10
…
G80
M99
```

Example for external subprogram

The G code file name is OTEST, and the subprogram file name is O123.

Main program	Subprogram 0123
%1001 G92 X0 Y0 Z50 G91 G01 Z10 F400 M98 P123; Call subprogram 0123 G4X1 M30	%1234; G01X10Y10Z10 … G80 M99

Canned cycle

There are two types of canned cycles. One is a general canned cycle, which is mainly used in turning, milling, drilling, etc., and the other is a user canned cycle, which is created based on the special needs of users.

Refer to Chapter 12 for the specific use of the general canned cycle.

User can add subprograms to the file of user canned cycle (USERDEF.CYC) as needed, and use the corresponding G command in the main program. (The correspondence between G command and user canned cycle subprogram number is shown in Chapter 14 and Chapter 15)

Open the user-defined cycle file USERDEF.CYC, index to the following contents, and add them in turn. For example, add user subprogram 1010.

```
%1010
G01X10Y10F1000
Z50
M99
```
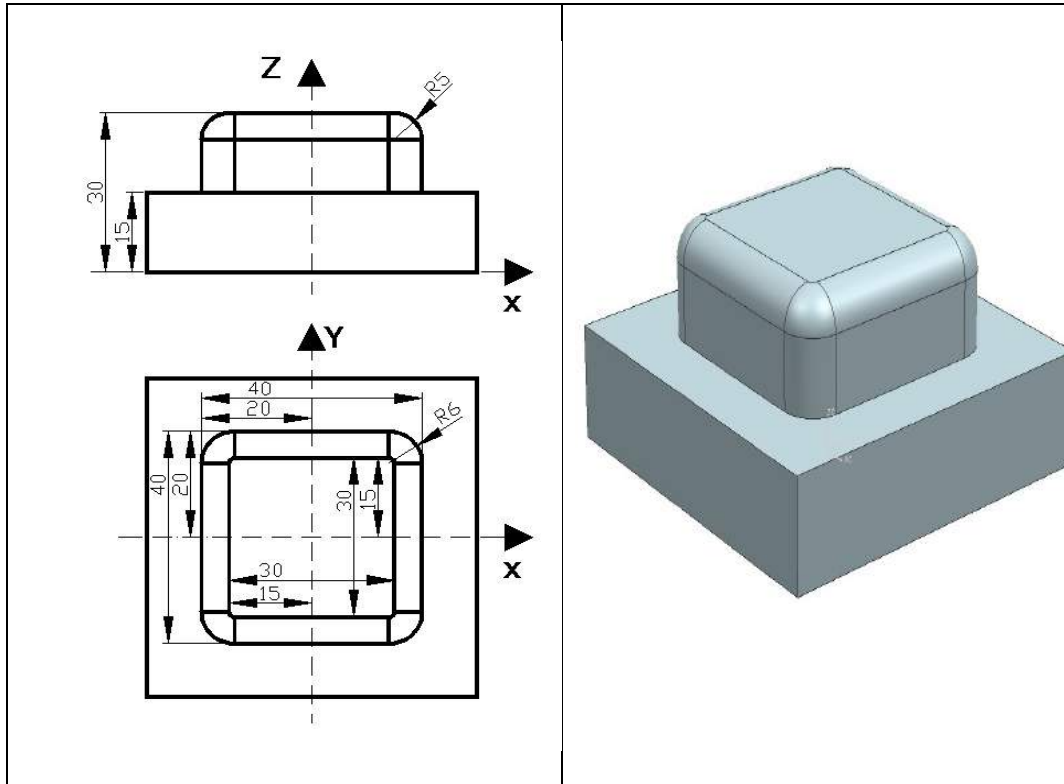
The main program uses command G1010 to call the above program.

19.2.6 Macro Program Example

Application example

As shown in the figure, use a ball-end milling cutter to process the R5 fillet surface.

%0001   （The tool position point is the ball center）
G92 X-30 Y-30 Z25
#0=5   （Fillet radius）
#1=4   （Radius of ball-end cutter）
#2=180   （The initial value of the step angle γ. Unit: Degree）
WHILE #2 GT 90
G01 Z[25+[#0+#1]*SIN[#2*PI/180]]   （Calculate Z axis height）
#3=ABS[[#0+#1]*COS[#2*PI/180]]-#0   （Calculate radius offset）
G10 L12 P3 R[#3]
G01 G41 X-20 D3
Y14
G02 X-14 Y20 R6
G01 X14
G02 X20 Y14 R6
G01 Y-14
G02 X14 Y-20 R6
G01 X-14
G02 X-20 Y-14 R6
G01Y30
G01 X-30
G40 Y-30
#2=#2-10
ENDW
M30

19.3   Manual Calling Subprogram

Function and Purpose

In JOG mode, user can customize the buttons on the MCP panel, set the G register to 1 by the button, and call the related suprograms, thereby realizing some more complex functions, such as tool change, spindle C/S switching, etc.

The correspondence of G register points to subprograms is shown in the following table

G register point	Canned cycle subprogram
G2637.1	Call canned cycle subprogram O1007
G2637.2	Call canned cycle subprogram O1008
G2637.3	Call canned cycle subprogram O1009
G2637.4	Call canned cycle subprogram O1010
G2637.5	Call canned cycle subprogram O1011
G2637.6	Call canned cycle subprogram O1012
G2637.7	Call canned cycle subprogram O1013
G2637.8	Call canned cycle subprogram O1014
G2637.9	Call canned cycle subprogram O1015
G2637.10	Call canned cycle subprogram O1016
G2637.11	Call canned cycle subprogram O1017
G2637.12	Call canned cycle subprogram O1018
G2637.13	Call canned cycle subprogram O1019
G2637.14	Call canned cycle subprogram O1020
G2637.15	Call canned cycle subprogram O1021
F2637.0	Status bit: subprogram is running in JOG mode

Description

(1) In PLC, the program can be created as shown below (user can customize the buttons based on the actual situation), the G command signal can be output, and the corresponding subprogram can be called.
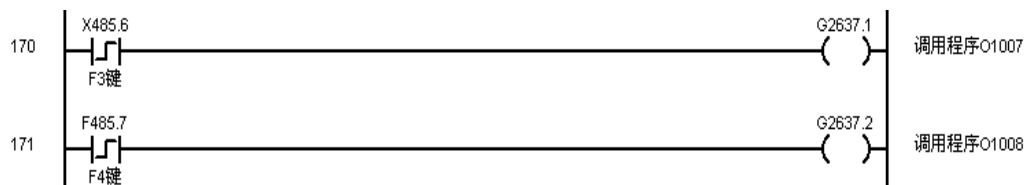
(2) In JOG mode, call the corresponding subprogram by clicking the button. Click the F3 button as shown in the figure above to call

O1007 subprogram.

Note

(1) During the execution of the program, F2637.0 is always 1, and the buttons on the panel are in the shielding state except for the reset and emergency stop buttons, and the clicks are invalid; when the corresponding G register signal (e.g. G2637.1) is 0, F2637.0 is 0 after the subprogram is executed.

(2) Currently, subprograms O1007 to O1021 must be placed in the file USERDEF.CYC. The subprogram corresponding to the O1007 program starts with %1007, and O1008 corresponds to %1008, and so on. In the line before M99 at the end of each subprogram, G80 must be added to clear the canned cycle mode.

(3) When the subprogram is called manually, the buttons on the MCP panel should be in the shielding state except for emergency stop button, but for the PLC output control buttons (such as spindle CW, spindle CCW, cooling, etc.), PLC needs to be changed to shield these buttons.

For example, shielding the function key of spindle CW:
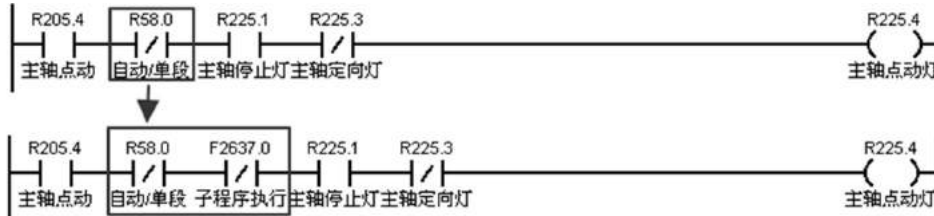
For the M command that is only executed in the auto/single-block mode, the normally open point of F2637.0 should be connected in parallel in the program, as shown in the figure below, to ensure that the M command can be executed normally during calling subprogram in JOG mode.
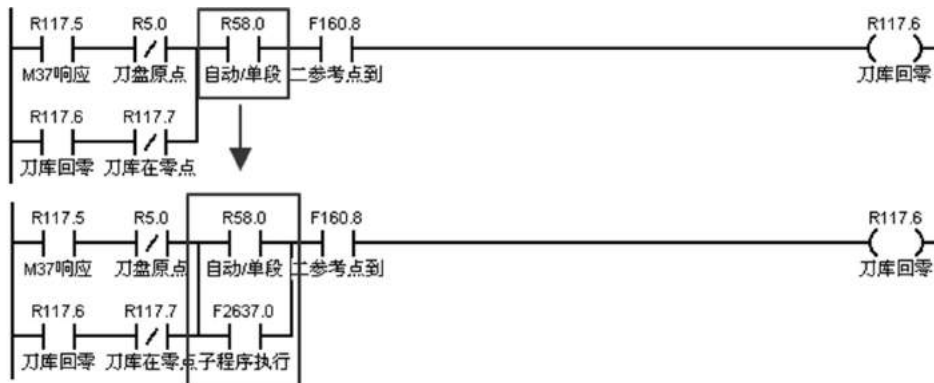
20　High-speed High-Precision Function

20.1　Machining Optimization Function G125/G126

Function and Purpose

If the processing optimization function is turned on by the G command (G125) in the program, the CNC system can optimize the speed of the related processing program (G code program) to improve the surface quality of the part. This function is mostly used for finishing of curved or mold parts.

Command Format

G125 ； Call machining optimization function

G126 ； Cancel machining optimization function

Example

The processing optimization function used to optimize standard curved surface parts processing procedures

```
%0001
G40 G17 G49 G80 G90
G54
N0010 (ROUGH_MILL)
G0 X46.694 Y-51.205 S6000 M03
G125                              ；  Call machining optimization function
Z5.
Z2.521
```
……………………………
```
G2 X-40.694 Y-53.651 I13.015 J58.936
G1 X-46.694 Y-51.382
G1 Z2.521
G0 Z5.
X46.694 Y-44.44
Z2.521
G1 Z-0.479
G1 X40.694 Y-47.014
G2 X16.255 Y-51.349 I-21.491 J50.099
G1 X-17.851 Y-51.371
```
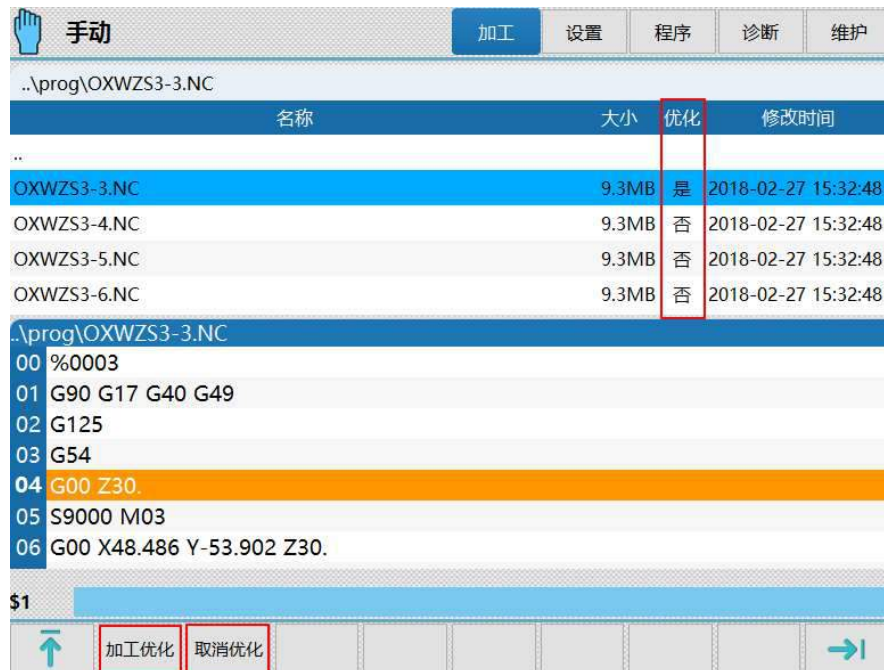…………………………………

G126   ； Cancel machining optimization function

M30

Description

Interface button and logo introduction

The following figure shows the program selection interface

Machining optimization

Optimize the processing program (G code) and generate the corresponding optimization file.

Operation

Select the corresponding program and click the key, then the program will be optimized. The interface prompt bar and pop-up box display the processing optimization progress, as shown below:

After the program is optimized, the optimization column will display "Yes", as shown in the figure below.

Cancel optimization

When the program optimization process is terminated, the optimization column displays "No".

Operation

Select the corresponding program and click the button, then the program optimization will be cancelled, and the optimization column will display "No", as shown in the figure below:

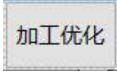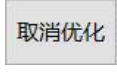**Optimization column**

"Yes" means that the processing optimization operation has been completed for G code program, "No" means that the processing optimization operation has been not completed for G code program.

Description of processing program optimization

G code optimization operation

All G codes can be optimized through the processing optimization key. The interface operation

and corresponding handling are shown in the following table.

Object	Operation	Result
Unoptimized G code	加工优化	The G code is optimized normally. After the optimization is completed, the optimization column displays "Yes".
Optimized G code	取消优化	G code optimization is canceled, and the optimization clolum displays "No"

Note

- Different programs have different processing optimization procedures. Generally, t he larger the program, the longer the optimization time, even as long as several minutes. The program interface will display "Processing optimization complete d..%, please wait patiently".

- In the process of processing optimization, the processing optimization interface is locked and cannot be switched to other interfaces until the optimization is compl eted, and then other operations can be performed.

- For ordinary programs (<20M), during the optimization process, the optimization completion progress is displayed in percentage (%) on the prompt bar; for super large programs (>20M), during the optimization process, the optimization progress is displayed as the number of program lines that has been optimized .

- If there is an alarm in the system or an alarm occurs during the optimization, th e optimization will fail, and user needs to perform the reset and cancel the opti mization. The following prompt will be given: the system alarm causes the proce ssing optimization to fail, press reset button to cancel the optimization.

Optimized G-code editing and corresponding optimization

When editing or modifying the optimized G code, the program needs to be re-optimized automatically. The specific operation and handling are shown in the following table:

Operation	Handling
Replace the optimized program with the program of the same name	Optimization of G code is canceled, and optimization column displays "No"
Delete optimized program	Optimization of G code is canceled, and optimization column displays"No"
Save as or copy an optimized program	The program is still being optimized, the optimization column indicates "Yes"

	The program saved or copied is not optimized, the optimization column indicates "No"
Rename the optimized program	The renamed program is still being optimized, the optimization column indicates "Yes"
Modify the optimized program, and save the modification	Optimization of G code is canceled, and optimization column indicates "No"
Click "cancel optimization" button for the optimized program	Optimization of G code is canceled, and optimization column indicates "No"

Optimization of called subprograms in the main program

In the main program/subprogram calling mode, the optimization processing mode can be called by adding G125 to the main program, so that all subprograms adopt the optimization processing mode. It is also possible to optimize a subprogram separately to realize the optimization processing of the subprogram.

Operation	Handling
Optimize main program	All subprograms called by the main program can be optimized synchronously, and the optimization attribute of each program is displayed as "Yes".
Optimize subprogram seperately	The optimization attribute of the corresponding subprogram is displayed as "Yes".
Modify the optimized main program	Optimization of main program is canceled, and optimization column indicates "No"
Modify the optimized subprogram	Optimization of G code is canceled, and optimization column indicates "No"

Note

- When the main program is used in combination with an external subprogram, if a certain subprogram needs to use the optimization processing mode, add the G125 command at the head of the subprogram, add the G126 command before M99 at the end of the program, and optimize the corresponding subprogram.

- Currently calling processing optimization function of internal subprogram is not supported.

- The processing optimization function of M98P_L_ mode is not supported.

Activation of machining optimization mode

The optimized program can call the mold optimization processing mode only through the G command calling. Generally, the G125 command is added to the program header.

Related G commands

G125: Indicates that the optimization processing function is turned on.

G126: Indicates that the optimization processing function is turned off.

Note

- For unoptimized G code, G125 is added to the program header.When the program is running, an alarm will be issued and prompts user "specified processing optimization code does not exist".

- If no G125 command is added to the optimized G code program, it will run in the original mode and the original speed planning algorithm is selected to run the program.

- If the G125 command is added to the optimized G code program, it will run in the optimized mode, and the new speed planning algorithm will be selected to run the program.

Parameter configuration

1) Internal parameters of the algorithm

The following parameters are used internally by the algorithm, and can be set according to the following instructions under normal circumstances; if the modification is needed, they must be modified under the guidance of the developer.

040045 Standard field radius 1.35

Note: This parameter is to set the neighborhood radius length based on neighborhood speed planning, the default value is 1.35.

040046 Single point deceleration angle scale factor 1.000

Along the tool path, when the tangent vector angle between two adjacent blocks is greater than a certain threshold (the default is 10°), the end of the block is used as the deceleration point at the end. This parameter is used to adjust the angular velocity threshold.

Actual threshold=10*single point deceleration angle scale factor

040047 Minimum angle ratio in the angle ratio criterion 3.000

This parameter is used to set the determination threshold of the angle ratio of front and back ends of the block in the end point angle ratio criterion.

040048 Minimum angle ratio in the relatively long line segment criterion 0

This parameter is used to set the determination threshold of the angle ratio of the front and back ends of the block in the relatively long line segment criterion.

040049 Criterion combination mode 0X0

This parameter is used to set the criteria combination method and the curvature calculation optimization method.

Bit 0

0: corner criterion, relatively long line segment criterion, and inflection point criterion are all effective.

1: Relatively long line segment criterion and inflection point criterion are effective

2: Relatively long line section criterion and corner criterion are effective

Bit 1

0: Curvature radius calculation mode 1, the default mode.

1: Curvature radius calculation mode 2.

040068 The second processing code spline merging enable 0

This parameter is used to enable the spline merging function of the second processing code.

0: Turn off the spline merging of the second processing code

1: Turn on the spline merging of the second processing code

040069 Speed planning mode 0

There is a motion planning mode for small line segment interpolation in the HNC-8 CNC system. Currently 0 is set.

(1) Recommended parameter of small line segment

Basic small line segment parameters (040069-040087) can be based on this, as shown in the following table

Maximum length of small line segment (mm)	Minimum smooth interior angle (°)	Contour allowance (mm)	Read-ahead blocks	Command smoothing period

1.5	0	0.015	800	20
Acceleration time proportional factor	Acceleration jerk time proportional factor	Centripetal acceleration (mm/s2)	Smooth closing of preprocessing	Collinear angle threshold (radians)
1	1	200~2000	1	0.017

G05.1Q1, G05.1Q2, and G05.1Q3 can be set.

(2) Axis parameter

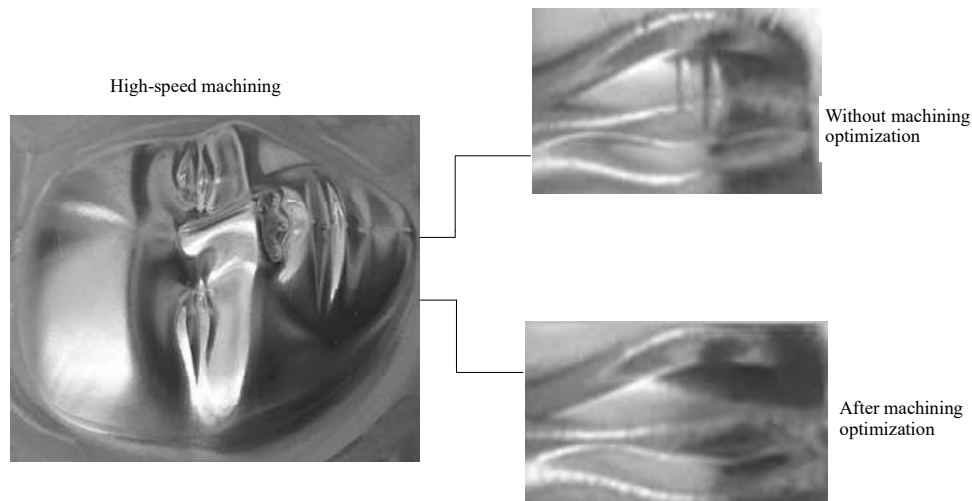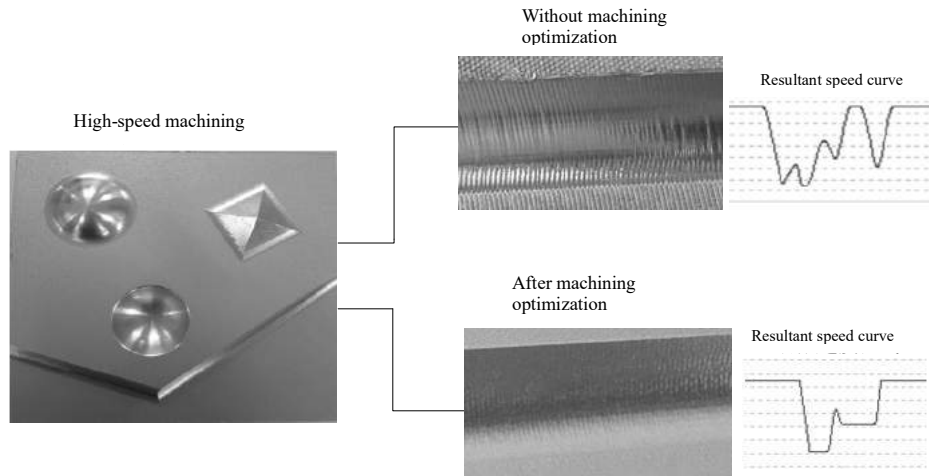| Acceleration deceleration time constant (ms)	Acceleration deceleration jerk time constant (ms)
16	8

 Example

Using of mold optimization processing mode

The mold optimization processing mode improves the processing speed planning, reduces the frequency of the speed fluctuations during the processing, improves the smoothness of the processing speed, and at the same time ensures the consistency of the speed of the adjacent tool path in the three-dimensional surface processing, improves the processed surface quality, and effectively solves the obvious cut marks in surface processing. In the case of the same processing effect, the processing efficiency can be improved.

High-speed machining

Without machining optimization

After machining optimization

Without machining optimization

Resultant speed curve

High-speed machining

After machining optimization

Resultant speed curve

Improvement of machining efficiency and quality
Comparing with ordinary machining, the optimization machining control can increase the surface quality when the machining efficiency is constant.

High

Surface quality

7min33s

Optimization machining control

9min6s

Ordinary machining control

Low

7min16s

Short   Machining time   Long
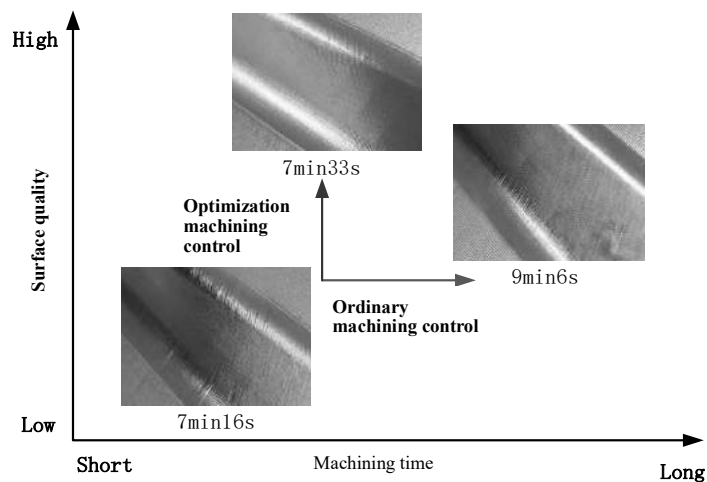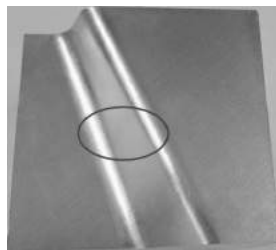
⚠ **Note**

(1) Use G125 in the program to turn on the optimization processing mode. If G125 is not used to turn on the optimization processing mode, the processing is the same as the normal version.

(2) The optimization processing mode is generally used for three-dimensional surface finishing, mainly to solve the obvious over-cutting in the processing of complex parts, and at the same time, it can improve the processing efficiency while ensuring the processing quality.

(3) The G125 command must be placed before the traverse command of the program head.

(4) When the main program is used in combination with an external subprogram, if a certain subprogram needs to use the optimization processing mode, then add G125 command at the head of the subprogram, add G126 command before M99 at the end of the program, and optimize the corresponding subprogram.

(5) The internal subprogram processing optimization function is not supported.

(6) The processing optimization function in M98P_L_ mode is not supported.

20.2 High Speed High Precision Mode Selection（M）（G05.1）

![icon] **Function and Purpose**

In modern CNC systems in order to ensure the quality of the contour processing, in addition to the good mechanical accuracy of the CNC equipment, a CNC system with high-speed processing capabilities and high-speed and high-precision functions is also required.

For the actual contour processing, it can generally be divided into roughing, semi-finishing and finishing according to the parts processing procedure. Regardless of the processing mode, the actual processing and finished product quality requirements can be decomposed as follows:

Machining requirement	Feature
High efficiency	Focus on the processing time and increase the processing speed of the free curve. There is no high requirement on dimensional precision of workpiece. It is generally required for roughing or intermediate processing.
Balance of efficiency and precision	There is no high requirement for processing time and accuracy, but both need to be controlled within a certain range.
High precision	Focus on the dimensional accuracy of the workpiece. The surface quality is high, and the processing efficiency is not considered. It is generally finishing.

As shown in the above table, in the actual machining process, both efficiency and accuracy cannot reach the optimal level, and only one of them can be emphasized or a balance between them is maintained based on different process characteristics and requirements. There are a series of control parameters that play a major role in processing efficiency and processing accuracy in the system. The combined application of these control parameters can meet different processing requirements.

Through this command, different processing modes can be switched to meet the processing requirements of different technological characteristics.

![icon] **Command Format**

G05.1 Q_ ； Specifying processing mode

G05.1 Q0； Default mode

Parameter	Meaning
Q_	Select the processing mode, the value can be 0, 1, 2, 3. The four groups of processing modes can be switched mutually by G05.1Q_.

Command	Description
G05.1Q0	Default mode: focus on a balance of efficiency and precision
G05.1Q1	High precision mode: focus on the machining surface and dimensional precision
G05.1Q2	High-speed high-precision mode: focus on smoothness of processing and a balance of efficiency and precision
G05.1Q3	High speed mode: focus on efficiency, improve processing speed of free curve

With G05.1Q1, G05.1Q2, and G05.1Q3, user selects different high-speed and high-precision machining modes and configures different parameters (speed smoothing, spline smoothing, etc.) to achieve high efficiency, high precision, or balance of efficiency and precision.

Example

High-speed high-precision mode is used to edit small line segment program

1）**G05.1Q0**　　　　　　　；　**Default mode; balance of efficiency and precision is focused**
  example
  G54
  G0 X48.689 Y-51.225 S6000 M03
  G05.1Q0　　　　　　　；　Default mode; balance of efficiency and precision is focused
  X46.694 Y-37.445
  ………………
  Z2.521
  M30

2）**G05.1Q1**　　　　　　；　**High-precision mode; machining surface and dimensional precision is focused**
  Example
  G54
  G0 X48.689 Y-51.225 S6000 M03
  G05.1Q1　　　　　　；　**Default mode; balance of efficiency and precision is focused**
  X46.694 Y-37.445
  ………………
  Z2.521
  M30

3）**G05.1Q2　；　High-speed high-precision mode; machining smoothness and balance of efficiency and precision is focused**
  Example
  G54
  G0 X48.689 Y-51.225 S6000 M03

G05.1Q2 ； **High-speed high-precision mode; machining smoothness and balance of efficiency and precision is focused**

X46.694 Y-37.445

………………

Z2.521

M30

4） **G05.1Q3 ； High-speed mode; machining efficiency is focused, and machining speed of free curve is improved**

Example

G54

G0 X48.689 Y-51.225 S6000 M03

G05.1Q3 ； **High-speed mode; machining efficiency is focused, and machining speed of free curve is improved**

X46.694 Y-37.445

………………

Z2.521

M30

Note

(1) G05.1Q_ can only be called in a single line, and cannot be called in the same line with other commands.

20.3   High Speed High Precision Parameter Setting

Function and Purpose

In the actual CNC machining process, different workpieces processed by the same machine tool or different parts and procedures of the same workpiece have their own technological characteristics and processing requirements. The use of a set of control parameters obviously cannot achieve good processing results. It is necessary to adopt a combination of different control parameters for different parts and processes for overall coordinated control, so as to achieve the best processing effect.
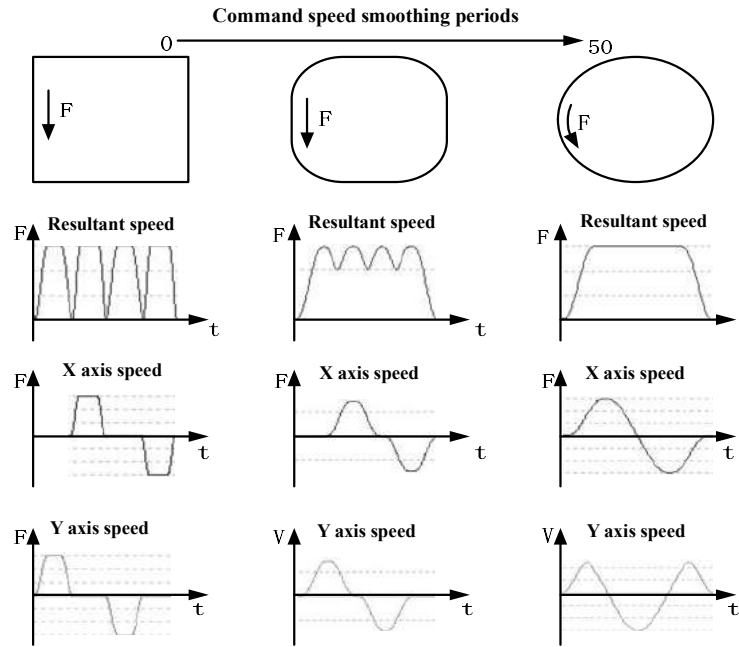

Description

Control parameters

Command speed smoothing periods

Parameter type	Parameter	Unit	Value range	Default value
Channel parameter	040082	ms	0 to 50	0

Through the command speed smoothing window, the smooth transition of the command speed is realized, the speed fluctuation is reduced, the speed stability in the high-speed control is ensured, the vibration of the machine tool is reduced, and the processing efficiency is improved. The working principle is shown in the figure below. For a small square (side length <0.5mm), as the smoothing period increases, the 90° corners of the square will gradually become a circular transition until it becomes a full circle; The amount of change in the composite speed gradually decreases until it becomes a constant speed section, and the uniaxial speed gradually becomes a smooth transition.

Command speed smoothing periods

It can be seen from the above principle that the command speed smoothing periods help to improve the smoothness of processing speed, reduce machine tool vibration and improve efficiency, and at the same time it will affect machining accuracy. The larger the command smoothing periods, the lower the accuracy. That is, when the accuracy is the priority, the lower value the better; when the efficiency is the priority, the larger value the better; when considering the smoothness (efficiency and accuracy are both balanced), the value is between 10 and 30.
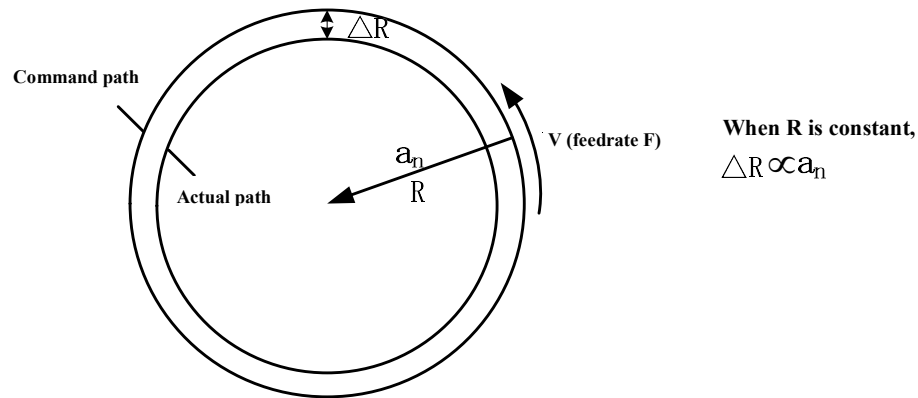
Centripetal acceleration

Parameter type	Parameter	Unit	Value range	Default value
Channel parameter	040084	mm/s2	1.0 to 100000.0	1000.0

According to Newton's second law, the force will cause the object to produce an acceleration. The acceleration produced by the centripetal force is the centripetal acceleration. Centripetal acceleration is a physical quantity that reflects the speed and the direction of circular motion. Centripetal acceleration only changes the direction of the velocity. The maximum centripetal acceleration is used to set the limit of the maximum centripetal acceleration.

As shown in the figure below, the centripetal acceleration (a_n), linear speed (v) and radius R satisfy the following relationship:

$$a_n = v^2/R$$

When R is constant,

$$\triangle R \propto a_n$$

The feed cutting speed of the circular arc can be limited by the centripetal acceleration. The table of comparison between the maximum feed speed of a circle with a certain radius (R) and the centripetal acceleration is as follows:

Centripetal acceleration (mm/s2)	500	1000	2000	3000	5000	6000
R5mm circular deceleration speed（mm/min）	3000	4242.64	6000	7348.47	9486.83	10392.3
R1mm circular deceleration speed（mm/min）	1341.64	1897.36	2683.28	3286.33	4242.64	4647.58

When the given feedrate F is larger than the circular deceleration speed of the current radius, the speed will reduce to the circular deceleration speed of the current radius; when the given feedrate F is lower than the circular deceleration speed of the current radius, the circular feed cutting is executed at the given speed F.

According to the acceleration principle diagram, for a circular arc with a certain radius (R), the radius error ($\triangle R$) is proportional to the centripetal acceleration. Therefore, in the case of accuracy priority, the feedrate on arc can be limited by reducing the acceleration; in the case of efficiency priority, the feedrate on arc can be increased by increasing the acceleration, but the centripetal acceleration needs to be adjusted based on the actual situation to meet the efficiency and precision requirements.

Example: When the centripetal acceleration is set to 1000 mm/s2 and the given feedrate is 3000mm/min, for an arc with a radius of 5mm, the feed cutting is performed at 3000mm/min; for an arc with a radius of 1mm, the feed cutting is performed at 1897.36mm/min.
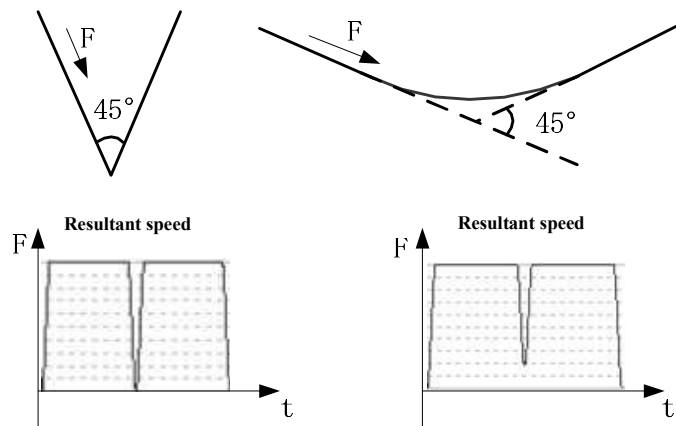
Minimum internal angle of corner smoothing

Parameter type	Parameter	Unit	Value range	Default
Channel	040071	Degree	0 to 180	100

parameter				

During continuous small line segment interpolation, local speed reduction can be performed based on the actual programmed path. For situations where the sharpness of contour corner needs to be highlighted, the speed will be reduced to 0 at the top of the sharp corner. This parameter is used to set the value of the angle. If the processing angle is smaller than this angle, the exact stop will be performed. If it is greater than this value, other determinations will be used to plan the speed reduction at this angle, as shown in the figure below;


If θ is larger than the set value, the speed will be reduced to 0 at the top of the sharp corner.
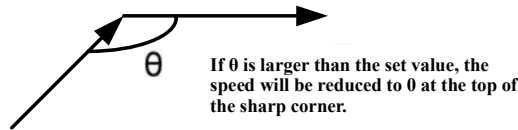
Example: When the minimum interior angle of corner smoothing is set to 90°, the 45° (<90°) and 135° (>90°) corner paths and speed curves are shown in the following figure:

Corner deceleration proportional factor

Parameter type	Parameter	Unit	Value range	Default value
Channel parameter	040074	%	0 to 150	100

For the polyline of which corner angle is greater than the minimum inner angle of corner smoothing, that is, it adopts arc transition mode to execute the feed at the corner. The corner deceleration speed can be controlled by the corner deceleration proportional factor. The smaller the setting value, the smaller the corner deceleration speed, the smaller the corner roundness, and the smaller the accuracy error in theory, but the milling time at the corner will be longer and the efficiency will be reduced.

F1>F2   d1>d2
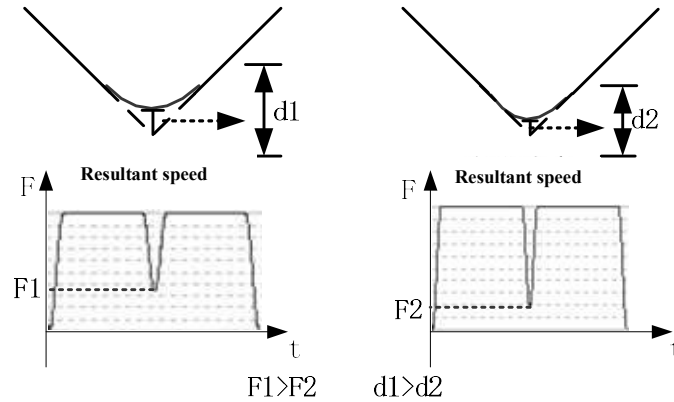
The corner deceleration proportional factor can be adjusted based on the specific processing requirements. In the case of accuracy priority, the value is set as small as possible; in the case of efficiency priority, the value is set as large as possible; for the case of both efficiency and accuracy being balanced, the value is between 90 and 99.

Whether the arc is discrete into straight lines

Parameter type	Parameter	Unit	Value range	Default value
Channel parameter	040079	-	0~1	0

If this function is turned on, the arc can be discretized into the connection of tiny line segments, then the situation where a straight line meets an arc or an arc meets an arc can be equivalent to the connection of a straight line to a straight line, as shown in below figure. The speed at the junction of the two can be processed by the corner deceleration.

Connection between straight line and arc

Connection between arc and arc
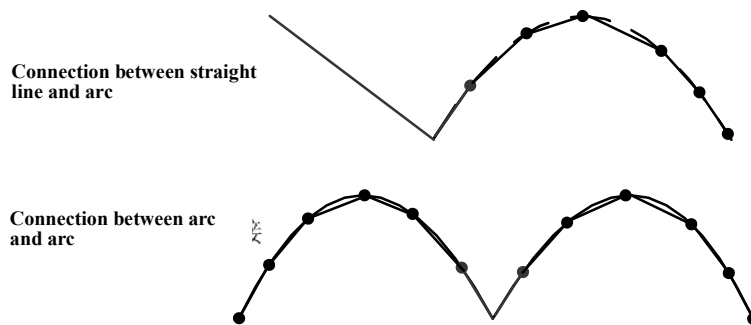
Spline smoothing function

Parameter type	Parameter	Unit	Value range	Default value
Axis parameter (servo parameter)	100209	-	-32767 to 32767	0

Spline curve refers to a curve obtained by a given set of control points, which can be used to describe free curves and curved surfaces. It is specified as a CAD/CAM data exchange standard by the International Organization for Standardization. The chord line is usually used for approximating the arc at the time of spline curve interpolation, and the accuracy depends on the number of the approximating chord. As shown in the figure below; the system interpolation period is 1ms, and the system is specified to run a full circle in 8ms, then it will approximate the circle to a regular octagon, and the servo position control period is 0.2ms. After the spline smoothing function is turned on (100 is set), the full circle spline can be approximated to a regular 40-sided polygon.
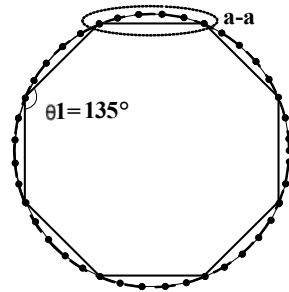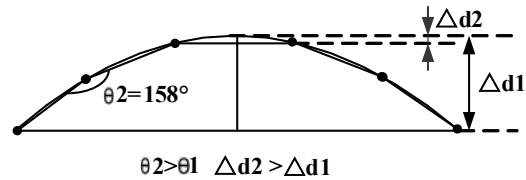

图（a）   图（a-a）

After the spline is smoothed, the included angle of the spline segment becomes larger, which is beneficial to reduce the speed fluctuation and improve the surface finish; the contour error is greatly reduced compared with the previous one, which reduces the loss of precision and improves the processing accuracy.

Machining acceleration/deceleration time constant

Parameter type	Parameter	Unit	Value range	Default value
Axis parameter	100038	ms	0~2000.0	32

"Machining acceleration/deceleration time constant" refers to the time for linear axis (G01, G02, etc.) to accelerate from 0 to 1000mm/min or decelerate from 1000mm/min to 0, as shown in the figure below (T represents the machining acceleration deceleration time constant). This parameter determines the axis's processing acceleration. The larger the processing acceleration/ deceleration time constant, the slower the acceleration/deceleration. This parameter is set based on the motor inertia moment, the load inertia, and the acceleration capability of the drive.
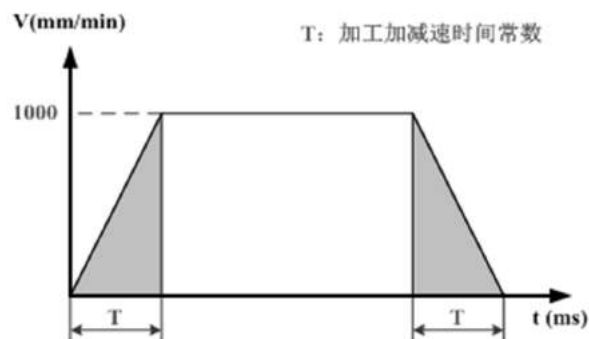
The comparison table of common machining acceleration/deceleration time constant and acceleration is as follows:

Machining acceleration/deceleration time constant	2ms	8 ms	16 ms	32 ms	64 ms
Acceleration	1g	0.2g	0.1g	0.05g	0.02g

Example: The machining acceleration/deceleration time constant is set to 6ms, then the processing acceleration is calculated as follows:
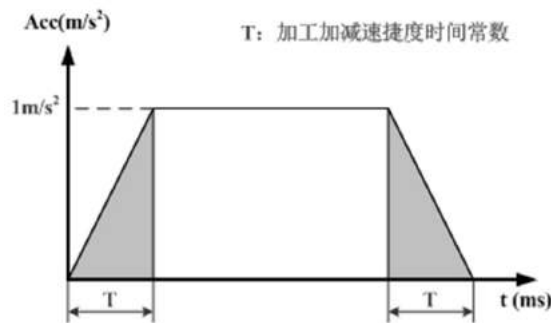
1000mm/60s≈16.667mm/s

16.667/0.006≈2778mm/s2≈0.283g   (1g=9.8m/s2)

Machining acceleration/deceleration jerk time constant

Parameter type	Parameter	Unit	Value range	Default value
Axis parameter	100039	ms	0~2000.0	128

"Machining acceleration/deceleration jerk time constant" refers to the time when the acceleration increases from 0 to 1m/s2 or decreases from 1m/s2 to 0 during axis machining movement (G01, G02, etc.), as shown in the figure below (T represents the machining acceleration deceleration jerk time constant). This parameter determines the machining jerk of the axis. The larger the time constant, the smoother the acceleration changes. This parameter is set based on the motor, the performance of the drive and the load, and is generally set between 8 and 150.

Example: Assuming that the processing acceleration is 0.05g (0.49m/s2) and the processing acceleration/deceleration jerk time constant is set to 128ms, the jerk is 0.49/0.128≈3.8m/s3.

Machining acceleration time coefficient

Parameter type	Parameter	Unit	Value range	Default value
Axis parameter	040156/040176/040196	-	0.01 to 100.0	1

Take the axis parameter "machining acceleration/deceleration time constant" as the reference

value, and use the "machining acceleration time coefficient" to convert the machining acceleration /deceleration time to change the acceleration. The formula is as follows:

Converted value of machining acceleration/deceleration time = machining acceleration/deceleration time constant * machining acceleration time coefficient

According to the characteristics of the processing program and the actual processing conditions, this parameter can realize the flexible switching of the acceleration, and optimize the processing efficiency or processing accuracy. When efficiency is the priority, this parameter can be appropriately reduced; when accuracy is the priority, this parameter can be appropriately increased.

Example: The machining acceleration/deceleration time constant is set to 8ms in the axis parameters, and the corresponding acceleration is 0.2g. When the machining acceleration time coefficient is 0.25, the machining acceleration/deceleration time is converted to 2ms, and the corresponding acceleration becomes 1g.

Machining jerk time coefficient

Parameter type	Parameter	Unit	Value range	Default value
Axis parameter	040157/040177/040197	-	0.01 to 100.0	1

Taking the axis parameter "machining acceleration/deceleration jerk time constant" as the reference value, the machining acceleration/deceleration jerk time is converted through the "machining jerk time coefficient" to change the time that the acceleration increases. The conversion formula is as follows:

Converted value of machining jerk time = machining acceleration/deceleration jerk time constant * machining acceleration time coefficient

Through this parameter, the acceleration speed can be flexibly switched, and the acceleration change can be reasonably controlled to ensure the stability of the machining.

Example: Assuming that the current processing acceleration is 0.05g (0.49m/s2) and the axis parameter "machining acceleration/deceleration jerk time constant" is set to 64ms, the jerk is 0.49/0.64≈7.6m/s3. When the machining acceleration time coefficient is 2, the converted value of the machining acceleration/deceleration time is 128ms, and the corresponding jerk becomes 3.8 m/s3.

 Example

In the actual CNC machining process, different machining processes and part contours will produce parts processing procedures with their own characteristics, and different machining requirements will also bring different control modes. Efficiency and precision are the two ends of the balance of CNC machining. Therefore, only the combination of the above parameters can be adjusted according to the specific processing object and customer needs, so that the efficiency

and accuracy can be biased or reach a suitable balance point, so that the overall processing effect can be optimized. The following explains the combination adjustment of parameters through practical application examples.
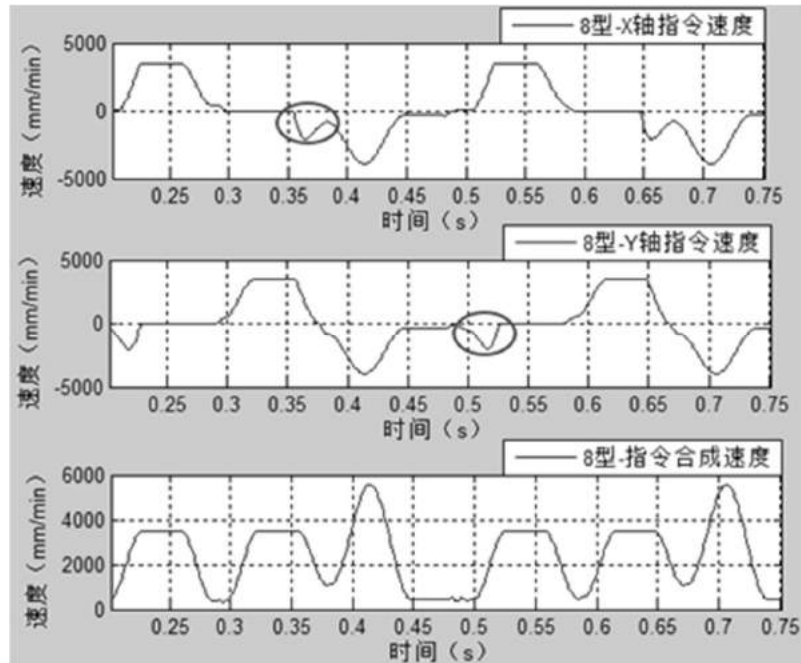
Cleanup machining parameter application

The cleanup machining is popular in 3C industries including mobile phone casings and circuit board fixtures.

The contour path is shown as below
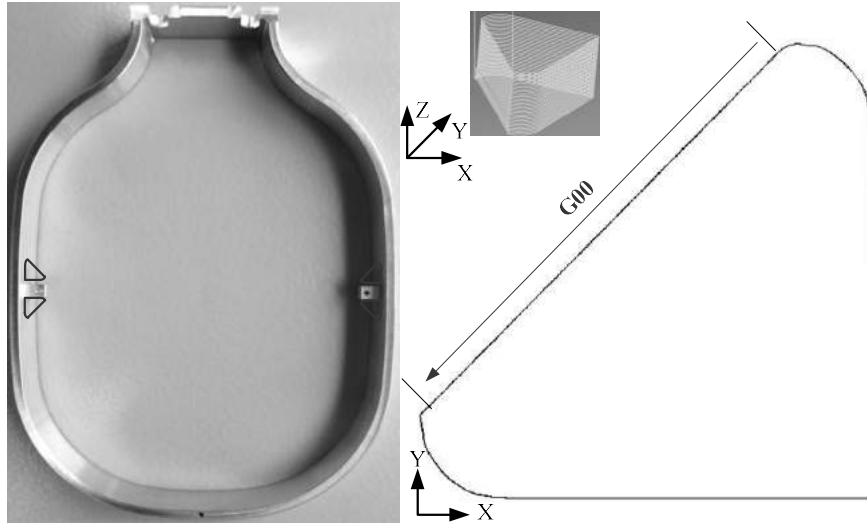
① In the actual processing process, the requirements for the cleanup machining efficiency is high, that is, the processing time is as short as possible.

② It can be seen from the machining process and machining contour that the machining requires high precision at 90-degree corners, and there is no actual effective cutting on the arc segment, so there is no need to consider the contour accuracy.

③ The overall machining contour is small, there are connections between straight lines and arcs, as well as G00 and G01, the speed changes quickly, and the speed fluctuates greatly. As shown in the figure below, the single-axis speed curve has a sudden change in speed. During the actual processing, this sudden change in speed is the main cause of severe vibration of the machine tool.

Parameter combination application

① To meet the high-efficiency requirements of machining, improve the system's acceleration capability, that is, reduce the machining acceleration/deceleration time constant and the machining acceleration/deceleration jerk time constant to the suitable values, but the values should not be too small to cause excessive acceleration and increase the vibration of the machine tool. For example, when the acceleration time constant and the jerk time constant are 4 and 1, 2 and 2 respectively in the actual debugging, and the machining time is the same. However, when the acceleration time constant is 2, the acceleration is too large and the machine vibration is severe, so choose the appropriate acceleration time constant as 4.

② For the sudden change of single-axis, increase the command speed smoothing coefficient to make the speed transition smoothly. As shown in the figure below, in the actual machining process, when the command speed smoothing coefficient is 40, the machine vibration almost disappears.
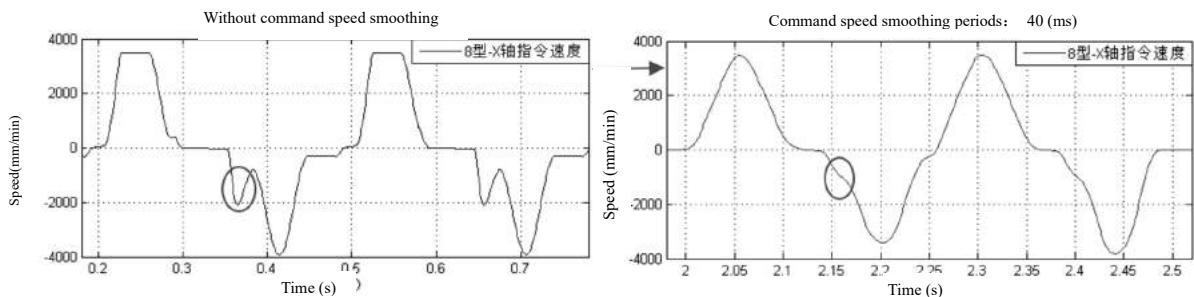
③ Since the accuracy of this machining arc is not high, the centripetal acceleration can be increased, and the circular cutting feedrate can be increased; at the same time, in order to improve the processing efficiency, reduce the minimum internal angle value of corner smoothing and increase the corner deceleration factor to increase the feedrate at the corner.

Parameter setting

Parameter	Set value
Machining acceleration time constant (ms)	4
Machining acceleration jerk time constant (ms)	1
Command speed smoothing periods (ms)	40
Min. interior angle of corner smoothing （°）	80
Corner deceleration factor (%)	90
Centripetal acceleration (mm/s2)	6000

Combination application of laser processing parameters

Laser cutting is widely used in contour processing. Take the "horse" contour as an example here;

The contour path is shown in the figure below:
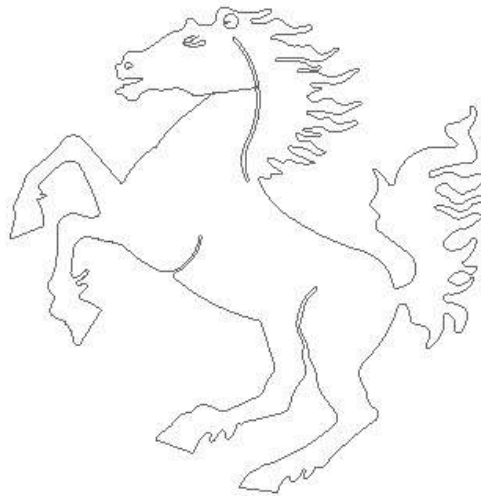
Analysis of processing requirements and characteristics:

① It can be seen from the outline of the horse that it has hair shape processing on the horse head and tail which need a large number of sharp corner processing, and the speed fluctuates greatly.

② According to the laser cutting processing requirements of the horse shape, when a certain processing time is ensured, the sharp corners of the processing contour must be sharp, and there is no rounded corners.

③ The processing speed is high and above F10000, and its acceleration is required to be close to 1g, but at the same time it is necessary to ensure that the machine vibration is small.

Parameter combination application

① For problems such as large speed fluctuations and strong machine tool vibration, and its sharp corners are processed into a large number of small line segments, the spline smoothing function is turned on to smooth the speed step changes in the high-speed section.

② Although the command speed smoothing function can effectively reduce speed fluctuations, but the excessive large value will change the sharp corners to rounded corners, so the command smoothing speed here should not be too high. In the actual debugging the value of 5 can reduce machine vibration , improve efficiency without affecting the shape of horse's hair.

③ For the acceleration requirement of 1g, the machining acceleration time constant must be reduced. The appropriate value is 2. However, the vibration of the machine tool increases due to excessive acceleration changes. Therefore, the acceleration jerk time constant is reduced to lower the acceleration change and increase the smoothness of the axis in the acceleration and deceleration stages, reducing machine vibration.

④ The processed parts require high precision and processing time. Appropriately reduce the centripetal acceleration so that the deceleration is performed at the small arc; increase the minimum interior angle of corner smoothing to improve the corner contour accuracy; adjust the appropriate corner deceleration factor to keep its value about 70, then the balance of accuracy and efficiency is ensured.

Parameter setting reference value

Parameter	Set value
Machining acceleration time constant (ms)	2
Machining acceleration jerk time constant (ms)	10
Command speed smoothing periods (ms)	5
Min. interior angle of corner smoothing (°)	130
Corner deceleration factor (%)	70
Centripetal acceleration (mm/s2)	2500
Spline smoothing coefficient	100

Note

1) In HNC8 CNC system, multiple sets of small line segment control parameters are used to achieve different parameter combination control, which includes the various control parameters described above that play a major role in processing accuracy and efficiency.

2) As shown in the above example, according to the specific processing technology and processing requirements, different parameter combinations are summarized to set the small line segment

control parameters of different groups. In actual processing, user only needs to call the corresponding small line segment combination parameter command to complete the change of control parameters, which can not only realize the combined use of multiple sets of control parameters, but also complete the quick switching of different control parameter combinations. Please find below for how to call the control parameter combinations.

G05.1 Q1 (The first group of small line segment control parameter combination)

……(Machining program 1)

G05.1 Q2 (The second group of small line segment control parameter combination)

……(Machining program 2)

G05.1 Q3 (The third group of small line segment control parameter combination)

……(Machining program 3)